CNC SYSTEMS

OSP5020M
OSP500M-G

SPECIAL FUNCTION MANUAL
(No. 2) (6th Edition)

ME51-019-R6

CNC SYSTEMS

OSP5020M
OSP5000M-G

SPECIAL FUNCTION MANUAL

(No. 2) (6th Edition)

MES1-0179-R6

SAFETY PRECAUTIONS

The machine is equipped with a variety of safety devices designed to protect prevent accidents and injuries from occurring. Still, operators must use the machine carefully and learn not to rely on these safety devices.

This section describes the general safety precautions required for machine operation. Every operator must read this section carefully and observe the guidelines it describes.

1. Before Turning on the Power

(1) Make sure that doors to the operation panel and the electric control cabinet are closed.

(2) Do not put any obstacles on or around the machine.

(3) Turn on the main power switch before turning on the **CONTROL ON** switch on the operation panel.

(4) Make sure that there is not dangerous elements around the machine.

2. In Manual or Continuous Operation

(1) Always follow the operation manual.

(2) Never operate the machine without closing the front shield.

(3) In initial grinding, check for operating conditions and interference according to the steps from no-load run to single block grinding to continuous operation.

(4) Ensure the safety before rotating or moving the spindle.

(5) Check the tool offset values.

(6) Check the zero offset values.

(7) Ensure that the spindle speed and feedrate override switches are set at 100 %. Operate the machine carefully and stop it immediately in case of abnormality. If necessary, set the feedrate override switch to 5, 10, and 50 % etc.

(8) Never attempt to touch the spindle or tool in rotation.

(9) Do not touch the spindle or the tool when indexing or changing speed. The spindle may turn.

(10) Never apply a rotating torque to the spindle by operations such as retightening of milling chuck before the spindle is completely indexed. The spindle may turn.

(11) Do not turn the spindle with the tool or other parts loosely fixed.

(12) Be sure to stop the spindle tool and use a brush or broom when removing chips adhering to the spindle.

(13) Cutting operation causes scatter of chips, coolant, and in some cases fragments of the tool.

Be sure to close the cover and stay away from the spindle.

(14) Load a workpiece securely on the table. After loading, ensure that there are no tightening tools left on the table.

(15) Do not move the table when an operator is on it.

(16) When two or more persons operate the machine, each operator should make sure of the other operators' safety.

(17) Movements of ATC, APC and AAC are large and dangerous. Operators must keep away from these units while in operation.

(18) If the ATC, APC or AAC stops in the middle of operation for some reasons and you inspect the machine without turning off the power, always secure an emergency escape zone and never touch the problem unit directly by your hands. The unit may start moving suddenly.

3. At the End of the Day

(1) Clean the area around the machine.

(2) Return the APC, ATC and AAC to the preset retracting position.

(3) Make sure that all the power switches are turned off before leaving your workplace.

(4) Turn off the **CONTROL ON** switch on the operation panel before turning off the main power switch.

4. In Maintenance Inspection or Abnormality Treatment

(1) Press the **EMERGENCY STOP** switch in case of emergency.

(2) The person responsible for maintenance should grasp the points of inspection or repair before maintenance work.

(3) When two or more persons conducts maintenance work, they should cooperate each other by exchanging signs and information.

(4) Use fuses or other parts of the specified rating for replacement.

(5) Turn off the power before replacing parts or checking wiring connections.

(6) After removing parts for inspection or repair, do not forget to remount and securely tighten them with the screws.

(7) Use calibrated measuring instruments in measuring inspections such as voltage check.

(8) Use great care in handling the following high-voltage units:

- Main breaker
- Servo drive unit
- VAC drive unit
- Power cables

(9) Do not leave any flammables or metallic parts inside the operation panel or the terminal box.

For the safety precautions in machine operation, refer to the Instruction Manual for the NC Machining Center.

5. Symbols

This manual uses the following symbols to emphasize the items which must be strictly observed:

REMARKS : Precaution in machine operations.

Default of this item affects the machining accuracy and smooth operation.

CAUTION : Default of this item may cause faulty machine operation, machine damage, or operator injury.

More care must be exercised to this item than the items indicated under "REMARKS".

WARNING :

Default of this item may cause not only machine damage but also serious accidents that affect human life.

Sufficient care must be taken to this item.

TABLE OF CONTENTS

PAGE

PAGE

PAGE

PAGE

PAGE

PAGE

PAGE

PAGE

PAGE

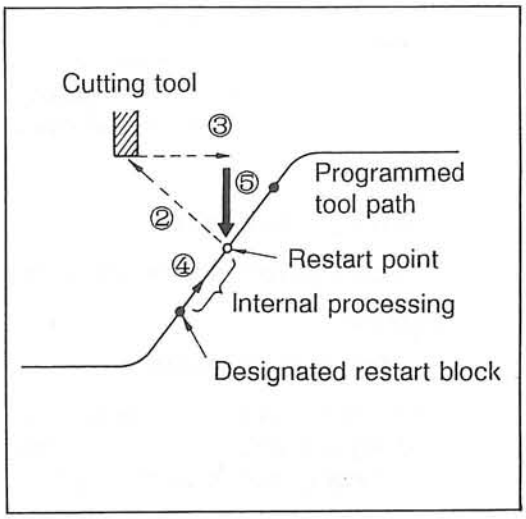
PAGE

SECTION 1 MID-BLOCK RESTART FUNCTION

The mid-block restart function covered by this section is closely related with the sequence restart function provided as a standard function of the OSP5020M/OSP500M-G. Therefore, basic knowledge of the sequence restart function is necessary to use this mid-block function. Before reading this section, it is recommended to read "Return Search and Sequence Restart" in OSP5020M/OSP500M-G Operation Manual (Publication No. 3354-E) - Operation - thoroughly.

1. Function Overview

If programmed cutting operation is interrupted halfway, it is possible to restart the operation from the beginning of the interrupted block with the standard sequence restart function. The mid-block sequence restart function allows the operation to restart from the point where the operation has been interrupted. In addition, the operator can designate the axis to return to the sequence restart point first using the option designation function.



The mid-block restart function allows the interrupted operation to restart from the interrupted point in the following sequence.

① Operation is interrupted halfway during the execution of a block of commands.

② Retract the cutting tool to a desired point.

③ Manually bring the cutting tool to a point close to the restart point.

④ After searching the block from which the sequence is to restart, press the **CYCLE START** button. The commands are internally processed* up to the restart point.

⑤ Press the **SEQ. RESTART** button. The cutting tool is positioned at the restart point in a manual cutting feedrate.

⑥ Press the **CYCLE START** button, and the interrupted operation is resumed from the restart point.

* In this processing, the commands are processed without actual machine operations like in the machine lock state operations.

2. Command Format for Sequence Restart Operation

This section explains the command formats used to designate sequence restart operation.

2-1. Restart from Designated Sequence

=RS␣sequence-name [,number-of-repetitions]
[;option-designation] **WRITE**

Note 1: *Designation in [ ] can be omitted.*

Note 2: *The following option designation is possible. Details of option designation is explained in Section 3.*

;±∗·····Axis movement priority
;M∗·····Restart point designation option
;TS·····Restart point designation option

Example 1:

=RS␣N100 **WRITE**

Sequence restart from sequence name N100

Example 2:

=RS␣N100,3 **WRITE**

When the same sequence name appears in the program more than one time, it is necessary to designate which of the sequence name should be searched for restart, first, second, etc. with a comma (,) placed preceding it.

If such data is not set, the sequence restart occurs from the designated sequence name appearing first.

The allowable maximum number for this setting is 9999.

With the example program above, sequence restarts from the third N100.

Example 3:

=RS␣N100; – Z **WRITE**

Sequence restarts from the N100 sequence. Positioning to the restart point is carried out by axes other than Z-axis first when the **SEQ. RESTART** button is pressed. Z-axis returns to the sequence restart point when the **SEQ. RESTART** button is pressed next.

2-2. Restarting from Designated Block

=RS␣block-count-data [;option-designation] **WRITE**

Note 1: Designation in [ ] can be omitted.

Note 2: Option designation as explained in 2-1 also applies in this case.

Note 3: Block count data must be smaller than 65535.

Example 1:

=RS␣127 **WRITE**

Sequence restarts from the 127th block.

Example 2:

=RS␣127; +Z **WRITE**

Sequence restarts from the 127th block; Positioning to the restart point is carried out by Z-axis first when the **SEQ.RESTART** button is pressed. The other axes return to the sequence restart point when the **SEQ.RESTART** button is pressed next.

2-3. Restart from The Designated Block Preceding The Restart Block

=RS_* [number-of-blocks] [;option-designation] **WRITE**

Note 1: Designation in [ ] can be omitted.

Note 2: The block count data corresponding to the block at which the control has been reset is registered in the NC and it is called out by the designation of an asterisk (*).

Example 1:

=RS_* **WRITE**

Sequence restarts from the sequence at which the cursor (↑) was located just before the control has been reset.

Example 2:

=RS_*−5 **WRITE**

Sequence restarts from the sequence five blocks ahead the sequence where the cursor (↑) was located just before the control has been reset.

3. Option Designation

Using the option designation, it is possible to specify the order in which axis positioning is carried out at the sequence restart point. Designation of the restart point is also possible by the option designation.

3-1. Axis Positioning Order Designation

;±*

(1) With this option code, designation of the axis movement order for carrying out positioning at the sequence restart point is possible.

(2) Designate the following items

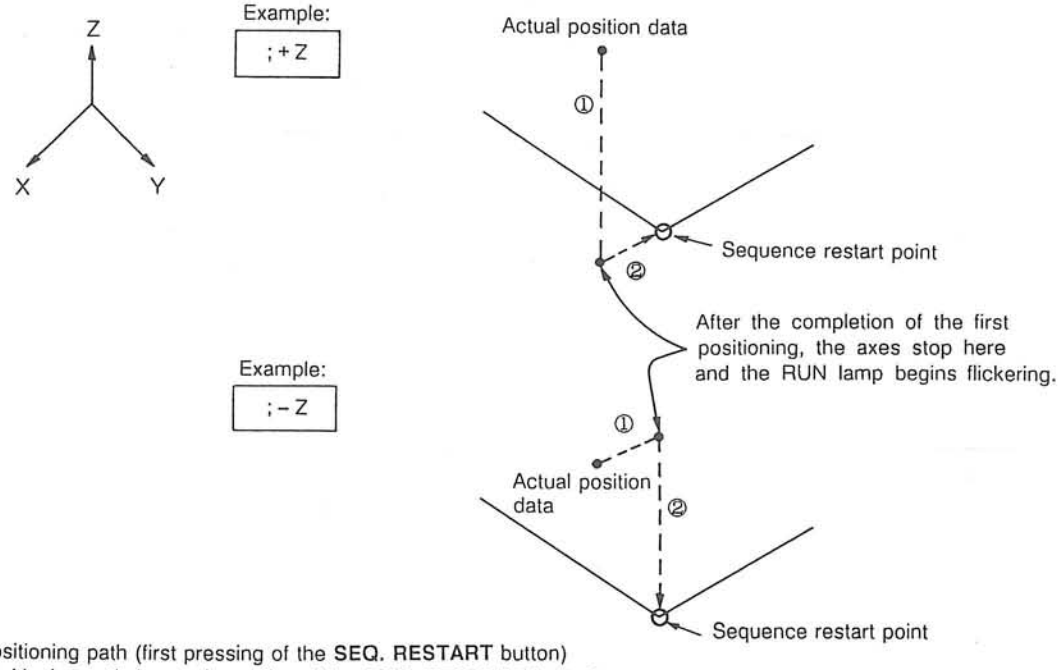
± ..... Either "±" "+" or "−" sign

* ..... X, Y, or Z

(3) The option code determines the axis positioning order in the following manner:

+* ..... The axis designated is first position at the sequence restart point when the **SEQ.RESTART** button is pressed. Positioning of the other axes occurs when the **SEQ.RESTART** button is pressed in the second time.

−* ..... The axis designated is positioned at the sequence restart point when the **SEQ.RESTART** button is pressed in the second time. With the first pressing of the **SEQ.RESTART** button,the other two axes are positioned at the sequence restart point.



Positioning path (first pressing of the **SEQ. RESTART** button)
Positioning path (second pressing of the **SEQ. RESTART** button)

3-2. Designation of Sequence Restart Point - ;M Option

;M∗

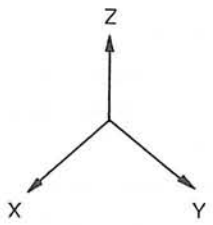
(1) Program is processed internally from the block one block preceding the designated sequence restart sequence or block and the point where the actual position data of the designated axis matches the data of the position where the axis is actually located is taken as the restart point.

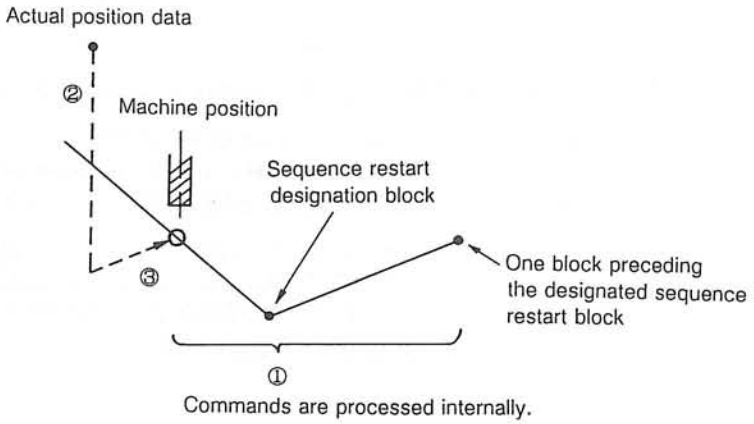
(2) Designate X, Y or Z at the symbol " ∗ ".

The order of axis movement can be designated by an option code

Example:

```
;MY + Z
```





3-3. Designation of Sequence Restart Point - ;TS Option

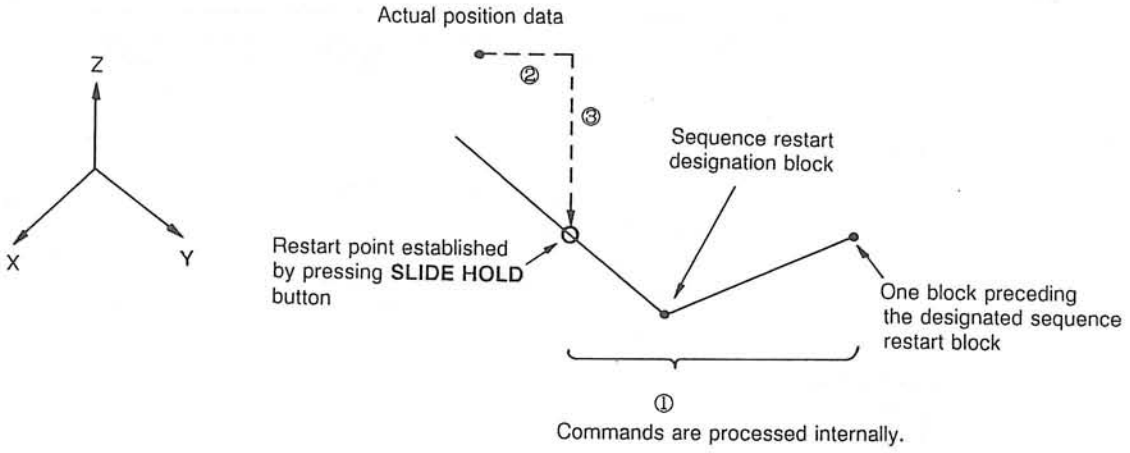
;TS

(1) Program is processed internally from the block one block preceding the designated sequence restart sequence or block and the point at which axis movements stop due to the pressing of the **SLIDE HOLD** button is taken as the restart point.

(2) The order of axis movement can be designated by an option code "±∗".

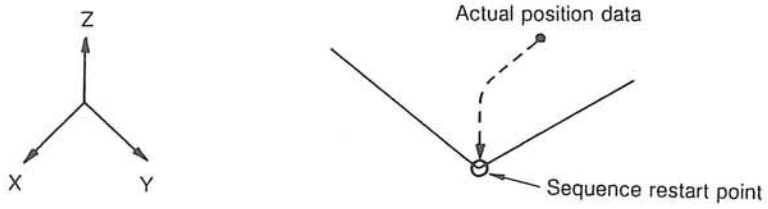
Example:

```
;TS–Z
```



3-4. Axis Movements without Option Designation

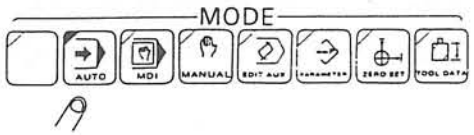
When the **SEQ. RESTART** button is pressed while no option code is designated, the three axes are positioned at the start point of the designated sequence or block at a manual cutting feedrate.



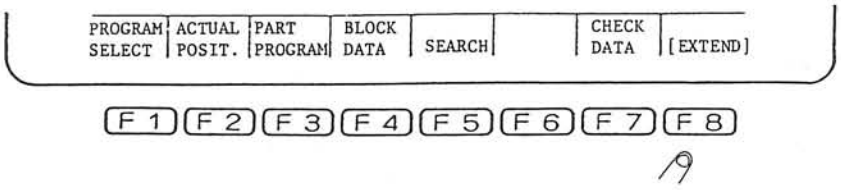
4 Operations

4-1. Sequence Restart from The Designated Sequence

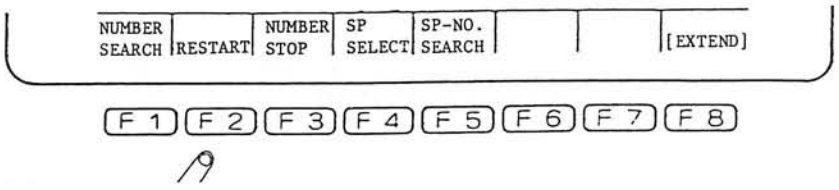
(1) Select the AUTO OPERATION mode by pressing the **AUTO** key.



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



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



(4) Input a sequence name.

(5) Press the **WRITE** key.

(6) Press the **SEQ. RESTART** button.

Positioning is carried out at the start point of the designated sequence in a manual cutting feedrate.

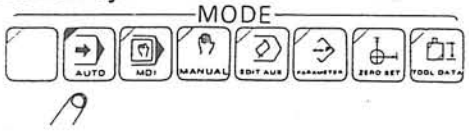
(7) Press the **CYCLE START** button.

Programmed operations are resumed.

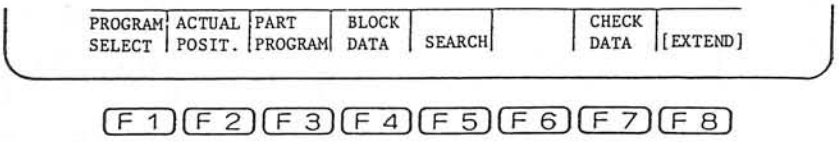
4-2. Sequence Restart from The Position Where Cutting Tool is Actually Located

(1) Position any of X, Y and Z axes at a point where programmed operations are to resume.

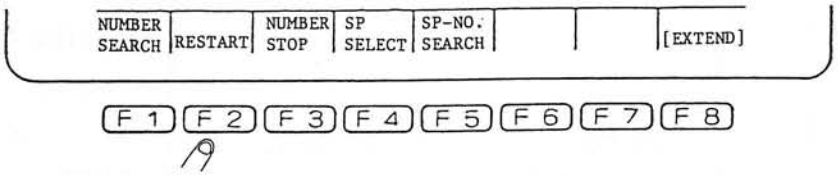
(2) Select the AUTO OPERATION mode by pressing the **AUTO** key.



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



(4) Press the function key [F2] (RESTART).



(5) Input the name of the sequence preceding the one in which the restart point is included and also the restart point designation option ";M∗". Input the name of axis which has been positioned in step 1).

(6) Press the **WRITE** key.

(7) Press the **CYCLE START** button.

Program is processed internally from the sequence one block preceding the designated sequence and when the actual position data of the designated axis matches the data of the axis position where the axis designated is actually located, the SLIDE HOLD lamp lights up and internal processing of the data completes.

This actual position is taken as the restart point.

(8) Press the **SEQ. RESTART** button.

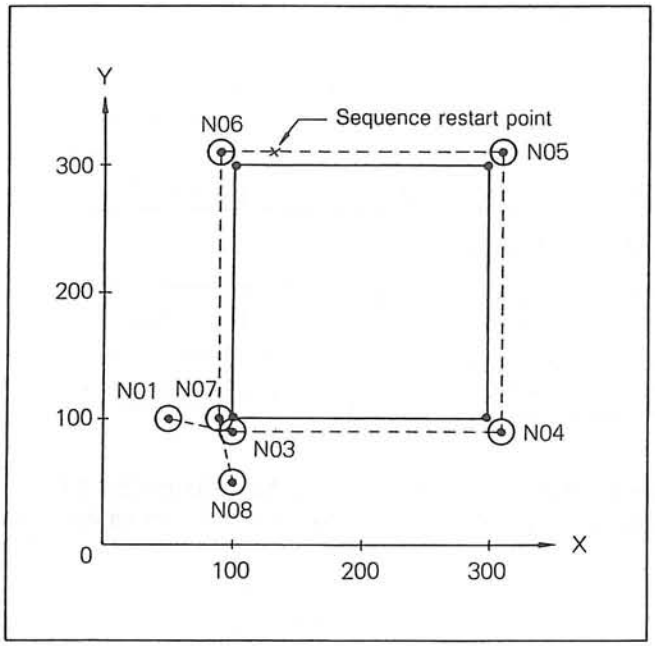
Positioning is carried out at the restart point in a manual cutting feedrate. After the completion of positioning, the control is placed in the slide hold state.

(9) Press the **CYCLE START** button.

Programmed operations are resumed.

Note: The restart point will be shifted a little in the tool advancing direction from the required restart point.

Example 1:



N01 G00 X100. Y50. S300 CR
N02 G56 Z-30. F200 CR
N03 G01 G41 Y100. D08 CR
(D08 = 10 mm)
N04 Y300. CR
N05 X300. CR
N06 Y100. CR
N07 X100. CR
N08 G40 X50. CR
N09 G00 G53 Z100. M05 CR
N10 M02 CR

The procedure to restart the sequence from X130. Y310. Z – 30. is explained below (The actually returned position is slightly ahead of the designated position.):

① Position X-axis at X130. point.

In this positioning operation, Y and Z axes may be located at any point.

② Input the command as indicated below.

RS␣N04;MX + Z **WRITE**

③ Press the **CYCLE START** button.

The program is processed internally from the commanded position (X90. Y100. Z – 30.) in sequence N03 and at a position a little shifted from the required restart point (X130. Y310. Z – 30.) in the tool advancing direction, the internal processing ends and the SLIDE HOLD lamp lights up.

The actual position reached in this process is taken as the restart point.

④ Press the **SEQ. RESTART** button.

Positioning of Z-axis occurs at the restart point in a manual cutting feedrate. After the completion of positioning, the RUN lamp begins flickering.

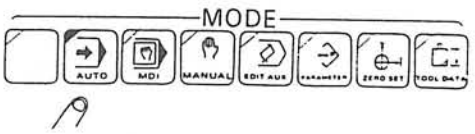
⑤ Press the **SEQ. RESTART** button again. X and Y axes are positioned at the restart point and after the completion of the positioning, the SLIDE HOLD lamp lights up.

⑥ Press the **CYCLE START** button.

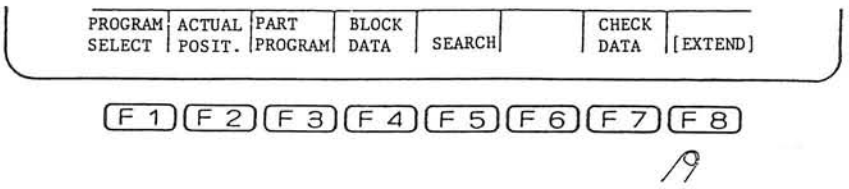
Programmed operations are resumed.

4-3. Sequence Restart from a Required Position in the Program

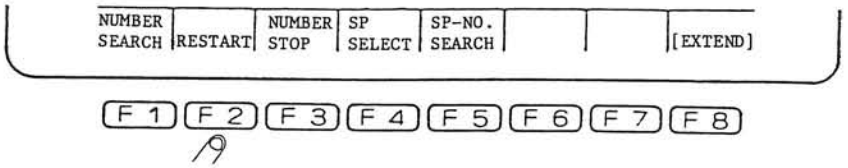
(1) Select the AUTO OPERATION mode by pressing the **AUTO** key.



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



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



(4) Input the name of a sequence before the one which contains the required restart point and the restart point designation option code ";TS".

(5) Press the **WRITE** key.

(6) Press the **CYCLE START** button.

Program is processed internally from the sequence one block preceding the designated sequence.

(7) When internal processing reaches the required restart point, press the **SLIDE HOLD** button. This point is taken as the sequence restart point.

(8) To change the sequence restart point press the **CYCLE START** button, and internal processing begins again. Press the **SLIDE HOLD** button when required point reached.

(9) Repeat pressing the **CYCLE START** and **SLIDE HOLD** buttons until the required restart point is reached.

(10) Press the **SEQ. RESTART** button.

The axes are positioned at the required sequence restart point in a manual cutting feedrate. After the completion of positioning, the control is placed in the slide hold state.

(11) Press the **CYCLE START** button.

Programmed operations are resumed.

5. Precautions

(1) Axis movement order option "+∗" and "−∗" cannot be specified at the same time.

(2) If shift amount by manual intervention operation differs from the previous cutting, sequence restarts from a point offset by this difference.

SECTION 2 MULTI-VOLUME OPERATION FUNCTION

1. Overview

If a program which cannot be saved in a sheet of IBM format floppy disk is to be handled, it will be necessary to save the program in two or more sheets of floppy disks. The function to operate the machine continuously using these floppy disks is called the "multi-volume operation function".

Multi-volume operation is available only for 8" floppy disk.

2. Multi-volume Operation Files

2-1. Multi-volume Identifier and Operation Order Numbers

If a part program is registered in more than one sheet of floppy disk, it is necessary to declare the execution order of the individual split files (to be referred to as the multi-volume file, hereinafter). For this purpose, identifier and execution order must be registered in the IBM format label information (to be explained in detail in Section 9) in addition to file name.

Main file name . . . . . Files split must be assigned with the identical file name.

Extension . . . . . Extension is not used and must be left in blanks ("␣␣").

Identifier . . . . . Multi-volume files must be assigned with character "C" or "L".
"C": Files other than the last file
"L": Last file

Execution order . . . . . Serial numbers indicating the order of file execution (01 - 99)

Example:

	File Name	Identifier	Execution Order Number
The first multi-volume file	MULTI.␣␣	C	1
The second multi-volume file	MULTI.␣␣	C	2
:		:	:
:		:	:
The "n – 1"th multi-volume file	MULTI.␣␣	C	n – 1
The last multi-volume file	MULTI.␣␣	L	n

2-2. Precautions in Splitting a Program

(1) Individual files must end at the end of a block.

Correct file registration example:

```
        :
G00Z250000LF
X – 130000Y – 280000LF
M01LF
                                  Files split here.
G90LF
G00Z250000LF
X0Y – 280000LF
G01Z215000LF
```

Wrong file registration example:

```
        :
X – 130000Y – 280000LF
M01LF
G
                                  Files split here.
90LF
G00Z250000LF
X0Y-280000LF
        :
```

(2) An alarm occurs, if files are split at a block where cutter radius compensation mode (G41, G42) is active.

```
        :                         Alarm 435
G42Y193000D50LF
G1X870000F500LF
                                  Files split here.
G0G40Y0LF
G53Z700000M5LF
        :
```

(3) If an M02 or M30 command is specified within a file assigned with identifier "C", alarm C (966) occurs if such file is executed.

(4) If an M02 or M30 command is not specified within the last file which is assigned with identifier "L", alarm B (435) occurs if such file is executed.

(5) During the execution of different files in succession, machine operation stops due to the automatic program selection processing for the next file to be executed. Therefore, it is necessary to avoid splitting files while a continuous cutting is in progress. Retract the axes to a fixed position by specifying the G30 command (return to home position) before splitting files, for instance.

3. Device

(1) Sector device is available for 8" or 3.5" floppy disk depending on the combination of specification.

Combination of Specification / Servo Device	Only 8" floppy disk is selected	Both 3.5" and 8" floppy disks are selected
FD0:	○	○ 3.5" floppy disk
FD1:		○ 8" floppy disk

FD0: and or FD1: is indicated on the floppy unit for built-in type floppy disk drive, and on the floppy interface for the portable floppy drive.

(2) Multi-volume operation is available only for 8" floppy disk.

4. Multi-volume Operation Procedure

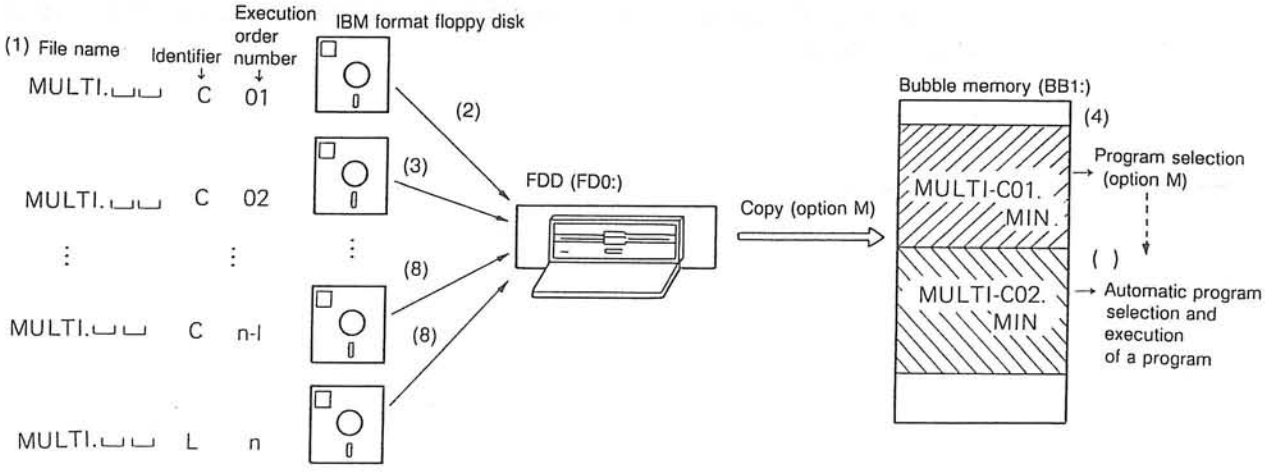


Fig.2-1 Automatic Program Selection and Execution of a Program

(1) Prepare floppy disks on which identifier, execution order number and identical file name are registered in accordance with the instructions given in 2., "Multi-volume Operation Files".

(2) The operator should copy the first multi-volume file into the user area bubble memory (BB1:) by designating the option M (;M) after selecting the transfer mode of the program operation. For details of the program copy operation procedure, refer to 5-1., "Copying Multi-volume Files".

(3) Copy the second multi-volume file in the same manner as with the first multi-volume file. If the bubble memory still has the capacity to store the third multi-volume file, register it as much as possible.

(4) Switch the operation mode to the automatic operation. Then, select the first multi-volume file, which has been registered in step (2) above, by specifying the option M (;M). For details of the program selection procedure, refer to 6., "Program Selection for Multi-volume Files".

Make sure that message "M-Mtd" is indicated at the lower left corner on the CRT screen, telling the operator that the control is in the multi-volume operation mode.

(5) Press the **CYCLE START** button to start the multi-volume operation.

(6) After the completion of the execution of the first file, the control automatically selects the next file to be executed and the operation is continued using the next file.

(7) Now, the operator should switch the operation mode from the automatic to the program operation and delete the first file and copy the file in succession in accordance with the execution order number by designating the option M.

The function key [F7] (RUN GUIDE) in the program operation mode is used to display the file execution status and thus the operator can confirm the files which have been executed. Details of the operation guide display are explained in 7., "Screen Display of Run Guide".

(8) Repeat the step (7) above until the last file is copied into the bubble memory.

(9) The operation completes when the execution of the last multi-volume is completed.

*Note: On the main program display screen, message **"RQNFL"** will be added to the last block of the multi-volume file. This is an instruction to automatically select the multi-volume file to be executed next. Therefore, this instruction word must not be used within a part program.*

5. Copying Multi-volume Files

5-1. Copying from Floppy Disk to Bubble Memory

(1) Multi-volume files in the IBM format floppy disk can be transferred to the bubble memory using the copy function in the PIP mode of program operation.

Example: >CO FD1:MULTI ;M **WRITE**

Option function to designate multi-volume file

(2) By the commands indicated above, the file name registered in the bubble memory is appended with identifier, execution order number and extension MIN.

MULTI – C01.MIN

Execution Order Number 01 - 99
(Refer to format file label information.)

Identifier (C or L)

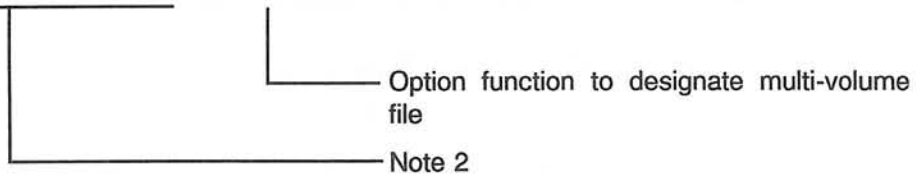
Note 1: Copy operation using the option M automatically determines the output file name and therefore, the operator cannot designate the output file name.

Note 2: Copy operation using the option M requires the input main file name (file name in the IBM format floppy disk) to be limited within six characters. Extension must be spaces ("␣␣").

5-2. Copying from Bubble Memory to Floppy Disk

(1) Multi-volume files in the bubble memory can be transferred to the IBM format floppy disk using the copy function in the PIP mode of program operation.

Example: >CO MULTI -C02.MIN, FD1:;M **WRITE**

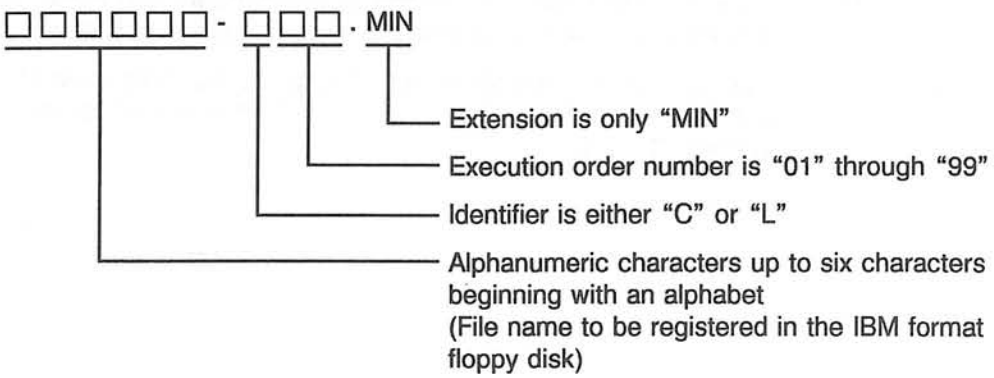


(2) The commands indicated above designate the file name, identifier, execution order number to be registered in the IBM format floppy disk as indicated below.

File name	:	MULTI.␣␣
Identifier	:	C
Execution order number	:	02

Note 1: *Copy operation using the option M automatically determines the output file name and therefore, the operator cannot designate the output file name.*

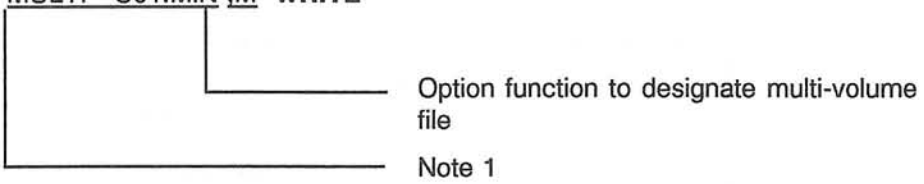
Note 2: *The format of the input file name to be designated for the copy using the option M is as indicated below and designation other than the format indicated below will cause an error (2605).*



6. Program Selection for Multi-volume Files

(1) The first multi-volume file is selected by the operator by specifying the option M code (;M).

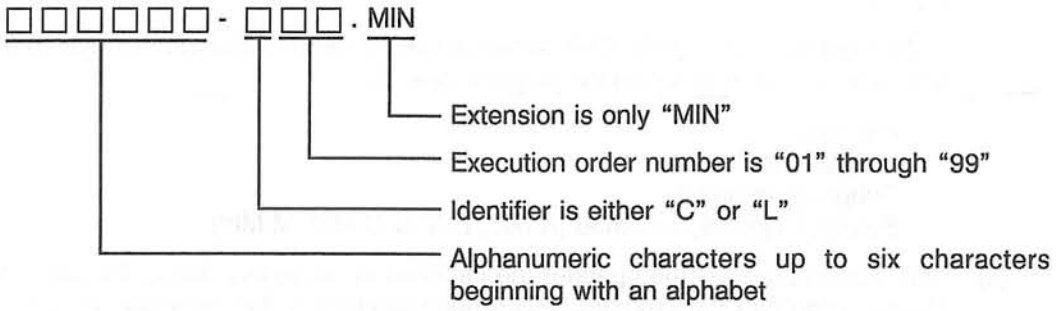
Example: =PS MULTI - C01.MIN ;M **WRITE**



(2) With the designation of option M code, the multi-volume files are automatically selected sequentially each time the operation of one file is completed.

MULTI-C01	.MIN	(selected by operator by option M)
↓		
MULTI-C02	.MIN	(automatic selection)
⋮		⋮
↓		⋮
MULTI-Cn – 1	.MIN	(automatic selection)
MULTI-Ln	.MIN	(automatic selection of the last file)

Note 1: The format of the file name to be designated for the program selection using the option M is as indicated below and designation other than the format indicated below will cause an error (2605).



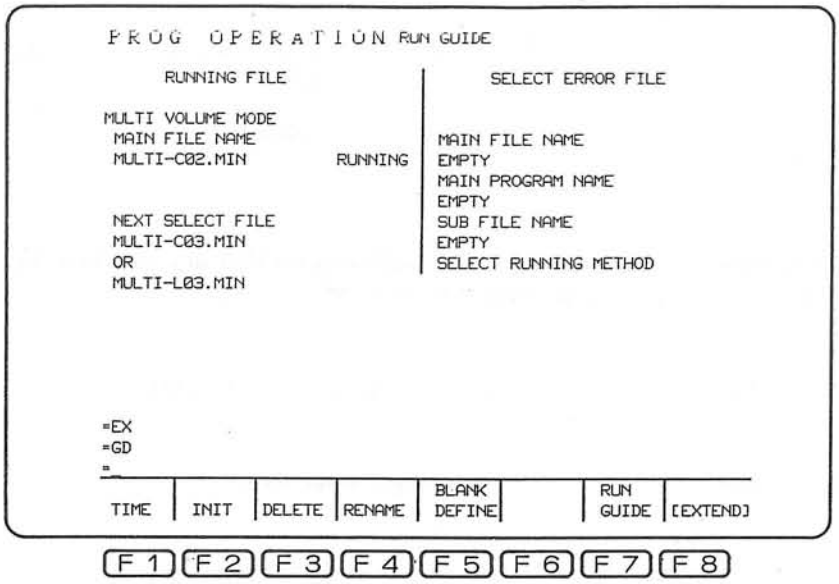
Note 2: The execution order number of the multi-volume file to be automatically selected is incremented continuously.

Note 3: The use of IF and GOTO statements is not permitted when option M is designated. If subprograms are to be called out, register them in the library in advance.

Note 4: If "3" is set at optional parameter (word) No. 11, designation of option M (;M) may be omitted.

7. Screen Display of Run Guide

(1) Pressing the function key [F7] (RUN GUIDE) in the program operation mode displays the RUN GUIDE screen indicated below.



(2) Information at the left half area on the screen titled RUNNING FILE indicates the following:

- Contents of present operation mode (1)
- The name of the selected file
- Operation status of the selected file (SELECTED, RUNNING, END)

(3) In the case of multi-volume operation mode, the name of the file to be executed next is displayed.

In the right half area on the CRT screen displayed is the following information at an occurrence of selection error during automatic program selection:

File name
Program name
Subprogram name
Selected operation method (A-Mtd, B-Mtd, S-Mtd, M-Mtd)

(4) This screen provides the operator the information necessary during the execution of multi-volume file operation and the program having been selected if the automatic program selection was not successful.

Note that the CRT screen displays the information of the related status when the function key [F7] is pressed and it is not updated even when the file operation status is changed. To display the latest operation status, press the function key [F7] again.

8. Parameter Setting

8-1. NC Optional Parameter (WORD) Data No. 11

This sets the default for program selection.

NC Optional Parameter (Word) No.	Contents
11	The operation method selected when option specification is not made for program selection 0: Normal operation (A method) 1: Large volume operation (B method) 2: Operations without branching and subprograms (S method) 3: Multi-volume operation (M method)

9. Error and Alarm Message

9-1. Error Message

The error related with the multi-volume operation specification is indicated below:

2605 Multi volume file information

In an attempt to copy or select a multi-volume operation file with the option M specification, the following designation is wrong.

Code	:	0	Main file name format is wrong.
		1	Extension other than "MIN" (for bubble memory) or "␣␣" (for IBM format floppy disk) is specified.
		2	A character other than "C" or "L" is specified as an identifier.
		3	The execution order number is other than "01" through "99"
		4	Output file is designated in copy operation.
		5	The floppy disk prepared is not formatted to the IBM format when copy operation is attempted.

9-2. Alarm Message

The alarm related with the multi-volume operation specification is indicated below:

(1) ALARM B

613 Multi volume next program load

Failure in the selection of the file to be executed next in the multi-volume operation.

Code : 10000002 ..... The file of the main program to be selected next is not registered in the bubble memory.

10000100 ..... The number of the sequence names in the selected program exceeds 31.

10000200 ..... The number of characters used in a sequence name in the selected program exceeds five including character N.

10004000 ..... The program selected contains a block which has more than 156 characters in it.

10400000 ..... Identifier of the multi-volume file is not found within eight characters in the main file name.

14000000 ..... The execution order number of the multi-volume file is greater than 99.

20000000 ..... Contents of the bubble memory cannot be read out due to bubble memory failure.

614 BFR task program load

An error other than program selection processing occurs within the BFR task.

Code : None

(2) ALARM C

966 Multi volume end code

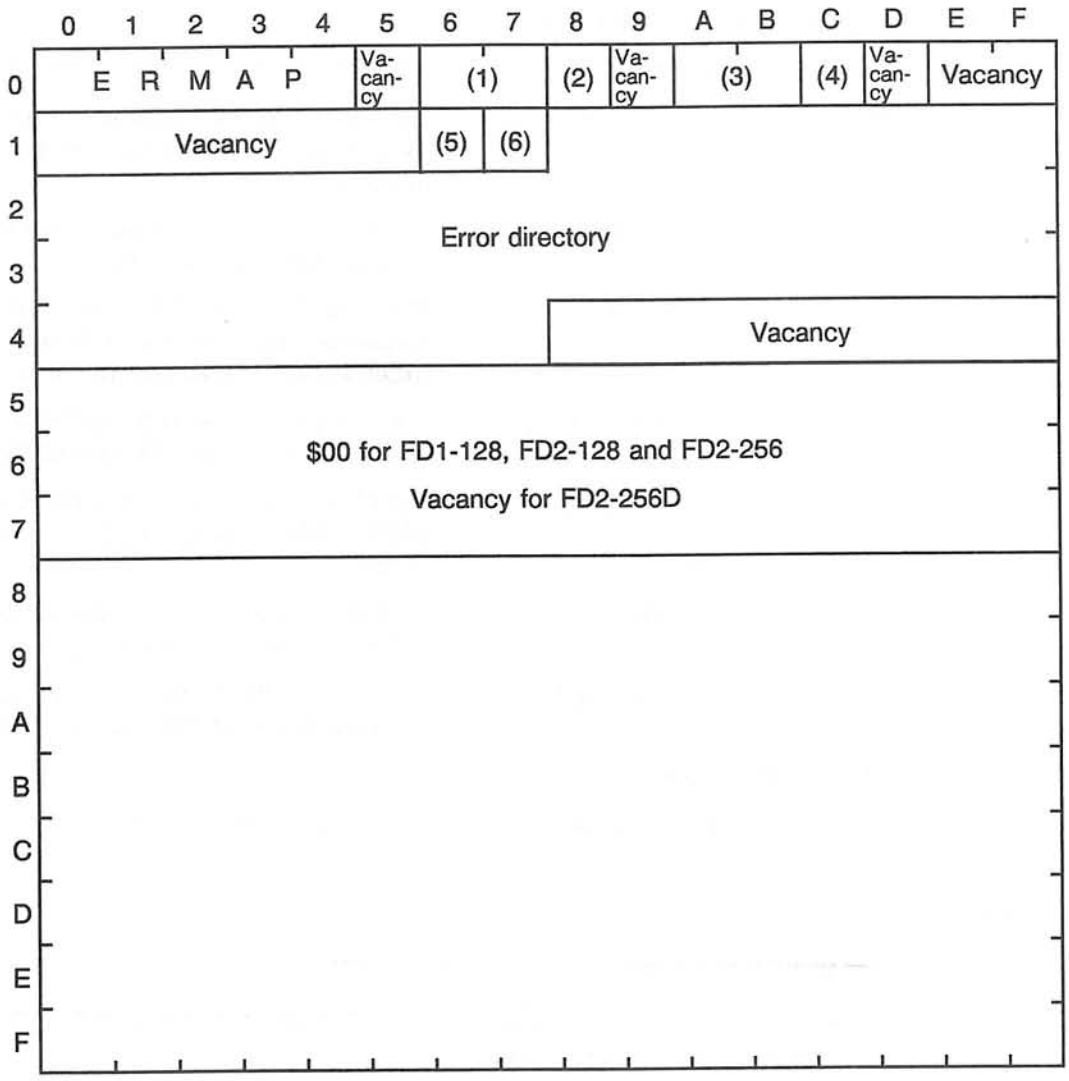
During the multi-volume operation, an end code (M02, M30) is written in a file* in which such a code is not permitted.

* File containing identifier "C"

Code : 1 ..... The file assigned with the identifier "C" contains an M02 or M30 command.

10. Appendixes

10-1. IBM Format Error Map (ERMAP)



(1) First error cylinder number

(2) Presence of the first error cylinder

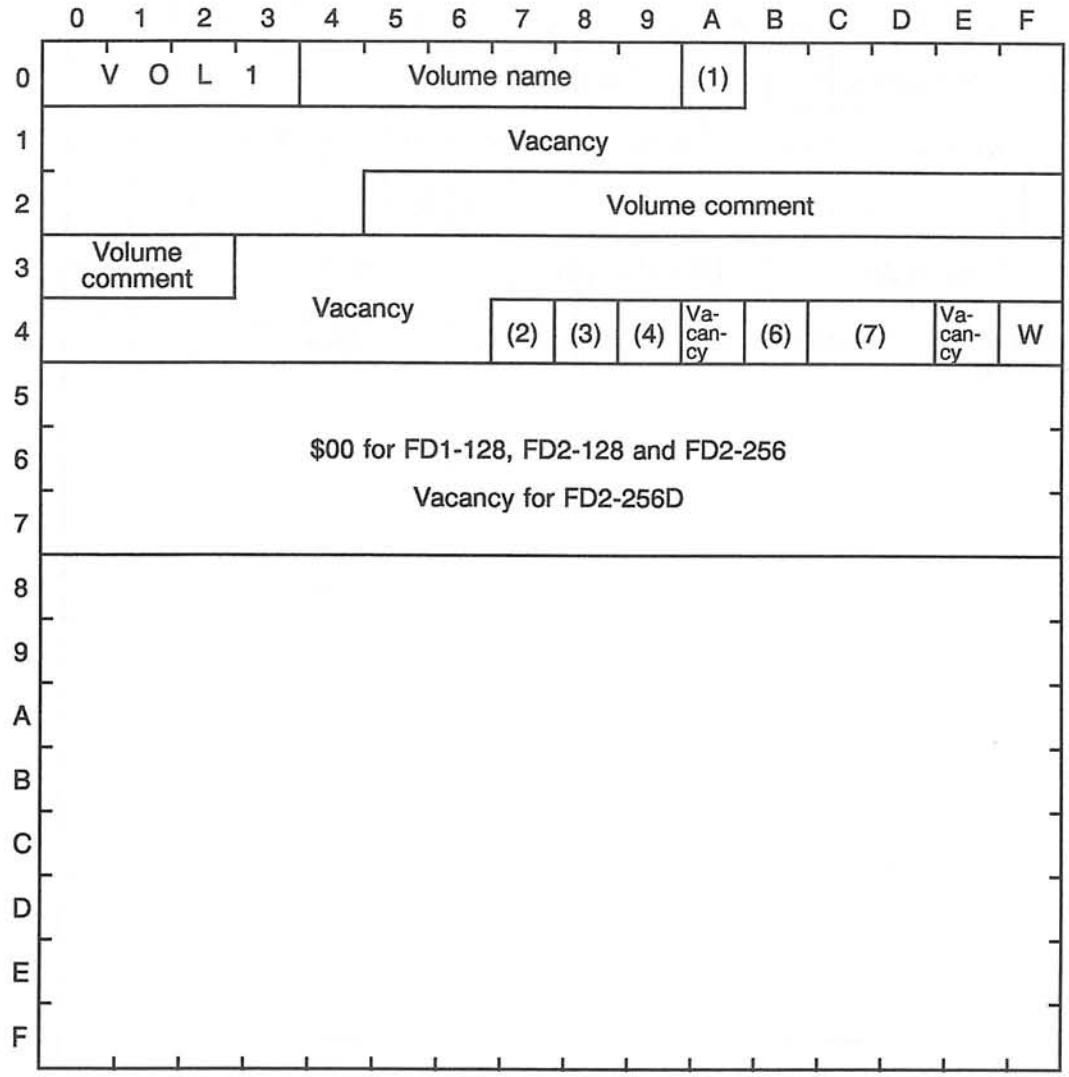
(3) Second error cylinder number

(4) Presence of the second error cylinder

(5) Fault indicator

(6) Error directory indicator

10-2. IBM Format Volume Label Information (VID)



(1) Access condition

(2) Recording face identifier

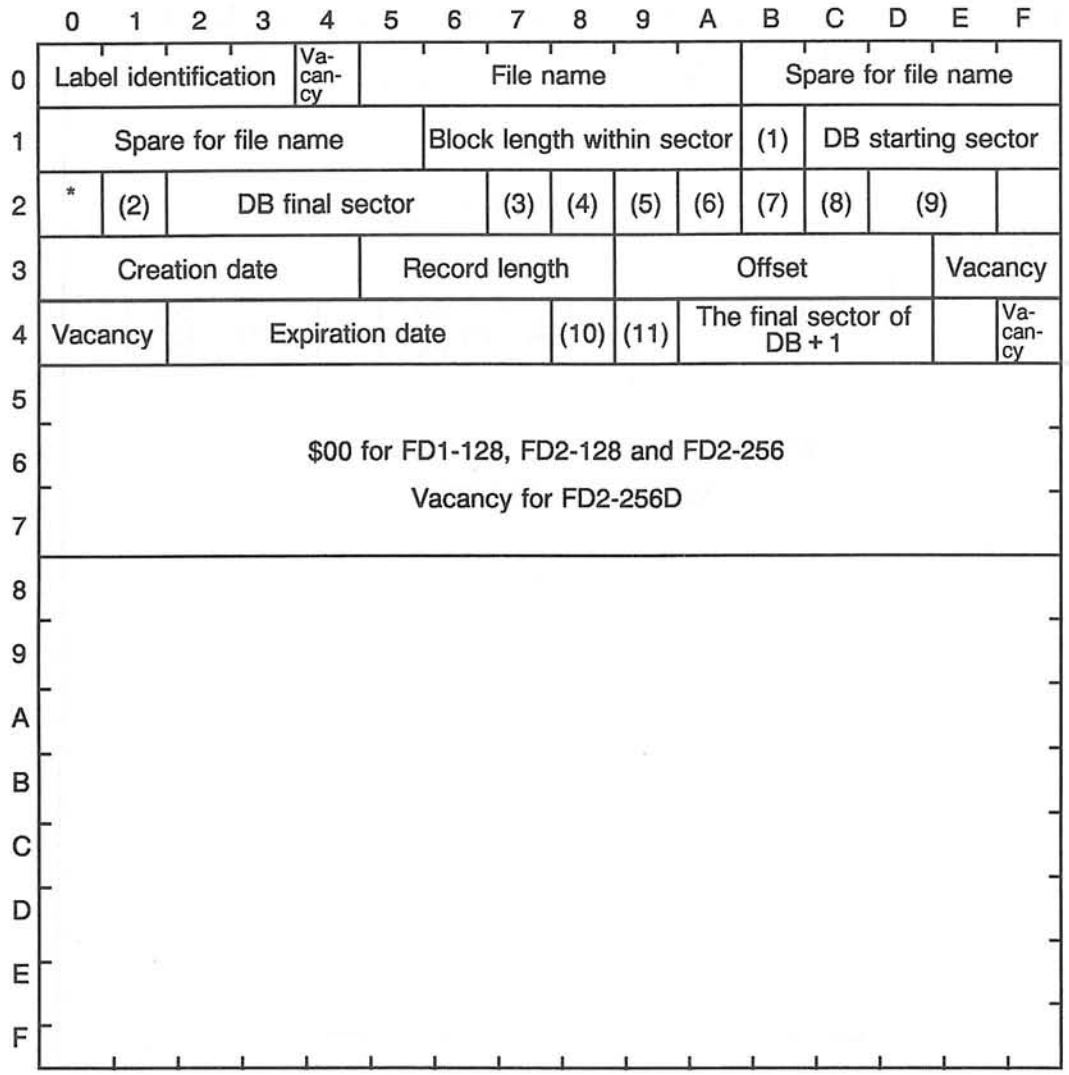
(3) List

(4) Access method

(5) Physical record length

(6) Physical record number

10-3. IBM Format File Label Information (HDR1)



* DB starting sector

(1) Record attribute
(2) Physical record length
(3) Format
(4) Bypass identifier
(5) Security
(6) Protect
(7) Exchange level
(8) Multi-volume identifier (C or L)
(9) Multi-volume execution order number (00 - 99)
(10) Verification
(11) File order

SECTION 3 DNC-A

1. Functions

DNC-A for the OKUMA OSP5020M/OSP500M-G controls the transmission of NC programs and data between a CNC machine and the host processor through an RS-232C interface. It provides the following two major functions:

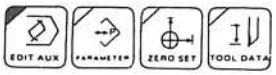
(1) Downloading [INPUT] ..... Receives NC programs (or files) from the host computer. The CNC machine can also verify a received NC program (file) by matching the received NC program (file) stored in its bubble memory with the NC program (file) that is sent from the host computer for verification.

(2) Uploading [OUTPUT] ..... Uploads NC programs (or files) to the host computer. The CNC machine can retransmit the NC programs (files) so that they can be verified at the host computer.

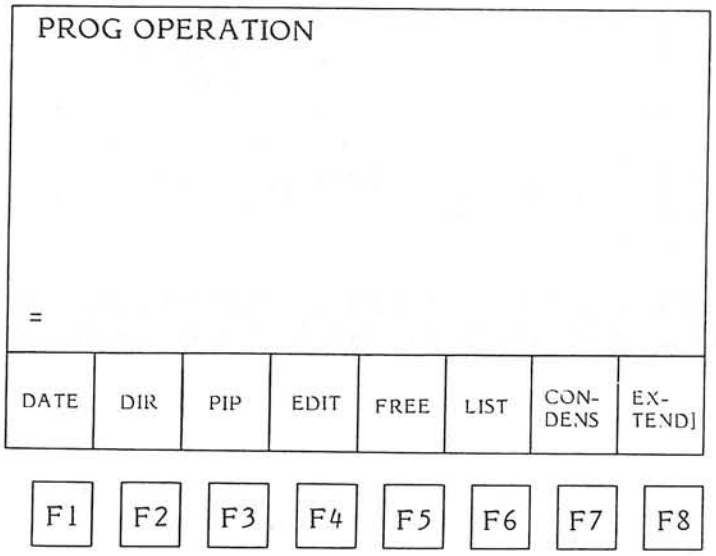
2. Operating Procedures

2-1. Operating Procedure for DNC

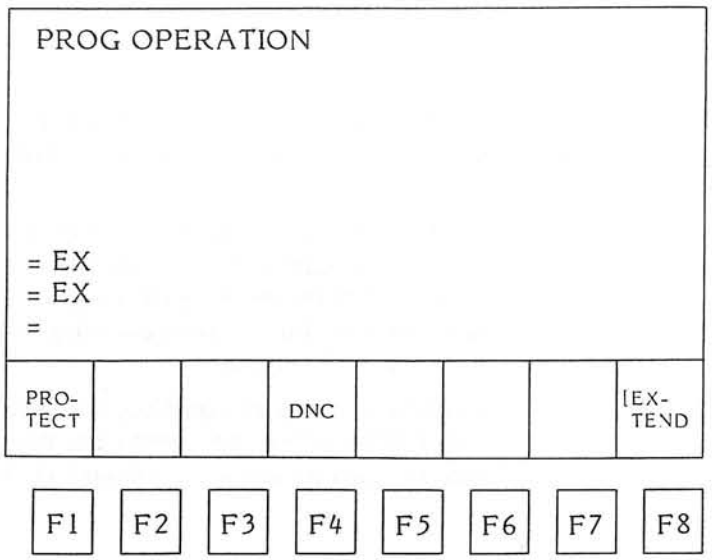
(1) Press the **EDIT AUX** key.



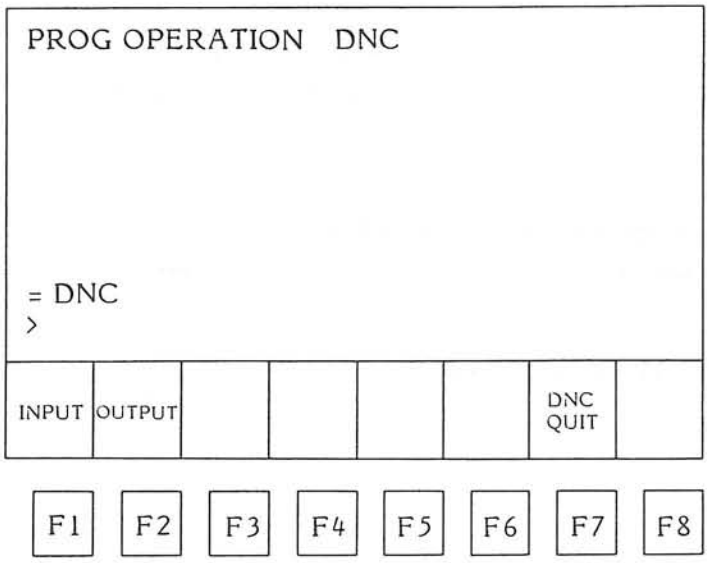
(2) The display shown below will appear on the CRT.



(3) Press the function key [F8] (EXTEND) twice to cause "DNC" to be displayed in the function key guide area corresponding to the [F4] key.



(4) Press the function key [F4] (DNC), and the display shown below will appear.



(5) Pressing the function keys [F1] and [F2] at this moment will enable input and output processing, respectively. Press the function key [F7] (DNC QUIT) to quit the DNC mode.

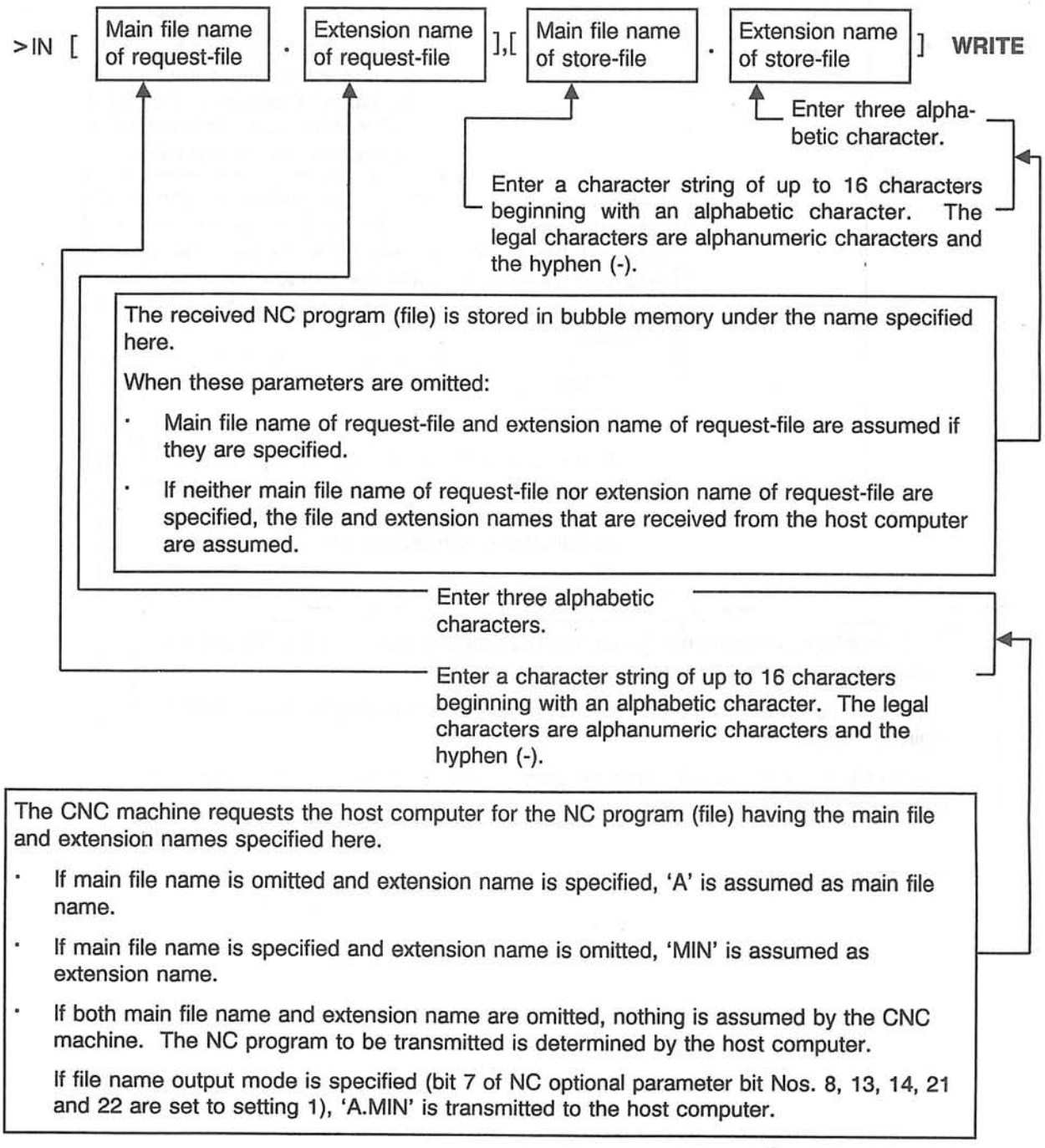
2-2. Operation Function in the DNC Mode

You can perform two types of functions in the DNC mode as explained below.

(1) Downloading an NC Program (File)

- Press the function key [F1] (Input).
- Enter the name of the NC program (file) to be downloaded.
- Press the **WRITE** key.

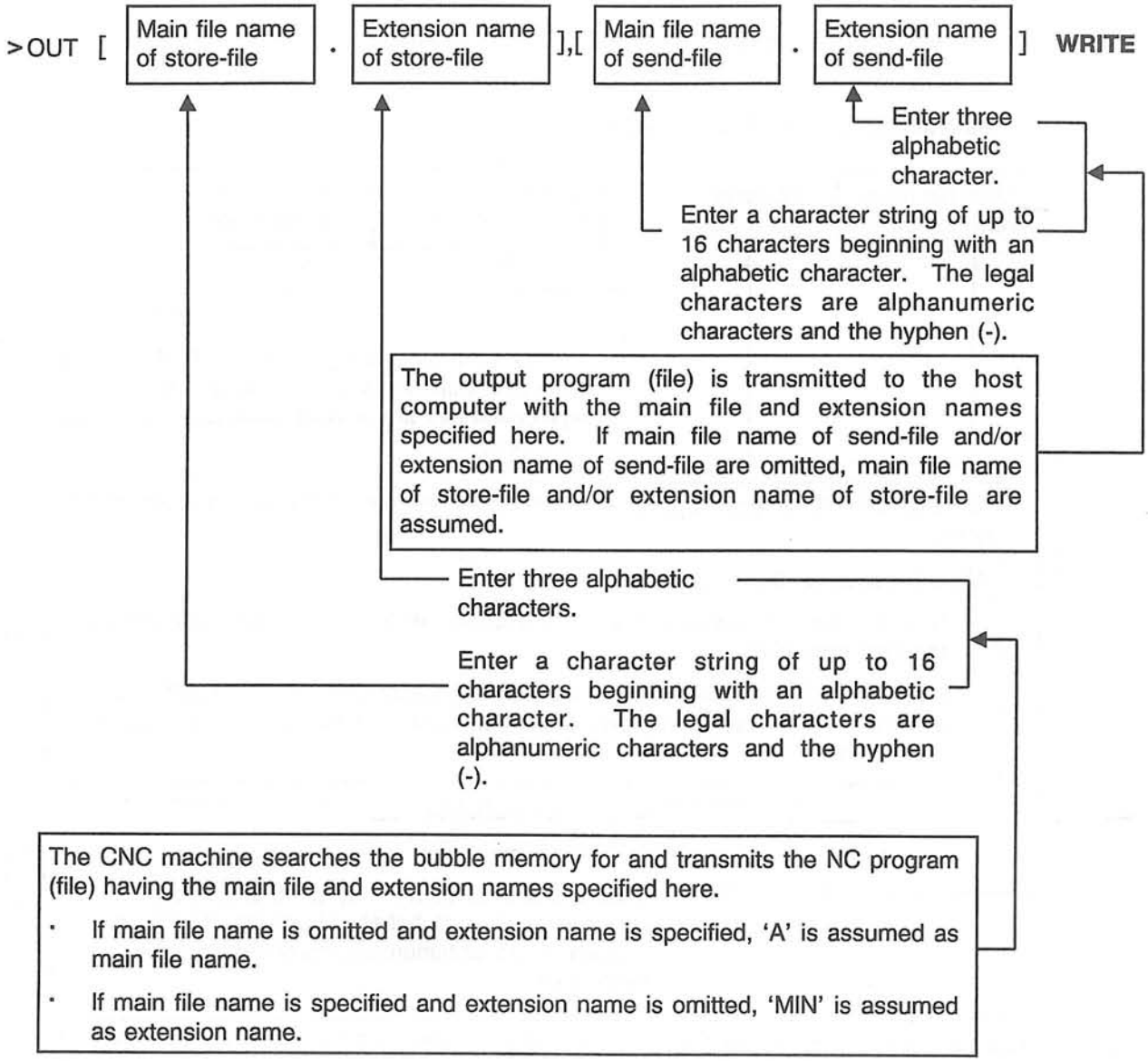
The general format of the IN command is shown below.



(2) Uploading an NC Program (File)

- Press the function key [F2] (Out put).
- Enter the name of the NC program (file) to be uploaded.
- Press the **WRITE** key.

The general format of the OUT command is shown below.



3. Setting Up NC Optional Parameters

(1) NC Optional Parameter (Bit) No. 10

The NC Optional parameter bit No. 10 specifies whether the verify data is to be transmitted in the upload mode and whether the NC program (file) name is to be transmitted.

	Bit 5	Bit 4	Mode
A	1	1	Verify data transmitted on output, no NC program (file) name transmitted
B	1	0	Verify data transmitted on output, NC program (file) name transmitted
C	0	1	No verify data transmitted on output, no NC program (file) name transmitted
D	0	0	No verify data transmitted on output, NC program (file) name transmitted

(2) NC Optional Parameter (Word) No. 43

The NC optional parameter word No. 43 selects the DNC-A input/output channel.

Parameter value	Channel
0	CN0: (TT:)
1	CN1:
2	CN2:
3	CN3:
4	CN4:

(3) NC Optional Parameter (Bit) No. 1

The NC optional parameter bit No. 1 defines the tape delimiter code.

	Bit 3	Mode
A	1	% (A5 in hex)
B	0	NUL (00 in hex)

NC optional parameters described in (1) through (3) are enabled immediately whenever they are reset; the CNC machine need not be switched off and on.

(4) NC Optional Parameter (Bit) Nos. 8, 13, 14, 21, 22

The NC optional parameter bit data Nos. 8, 13, 14, 21, 22 specify whether the request file name is to be transmitted from the host computer in the download mode.

	Bit 7	Mode
A	1	Request file name transmitted on input.
B	0	No request file name transmitted on input.

(5) RS-232C Interface Parameters for CN0: (TT:) through CN4:

Refer to Section 3., "Tape Puncher Interface" in Special Functions Manual (No. 1) (Publication No. 3294-E) for the parameter settings of the RS-232C interface parameters for channels CN0: (TT:) through CN4:

4. Error Messages

Two types of errors may occur in the DNC mode.

2557 DNC device

An invalid device (other than CN0: (TT:) through CN4:) was specified for DNC-A or a device name was specified as the request-file or transmit file name for DNC-C.

Character-string	:	None	An invalid device (other than CN0: (TT:) through CN4:) was specified for DNC-A.
Code	:	1	A device name was specified as the request-file or transmit file name for DNC-C.

2558 DNC verify

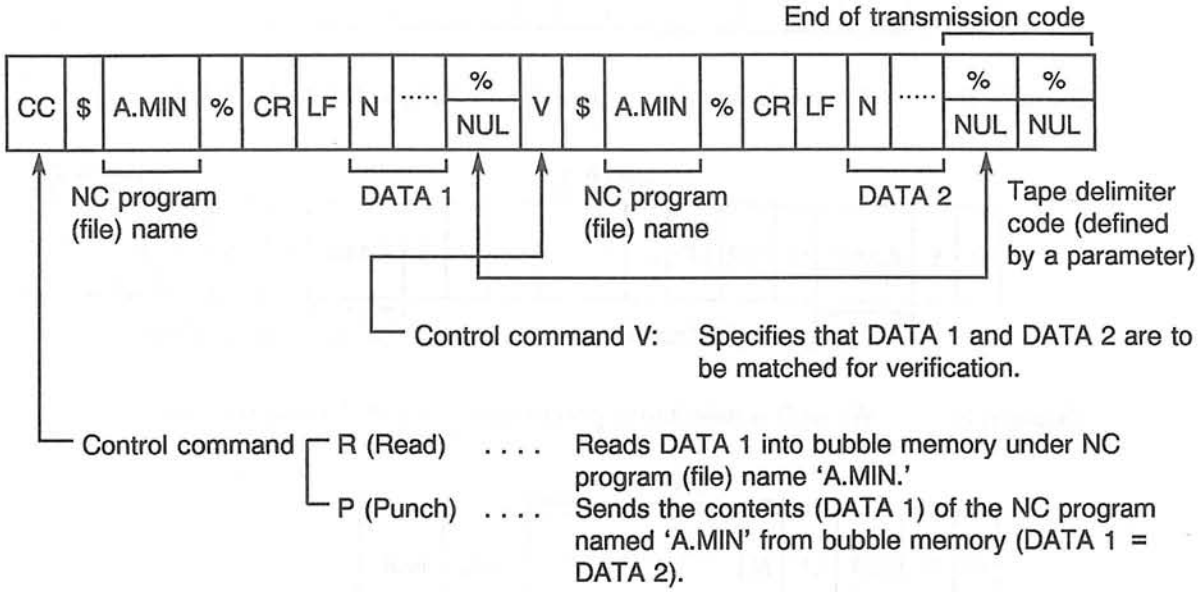
A verify error was found during DNC verify processing

Character-string	:	None	
Code	:	1	The file is longer than the verify data.
		2	The file is shorter than the verify data.
		Others	Indicates the number of mismatching characters.

5. Specifications

5-1. Data Formats

Data is transferred all in ISO code. The end of data is identified by two consecutive NUL (00) or % (A5 in hex) codes.



(1) When transmission of verify data is not specified, the control command V and the subsequent data are not transmitted; instead, two consecutive % or NUL codes are transmitted and transmission terminates (see Examples 3 and 4 in (1)).

(2) When transmission of the NC program (file) name is not specified, the NC program (file name following the control command is not transmitted (see Examples 1 and 3 in (1)).

(3) The NC program (file) name may be omitted in the download mode. In this case, the program (file) name 'A.MIN' is assumed if it is not specified on the CNC machine side (see Example 2 in (2)).

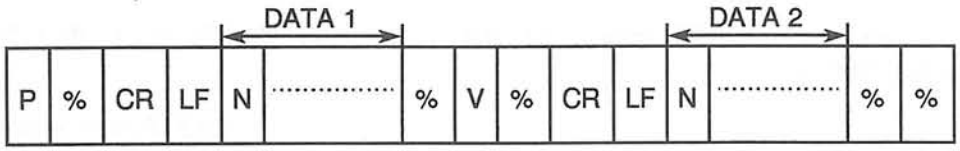
(4) Either % or NUL code can be defined in a parameter as the tape delimiter code. The host computer must use the defined tape delimiter code when sending a NC program (file) to the CNC machine.

(5) The host computer need not transmit the V control command and subsequent data in the download mode if verify processing is not required. In this case, the host computer must send two consecutive % (or NUL) codes instead of the V control command (see Example 2 in (2)).

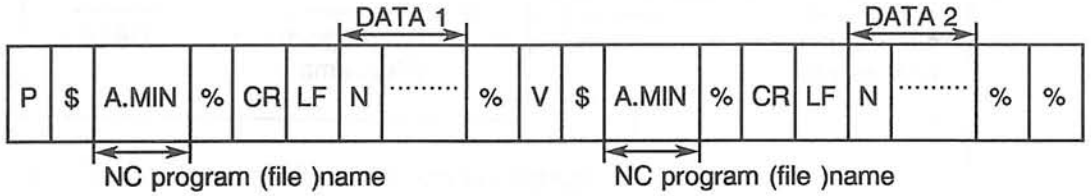
(6) Only the data between the LF code and the tape delimiter code (i.e., DATA 1) is stored in bubble memory.

5-1-1. Output Data Format Examples

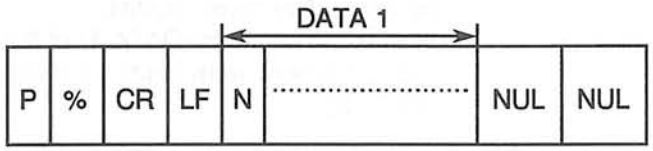
Example 1: When A is selected in parameter settings in 3-1 and 3-3



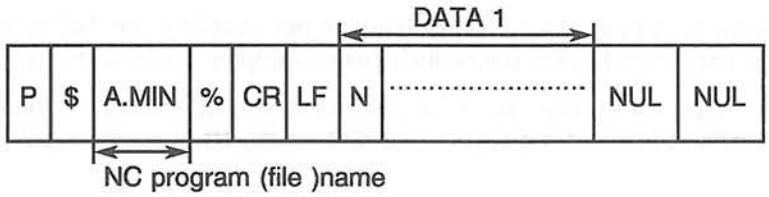
Example 2: When B is selected in parameter setting in 3-1 and A in 3-3



Example 3: When C is selected in parameter setting in 3-1 and B in 3-3

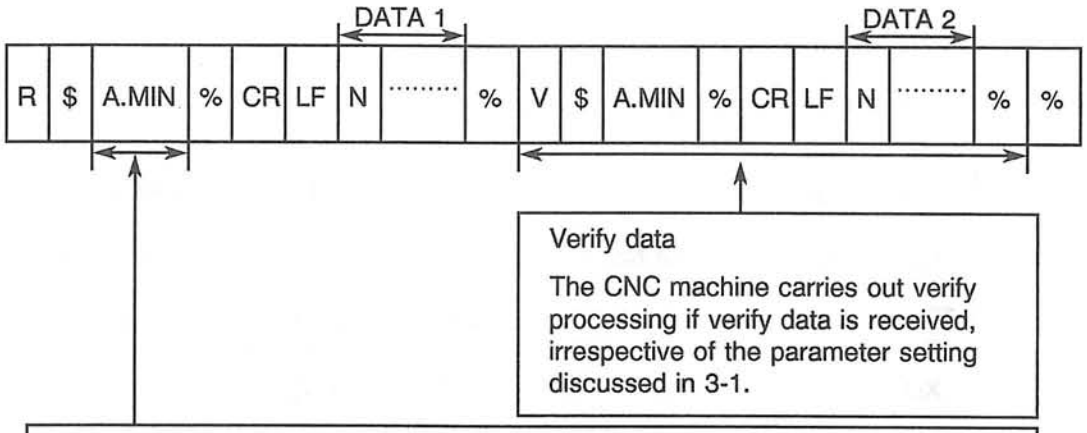


Example 4: When D is selected in parameter setting in 3-1 and B in 3-3



5-1-2. Input Data Format Examples

Example 1: When A is selected in parameter setting in 3-3 and verify data is present



Verify data

The CNC machine carries out verify processing if verify data is received, irrespective of the parameter setting discussed in 3-1.

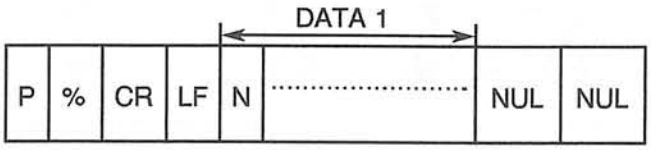
NC program (file) name

(a) This file name is verified if the request file name is specified.

(b) This file name is not verified and used as is if the request file name is omitted.

(c) This file name is assigned to the file stored in bubble memory if both request file name and store file name are omitted.

Example 2: When A is selected in parameter setting in 3-3 and no verify data is present



When the NC program (file) name is not present

(a) If the store file name is specified, the received data is stored in bubble memory under that name.

(b) If the request file name is specified and the store file name is omitted, the received data is stored under the request file name.

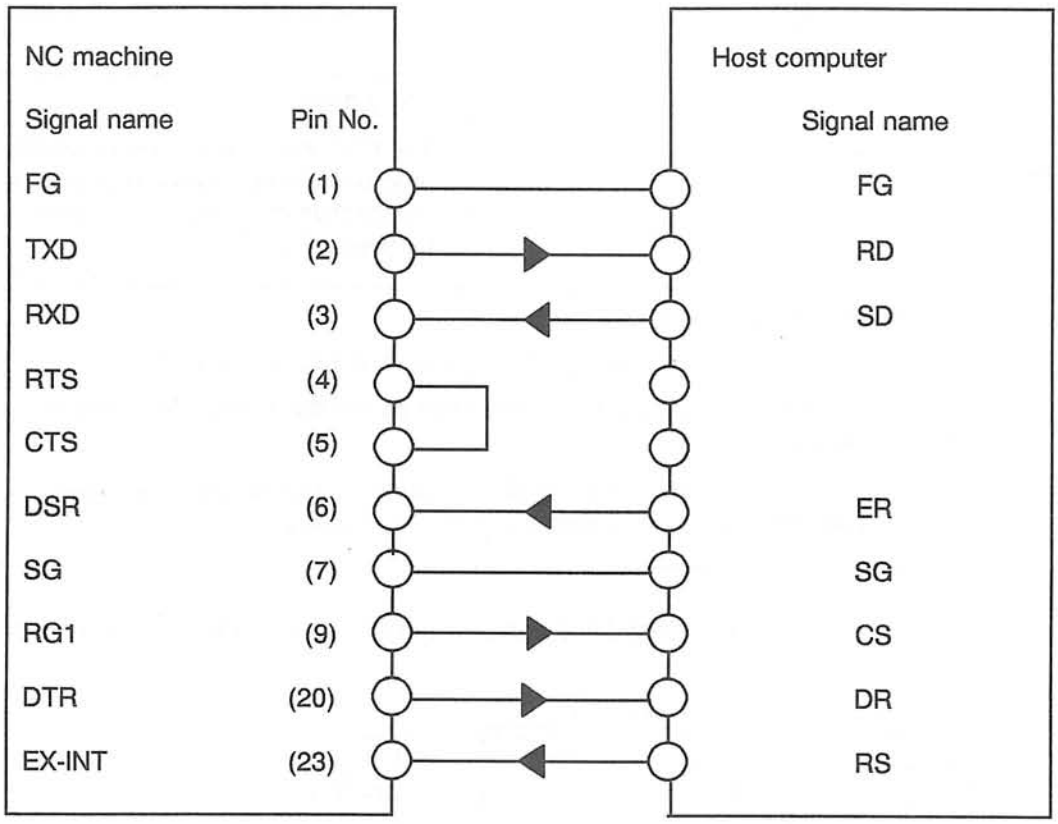
(c) If both the request and store file names are omitted, 'A.MIN' is assumed as the name of the received file.

5-2. Example RS-232C Interface Connections and Timing Charts

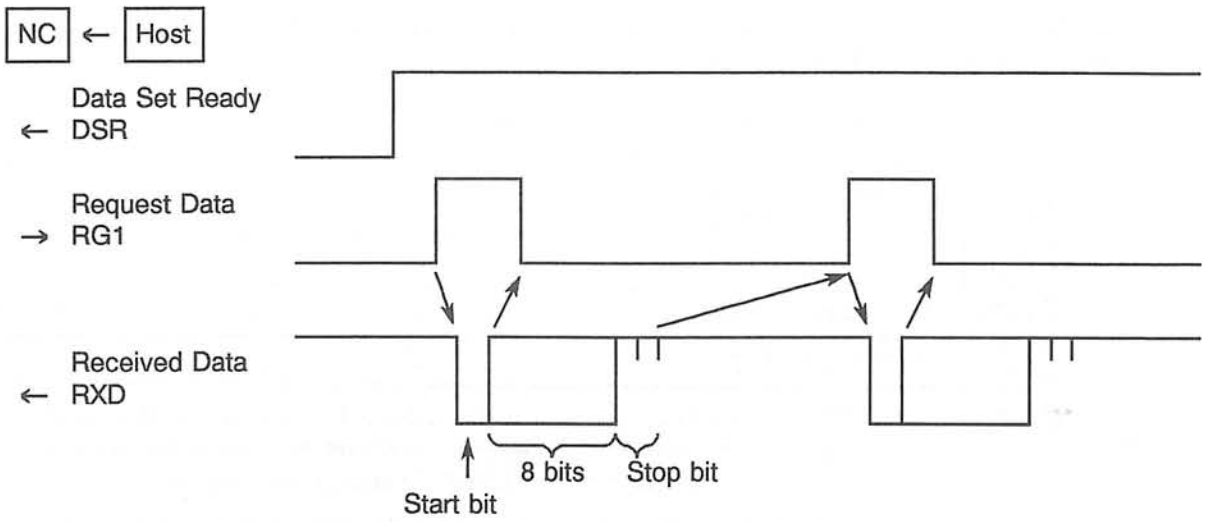
Since the CNC machine and the host computer are likely to use different RS-232C interface lines, it is necessary to check the RS-232C interface lines used in the host computer system and use an appropriate, dedicated RS-232C cable.

5-2-1. Start/Stop Synchronization Mode

(1) Sample Connection Diagram:



(a) Download mode timing chart:

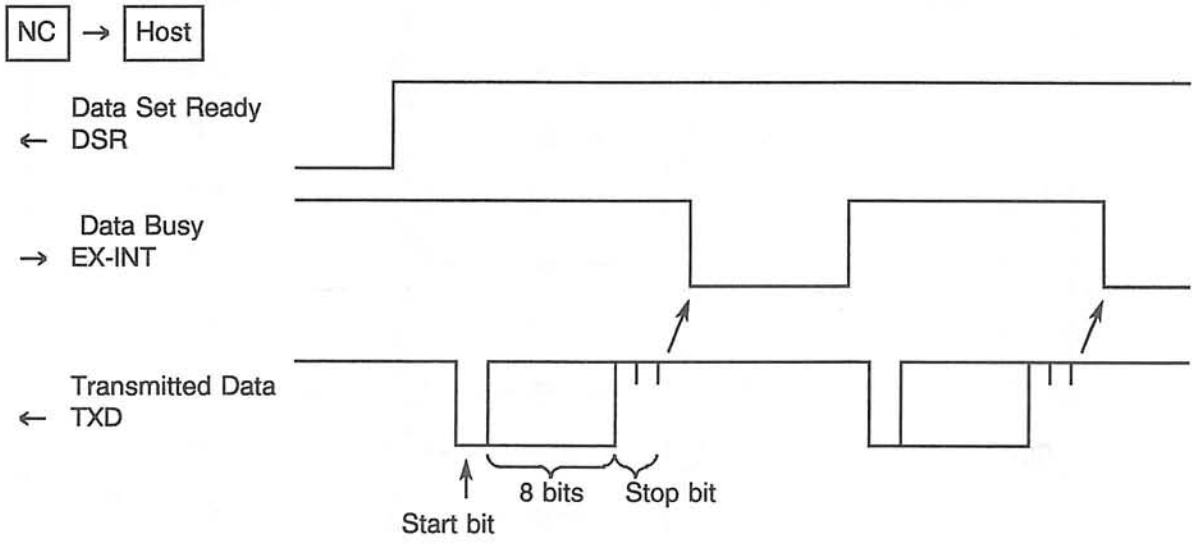


① The CNC machine sends the Data Request signal RG1.

② The host computer, on receiving RG1, starts transmitting a byte of serial data.

③ The Data Request signal is reset on the start bit of the serial data byte.

(b) Upload mode timing chart:



① The CNC machine sends a byte of transmitted data when the Data Busy signal EX-INT is high.

② The host computer resets the Data Busy signal when it reads the stop bit of the received serial data byte. The Data Busy signal must be reset each time a serial data byte is received.

③ The host computer sets the Data Busy signal when it completes processing of the received data byte.

Note: *Since this example configuration uses the EX-INT signal, NC optional parameter bit for the "no EX-INT signal" mode (bit 1 of NC optional parameter bit Nos. 8, 13, 14, 21, 22) must be set to "0".*

5-2-2. DC Code Control

The DC (Device Control) codes are defined as follows irrespective of the code of the NC program data to be transmitted (ISO or EIA):

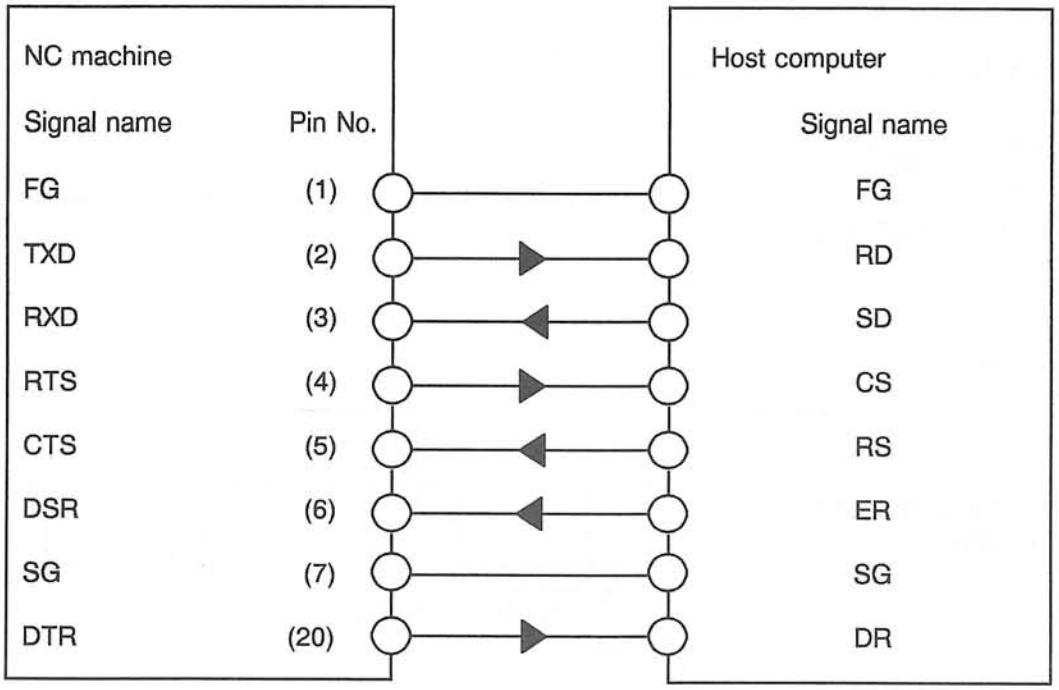
Character		8	7	6	5	4	3	2	1
DC1	Tape Reader Start				○	o			○
DC2	Tape Punch Start				○	o		○	
DC3	Tape Reader Stop	○			○	o		○	○
DC4	Tape Punch Stop				○	o	○		

Note 1: *The RS-232C interface parameters must be set to use DC codes (refer to Section 3,"Tape Puncher Interface" in Special Functions Manual (Publication No. 3294-E) for the parameter settings of the RS-232C interface parameters).*

Note 2: *DC codes are automatically generated by the NC machine. The program need not consider the DC codes.*

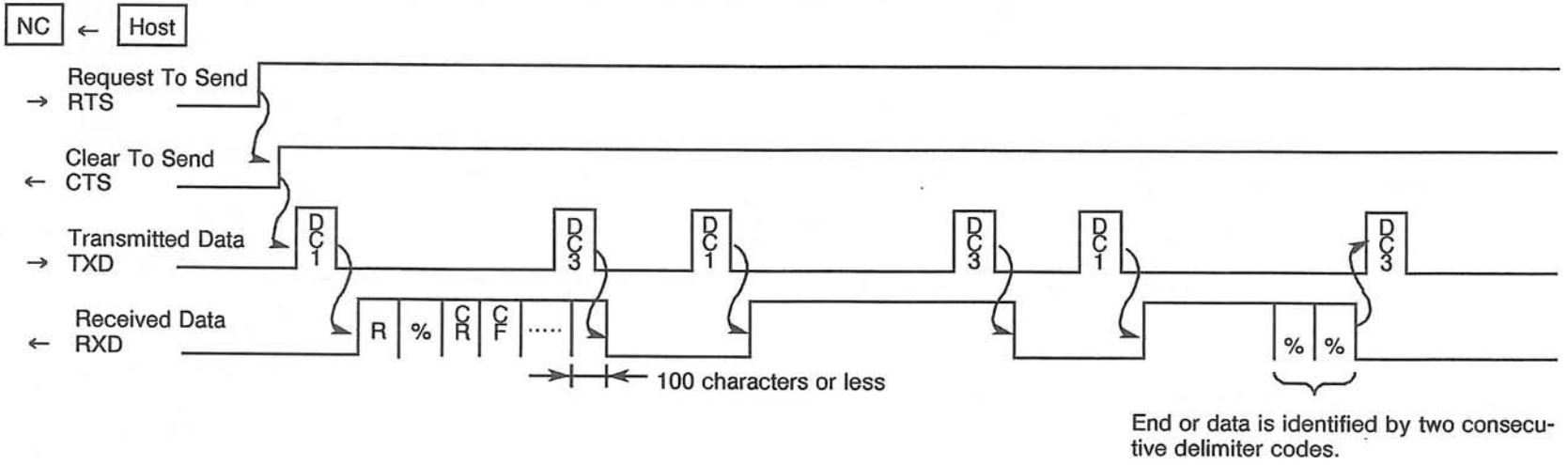
Sample Connection Diagrams

(1) Example 1:

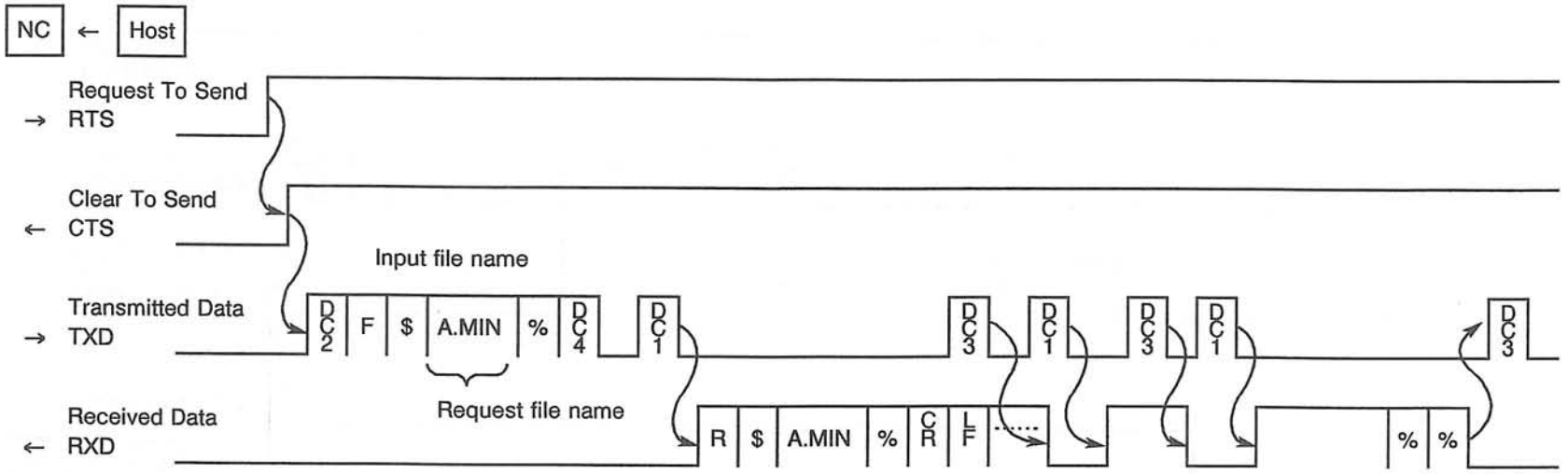


(a) Download mode timing charts

1) When request-file name output mode is not specified (NC optional parameter bit Nos. 8, 13, 14, 21, 22 in 3-4 set to setting B)

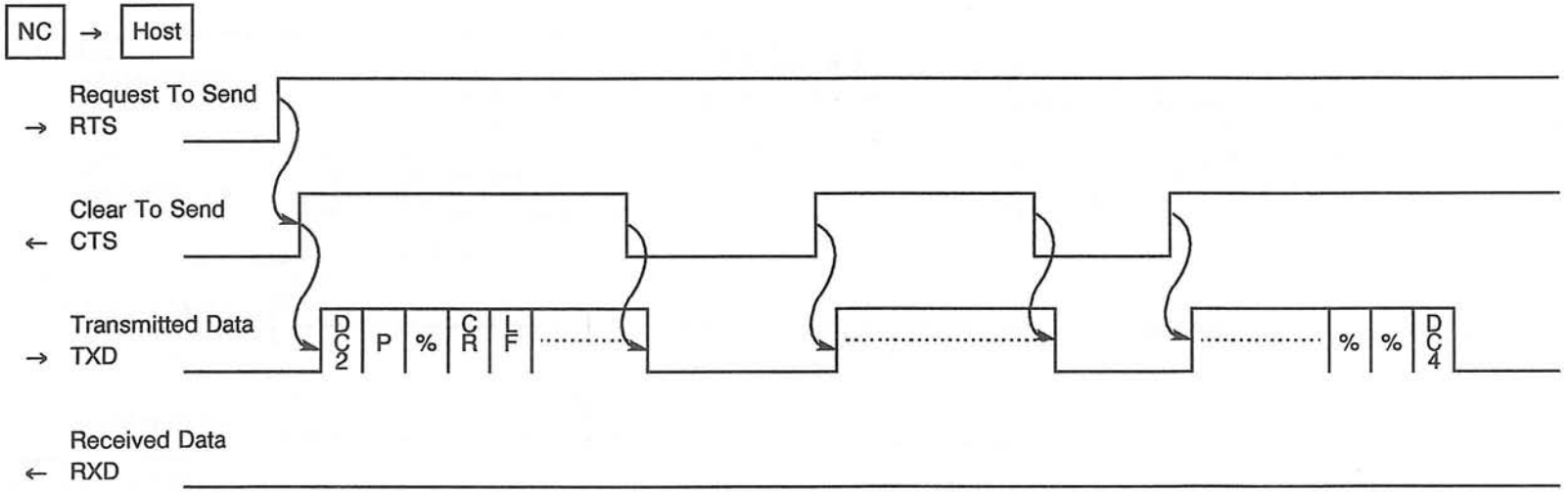


2) When request-file name output mode us bit specified (NC optional parameter bit Nos. 8, 13, 14, 21, 22 in 3-4 set to setting A)

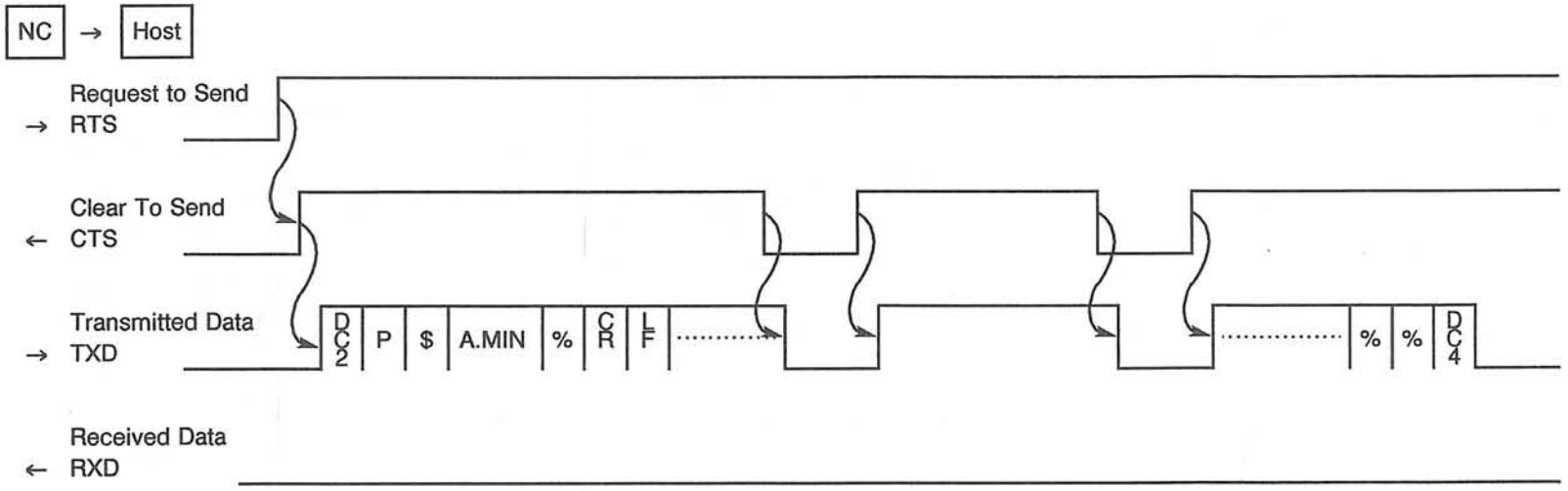


(b) Upload mode timing charts

1) When file name output mode is specified (NC optional parameter bit No. 10 in 3-1 set to setting A or C)



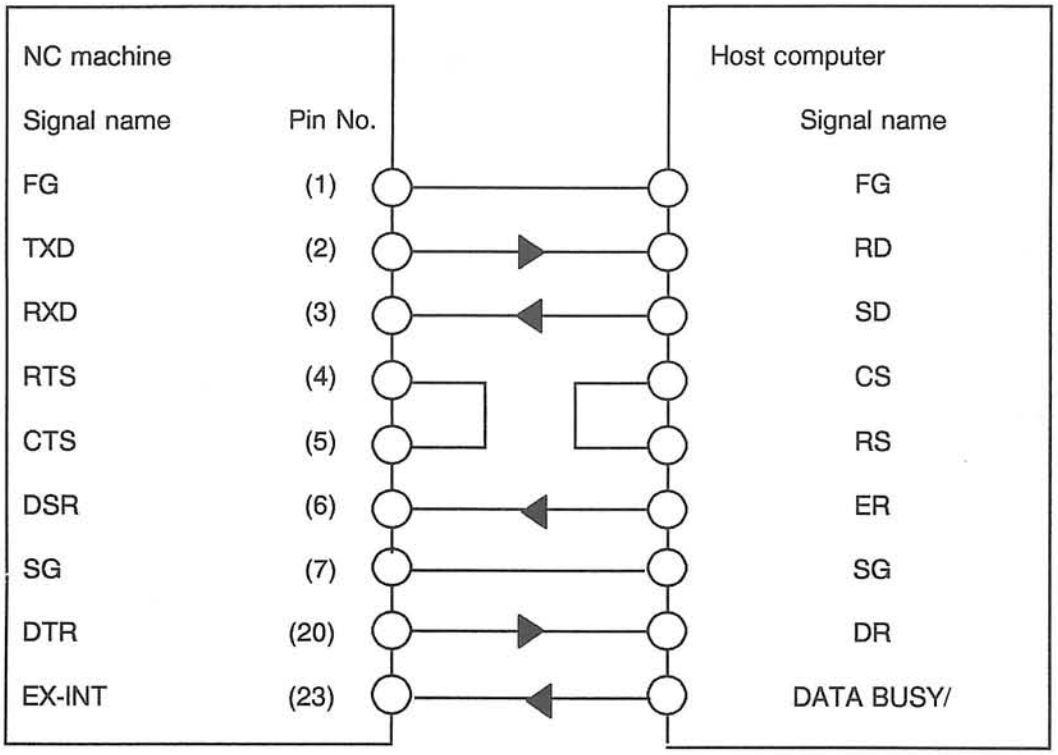
2) When file name output mode is specified (NC optional parameter bit No. 10 in 3-1 set to setting B or D)



Note 1: When CTS is set off during data transfer, the data transfer is suspended within two characters.

Note 2: Since this example configuration does not use the EX-INT signal, NC optional parameter bit for the "no EX-INT signal" mode (bit 1 of NC optional parameter bit Nos. 8, 13, 14, 21, 22) must be set to "1".

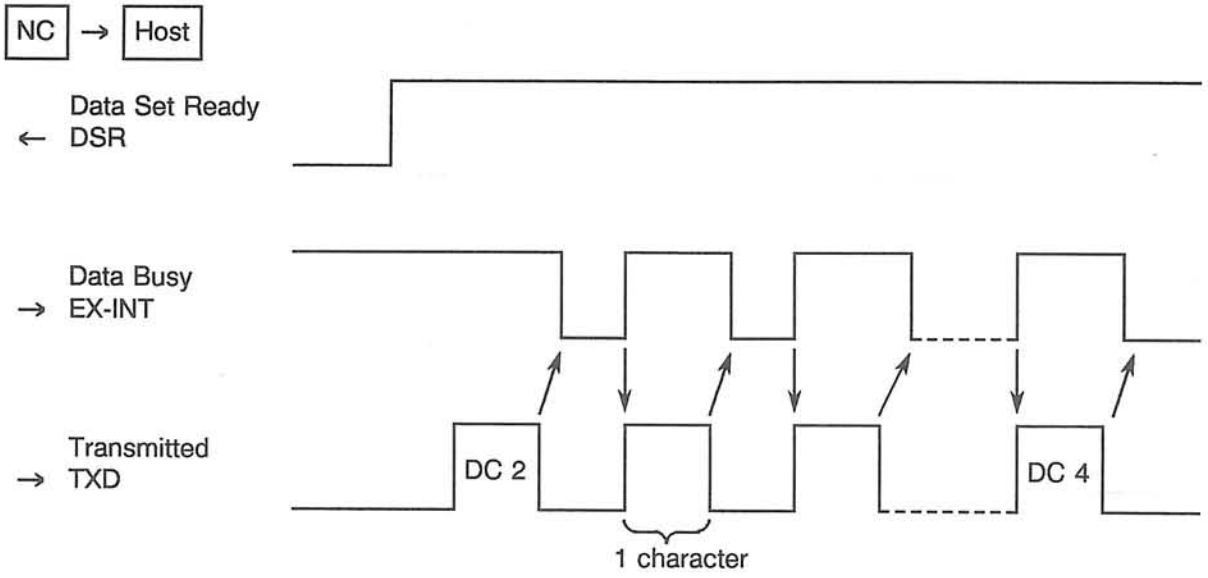
(2) Example 2:



(a) Download mode timing charts:

The timing chart in this mode is identical to that given in Example 1.

(b) Upload mode timing charts:



Note : *Since this example configuration uses the EX-INT signal, NC optional parameter bit for the "no EX-INT signal" mode (bit 1 of NC optional parameter bit Nos. 8, 13, 14, 21, 22) must be set to 0.*

5-2-3. DC Code Control Type 2

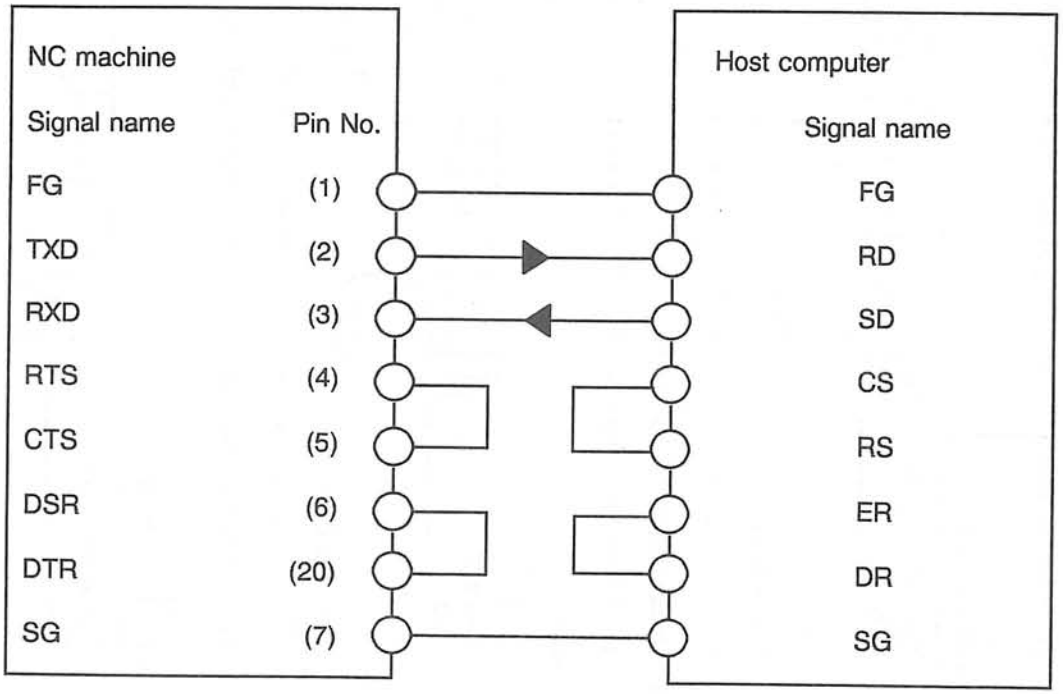
The DC code control type 2 configuration uses only DC codes to control data transfer between the host computer and a CNC machine through an RS232C interface.

(1) While the standard DC code control configuration allows only the CNC machine to generate DC codes, this configuration also allows the host computer to generate DC codes.

(2) The CNC machine uses four device control codes (DC1, DC2, DC3, and DC4) and the host computer two codes (DC1 and DC3).

DC code	NC machine	Host computer
DC1	Enables data read. (a) Initiates a read. (b) Resumes a read.	Enables data read. (a) Responds to DC2. (b) Resumes a read.
DC2	Placed before the transmitted data to request the host computer to receive the transmitted data.	
DC3	Requests suspension of data transmission.	Requests suspension of data transmission.
DC4	Identifies the end of data transmission.	

(3) Sample Connection Diagrams

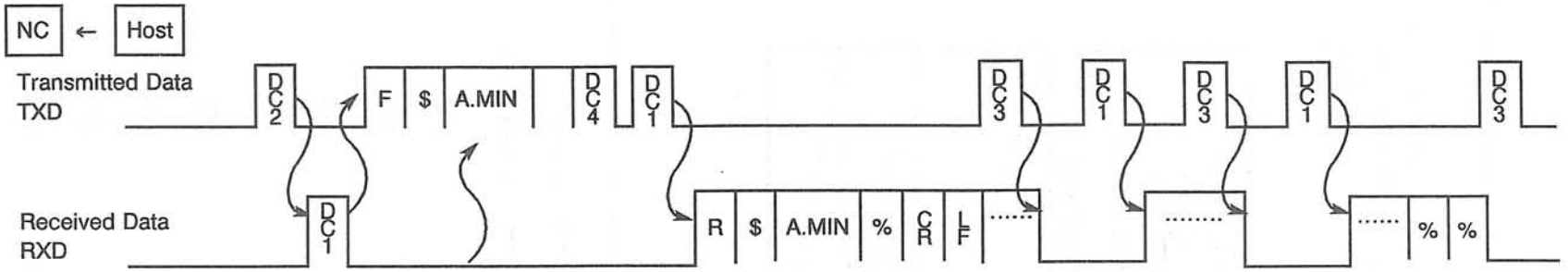


Note : *Since this example configuration does not use the EX-INT signal, NC optional parameter bit for the "no EX-INT signal" mode (bit 1 of NC optional parameter bit Nos. 8, 13, 14, 21 and 22) must be set to 1.*

(a) Download mode timing charts

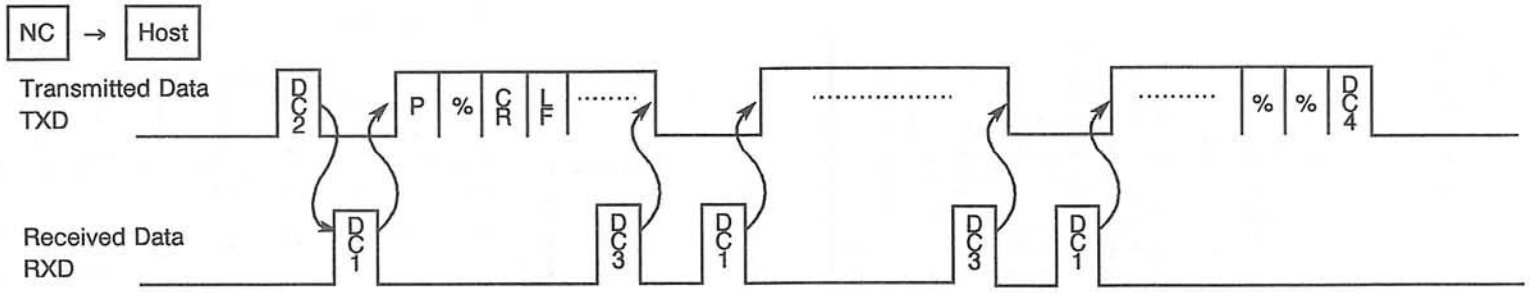
1) When request-file name output mode is not specified (NC optional parameter bit Nos. 8, 13, 14, 21, 22 in 3-4 set to setting B)
The timing chart in this mode is identical to that given in 2, (2), "DC Code Control".

2) When request-file name output mode is specified (NC optional parameter bit Nos. 8, 13, 14, 21, 22 in 3-4 set to setting A)

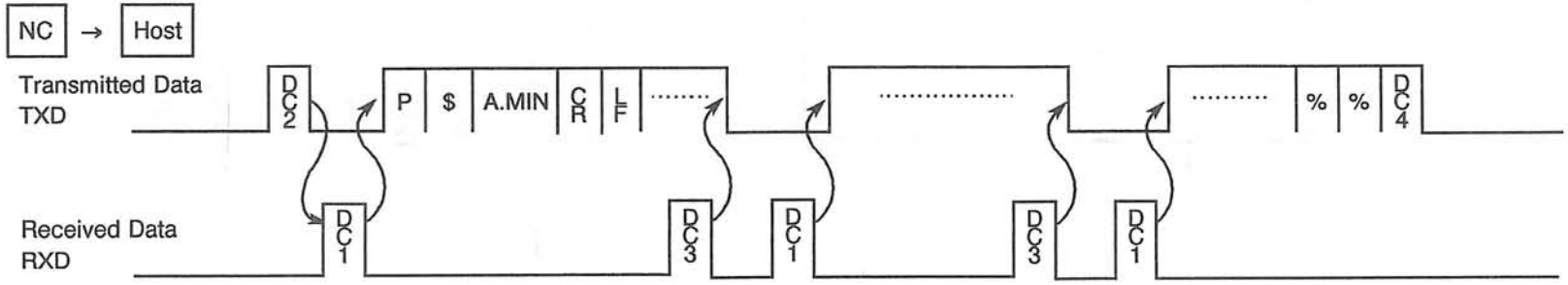


(b) Upload mode timing charts

1) When file name output mode is not specified (NC optional parameter bit No. 10 in 3-1 set to setting A or C)



2) When file name output mode is specified (NC optional bit No. 10 in 3-1 set to setting B or D)



SECTION 4 DNC-B HIGH-SPEED RM BUFFER METHOD

1. Overview

This section explains the communication link between the OSP and the host computer to transfer part programs between these two devices.

(1) There are two types of program transfer methods. One is the batch transfer method in which part programs are transferred independently of operation status of the OSP and the other is on-line mode program transfer in which part programs are transferred from the host computer to the buffer memory of the OSP and the machine is controlled by the program being sent. The latter mode of program transfer is called "buffer operation" in this manual.

Note that these two methods of program transfer cannot be selected at the same time.

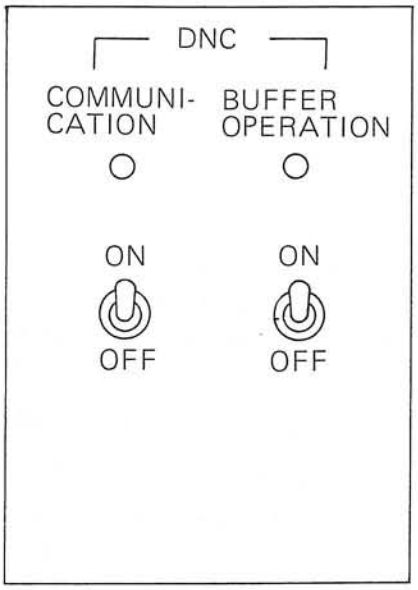
(2) The schedule operation is possible in the buffer operation mode by setting the parameters.

(3) Two types of protocol used for communication between the host computer and the OSP are selectable by the setting of parameter. These are protocol A and protocol B.

Protocol A . . . Shake hand method in which transmission and reception of data are repeated each other.

Protocol B . . . Communication is controlled by the DC (device control) codes which are output from the OSP.

2. DNC Operation Panel



(1) **COMMUNICATION ON/OFF** switch

(a) COMMUNICATION ON

Turning on power supply after placing the switch in the ON position establishes the communication mode between the OSP and the host computer. The COMMUNICATION lamp lights up.

(b) COMMUNICATION OFF

Turning on power supply after placing the switch in the OFF position disconnects communication line between the OSP and the host computer.

① Once power supply has been turned off, changing the switch setting position has no effect and the established state cannot be changed.

② With protocol A, turn the switch in the OFF position once and then back to the ON position to recover the communication alarm state* from the NC. Even in this case, the previously established communication ON state is not changed.

* The state in which the alarm W28 or W29 is displayed.

(2) **BUFFER OPERATION ON/OFF** switch

(a) BUFFER OPERATION ON

By setting the switch in the ON position with the DNC COMMUNICATION ON state, buffer operation is enabled and the BUFFER OPERATION ON lamp lights up.

(b) BUFFER OPERATION OFF

Buffer operation is disabled.

① If the switch is set at the BUFFER OPERATION OFF position during buffer operation, the buffer operation enabled state is continued until the buffer operation has completed.

② With protocol A, ON/OFF state of the buffer operation can be changed by the command from the host computer. If the command from the host computer does not match the switch setting, the command from the host computer is given priority.

However, if communication alarm occurs, ON/OFF state of the buffer operation is controlled by the setting of the **BUFFER OPERATION ON/OFF** switch setting at the OSP operation panel.

3. Buffer Operation

Buffer operation is possible in the automatic mode while the BUFFER OPERATION ON lamp at the DNC operation panel lights up.

Buffer operation function varies depending on the NC program transmission method (with or without file name designation).

Operation Type	Program Type	Operation Method	Buffer Operation (DNC mode: ON, Buffer operation: ON)				Normal Memory Operation	
			Without File Name Designation		With File Name Designation			
			Bubble Memory	Host Computer	Bubble Memory	Host Computer	Bubble Memory	Host Computer
Manual Mode Operation	Main Program	A	×	×	○	×	○	×
		B	×	×	○	×	○	×
		S	×	○	○	○	○	×
		M	×	×	○	×	○	×
Operation According to Schedule Program	Schedule Program	–	×	×	○	○	○	×
	Main Program	A	×	×	○	×	○	×
		B	×	×	○	×	○	×
		S	×	×	○	○	○	×
		M	×	×	×	×	×	×

Note 1: *"○" indicates "operation possible" and "×" indicates "operation impossible".*

Note 2: *Operation Methods:*

A .... Normal operation
B .... Large-volume operation
S .... Subprogram and branch function are not checked in operation method B.
M ... Multi-volume operation

The following operations are possible depending on "with" and "without" file name designation.

(a) Without file name designation .... Buffer operation of NC program

(b) With file name designation ..... Buffer operation of NC Program

Scheduled operation based on schedule program

Memory mode operation based on programs stored in bubble memory

3-1. Buffer Operation of NC Programs

3-1-1. Operation Methods

(1) If "file name not used" state is selected,

(a) Place the **DNC COMMUNICATION ON/OFF** switch at ON.

(b) Turn on power supply

(c) Select the AUTO OPERATION mode by pressing the **AUTO** key.

(d) Place the **BUFFER OPERATION ON/OFF** switch at ON.

(e) Press the **CYCLE START** button. (PROGRAM SELECT operation is not necessary.)

(2) If "file name used" state is selected,

(a) Place the **DNC COMMUNICATION ON/OFF** switch at ON.

(b) Turn on power supply

(c) Select the AUTO OPERATION mode by pressing the **AUTO** key.

(d) Place the **BUFFER OPERATION ON/OFF** switch at ON.

(e) After pressing the function key [F1] (PROGRAM SELECT), input the main program file name.

(f) Press the **WRITE** key

(g) The CRT displays prompt "Request file name data?!", requesting the operator to input the file address character-string managed by the host computer.

(h) Press the **WRITE** key.

(i) Press the **CYCLE START** button.

3-1-2. Command Format

(Possible only when "file name used" state is selected.)

=PS [main-program-file-name] **WRITE**

Request file name data?! file-address-character-string requested **WRITE**

(1) Main Program File Name
[program file name]. [extension]

(a) Designate a character-string of up to 16 characters beginning with an alphabet. Alphabets, numbers and hyphen (-) can be used.

(b) The extension is MIN.

(c) The main program file name designated in this step is registered as the file name for buffer operation.

(d) If a main program file name is omitted, A.MIN is used.

(e) To designate a device name in the program file name, designate the host computer (DNC:).

(2) File Address Caracter-string Requested

Character-string of up to 30 characters;

following characters can be used:

0 - 9, A - Z, !, ", #, $, %, &, ', <, >, (,), @,

?, ¥, , [,], _, ;, :, ,, +, −, *, /, =, ., **SP**

Transmission of the NC program of the request-file-name designated in this step is requested to the host computer. If the request-file-name is omitted, the main program file name is designated.

3-2. Scheduled Operation by Schedule Program
(Possible only when "file name used" state is selected.)

3-2-1. Operation Methods

(1) Place the **DNC COMMUNICATION ON/OFF** switch at ON.

(2) Turn on power supply.

(3) Select the AUTO OPERATION mode by pressing the **AUTO** key.

(4) Place the **BUFFER OPERATION ON/OFF** switch at ON.

(5) After pressing the function keys [F8] (EXTEND) two times and then [F4] (SP SELECT), input the schedule program file name.

(6) Press the **WRITE** key.

(7) The CRT displays prompt "Request file name data?!", requesting the operator to input the file address character-string managed by the host computer.

(8) Press the **WRITE** key.

(9) Confirm that the schedule program is selected on the program display screen.

(10) Press the **CYCLE START** button.

For the details of schedule program, refer to "OSP5020M/OSP500M-G Programming Manual" and attached Programming Supplementary Information.

3-2-2. Command Format for Schedule Program Selection

=SS [Schedule-program-file-name] **WRITE**

Request file name data?! file-address-character-string requested **WRITE**

(1) Schedule Program File Name

[program file name] [extension]

(a) Designate a character-string of up to 16 characters beginning with an alphabet.

Alphabets, numbers and hyphen (-) can be used.

(b) The extension is SDF.

(c) The schedule program file name designated in this step is registered as the file name for schedule operation.

(d) If a schedule program file name is omitted, S.SDF is used.

(e) To designate a device name in the program file name, designate the host computer (DNC:).

(2) File Address Character-string Requested

Character-string of up to 30 characters;

following characters can be used:

0 - 9, A - Z, !, ", #, $, %, &, ', <, >, (,), @,

?, ¥, , [,], _, ;, :, ,, +, −, *, /, =, ., **SP**

Transmission of the schedule program of the request-file-name designated in this step is requested to the host computer. If the request-file-name is omitted, the schedule program file name is designated.

3-3. Memory Mode Operation Using A Program Stored in Bubble Memory

The operation method is quite the same as used for normal memory mode operation.

Only the difference is the designation of the bubble memory (BB1:) as the device name when designating the schedule program file name and the main program file name (including main programs in the schedule program).

3-4. Precautions

(1) Subprograms only registered in the library program can be used.

(2) Size of part program has no limit for program transfer.

(3) Only schedule programs or main programs can be transmitted.

If subprograms and/or other main programs continue following the M02 code of the main program, do not forget to set the "M02 reset output used" state by the parameter.

(4) At the instant the **CYCLE START** button is pressed, transfer of a part program begins (GTD sent for protocol A, DC1 or DC2 sent for protocol B) and then the program execution begins.

(5) Change of parameter setting for file name used or not-used status becomes active when power supply is turned off and then on again.

(6) Even in the buffer operation mode, intervention of manual operation is possible as usual,

(7) Even in the buffer operation ON mode, the return search and sequence restart commands can be executed.

(8) In the buffer operation mode, when "file name not used" is selected, the following commands cannot be executed:

- Schedule program selection
- Number search
- Sequence search using the cursor
- Schedule program search

(9) When "file name not used" state is selected, program selection is always required when buffer operation mode is switched form ON to OFF. If program selection is not carried out, program execution does not begin.

(10) When "file name used" state is selected, program selection is always required when buffer operation mode is switched between ON and OFF. If program selection is not carried out, program execution does not begin.

(11) When selecting a program in the file name used state, designation of a main program name and subprogram name is not possible.

(12) Irrespective of the setting of the **BUFFER OPERATION ON/OFF** switch, the selected buffer operation on or off mode is continued until the operation in progress has been completed even if the setting of the switch is changed.

This function is effective for program transfer operation.

If the buffer operation on/off mode is switched by the command using protocol A, an alarm occurs if such an attempt is made during operation irrespective of the setting of the buffer operation on/off mode.

(13) Batch transfer of NC program is impossible during buffer operation (buffer operation mode on and process in between cycle start and M02 execution, or NC reset).

(14) After the execution of MDI mode operation, buffer operation is not possible unless the NC is once reset.

Interruption by MDI mode operation during buffer mode operation may be executed in normal operation intervention manner.

(15) Automatic program selection when power is turned on:

(a) File name not used

BUFFER OPERATION ON . . . . . Program selection is not executed.

BUFFER OPERATION OFF . . . . . Program selection is executed.

(b) File name used

The buffer operation on/off mode and the file name being selected when power has been turned off are stored in memory. If the buffer operation on/off mode currently active and the mode active before power off match, automatic program selection is executed. When buffer operation mode is on, the main program of the file name stored is requested to the host computer when the **CYCLE START** button is pressed.

If the previous and current buffer operation on/off modes do not match, automatic program selection is not executed. When buffer operation mode is on, no operation occurs when the **CYCLE START** button is pressed.

4. Batch Transfer of NC Programs

NC programs can be transferred from the NC to the host computer or from the host computer to the NC in batch.

4-1. Operation Procedure

Follow the steps below:

(1) Place the **DNC COMMUNICATION ON/OFF** switch at ON.

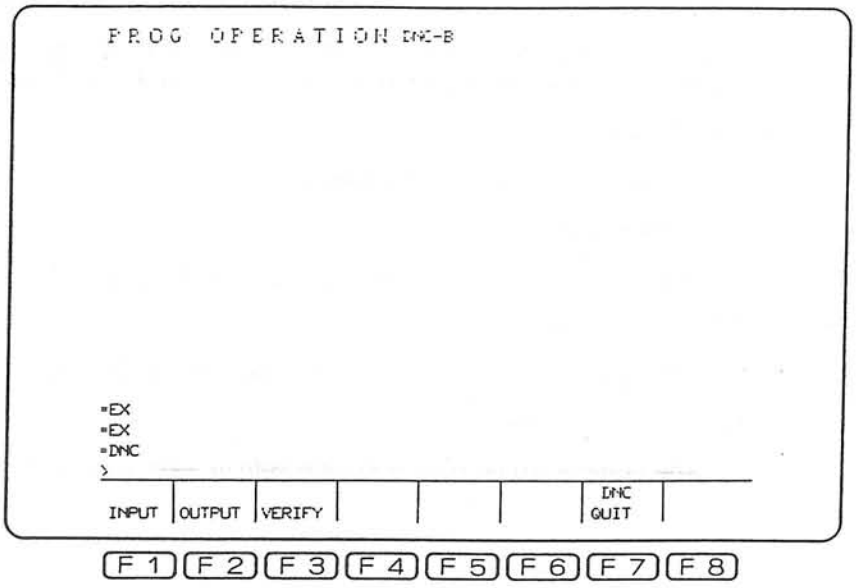
(2) Turn on power supply.

(3) Place the **BUFFER OPERATION ON/OFF** switch at OFF.

(4) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.

(5) Press the function keys [F8] (EXTEND) and then [F4] (DNC). This displays the DNC screen.

In this status, batch transfer of the program is possible.



(6) (a) For program request to the host computer

After pressing the function key [F1] (INPUT), input the file name of the file to be stored in the bubble memory.

(b) For sending NC program to the host computer

After pressing the function key [F2] (OUTPUT), input the file name of the file stored in the bubble memory.

(c) For program verification

To verify the NC program already stored in the bubble memory, press the function key [F3] (VERIFY) and input the file name of the file stored in the bubble memory.

(7) Press the **WRITE** key.

If "file name used" state is selected by the parameter, follow the step (8) and (9) below.

If such state is not selected, batch transfer of the designated file begins.

(8) (a) For program request to the host computer

Key in the request-file name designation data.

(b) For sending NC program to the host computer

Key in the transmission-file-name designation data.

(c) For program verification

Key in the request-file-name designation data.

(9) Press the **WRITE** key.

This begins transmission of the NC program in batch.

(10) After the completion of program transmission, press the function key [F7] (DNC QUIT).

Note 1: As request-file-name designation data and transmission-file-name designation data, input the program file name data (character-string) which is managed by the host computer. This allows the input data to be sent to the host computer.

Note 2: For program verify operation, the following message is displayed after the completion of verification in accordance with the results of the verification.

data all same

File and received NC data match.

DNC verify unsame-2

Although the file has terminated, received NC data is left.

DNC verify unsame-1

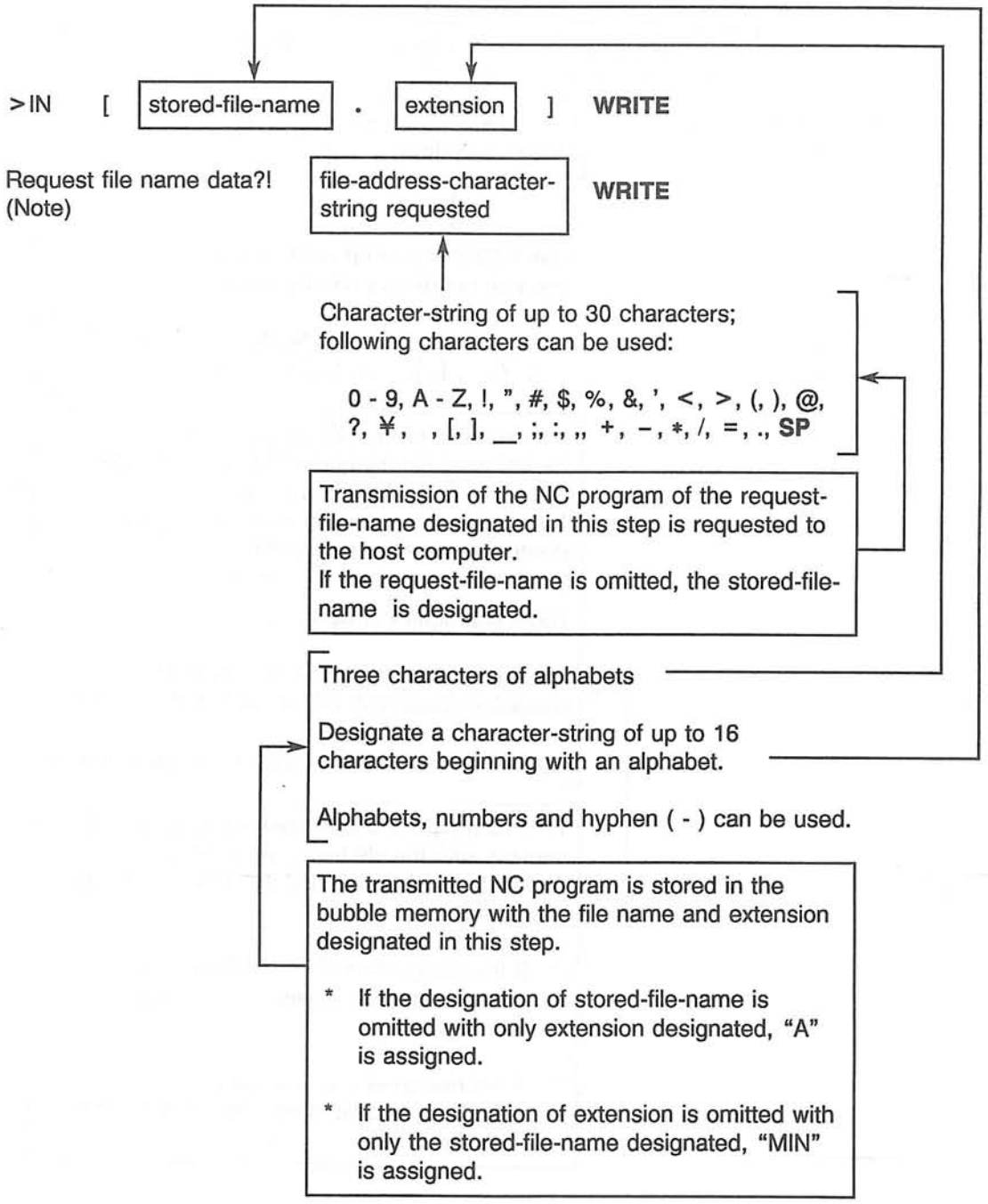
Although the received NC data has terminated, file data is left.

DNC verify unsame-n

The number (n) of mismatch data sets in verify operation.

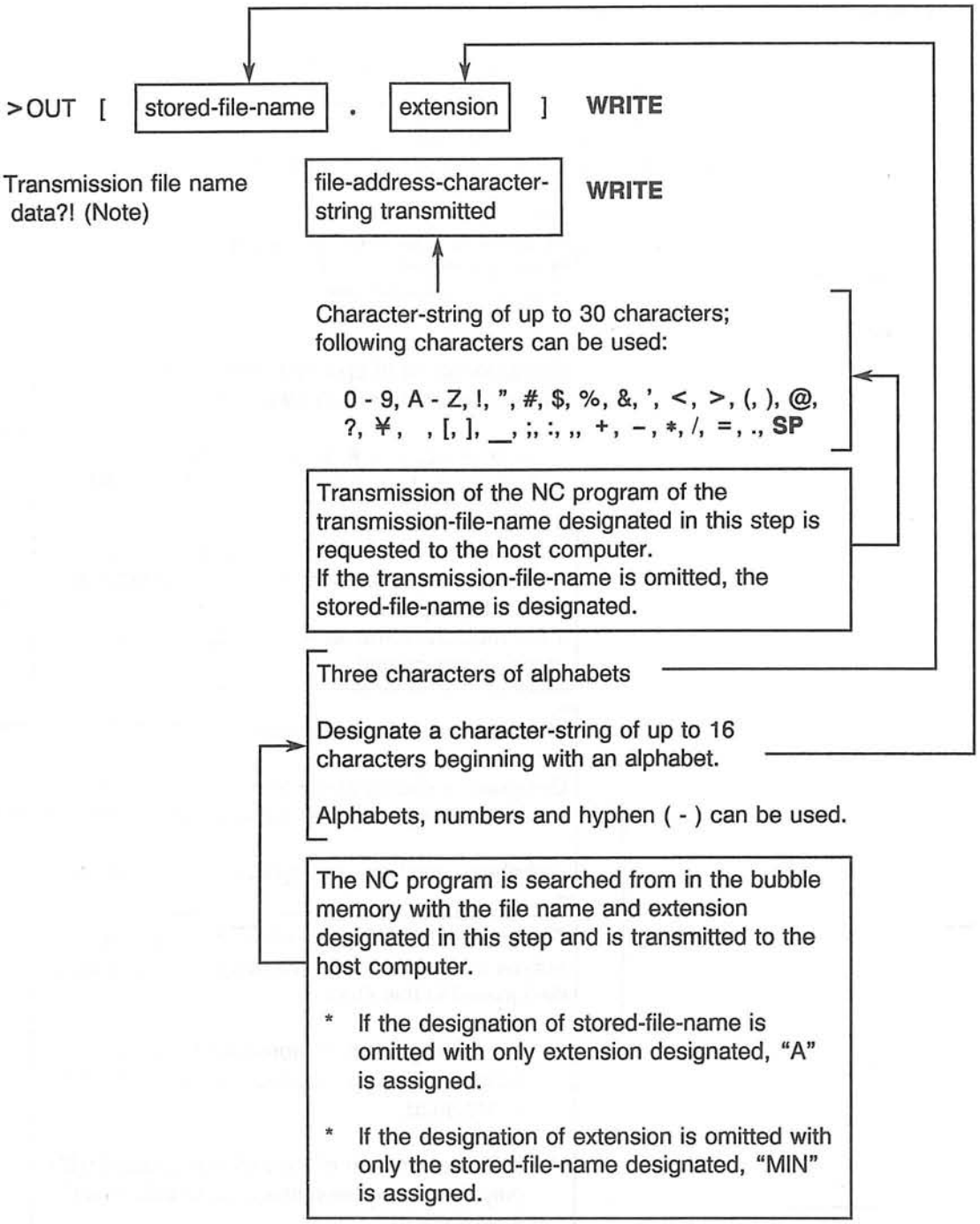
4-2. Command Format

(1) Input



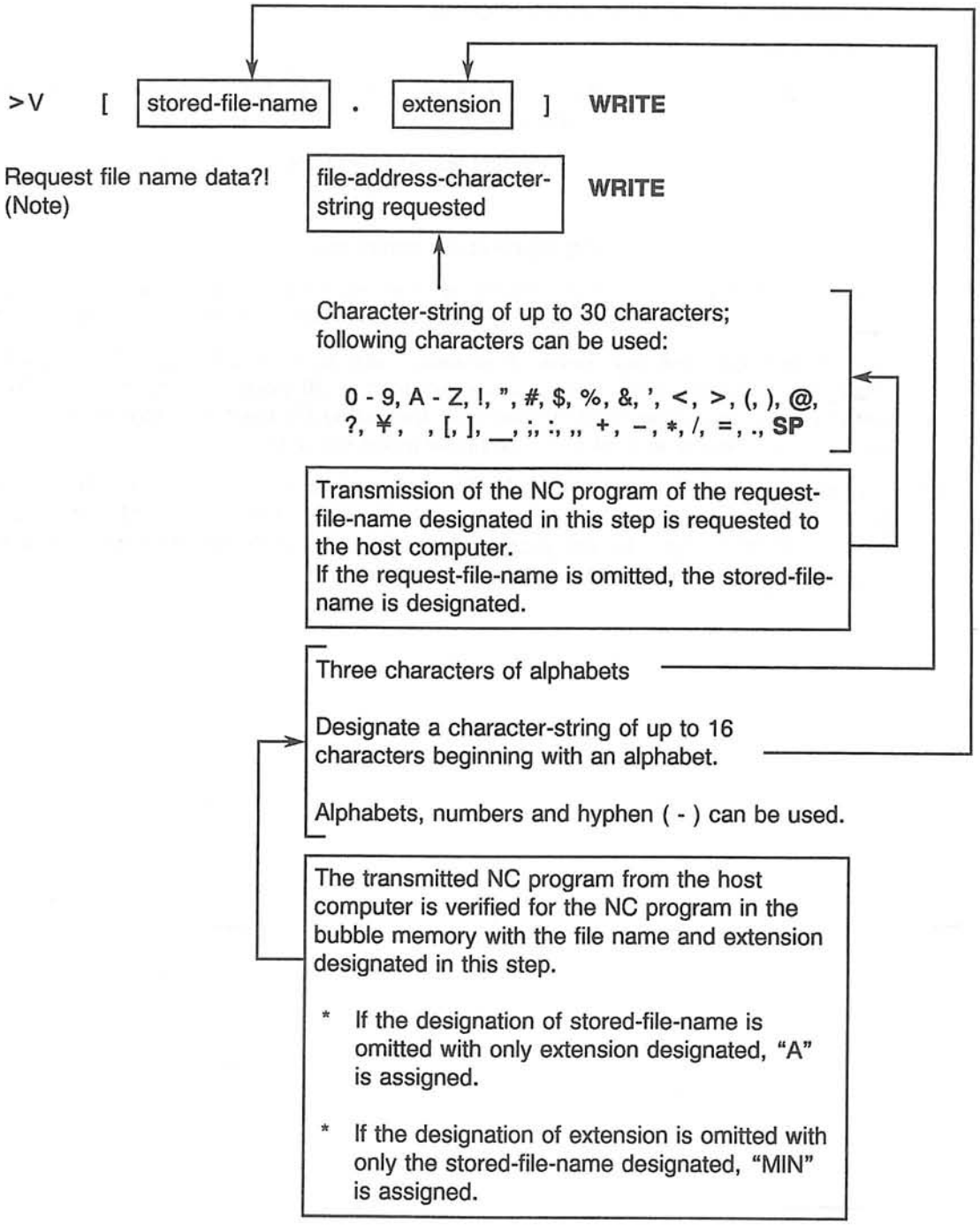
Note: The "file-address-character-string requested" column becomes effective when "file name used" state is selected by parameter. If "file name used" state is not selected, it should be omitted.

(2) Output



Note: The "file-address-character-string transmitted" column becomes effective when "file name used" state is selected by parameter. If "file name used" state is not selected, it should be omitted.

(3) Verify



Note: The "file-address-character-string requested" column becomes effective when "file name used" state is selected by parameter. If "file name used" state is not selected, it should be omitted.

4-3. Precautions

(1) Transmission and reception of NC programs

(a) Protocol A:

Method 1 . . . . File name is not added to RTD command and the first GTD command.

Method 2 . . . . File name is added to RTD command and the first GTD command.

(b) Protocol B:

Method 1 . . . . File name is not transmitted.

Method 2 . . . . For program transmission, file name is first transmitted; for program reception, DC2 code, file name and DC4 code are transmitted.

In each protocol, switching between method 1 and method 2 is made by the parameter. After changing parameter setting, it is necessary to turn off power supply and then turn it back on. Note that the parameter used for switching the method is used in common with the one used for switching the operation method in the buffer mode operation.

(2) For program "input" operation, if the file assigned with the file name and extension same as those designated is already existing, prompt "file exist overwrite? (Y/N)!" appears on the CRT screen. If "Y" is input for this prompt, the existing file is deleted and new file is input from the host computer. If "N" is input, no processing occurs.

5. Parameters

5-1. Parameters Which Cannot be Set At OSP

The parameters indicated below cannot be set at the OSP and setting is possible only by using the SET command from the host computer.

(Parameters in the table below, conversion codes, codes after conversion (45th - 48th bytes of SET command))

(For protocol A only)

Parameter	Function	Unit	Range	Initial Value when Power is Turned On
Nb	Minimum buffer area of CNC when transmitting [GTD]	bytes	1 - 4000	2000
No	Maximum overrun amount in reception	bytes	2 - 2000	50
Ne	Number of retries allowed when transmission error is detected	times	0 - 100	10
Tp	[SAT] transmission polling time for DNC, or response time-out duration by CNC for host computer	sec	1 - 999	5
To	Response time-out duration by host computer for CNC	sec	1 - 999	20
Ti	Minimum time interval between transmission bytes	msec	0 - 10	10
Tx	Minimum switching time from reception to transmission	msec	0 - 100	100
Tw	Standby time for reception of [WAT] used by both of CNC and host computer; upper limit value is Tp or To, whichever smaller.	sec	0 - Tp 0 - To (Whichever smaller)	5

Tw: Communication is executed in the following manner when "Tw" value exceeding the specified range is set by the host computer using the SET command.

(1) When Tw value > Tp value

In the remote buffer mode operation or batch transmission mode, the OSP sends the GTD command after waiting for "Tw" when it received the WAT command from the host computer. The SAT command is not sent out during this period.

(2) When Tw value > To value

If program data to be transmitted is not ready in the NC program for transmission from the OSP to the host computer operation, the OSP sends the WAT command after and elapse of "Tp". If the host computer sends the GTD command in response to this after waiting for "Tw", the OSP regards that, after an elapse of "To", there is no answer to the WAT command. In this case the communication error occurs.

5-2. Parameters Which Can be Set At OSP

Some parameters are required to be changed frequently during operation while some are not changed once they have been set. The OSP differentiates these two types of parameters to allow easy setting of parameters. The former is assigned to NC optional parameter (long word), (word) and (bit), and the latter to the communication parameter (dedicated screen).

(1) NC Optional Parameter (bit)

(a) Bit 0 of No. 1

Coding system of part programs

1: ISO

2: EIA

Initial setting : 1

(b) Bit 2 of No. 1

TV check, creation designation

This parameter determines whether or not the TV check, creation is made for the NC program to be received or transmitted--a space is inserted before LF (or CR LF) so that a total number of characters in a record including the LF (or CR LF) will be even, or whether the record contains an even number of characters is checked.

1: TV check, creation is executed.

2: TV check, creation is not executed.

Initial setting : 0

(c) Bit 3 of No. 11

End of record code

1: LF

0: CR LF

Initial setting : 0

(d) Bit 0 - bit 7 of No. 43 and No. 44

Data at the 41st to 44th bytes of [SAT] or [SET] command.

[SAT] command . . . . From OSP to the host computer

[SET] command . . . . From the host computer to OSP

Bit 0 of No. 44

1: Buffer is cleared when NC is reset.

0: Buffer is not cleared when NC is reset.

Initial setting : 0

Note: Other bits have not been defined.

(2) NC Optional Parameter (word)

(e) No. 16

Number of bytes in one block for transmission of NC program from OSP to the host computer in protocol A.

If setting is "0", it follows Nb sent from the host computer.

Initial setting : 0

Setting range: 0 - 4000

(f) No. 24

File read-out waiting time T2 used for protocol B
(Time duration between transmission of DC4 code and transmission of DC1 code.)

Initial setting : 0 (sec)

Setting range: 0 - 9999 (sec)

If setting is "0", time duration of T1 (Parameter No.35) is automatically adopted.

(g) No. 35

Response timer used for protocol B
(Time duration between transmission of DC1 code and input of NC program.)

Initial setting : 10 (sec)

Setting range: 1 - 9999 (sec)

(3) NC Optional Parameter (long word)

(h) No. 13

In the remote mode operation, program is started after the specified number of character of the NC program is written to the buffer.

Initial setting : 0

Setting range: 0 - 99999999

Note: If the setting is smaller than 256, it is handled as 256. If the setting is greater than the main program buffer capacity, program is started when the main program buffer became full.

(4) Communication Parameters

Communication parameters are displayed and set in the parameter setting mode.

Note that the communication parameter screen cannot be displayed in conversional method with the function keys. Follow the steps below to display the communication parameter screen.

① Select the PARAMETER SET mode by pressing the **PARAMETER** key.

② Key in CMPS and press the **WRITE** key. This displays the COMMUNICATION PARAMETER screen.

③ Set the required parameters

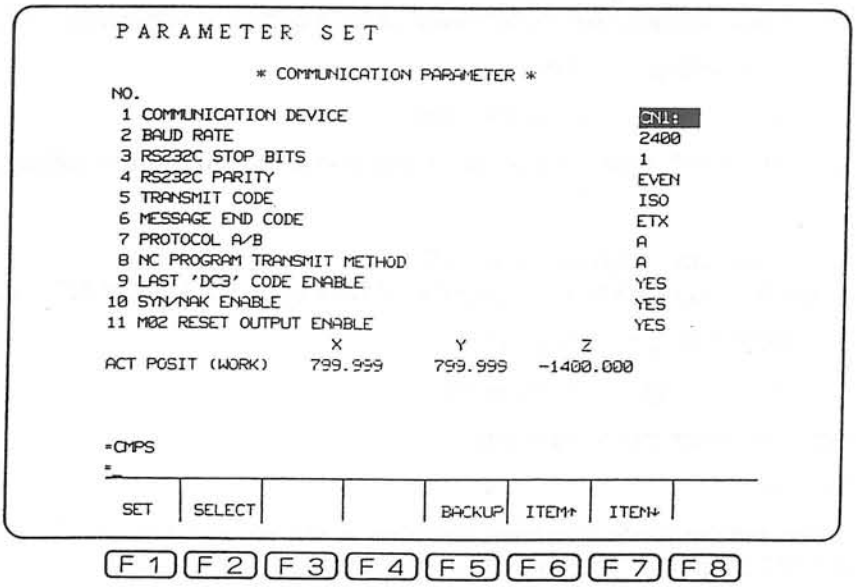
This setting is possible by using the function key [F2] (SELECT). Each time this key is pressed, contents are automatically changed within a setting range.

④ Press the function key [F5] (BACKUP). This operation is always required when data is set or changed.

⑤ Turn off power supply and then turn it back on

The communication parameters become effective when power reapplied.

(5) Description of Communication Parameters



No.	Name	Description
1	COMMUNICATION DEVICE	Designation of device name used for communication Select from CHA: and CHB: Initial setting : CHA:
2	BAUD RATE (BPS)	Designation of baud rate Select from 110, 150, 300, 600, 1200, 2400, 4800, 9600, 19200, 38400, 89300. Initial setting : 2400
3	RS-232C STOP BITS	Designation of number of stop hits Select from 1 and 2. Initial setting : 1
4	RS-232C PARITY	Designation of parity added at RS-232C interface Select from NON, EVEN and ODD. Initial setting : EVEN

No.	Name	Description
5	TRANSMIT CODE	Designation of transmission code (excluding NC programs) Select from ISO and ASCII. Initial setting : ISO *Note: The transmission code is not used for protocol B.*
6	MESSAGE END CODE	Designation of message end code Select from ETX and CR. Initial setting : ETX *Note: The end code is not used in protocol B.*
7	PROTOCOL A/B	Selection of protocol Select from A (protocol A) and B (protocol B). Initial setting : A
8	NC PROGRAM TRANSMIT METHOD	Designation of "file name used" or "file name not used" state for program transmission Select from A (not used) and B (used). Initial setting : A
9	LAST 'D3' CODE ENABLE	Designation whether or not DC3 code at the end of NC program reception is transmitted Select from YES (transmitted) and NO (not transmitted). Initial setting : YES *Note: The DC3 code is not used in protocol A.*
10	SYN/NAK ENABLE	Designation whether or not [SYN] and [NAK] are transmitted at an occurrence of NC alarm or NC reset Select from YES (transmitted) and NO (not transmitted). Initial setting : YES *Note: This is not used in protocol A.*
11	M02 RESET OUTPUT ENABLE	Designation whether or not reset signal ([RST] command for protocol A, [SYN] code for protocol B) is transmitted to the host computer upon execution of M02 and M30. Select from YES (transmitted) and NO (not transmitted). Initial setting : YES

6. Protocol [A]

6-1. Communication Format

When the power is turned ON, the NC and Host Computer are prepared for operation. When this preparation is completed, transmissions from the NC and reception from the Host Computer begins. From this point on, transmissions and receptions are repeated in an alternating manner.

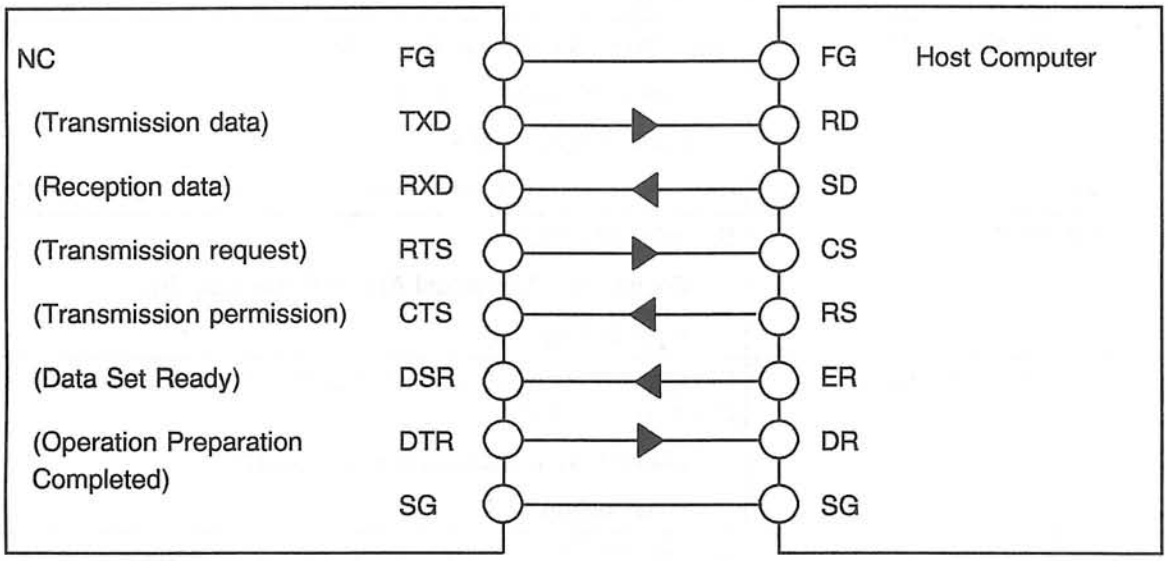


Fig. 6-1-1 Example of RS-232C Link

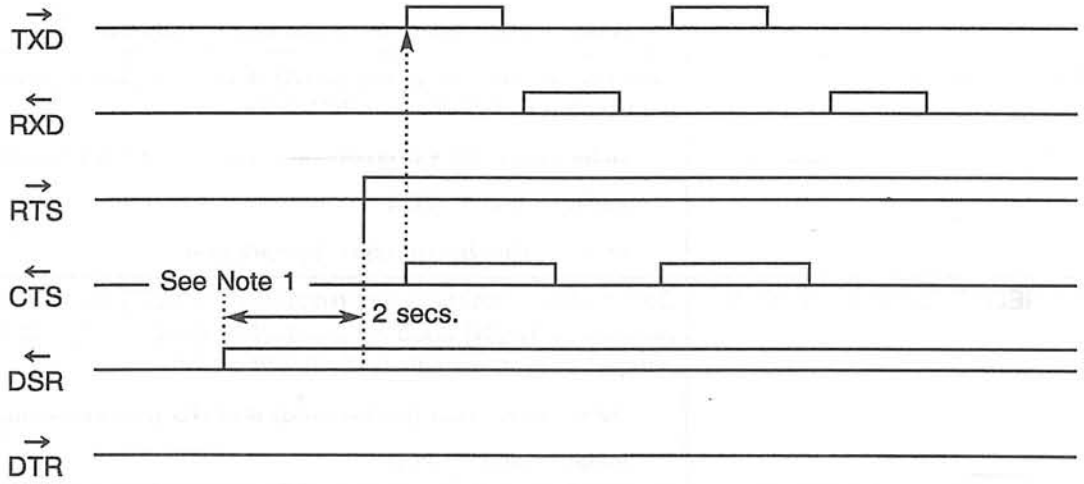


Fig. 6-1-2 Timing Chart at Communication START

Note 1: The initial transmission request occurs 2 seconds after a READY status is established at both the NC and Host Computer.

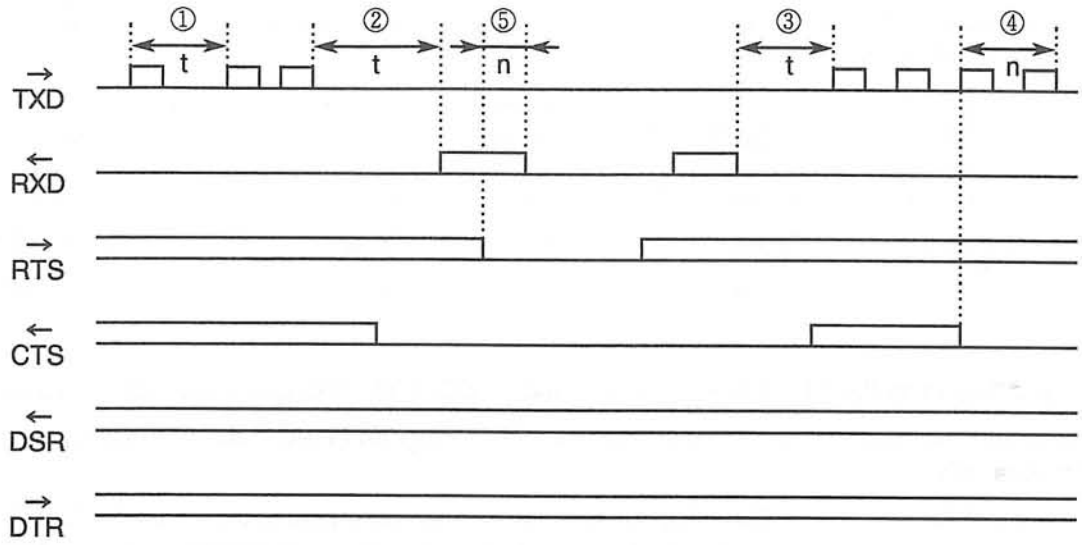


Fig. 6-1-3 Timing Chart During Communication

①	Minimum time interval between transmission bytes	:	Ti≦t
②	"Transmission to reception" switching time	:	0≦t≦To
③	"Reception to transmission" switching time	:	Tx≦t≦Tp
④	Overrun which follows CTS signal OFF	:	n≦2 bytes
⑤	Overrun amount which follows reception RTS signal OFF	:	n≦No

To: Timeout period

Tp: Polling time interval

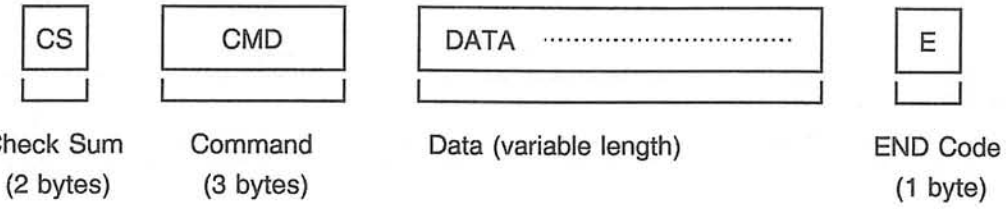
Tx: Minimum "reception to transmission" switching time

No: Maximum overrun amount during reception

Note 1: There is no specific designation for "Ti".

6-2. Message Format

Data exchanges (hereafter referred to as "messages") between the NC and Host Computer occurs using the data format shown below.



All data, including the Check Sum, is comprised of ISO or ASCII codes (parameter switching).

However, when the "DATA" portion consists of NC Program or NC data, EIA codes can be used for that data only.

① Check Sum ..... The COMMAND to END code data total is shown at the Check Sum area, where the final 1-byte of the total is expressed as a hexadecimal, 2-digit value. (Transmission occurs in order, from high to low.)

② Command ..... The COMMAND is comprised of 3 characters, and indicates the message type.

③ Data ..... This is the text data of variable length. The format and content of this data is determined according to the COMMAND. (Numeric data is transmitted in order, from high to low.)

④ END Code ..... This code indicates the end of the message. Either "ETX" or "CR" may be used as this code (parameter switching).

When ISO code is used : ETX (03H) or CR (8DH)

When ASCII code is used: ETX (03H) or CR (0DH)

The message must not contain any data which appears in the same pattern as the END code.

6-3. Command List

(1) Transmission Commands (NC→Host Computer)

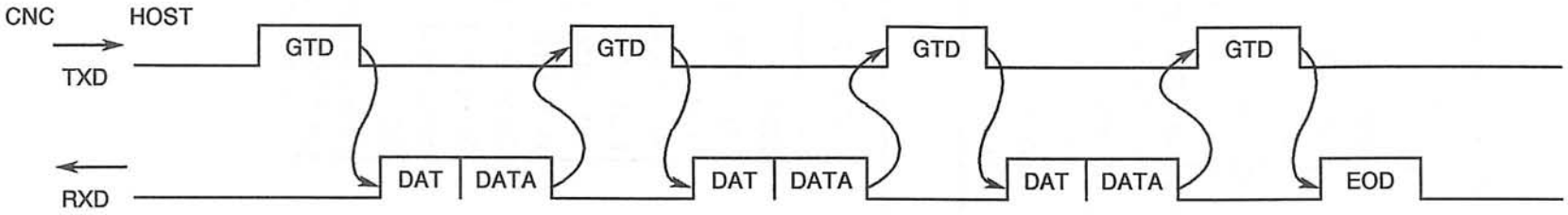
Command	Description	Data Area	Reply From Host
SYN	Initialize command	None	SYN
RDY	Initialize completed	None	SYN, RDY
RST	DNC reset notification	None	ARS
ALM	DNC alarm notification	None	AAL
SAT	DNC status notification	Status	SET, CLB, SYN
GTD	NC Program data transmission request	NC Program file name	DAT, EOD, WAT
RTY	Repeat request (retry)	Error status	Previous message
RTD	NC Program data upload request	NC Program file name	GTD
DAT	NC Program data upload	NC Program data	GTD
WAT	Wait	None	GTD
EOT	NC Program data uploading completed	None	SET

(2) Reception Commands (Host Computer→NC)

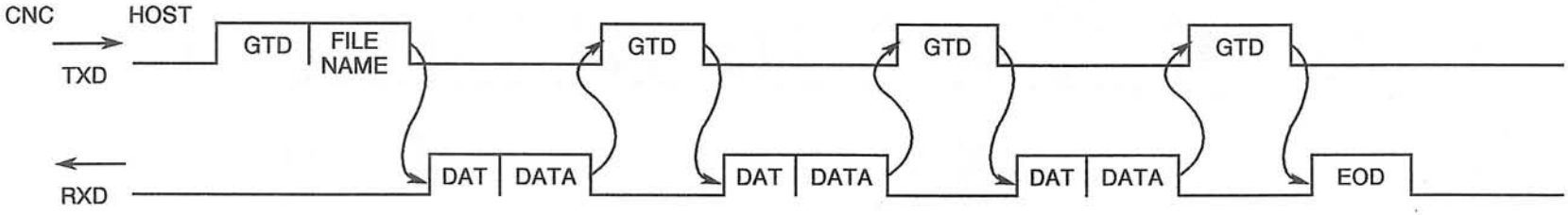
Command	Description	Data Area	Reply From NC
SYN	Initialize command	None	RDY
RDY	Initialize completed	None	RST, ALM, SAT, GTD
ARS	RST reply	None	Same as above
AAL	ALM reply	None	RST, SAT
CLB	Buffer delete	None	RST, ALM, SAT, GTD, RTD
SET	NC status change	Status	Same as above
DAT	NC Program data transmission	NC Program data	Same as above
EOD	NC Program data transmission completed	None	Same as above
WAT	Wait command	None	GTD
RTY	Repeat transmission (retry)	Error status	Previous message
GTD	NC Program data transmission request	None	DAT, WAT, EOD

6-4. Data Reception

(1) When File Name is not Designated:

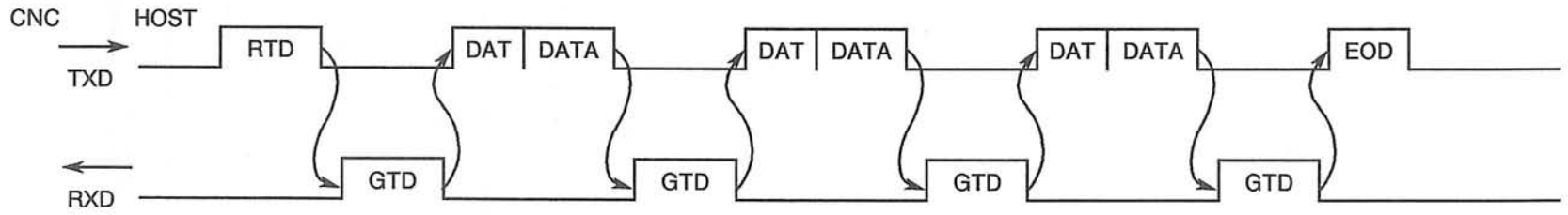


(2) When File Name is Designated:

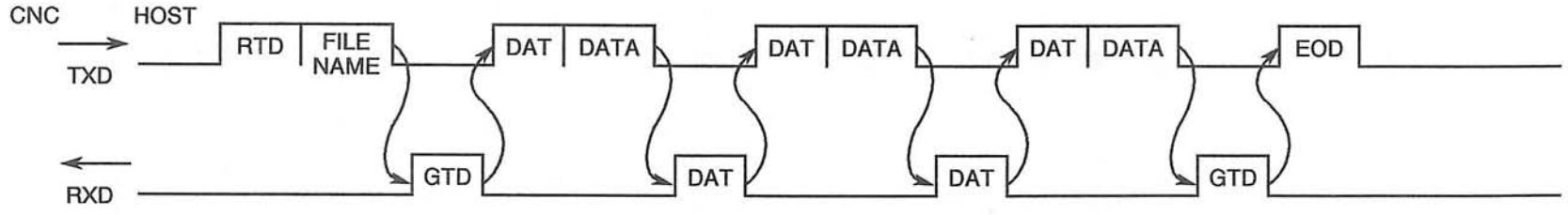


6-5. Data Transmission

(1) When File Name is not Designated:



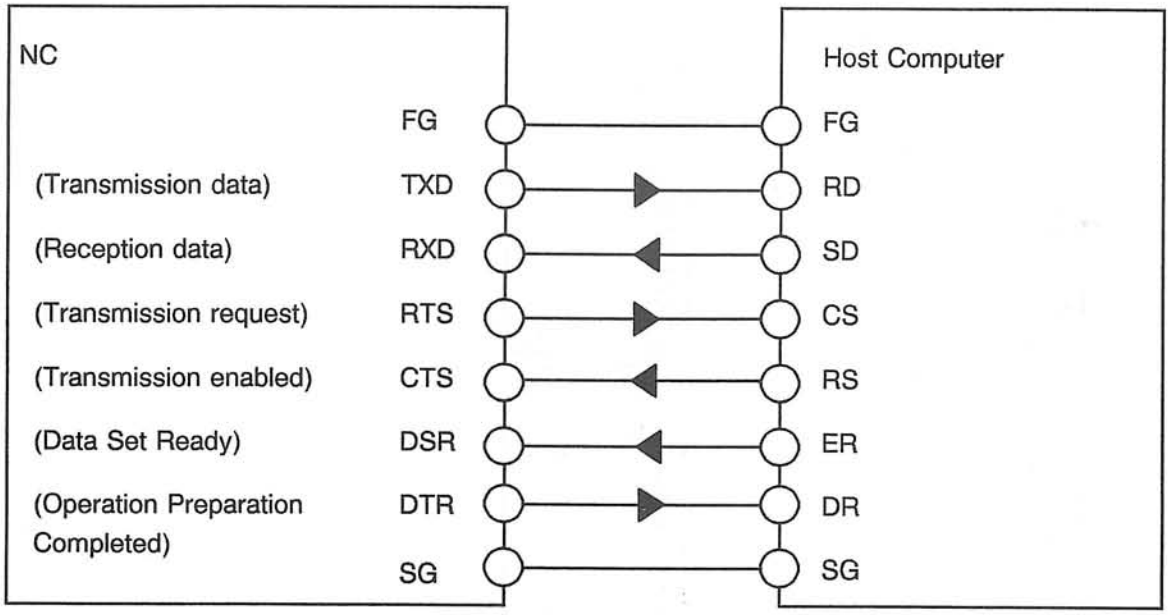
(2) When File Name is Designated:



7. Protocol [B]

7-1. Communication Format

Data transmission/reception requests are always sent from the NC to the Host Computer using DC codes (Device Control Code).



<Fig.7-1-1. Example of RS-232C Link>

7-2. DC (Device Control) Codes

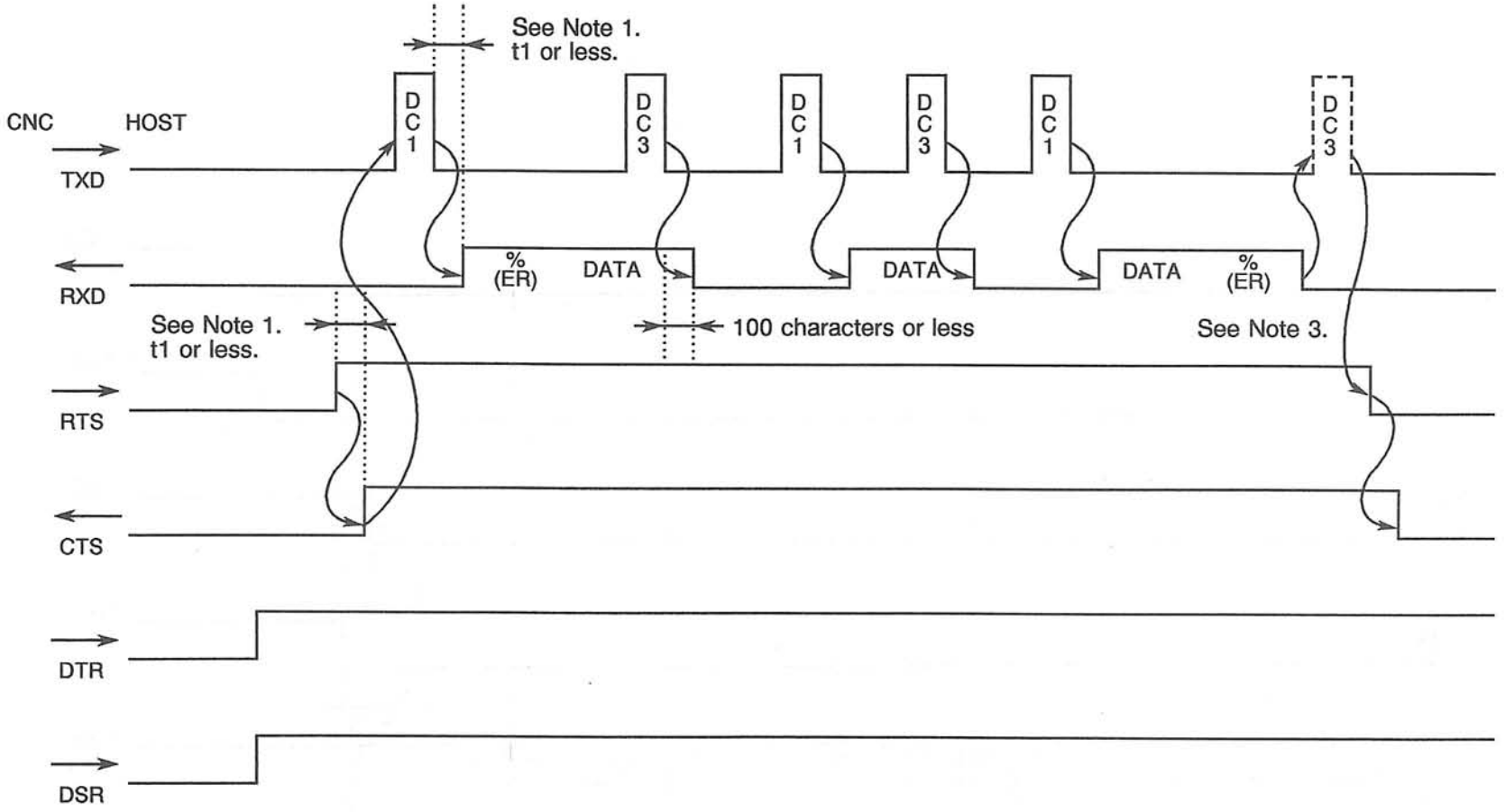
The DC codes will be as shown in the Table below, regardless of the code system (ISO or EIA) of the NC Program data which is being transmitted.

Character		8	7	6	5	4		3	2	1
DC1	Tape Reader START				○		∘			○
DC2	Tape Puncher START				○		∘		○	
DC3	Tape Reader STOP	○			○		∘		○	○
DC4	Tape Puncher STOP				○		∘	○		

Note 1: DC codes are generated automatically by the NC, and need not be included in the Program.

7-3. Data Reception

(1) When File Name is not Designated:



Note 1: *t1 may be switched by parameter setting.*

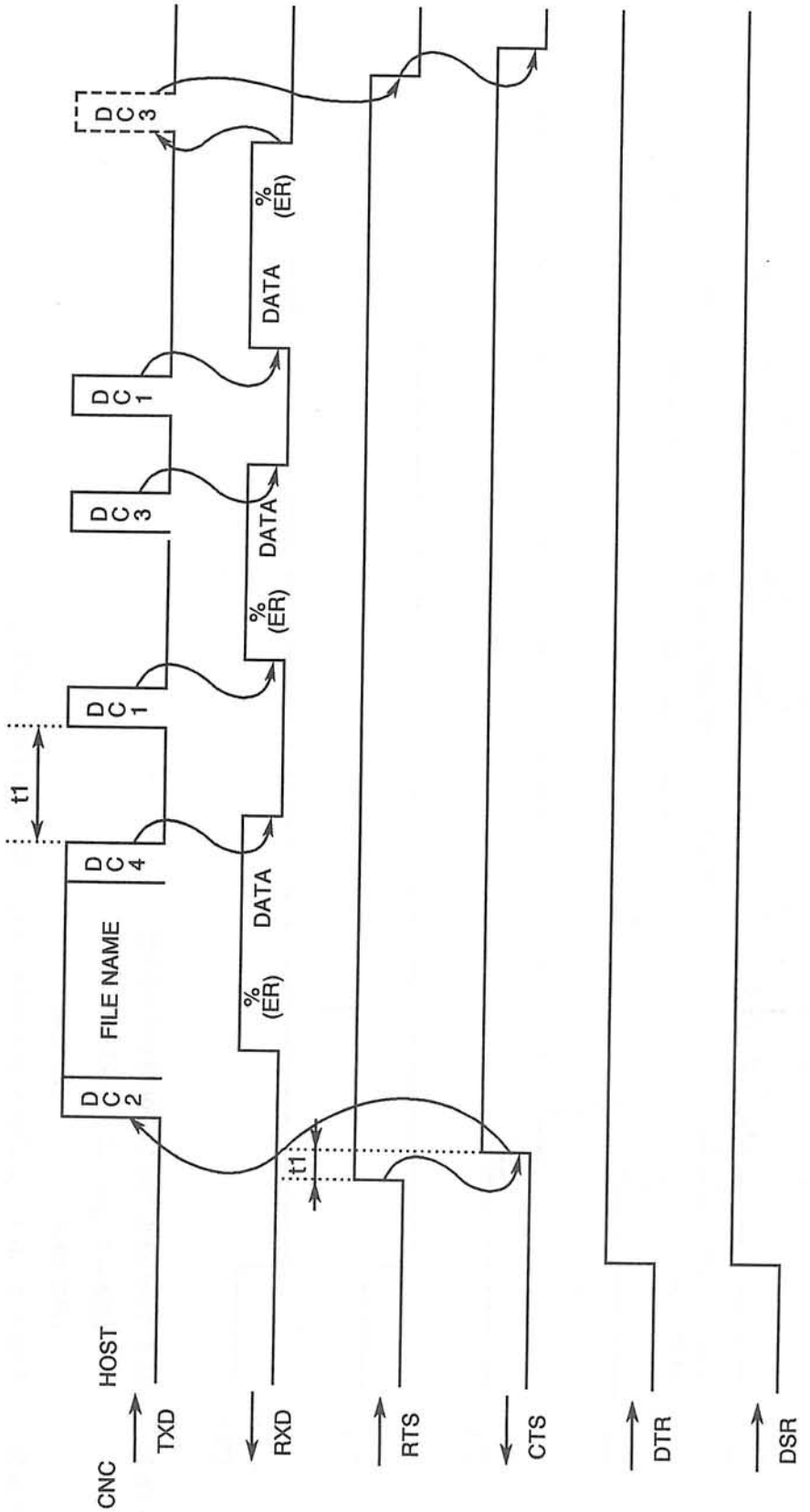
Setting range : 1-9999 secs

Initial value : 10 secs.

Note 2: *Parameter setting determines whether or not this DC code is used.*

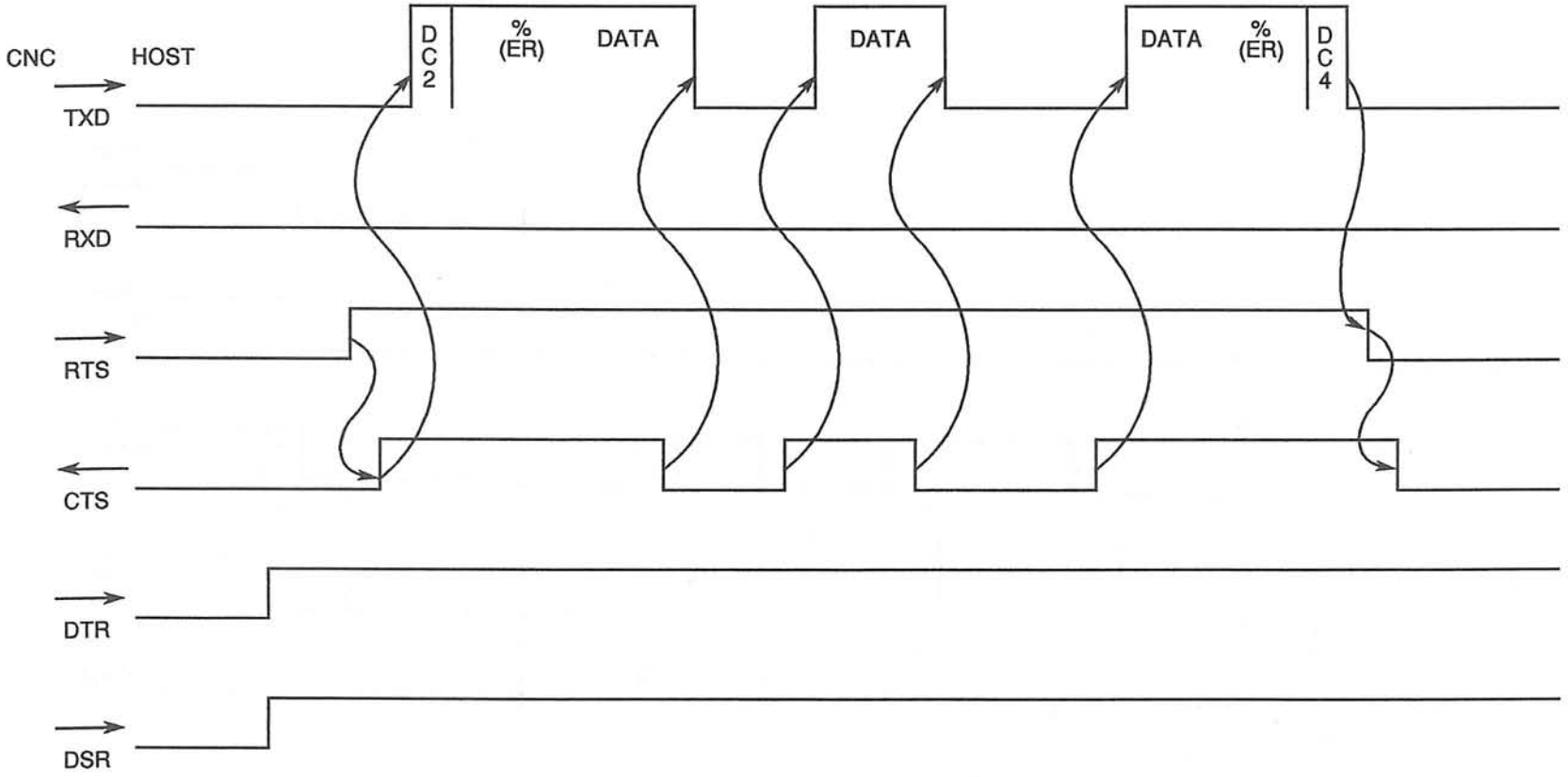
Note 3: *The transmission data must begin with either a "%" or "ER" code, and must also end with a "%" or "ER" code.*

(2) When File Name is Designated:

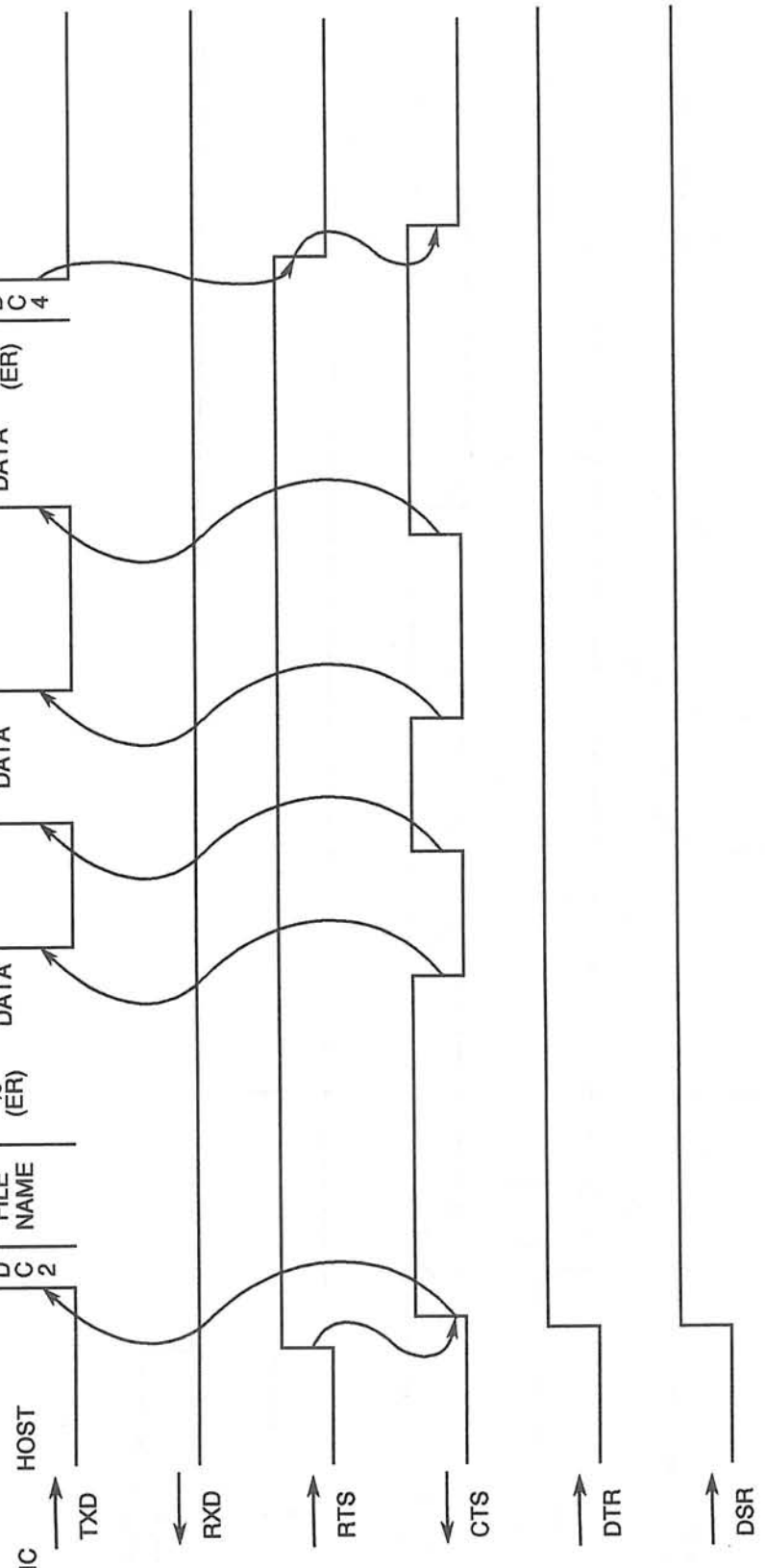


7-4. Data Transmission

(1) When File Name is not Designated:



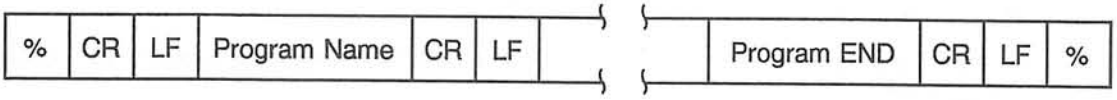
(2) When File Name is Designated:



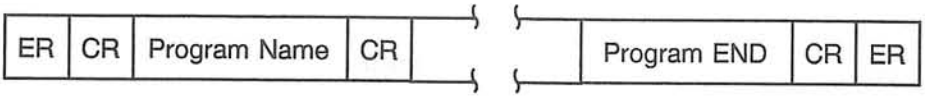
8. Data Format

8-1. Input Format

<For ISO Code>



<For EIA Code>



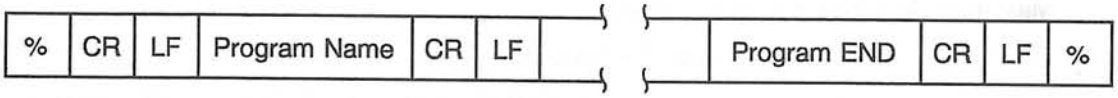
- The transmission data must begin with a "%" or "ER" code, and must also end with a "%" or "ER" code. If the data does not begin with either a "%" or "ER" code, all the data (from the very beginning) will be processed as significant data, and the initial "%" or "ER" code which appears will be processed as the END code.
- When ISO code is used, an [LF] or [CR LF] code should be designated at the end of each record within the Program. Switching between the "[LF] only" and the "[CR LF]" format is possible (parameter setting).
- When EIA code is used, a [CR] code should be designated at the end of each record within the Program.

Note 1: *The continuing transfer of a subprogram following a main program is possible only during batch transfer operations. In such cases, a breakpoint code should not be used between the main program and the subprogram, or between two subprograms.*

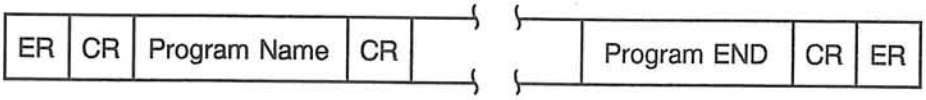
Note 2: *The maximum record length is 156 bytes.*

8-2. Output Format

<For ISO Code>



<For EIA Code>



- When ISO code is used, the output will begin with a [CR LF] or [% LF] code. When EIA code is used, output will begin with a [ER CR] code.
- The machining program data will be output following the Program Name (No.).
- When ISO code is used, the output will end with a [CR LF] or [% LF] code. When EIA code is used, output will end with an [ER CR] code.

9. Specifications

9-1. RS-232C Interface

(1) Communication Format

Asynchronous Transmission

With this format, data is sent by preceding each character with a start bit and following it with a stop bit.

1 character consists of the ① START Bit (1 bit), ② Information Bit (8 bits), ③ Parity Bit (1 bit), and the ④ STOP Bit (2 bits), as shown in Fig.1 below.

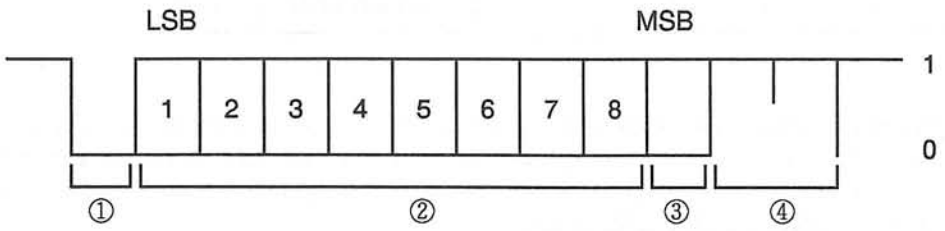


Fig. 1 Bit Configuration

(2) Baud Rate (BPS)

110, 300, 600, 1200, 2400, 4800, 9600, 19200, 38400, 89300

(3) Data Configuration

START Bit (1 bit)

Data Bits (8 bits)

Parity Bit ("1 bit" or "none" → selected by parameter setting)

STOP bits ("1 bit" or "2 bits" → selected by parameter setting)

(4) Parity Check (Character Parity)

"Odd/Even parity" or "no parity" (selected by parameter setting).

(5) Maximum Data Transmission Distance

RS-232C : Maximum cable length of 15 meters.

RS-422 : Maximum cable length of 1200 meters (an RS-232C - RS422 converter is used)

Fiber Optics : Maximum cable length of 2000 meters (a Fiber Optics - RS232C converter is used)

(6) RS-232C Signal Level

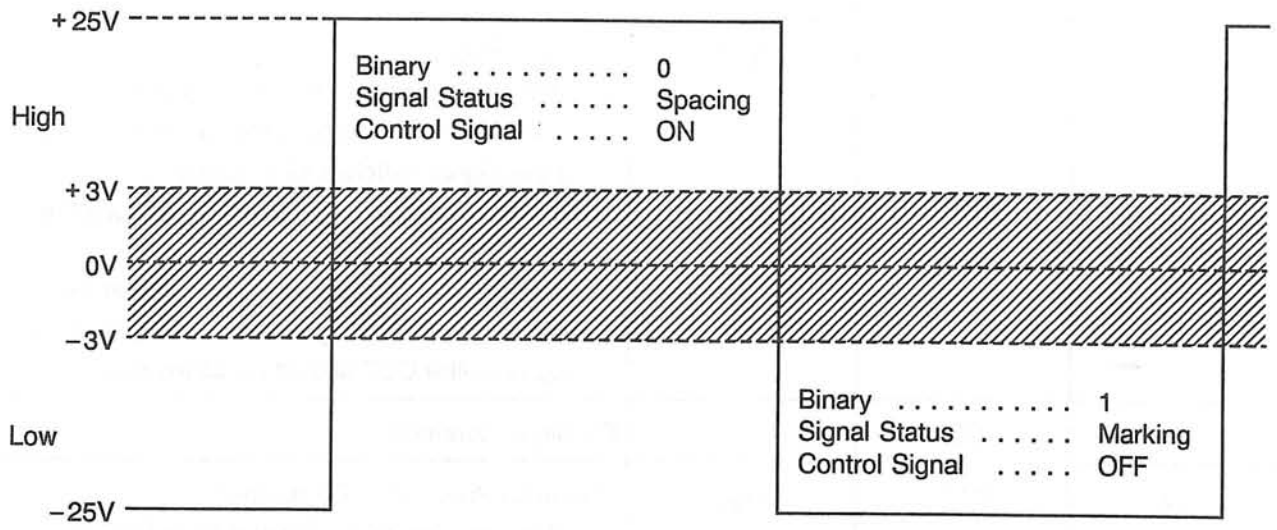


Fig.2 RS-232C Signal Level

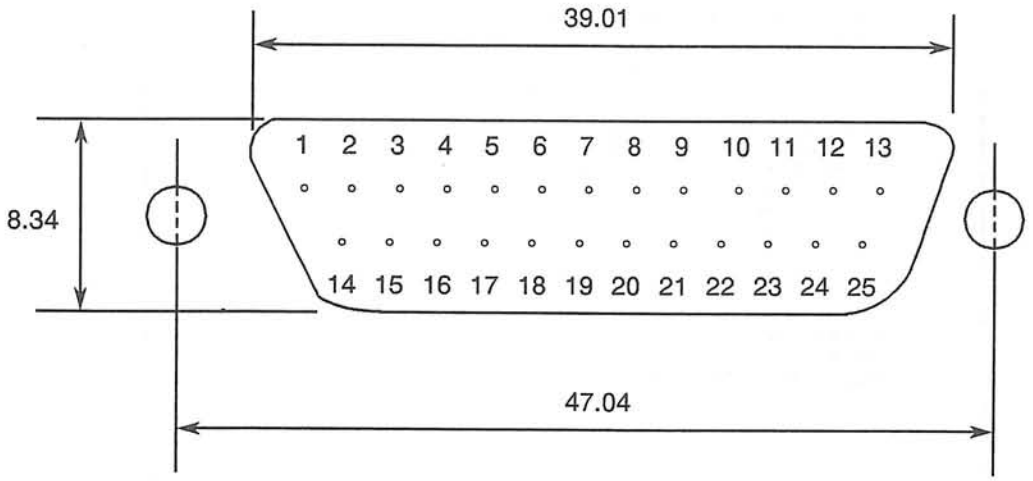
(7) Signal Descriptions

Pin No. (DB25)	Signal Name	Signal Direction	Description
1	FG	–	Safety ground
2	TXD	Output	Transmission Data Data line from OSP to external device
3	RXD	Input	Transmission Data Data line from external device to OSP
4	RTS	Output	Transmission Request Switches ON when transmission or reception begins. It then remains ON.
5	CTS	Input	Transmission Possible When this signal is OFF, no data will output from the OSP. This signal is used for BUSY/READY control. When this signal is not used, the RTS signal at the NC should be connected.

Pin No. (DB25)	Signal Name	Signal Direction	Description
6	DSR	Input	Data Set Ready This signal indicates that a "communication enabled" status exists at the external device. If this signal switches OFF during communications, an error will be activated at the OSP. BUSY/READY control is not possible at this signal. When this signal is not used, the DTR signal at the OSP should be connected.
7	SG	–	For signal connection
20	DTR	Output	Operation Preparation Completed This signal switches ON when the OSP is ready for operation. When this signal is OFF, any data which is transmitted to the OSP will not be received.

(8) Connector Layout

OSP5020M/OSP500M-G Connectors DB25S
(Female)



DB25S
(Female)

1	2	3	4	5	6	7	8	9	10	11	12	13
FG	TXD	RXD	RTS	CTS	DSR	SG	CD					

14	15	16	17	18	19	20	21	22	23	24	25
						DTR					

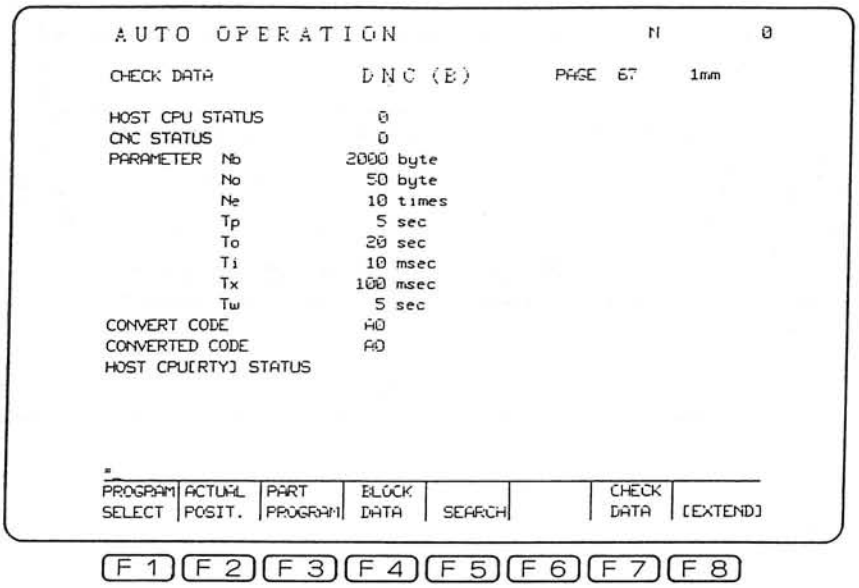
External
Device

Nihon Koku Deshi Connector DB-25S Female (standard type shell)
Lock D20418-J (hexagonal type)

10. CRT Screens

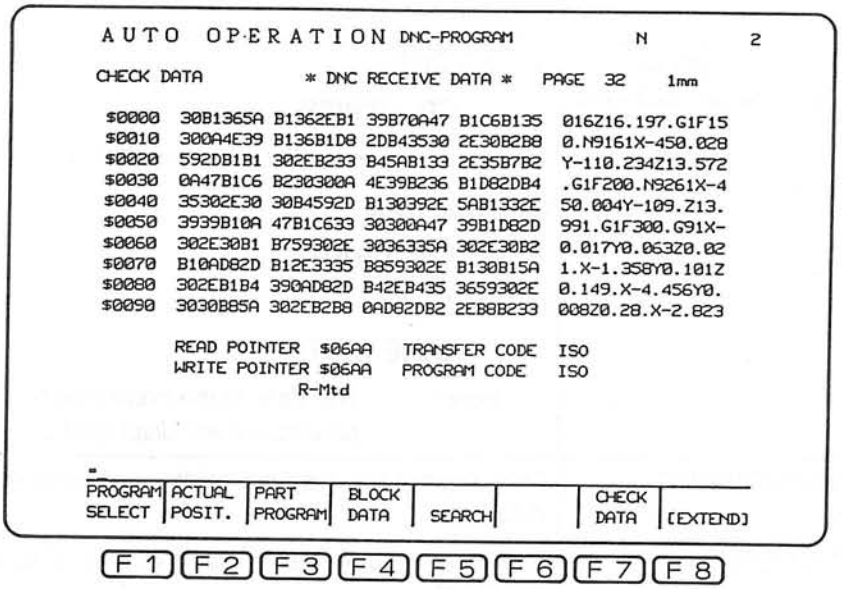
10-1. Check Screens

(1) Display the communication status and parameters on the CHECK DATA screen. (This screen is available only for protocol A.)



No.	Name	Description
①	HOST CPU STATUS	Displays the second byte data in [SET] command. Select from CHA: and CHB: Set "CHB:". Initial setting : CHA:
②	CNC STATUS	Displays the second byte data in [SAT] command. 0: Not ready status 1: CNC in normal status 2: Buffer operation status 3: Alarm status
③	PARAMETER Nb - Tw	Displays contents of [SET] and [SAT] commands from 9th to 40th bytes.
④	CONVERT CODE/CONVERTED CODE	Displays code conversion designation for NC programs.

(2) Display the RECEIVE DATA on the CHECK DATA screen.

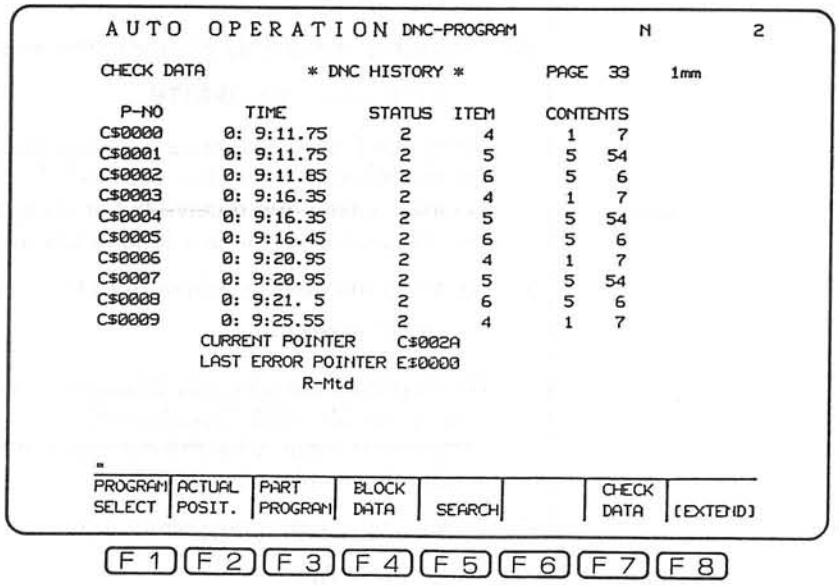


The list of the data received from the host computer is displayed.

No.	Contents	
①	The following four methods can be used to change the display pointer.	(a) To position the display pointer at "0" CP **WRITE** (b) To position the display pointer at the required position CP Pointer-value **WRITE** Designate the pointer value in hexadecimal number ($ symbol is not necessary). When the designated value causes the pointer to move outside the receive buffer area, the pointer is replaced with the pointer of the last area of the receive buffer. (c) To return the display pointer from the current position CP ;B **WRITE** This replaces the currently displayed pointer with the "current display pointer - $80 (hexadecimal)". If the above indicated command is input while the current display pointer value is 0 or smaller, it is replaced with "0". (d) To advance the display pointer from the current position. CP ;N **WRITE** This replaces the currently displayed pointer with the "current display pointer + $80 (hexadecimal)". If this replacement causes the pointer to move beyond the receive buffer, the pointer is replaced with the pointer of the last area in the receive buffer.

No.	Contents	
	The display right character coding system can be changed as required.	(a) ISO Code CP ;I **WRITE** When the power is turned on, this code is selected. (b) ASCII Code CP ;S **WRITE** (c) EIA Code CP ;E **WRITE** *Note: The data of the code which cannot be expressed in characters are displayed in " . " .*
②	READ POINTER/WRITE POINTER	This displays the read-out pointer and write pointer of the received data.
③	TRANSFER CODE	The code used for transmission of data other than NC program is displayed.
④	PROGRAM CODE	The code used for transmission of NC program is displayed.

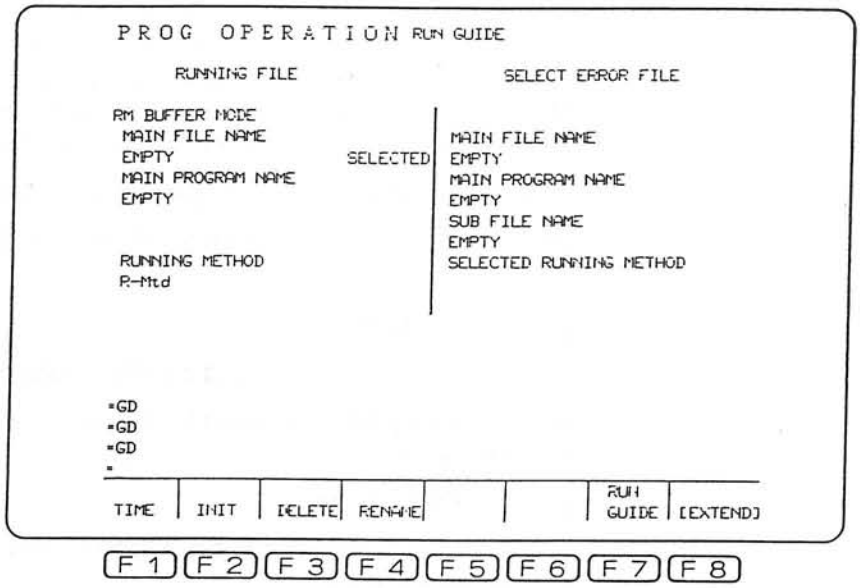
(3) Display the DNC HISTORY page on the CHECK DATA screen.



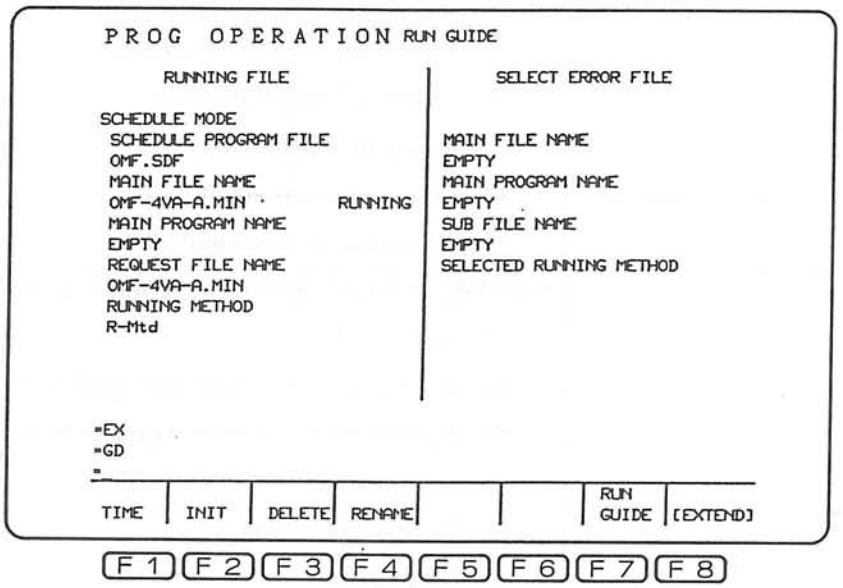
No.	Description	
①	P-NO	The entry pointer of the DNC HISTORY saving area is displayed. Character C at the beginning of the pointer indicates that the information displayed is the contents in the information saving area for normal operation. Character E at the beginning of the pointer indicates that the information displayed is the contents in the information saving area for the occurrence of abnormality. The display area can be changed in the following method. (a) To display the information for the normal DNC HISTORY saving area. CP ;C **WRITE** When the power is turned on, this display area is selected. (b) To display the information for the abnormal DNC HISTORY saving area. CP ;E **WRITE** The display pointer can be changed in the same manner as explained in (1).
②	TIME	The time elapsed from the power on is displayed in hour: minute: second.
③	STATUS	The communication operation status is displayed in a coded number. 0: Initial state 1: Not ready for communication 2: Ready for communication 3: Communication in execution 4: Communication alarm 5: Communication system alarm
④	ITEM	The DNC HISTORY kind is displayed in a coded number. 1: Command 2: Notification for correct ending of command 3: Notification for abnormal ending of command 4: Internal request 5: Transmission message 6: Line event
⑤	CONTENTS	Details of DNC HISTORY kind is indicated in a coded number. For details, refer to Appended Table.
⑥	CURRENT POINTER	The latest information saving point for normal DNC HISTORY
⑦	LAST ERROR POINTER	The latest information saving point for abnormal DNC HISTORY

10-2. Run Guide Screens

(1) BUFFER MODE



(2) SCHEDULE MODE



11. Error Messages

2611 DNC can not transmit

An attempt is made to transfer a program in batch although DNC communication is impossible.

Character-string : None

Code : 1 ..... An attempt to transfer a program during communication alarm

: 2 ..... DNC communication OFF

: 4 ..... Operation preparation not completed

2622 DNC transmitting

A communication error has occurred during batch transmission of the NC program and data transmission has terminated abnormally.

Character-string : "W28","W29","W30", or "W39", warning massage number of the communication alarm explained later is displayed.

Code : The completely identical to the code which is displayed for each warning message.

For details, refer to 12-1. "Communication alarm".

2623 Remote operation mode

An attempt is made to execute the following processing during buffer operation mode ON in DNC-B specification or during buffer operation. (common to protocol A and B)

Character-string : None

Code : 1 ..... And attempt to transfer program in batch (impossible only during buffer operation)

: 3 ..... Main program name or subprogram file name is designated when selecting a program in NC program transmission method (2) during buffer operation.

2629 Check point setting

Setting error of check points

Character-string : None

Code : 1 ..... Data other than a pointer value (hexadecimal) and an option is input.

12. Alarm Message Chart

12-1. Communication Alarm

Communication alarm is the alarm which has occurred in communication link (protocol A) between the host computer and the OSP and does not have influence on NC operations. Once this alarm has occurred, it can be reset either by the reception of [SYN] command from the host computer or switching **COMMUNICATION ON/OFF** switch of the CNC from OFF to ON.

W28 DNC-B communication error

A communication error has occurred in communication protocol A of the host computer.

Character-string : None

Index : None

Code : YZZ (hexadecimal)

Y: Status of the OSP when communication error has occurred

- 0 ..... Not ready for operation
- 1 ..... OSP in normal operation state
- 2 ..... Buffer operation state
- 3 ..... Alarm state

ZZ: Cause

- 0 ..... An alarm is activated before the "Cause" code is set. Refer to 13-1. "DNC HISTORY Contents Code Table" in this section.
- 1 ..... Check sum error has occurred by Ne times in succession, set by parameter.
- 2 ..... Parity error has occurred by Ne times in succession, set by parameter.
- 3 ..... --
- 4 ..... No answer signal
- 5 ..... DSR signal OFF
- 6 ..... Ineffective answer signal
- 7 ..... --
- 8 ..... Reception buffer overflow
- 9 ..... Reception from the host computer while not in answer standby state

W29 DNC-B message format error

An error in received message in protocol A. This alarm is notified to the host computer as reception of unexpected message.

Character-string : None

Index : None

Code : None ..... Message length has exceeded 72 bytes (or 4000 bytes).

W30 DNC-B communication system abort

An unrecoverable error has occurred in protocol A. This alarm cannot be cleared unless power is turned off and then on.

Character-string	:	None	
Index	:	None	
Code	:	10	Overrun error has occurred.
		11	Framing error has occurred.

W39 CCP-CPU illegal

An alarm occurred with the CCP-CPU.

This alarm cannot be cleared unless power is turned off and then on.

Character-string	:	None	
Index	:	None	
Code	:	1	Memory parity or write error is detected in CCP memory test.
		2	The CCP control program cannot be loaded correctly.
		3	The CCP control program cannot be started correctly.
		4	The CCP communication channel opening does not end correctly.
		5	A CCP self-diagnosis error is detected.
		6	An exception error is detected with the CCP-CPU.
		7	An error is detected in the CCP control program operation. (task start control error)
		8	An error is detected in the CCP control program operation. (real time task loop error)
		9	An error is detected in the CCP control program operation. (switchover control error for time sharing task)

12-2. NC Alarm

ALARMC

998 DNC-B Buffer operation change

An attempt is made to switch the buffer operation ON/OFF state by a command during operation (for protocol A).

Character-string : None

Index : None

Code : None

927 DNC start condition

Conditions for starting operations are not satisfied.

Character-string : None

Index : None

Code : =1 ..... Not-ready state or communication error state in buffer operation by DNC-B specification, protocol A

: =2 ..... Schedule program has been being executed in buffer operation in DNC-B specification.

: =5 ..... Main program buffer size is less than 12 m for buffer operation

ALARMB

652 DNC-B Remote operation

An error has occurred during transmission of a program.
(Alarm message is not displayed at the console line.)

Character-string : When communication alarm has occurred, the warning number indicated in Section 12-1.

When an error has occurred, four digits of the error number

Example:

2504 No tape data
2527 Program end code
2530 Program buffer overflow
2541 Record buffer overflow

Index : None

Code : The completely identical to the code which is displayed for each warning message when the communication alarm has occurred. For details, refer to 12-1. "Communication alarm".

Related error code when an error has occurred

13. Appendix

13-1. DNC HISTORY Contents Code Tables

(1) Kind = 1 (Command)

Upper Code	Lower Code
Command kind	Appended data length
1: Open	
2: Close	
3: Request for NC program	File name length
4: Call of NC program	
5: Transmission of NC program	File name length
6: Notification of NC reset	
7: Notification of NC alarm	

(2) Kind = 2 (Notification for correct ending of command)

Upper Code	Lower Code
Normal end code	Not used = 0
0: Processing completion	
1: Request for NC program	
2: Request for NC program transmission	
3: NC reset	
4: NC alarm	
5: NC reset/alarm	
6: Forced termination	

(3) Kind = 3 (Notification for abnormal ending of command)

Upper Code	Lower Code
Abnormal end code	Error detail code
1: Temporary error	1: Check sum error
2: Permanent error	2: Parity error
3: NC reset	3:
4: NC alarm	4: No answer signal
5: Forced termination	5: DSR OFF
6: Buffer clear	6: Ineffective answer signal
7: Initialization	7: Data length overflow
8: NC reset/alarm	8: Reception buffer overflow
	9: Data reception in other than data reception standby state
– 1: Command issuing sequence error	10: Overrun error
– 14: Data error	11: Framing error
– 15: Parameter error	
– 16: Command error	– 59: Appended data length range error
	– 60: Command issuing error
	– 62: Error in selecting with/without significant data
	– 63: Command kind selection error
	– 64: Command request selection error
	– 91: Answer error for SAT
	– 92: NC program end code selection error
	– 93: EIA/ASCII conversion error
	– 94: NC program length range error
	– 95: NC program record end code selection error
	– 96: NC program data code selection error.
	– 112: Answer signal monitoring time range error
	– 128: SAT, SET command parameter selection error

(4) Kind = 4 (Internal request)

Upper Code	Lower Code
Macro status 0: Initial state 1: Awaiting macro start 2: Awaiting request for reception 3: Awaiting request for transmission 4: Awaiting reception of the first block *4 Only for protocol A	Macro event 1: Open 2: Close 3: Request for reception 4: Request for transmission 5: NC alarm 6: NC reset 7: Time-out 8: request for forced termination 9: 10: Code conversion error

(5) Kind = 5 (Transmission message)

Upper Code	Lower Code
Message kind	Command data length
a) Protocol A	
1: GTD	
2: DAT	DAT : data length/16
3: RST	Others : data length
4: ALM	
5: SAT	
6: RTD	
7: RTY	
8: SYN	
9: RDY	
10: WAT	
11: EOD	
12: ARS	
13: AAL	
14: CLB	
15: SET	
b) Protocol B	Data length/16 (incl. DC code)
– 127: DC1	
– 126: DC3	
– 125: Data	
– 124: SYN	
– 123: NAK	

(6) Kind = 6 (Line event)

Upper Code	Lower Code
Line status a) Protocol A 0: Awaiting answer signal for initialization 1: Awaiting end of initialization 2: Data being received 3: Awaiting reception after transmitting DAT/RTD 4: Awaiting reception after transmitting WAT 5: Awaiting reception after transmitting SAT 6: Awaiting reception after transmitting ALM 7: Awaiting reception after transmitting RST 8: Awaiting reception after transmitting EOD b) Protocol B 0: Reception being requested 1: Data being received 2: Transmission being requested 3: Data being transmitted 4: Alarm signal being requested 5: Reset signal being requested	Line event 1: Reception of SYN 2: Reception of RDY 3: Reception of ARS 4: Reception of AAL 5: Reception of CLB 6: Reception of SET 7: Reception of DAT 8: Reception of EOD 9: Reception of WAT 10: Reception of GTD 11: Reception of RTY 12: Reception of data other than answer signal 13: No answer back 14: DSR signal off 15: Transfer error 1 16: Transfer error 2 1: Normal character 2: Transfer error 1 3: Transfer error 2 4: No answer signal 5: CTS signal on 6: CTS signal off 7: DSR signal off 8: Forced termination

13-2. Programming Supplementary Information

The following is supplementary information to "OSP5020M Programming Manual".

For general information for programming, refer to the above indicated manual.

13-2-1. Schedule Program (PSELECT Block)

The command format to execute the schedule program in the buffer operation ON status for the DNC-B specification is indicated below. ([ ] is omissible.)

PSELECT [fm] [(fd)],, [; S], [n]

fm : | main-program-file-name | . | extension |

- Designate a character-string of up to 16 characters beginning with an alphabet.
- (Alphabets, numbers and hyphen (-) can be used.)
- The extension is MIN.
- The main program file name designated in this step is registered as the file name for buffer operation. If a main program file name is omitted, A. MIN is used.
- To designate a device name in the program file name, designate the host computer (DNC:).

(fd) : Request main program file name data

File-address-character-string requested

Character-string of up to 30 characters; following characters can be used:

0 - 9, A - Z, !, ", #, $, %. &, ', <, >, (,), @, ?, ¥, , [,], _, ;, :, ,, +, −, *, /, =, ., **SP**

Transmission of the NC program of the request-file-name designated in this step is requested to the host computer. If the request-file-name is omitted, the main program file name is designated.

;S : Option designation

In buffer operation mode in the DNC-B specification, only operation S (search is not made for subprograms) is possible.

n : Number of repetitions

The number of repetitions is designated with address 'Q'. Designation range is 1-9999 and omission is regarded as "1". Designation of a value outside the indicated range causes an alarm.

Note: In the PSELECT block of the schedule program in buffer operation mode for DNC-B specification, designation of a main program name and subprogram file name is not possible.

SECTION 5 SEMI-SYNCHRONIZED OPERATION OF MACHINING CENTER WITH DIGITIZER (TD)

1. Overview

(1) The semi-synchronized operation function links the OSP5020M/OSP500M-G and the OSP5000N to allow semi-synchronized operation of the machining center and the TD digitizer.

(2) The semi-synchronized operation means that the digitized data obtained during scanning processing carried out by the TD digitizer is transferred to the NC machine so that machining a workpiece is machined in parallel with scanning while scanning is being conducted.

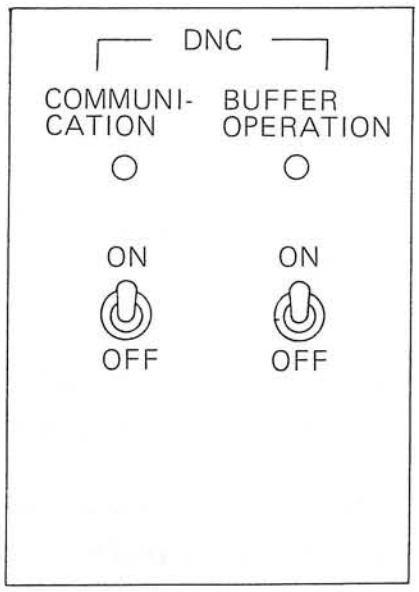
(3) In this mode of operation, lag between the machining position and scanning position is less than one line of scanning.

(4) At the OSP5020M/OSP500M-G, delay is controlled to be less than 1K bytes in character length.

(5) In addition, the DNC operation in the ordinary DNC-B high speed RM buffer method is also possible. For details of the high-speed RM buffer method, refer to the instruction manual for "DNC-B high speed RM buffer method". Operation mode can be selected between the "semi-synchronized" and the "DNC-B high-speed RM buffer" by setting proper parameter; the DNC operation panel is used in common to these two modes.

(6) The information in this section covers only the NC related information. For the operations of the TD digitizer, refer to the instruction manual for the TD.

2. DNC Operation Panel



(1) **COMMUNICATION ON/OFF** switch

(a) COMMUNICATION ON

Turning on power supply after placing the switch in the ON position establishes the communication mode between the TD and the host computer. The COMMUNICATION lamp lights up.

(b) COMMUNICATION OFF

Turning on power supply after placing the switch in the OFF position disconnects communication line between the TD and the host computer.

Once power supply has been turned off, changing the switch setting position has no effect and the established state cannot be changed.

Turn the switch in the OFF position once and then back to the ON position to recover the communication alarm state* from the NC. Even in this case, the previously established communication ON state is not changed.

* The state in which alarm W28 or W29 is displayed.

(2) **BUFFER OPERATION ON/OFF** switch

(a) BUFFER OPERATION ON

By setting the switch in the ON position with the TD COMMUNICATION ON state, buffer (semi-synchronized) operation is enabled and the BUFFER OPERATION ON lamp lights up.

(b) BUFFER OPERATION OFF

Buffer (semi-synchronized) operation is disabled.

If the switch is set at the BUFFER OPERATION OFF position during semi-synchronized operation, the buffer operation enabled state is continued until the buffer operation has completed.

However, if communication alarm occurs, ON/OFF state of the buffer (semi-synchronized) operation is controlled by the setting of the **BUFFER OPERATION ON/OFF** switch setting at the OSP operation panel.

Note: *The DNC operation panel is used in common to the DNC-B high-speed RM buffer operation and the semi-synchronized operation.*

In the high-speed RM buffer operation mode (NC optional parameter (bit) No. 37, bit 2: 0), the operation panel is used for the high-speed RM buffer operation, and in the semi-synchronized operation mode (NC optional parameter (bit) No. 37, bit 2: 1), it is used for semi-synchronized operation.

3. Semi-synchronized Operation

Semi-synchronized operation is possible in the automatic mode while the BUFFER OPERATION ON lamp at the DNC operation panel lights up.

Semi-synchronized operation function varies depending on the NC program transmission method (with or without file name designation).

Operation Type	Program Type	Operation Method	Buffer Operation (DNC mode: ON, Buffer operation: ON)				Normal Memory Operation	
			Without File Name Designation		With File Name Designation			
			Bubble Memory	TD	Bubble Memory	TD	Bubble Memory	TD
Manual Mode Operation	Main Program	A	×	×	○	×	○	×
		B	×	×	○	×	○	×
		S	×	○	○	×	○	×
		M	×	×	○	×	○	×

Note 1: "○" indicates "operation possible" and "×" indicates "operation impossible".

Note 2: Operation Methods:

A ..... Normal operation
B ..... Large-volume operation
S ..... Subprogram and branch function are not checked in operation method B.
M ..... Multi-volume operation

Note 3: In the semi-synchronized operation mode, operation using a schedule program is not possible.

The following operations are possible depending on "with" and "without" file name designation.

(a) Without file name designation ..... Buffer operation of NC program (digitized data)

(b) With file name designation ..... Buffer operation of NC program

Memory mode operation based on programs stored in bubble memory

3-1. Operation Procedure for Semi-synchronized Operation

(1) Selecting the Semi-synchronized Operation Mode

Both the DNC-B high-speed RM buffer operation and the semi-synchronized operation use the same DNC operation panel. Selection of the operation mode is possible by the setting for NC optional parameter (bit) data (No. 37, bit 2).

The procedure to select the semi-synchronized operation mode is indicated below:

① Press the **PARAMETER** key.

② Press the function key [F7] (ITEM) to display the NC optional parameter (bit) data setting screen.

③ Move the cursor to No. 16 bit 7 and set "1".

④ Press the page key to display the screen which contains No. 37.

⑤ Move the cursor to No. 37 bit 2 and set "1".

The setting should be "0" to select the high-speed RM buffer operation mode.

⑥ Press the backup function key to back up the data. After that turn off the power once and turn it back on again.

Note: Once the parameter is set, it is not necessary to repeat the same setting operation when the power is turned on or off. Setting is necessary only when the operation mode should be changed.

(2) Starting the Semi-synchronized Operation

Operation Procedure:

① Place the **DNC COMMUNICATION ON/OFF** switch at ON.

② Turn on power supply.

③ Select the AUTO OPERATION mode by pressing the **AUTO** key.

④ Place the **BUFFER OPERATION ON/OFF** switch at ON.

⑤ Press the **CYCLE START** button. (PROGRAM SELECT operation is not necessary.)

3-2. Memory Mode Operation Using A Program Stored in Bubble Memory

The operation method is quite the same as used for normal memory mode operation.

Only the difference is the designation of the bubble memory (BB1:) as the device name when designating the main program file name.

3-3. Precautions

(1) Subprograms only registered in the library program can be used.

(2) Size of part program has no limit for program transfer.

(3) Only main programs can be transmitted.

If subprograms and/or other main programs continue following the M02 code of the main program, do not forget to set the "M02 reset output used" state by the parameter.

(4) At the instant the **CYCLE START** button is pressed, transfer of the NC program (digitized data) begins (GTD sent for protocol A) and then the program execution begins.

(5) Change of parameter setting for file name used or not-used status becomes active when power supply is turned off and then on again.

(6) Even in the semi-synchronized operation mode, intervention of manual operation is possible as usual.

(7) After the buffer operation mode is turned off, it is always necessary to select a program. Unless a program is selected, cycle start is impossible.

(8) Regardless of the on or off state of the buffer operation mode, the state does not change even if the setting for the **BUFFER OPERATION ON/OFF** switch. This is also applied when a program is being transmitted for buffer operation.

If the buffer operation mode is changed using a command, in the "buffer operation mode on" always causes an alarm regardless of the current mode.

(9) Batch transfer of NC programs using the DNC function is not possible. (Ignored, "No Command" message is displayed.)

(10) The RS command (restart search) cannot be used. (Ignored, "No Command" message is displayed.)

(11) Schedule program selection is not possible. (Ignored, "No Command" message is displayed.)

(12) After the execution of MDI mode operation, semi-synchronized operation is not possible unless the NC is once reset.

Interruption by MDI mode operation during buffer mode operation may be executed in the same manner as normal operation intervention.

(13) Automatic program selection when power is turned on:

BUFFER OPERATION ON . . . . . Program selection is not executed.
BUFFER OPERATION OFF . . . . . Program selection is executed.

(14) In the semi-synchronized operation mode, three blocks of commands are buffered.
In the cutter radius compensation mode, eight blocks of commands are buffered.

4. Parameters

4-1. Parameters Which Cannot be Set At OSP

The parameters indicated below cannot be set at the NC and setting is possible only by using the SET command from the TD.

(Parameters in the table below, codes after conversion (45th - 48th bytes of SET command))

Parameter	Function	Unit	Range	Initial Value when Power is Turned On
Nb	Minimum buffer area of CNC when transmitting [GTD]	bytes	1 - 60	30
No	Maximum overrun amount in reception	bytes	2 - 2000	50
Ne	Number of retries allowed when transmission error is detected	times	0 - 100	10
Tp	[SAT] transmission polling time for DNC, or response time-out duration by CNC for host computer	sec	1 - 999	5
To	Response time-out duration by host computer (TD) for CNC	sec	1 - 999	20
Ti	Minimum time interval between transmission bytes	msec	0 - 10	10
Tx	Minimum switching time from reception to transmission	msec	0 - 100	100
Tw*	Standby time for reception of [WAT] used by both of CNC and host computer (TD); upper limit value is Tp or To, whichever smaller.	sec	0 - Tp 0 - To (Whichever smaller)	5

*Tw: Communication is executed in the following manner when "Tw" value exceeding the specified range is set by the host computer (TD) using the SET command.

(1) When Tw value > Tp value

In the remote buffer mode operation or batch transmission mode, the OSP sends the GTD command to the host computer (TD) after waiting for "Tw" when it received the WAT command from the host computer (TD). The SAT command is not sent out during this period.

(2) When Tw value > To value

If program data to be transmitted is not ready in the NC program transmission from the OSP to the host computer operation, the OSP sends the WAT command after and elapse of "Tp". If the host computer sends the GTD command in response to this after waiting for "Tw", the OSP regards that, after an elapse of "To", there is no answer to the WAT command. In this case the communication error occurs.

4-2. Parameters Which Can be Set At OSP

The parameters indicated below can be set at the NC

Some parameters are required to be changed frequently during operation while some are not changed once they have been set. The OSP differentiates these two types of parameters to allow easy setting of parameters. The former is assigned to NC optional parameter (long word), (word) and (bit), and the latter to the communication parameter (dedicated screen).

(1) NC Optional Parameter (bit)

(a) Bit 0 of No. 1

Coding system of NC programs

1: ISO
0: EIA

Initial setting : 1

(b) Bit 2 of No. 1

TV check, creation designation

This parameter determines whether or not the TV check, creation is made for the NC program (digitize data??) to be received or transmitted--a space is inserted before LF (or CR LF) so that a total number of characters in a record including the LF (or CR LF) will be even, or whether the record contains an even number of characters is checked.

1: TV check, creation is executed.
0: TV check, creation is not executed.

Initial setting : 0

(c) Bit 3 of No. 11

End of record code of NC program data (digitized data)

1: LF
0: CR LF

Initial setting : 0

(d) Bit 3 of No. 37

Selection of operation mode - DNC-B high-speed RM buffer method or semi-synchronized operation mode

1: Semi-synchronized operation
0: DNC-B high speed buffer

Note: After changing the setting, it is necessary to turn off the power once and then turn it back on again to make the new setting effective.

(e) Bit 0 - bit 7 of No. 43 and No. 44

Data at the 41st to 44th bytes of [SAT] (CNC→TD) or [SET] (TD→CNC) command.

Bit 0 of No. 44

1: Buffer is cleared when NC is reset.
0: Buffer is not cleared when NC is reset.

Initial setting : 0

Note: Other bits have not been defined.

(2) Semi-synchronized Operation Communication Parameters

Communication parameters are displayed and set in the parameter setting mode.

Note that the communication parameter screen cannot be displayed in normal operation method. (screen selection using the ITEM function keys.) Follow the steps below to display the communication parameter screen.

① Select the PARAMETER SET mode by pressing the **PARAMETER** key.

② Key in CMPS and press the **WRITE** key. This displays the COMMUNICATION PARAMETER screen.

③ Press the page key to display the COMMUNICATION PARAMETER screen.

④ Set the required parameters.

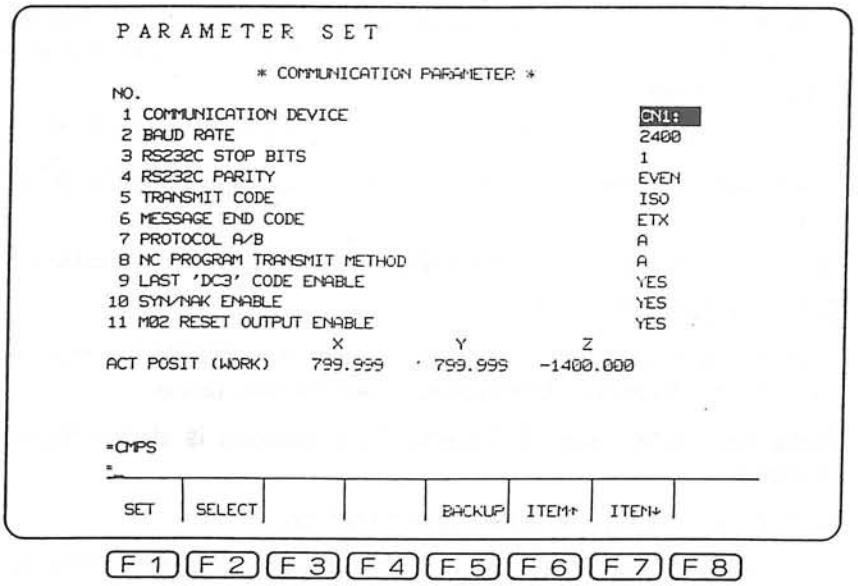
This setting is possible by using the function key (SELECT). Each time this key is pressed, contents are automatically changed within a setting range.

⑤ Press the function key (BACKUP). This operation is always required when data is set or changed.

⑥ Turn off power supply and then turn it back on.

The communication parameters become effective when power reapplied.

(3) Description of Communication Parameters



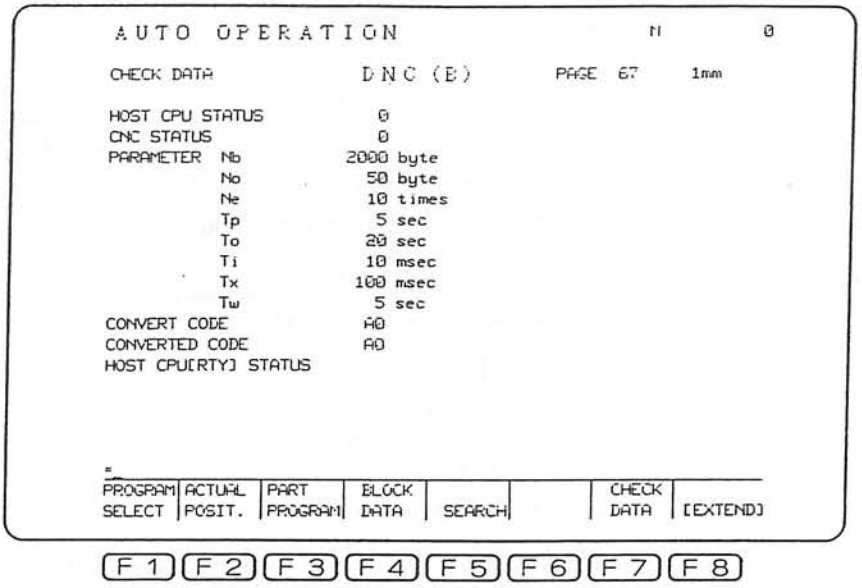
No.	Name	Contents
1	COMMUNICATION DEVICE	Designation of device name used for communication through RS-232C interface Select from CHA: and CHB: Set "CHB:". Initial setting : CHA:
2	BAUD RATE (BPS)	Designation of baud rate Select from 110, 150, 300, 600, 1200, 2400, 4800, 9600, 19200, 38400, 89300. Initial setting : 2400
3	RS-232C STOP BITS	Designation of number of stop bits Select from 1 and 2. Initial setting : 1
4	RS-232C PARITY	Designation of parity added at asynchronized communication Select from NON, EVEN and ODD. Initial setting : EVEN
5	TRANSMIT CODE	Designation of transmission code (excluding NC programs) Select from ISO and ASCII. Initial setting : ISO *Note: The transmission code is not used for protocol B.*

No.	Name	Contents
6	MESSAGE END CODE	Designation of message end code Select from ETX and CR. Initial setting : ETX *Note: The end code is not used in protocol B.*
7	PROTOCOL A/B	Protocol A only (setting is not possible.) Initial setting : A
8	NC PROGRAM TRANSMIT METHOD	Designation of "file name used" or "file name not used" state for NC program (digitized data) transmission Select from A (not used) and B (used). Initial setting : A
9	LAST 'D3' CODE ENABLE	Setting is not possible. Initial setting : YES *Note: Not used*
10	SYN/NAK ENABLE	Setting is not possible. *Note: Not used*
11	M02 RESET OUTPUT ENABLE	Designation whether or not reset signal ([RST] command for protocol A, [SYN] code for protocol B) is transmitted to the host computer (TD) upon execution of M02 and M30. Select from YES (transmitted) and NO (not transmitted). Initial setting : YES

5. CRT Screens

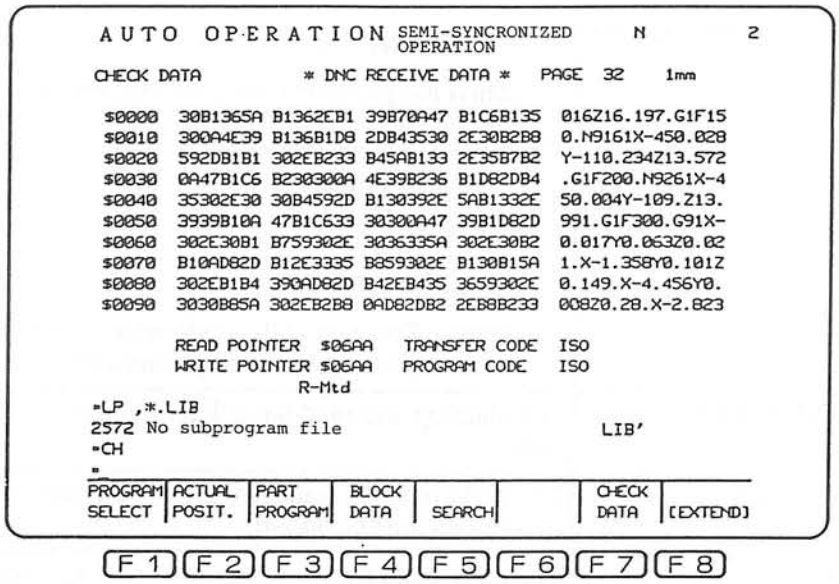
5-1. Check Screens

(1) Display the communication status and parameters page on the CHECK DATA screen.



No.	Name	Contents
①	HOST CPU STATUS	Displays the second byte data in [SET] command for TD transmission.
②	CNC STATUS	Displays the second byte data in [SAT] command for NC transmission. 0: Not ready status 1: CNC in normal status 2: Buffer operation status 3: Alarm status
③	PARAMETER Nb - Tw	Displays contents of [SET] and [SAT] commands from 9th to 40th bytes.
④	CONVERT CODE/CONVERTED CODE	Displays code conversion designation for NC programs (digitize data).

(2) Display the RECEIVE DATA on the CHECK DATA screen.

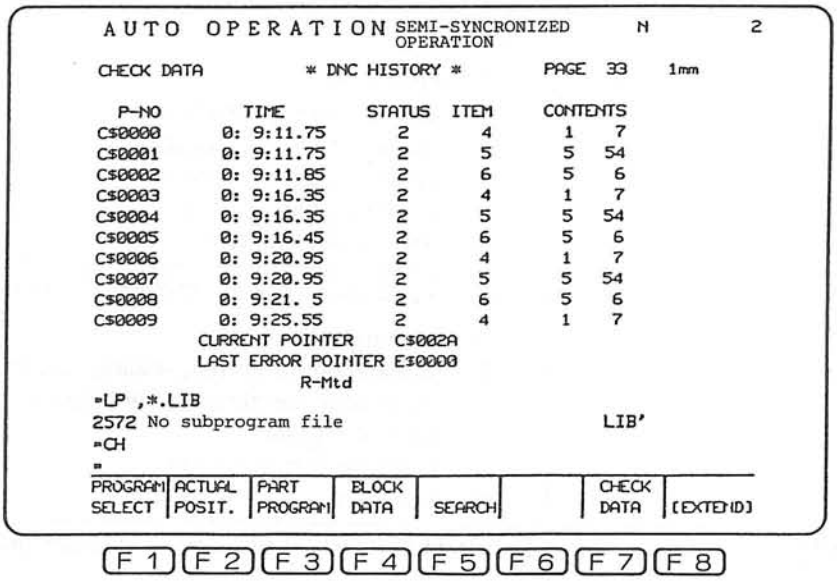


The list of the data received from the TD is displayed.

No.	Contents	
①	The following four methods can be used to change the display pointer.	(a) To position the display pointer at "0" CP **WRITE** (b) To position the display pointer at the required position CP Pointer-value **WRITE** Designate the pointer value in hexadecimal number ($ symbol is not necessary). When the designated value causes the pointer to move outside the receive buffer area, the pointer is replaced with the pointer of the last area of the receive buffer. (c) To return the display pointer from the current position CP ;B **WRITE** This replaces the currently displayed pointer with the "current display pointer - $80 (hexadecimal)". If the above indicated command is input while the current display pointer value is 0 or smaller, it is replaced with "0". (d) To advance the display pointer from the current position. CP ;N **WRITE** This replaces the currently displayed pointer with the "current display pointer + $80 (hexadecimal)". If this replacement causes the pointer to move beyond the receive buffer, the pointer is replaced with the pointer of the last area in the receive buffer.

No.	Contents	
①	The display right character coding system can be changed as required.	(a) ISO Code CP ;I **WRITE** When the power is turned on, this coding system is selected. (b) ASCII Code CP ;S **WRITE** (c) EIA Code CP ;E **WRITE** *Note: The data of the code which cannot be expressed in characters are displayed in " . ".*
②	READ POINTER/WRITE POINTER	This displays the read-out pointer and write pointer of the received data.
③	TRANSFER CODE	The code used for transmission of data other than NC program (digitize data) is displayed.
④	PROGRAM CODE	The code used for transmission of NC program (digitize data) is displayed.

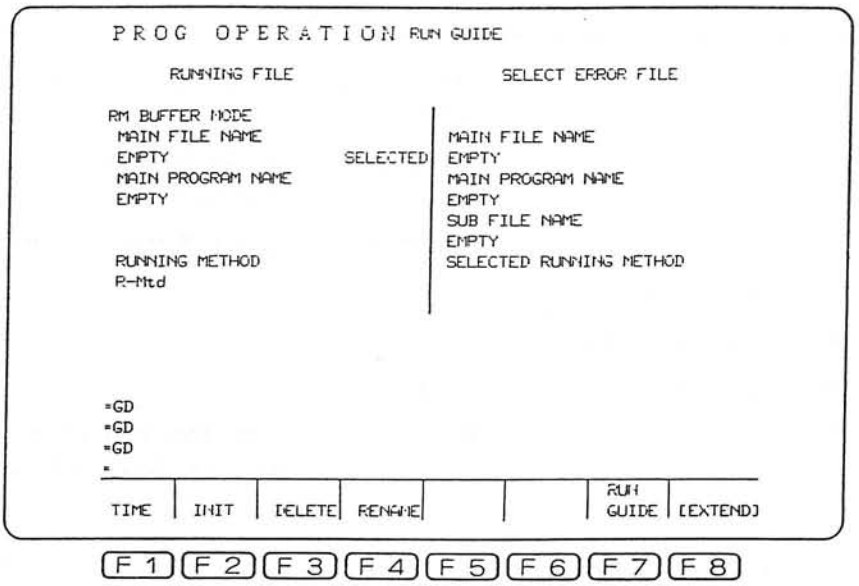
(3) Display the DNC HISTORY on the CHECK DATA screen.



No.	Description	
①	P-NO	The entry pointer of the DNC HISTORY saving area is displayed. Character C at the beginning of the pointer indicates that the information displayed is the contents in the information saving area for normal operation. Character E at the beginning of the pointer indicates that the information displayed is the contents in the information saving area for the occurrence of abnormality. The display area can be changed in the following method. (a) To display the information for the normal DNC HISTORY saving area. CP ;C **WRITE** When the power is turned on, this display area is selected. (b) To display the information for the abnormal DNC HISTORY saving area. CP ;E **WRITE** The display pointer can be changed in the same manner as explained in 5-1, (2).
②	TIME	The time elapsed from the power on is displayed in hour: minute: second.

No.	Description	
③	STATUS	The communication operation status is displayed in a coded number. 0: Initial state 1: Not ready for communication 2: Ready for communication 3: Communication in execution 4: Communication alarm 5: Communication system alarm
④	ITEM	The DNC HISTORY kind is displayed in a coded number. 1: Command 2: Notification for correct ending of command 3: Notification for abnormal ending of command 4: Internal request 5: Transmission message 6: Line event
⑤	CONTENTS	Details of DNC HISTORY kind is indicated in a coded number. For details, refer to Table 5-1.
⑥	CURRENT POINTER	The latest information saving point for normal DNC HISTORY
⑦	LAST ERROR POINTER	The latest information saving point for abnormal DNC HISTORY

5-2. Run Guide Screens



6. Error Messages

2622 DNC transmitting

A communication error has occurred during batch transmission of the NC programs and data transmission has terminated abnormally.

Character-string : "W28", "W29", "W30" or "W39"; warning message number of the communication alarm explained later is displayed.

Code : The contents completely identical to the code which is explained for each warning message.

For details, refer to 7-1 "Communication Alarm".

2629 Check point setting

Setting error of check points

Character-string : None

Code : 1 . . . . . Data other than a pointer value (hexadecimal) and an option is input.

7. Alarm Message Chart

7-1. Communication Alarm

Communication alarm is the alarm which has occurred in communication link (protocol A) between the host computer (TD) and the OSP and does not have influence on NC operations. Once this alarm has occurred, it can be reset either by the reception of [SYN] command from the host computer (TD) or switching **COMMUNICATION ON/OFF** switch of the CNC from OFF to ON.

W28 DNC-B communication error

A communication error has occurred in communication protocol A of the host computer (TD).

Character-string : None

Index : None

Code : YZZ (hexadecimal)

Y:

Status of the NC when communication error has occurred

0 ..... Not ready for operation

1 ..... NC in normal operation state

2 ..... Buffer operation state

3 ..... Alarm state

ZZ:

Cause

1 ..... Check sum error has occurred by Ne times in succession, set by parameter.

2 ..... Parity error has occurred by Ne times in succession, set by parameter.

3 ..... --

4 ..... No answer signal

5 ..... DSR signal OFF

6 ..... Ineffective answer signal

7 ..... --

8 ..... Reception buffer overflow

9 ..... Reception from the host computer (TD) while not in answer standby state

W29 DNC-B message format error

An error in received message in protocol A. This alarm is notified to the host computer (TD) as reception of unexpected message.

Character-string : None

Index : None

Code : None ..... Message length has exceeded 72 bytes (or 4000 bytes).

W30 DNC-B communication system abort

An unrecoverable error has occurred in protocol A. This alarm cannot be cleared unless power is turned off and then on.

Character-string	:	None	
Index	:	None	
Code	:	10	Overrun error has occurred.
		11	Framing error has occurred.

W39 CCP-CPU illegal

An alarm occurred with the CCP-CPU.

This alarm cannot be cleared unless power is turned off and then on.

Character-string	:	None	
Index	:	None	
Code	:	0	An alarm is activated before the "Cause" code is set. Refer to 13-1. "DNC HISTORY Contents Code Table" in this section 4.
		1	Memory parity or write error is detected in CCP memory test.
		2	The CCP control program cannot be loaded correctly.
		3	The CCP control program cannot be started correctly.
		4	The CCP communication channel opening does not end correctly.
		5	A CCP self-diagnosis error is detected.
		6	An exception error is detected with the CCP CPU.
		7	An error is detected in the CCP control program operation. (task start control error)
		8	An error is detected in the CCP control program operation. (real time task loop error)
		9	An error is detected in the CCP control program operation. (switchover control error for time sharing task)

7-2. NC Alarm

ALARM C

998 DNC-B buffer operation change

An attempt is made to switch the buffer operation (semi-syncronized operation) ON/OFF state by a command during operation (for protocol A).

Character-string : None

Index : None

Code : None

927 DNC start condition

Conditions for starting operations are not satisfied.

Character-string : None

Index : None

Code : 1 ..... Not-ready state or communication error state in semi-synchronized operation

: 2 ..... Schedule program has been being executed in semi-synchronized operation

: 5 ..... Main program buffer size is less than 12 m in semi-synchronized operation

ALARM B

652 DNC-B remote operation

An error has occurred during transmission of a NC program (digitize data).
(Alarm message is not displayed at the console line.)

Character-string : None ..... When communication alarm has occurred, a warning number explained in section 7-1.

When an error has occurred, four digits of the error number

Example:

2504 No tape data
2527 Program end code
2530 Program buffer overflow
2541 Record buffer overflow

Index : None

Code : Communication alarm : The contents completely identical to the code which is explained for each warning message.
For details, refer to 7-1., "Communication Alarm"

Error : Related error code

8. Appendix

8-1. DNC HISTORY Contents Code Tables

(1) Kind = 1 (Command)

Upper Code	Lower Code
Command kind	Appended data length
1: Open	
2: Close	
3: Request for NC program	File name length
4: Call of NC program	
5: Transmission of NC program	File name length
6: Notification of NC reset	
7: Notification of NC alarm	

(2) Kind = 2 (Notification for correct ending of command)

Upper Code	Lower Code
Normal end code	Not used = 0
0: Processing completion	
1: Request for NC program	
2: Request for NC program transmission	
3: NC reset	
4: NC alarm	
5: NC reset/alarm	
6: Forced termination	

(3) Kind = 3 (Notification for abnormal ending of command)

Upper Code	Lower Code
Abnormal end code	Error detail code
1: Temporary error	1: Check sum error
2: Permanent error	2: Parity error
3: NC reset	3:
4: NC alarm	4: No answer signal
5: Forced termination	5: DSR OFF
6: Buffer clear	6: Ineffective answer signal
7: Initialization	7: Data length overflow
8: NC reset/alarm	8: Reception buffer overflow
	9: Data reception in other than data reception standby state
– 1: Command issuing sequence error	10: Overrun error
– 14: Data error	11: Framing error
– 15: Parameter error	
– 16: Command error	– 59: Appended data length range error
	– 60: Command issuing error
	– 62: Error in selecting with/without significant data
	– 63: Command kind selection error
	– 64: Command request selection error
	– 91: Answer error for SAT
	– 92: NC program end code selection error
	– 93: EIA/ASCII conversion error
	– 94: NC program length range error
	– 95: NC program record end code selection error
	– 96: NC program data code selection error
	– 112: Answer signal monitoring time range error
	– 128: SAT, SET command parameter selection error

(4) Kind = 4 (Internal request)

Upper Code	Lower Code
Macro status 0: Initial state 1: Awaiting macro start 2: Awaiting request for reception 3: Awaiting request for transmission 4: Awaiting reception of the first block *4 Only for protocol A	Macro event 1: Open 2: Close 3: Request for reception 4: Request for transmission 5: NC alarm 6: NC reset 7: Time-out 8: Request for forced termination 9: 10: Code conversion error

(5) Kind = 5 (Transmission message)

Upper Code	Lower Code
Message kind	Command data length
a) Protocol A	
1: GTD	
2: DAT	DAT : data length/16
3: RST	Others : data length
4: ALM	
5: SAT	
6: RTD	
7: RTY	
8: SYN	
9: RDY	
10: WAT	
11: EOD	
12: ARS	
13: AAL	
14: CLB	
15: SET	
b) Protocol B	Data length/16 (incl. DC code)
– 127: DC1	
– 126: DC3	
– 125: Data	
– 124: SYN	
– 123: NAK	

(6) Kind = 6 (Line event)

Upper Code	Lower Code
Line status	Line event
a) Protocol A	
0: Awaiting answer signal for initialization	1: Reception of SYN
1: Awaiting end of initialization	2: Reception of RDY
2: Data being received	3: Reception of ARS
3: Awaiting reception after transmitting DAT/RTD	4: Reception of AAL
4: Awaiting reception after transmitting WAT	5: Reception of CLB
5: Awaiting reception after transmitting SAT	6: Reception of SET
6: Awaiting reception after transmitting ALM	7: Reception of DAT
7: Awaiting reception after transmitting RST	8: Reception of EOD
8: Awaiting reception after transmitting EOD	9: Reception of WAT
	10: Reception of GTD
	11: Reception of RTY
	12: Reception of data other than answer signal
	13: No answer back
	14: DSR signal off
	15: Transfer error 1
	16: Transfer error 2
b) Protocol B	
0: Reception being requested	1: Normal character
1: Data being received	2: Transfer error 1
2: Transmission being requested	3: Transfer error 2
3: Data being transmitted	4: No answer signal
4: Alarm signal being requested	5: CTS signal on
5: Reset signal being requested	6: CTS signal off
	7: DSR signal off
	8: Forced termination

SECTION 6 WARM-UP FUNCTION

This section deals with the information necessary to effectively use the warm-up function incorporated in the OSP5020M/OSP500M-G.

Since operation procedures for the calendar timer are not discussed in this section, consult the machine manufacture for them.

1. Overview

The warm-up function described in this section allows to warm up the machine through the execution of the predetermined program (warm-up program) by automatically turning on power supply to the machine in accordance with the setting of the calendar timer.

That is, when power supply to the machine is turned on in accordance with the setting of the calendar timer, the OSP5020M/OSP500M-G automatically selects and executes the warm-up program prepared by the user.

2. Registration of Warm-up Program

Registration of a warm-up program in the user area of bubble memory (BB1:) by the file name of "W.WAF" is necessary to activate the warm-up function.

2-1. Registering Warm-up Program

The procedure to register the warm-up program is identical to the procedure to register part programs in the bubble memory, which is explained in Section 7, "Program Management" of Part II in the Operation Manual for OSP5020M/OSP500M-G (Publication No. 3354-E).

2-2. Format

The warm-up program format is identical to the program format for main programs. The program should begin with the program name, beginning with character O, and end with code M02 or M30.

Example:

```
O9999

    S200        M3
    VC128=50
    G0        Z500

NA1 X-200   Y-200
    X200    Y200

    G4  F5
    VC128=VC128-1
    IF [VC128 GT 0] NA1
    M5
    M2
```

3. Operations

Machine warm-up is carried out in the order described below:

(1) Set the cycle on and cycle off time on the calendar timer in accordance with the operation schedule.

Duration between cycle on to cycle off should be more than three minutes. For cycle power off, refer to items (2) and (3) in 5, "Precautions".

(2) Power supply to the NC is automatically turned on in accordance with the setting of the calendar timer.

(3) After an elapse of 90 seconds after power supply to the control has been turned on, operation end alarming buzzer will sound for the duration set by parameter. After the cycle end alarming buzzer has stopped, the control is automatically placed in the automatic mode and the warm-up program* is automatically selected and executed.

After the completion of the warm-up program (the execution of M02 or M30 in the warm-up program), or when the **RESET** button on the operation panel is pressed, machine stops and the program having been selected before the power has been turned off is automatically selected**.

* warm-up program:

Warm-up program should have the file name of W.WAF and program name may be designated as required.

** automatic program selection:

Whether the program is to be selected or not after the completion of the warm-up program is determined by the setting of parameter.

4. Timing Chart

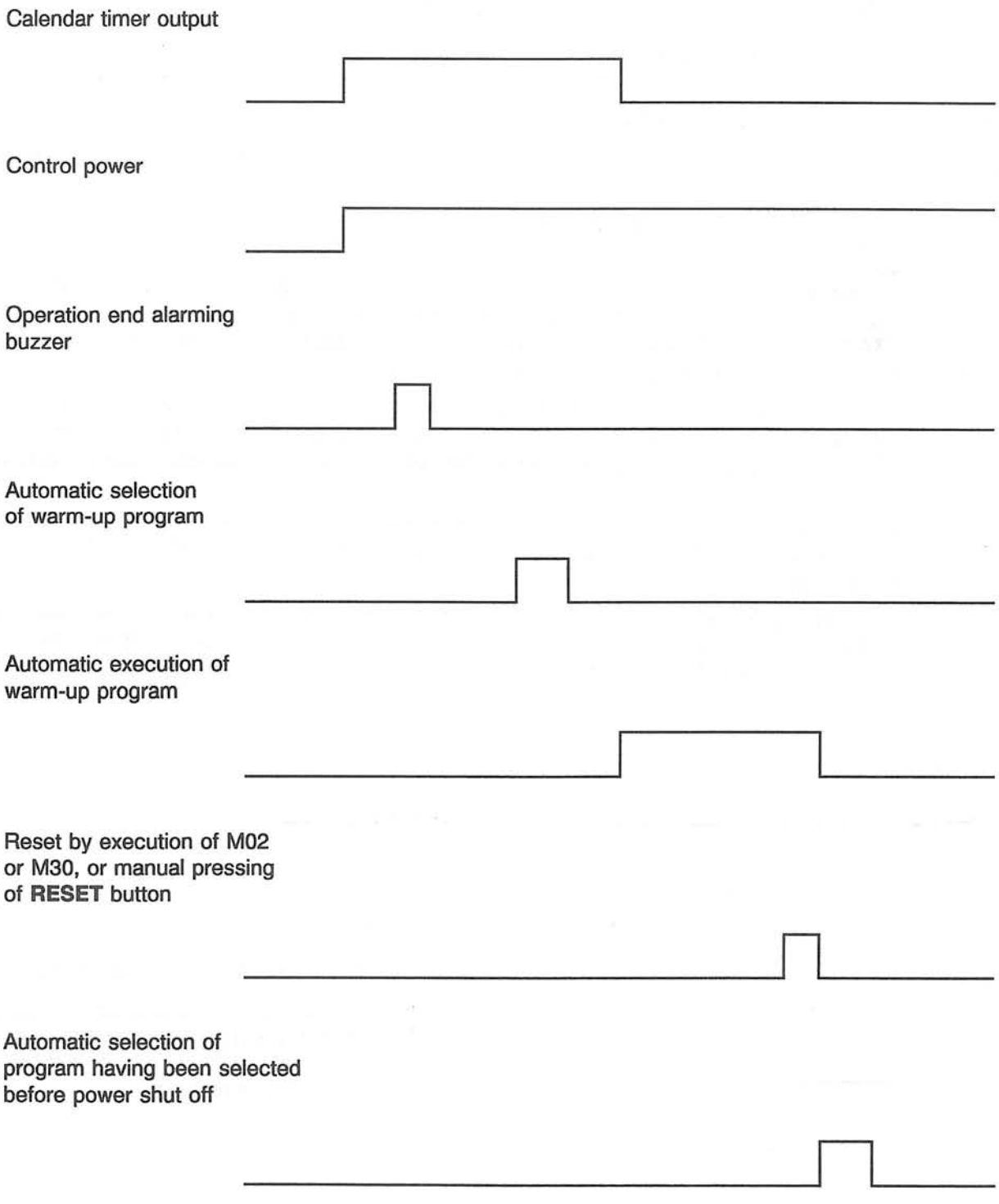


Fig. 6-1 Timing Chart for Warm-up Program Execution

5. Precautions

(1) After the completion of the warm-up program, the program selected through the operation panel is automatically selected. Therefore, if the warm-up program (file name W.WAF) is selected through the operation panel before turning off power supply, W.WAF is selected again after the completion of warm-up program.

(2) If the operator turns off control power while the calendar timer output is on and the control power is turned on again, the warm-up program is executed again.

(3) If the calendar timer output is turned off while the warm-up program is being executed, the warm-up program is not interrupted and executed up to the end of the program.

(4) If the file W.WAF is not existent in the bubble memory when power is automatically turned on by the calendar timer function, an alarm will occur. See 6. "Alarm".

(5) For the period from automatic power on to the cycle start of the warm-up program, controls (mode selection keys, function keys, alphanumeric keys) on the NC operation panel, **CYCLE START** button and the external cycle start button are all disable. However, if the selection of the warm-up program is impossible, all controls are operative.

(6) If the operator presses the **RESET** button while the warm-up program is being executed, the program having been selected before power shutoff is automatically selected. Note that when "0" is designated at the optional parameter (bit) No. 4, the automatic program selection is ineffective.

(7) The warm-up function is ignored even if the calendar timer is actuated, if its actuation is after the display of the ACTIVE POSITION data display with power supply to the OSP5020M turned on by the operator.

(8) The end of program command in the warm-up program is ignored by the automatic power shutoff function. That is, power supply will not be turned off even when the M02 or M30 in the warm-up program is executed.

6. Alarm

The following alarm is added when the warm-up function is selected.

ALARM C

958	Warming up			
	Index	:	None	
	Code	:	1	No W.WAF file in the bubble memory
			2	File selection is impossible although the warm-up program file is existent. (Ex.: M02 or M30 not specified at the end of warm-up program.)

7. Parameter Setting

For the activation of the warm-up program, setting of the following parameter is necessary.

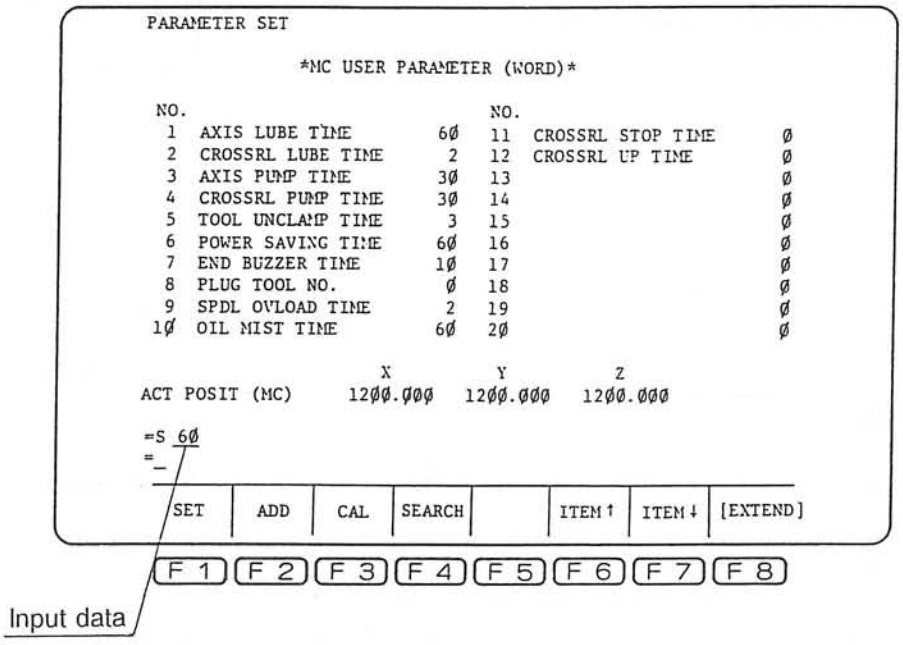
MC User Parameter (Word)						
Parameter No.	CRT Display	Function	Effective commands	Initial Value	Setting Range	Setting Unit
7	END BUZZER TIME	Time duration in which the operation end alarming buzzer sounds	SET ADD	10	0 to 600	(sec)

Operation Procedure:

(1) Select the PARAMETER SET mode by pressing the **PARAMETER** key.

(2) Press the function key [F6] (ITEM ↑) or [F7] (ITEM ↓) until the CRT displays the *MC USER PARAMETER (WORD) * page.

(3) Locate the cursor on parameter No. 7 using the cursor control keys.



(4) After pressing the function key [F1] (SET), key in the required duration through the keyboard.

(5) Press the **WRITE** key.

Check the data of parameter No. 7.

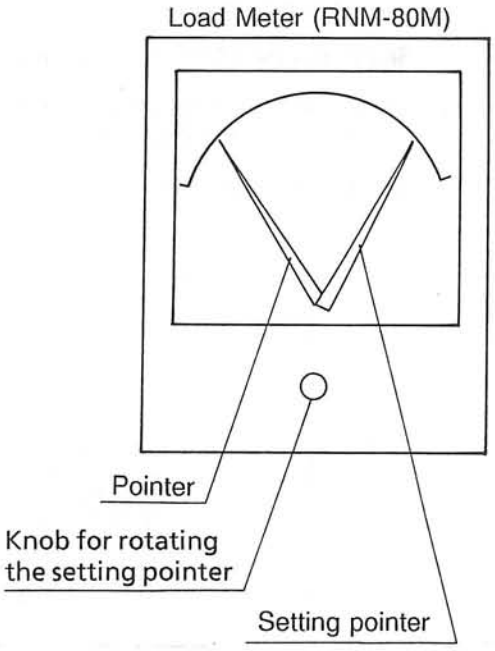
SECTION 7 SPINDLE OVERLOAD DETECTION FUNCTION

1. Overview

(1) The meter relay is used for the spindle motor load meter to trigger an alarm signal if the spindle load exceeds a preset level. This overload detection function detects excessively heavy cutting condition or collision of the cutting with the workpiece due to operation error immediately. Upon detection, the control triggers level A alarm and stops machine operation.

(2) The load meter with the meter relay as illustrated at the right is connected to the motor drive circuit.

(3) Set the maximum load setting pointer at the required load value position by rotating the knob. When the motor starts, the meter pointer moves according to the motor load; if the pointer swings beyond the setting pointer, the relay contact closes. This is detected as level A alarm and stops all axes rotation.



2. Operations

(1) This function is made effective and ineffective by designating the following M codes in the automatic or MDI mode.

M142 . . . . . Spindle overload detection function ineffective
M143 . . . . . Spindle overload detection function effective.

(2) When power supply to the control is turned on or when the control is reset, M142 is effective. Therefore, it is necessary to designate M143 to make the spindle overload detection function effective.

(3) In the manual operation mode, the spindle overload detection function is always ineffective and thus the related alarm does not occur.

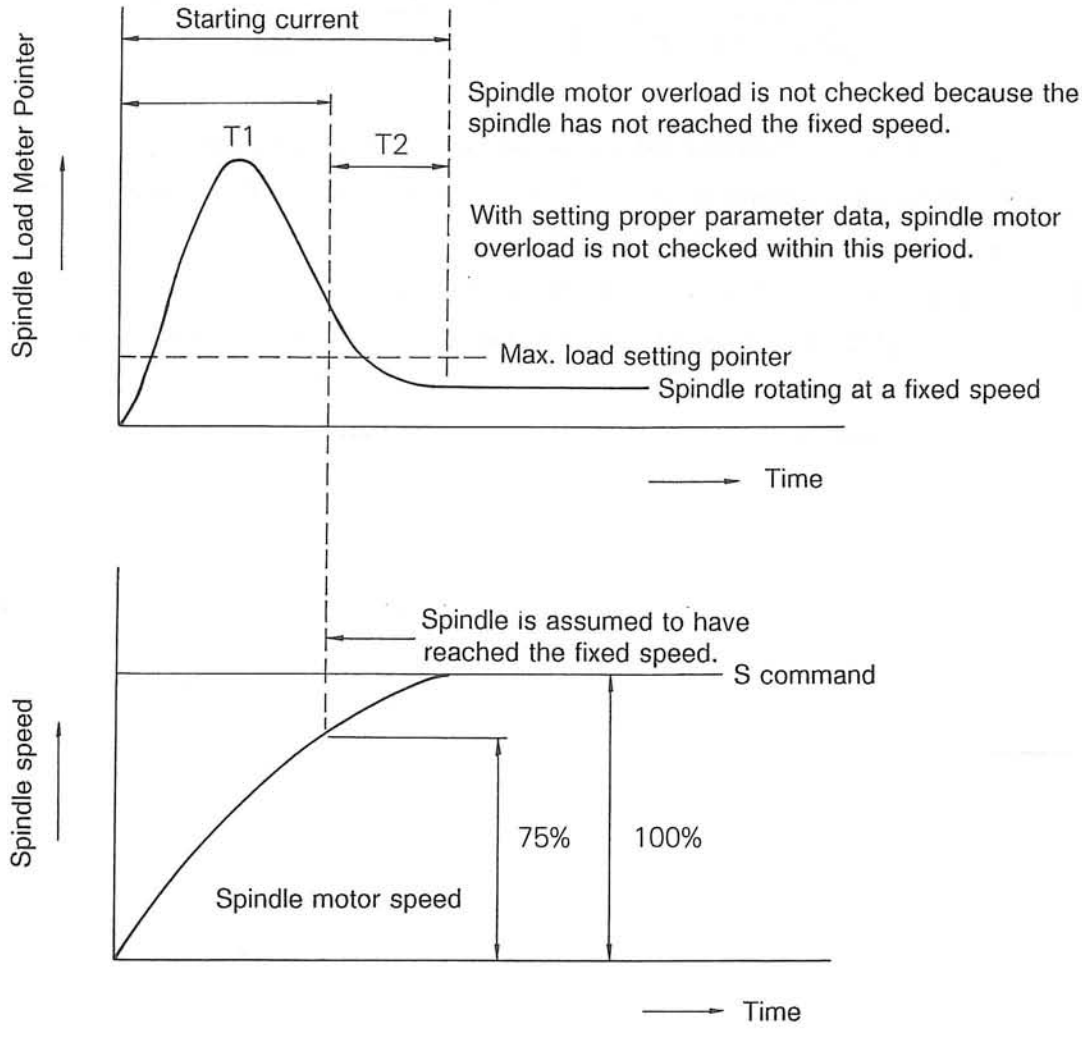
3. Spindle Drive Motor Starting Current

(1) When the spindle motor starts, the starting current flows to cause the load meter pointer to swing beyond the max. load setting pointer. The control ignores this overcurrent status so that alarm will not occur. For this purpose, the control establishes the immune period as Indicated below.

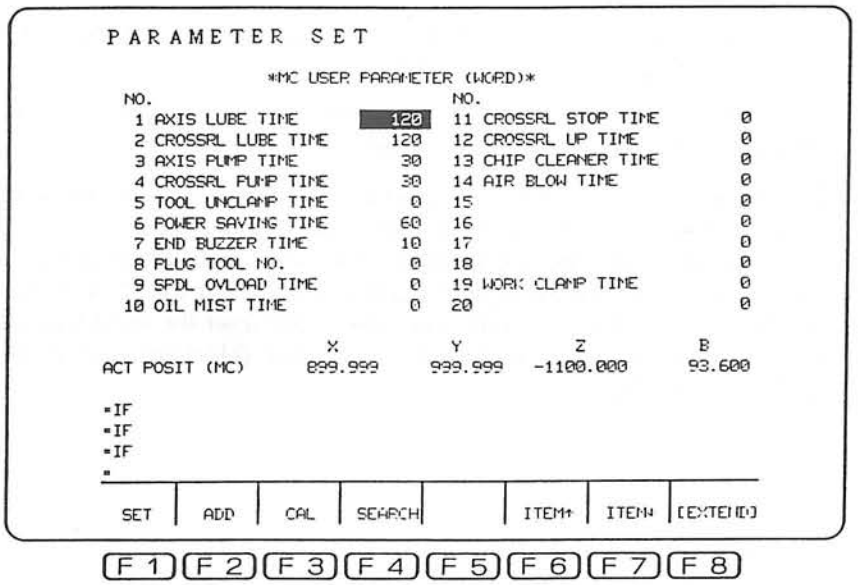
(a) The control determines that the spindle is rotating at a fixed speed when an actual spindle speed is 75-125 percent of the programed speed.

(b) Spindle overload condition is not checked until the spindle speed reaches the fixed speed, i.e., within duration T1 from the spindle start.

(c) However, there are cases where the spindle load meter pointer still lies at the right side of the preset point even after the spindle speed reaches the fixed speed because the response of the spindle load meter pointer is slow. This also might occur if the setting value is relatively small. Thus the control does not check the spindle overload condition for a period T2 until the spindle rotation reaches the stationary state after the fixed speed condition has been reached. This period T2 is set with the parameter (Machine user parameter (word) No. 9 SPDL OVLOAD TIME).



(2) The CRT screen display for the spindle overload timer is shown below. Time setting range is 0 - 60 seconds in increments of 1 second. Setting is made using the [F1] (SET) or [F2] (ADD). The Initial value is one second (1).



(3) The following states are assumed that the spindle has not reached the fixed speed:

(a) The spindle is not rotating, or the spindle start (forward or reverse) command is given.

(b) The spindle stop command is given while the spindle is rotating.

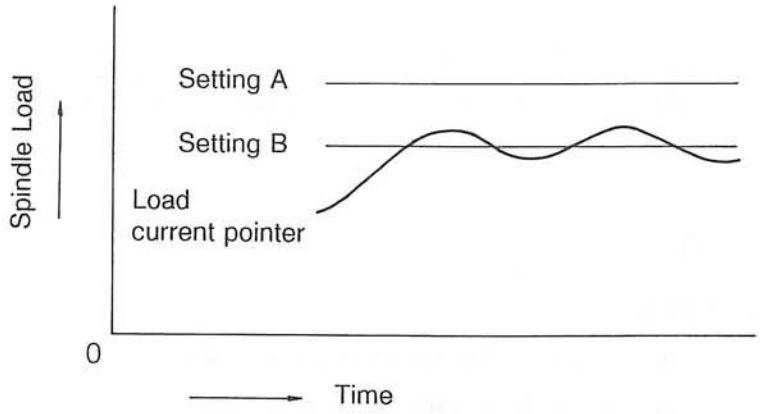
(c) A new S command is given while the spindle is rotating.

(d) When the spindle speed override switch setting is changed while the spindle is rotating.

4. Spindle Overload Detection

When the spindle overload detection is effective, the spindle overload alarm occurs immediately if the spindle load meter pointer moves beyond the setting pointer.

If the spindle load meter pointer fluctuates as illustrated below when the setting is at "B", the spindle overload alarm occurs. Therefore, the setting should be at "A".



5. Alarm

ALARM A

147 Spindle overload

Spindle load exceeds the preset limit.

SECTION 8 SIMPLIFIED LOAD MONITORING FUNCTION

1. Outline

This function monitors the spindle load and activates an alarm if the parameter-designated load value is exceeded.

A total of five parameter sets is used to control this function. The desired parameter set is selected using system variable VSLNO, with monitoring ON/OFF switching executed by the M142/M143 commands.

2. Details of Function

(1) Monitoring ON/OFF Switching

M142 . . . . . "Simplified Load Monitoring Function" OFF

M143 . . . . . "Simplified Load Monitoring Function" ON

(2) Selecting The Spindle Load Monitoring Parameter No.

There are 5 sets of setting values for the "LOAD MAX", "LOAD TIMER", and "MONIT. MOD (Monitoring Mode)" items. The desired parameter set is selected from these 5 sets.

VSLNO . . . . . Monitoring parameter No. selection System Variable

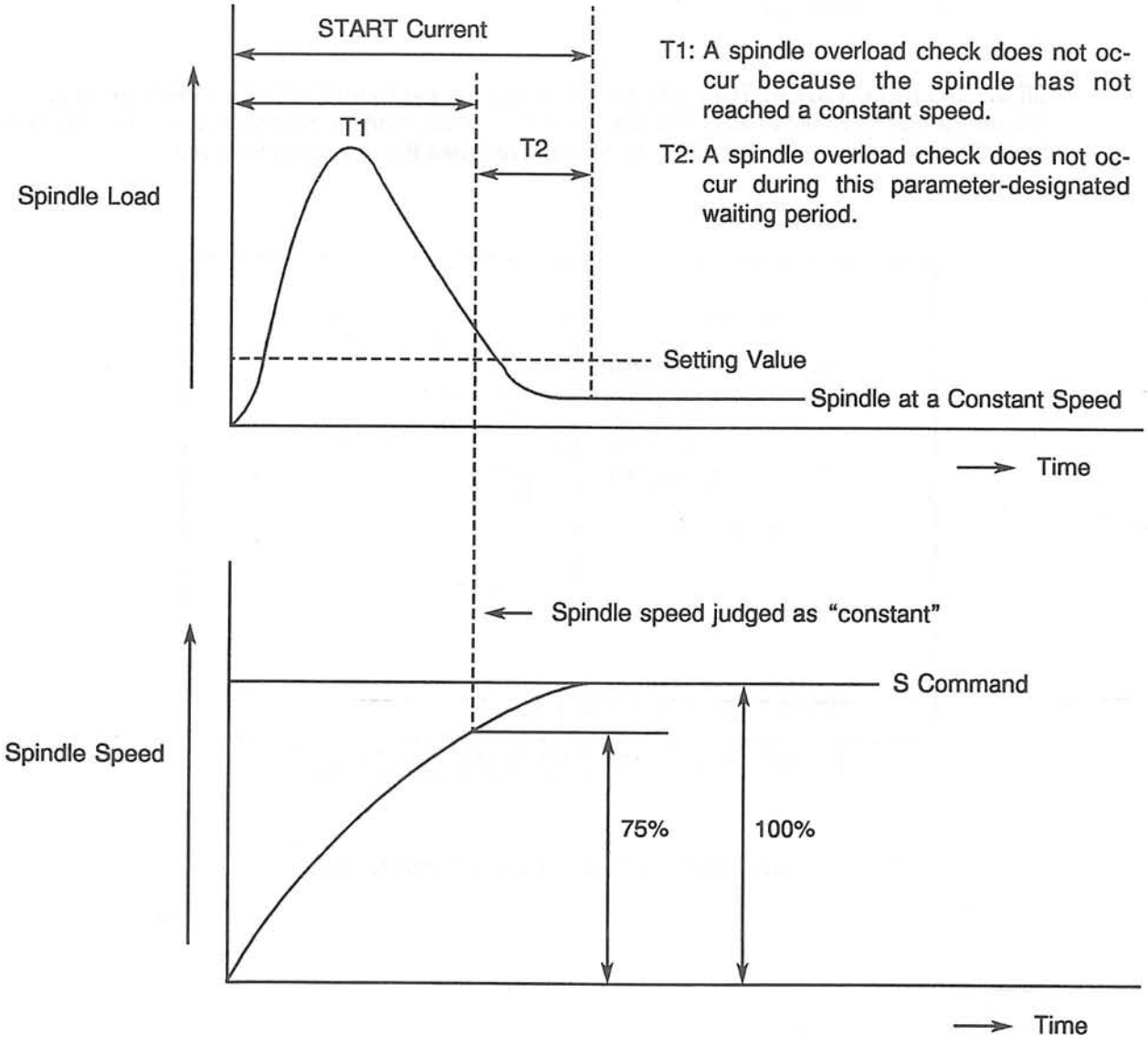
A VSLNO backup is executed. Therefore, if no designation is made, the value designated for the previous operation will be adopted. If no previous designation exists, "1" will be adopted.

(3) Ignoring The Spindle Motor's START Current

When spindle rotation is executed from a stopped condition, there will be cases in which the "maximum load value" will be exceeded due to the START current at the spindle motor. This START overload condition is handled by the following 2 actions:

① The overload is ignored until the actual spindle speed is within 75% to 125% of the speed command value.

② When the actual spindle speed enters this 75% to 125% range, the system then waits for the spindle to reach a constant speed. During this waiting period (designated by machine user parameter(word) No.9), the overload condition is ignored.



(4) Overload Monitoring Format

① When the monitoring parameter is selected at VSLNO, and the M143 command is executed to switch the monitoring function ON, overload monitoring will then begin from the point when the spindle reaches the constant speed.

② If the spindle load exceeds the parameter-designated maximum load value, and that overload is maintained continuously until the "continuous overload time" is exceeded, the spindle overload alarm will be activated.

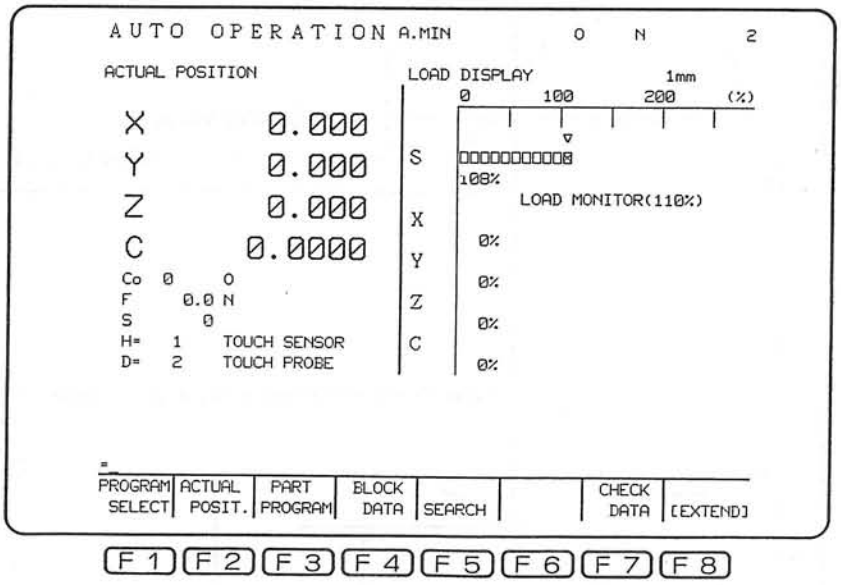
The alarm level can be changed by changing the monitoring mode.

0 : Alarm [A]

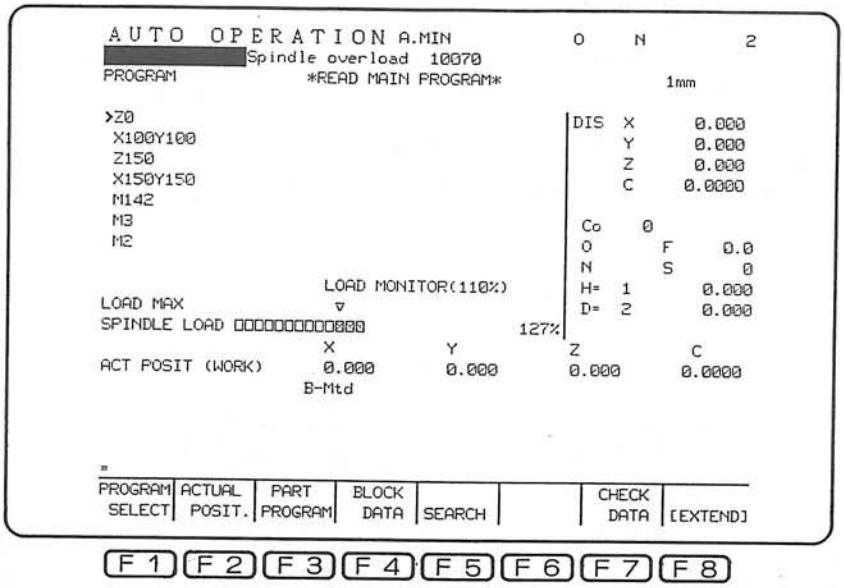
30 : Alarm [D]

(5) Display

On the 2nd page of the ACTUAL POSITION display, on the CURRENT MAIN PROGRAM screen, and on the READ MAIN PROGRAM screen, a "▽" scale mark is displayed at the 1st line of the "spindle load" bar-graph. This "▽" scale mark indicates the maximum load value.



2nd Page of the ACTVAL POSITION Screen



READ MAIN PROGRAM Screen

(Display is identical for CURRENT MAIN PROGRAM)

When the monitoring function is ON (M143), LOAD MONITOR (*** %) (*** : Maximum Load Value) items are displayed to indicate that the monitoring function is operating.

If the spindle load exceeds the maximum load value, the LOAD MONITOR (*** %)' item will change from white to red. (In monochrome systems, it will become brighter.)

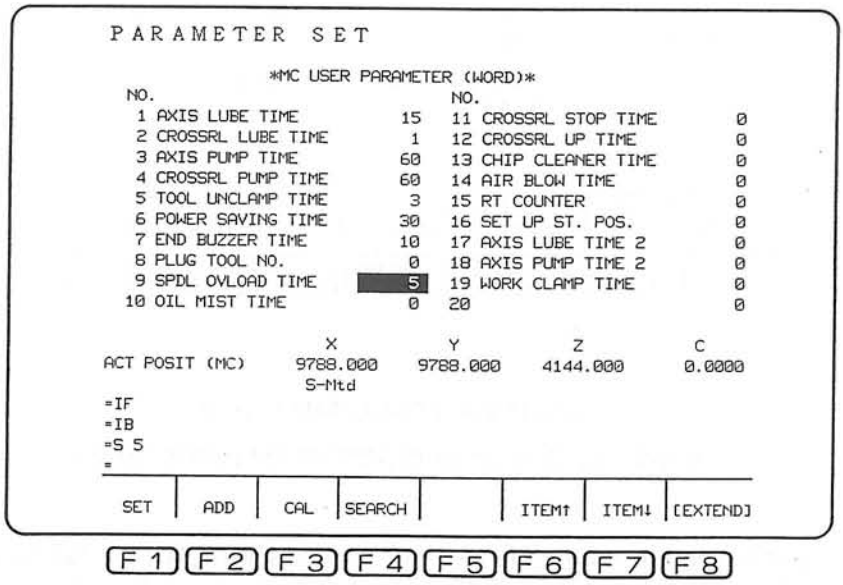
(6) Parameters

① Constant Spindle Speed Waiting Period Setting

This parameter designates the waiting period during which the system waits for the spindle to reach a constant speed. (For details, refer to item (3) above.)

- Machine user parameter (word) No.9

  Spindle Overload Timer (SPDL OVLOAD TIME)

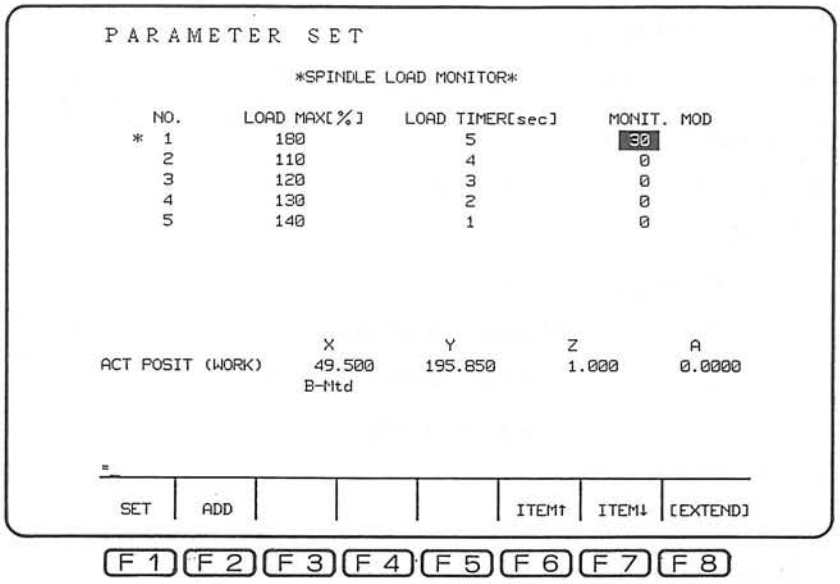


Setting range : 0 - 60 secs.

Initial value : varies depending on machine models

② Spindle Overload Monitoring Parameter Setting

Values are set at LOAD MAX, LOAD TIMER, and MONIT. MOD in order for spindle overload monitoring to occur.



- No. : One of the possible five parameter sets is selected at VLSNO, and an asterisk is displayed to the left of the selected parameter No.
- LOAD MAX : The maximum allowable load (in percentage) is designated.

  Setting range .... 0 - 280 (%)

  Command ..... [SET], [ADD]

  Initial value ..... 0 (%)
- COAD TIMER: The maximum permissible period during which the spindle load can continuously exceed the maximum load value is designated (secs.).

  Setting range .... 0 - 120 (secs.)

  Command ..... [SET], [ADD]

  Initial value ..... 0 (secs.)
- MONIT. MOD: The overload alarm level is designated.

  Setting value = 0 .... Alarm [A]

  30 .... Alarm [D]

  Initial value 0

* These parameters are stored (backup) at the MAU **_**_*.PB_.

(7) System Variables

① Spindle Overload Monitoring Parameter No.

[VSLNO]

- Reading/writing possible
- Data value range: 1 to 5
- VSLNO is stored in memory (backup). Therefore, if no designation is made, the value designated for the previous operation will be adopted. If no value has been designated previously, "1" will be adopted.

② Spindle Overload Monitoring Parameter

[VSLDT (i,j)]

- i : Parameter No. (1-5)
- j : 1 ..... Maximum load value
  2 ..... Continuous overload time
  3 ..... Monitoring mode
- Only reading possible

(8) Alarms

① Alarm A 147 Spindle overload

② Alarm D W47 Spindle overload

Codes (for both 1) and 2))

XXXXYYYY

XXXX : Overload monitoring parameter No.

YYYY : Overload (%) when alarm was activated

SECTION 9 EXTERNAL PROGRAM SELECTION FUNCTION

1. External Program Selection A (Pushbutton Type)

This function selects the main and schedule programs by pressing the pushbutton switch corresponding to the desired program.

1-1. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule program is selected by the setting of the parameter (bit) data.

Parameter (bit) No.17	Main /Schedule Program
Bit 5 = 1	Schedule programs
Bit 5 = 0	Main programs

1-2. Inhibiting the Selection of the Same Program

By setting proper parameter (bit) data, it is possible to inhibit the loading of the selected program if the same program resident in the memory is selected again. This can save the program loading time.

Parameter (bit) No.17	Same Program Loading
Bit 6 = 1	Inhibited
Bit 6 = 0	Allowed

1-3. Program Selection

Selection of a desired program is possible by pressing the pushbutton switch corresponding to that program.

Selection of a program from P1.MIN (or SDF) to P16. MIN (or SDF) is possible.

Note that the number of selectable programs differs depending on the specification.

After the completion of program selection, the lamp corresponding to the program number lights up. This lamp, while lit, indicates that the corresponding program is selected.

1-4. Precautions

(1) Program selection is possible only in the automatic mode.

(2) While a program is being selected, pressing another program selection button is ignored.

(3) While the control is in the alarm state, program selection is not carried out.

(4) An alarm occurs if the program designated for selection is not found.

(5) An attempt to select another program during the selected program execution causes an alarm.

(6) Program selection during the editing or copying execution is suspended until the completion of the editing on copying.

2. External Program Selection B (Rotary Switch Type)

This function selects the main and schedule programs using the rotary switch and the program selection pushbutton switch.

2-1. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule program is selected by the setting of the parameter (bit) data.

Parameter (bit) No.17	Main /Schedule Program
Bit 5 = 1	Schedule programs
Bit 5 = 0	Main programs

2-2. Inhibiting the Selection of the Same Program

By setting proper parameter (bit) data, it is possible to inhibit the loading of the selected program if the same program resident in the memory is selected again. This can save the program loading time.

Parameter (bit) No.17	Same Program Loading
Bit 6 = 1	Inhibited
Bit 6 = 0	Allowed

2-3. Program Selection

Selection of a program is possible by specifying the desired program using the rotary switch.

Selection of a program from P1.MIN (or SDF) to P16. MIN (or SDF) is possible.

Note that the number of selectable programs differs depending on the specification.

After the completion of program selection, the program selection indicating lamp lights up. This lamp lights up when the program number selected by the rotary switch and the program number actually selected coincide each other.

2-4. Precautions

(1) Program selection is possible only in the automatic mode.

(2) While a program is being selected, pressing another program selection button is ignored.

(3) While the control is in the alarm state, program selection is not carried out.

(4) An alarm occurs if the program designated for selection is not found.

(5) An attempt to select another program during the selected program execution causes an alarm.

(6) Program selection during the editing or copying execution is suspended until the completion of the editing or copying.

3. External Program Selection C (BCD Type)

3-1. Overview

This function selects the main and schedule programs using the BCD (binary coded decimal) number input from an external device.

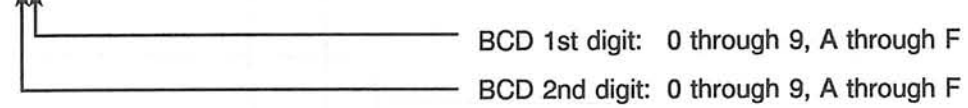
3-2. Program Selection Range

(1) 4-digit BCD numbers

P* * * *.MIN (main program)
P* * * *.SDF (schedule program)

(2) 2-digit BCD numbers

P* *.MIN
P* *.SDF



Note: *Designation of "P0000.MIN (or P00.MIN)" is not allowed.*
Selectable range is from P0001.MIN to PFFFF.MIN or from P01.MIN to PFF.MIN.

3-3. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule program is selected by the setting of the parameter (bit) data.

Parameter (bit) No.17	Main /Schedule Program
Bit 5 = 1	Schedule programs
Bit 5 = 0	Main programs

3-4. Inhibiting the Selection of the Same Program

By setting proper parameter (bit) data, it is possible to inhibit the loading of the selected program if the same program resident in the memory is selected again. This can save the program loading time.

Parameter (bit) No.17	Same Program Loading
Bit 6 = 1	Inhibited
Bit 6 = 0	Allowed

3-5. Precautions

(1) Program selection is possible only in the automatic mode.

(2) While a program is being selected, pressing another program selection button is ignored.

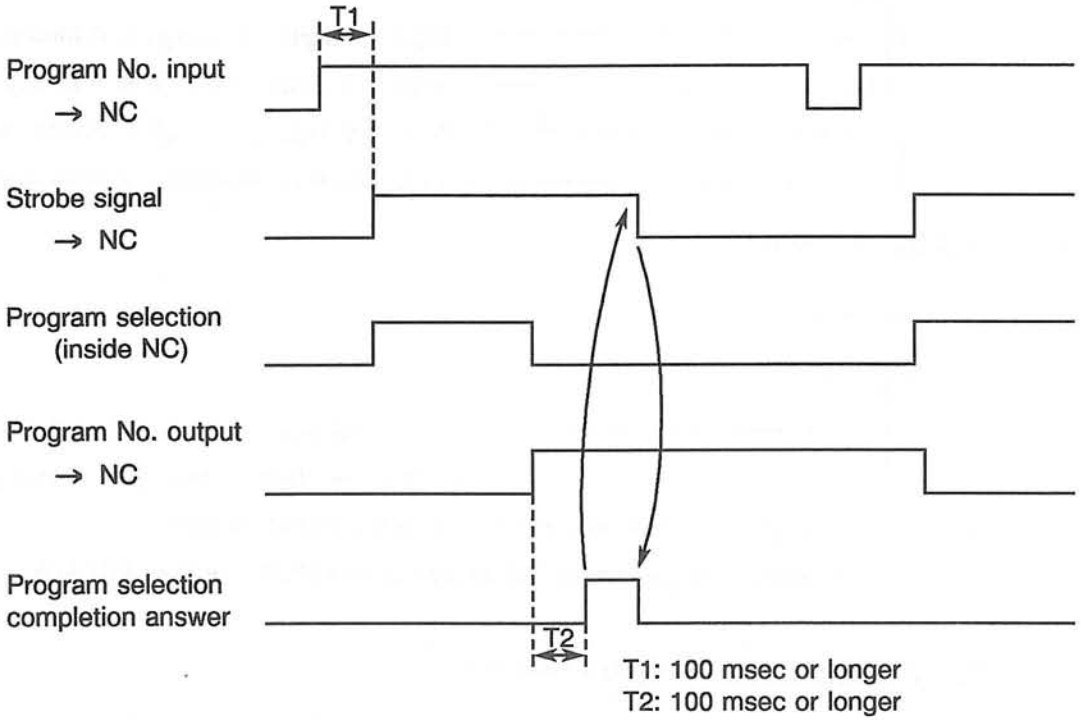
(3) While the control is in the alarm state, program selection is not carried out.

(4) An alarm occurs if the program designated for selection is not found.

(5) An attempt to select another program during the selected program execution causes an alarm.

(6) Program selection during the editing or copying is being executed is suspended until the completion of the job being in execution.

3-6. Timing Chart

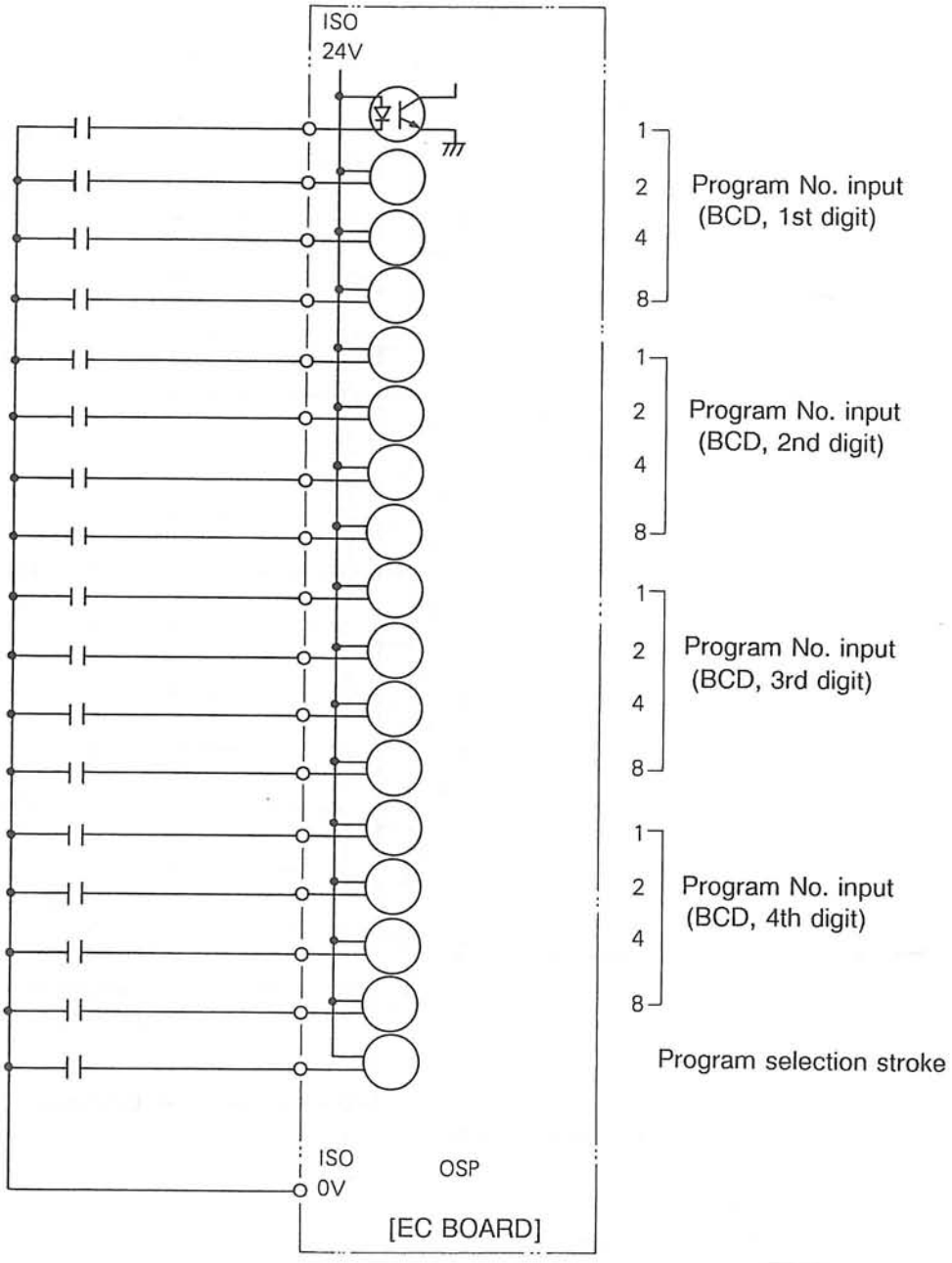


(1) The strobe signal is maintained the program selection completion answer is output from the NC. The strobe signal is turned off when the program selection completion answer is received.

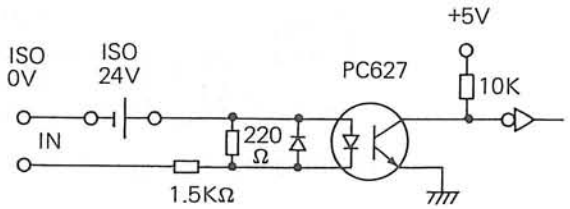
(2) The program number output signal is turned on after the completion of program selection until the next program selection begins. This signal is not output while a program is being selected.

3-7. Input/output Signals

(1) Input signals

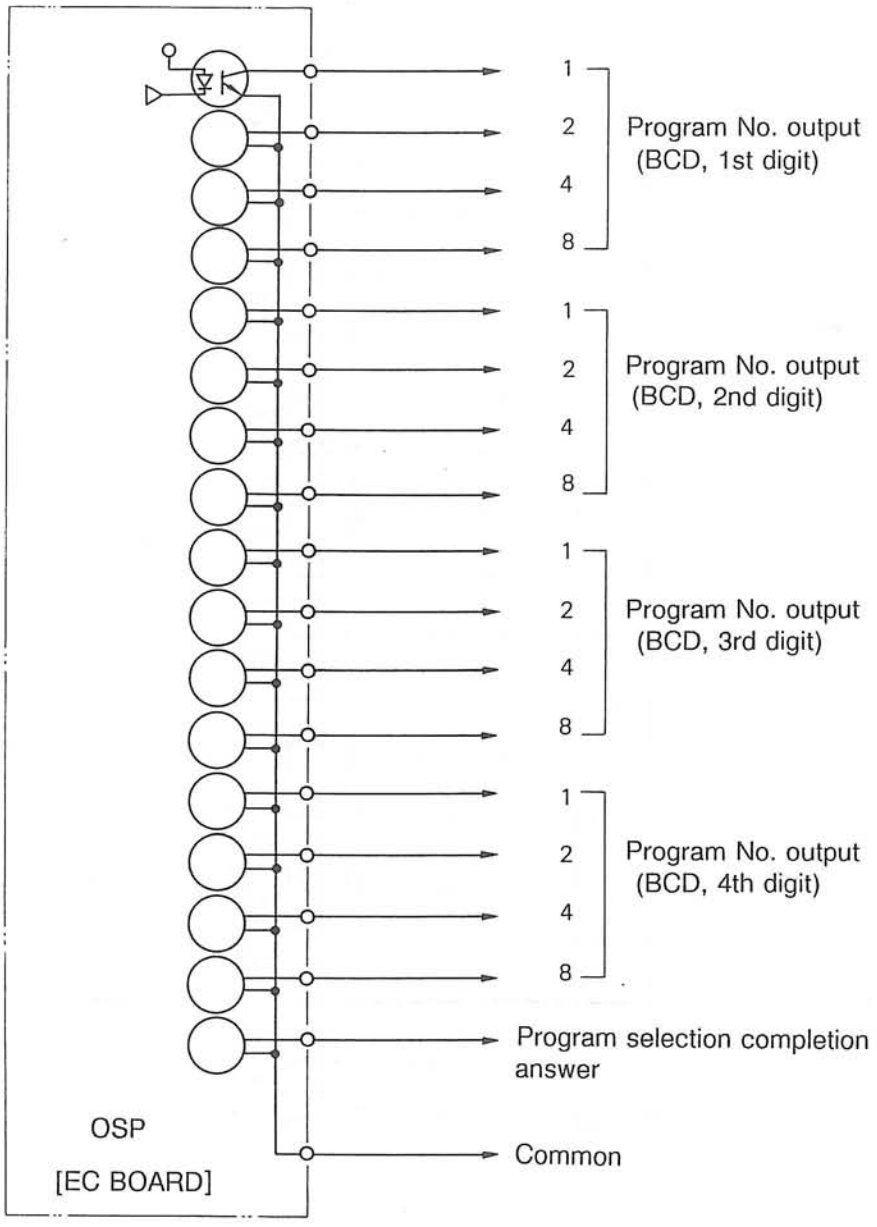


Phtocoupler Input Circuit

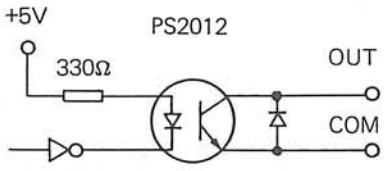


Input contact rating: DC 24V, 15 mA

(2) Output signal



Transistor Output Circuit



Output current rating: DC 24V, 60 mA

3-8. Parity Check

The parity check function can be added to external program selection C (BDC type). By adding this function, the reliability of signals can be improved.

This function is a standard feature for NC versions from A709K (NC main: MAA07.09AK.POL).

3-8-1. Parameters

NC optional parameter (bit) No. 20, bit 6

Setting

1: Parity check is performed for external program selection C.

0: Parity check is not performed for external program selection C.

NC optional parameter (bit) No. 20, bit 7

1: An odd parity check is performed for external program selection C.

0: An even parity check is performed for external program selection C.

3-9. Alarms

ALARM C

944 Program select

In external program selection operation, an error occurred during the selection of a file-name which was input from the ECP or EC board.

Index	:	None
Character-string	:	None
Code	:	X

4 ..... A parity error occurred with external program selection C.

(For the explanation of other error codes for X, refer to the Alarm & Error List (publication No. 3353-E).

SECTION 10 EXTERNAL M SIGNAL OUTPUT FUNCTION

1. Overview

The function outputs contact point signals to an external device, corresponding to the external M code written in a program. The external device conducts processing in response to the reception of the signal and after the completion of the processing, it returns the answer back signal to the NC. It then proceeds the program to the next step assuming that the commanded processing is complete upon the reception of the answer back signal from the external device.

For external M codes, two different numbers of output points, four or eight, are prepared. In either case, only one point is provided for an answer back signal.

1-1. External M Codes

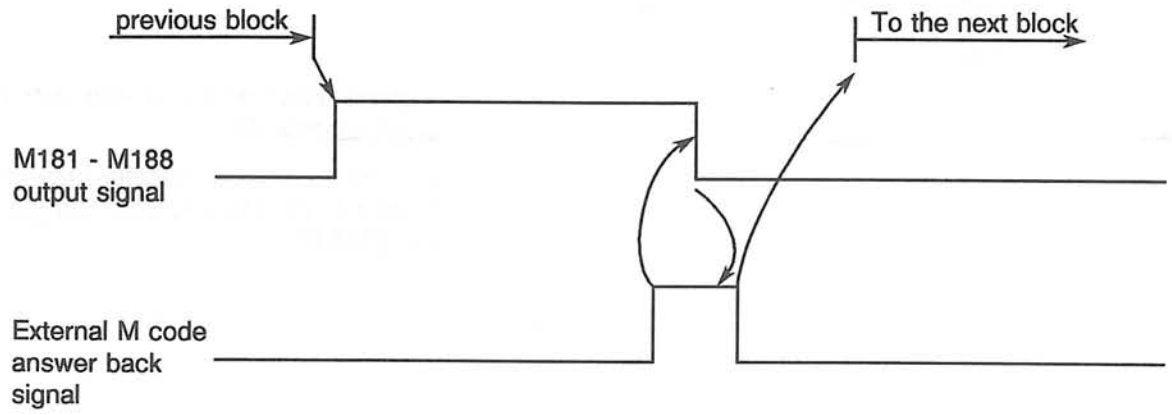
For external M code outputs, M181 - M188 are provided:

Four points ..... M181 - M184

Eight points ..... M181 - M188

Note: In a program, one block can contain only one external M code.

1-2. Timing Chart



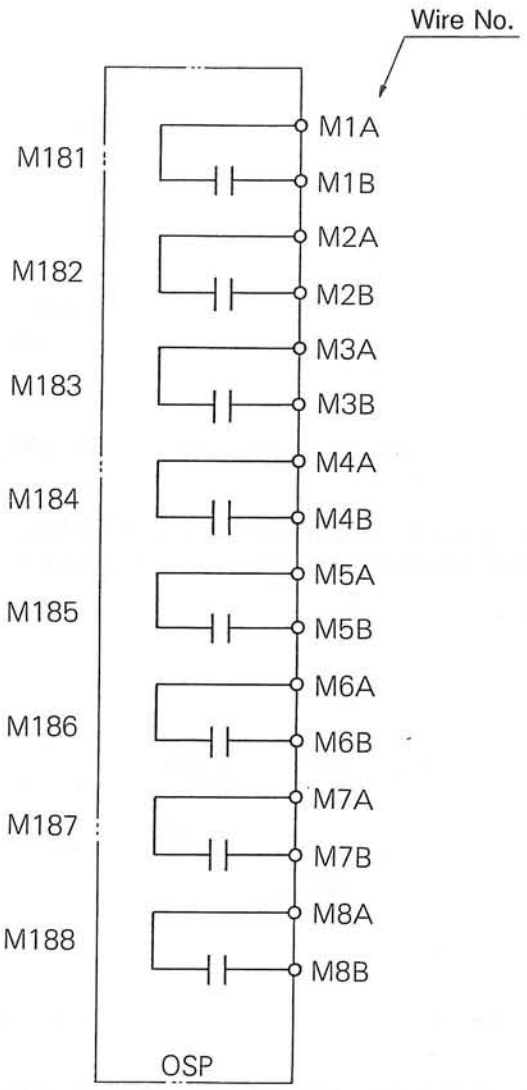
Note 1: M181-M188 output signals are turned on by the corresponding external M code and turned off by the external M code answer back signal.

Note 2: The answer back signal should be turned off only after checking that the corresponding M181-188 output signal has been turned off.

Note 3: The control is placed in the S.T.M. function execution status while an external M signal is being output and program execution stops.

2. Input/Output Signals

2-1. Output Signals

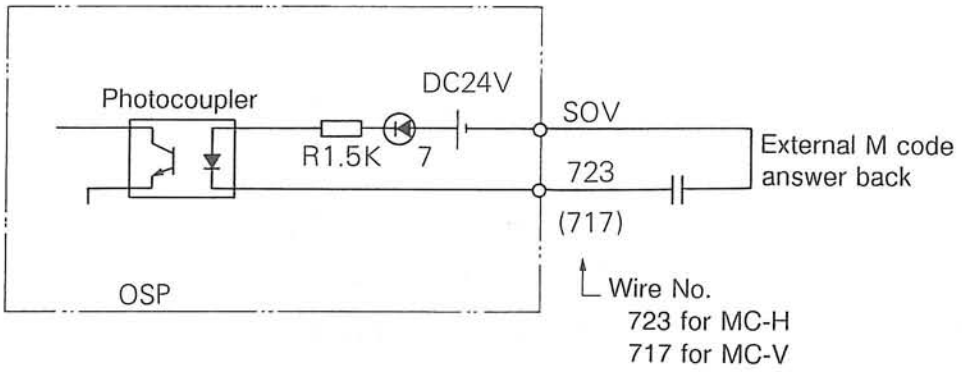


Note: *Signals are output via relays (Mini-power relay MY2-1024, OMRON). The contact point ratings are indicated below.*

Item / Load	Rated Load	Rated Current	Rated Maximum Voltage	Rated Maximum Current	Maximum Switching Capacity
Resistance load (cosØ = 1)	AC 220V 5V DC 24V 5A	5A	AC 250V DC 125V	5A	1,100VA 120W
Induction load (cosØ = 0.4, VR = 7 msec)	AC 220 2A DC 24V 2A	5A	AC 250V DC 125V	5A	440 VA 48W

2-2. Input Signals

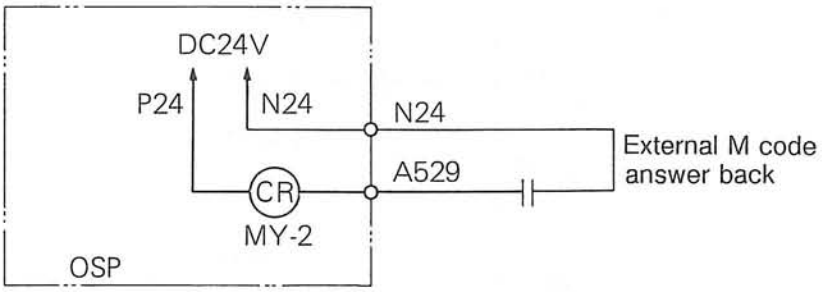
(1) MC-H/MC-V Series



Note 1: Input signals should use dry contact point with capacity of DC 24 V, 16 mA minimum.

Note 2: Input signals are effective when it is on for longer than 200 msec. Chattering at signal rising and falling must be less than 10 msec.

(2) Double-column Series



Note 1: Input signals should use dry contact point with capacity of DC 24 V, 37 mA minimum.

Note 2: Input signals are effective when it is on for longer than 200 msec. Chattering at signal rising and falling must be less than 10 msec.

3. Connection

The machine has the terminal box used for external signals. Connect the cables from this terminal box to the external devices.

The terminal blocks are labeled with the wire numbers explained in 2., "Input/Output Signals". The screw size used at the terminal block is M4.

Illustrations in Fig. 10-1 shows the external terminal box installation locations on each machining center model. Note that these show the standard specification mode and the terminal box installation locations might differ from the illustrated locations, if special specifications are selected.

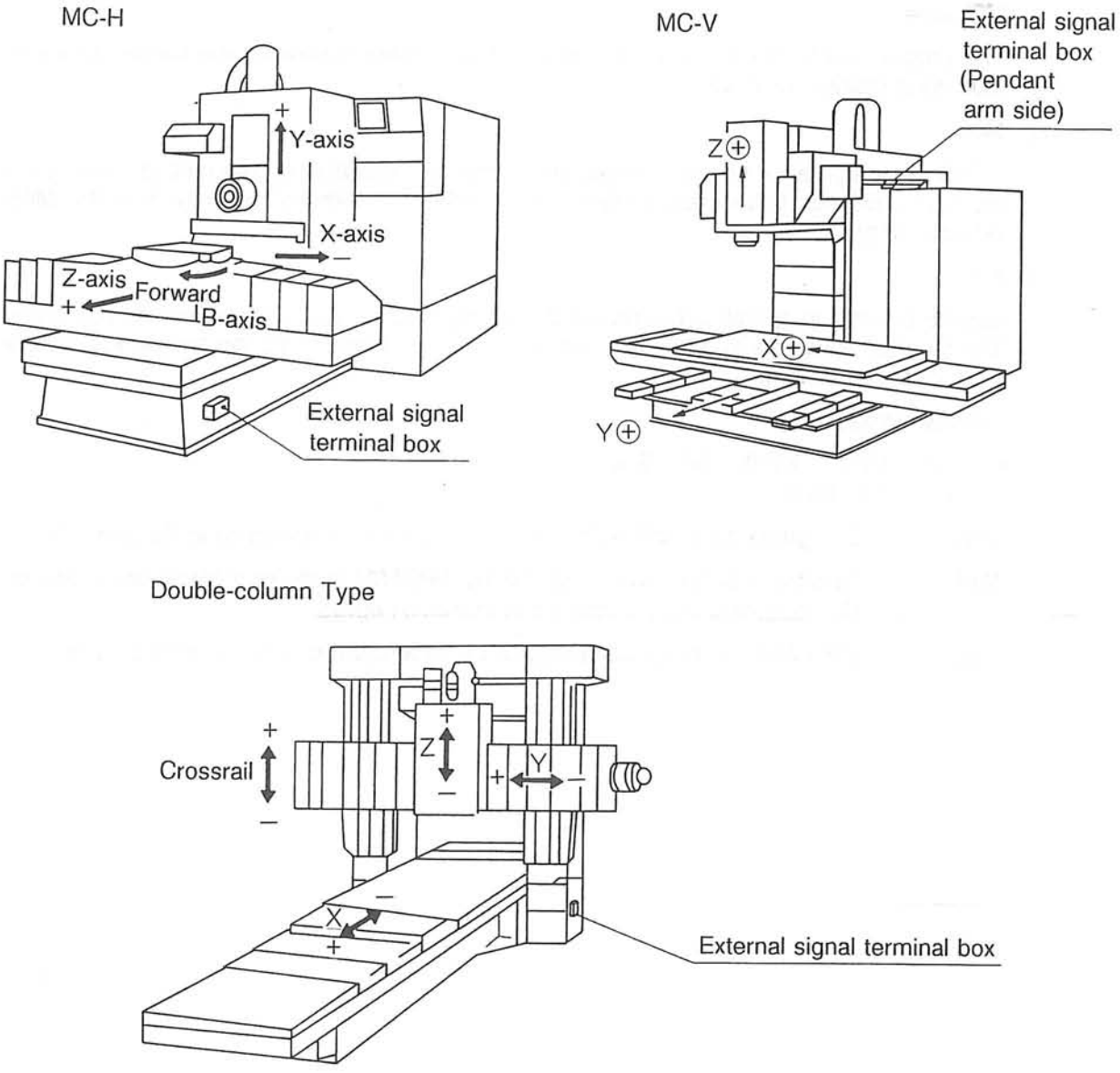


Fig. 10-1

SECTION 11 CYCLE TIME REDUCTION FUNCTION

This function reduces non-cutting time by executing spindle rotation command and axis motion command or other command simultaneously to shorten total machining cycle time.

1. Ignoring Spindle Rotation Answer: M300 (1 Block)

When the M300 code is designated in a program, answer signals for spindle start/stop command is ignored:

(1) M03/M04

The program advances to the next program block without confirming the answer to the M03 or M04 code (spindle rotation).

(2) M05

Spindle stop operation begins simultaneously with the execution of axis motion commands. The program advances to the next program block without confirming the answer to the M05 code (spindle stop).

(3) M19

Spindle orientation operation begins simultaneously with the execution of axis motion commands. The program advances to the next program block without confirming the answer to the M19 code (spindle stop position).

(4) Example Program

```
N100  G00   X100   M3 M300
N101  Z10 (M3)
```

Note 1: Designate the M300 code with a spindle rotation command in the same block.

Note 2: To confirm the answer signal for the M03/M04 in the next block, designate the spindle start (M03/M04) command in that block again.

Note 3: If the M300 code is designated in the fixed cycle mode, an alarm occurs.

2. Ignoring/Confirming Answer Signal Other Than Spindle Rotation Answer: M301, M302 (1 Block)

When the M301 code is designated in a program, answer signals other than the answer signal for the M03/M04 code are ignored. The M302 code cancels the M301 mode and the answer signals for the programmed M codes are confirmed.

These M codes (M301/M302) are effective to the following M codes:

- M32 (door close)
- M33 (door open)
- M181 - M188 (external M signal)

(1) Example program

```
        :
N200    M33M301         (Ignoring M33 answer signal)
N201    M181    M301    (Ignoring M181 answer signal)
N202    M302            (Confirming M33/M181 answer signal)
        :
```

Note: The M301 code is a one-shot M code. Therefore, designate it with the M code for which an answer signal should be ignored in the same block.

SECTION 12 F1-DIGIT FEED COMMAND FUNCTION (BY PARAMETERS)

1. Outline

Conventionally, as an alternative method, the cutting feedrate has been able to be designated by the F address character followed by a 1-digit integer. The feedrate can then be controlled by the feed switch setting value which corresponds to the designated 1-digit number.

This method is referred to as the "F1-digit feed function".

The function discussed here enables the feedrate to be designated by parameter setting through the CRT screen instead of by feed switches on the operation panel.

This function has the following features:

(1) Designation of the feedrate through the CRT screen enables a "software type" designation format which eliminates the need for conventional feed switches.

(2) All setting data are handled as system variables, thus making reference and setting possible from the program as well.

(3) In addition to using function keys [SET] and [ADD] to set the feedrate, the page keys can be used to update data in a continuous manner.

2. Number of Parameter Sets Used for F1-Digit Feed Control And Command Method

- Number of parameter sets used for F1-digit feed control:

  F1 to F9 : 9 sets

- F1-digit feed command method

  Address F followed by integer between 1 and 9

  Any other designation will be processed as a regular F command which directly designates a cutting feedrate.

- Real numbers (including variables) such as F1. etc., will be processed as regular F commands. Therefore, variables may not be used in the F1-digit command format.

  Example:

  F1 F1-digit . . . . . Operation will be according to the F1 feed switch setting.

  F200 200 mm/min. feed.

  F5. 5 mm/min. feed.

  ```
  LA1 = 8
  F = LA
  ```

  8 mm/min. feed.

3. Operating Procedure

The data setting procedure for F1-digit feed by parameters feed format is described below.

(1) On-Screen Display

When in the automatic or MDI operation mode, press the function key [F2] shown in Fig. 12-1 to proceed to the Parameter F1-Digit Setting screen as shown in Fig. 12-2 below.

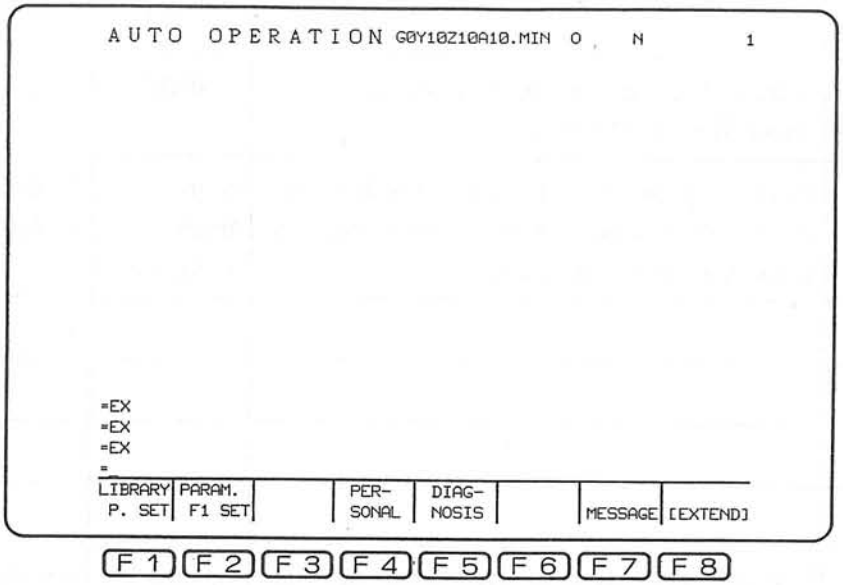


Fig. 12-1 Function Keys in the Automatic (MDI) Operation Mode

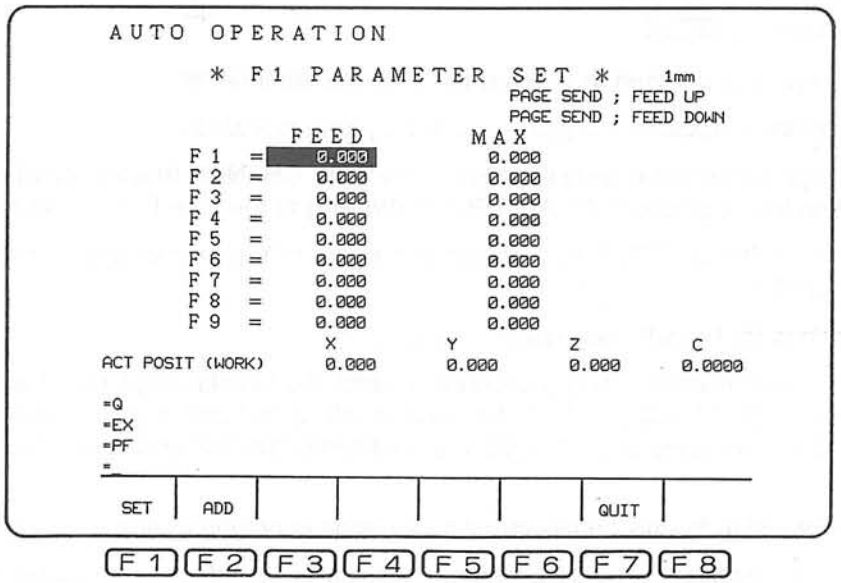


Fig. 12-2 Parameter F1-Digit Setting Screen

(2) Unit System Setting

In the same manner as for other data setting modes (zero setting mode, tool data setting mode, and parameter setting mode) the unit system is determined by the setting at NC optional parameter (bit) No. 9.

The unit system which is currently selected will be displayed at the 3rd line of the on-screen display.

Parameter No.	Bit No.	Description	When "1" is set:	When "0" is set:	Initial Factory Setting
9	0	In the data setting mode, the length is designated as "INCH" or "MM".	INCH	MM	0
	1	In the data setting mode, data is set in units of "1mm", "1 inch", "1 deg", and "1 sec", or in the minimum setting unit.	1mm, 1inch, 1deg, 1sec	Minimum setting unit	1
	2				0
	3				0
	4				0
	5				0
	6	For settings involving decimal points, the unit system for decimal point position is designated as "1 mm", "1 inch", "1 deg", "1 sec".	Yes	According to bit 1	0
	7				

(3) Function Key Commands

Function keys may be used to execute the following commands:

For data setting purposes, the [SET] and [ADD] keys are used.

If the number of controlled axes exceeds 4, the AXIS CHANGE function can be displayed. When this function key is pressed, the ACT POSIT (WORK) at the 17th line can be switched.

To end the F1 PARAMETER SET screen and return to the original display page, press the function key [QUIT].

(4) Updating Data By Page Forward/Back Operation

In addition to the normal setting procedure in which the function keys [SET] and [ADD] are used, the page keys [P ↓] and [P ↑] can be used to add or subtract a given value (designated at the optional parameter (long word) No.22) in a continuous manner over a number of pages. (Details follow.)

A timer is provided for this continuous data updating operation to prevent abrupt data changes.

Moreover, the addition and subtraction of the variable speed data designated at the optional parameter (long word) No.22 is always executed using the "minimum setting unit" value ("1 μm" for MM, "1/10,000 inch" for INCH).

4. Minimum and Maximum Values of Setting Data

① Maximum Feed Value (right-side data in Fig. 2)

- Minimum value : 0 (For both "inch" and "mm" unit systems)
- Maximum value : Maximum F command value (4,000 mm/min or 157.4803 ipm)

② Feedrate (left-side data in Fig.2)

- Minimum value : 0 (For both "inch" and "mm" unit systems)
- Maximum value : The maximum value in (1) above (maximum feedrate)

If the maximum value is exceeded during data addition operations executed using the keys, processing will occur as follows:

- Feedrate : The calculated feedrate will be changed to the "maximum feed value", and set accordingly.
- Maximum Feed Value : If the "maximum feed value" is exceeded, the following error will be activated:
  2509 Input data overflow

5. Other Items

(1) System Variables

All setting data are held in system variables and can be referred to and set in programs.

System variable name	Description	Subscript command	Remarks
VPF?F[*]	F1-digit parameter Feedrate	With 1 to 9	Unit conversion data
VPF1M[*]	F1-digit parameter Maximum feed value	With 1 to 9	Unit conversion data
VPF1C	F1-digit parameter Variable speed data	Without	Optional parameter (long word) No.22

* Subscript from 1 to 9

(2) Backup

The F1-digit parameter setting data is automatically backed up in memory at fixed intervals.

Therefore, this setting data will be saved even if the power is switched OFF.

(3) Variable Speed Data Setting

Acceleration/deceleration data of F1-digit parameters "feedrate" and "maximum feed value" is set at optional parameter (long word) No. 22 within the following setting range:

Minimum value : 0
Maximum value : 4,000,000

(4) F Displays on Actual Position Display Screen and Other Screens

On the ACTUAL POSITION and BLOCK DATA screens in the automatic operation mode, the F1-digit command is displayed in the same manner as the normal F4-digit command.

Metric system .... Displayed to one digit following the decimal point (the numeral at the 2nd decimal place is rounded off)

Inch system ..... Displayed to two digits following the decimal point (the numeral at the 3rd decimal place is rounded off)

6. Precautions

(1) The F1-digit feed status (F1-F9) is not cleared when the NC is reset. It can be cleared by designating a direct F command or by switching the power OFF and back ON again.

(2) The F1-digit feedrate setting may be changed while cutting is in progress.

(3) The feedrate override function is inoperative while the F1-digit feed command is being executed.

(4) An alarm will be activated if the F1-digit feed command is executed while in the "feed per revolution (G95)" mode.

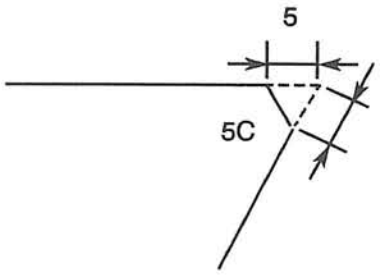
SECTION 13 ARBITRARY-ANGLE CHAMFERING FUNCTION

1. Outline

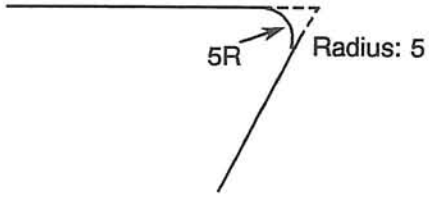
For some workpieces, chamfering (C-chamfer, R-chamfer) is desirable at the corners.

If the corners have arbitrary angles, however, this presents difficulties when chamfering is executed using usual programs (G01, G02, G03).

The "Arbitrary-angle chamfering function" enables the start and end points of chamfering to be determined automatically simply by designating the corner's apex and the desired chamfer amount.



C-chamfer (chamfer amount: 5 mm)



R-chamfer (chamfer amount: 5 mm)

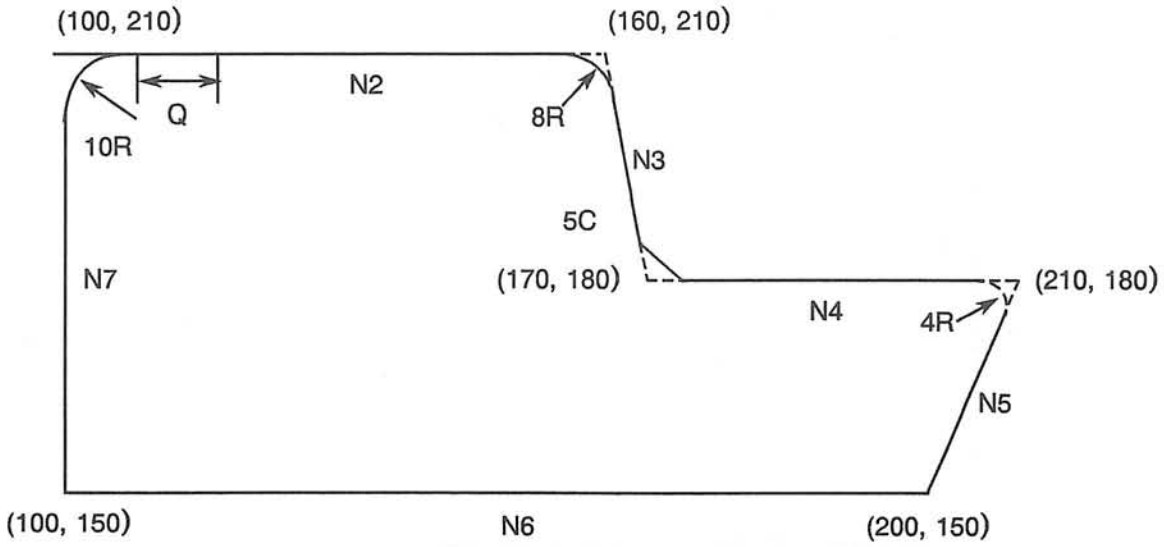
2. Programming Format

CHFC (CHFR) Xp__ Yp__ L__ Q__ I__ J__ ..... On the G17 plane

CHFC (Chamfer C)	C-chamfer command, one shot
CHFR	R-chamfer command, one shot
Xp, Yp	Corner apex, target value
L	Chamfer amount, one shot (positive number) L = 0 - 99999999
Q	Travel amount after chamfer (travel amount toward next block's target value after completion of corner chamfer), one shot (positive number) Q = 0 - 99999999
I, J	Virtual advancing direction (vector), one shot Direction is indicated, but command value is according to the unit system.)

3. Function

The chamfering command (C-chamfer, R-chamfer) is executed at the corner area between the block in which the chamfer command has been designated and the block following it.



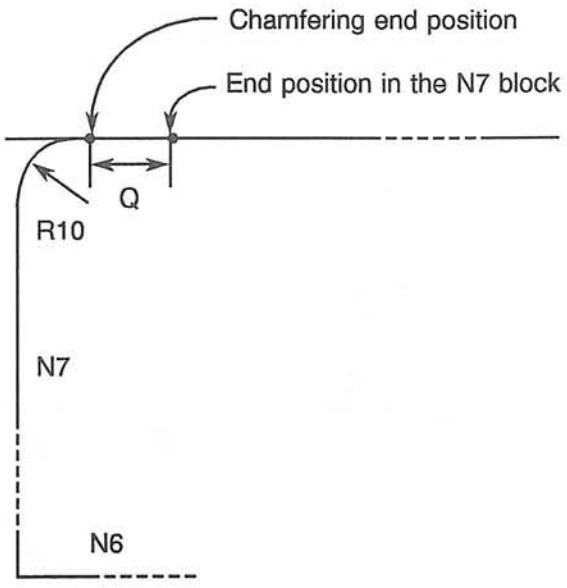
(Program Example)

```
                :
                :
N1        G0  X100    Y210
N2  CHFR  G1  X160            L8      F2000
N3  CHFC      X170    Y180    L5
N4  CHFR      X210            L4
N5            X200    Y150
N6            X100
N7  CHFR                Y210    L10     Q10     I1
                :
                :
```

When a Q command is present:

In order to prevent uncut and cutter marks following the chamfering of the final corner (N7 & N2 corner) in multi-cornered shapes such as that shown in Fig.1 above, a slight amount of overlapping cutting is required.

In such cases, use a Q command to designate the amount of overlapping cutting. After corner chamfering is completed, axes will amount toward the point specified in the next block only by this Q amount.



When the I or J command is present (virtual advancing direction):

When chamfering the final corner of a multi-cornered workpiece as shown in Fig.1, the I or J command can be used to execute the chamfering operation by designating the virtual advancing direction.

When the I or J command is designated in the chamfering command block, operation will occur based on the designated this virtual advancing direction.

Note 1: *The above explanation is made assuming that the G17 plane (X-Y plane) is selected. The following applies when other planes are selected:*

G18 plane . . . Zp, Xp, K, I

G19 plane . . . Yp, Zp, J, K

4. Precautions

(1) The chamfering command is effective only in the G01 mode. If attempted in another mode, an alarm will be activated.

(2) Chamfering will be executed on the plane which has been designated in the plane selection operation.

(3) In order to ensure that chamfering is executed at the corner area between two straight lines, the mode for the next block must be either G00 or G01. If any other mode is designated, an alarm will be activated.

(4) An axis travel command must be present in the next block following a chamfering command.

(5) A chamfering command is also effective during an AG command, or during cutter radius compensation.

(6) When a chamfering command is designated in the cutter radius compensation mode, virtual advancing direction (I and J commands) of the cutter in regard to cutter radius compensation cannot be designated.

Regardless of the mode selected, any Q, L, I, or J commands which have been designated in a chamfering block will be taken as chamfering Q, L, I, or J commands.

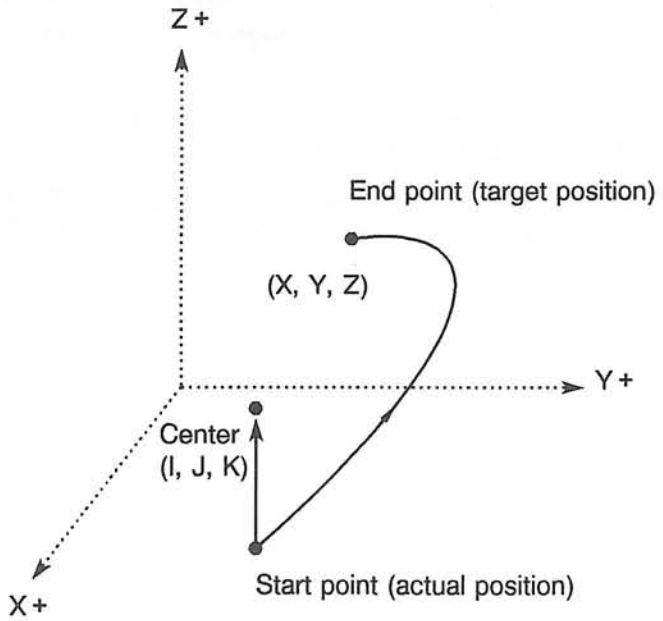
(7) To execute a chamfering command in the incremental mode, designate the travel amount in the next sequence in terms of the travel amount from the end point of chamfering.

SECTION 14 3-D CIRCULAR INTERPOLATION FUNCTION

1. Overview

(1) The 3-D circular interpolation function executes circular interpolation in a required plane in addition to XY, YZ, and ZX planes in which circular interpolation is possible with conventional function.

3-D circular interpolation function:



(2) Designation of the center and end point of the required arc makes axis motion possible along an arc in a spatial plane.

2. Command Format

G02/G03 Xp__Yp__Zp__I__J__K__(P__Q__R__)

G02, G03 : Rotating direction (see 3., "3-D Arc Rotating Direction")

Xp, Yp, Zp : Coordinate values of end point

Xp ..... X or U axis
Yp ..... Y or V axis
Zp ..... Z or W axis

I, J, K : Coordinate values of arc center

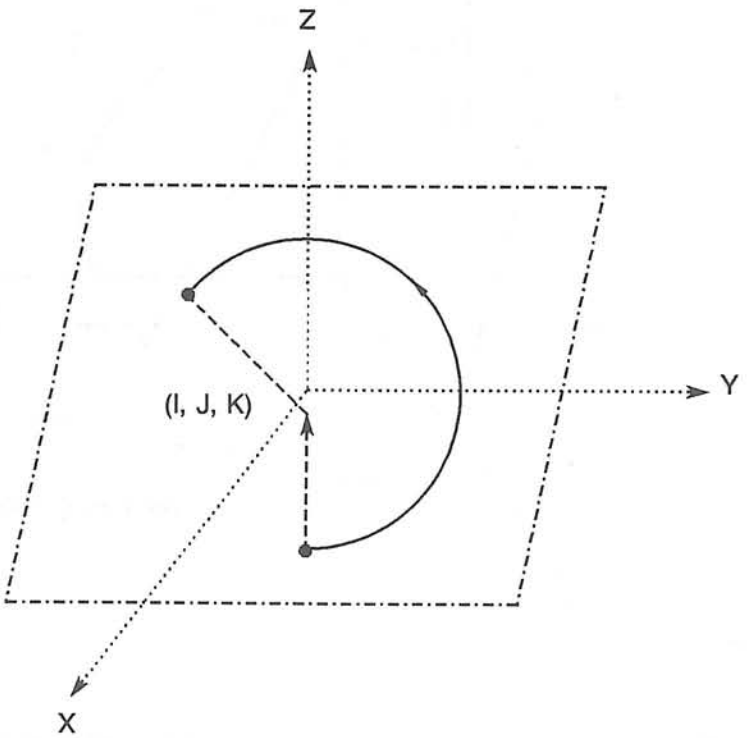
(P, Q, R) : Coordinate values of a point in the arc in reference to the arc start point

These are necessary if a plane cannot be determined by Xp, Yp, Zp, I, J, and K.

(1) When all of I, J, and K are designated, the arc is assumed to be 3-D arc. In other cases, the arc is assumed to be 2-D arc in XY, YZ, or ZX plane.

(2) An arc is defined in the plane where the following three points lie:

- Start point (actual position)
- Arc center
- End point (target position)



(3) The end point can be designated in either absolute value or incremental value according to the G code (G90 or G91).

(4) The coordinate values of the arc center should be designated in incremental value regardless of G90 or G91.

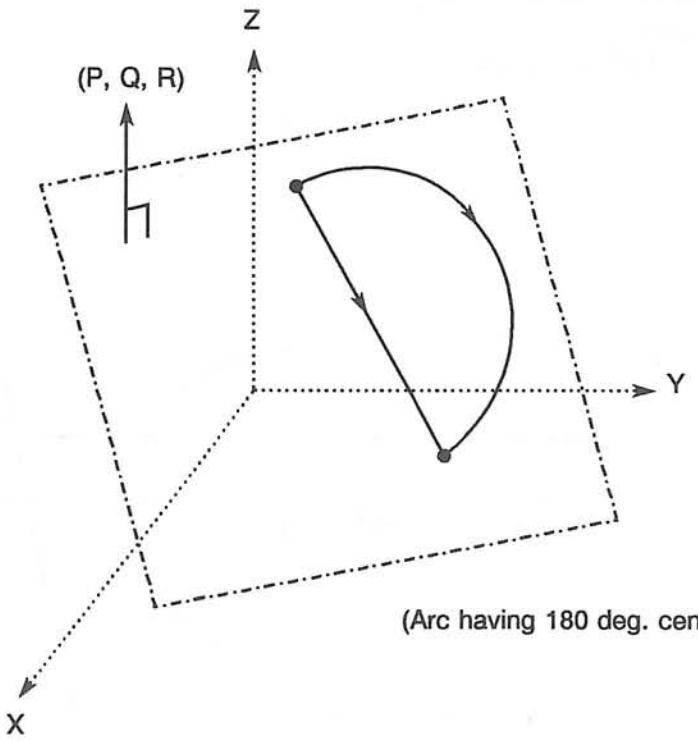
(5) If the start point, arc center, and end point are on a straight line (arc to be defined has 180 deg. or 360 deg. central angle), the plane in which the arc should be defined cannot be determined. In this case, either the following items by designating (P, Q, R) is necessary so that the plane can be determined.

- Coordinate values outside the straight line
- Normal vector on the plane including the arc set "0" or "1" at NC optional parameter (bit) No. 37 bit 5 for selecting either the above items.

(6) When P, Q, and R commands are designated with setting "0" at the optional parameter (bit) No. 37, but 5, an arc is defined in the plane where the following three points lie:

- Start point (actual position)
- Arc center
- Point defined by (P, Q, R)

Example:

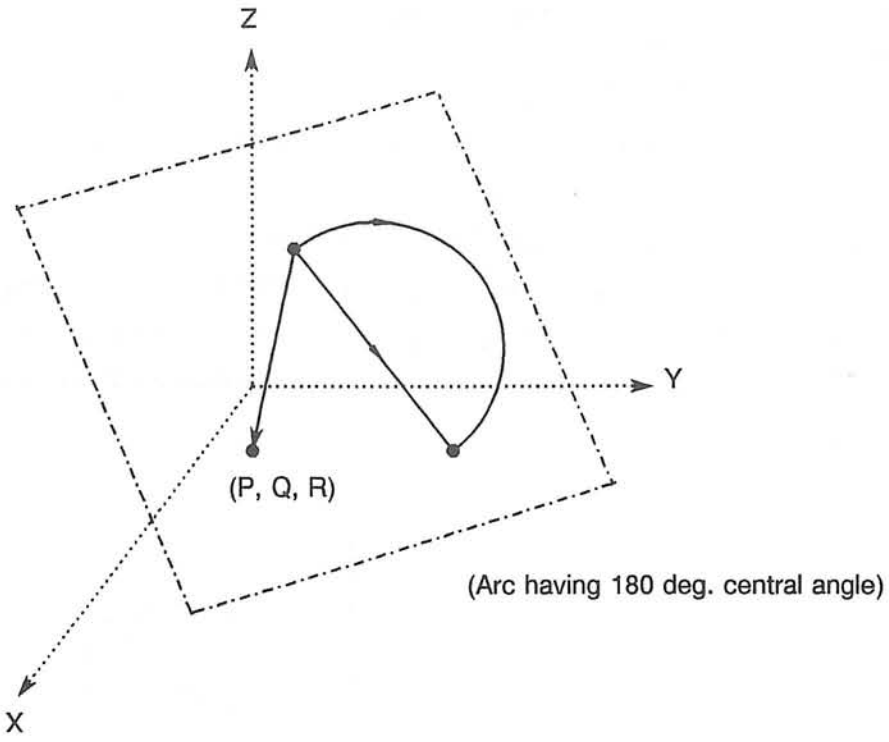


(Arc having 180 deg. central angle)

When P, Q, and R commands are designated with setting "1" at theoptional parameter (bit) No. 37, but 5, an arc is defined in the plane where the following two points and a vector lie:

- Start point (actual position)
- Arc center
- Vector defined by (P, Q, R)

Example:



(7) The coordinate values of (P, Q, R) used to define the plane should be designated in incremental value regardless of G90 or G91.

Note 1: *If any of Xp, Yp, and Zp is not designated, the coordinate value designated in a preceding block becomes effective for the omitted axis.*

Note 2: *If (P, Q, R) is designated, the plane is defined using the start point, arc center, and (P, Q, R) even when the plane can be defined using the start point, arc center, and end point.*

3. 3-D Arc Rotating Direction

The arc rotating direction should be designated based on the arc rotating direction when the 3-D arc is projected on the 2-D plane.

Command Format: G256 I__(J__,K__) (designating the plane of projection)

I+: The arc projected on the G19 plane is viewed from the X-axis positive direction.

I−: The arc projected on the G19 plane is viewed from the X-axis negative direction.

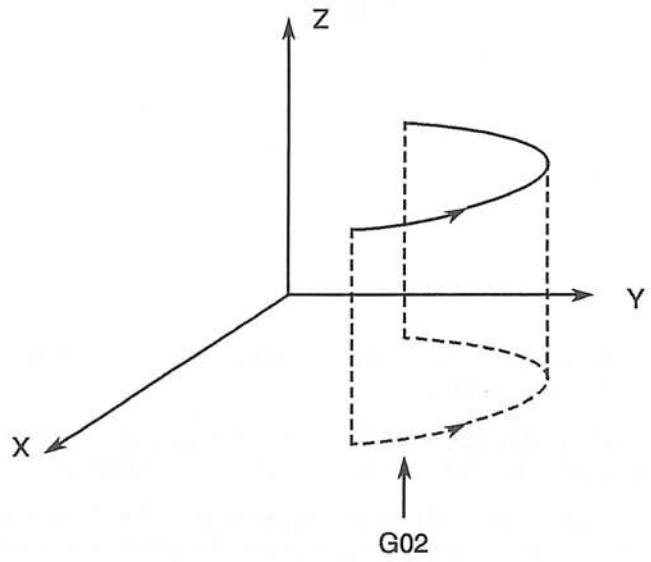
J+: The arc projected on the G18 plane is viewed from the Y-axis positive direction.

J−: The arc projected on the G18 plane is viewed from the Y-axis negative direction.

K+: The arc projected on the G17 plane is viewed from the Z-axis positive direction.

K−: The arc projected on the G17 plane is viewed from the Z-axis negative direction.

Example:



G256 K-1

G02 X__Y__Z__I__J__K__

In the illustration above, when the arc projected on the G17 plane is viewed from the Z-axis negative direction, the rotating direction is G02 (CW).

To cancel the plane of projection, designate as follows:

G256 or G256 I0 (J0,K0)

Note 1: Designate an arbitrary numeral following I, J, and K. Any of the I, J, and K words must be designated.

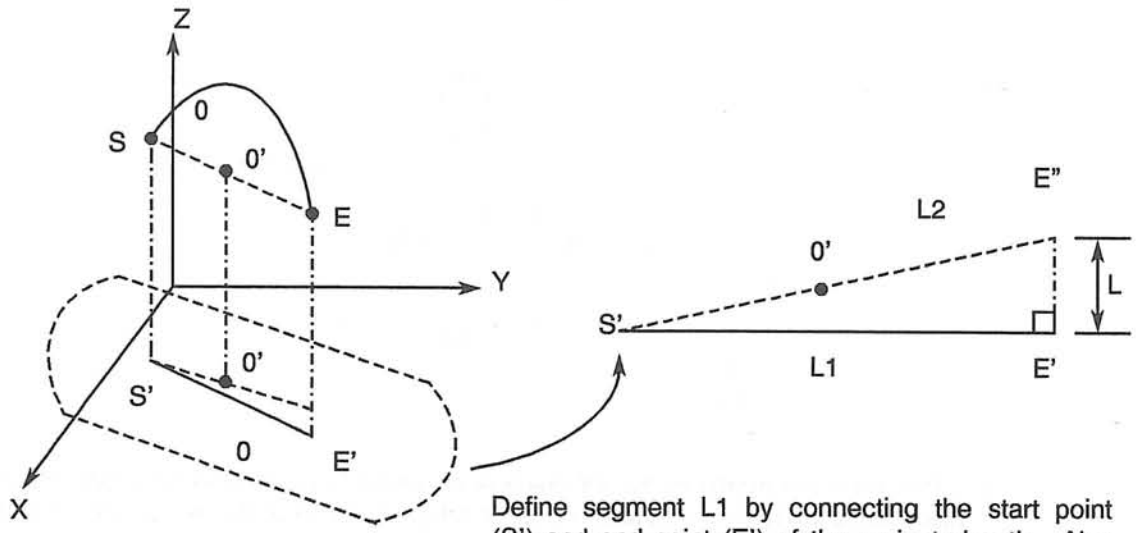
Note 2: When another plane of projection has been designated while a plane of projection is selected, the plane designated later becomes effective.

```
    ⋮
G256   I-100
G02    .....
G256   J100        I-100 is canceled and the J100 mode is established.
    ⋮
```

Note 3: *If P, Q, and R words are designated simultaneously of G256, G256 is ignored.*

Note 4: *When an arc rotating direction cannot be determined on the plane designated by G256, an alarm occurs. An arc rotating direction cannot be determined when the projected path of the 3-D arc is taken as a straight line.*

Note 5: *Whether or not the projected path of a 3-D arc is a straight line is judged while referencing the tolerance (allowable amount) set at the NC optional parameter (word) No. 59. Here, the tolerance refers to the allowable width within which the projected path is taken as a straight line. This allows the arc rotating direction to be determined correctly.*



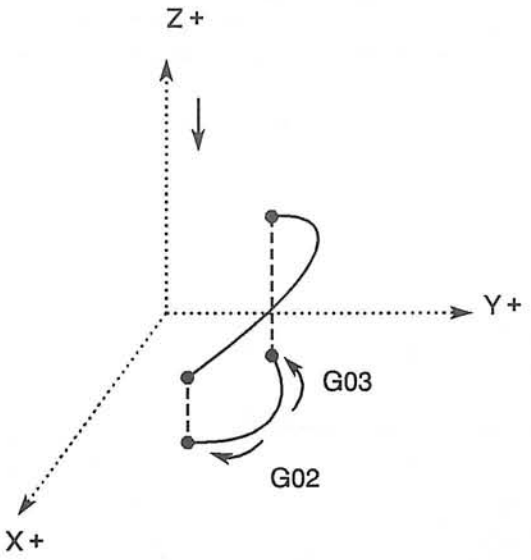
Define segment L1 by connecting the start point (S') and end point (E') of the projected path. Also define segment L2 by connecting S' and the center point (0'). Draw vertical line at point E' to L2; the point of intersection between L2 and the vertical line is defined as E". Length E'E" is defined as L which is compared to the value set for optional parameter (word) No. 59.

$\lvert L \rvert \leqq$ Optional parameter (word) No. 59	$\lvert L \rvert >$ Optional parameter (word) No. 59
The projected path is assumed to be straight line.	The projected path is not assumed to be straight line.

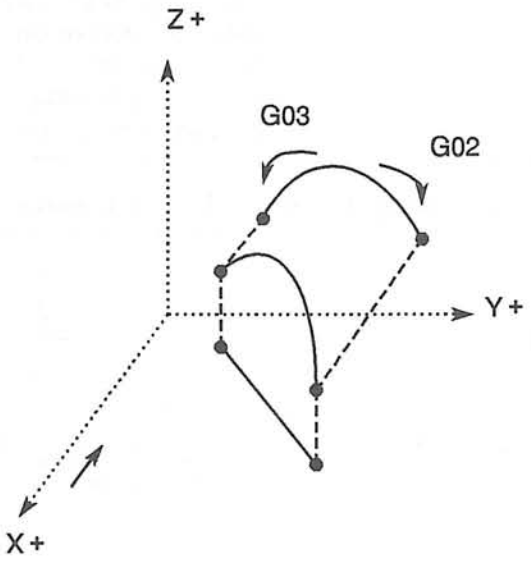
(1) Arc Rotating Direction Definition without G252 Command

(a) The arc rotating direction (G02, G03) is defined based on the rotating direction of the arc projected on the XY plane and viewed from the +Z direction. See the illustration below.

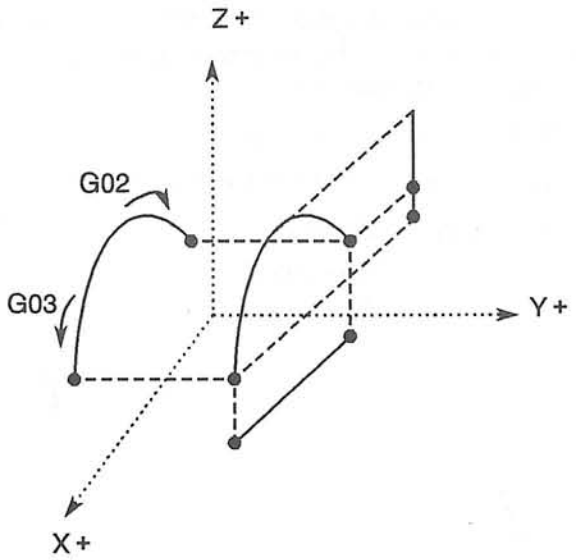
G02 . . CW
G03 . . CCW



(b) If the projected profile in the XY plane is assumed to be straight line (See Note), the arc rotating direction is defined based on the rotation direction of the arc projected on the YZ plane and viewed from the +X direction.



(c) If the projected profile in the XY plane and that in the YZ plane are both assumed to be straight line (See Note), the arc rotating direction is defined based on the rotation direction of the arc projected on the ZX plane and viewed from the +Y direction.



Note: *Optional parameter (word) No. 59 is used as the "tolerance to determine whether the projected line of 3-D arc on a plane is straight line or not" (Refer to Note 5.)*

(2) The projection plane where the 3-D arc is projected can be fixed to a required plane by designating G17, G18, or G19. Whether this function is effective or not can be changed by the setting for optional parameter (bit).

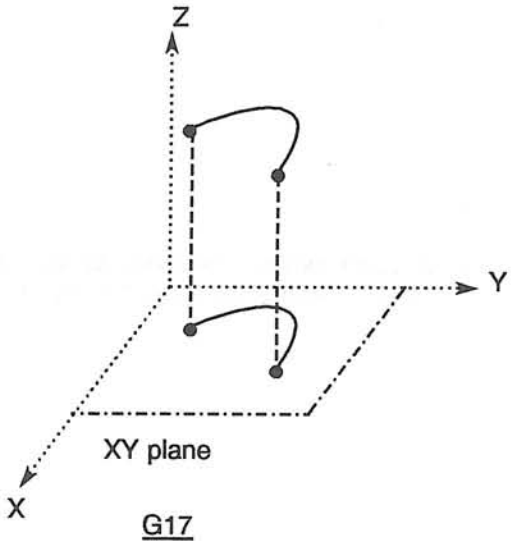
(a) The G17, G18, and G19 codes are used to designate the plane on which the 3-D arc is projected to determine the rotating direction of the 3-D arc. In this case, the following alarm occurs if the rotating direction cannot be determined by the projected path because the projected path is determined to be straight line.

ALARM B 454 Data word: arc calculation 9

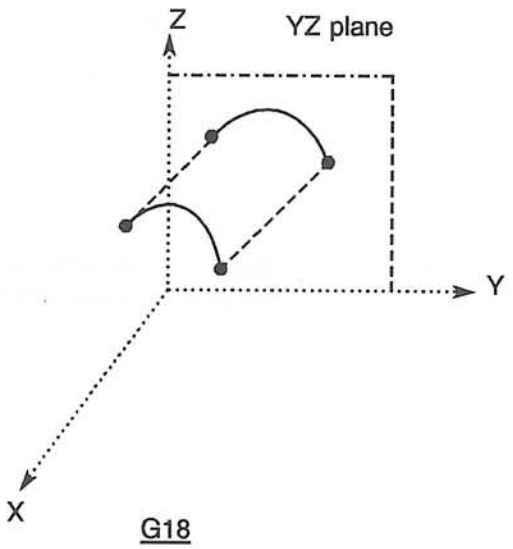
(b) The following parameter is used to set this function effective or ineffective.

Optional parameter (bit) No. 37, bit 3

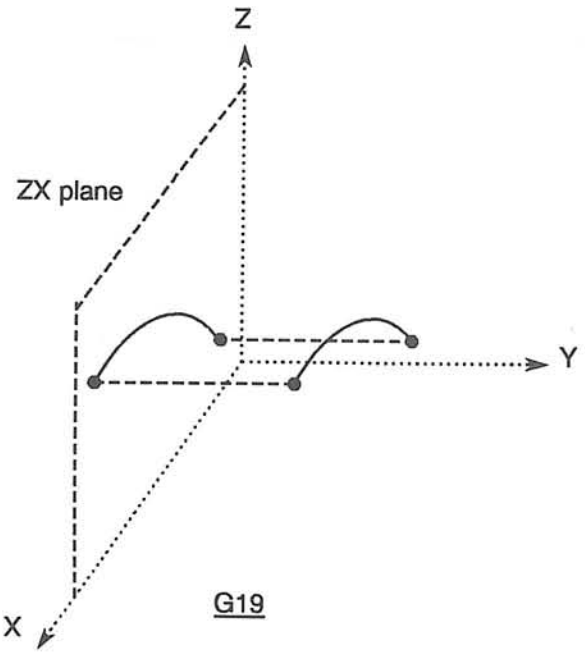
0 . . . . . Ineffective
1 . . . . . Effective



G17



G18



G19

Example:

```
N1  G0  X100  Y0  Z0
N2  G17
N3  G02  X0  Y0  Z100  I-33.333  J66.667  K66.667  F1000
N4  G18
N5  G02  X0  Y100  Z0  I66.667  J66.667  K-33.333
N6  G19
N7  G02  X100  Y0  Z0  I66.667  J-33.333  K66.667
N8  G0  X0  Y0  Z0
N9  M02
```

The 3-D arc rotating direction defined by the N3 block is determined by the G02 code using the path projected on the XY plane. Similarly, the 3-D arc rotating direction defined by the N5 block and that by the N7 block are determined by the G02 code using the path projected on the YZ and ZX plane, respectively.

4. Shorter Arc and Longer Arc

The circular interpolation will be performed from the are starting point to the are end point for shorter or longer are wite the designated center.

Command Format: G172/G173 X_Y_Z_I_J_K_(P_Q_R_)

G172 : Shorter arc

G173 : Longer arc

X, Y, Z : Coordinate values of end point

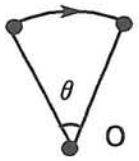
I, J, K : Coordinate values of arc center

P, Q, R : Normal vector of the plane which includes the arc

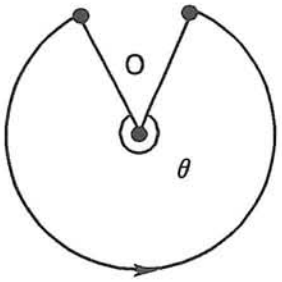
(When "0" is set at the optional parameter (bit) No. 37, bit 5)

Arbitrary vector from the start point on the arc plane

(When "1" is set at the optional parameter (bit) No. 37, bit 5)



G172 : $\theta < 180°$



G173 : $\theta > 180°$

Note 1: *When angle $\theta = 180°$, the rotating direction may be reverse depending on the command error. In such cases, designate G256 (designation of projection plane) or any of P, Q, and R. When this is done, circular interpolation will be executed as follows.*

G172 → G02

G173 → G03

5. Caution

(1) The cutter radius compensation function (G41, G42) cannot be used.

(2) High speed drawing of line drawing and animated simulation is not possible.

(3) Envelope of animated simulation is not possible.

(4) The tool length offset function must be set effective in a block before the 3-D arc is defined.

In other words, the G53 - G59 codes must not be designated in the same block as the 3-D arc definition block.

6. Alarm Codes

ALARM B

445 Data word: 'I', 'J' or 'K'

Code :

None ..... In the G256 mode, two or more of I, J, and K commands have been designated.

454 Data word: arc calculation

Code :

11 ..... In 3-D circular interpolation, the designated axis has been canceled.

8 ..... In 3-D circular interpolation, no P, Q, or R command is designated while the start point, center point, and end point are on a straight line.

12 ..... In 3-D circular interpolation, the 3-D coordinate conversion command or an axis other than the rotary axis which was copied in the G11 mode has been designated.

4 ..... In the G172 or G173 mode, I, J, and K commands have not been designated. Or 0 has been designated for I, J, and K.

455 Data word: plane selection

Code : Commanded axis number

None ..... X- and U-axes

1 ..... Y- and V-axes

2 ..... Z- and W-axes

458 Data word: coordinate conversion

Code :

1 ..... An error has been generated when the normal vector which had been copied in the G11 mode was being rotated.

565 Data word: rotating axis command

Code :

1 ..... A rotary axis has been designated in 3-D circular interpolation.

7. Parameters

(1) Optional Parameter (word) No. 59

Tolerance to determine whether the projected path of 3-D arc on a plane is straight line or not

Initial value	:	0
Minimum value	:	0
Maximum value	:	32767
Setting unit	:	μm
Effective command	:	S (setting) only

(2) Optional Parameter (bit) No. 37, bit 3

1 : 3-D arc (G17, G18, G19) projection plane fixing function effective

0 : 3-D arc (G17, G18, G19) projection plane fixing function ineffective

Initial value : 0

(3) Optional Parameter (bit) No. 37, bit 5

1 : Coordinate value (P, Q, R) on the plane including arc.

0 : Normal vector (P, Q, R) on the plane including arc.

SECTION 15 AUTOMATIC ATTACHMENT INDEXING FUNCTION

1. Overview

This function indexes an attachment in intervals of 5 degrees, 15 degrees, or 45 degrees.

The index angle intervals (5, 15 or 45 degrees) are determined by the machine specification.

Refer to the specification for approval.

2. Index Commands

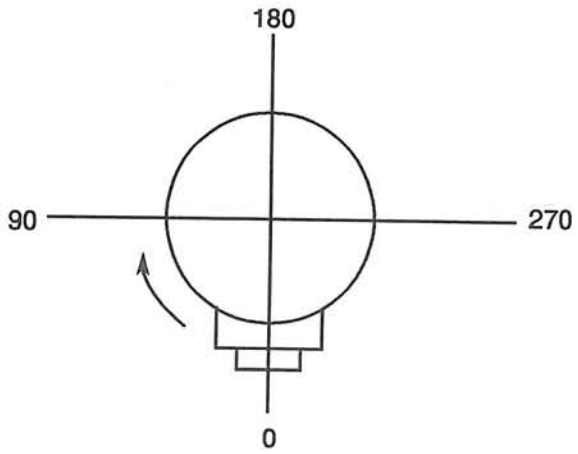
The attachment index command includes the direction through which the attachment is rotated.

M94 . . . Attachment indexing in the forward direction
M95 . . . Attachment indexing in the reverse direction

3. Angle Commands

Attachment index angle is defined as absolute angle position in the forward rotation direction with the front position taken as 0 degree.

RH = ***
Angle 0 - 359



4. Command Format

M ** RH = ***
Rotating direction
Angle

(1) The M code and an angle command must be designated in the same block.

(2) The angle command should be given after the spindle rotation has stopped.

5. Alarm

ALARM B

618 Data word: 'RH'

Attachment indexing angle value is outside the range of 0 to 359, or an attachment angle not meeting the specification is designated.

Index	:	None
Code	:	Hexadecimal of commanded value

SECTION 16 AXIS NAME DESIGNATION FUNCTION

1. Outline

As a rule, coordinate axis names used in the NC program are specific to each machine model. For example, table motion is defined as the X-axis

However, the axis name designation function enables the axis names which are used in a program to be selected by the programmer.

When this function is used for machining with an angular attachment, or for MCM five-face machining models, the tool travel axis (cycle axis) can be handled as the Z-axis at all the machining surfaces.

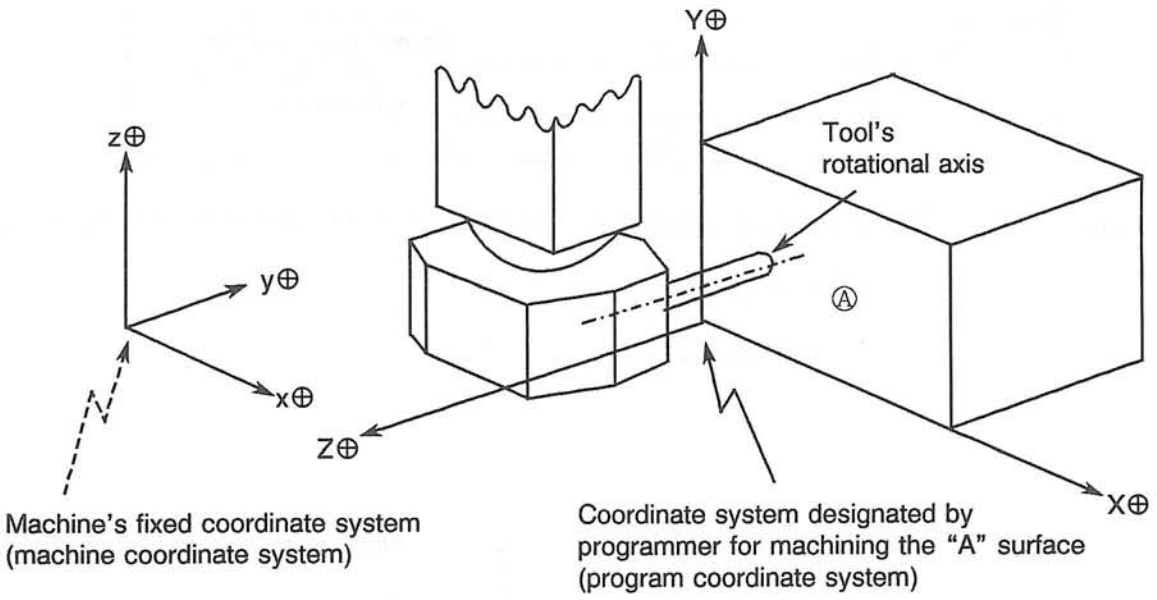


Fig. 16-1 Program Coordinate System

By designating the axis names as shown above, the Z-axis and the Y-axis in the program coordinate system will correspond to the Y-axis negative direction and the Z-axis positive direction in the machine coordinate system, respectively.

2. Axis Name Designation

The names of the standard axes (X, Y, Z) are designated by G-code commands.

2-1. Renaming a Machine Axis

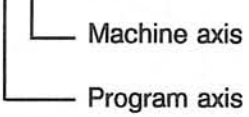
Programming Format [G14 Xx Yy Zz]

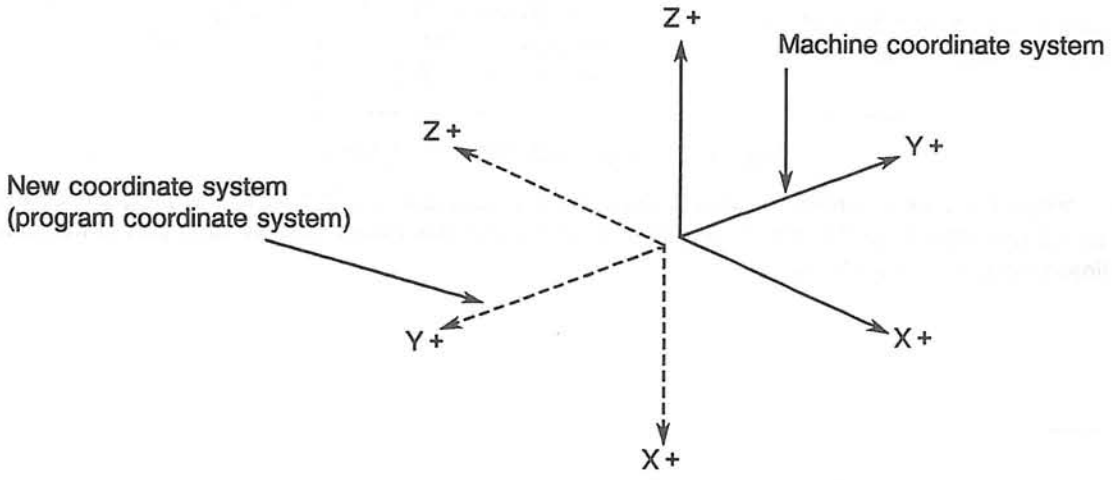
X,Y,Z ..... Program axes (new axis names)

x,y,z ..... Machine axis No.

x,y,z values	Corresponding machine-specific axis names	Remarks
±1	X	A minus (−) symbol indicates the opposite axis direction.
±2	Y	
±3	Z	

Example: For a "G14 X-3 Y-2 Z-1" designation, the new coordinate system will be as shown below.





The following command is used to cancel the axis switching function (return to the machine coordinate system):

Programming format: [G14] or [G14 X1 Y2 Z3]

Note 1: The rotating direction of 3-D circular interpolation is judged taking the selected plane as the projection plane. (same as when the optional parameter (bit) No.37 bit3 is "1".)

- The following alarm will be activated if the program axis names are not designated for all of the X, Y, and Z axes (except for the cancel command).

  Alarm B 452 Data word: axis command

Codes	:	01	X-axis command only
		02	Y-axis command only
		03	X- and Y-axis command only
		04	Z-axis command only
		05	Z- and X-axis command only
		06	Z- and Y-axis command only

- The following alarm will be activated if the machine axis No. designation is other than ±1, ±2 and ±3 or if the same numbers exist.

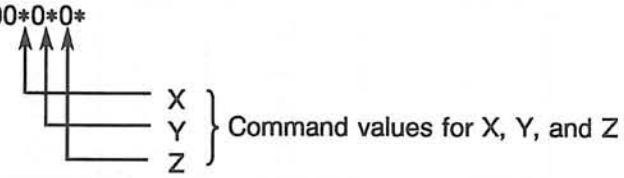
  Alarm B 452 Data word: axis command

  Code : Command value

- Machine axis No. combinations are shown in Table 1. and in Fig. 2. If other combinations are used, the following alarm will be activated:

  Alarm B 452 Data word: axis command

  Code : 000∗0∗0∗

- The following alarm will be activated if designation is attempted during the tool length offset, cutter radius compensation, fixed cycle, 3-D tool offset, and/or arc mode.

  Alarm B 462 Data word: G code 18

- Once designated, a coordinate system will not be changed when the power is switched ON or OFF, or when the NC is reset.

Table 1 Coordinate Systems And Axis Designation Nos.

Machining surface	Coordinate system No.	Axis designation No.			Remarks
		X	Y	Z	
A	1	1	2	3	
	2	−1	−2	3	
	3	2	−1	3	
	4	−2	1	3	
B	5	1	−3	2	
	6	−1	3	2	
	7	−3	−1	2	
	8	3	1	2	
C	9	1	3	−2	
	10	−1	−3	−2	
	11	3	−1	−2	
	12	−3	1	−2	
D	13	−3	2	1	
	14	3	−2	1	
	15	2	3	1	
	16	−2	−3	1	
E	17	3	2	−1	
	18	−3	−2	−1	
	19	2	−3	−1	
	20	−2	3	−1	

Machine coordinate system

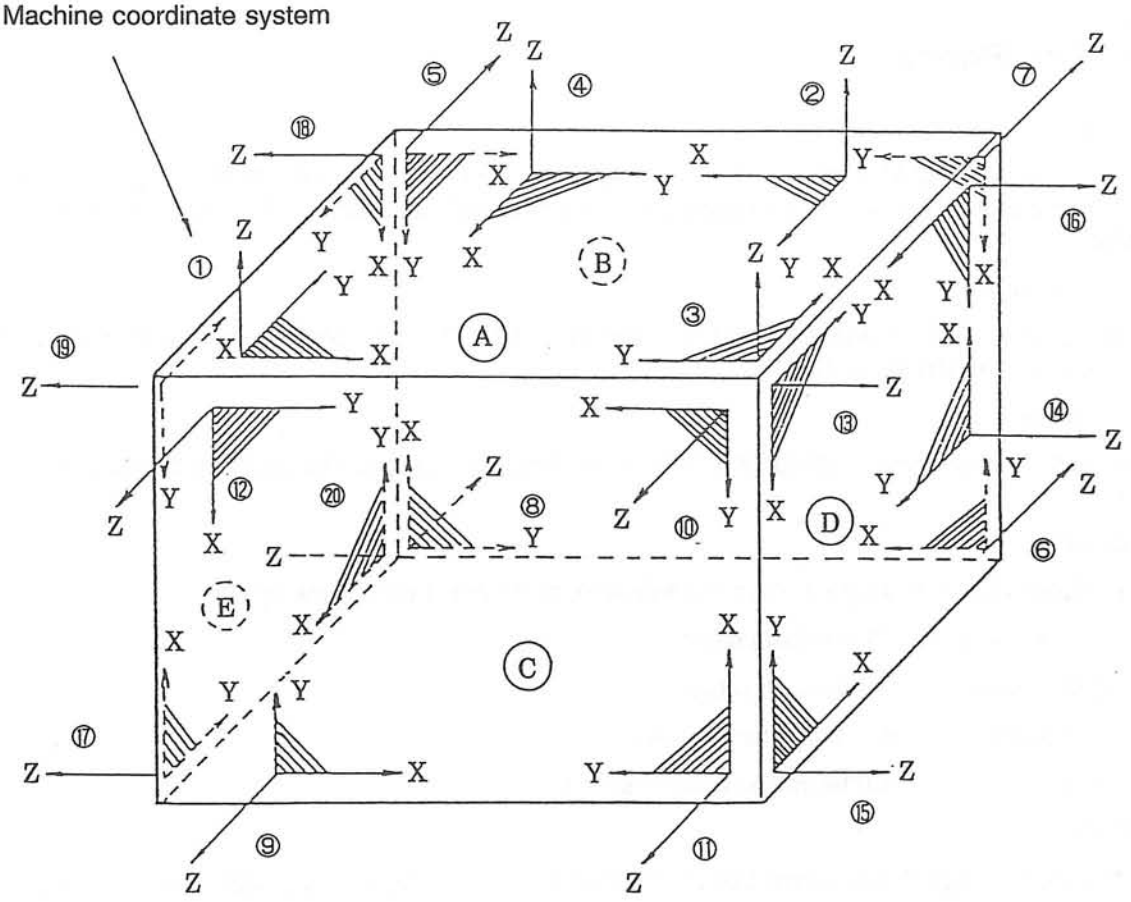


Fig. 16-2 Coordinate System Types

3. Application Range

(1) Modes

In the automatic and MDI modes, the NC program is processed according to the program coordinate system. In other modes, processing will be performed according to the machine coordinate system.

(2) Command Axis

Standard axes (X,Y,Z) and axis-related commands (I,J,K,P,R) are also processed according to the program coordinate system.

(3) External Axis

The mirror image switch setting (X,Y,Z) is processed according to the program coordinate system.

(4) G Codes

Axis related G codes are processed according to the program coordinate system.

G17, G18, G19 (Plane designation)

G54, G55, G56 (Tool length offset)

G02, G03 (Circular interpolation)

G41, G42 (Cutter radius compensation)

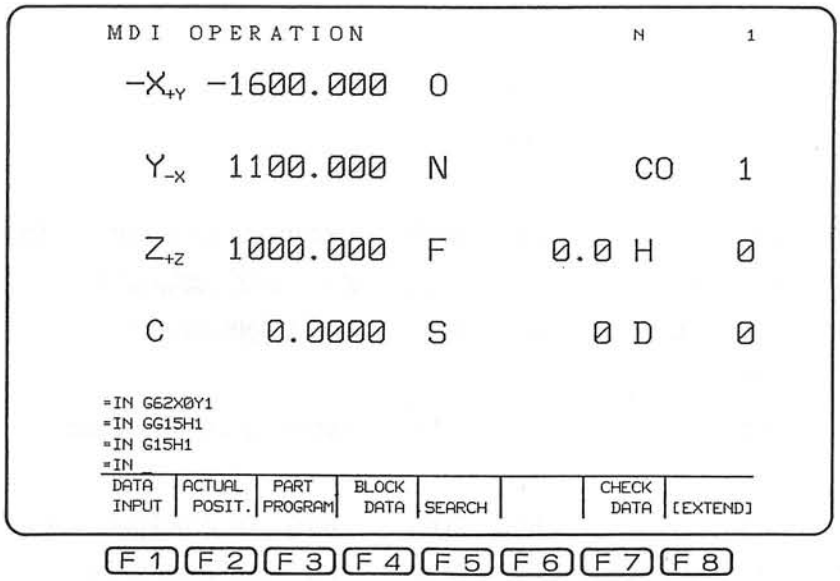
(5) M Codes

All M codes except those shown below are processed according to the program coordinate system.

M22, M23, M24, M25 (Clamp/unclamp)

4. Display

The program axes are displayed at the lower right of the machine axes X, Y, and Z on the ACTUAL POSITION screen.

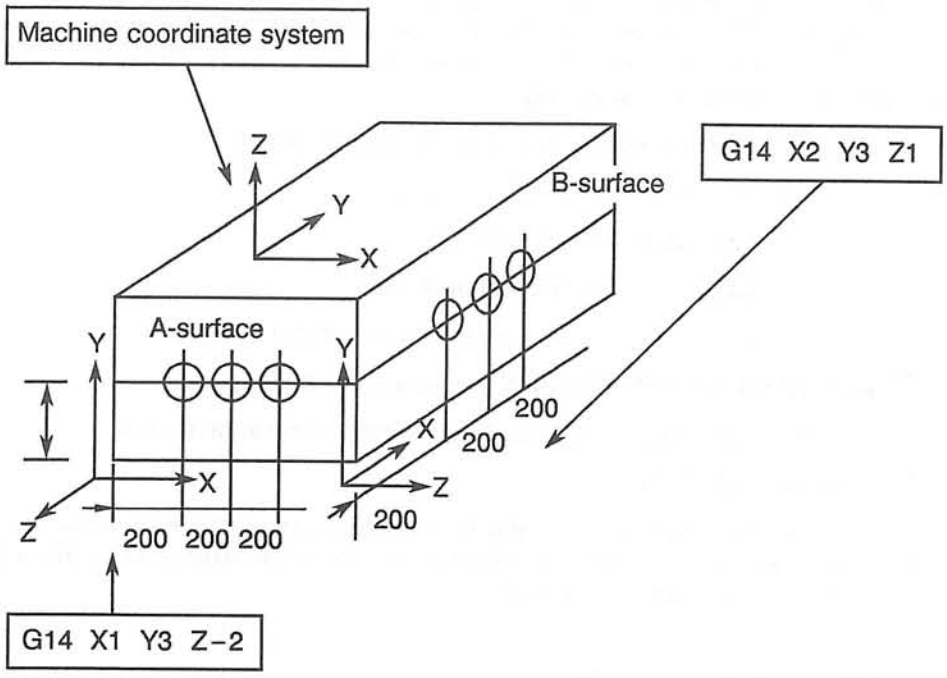


x 4 Display (or x 16 display)

5. Program Example (for MCM models)

(1) Fixed Cycle (Drilling) Program Example

```
O100
N01   G0    Z100
N02                    T1
N03                    M77
N04                    M63
N05                    M75
N06   G15               H1      N06:   Workpiece coordinate system selection 1
N07   G14   X1   Y3      Z-2     N07:   Axis name designation
N08   G56   Z100          H1      N08:   Tool length offset Z-axis
N09   CALL 01
N10   G53                        N10:   Tool length offset cancel
N11                    M74
N12   G15               H2      N12:   Workpiece coordinate system selection 2
N13   G14   X2   Y3      Z1      N13:   Axis name designation
N14   G56   Z100          H1      N14:   Tool length offset Z-axis
N15   CALL 01
N16   G53   G0   X1000   M77
N17   G14                        N17:   Axis name cancel
N18                    M2
O1
N1    G71   Z100
N2    G81   X20  Y20    Z-5   R40   F100
N3    X40
N4    X60
N5    G80
RTS
```



As shown in the above program, the same machining operation can be executed for all machining surfaces using a single subprogram by switching axes.

6. Precautions

(1) There are cases when G codes and actual position data in the BLOCK DATA screen differ from those in the NC program. This is because the BLOCK DATA screen shows the machine axis motion, and differences occur between the machine coordinate system and the program coordinate system due to axis switching.

G codes which may differ from those in the NC program:

G17, G18, G19	(plane designation)
G54, G55, G56	(tool length offset)
G02, G03	(circular interpolation)
G41, G42	(cutter radius compensation)

Axes which may differ from the program command:

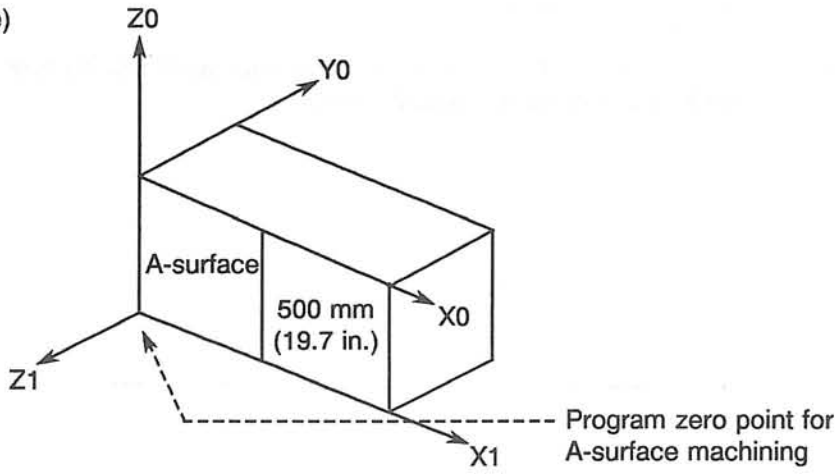
Standard axes (X, Y, Z) and axis-related commands (I, J, K)

(2) Zero Point Offset (G92)

A command value designated in the G92 mode is processed as a command value designated in the new coordinate system. Therefore, care should be exercised to the machining surface and axis to which the command applies.

(Program Example)

```
      :
      :
G0  X0  Y0  Z0
G14 X1  Y3  Z-2
G92     Y500
```



Before G92 is executed

Zero point offset	
X	2000
Y	2000
Z	2000

→

After G92 is executed

Zero point offset	
X	2000
Y	2000
Z	1500

- The value which is entered in zero offset mode is stored in memory as machine axis data. In the above example, the Z-axis (machine axis) zero point will be rewritten as shown above.

(3) Z-Axis Retract End (Upper Limit) Return

The M52 command which is designated for fixed cycle, coordinate calculations, and others is effective for the Z-axis (machine axis) in the machine coordinate system. If the Z-axis in the program is designated as the X or Y machine axis, an alarm of level B will be activated when the M52 command is designated.

(4) Feedrate

Normally, the Z-axis feedrate differs from that of the other axes (X, Y). Therefore, the following precautions should be observed during the programming operation:

- To designate a feedrate command (F-command), always verify the machining surface and the axis (machine axis) which will actually move. Special care must be exercised when designating the X- or Y-axis as the Z machine axis in the program, so that the maximum Z-axis feedrate is not exceeded.
- When the same part program is performed repeatedly by switching the coordinate system, if rapid feed commands are designated for multiple axes in the program, various positioning paths will be generated, and, depending on the command value, there may be cases in which the tool axis (Z-axis) will interfere with the workpiece. Use extra care to avoid such an occurrence.

(5) G Codes Effective for Machine Coordinate System

When a G code, for example G22 (stored travel limit), G30 (home position return), and others which are effective for the machine coordinate system, has been designated, it causes axis motion in the machine coordinate system, and does not cause axis motion in the program coordinate system.

[Example]

```
G22  X1000  Y1000   Z1000   I-1000 J-1000
     K-1000
```

↓

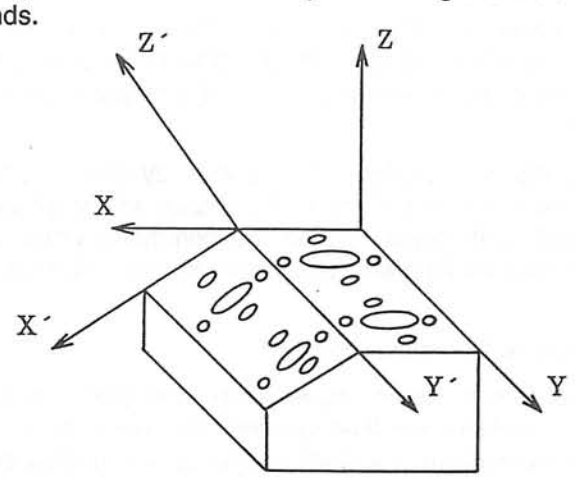
Positive travel end limits for X, Y, and Z = 1,000

Negative travel end limits for X, Y, and Z = −1,000

SECTION 17 SLOPE MACHINING FUNCTION

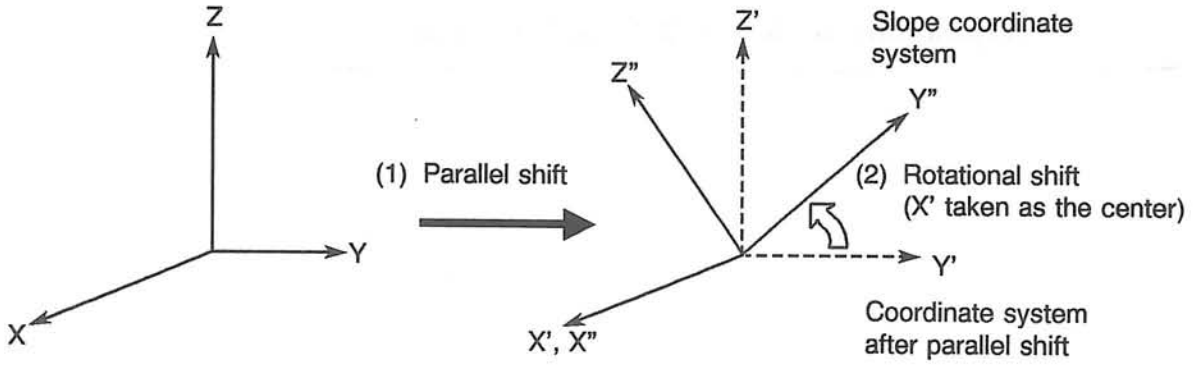
1. Outline

This function enables a part program created in the X-Y-Z coordinate system to be executed in any arbitrary coordinate system. This is done by converting the arbitrary coordinate system commands to machine commands.



X-Y-Z : Basic coordinate system
X'-Y'-Z' : Slope coordinate system

The arbitrary coordinate system shown above (created by executing a parallel or rotational shift of the basic coordinate system) is called a "slope coordinate system".



(1) Parallel shift of a coordinate system parallel is executed by designating the desired X, Y, and Z shift amount from the zero point of the basic coordinate system to the zero point of the arbitrary coordinate system.

(2) With either the X', Y', or Z' axis of the parallel shifted arbitrary coordinate system serving as the pivot point (center), the desired rotation angle is then designated to execute a rotational shift.

(3) The X"-Y"-Z" coordinate system created by steps 1 and 2 above is referred to as the "slope coordinate system".

This function permits up to 3 coordinate shift conversions, with steps 1 and 2 above being counted as 1 conversion.

2. Coordinate Conversion Function in the Automatic and MDI Operation Modes

To perform 3-D coordinate conversion, first create a slope coordinate system, and then designate machining commands in the normal manner in the slope coordinate system.

2-1. Setting of Slope Coordinate System

Either of the following two command formats may be used:

(1) [G69 [X_] [Y_] [Z_] { P_ / Q_ / R_ }]

X, Y, Z : Parallel shift amount for each coordinate axis.

P : Rotation angle in the X-Y plane (Z-axis pivot)

Q : Rotation angle in the Z-X plane (Y-axis pivot)

R : Rotation angle in the Y-Z plane (X-axis pivot)

P, Q, R: −9999.9999 to 9999.9999 (including cases of rotational axis with limit)

* The rotation direction in the P, Q, and R commands is CCW as viewed from the positive direction of an axis not included in the plane.

(2) [G69 H1]

Note 1: When settings of X, Y, and Z are omitted, "0" is set.

Note 2: Either P, Q, or R must be designated.

Note 3: This command executes the coordinate conversion by using the coordinate conversion parameter designated in the manual operation mode (refer to 3. "Coordinate Conversions in the Manual Operation Mode" which follows).

As stated earlier, up to 3 coordinate conversion operations are possible with this function.

At the (1) command shown above, the number of conversions is counted at each G69 code. If a 4th conversion is attempted, this will be processed as a new 3rd conversion, and the old 3rd conversion will be canceled.

When the (2) command shown above occurs, all the current settings will be deleted (even if a coordinate conversion is currently in progress), with the setting operation then beginning from the 1st conversion. The number of conversions will be the same as the number set by the parameter.

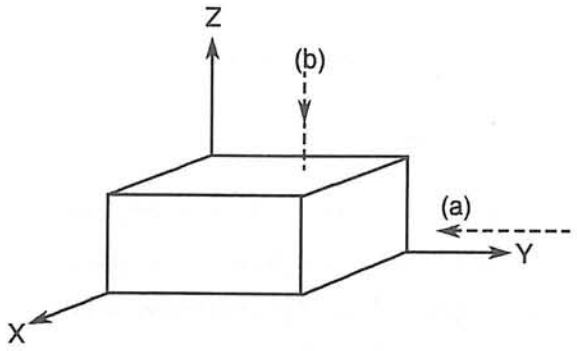
If the (1) command is executed following the (2) command, the current "number of conversions" count will be updated accordingly, or, if it is a 4th conversion, the 3rd conversion setting will be changed.

The G68 command is used to cancel 1 of the coordinate conversion settings. The "number of conversions" count will be reduced accordingly. This command can be used in both the (1) and (2) formats.

Note 4: All G69 commands will be canceled if an NC reset or the M02 command is executed.

Note 5: *With the graphic display specification, there are cases in which the tool path immediately following a coordinate conversion differs from the path immediately preceding the conversion.*

Example:




```
G69 H1
    :
G00 X_Y_Z_ ..... (a)
G68 (Coordinate conversion cancel)
G01 Z_ ..... (b)
```

After the execution of G68, the tool path is displayed from (b).

2-2. G Codes Usable During Coordinate Conversion

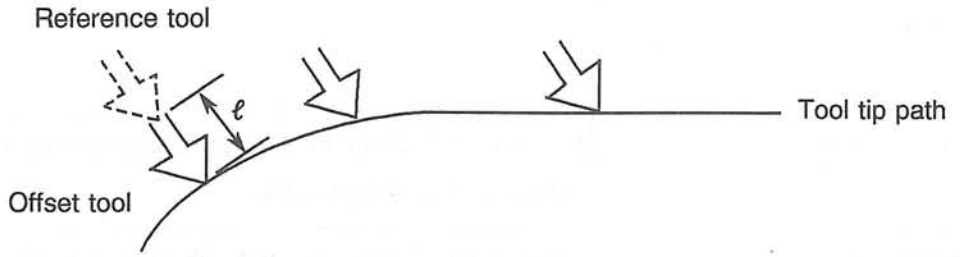
G Code	Operation During Coordinate Conversion
G00 Positioning G01 Cutting feed G31 Skip	Linear interpolation in the slope coordinate system
G02 Circular interpolation G03	Circular interpolation in the slope coordinate system Helical cutting is impossible.
G10 Local conversion G11	Local conversion is executed with respect to the slope coordinate system.
G15 Work coordinate system selection G16	The work coordinate system which is to be the object of 3-D coordinate conversion is selected.
G17 to G19 Plane designation	A plane in the slope coordinate system is designated. Planes having an additional axis cannot be designated.
G22, G23 Programmable stroke limit	Stroke limit is designated in the work coordinate system (coordinate system before 3-D coordinate conversion occurs).
G40 to G42 Cutter radius compensation	Cutter radius compensation is executed in the slope coordinate system.
G43, G44 3-D offset	3-D offset is executed in the slope coordinate system.
G50, G51 Geometry enlarge/reduce	Geometry is enlarged or reduced in the slope coordinate system.
G53 to G59 Tool length offset	Refer to following page.
G60 One-direction positioning	One-direction positioning is executed in the machine coordinate system. (Verify movement before executing this command.)
G62 Programmable mirror image	Mirror image is executed according to parameter bit setting. Identical operation can also be executed using switches. (Refer to following page.)
Fixed cycle	Refer to following page.

G14 (axis name designation) and G92 (work coordinate system setting) cannot be designated while coordinate conversion is in progress.

G codes other than those shown above can be executed regardless of the coordinate system. (G04, G09, G20, G21, G30, G61, G64, G90, G91, G94, G95)

- Tool length offset (G53 to G59)

  When tool offset the X- Y- or Z-axis direction is designated while a coordinate conversion is in progress, the coordinate system in which tool length offset is executed can be selected by the NC optional parameter bit No. 48, bit 1 (refer to 5. "Parameter Setting").

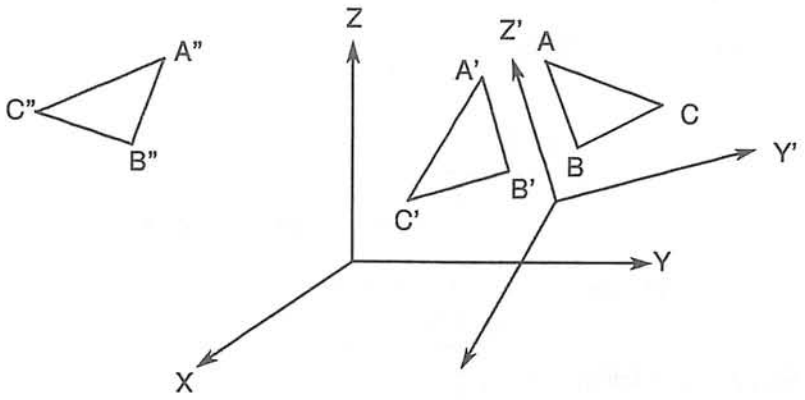


  If tool length offset along an additional axis (G57 to G59) is designated while a coordinate conversion is in progress, tool length offset will be executed in the coordinate system which includes the additional axis, irrespective of the slope coordinate system.

- Programmable mirror image (G62)

  The desired mirror image coordinate system can be selected according to the bit status when the power is switched ON. The bit settings and the corresponding coordinate systems are shown in the Table below. (This applies to the mirror image function using switches.)

Optional Parameter Bit Status at Power ON		Corresponding Coordinate System
No.34, bit 2	No. 33, bit 2	
1	1	Work coordinate system
1	0	
0	1	Slope coordinate system
0	0	Local coordinate system



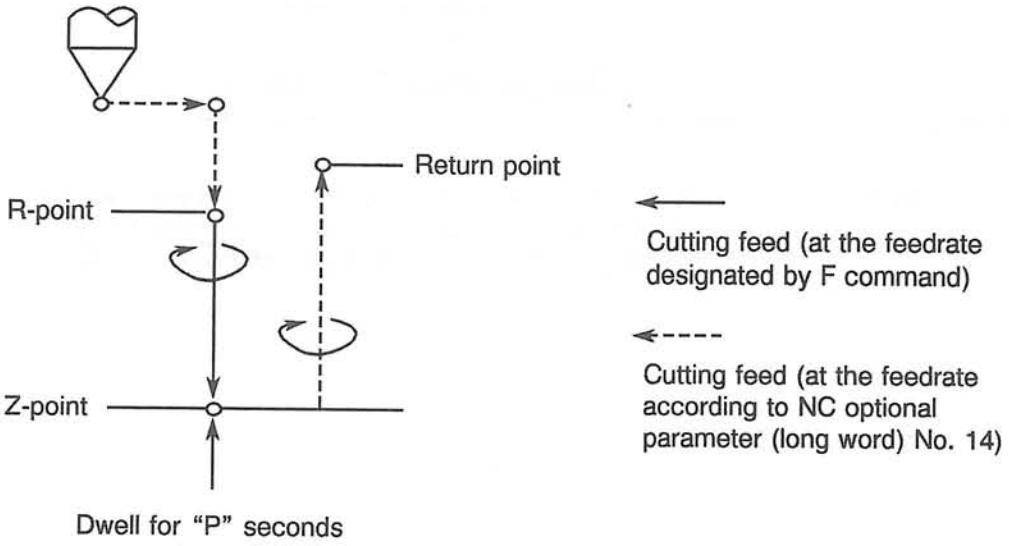
X-Y-Z : Work coordinate system
X'-Y'-Z': Slope coordinate system

A B C : Original command
A' B' C' : Slope coordinate system mirror image (Y-axis)
A" B" C" : Workpiece coordinate system mirror image (Y-axis)

- Fixed cycles (G71, G73, G74, G76, G80 to G87, G89, G274 and G284)

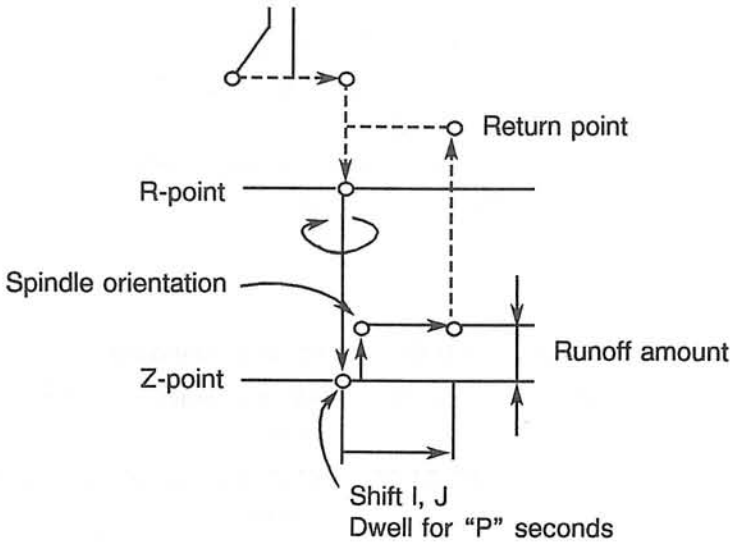
  During coordinate conversions, rapid axis feed in the usual fixed cycle is converted to cutting feed. This cutting feedrate is designated by the NC optional parameter (long word) No. 14 (refer to 5, "Parameter Setting").

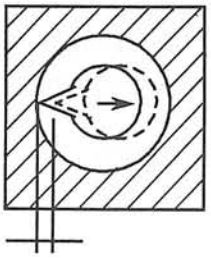
  Example: Drilling cycle (G81, G82)



In fine-boring (G76) and back-boring (G87) cycles, a shift is executed to retract the tool tip. During coordinate conversions, this shift is executed in either the machine coordinate system or the slope coordinate system, according to the setting at NC optional parameter (bit) No. 11, bit 4 (refer to 5. "Parameter Setting").

Example: Fine boring cycle (G76)





Designated by shift amount I, J

Note 1: *When an attachment is used, the tool indexing position will vary according to the direction in which the tool is facing. Therefore, always verify the tool tip direction before designating the shift amount using I and J commands.*

2-3. Mnemonic Codes Usable During Coordinate Conversion

Mnemonic Code	Description
OMIT RSTRT LAA ARC GRDX GRDY DGRDX DGRDY SQRX SQRY BHC	Can be used as a coordinate calculation function in the slope coordinate system
FMILR FMILF PMIL RMILR RMILO RMILI	Can be used as an area machining function in the slope coordinate system
CHFC CHFR	Can execute arbitrary angle chamfering in the slope coordinate system
COPY COPYE	Can be used in the slope coordinate system

Mnemonic codes other than those listed above can be used regardless of the coordinate system. (NOEX, CALL, RTS, MODIN, MODOUT, GOTO, IF, MSG, NMSG, and NCYL)

Note: Additional patterns of I-MAP may not be used.

2-4. Upper Limit Return (M52)

During coordinate conversions, the "upper limit return (M52)" command used in the fixed cycle, milling cycle, and other cycles is converted to the "R-point return (M54)" command. Moreover, the "upper limit return" command which is generated at the final cycle of the coordinate calculation function is also ineffective.

The "designated point return (M53)" command can be used.

2-5. Axis Command Cancel

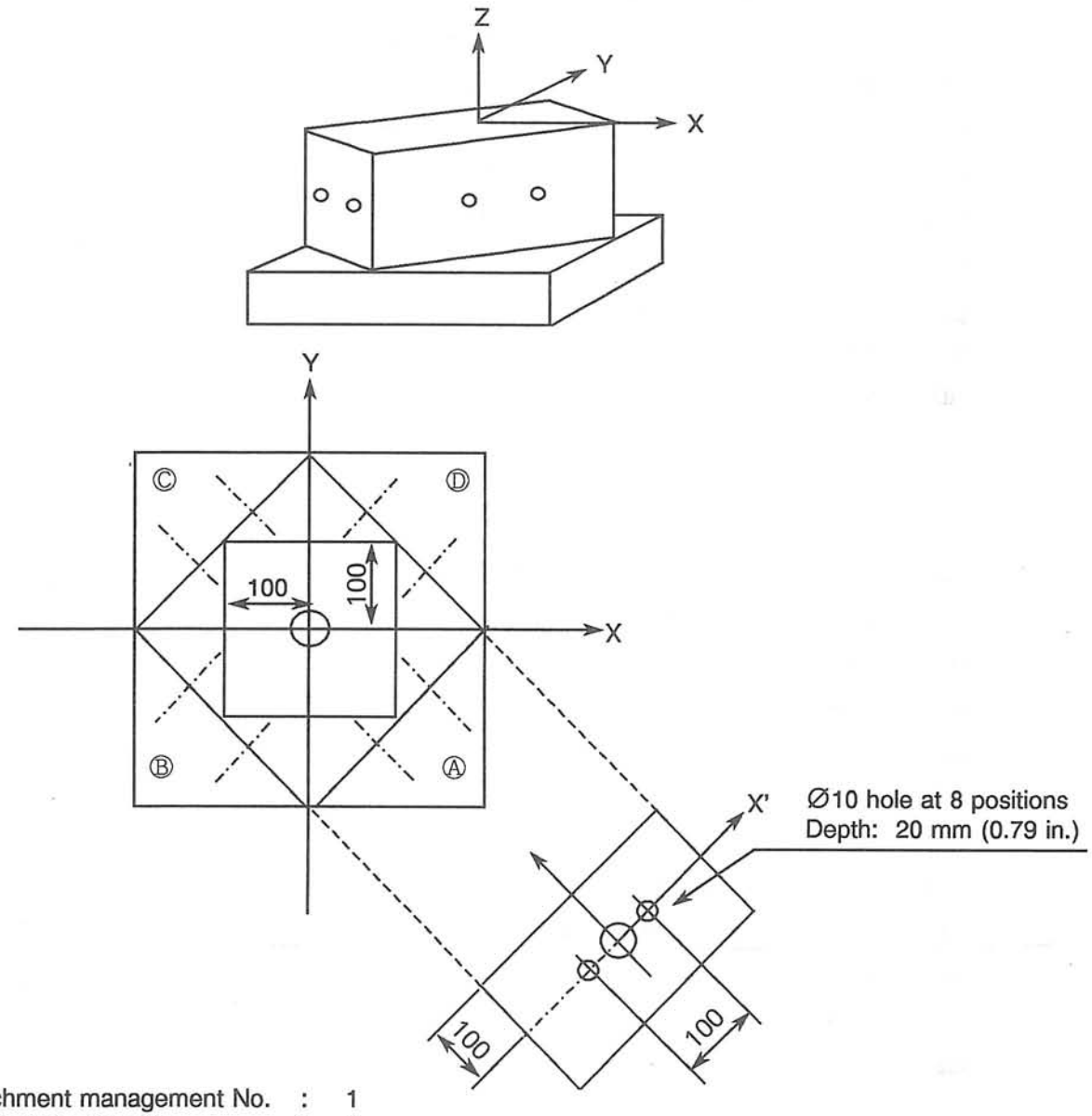
The "axis command cancel" command is not effective during coordinate conversions.

2-6. Pulse Handle Overlap

Regarding pulse handle overlap during coordinate conversions, select the coordinate system according to the ON/OFF status of the slope selection switch.

Slope Selection Switch	Coordinate System
ON	Slope coordinate system
OFF	Machine coordinate system

- Slope Machining Example:



Attachment management No.	:	1
Tool length offset No.	:	6
X-Y-Z	:	Workpiece coordinate system
X'-Y'-Z'	:	Slope coordinate system

```
      :
Ⓐ  G333   PAT=1   PAC=45   H6            OSUB
   G69    X100    Y-100    Z-100   P45    S700
   G69    R90                             G0  X0  Y300    Z100
   CALL   OSUB                            G81 G17 X-50    Y0  R5  Z-20    F84
   G68                                        X50
   G68                                    G80
   G334                                   RTS
Ⓑ  G333   PAT=1   PAC=135  H6
   G69    X-100   Y-100    Z-100   P-45
   G69    R90
   CALL   OSUB
   G68
   G68
   G334
Ⓒ  G333   PAT=1   PAC=225  H6
   G69    X-100   Y100     Z-100   P-135
   G69    R90
   CALL   OSUB
   G68
   G68
   G334
Ⓓ  G333   PAT=1   PAC=315  H6
   G69    X100    Y100     Z-100   P135
   G69    R90
   CALL   OSUB
   G68
   G68
   G334
      :
```

3 Coordinate Conversions in the Manual Operation Mode

By designating coordinate conversion parameters in the manual operation mode and by turning the "slope selection switch" ON, manual operation can be executed in the slope coordinate system.

Moreover, in the automatic and MDI operation modes, the designated coordinate conversion parameters can be transferred by executing the G69 H1 command.

3-1. Setting Coordinate Conversion Parameters

Call the 3-D coordinate setting screen using the following procedure:

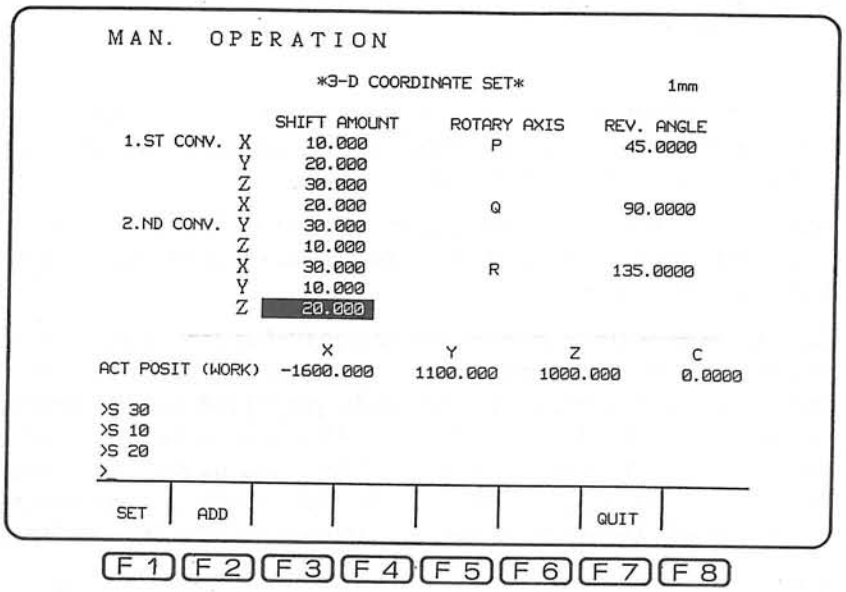
① Press the MANUAL key to select the manual operation mode.

② Press the function key [F8] (EXTEND) twice. (The function key [F1] will be assigned to the function key [F1].)

③ Press the function key [F1] (3-D CD. CONVERT).

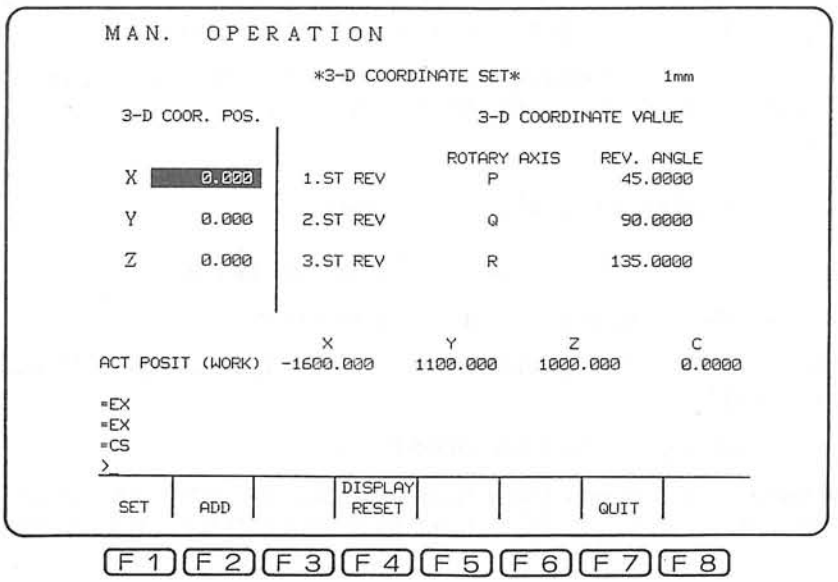
The 3-D coordinate setting screen consists of 2 pages, with either the 1st or 2nd page being displayed, depending on the setting at NC optional parameter (bit) No. 11, bit 5. (Refer to 5. "Parameter Setting").

1st Display Page



The parallel shift amount, rotational axis name (P, Q, R), and the rotation angle are designated in the same manner as when the G69 command is used.

2nd Display Page



The parallel shift amount is designated as a coordinate point position* in the slope coordinate system, and the rotational axis name and rotation angle are designated in the same manner as on the 1st display page.

* "Coordinate point position in the slope coordinate system" refers to the spindle position in coordinate system which follows the 3-D coordinate conversion operation. In other words, it is the current position in the slope coordinate system.

Note 1: *Although the coordinate conversion parameter displays on the 1st and 2nd display pages are different, the content is the same. Differing parameter settings on the two pages are not permitted.*

Note 2: *Coordinate conversion parameters designated in the manual operation mode can be transferred to the automatic and MDI operation modes by designating the G69 H1 command. The parameters on the 1st display page (shift amount, rotational axis name, rotation angle) are transferred as they are. However, as for the parameters on the 2nd display page, the "total shift amount" is transferred as the "1st conversion" shift amount, and the shift amounts for the 2nd and 3rd conversions are transferred as "0". This should be noted when using the G68 command on the 2nd display page.*

Note 3: *If an asterisk (*) is entered as the rotational axis name for a given coordinate conversion, all subsequent coordinate conversion settings will be ignored. For example, if the 1st conversion axis name is designated as " * ", the 2nd and 3rd conversion settings will be ignored, with only the 1st conversion being effective.*

Note 4: *An alarm will be activated if the G69 H1 command is attempted when the "coordinate conversion parameter inoperative" status (1st conversion axis name is " * ") is established.*

3-2. Coordinate Conversion Parameter Settings When Switching To The Manual Operation Mode

There are occasions during automatic or MDI operation when it is necessary to switch to the manual operation mode to make a correction, etc. When switching to the mode at such a time, the data which exists immediately before switching occurs is automatically transferred and is displayed as coordinate conversion parameter data. The setting at NC optional parameter (bit) No. 11, bit 6 determines whether or not this automatic data transfer occurs when switching to the manual operation mode.

```
G69   X10   Y10   Z10   P45 .....  ⓐ
G69   X20   Y20   Z20   Q45 .....  ⓑ
G69   X30   Y30   Z30   R45 .....  ⓒ
  ⋮
  ①
  ⋮
G68
  ⋮
  ②
  ⋮
G68
  ⋮
  ③
  ⋮
G68
  ⋮
  ④
  ⋮
M02
```

If a switch to the manual operation mode occurs at steps ① to ④ during execution of the above program, data will be transferred as coordinate conversion parameter data as shown in the table below. (When NC optional parameter (bit) No. 11, bit 6 is "1".)

Conversion data not canceled by the G68 command will be transferred even if the M02 or RESET command is executed.

Mode Switching Points	Data to be Transferred		
	1st conversion	2nd conversion	3rd conversion
①	ⓐ	ⓑ	ⓒ
②	ⓐ	ⓑ	–
③	ⓐ	–	–
④	–	–	–

When a data transfer occurs, all the current coordinate conversion parameter settings will be deleted. Therefore, the "no setting" (indicated by "–" marks in the above table) status exists at steps where no data transfer occurs.

3-3. Executing Coordinate Conversion

In the manual operation mode, turn the "slope selection switch" ON in order to make the designated coordinate conversion parameter settings effective.

The coordinate conversion operation will then be executed.

During coordinate conversion operation, rapid feed commands, cutting feed command, and pulse handle feed commands for X-, Y-, and Z-axes are processed in the slope coordinate system.

The feedrates and feed amounts are as shown below:

- Rapid feedrate . . . . . The slowest machine axis speed is adopted for all machine axes.
- Cutting feedrate . . . . . The designated speed is adopted.
- Pulse handle feed amount . . . . The designated distance is adopted.

Note 1: Never switch the axis or change the coordinate conversion parameter setting while axis travel is in progress.

Note 2: All machine axes will be stopped if any axis reaches the limit position during a coordinate conversion operation.

4. Actual Position Data Display

Actual position data in the slope coordinate system can be displayed. Switching between the slope coordinate system and the work coordinate system can be executed at the NC optional parameter (bit) No.6, bit 6 setting.

However, to display actual position data in the slope coordinate system in the manual operation mode, the "slope selection switch" must be turned ON together with the bit switching operation.

Note 1: *This function applies only to the ACTUAL POSITION display screen called by pressing the function key [F2]. On the program display screen, actual position data is displayed only in the work coordinate system.*

Note 2: *There are cases when a discrepancy may exist (never exceeding the "minimum setting unit" amount) between the designated coordinate value and the displayed value, due to limitations in the OSP's calculating accuracy.*

Note 3: *If a local coordinate conversion (G11) is designated during 3-D coordinate conversion operation, the local coordinate's current position will be displayed at the local-converted position relative to the work coordinate system. In other words, actual position data in the local coordinate system represents the [actually designated value] - [3-D coordinate conversion].*

Example:

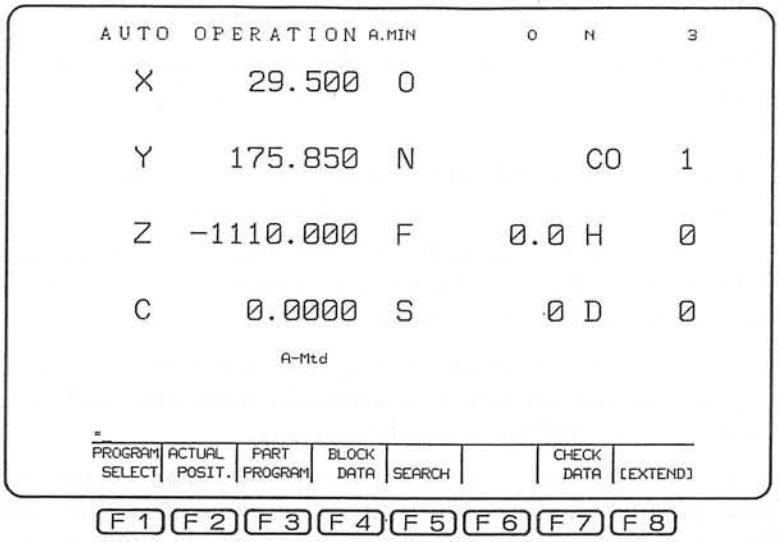
```
G69 X10     Y10       Z10       P0
G11 X-20    Y-20      Z-20
G00 X49.5   Y195.85    Z-1090 .....
```

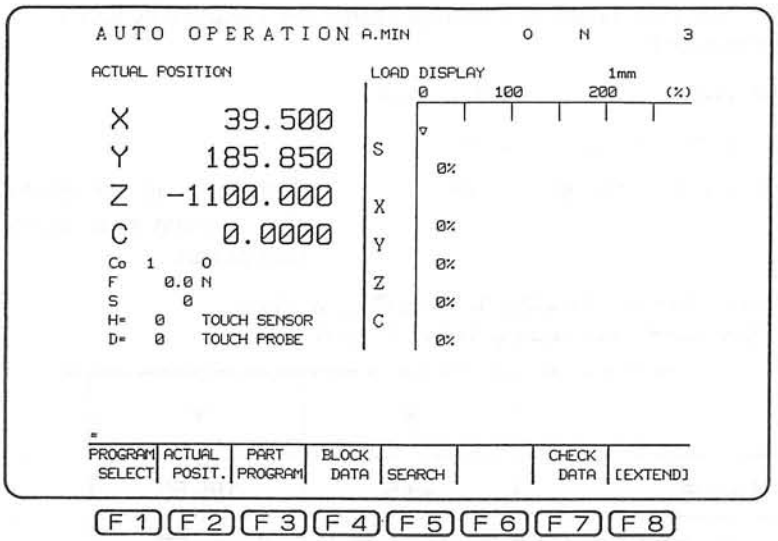
Command at coordinates where local conversion is being executed following 3-D coordinate conversion.

Current Position Display At Each Display Page
(NC optional parameter (bit) No. 6, bit 6 is 1)

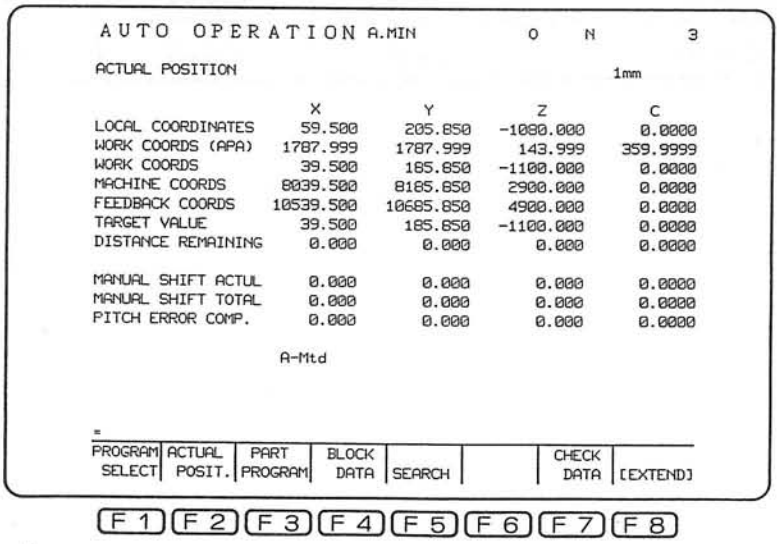
	X	Y	Z
Command value	49.5	195.85	−1090
x 4 (x 2) enlarged display (slope coordinate system)	29.5	175.85	−1110
Local coordinates	59.5	205.85	−1080
Work coordinates	39.5	185.85	−1100

x 4 (x 2) Enlarged Display (Slope Coordinate System)





Local Coordinate Current Position Display



5. Parameter Setting

- NC optional parameter (long word)

  No.14 Cutting feedrate to which the rapid feedrate in the fixed cycle is converted during coordinate conversion operations.

  Setting units : mm/min.

  Setting range : 1 - 20,000

  Initial value : 20,000

- NC optional parameter (bit) (factory setting: 0)

  No. 6, bit 6 . . . . . Actual position data display in the slope coordinate system

  =0 : Actual position data in the work coordinate system

  =1 : Actual position data in the slope coordinate system

  No. 11, bit 4 . . . . . Tool tip retract motion during fixed cycle operation

  =0 : Axis retract command in the machine coordinate system

  =1 : Axis retract command in the slope coordinate system

  No. 11, bit 5 . . . . . Coordinate conversion parameter setting screen in the manual operation mode

  =0 : 1st page

  =1 : 2nd page

  No. 11, bit 6 . . . . . Coordinate conversion parameter transfer when switched to the manual operation mode

  =0 : Not transferred

  =1 : Transferred

  No. 33, bit 2 . . . . . Mirror image in the slope coordinate system

  =0 : According to the setting at NC optional parameter (bit) No.34, bit 2

  =1 : Slope coordinate system

  No. 48, bit 1 . . . . . Tool length offset in slope coordinate system

  =0 : Workpiece coordinate system

  =1 : Slope coordinate system

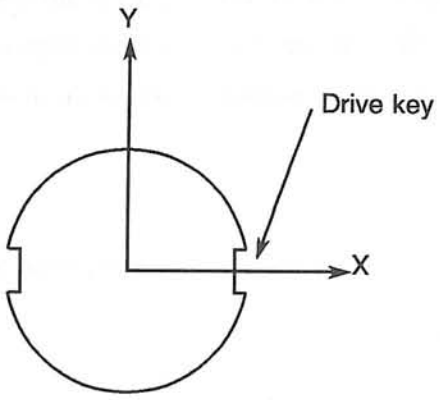
SECTION 18 MULTIPLE-POINT SPINDLE ORIENTATION FUNCTION

1. Function Outline

When executing the tool breakage detection and automatic tool length compensation, the tool tip must be brought into contact with the touch sensor. However, with some cutting tools, it becomes necessary to orient the spindle in a position other than the one supported as the standard spindle orientation position.

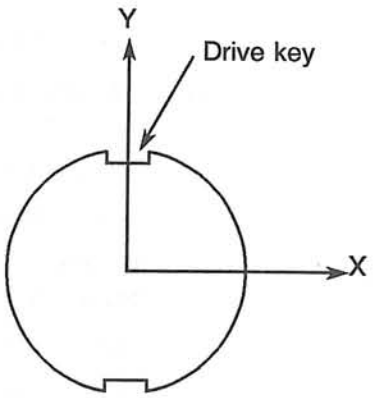
The function covered by this specification orients the spindle at such positions so that the tool breakage detection and automatic tool length offset function can be used for special tools.

The illustration below shows the spindle orientation positions.



Normal spindle orientation position

Fig, 18-1



Spindle orientation position for tool breakage detection for boring bar

Fig, 18-2

2. Multiple-point Spindle Orientation Function

(1) Spindle Orientation Direction

The direction in which the spindle is rotated to orient the spindle is determined by the M code commands.

M19 . . . . Spindle orientation is executed in the spindle forward rotation direction.

M118 . . . Spindle orientation is executed in the spindle reverse rotation direction.

M119 . . . Spindle orientation direction is not definite; to shorten the spindle orientation time, spindle orientation is executed in the direction the spindle orientation position lies further from the present spindle position.

(2) Multiple-point Spindle Orientation

The multiple-point spindle orientation function is called by M118 and M119 in addition to M19. The explanation is given assuming M19.

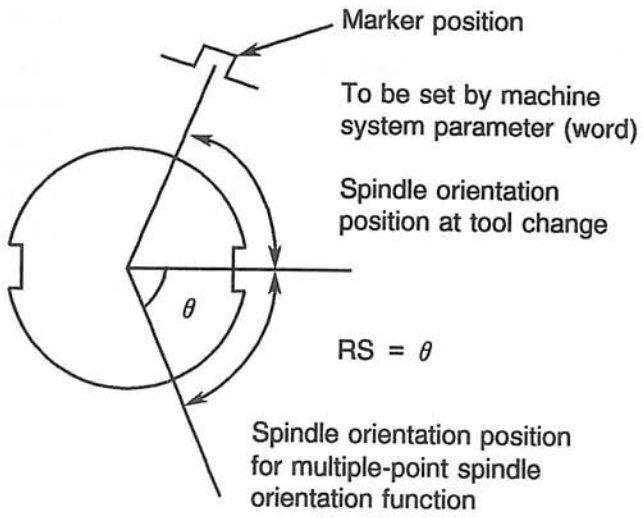
(a) The command format is:

M19 RS = θ

(b) θ represents an angle in units of degrees with decimal fractions rounded off.

(c) Programmable range is from 0° to 360°

The required orientation angle θ is defined in reference to the spindle position when the tool change is carried out and the angle is measured in the spindle forward rotation direction.



Note 1: *When RS command is omitted after M19 (or M118 or M119), the spindle position selected for the tool change cycle execution is used as the spindle orientation position.*

In other words, omission of an RS command is equivalent to M19 RS = 0.

Note 2: *Note that the RS command must always be programmed in the same block with the spindle orientation M code command (M19, M118, M119).*

If an RS command is designated independently, it does not generate an alarm and the answer signal is turned on immediately. This command is cleared by the M19 command designated next.

RS commands are handled as a modal command.

3. Tool Breakage Detection and Automatic Tool Length Offset Functions

The spindle can be oriented in the direction so that the special tool tip (boring bar, etc.) may be brought into contact with the touch sensor to execute the tool breakage detection and automatic tool length offset functions.

(1) Command Format

To execute the tool breakage detection or automatic tool length offset program using the multiple-point spindle orientation function, following format is used:

CALL OO30 PRS = θ

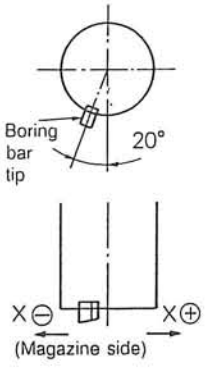
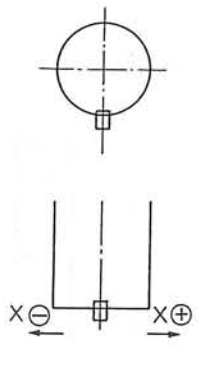
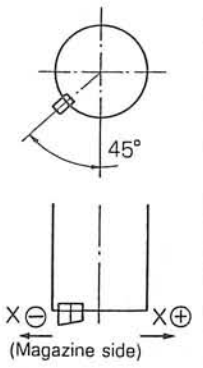
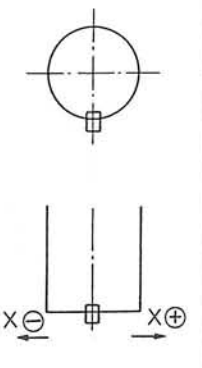
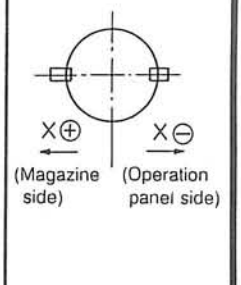
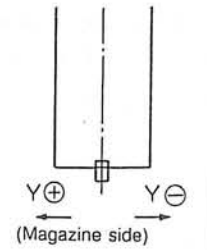
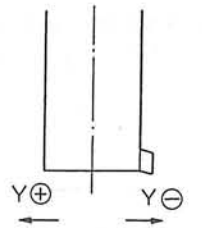
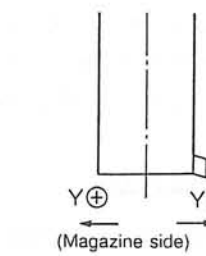
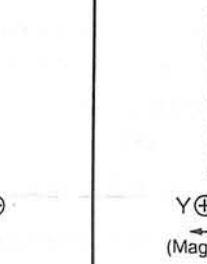
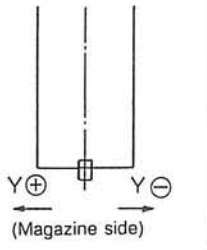
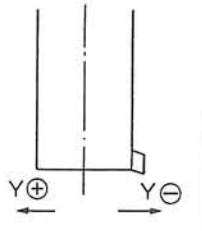
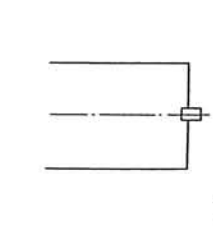
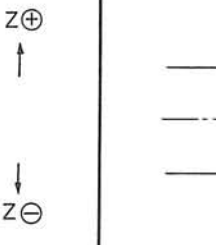
Designate the required spindle orientation angle in reference to the spindle position selected for tool change cycle and measured in the spindle forward rotation direction.

Note: *Difference between CALL OO30 and CALL OO30 PRS = θ*

CALL OO30 . . . . . The spindle is orient stopped at the position where the spindle has been oriented for the execution of the tool change cycle. If the spindle orientation function is selected, the spindle orientation pin is inserted at the spindle orientation position.

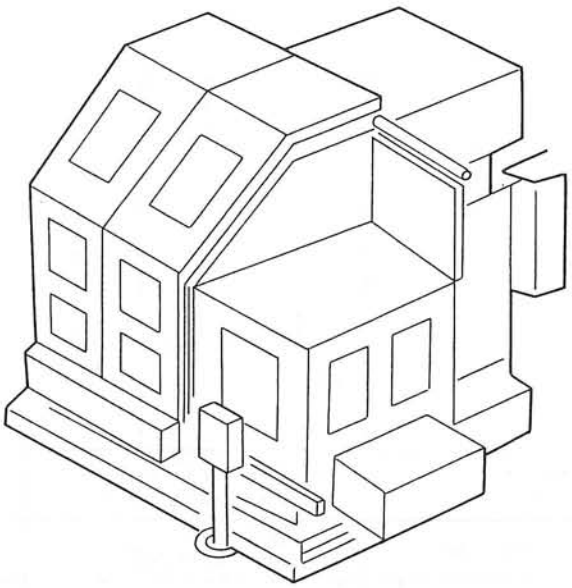
CALL OO30 PRS = θ . . . The spindle is orient stopped at the designated angle (θ). The spindle orientation pin is not inserted after the orientation even when the spindle orientation pin is available. Note that the pin is inserted when the orientation angle is 0 deg. ($\theta = 0$).

(2) An example of spindle orientation positions for special tools to execute the tool breakage detection and automatic tool length offset functions is indicated below for individual machining center models.

<table>
<tr><th colspan="2">MC-4VA</th><th colspan="2">MC-5VA</th><th>MC-400/500
/6000/800H</th></tr>
<tr><th>Before change</th><th>After change</th><th>Before change</th><th>After change</th><th>Before change
(no change)</th></tr>
<tr>
<td> 
</td>
<td> 
</td>
<td> 
</td>
<td> 
</td>
<td> 
</td>
</tr>
<tr><th colspan="2">MCV-A 24-tool
MCV-B 24-tool</th><th colspan="2">MCV-A 50/70-tool
MCV-B 50/70/100-tool</th><th>MCR-B</th></tr>
<tr><th>Before change</th><th>After change</th><th>Before change
(no change)</th><th>Before change
(no change)</th></tr>
<tr>
<td> 
</td>
<td> 
</td>
<td> 
</td>
<td> 
</td>
</tr>
<tr><th colspan="2">MCM, MCR</th><th colspan="2">MCM (Horizontal spindle)</th></tr>
<tr><th></th><th></th><th>Before change</th><th>After change</th></tr>
<tr>
<td> 
</td>
<td> 
</td>
<td> 
</td>
<td> 
</td>
</tr>
</table>

SECTION 19 INDEX TABLE EXTERNAL MANUAL OPERATION FUNCTION

1. Overview



The external manual operation panel is mounted at the front of the machine to turn the index table manually at the setup position to facilitate setup for the machine equipped with shield cover, etc.

(1) Turn the **B-AXIS MANUAL** switch in the ON position and the control is placed in the manual B-axis operation mode.

(2) Note that the control is reset in this case.

(3) In the manual B-axis operation mode, the control is in the following alarm state.

ALARM C 949 External manual operation

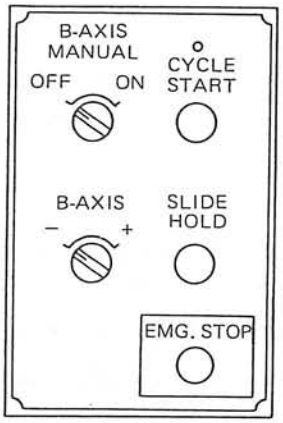
(4) Turn the B-AXIS +/– switch in the required position and press it.

(5) The B-axis rotates in the selected direction while the switch is held down.

(6) Positioning of B-axis is completed after command pulse is turned off.

(7) After the completion of setup, return the switch to the OFF position.

2. Operation



(1) B-axis Manual Switch

The manual B-axis operation mode is turned on and off according to the setting of the **B-AXIS MANUAL** switch.

Note that changing this switch setting resets the control.

In the manual B-axis operation mode, the control is in the following alarm state.

ALARM C 949 External manual operation

In this alarm state, index table operation using the operation panel has the top priority to any other axis motion commands. In the manual mode, for example, normal manual rapid feed, manual cutting feed, or pulse handle feed is not possible. This is to ensure safety of operation.

Place the switch to the OFF position only after confirming that the index table has been clamped.

(2) B-axis +/– Switch

This switch determines the direction of index table rotation and rotates the index table.

(a) Rotating direction

Determine the index table rotating direction by placing the switch in the "+" or "–" position.

(b) Index table rotation

① When the switch is pressed, the index table rotation signal is turned on.

② The index table keeps rotating in the selected direction while the switch is held down.

③ When the switch is released, the rotation signal is turned off; the table is positioned and clamped at the first 1 deg. or 5 deg. interval position.

④ The indexing angle interval conforms to the machine specification.

(3) Cycle Start Button

This button is the same as the **CYCLE START** button on the machine operation panel.

(4) Slide Hold Button

This button is the same as the **SLIDE HOLD** button on the machine operation panel.

3. Caution

(1) When the B-axis manual switch is turned from ON to OFF or from OFF to ON, the control is reset.

Therefore, place this switch in the OFF position after confirming that the index table has been clamped. If it is placed to the OFF position while the index table is rotating, the index table stops rotating in the unclamped state.

To release this table unclamped state, refer to (5).

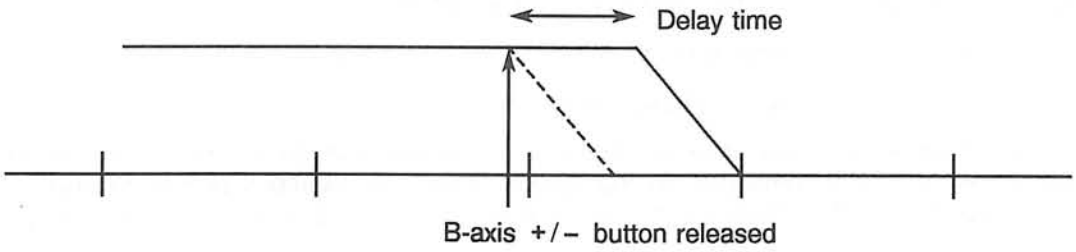
(2) When the manual B-axis mode is selected, axis feed (manual rapid feed, cutting feed, pulse handle feed, etc.) is not possible even in the manual mode.

Set the B-axis manual mode setting switch OFF to carry out normal manual operation.

This interlock is provided to ensure the safety during operation; when the manual B-axis mode is selected, an operator will be carrying out setup in the machine.

(3) Index table positioning is carried out in intervals of 1 deg. or 5 deg. according to the 1 deg. index or 5 deg. index specification.

Therefore, there may be slight delay until the B-axis starts decelerating after the B-axis motion button is released.



(4) When the manual B-axis operation mode is selected, the axes being unclamped for pulse handle operation, etc. are clamped.

(5) If the control is reset while the index table is rotating, the index table stops and remains unclamped.

In this state, although manual B-axis operation is possible, B-axis is not clamped.

(6) To clamp the B-axis, turn the **B-AXIS MANUAL SWITCH** off, select the MDI or automatic mode and designate an index table operation command. The B-axis is clamped after the completion of positioning.

SECTION 20 AUTOMATIC SCHEDULE PROGRAM UPDATE FUNCTION

1. Overview

(1) The OSP5020M/OSP500M-G can use a schedule program so that different types of workpieces can be machined continuously using a pallet changer, etc. without operator's intervention.

(2) Recently, flexibility of a schedule program, i.e., addition and/or change of a schedule program, is required for schedule operation using a schedule program to meet diversified needs in production.

(3) The automatic schedule program update function allows an operator to add or change the schedule program safely without stopping schedule operation by designating the automatic update and registering the schedule program.

2. Restrictions to Schedule Programs

Because the automatic schedule program update function, updates the schedule program while the machine is operating according to the schedule, the following restrictions must be observed to ensure safe operation.

The blocks which can be used in the schedule program are:

- PSELECT block . . . Selects and executes the main program.
- VSET block . . . . . Sets variables.
- END block . . . . . Ends the schedule program.

Other blocks such as GOTO and IF blocks cannot be used.

3. Operation

3-1. Designating Automatic Schedule Program Update

Designation of automatic schedule program update is made using the option designation when selecting the schedule program.

Procedure:

(1) Select the automatic mode.

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

The functions assigned to the function keys are changed; the SEARCH NUMBER function is assigned to the function key [F1].

(3) Press the function key [F4] (SP-SELECT).

The "SS" prompt is displayed on the screen.

(4) Key in [*] and [;].

SS␣*;

(5) Press the **WRITE** key.

The screen transfers to program selection directory screen, and the schedule program files in the bubble memory are displayed.

(6) Move the cursor to the required file name.

(7) Press the **WRITE** key.

(8) The following message is displayed on the screen.

What is the option?

Key in "U".

The required schedule program is selected and at the same time the automatic update function is called.

Note 1: *Optional designation*

U: Automatic update function effective
N: Automatic update function ineffective

The option designation can be omitted. Default for option designation is set with the NC optional parameter (word) No. 13.

No. 13: 0 . . . . N (automatic update function ineffective)
1 . . . . U (automatic update function effective)

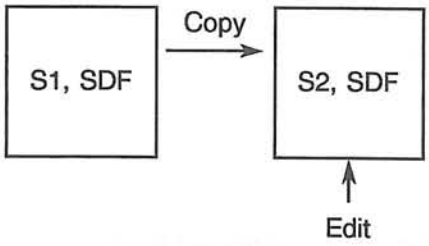
Note 2: *It is possible to designate the option using the keyboard directly.*

SS␣ (schedule program file name);U ***WRITE***

3-2. Editing Schedule Program

A schedule program can be edited in the same manner as editing machining programs, etc. However, the schedule program which is currently run or the schedule program registered for automatic editing cannot be edited directly. Copy the schedule program to other file to edit it.

Schedule program being run or registered



Note: *The automatic update function compares the schedule program being run when it is registered and with the schedule program to be registered. If the contents in the blocks preceding the currently executed block do not agree with each other, registration is not possible. Therefore, addition and change for blocks to be executed later are allowed in program editing operation.*

3-3. Registering Schedule Program to be Updated

Procedure:

(1) Select the automatic mode by pressing **AUTO** key.

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

The functions assigned to the function keys are changed; the NEW, SP ENTRY function is assigned to the function key [F6].

(3) Press the function key [F6] (NEW SP ENTRY).

The "SE" prompt is displayed on the screen.

(4) Key in [*].

SE *

(5) Press the **WRITE** key.

The screen transfers to program selection directory screen, and the schedule program files in the bubble memory are displayed.

(6) Move the cursor to the required file name.

(7) Press the **WRITE** key.

After the completion of registration, the following message is displayed.

****. SDF entry finished*

This registers the schedule program used to update the current schedule program. After the completion of execution of the current block, and before the execution of the next block, the registered update schedule program is selected automatically. Machining is continued with the new schedule program.

Note 1: *The schedule program file name can be directly keyed in to be registered.*

SE␣ (schedule program file name) ***WRITE***

Note 2: *To cancel the registered schedule program file, append option ;C to the registration command in the same manner as registering a schedule program.*

Example: *SE␣ *; C*
SE␣ S1. SDF; C

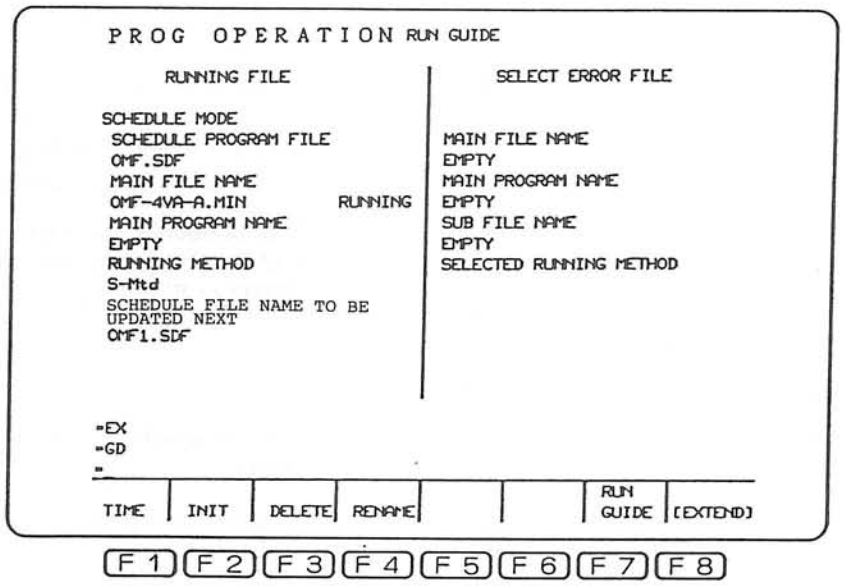
When the registered schedule program file is canceled, the following message is displayed.

****. SDF entry cancel finished*

Note 3: *The registered schedule program file is canceled when program selection or schedule program selection is carried out newly.*

4. CRT Display

The registered schedule program file names, etc. are displayed on the RUN GUIDE screen in the program operation mode.



5. Error Messages

2707 Schedule program update mode

Character-string : None

Code :

1 ..... Registration of automatic update is made for the schedule program which has been selected without the "automatic update" designation.

2 ..... SP number search, using the function key or cursor key, was made for the schedule program which has been selected with the "automatic update" designation.

3 ..... The schedule program selected with the "automatic update" designation contains the GOTO or IF block.

2708 Schedule program update entry

Character-string : None

Code :

1 ..... The designated schedule program is not found.

2 ..... A schedule program has already been registered.

3 ..... The GOTO or IF block is used.

4 ..... The program (machining processes) is not identical in the range from the head of the program to the registration point.

2533 File change inhibit

Character-string : None

Code :

1 ..... An attempt is made to edit the schedule program currently selected while the automatic update function is executed.

2 ..... An attempt is made to edit the schedule program which is registered for automatic update.

None ..... With no code, not related to automatic Update. (See standard alarm/error list.)

SECTION 21 ADDITIONAL AXIS (ROTARY AXIS) FUNCTION

1. Normal Operation Specification

1-1. Outline

Rotary tables have infinite movement range, and, differing from linear axes (X, Y, Z), they return to their original position in a rotation of 360 degrees. From this special feature, rotary axes are programmed in the different manner from linear axes. This section provides programming rule for rotary axes so that readers can program rotary axis movement correctly and easily.

1-2. Axis Nomenclature

A-axis: Axis rotating around X-axis
B-axis: Axis rotating around Y-axis
C-axis: Axis rotating around Z-axis

1-3. Program Format

1-3-1. Least Setting Unit

The least setting unit for the NC rotary table is 0.001°.

1-3-2. Programmable Range

Absolute value : 0 - 360.000
Incremental value : ± 360.000

1-3-3. Rotation Direction Command

Rotation direction commands should be programmed in conformity with ISO-841, ISO-1056 and JISB6310.

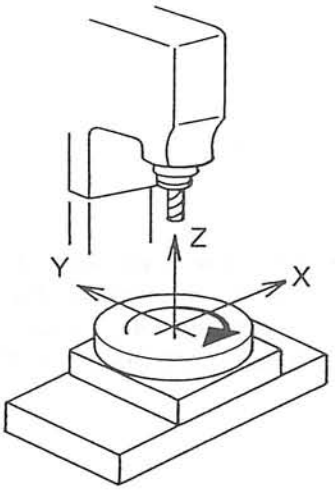
Command Method	Tape Command	Rotation Direction
Absolute command (4th axis)	M15	Forward direction
	M16	Reverse direction
Absolute command (5th axis)	M115	Forward direction
	M116	Reverse direction
Incremental command	+	Forward direction
	–	Reverse direction

Note: M codes used for determining the rotation direction of a rotary axis are all modal.
M15, M16, M115, and M116

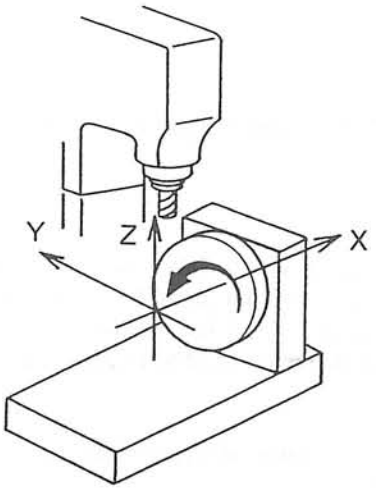
Installation Position and Rotation Direction of NC Rotary Table:

(1) Vertical Machining Center (MC-V)

C-axis



A-axis

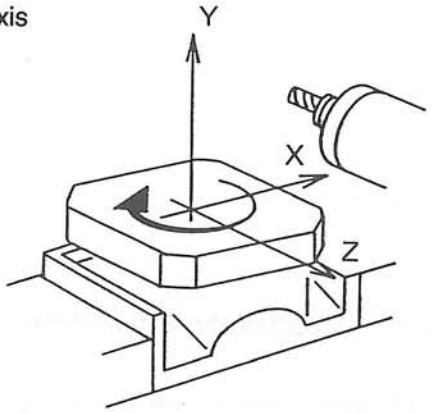


When controlling the NC rotary table as A-axis, it may not be installed to the left side of the spindle viewing the spindle from front due to the interference with the ATC.

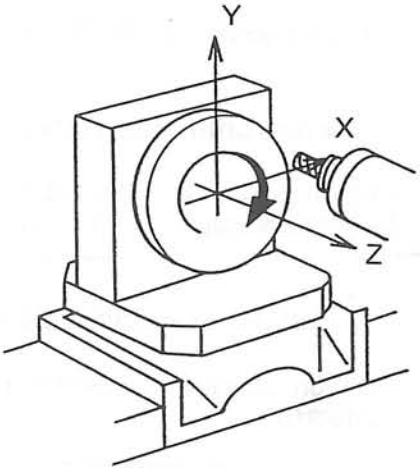
REMARKS

(2) Horizontal Machining Center (MC-H)

B-axis

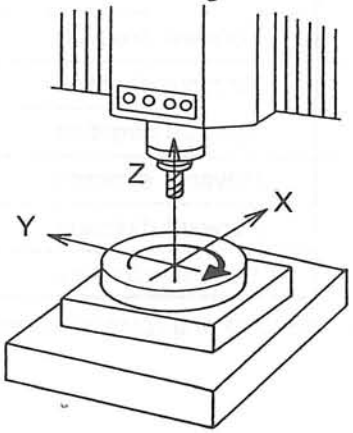


C-axis

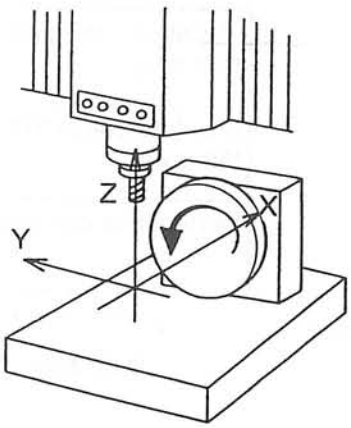


(3) Double-column Machining Center

C-axis



A-axis



Parameters for Rotary Axes

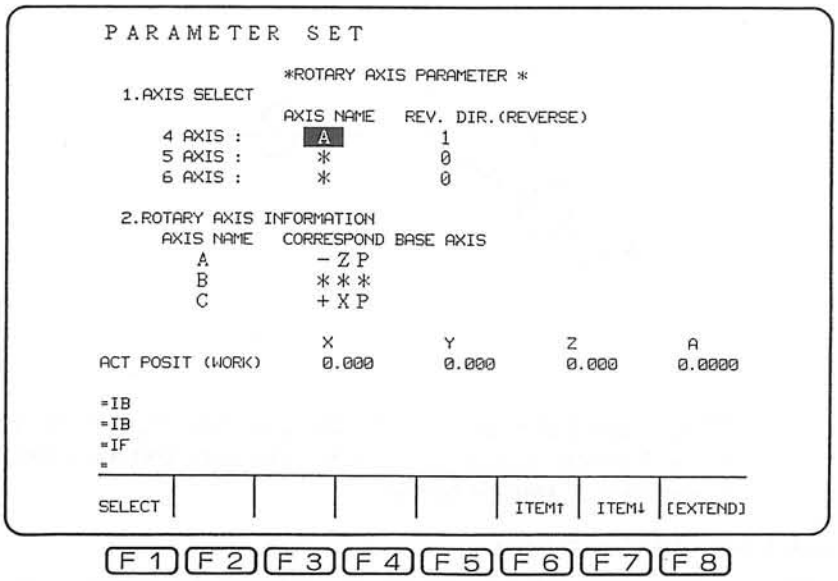
Rotary axes (the B-axis of the MC-H series machining centers not included) can be defined as A-, B-, or C-axis depending on the installation direction.

Axis name, rotation direction, and correspondence to the basic axis can be set using parameters.

Procedure:

① Press the **PARAMETER** key to select the parameter setting mode.

② Press the function key [F8] (EXTEND) repeatedly until the "ROTASRY AXIS PARAMETER" screen is displayed.



③ Move the cursor to the data column where data needs to be set using the cursor control keys and press the function key [F1] (SELECT).

Set data at the following parameters;

AXIS NAME	:	A, B, C, * (no setting)
REV. DIR.	:	0 (forward rotation), 1 (reverse rotation)
CORRESPOND BASE AXIS	:	±Xp, ±Yp, ±Zp, *** (no setting)

1) AXIS NAME

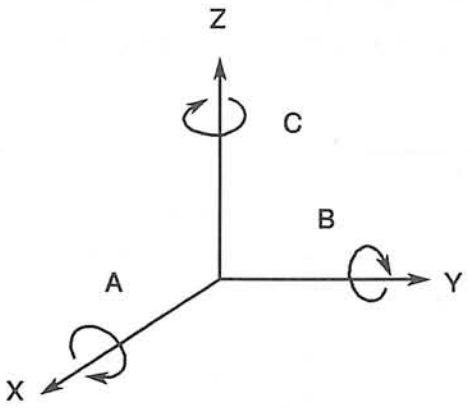
Select a name for the rotary axis to be used, Axis names for rotary axes can only be used.

Note 1: When more than one rotary axis is used, do not apply the same axis name to these axes.

Note 2: With the animated drawing specification, the rotary axis designated as drawing data is canceled when the axis name in parameter data has been changed. Reset the rotary axis in drawing data.

2) REV. DIR.

Arrow directions as indicated below are defined as forward rotation directions of rotary tables with respect to the basic axes by ISO. Refer to these directions when selecting a rotation direction.



Note: When the rotation direction Is changed, actual position data is also changed. In such cares, reset machine zero point data and work zero point data after the completion of step ④ below.

3) CORRESPOND BASE AXIS

Select a corresponding basic axis when machining the periphery of a cylinder. For details, refer to Section 22.

④ After necessary data has been set, press the function key [F7] (BACKUP), turn off the power, and then turn on the power again.

Note 1: Data set on the screen is not effective unless the power is once turned off and then turned back on again.

Note 2: When the power is turned off without pressing, the function key [F7] (BACK-UP), data set on the screen will be lost.

1-3-4. Program Examples

Example 1:

```
N1  G90   G0   C0          M15
N2              C0
```

In N2 block, rotary table does not rotate since the same point is specified for C-axis.

Example 2:

```
N3  G90   G0   C0          M15
N4  G91   G1   C360   F36
```

In N4 block, the table rotates in the forward direction since positive value is specified for C-axis command in incremental mode. It makes one full turn in 10 minutes.

Example 3:

```
N5  G90   G0   C0          M15
N6             C200        M16
```

Example 4:

```
N7  G90   G0   C0          M15
N8  G91        C-160
```

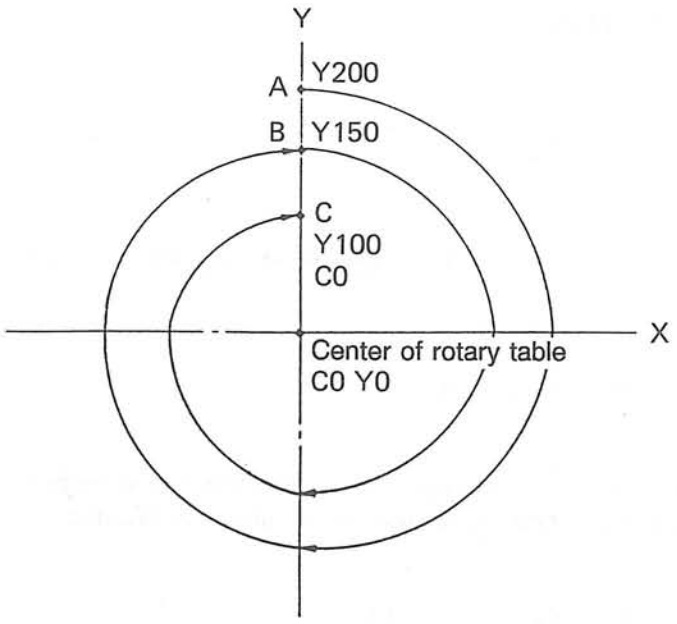
The rotary table carries out the same operations in the programs indicated in examples 3 and 4.

In example 3, the rotary table rotates in the reverse direction (M16) up to 200° position. That is, the rotary table rotates in the reverse direction by 160°.

In example 4, the commands are given in incremental values. The minus sign of the C command designates the reverse rotation direction, and thus the rotary table rotates in the reverse direction by 160°.

Example 5:

```
(A) N9    G0          C0    Y200       M16
(B) N10   G91   G1    C360  Y-50       F36
(C) N11               C360  Y-50
```

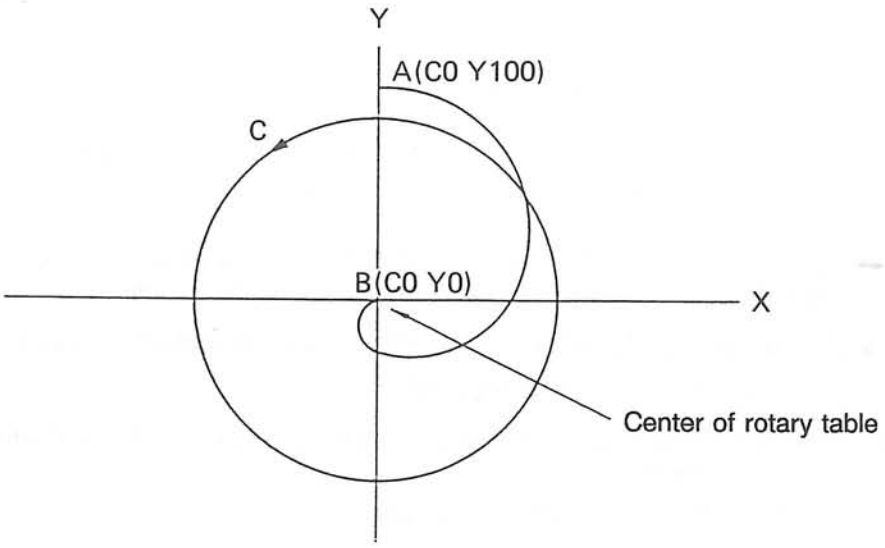


With the commands in blocks N10 and N11, Y-axis moves 100 mm (3.94 in.) while the rotary table makes two full turns (720°) and the resultant tool paths are as indicated above.

1-3-5. Feedrate in Simultaneous Two-axis Control

Example:

```
(A) N1    G90  G0    C0    Y100       M16
(B) N2    G91  G1    C360  Y-100      F36
```



The commands in block N2 generate tool paths as indicated above.

Y-axis moves from point A to point B while the rotary table makes one turn.

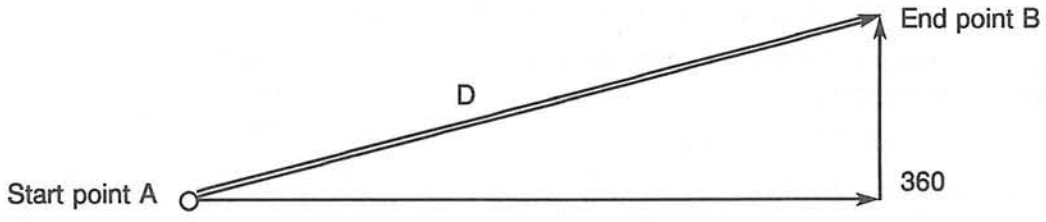
The time T required for the execution of the commands is calculated as indicated below:

In the simultaneous interpolation mode of the C-axis with other linear axis, feedrate unit may be considered as:

1 mm/min = 1 deg./min

For the program example, time T may be calculated as below:

C-axis : 360 mm (=360 deg.)
Y-axis : 100 mm



Distance between point A and point B:

$$D = \sqrt{(360)^2 + (100)^2} = 373.631$$

The control assumes the axis travel amount as D = 373.631 mm.

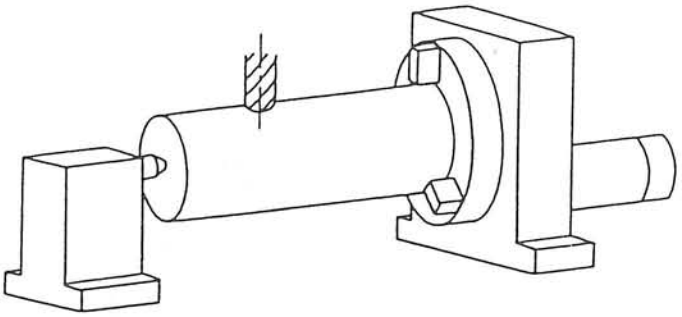
Since feedrate is programmed as F36 (mm/min), required time T is

$$T = D/F = 373.631/36 = 10.379 \text{ min}$$

Therefore, the time required for the execution of the programmed commands is 10 min 23 sec.

1-3-6. Calculating Feedrate Command F

(1) When Cutting is Made with Feeding Only A-axis



Assume the workpiece diameter is D = 200 mm, to obtain a feedrate of 150 mm/min on the workpiece OD, the feedrate to be programmed should be calculated in the following manner.

(a) One turn of a workpiece is equal to the cutter travel amount (L) as calculated below.

$$L = \pi \times D = 3.14 \times 200 = 628 \text{ mm}$$

(b) Time required for this cutting is calculated as indicated below if feedrate at the cutting point is f = 150 mm/min.

$$T = L/f = 628 \div 150 = 4.19 \text{ min}$$

(c) The travel amount L_{NC} calculated inside the control is

$$L_{NC} = 360° = 360 \text{ mm}$$

(d) The feedrate to be programmed as an F word is calculated as below:

$$F = L_{NC}/T = 360 \div 4.19 = 86$$

That is, programming F86 will provide the expected actual feedrate, 150 mm/min.

To simplify the calculation of a feedrate F, use the following formula:

$$F = \{360/(\pi \times D)\} \times f \quad \ldots\ldots \quad (1)$$

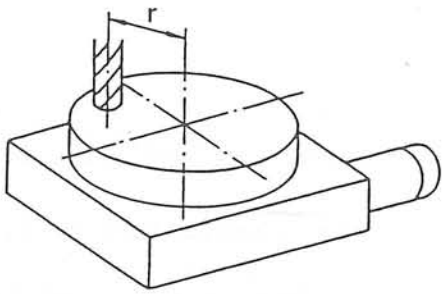
where,

f = feedrate required at the cutting point
D = workpiece diameter
π = ratio of circumference of circle to its diameter

With this formula, feedrate in the example above is calculated as

$$F = \{360/(3.14 \times 200)\} \times 150 = 86$$

For B- and C-axis, feedrate can be calculated using the same formula, For C-axis however, use two times the distance "r" between the table rotation center and the cutter position as "D".



(2) When Cutting is Made by Simultaneous A- and X-axis Movement

The example below gives the procedure how to determine the feedrate when feeding A-axis by 120° and X-axis by 150 mm in the simultaneous 2-axis control mode at a feedrate of 150 mm/min. In this example, workpiece diameter is D = 200 mm.

(a) The actual cutter travel distance L on the workpiece is

$$L = \sqrt{(\pi \times D \times a/360)^2 + x^2} = \sqrt{(3.14 \times 200 \times 120/360)^2 + 150^2} = 258 \text{ mm}$$

(b) Time required for this cutting is calculated as indicated below if feedrate at the cutting point is f = 150 mm/min.

$$T = L/f = 258 \div 150 = 1.72 \text{ min}$$

(c) The travel amount L_{NC} calculated inside the control is

$$L_{NC} = \sqrt{a^2 + x^2} = \sqrt{120^2 + 150^2} = 192 \text{ mm}$$

(d) The feedrate to be programmed as an F word is calculated as below:

$$F = L_{NC}/T = 192 \div 1.72 = 112$$

That is, programming F112 will provide the expected actual feedrate, 150 mm/min.

To simplify the calculation of a feedrate F, use the following formula:

$$F = \{\sqrt{a^2 + x^2} / \sqrt{(\pi \times D \times a/360)^2 + x^2}\} \times f \quad \ldots\ldots\ldots \quad (2)$$

where,

f = feedrate required at the cutting point
D = workpiece diameter
π = ratio of circumference of circle to its diameter
a = incremental value of A-axis command (deg.)
x = incremental value of B-axis command (mm)

Entering a = 360° and x = 0 mm, will yield formula (1).

If the simultaneous 2-axis control with B- and Y-axis is required, replace "a" and "x" with "b" and "y", respectively. Here, "b" represents the incremental value of B-axis and "y" the incremental value of Y-axis.

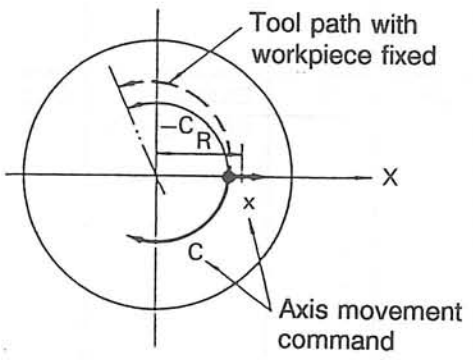
Simultaneous control including C-axis cannot provide a constant cutting speed at the cutting point because distance between the rotation center of C-axis and cutter changes as cutting progresses. In this case, divide the total cutting area into several segments in which change of distance is relatively small and calculate approximate feedrate F for each of these segments.

$$F' = \{\sqrt{c^2 + x^2} / (2 \times \pi \times R' \times C/360)\} \times f \quad \ldots\ldots\ldots \quad (3)$$

where,

C = incremental value of C-axis command (deg.)
R' = average value of distances between the C-axis rotation center and the cutter within a segmented area

Note that formula (3) is applicable only to the cases where simultaneous 2-axis control between C-axis and X-axis, and that X-axis passes the center of the C-axis rotation, and R' is relatively large compared with x.



1-4. Alarm

If an alarm occurs with a rotary axis, the ALARM lamp lights up and the corresponding alarm number and alarm message are displayed on the CRT.

Alarm No.	Alarm Level	Alarm Name	Cause
.100	A	DIF over	Follow-up error of the axis movement exceeds the permissible value. Index : AXIS Code : Follow-up error value
.101	A	DA over	Follow-up error of the axis movement exceeds the permissible value. Index : AXIS Code : Follow-up error value
.102	A	CON velocity	Variation amount of the position encoder exceeds the permissible value. Index : AXIS Code : Variation amount of CON
.103	A	APA velocity	Variation amount of the position encoder exceeds the permissible value. Index : AXIS Code : Variation amount of APA
.104	A	APA BCD data	Data read in the position encoder is not 0 to 9. Wrong data is read more than one time consecutively. Index : AXIS Code : XY X: Read-in digit None ..... Divided by "5" 1 ..... 1st digit 2 ..... 2nd digit 3 ..... 3rd digit 4 ..... 4th digit 5 ..... 5th digit 6 ..... 6th digit Y: Read-in BCD data A through E .... Interface error F ..... Position encoder error* * Digits other than 0, 1, 3, 7, F given for 5-divided digit
.565	B	Data word: rotating axis	Code : 1 ... Rotary or index table command was programmed in the following modes or programmed simultaneously with G code establishing them. - Circular interpolation mode (G02, G03) - Cutter radius compensation mode (G41, G42) - Three dimensional offset mode (G44) - Shape enlargement/reduction mode (G51) - Coordinate calculation mode - Area machining mode

2. Multi-Turn Specification

2-1. Outline

Operations of the multi-turn NC rotary table can be commanded in the same manner as linear axes. The rotary table rotation direction commands (M15 and M16) are no longer effective, and the rotation direction is determined based on the relationship between the actual position coordinates and the target position coordinates.

Mirror image function is also usable for the multi-turn NC rotary table.

2-2. Terminology

In this section, the terms "position" and "coordinate" are defined as indicated below.

Position:

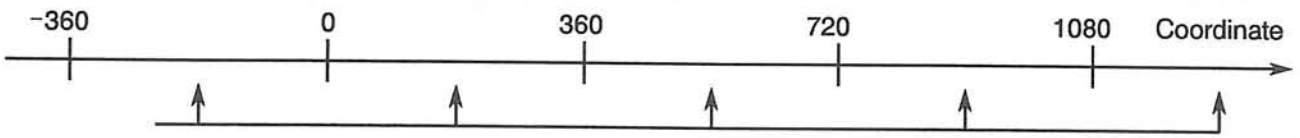
This indicates the physical position of the NC rotary table. That is, position data is expressed by any value between 0° and 360°.

0°, 360°, and 720° indicate the same position.

Coordinate:

This indicates the theoretical coordinate position of the NC rotary table.

0°, 360°, and 720° indicate different coordinate position.



These indicate physically the same position.

2-3. Program Format

2-3-1. Least Setting Unit

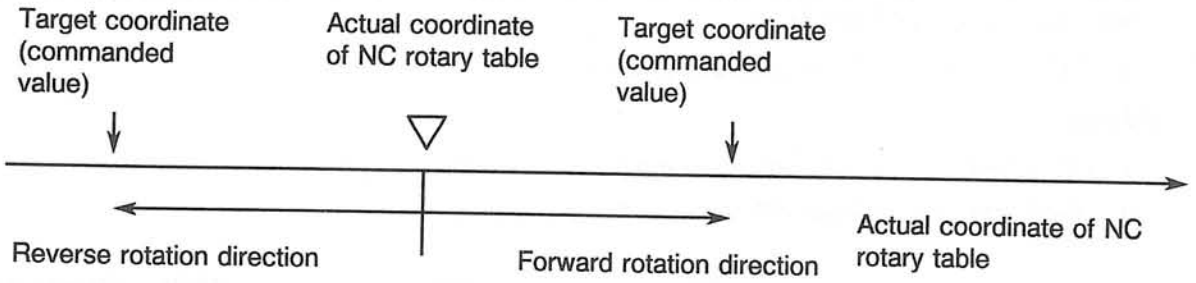
The least setting unit is 0.001°.

2-3-2. Programmable Range

-9999.999° - +9999.999°

2-3-3. Rotation Direction Command

If the programmed target position coordinate is larger than the actual position coordinate, the rotary table rotates in the forward direction. Conversely, if the target position coordinate is smaller than the actual position coordinate, the rotary table rotates in the reverse direction.



2-3-4. Feedrate Commands

Refer to Section 1, 3-5. and 3-6.

2-4. Operations

2-4-1. Automatic and MDI Modes

Commands, operations, and actual position display greater than 360° (one turn) is possible regardless of the absolute or incremental mode.

Example:

```
N001    G90   G0    C0.0
N002    C540.0
N003    G91   C540.0
```

N001 . . . . . Positioning of C-axis is made at 0° coordinate.

N002 . . . . . Positioning of C-axis is made at 540° coordinate. That is, the rotary table rotates by one and a half turns to make positioning at 180° position.

N003 . . . . . Positioning of C-axis is made at 1080° coordinate. That is, the rotary table rotates by one and a half turns from the current position to carry out positioning at 0° position.

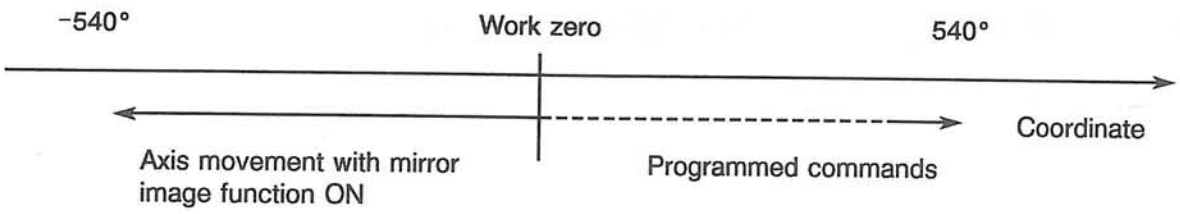
Mirror image:

(1) In Absolute Mode (G90)

Dimension commands are reversed with the work zero point taken as the center of mirror image function.

Example: Mirror image C-axis ON

```
N001    G90   C-20
N002          C540
```



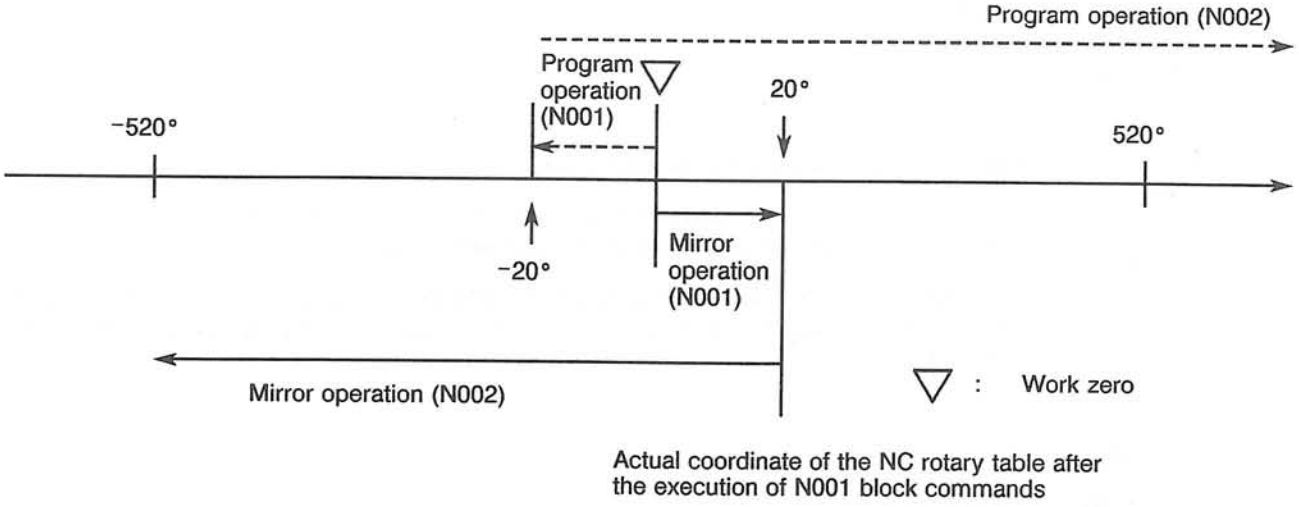
With the program above, C-axis is first positioned at +20° coordinate and then rotates by 560° in the reverse direction to carry out positioning at -540° coordinate.

(2) In Incremental Mode (G91)

Dimension commands are reversed with the current NC rotary table position taken as the center of mirror image function.

Example: Mirror image C-axis ON

```
N001    G90   C-20
N002    G91   C540
```



Actual coordinate of the NC rotary table after the execution of N001 block commands

2-4-2. Manual Mode

Positioning is carried out based on the coordinate values in the same manner as with a linear axis.

2-4-3. Manual Interruption and Pulse Handle Overlap

Same as in the manual mode

2-5. Parameter Setting

2-5-1. Parameters

NC rotary table multiple turn effective/ineffective is set at the NC optional parameter (bit) No. 39.

Bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
No. 39			Multi-turn C-axis	Multi-turn B-axis	Multi-turn A-axis			

Note: After the completion of setting, turn off the power supply once and then back it on again.

2-5-2. Rotary Axis with Limit

The operation of the NC rotary table with limit can be commanded in the same manner as linear axes, as in the case of the multi-turn NC rotary table. The relationship between the rotary table with limit and the multi-turn NC rotary table is as indicated below.

(1) When both the rotary table with limit and the multi-turn NC rotary table are selected, the rotary table with limit is given priority.

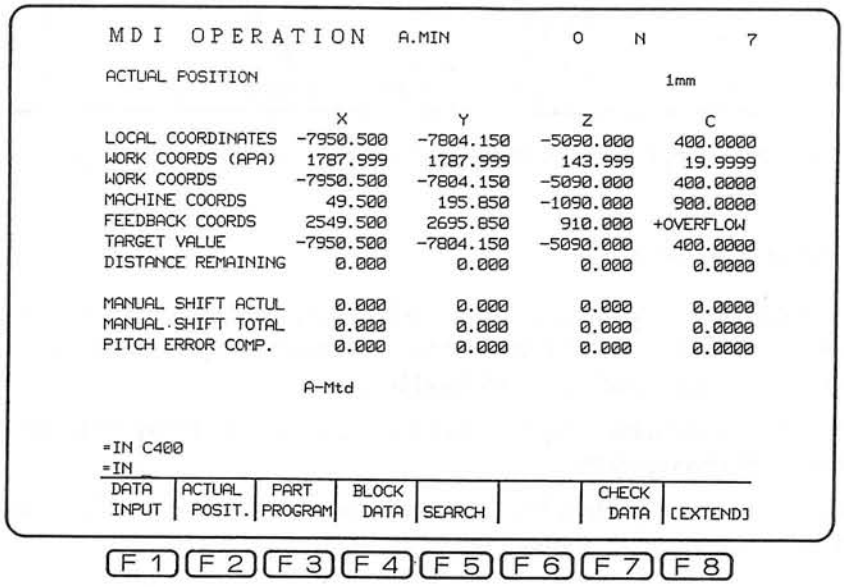
(2) When more than one multi-turn NC rotary table is used and the one is installed on the other, the lower table is recognized as a rotary table with limit.

2-6. Actual Position Data Display

The actual position data of NC rotary tables is displayed in coordinate values (-9999.999 - +9999.999°).

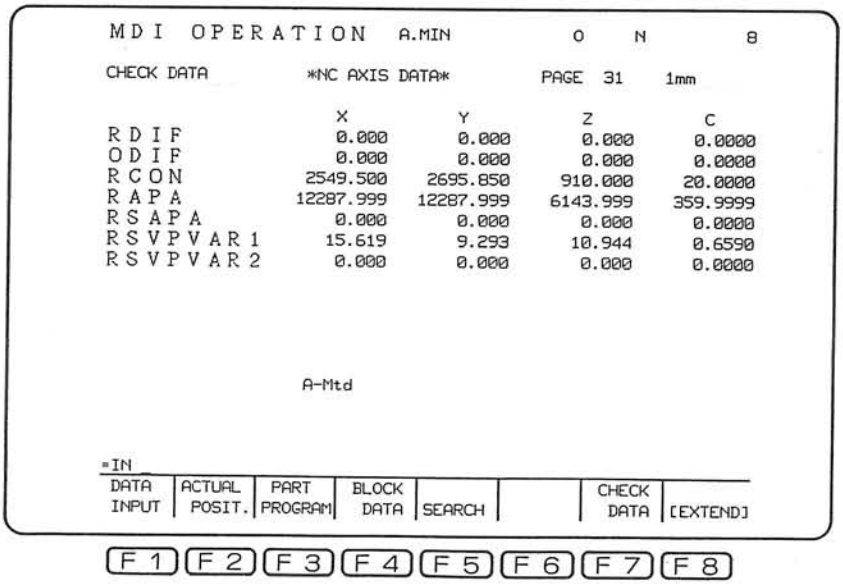
However, on the following screens, the data is displayed in "position".

(1) Actual Position Display (2nd page)



WORK COORDS (APA) is the position data display within a range of 0 - 359.999°.

(2) NC Axis Data



RCON and RAPA are the position data display within a range of 0 - 359.999°.

2-7. Power On/Off and NC Reset

Coordinate value of the NC rotary table on the work coordinate system is changed to any value between 0° and 359.999° corresponding to the actual coordinate value by the following operations.

(1) Power ON/OFF

(2) Machine lock ON to OFF

(3) NC reset to clear alarm A

It is possible to select whether or not the coordinate value (on the work coordinate system) of the NC rotary table is changed to any value between 0° and 359.999° by the NC reset operation for other than clearing alarm A, using the parameter (NC optional parameter (bit) No. 2).

Bit 1 of NC optional parameter (bit) No. 2:

1 . . . . . Reset to a value between 0° and 359.999°
0 . . . . . Not reset

Note that the "NC reset" includes the following operations:

- Pressing the **NC RESET** button
- Mode reset operation
  (Resetting the mode by selecting the manual mode on the operation panel)
- Machine lock ON/OFF operation
- Reset by M code (M02, M30)

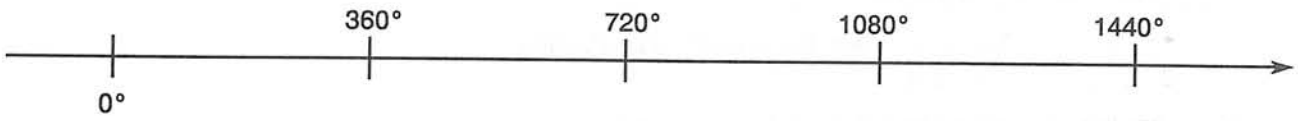
2-8. Sequence Restart and Home Position

2-8-1. Sequence Restart

With the multi-turn NC rotary table, return to the restart coordinate is accomplished within 360° movement by the sequence restart operation. In other words, positioning at the sequence restart point is carried out within one turn from the actual position and that position is taken as the restart coordinate.

Direction of the table rotation is determined based on the relationship between the actual coordinate and the sequence restart coordinate.

Example:



Assume that the actual coordinate is "1080".

```
N198    X50
N199    G90   C540
```

If restart is to be carried out from N199, the positioning at the sequence restart position occurs in the following manner.

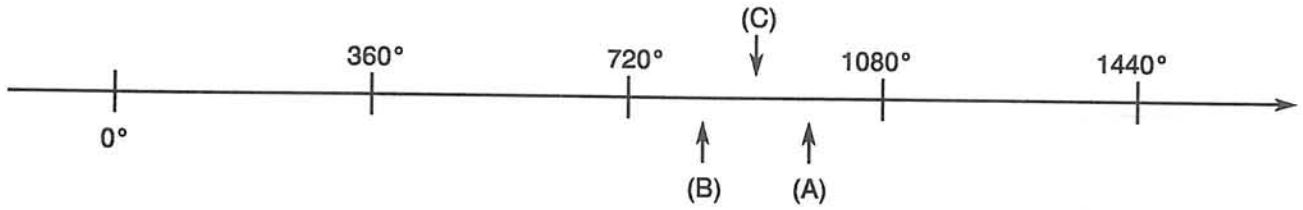
Since the restart coordinate is 540°, the table rotates in the reverse direction by 180° from 1080° coordinate to carry out positioning at 900°. Then, the actual coordinate is changed from 900° to 540°.

2-8-2. Home Position

(1) Positioning at Home Position

The multi-turn NC rotary table carries out positioning at the home position in the shortest path (within 360° turn).

Example:



Assume that 0° position is the home position.

(a) When the actual coordinate is 960°:

Positioning is carried out at the home position coordinate 1080° (0° position) in the forward rotation.

(b) When the actual coordinate is 840°:

Positioning is carried out at the home position coordinate 720° (0° position) in the reverse rotation.

(c) When the actual coordinate is 900°:

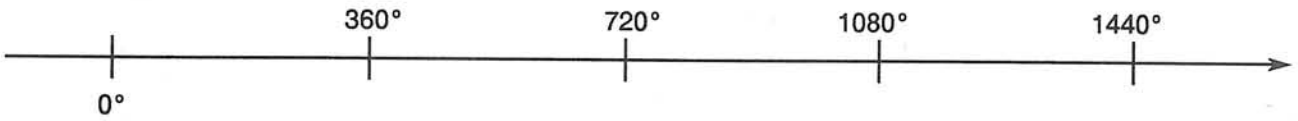
The case where the actual coordinate is equidistant from the two home position coordinates:

Positioning is carried out at the home position coordinate 1080° (0° position) in the forward rotation. That is, if the rotary table is equidistant from the two home positions, positioning at the home position is carried out in the forward direction.

(2) Incremental Command After Positioning at Home Position

When an incremental command is specified in the block immediately following the home position command (M60, G30), the incremental command is executed in reference to the coordinate which the rotary table took just before the execution of the home position return command.

Example:



Assume that 0° position is taken as the home position.

```
N101    G90  C960
N102    M60
N103    G91  C100
```

N101 . . . . . Positioning at 960° position

N102 . . . . . Positioning at the home position (1080°)

N103 . . . . . Positioning at 1060° (960 + 100) from 1080°

2-9. Work Zero Offset

The setting range of the work zero offset is indicated below:

−9999.999 - +9999.999°

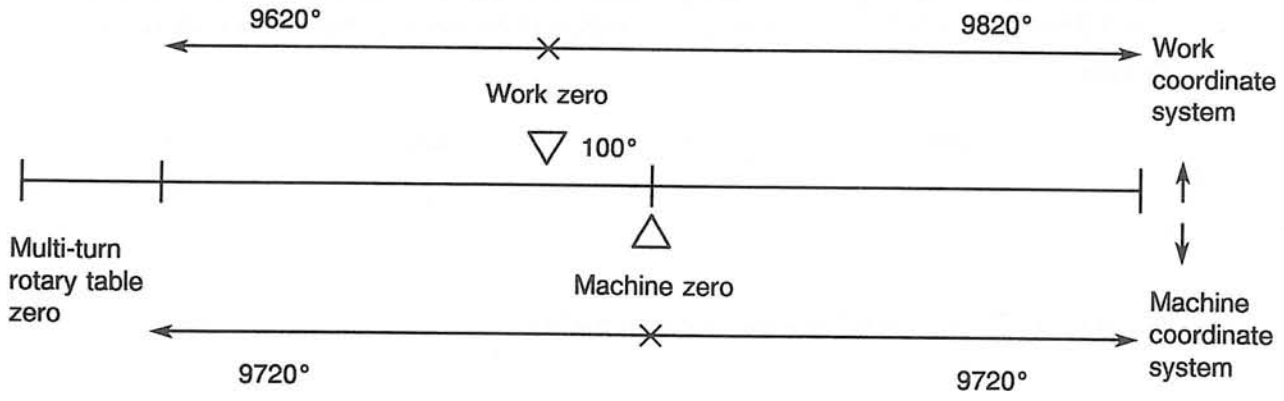
When the G92 command is specified, the work zero offset value is calculated so that the actual coordinate will take the coordinate value specified.

2-10. Limit Check

The setting is made so that rotary table movable range is as indicated below on the machine coordinate system (H = 0).

−9720.000 - +9720.000° (actual coordinate display, for ±27 turns)

Therefore, on work coordinate systems, the limit value is increased/decreased by the value equivalent to the work zero offset amount.



2-11. Installation of Multi-turn NC Rotary Table

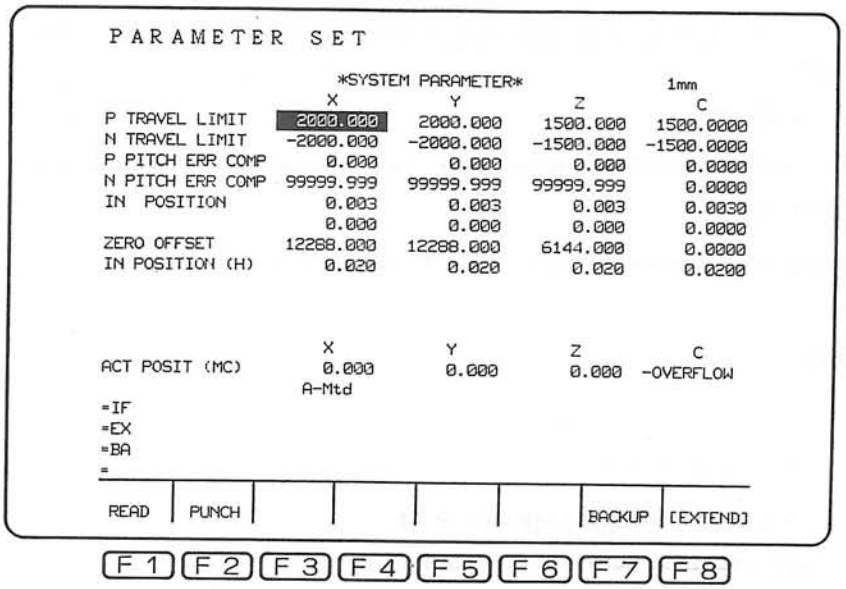
Use of a multi-turn NC rotary table requires the setting of system parameters as explained in 11-1.

<u>If the NC rotary table is to be rotated while the machine zero offset has not been set, rotate it only in the forward direction.</u>

If it is rotated in the reverse direction while the machine zero offset has not been set, an alarm occurs (alarm A 148 CON 0 pass). When the alarm lamp is illuminated, turn off the power supply to the NC once and then turn it back on.

After the installation of a multi-turn NC rotary table, set the system parameters. (Refer to 2-11-1.) After the system parameters have been set, always backup the set data by pressing the function key [BACKUP]. Note that the set system parameter data will be cleared if power to the NC is turned off before the data backup operation.

The data backup operation is explained below:



(1) Press any one of the mode selection keys; ZERO SET, TOOL DATA, and PARAMETER.

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

This displays the screen as indicated above.

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

A prompt "=BA" is displayed on the console line and data backup is executed continuously. After the completion of backup of data, a prompt "=" appears on the console line.

This backs up the data.

After setting system parameters, carry out data backup operation, turn off the power supply once and then turn it back on again.

REMARKS

2-11-1. Setting System Parameters

(1) Machine Zero Offset

Calculation, setting, and addition of the machine zero offset data for a multi-turn NC rotary table are carried out as explained below.

(a) Calculation

Machine zero offset (0 - 359.999°) is calculated from the actual position (CON 0 - 359.999°) of the multi-turn NC rotary table.

Calculation is made so that the machine zero offset data always falls within the range of 0 and 359.999°.

(b) Setting

Set the value which will fall within the range of 0 and 359.999°.

(c) Addition

The value set is added to the present machine zero offset data.

Carry out this operation in such a way that the value always falls within the range of 0 and 359.999°.

(2) Home Position Data

The home position data for the rotary table should be within the range below:

0 - 359.999°

Note that the setting of home position is not necessary if not required.

2-12. Alarm List

ALARM-A

148-* CON 0 pass 'Code'

Detected coordinate (CON) takes a negative value.

Check the setting of machine zero and P/N stroke limits.

* : represents an axis involved with the alarm.

Code: CON value

3. 0.0001° Specification

3-1. Outline

With the 0.0001° specification, axis commands for rotary axes (A, B, and C axes) can be specified in units of 0.0001°. The least setting unit is also 0.0001°.

3-2. Parameters

Set "1" at the NC optional parameter (bit) No. 4, bit 0.

3-3. Different Points from 0.001° Specification

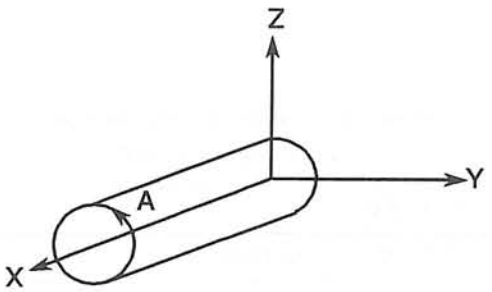
(1) Programmable Range

0.001° Specification	0.0001° Specification
0 - 359.000 (absolute value)	0 - 359.0000
−360.000 - +360.000 (incremental value)	−360.0000 - +360.0000
−9999.999 - +9999.999 (multi-turn)	−9999.9999 - +9999.9999

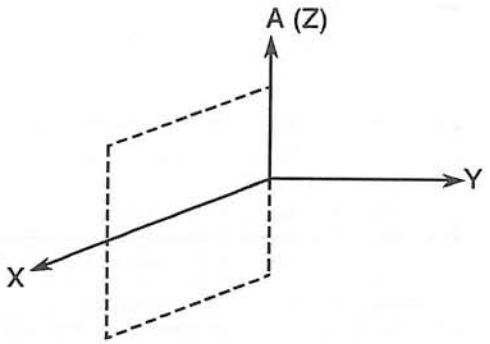
SECTION 22 CYLINDER SIDE-SURFACE MACHINING FUNCTION

1. Outline

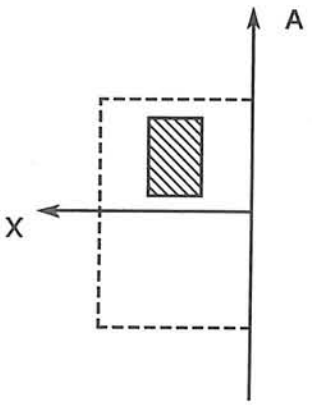
This function processes rotational axes in the same manner as linear axes, enabling the planes including the rotational axis to be selected. Cylinder side-surface machining can be executed by simply designating the usual plane machining image on the plane which is obtained by developing the cylinder side-face.



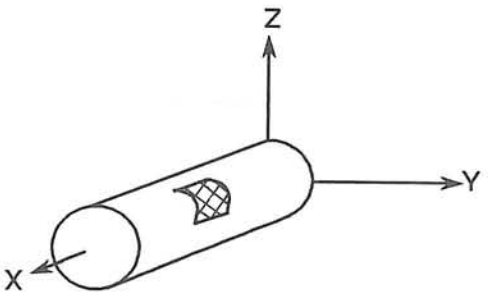
a) Rotational axis is used to execute plane development of the cylinder side-surface.



b) The plane which is obtained by developing the cylinder side-surface is designated using a G code (G17, G18, or G19).



c) The same image as that used for normal plane machining is designated, and machining begins.



d) Cylinder side-surface machining proceeds.

2. Programming Format

Follow the procedure below to execute cylinder side-surface machining:

(1) Select the plane which includes the rotational axis (G17, G18, or G19).

(2) Turn on the cylinder side-surface machining mode (G175R__).

(3) Designate machining commands. (For details regarding machining commands, refer to 2-3. "Machining Commands".)

- Designate rotational axis commands in units of "degree".
- Designate F commands in terms of the feedrate on the plane which is obtained by developing the cylinder side-surface.

(4) Turn off the cylinder side-surface machining mode (G174).

2-1. Cylinder Side-Surface Machining Mode

- Cylinder Side-Surface Machining Mode ON

```
G17 Xp_Yp_
G18 Zp_Zp_   } G175 R_
G19 Yp_Zp_
```

G17 : XpYp Plane selection.

G18 : ZpXp Plane selection.

G19 : YpZp Plane selection.

Xp : X-axis, U-axis, or the rotational axis whose corresponding basic axis is X-axis

Yp : Y-axis, V-axis, or the rotational axis whose corresponding basic axis is Y-axis

Zp : Z-axis, W-axis, or the rotational axis whose corresponding basic axis is Z-axis (For details regarding the "corresponding basic axis", refer to 2-2. which follows.)

G175 : Cylinder side-surface machining mode ON

R : Cylinder radius (1 - 5,000 mm)

The axes included in the selected plane are determined by the axis address in the block where plane selection is designated.

Example:

```
G17 A_X_ (A = -Y) ..... AX plane
G17 B_Y_ (B = X) ..... BY plane
G18 C_Z_ (C = -X) ..... CZ plane
G18 A_X_ (A = Z) ..... AX plane
G19 B_Y_ (B = -Z) ..... BY plane
G19 C_Z_ (C = Y) ..... CZ plane
```

- Cylinder Side-Surface Machining Mode OFF

  G174

Note 1: *In the cylinder side-surface machining mode, be sure to designate the plane which includes the rotational axis.*

Note 2: *A plane cannot be selected using the basic axes, corresponding rotational axes, and parallel axes.*

Note 3: *The rotational axis name, rotation direction, and the corresponding basic axis must be designated in advance at the "rotation axis" parameter. (For details regarding the "rotational axis" parameter, refer to Section 21 of this manual.)*

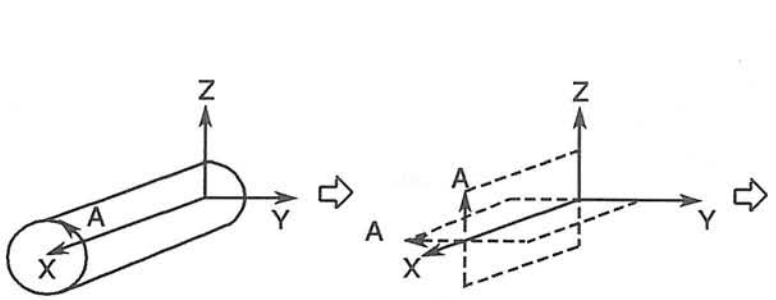
Note 4: *Rotational axis addresses "A", "B", and "C" are designated as angle commands.*

Note 5: *Do not designate both the plane selection command (G17, G18, or G19) and the cylinder side-surface machining mode ON command (G175) in the same block.*

2-2. Corresponding Basic Axis

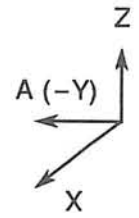
In cylinder side-surface machining, the rotational axis is replaced by a basic axis. The basic axis which is used to replace the rotational axis is referred to as the "corresponding basic axis". This corresponding basic axis can be either of the two basic axes which are not used as the center (pivot) axis of the rotational axis.

The plane which is formed by the corresponding basic axis and the center (pivot) axis is identical to the plane which is obtained by developing the cylinder side-surface.

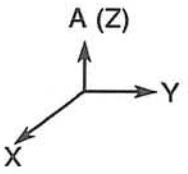


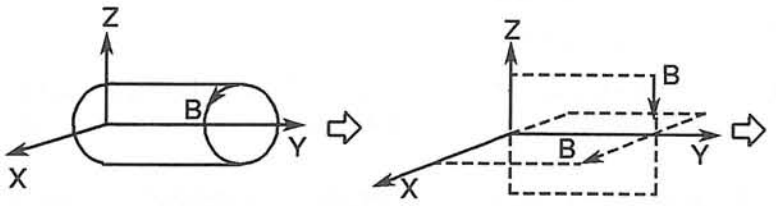
A-X plane

When the corresponding basic axis of the A-axis is −Y



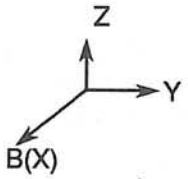
When the corresponding basic axis of the A-axis is Z



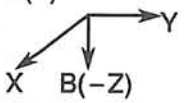


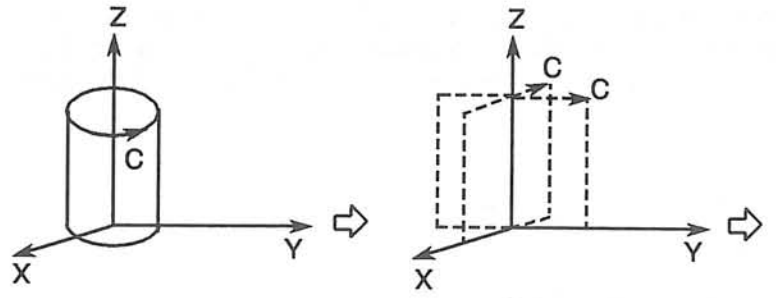
B-Y plane

When the corresponding basic axis of the B-axis is X



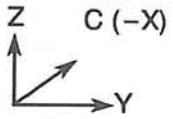
When the corresponding basic axis of the B-axis is -Z



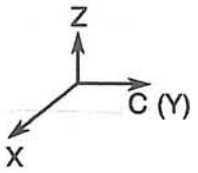


C-Z plane

When the corresponding basic axis of the C-axis is -X



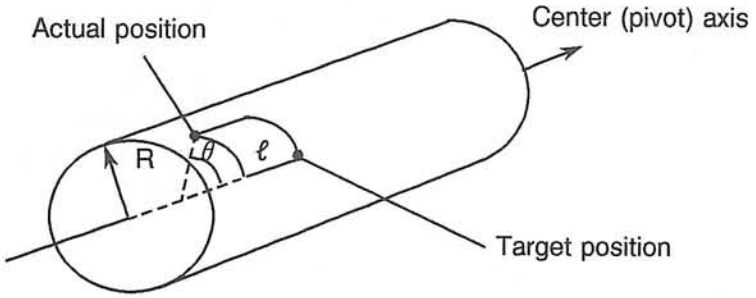
When the corresponding basic axis of the C-axis is Y



The directions of the corresponding basic axes shown above conform to the ISO Standard.

2-3. Machining Commands

The machining command is executed after the cylinder side-surface machining mode has been switched ON. The cylinder side-surface length is then converted to degrees in order to designate the rotational axis. The conversion formula is given below.



Rotational axis command value [deg] : $\theta = \frac{360\ell}{2\pi R}$

ℓ : Length of plane which is obtained by developing the cylinder side-surface

R : Cylinder radius

The F command is designated by the speed on the plane which is obtained by developing the cylinder side-surface. For details regarding the method for obtaining the cutting feedrate, refer to Section 21 of this manual.

Note: *When referring to the "Programming Manual for OSP5020M/OSP500M-G" regarding the command procedures, it should be noted that, unless otherwise stated, the X-axis is the horizontal axis, the Y-axis is the vertical axis, and the Z-axis is the axis which is vertical to the selected plane. The I, J, and K commands should be considered in the same manner.*

(1) Positioning (G00) and Linear Interpolation (G01)

Except for the feedrate command, program commands can be designated in the usual manner in the positioning and linear interpolation modes. Angle commands (AG) can also be used in the usual manner. For further details, refer to the Programming Manual for OSP5020M/OSP500M-G.

(2) Circular Interpolation (G02, G03)

Arc on the G17 (Xp-Yp) plane	G02 G03	Xp__Yp__I__J__F__
Arc on the G18 (Zp-Xp) plane	G02 G03	Zp__Xp__K__I__F__
Arc on the G19 (Yp-Zp) plane	G02 G03	Yp__Zp__J__K__F__

I : Plus or minus distance between the start point and the center of the Xp-axis

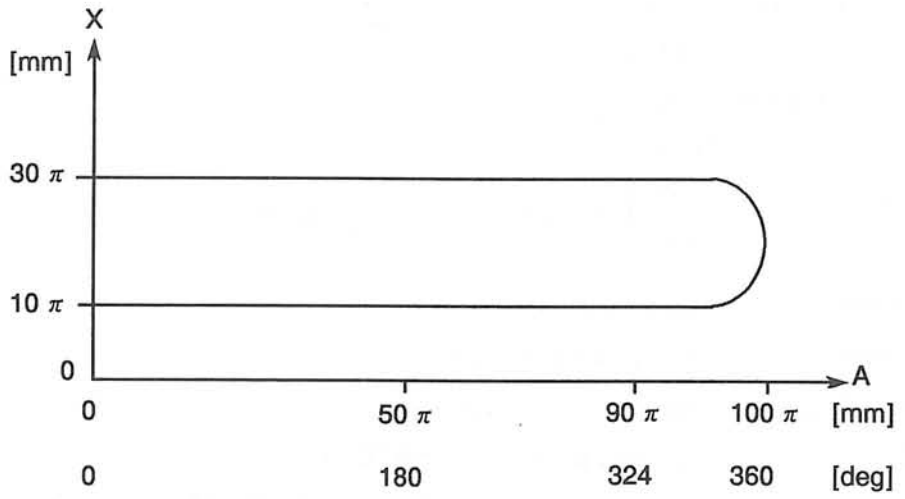
J : Plus or minus distance between the start point and the center of the Yp-axis

K : Plus or minus distance between the start point and the center of the Zp-axis

F : Cutting feedrate (speed on the plane which is obtained by developing the cylinder side-surface)

Example

When cylinder radius R = 50 mm




```
        :
G17     A0      X=10*3.14
G175    R50
G01     A324    F500
G03     A324    X=30*3.14       I=10*3.14 J0
G01     A0
G174
        :
```

The rotational axis designation cannot be made until the cylinder side-surface length has been converted to an angle value (degrees).

For the above example:

- Rotational axis is the A axis.
- Corresponding basic axis is the −Y axis.

Therefore, the A and J address characters are designated in units of degrees.

Rotational axis parameters

Axis Name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	−Y axis

(3) Cutter Radius Compensation (G40, G41, G42)

Cutter radius compensation is executed on the plane which is obtained by developing the cylinder side-surface. Function and commands are the same as usual. Refer to the Programming Manual for OSP5020M/OSP500M-G for details.

(4) Arbitrary Angle Chamfering

When chamfering (C-chamfer, R-chamfer) a corner having an arbitrary angle, simply designate the corner apex and the chamfer amount. The start and end points for chamfering will then automatically be determined, and chamfering will be executed accordingly.

G17 (Xp-Yp) plane { CHFC / CHFR } Xp_Yp_L_[Q_][I_][J_]

G18 (Xp-Yp) plane { CHFC / CHFR } Zp_Xp_L_[Q_][K_][I_]

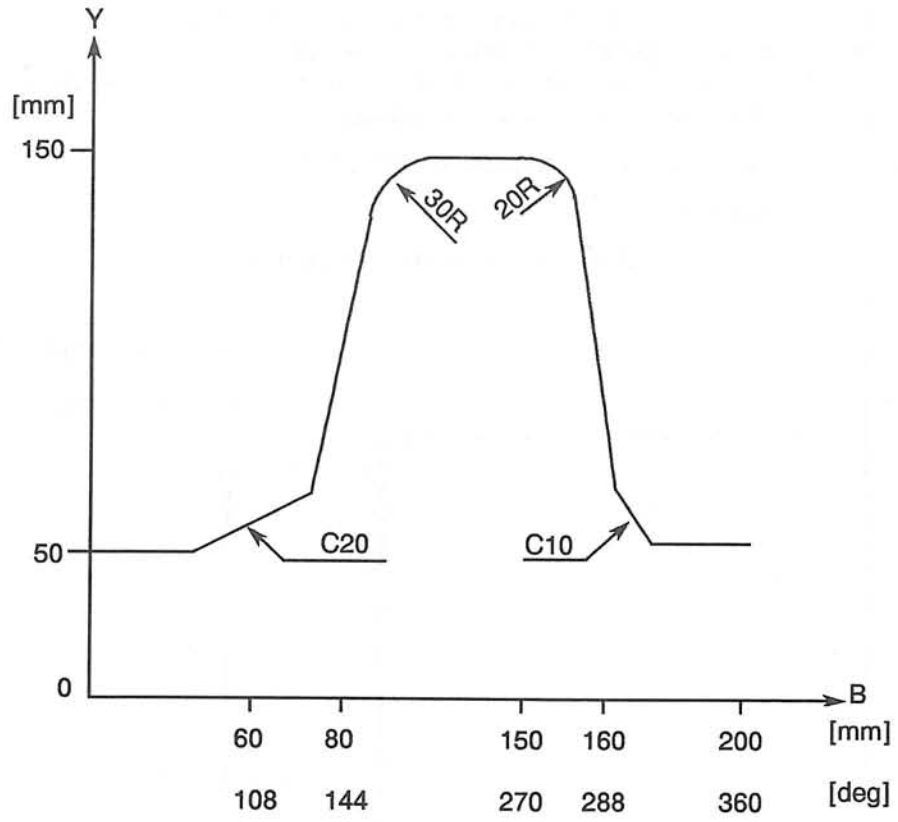
G17 (Xp-Yp) plane { CHFC / CHFR } Yp_Zp_L_[Q_][J_][K_]

CHFC	:	C-chamfer command
CHFR	:	R-chamfer command
Xp, Yp, Zp	:	Corner apex coordinate value
L	:	Chamfer amount (0 - 2099999999 mm)
Q	:	Travel amount after chamfering (0 - 99999999 mm)
I, J, K	:	Virtual advancing direction (direction vector of Q)
		I . . . . . Xp-axis direction
		J . . . . . Yp-axis direction
		K . . . . . Zp-axis direction

Note 2: The "L" chamfer amount and the "Q" travel amount after chamfering are always designated as length commands.

Example:

When cylinder radius R = 32.831 mm




```
    :
G17          B0       Y50
G175  R31.831
CHFC  G01    B108     Y50     L20     F2000
CHFR         B144     Y150    L30
CHFR         B270     Y150    L20
CHFC         B288     Y50     L10     Q30     I1  J0
G174
    :
```

Address characters B and I are designated as angle values (degrees).

Rotational axis parameters:

Axis name	:	B-axis
Rotation direction	:	Plus
Corresponding basic axis	:	X-axis

(5) Fixed Cycle

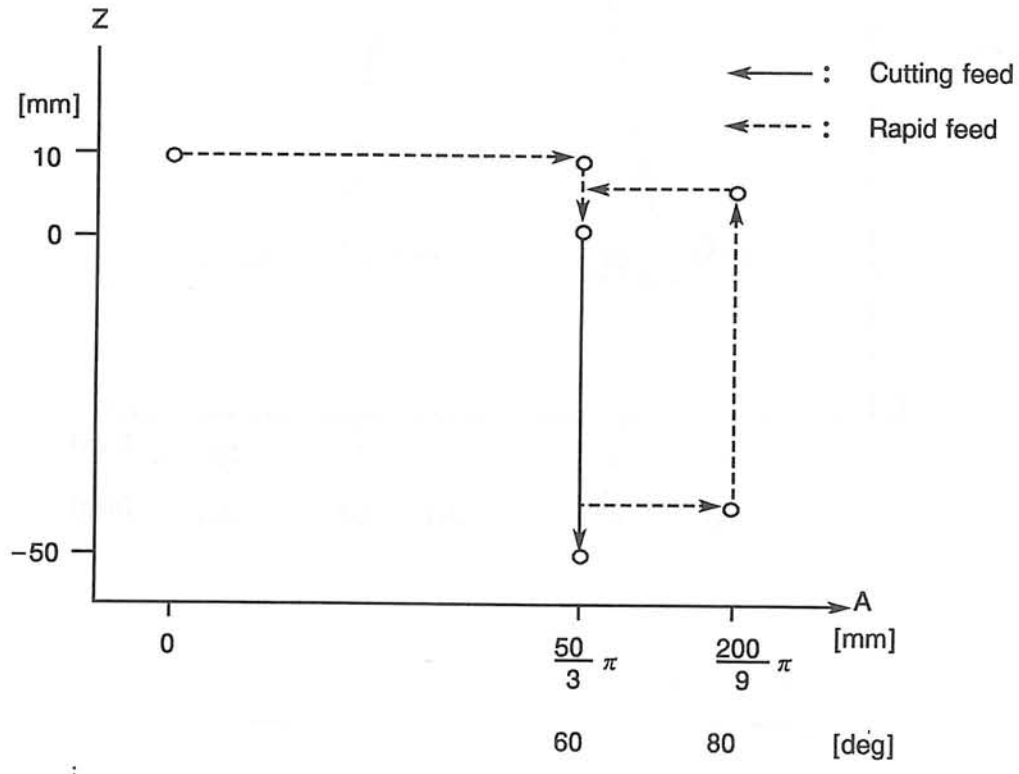
Fixed cycle machining is executed in the cylinder side-surface coordinates. A conventional linear axis is used as the cycle axis.

Shift amounts I, J, and K in the fine-boring (G76) and back-boring (G87) cycles are designated as angle values when corresponding rotational axes are present. However, "I" (depth of cut per pass) and "J" (tool tip retract amount) in the deep-boring cycle (G83) are designated as length values, regardless of the plane which has been selected.

For details, refer to the Programming Manual for OSP5020M.

Example: Fine-boring cycle

When cylinder radius R = 50 mm




```
    :
G00     Z10
G17     A0     X0
G175    R50
G76     A60    X50    Z-50    R0    I0    J20    P1    F100
G174
    :
```

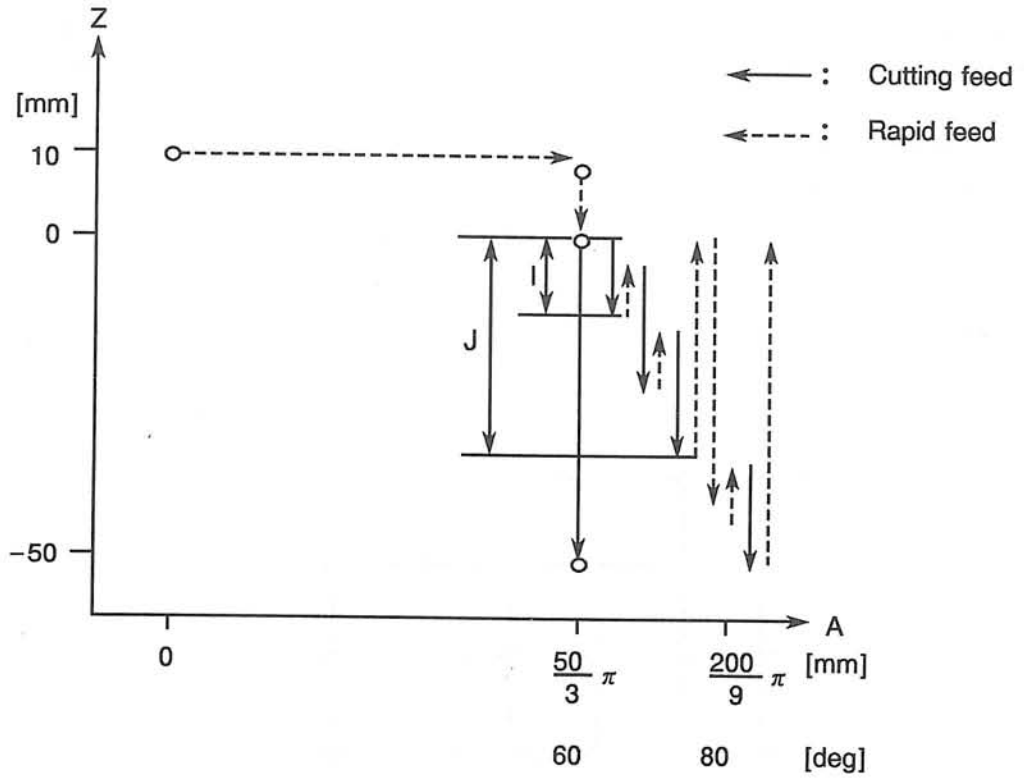
Address character "J" represents the shift amount in the A-axis (-Y axis) direction, and is therefore designated as an angle value (degrees).

Rotational axis parameters:

Axis name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	−Y

Example: Deep-boring cycle

When cylinder radius R = 50 mm




```
    :
G17     G00 A0   X0   Z10
G175    R50
G83     A60 X50 Z-50   R0  I0  J30  P1  F2000
G174
    :
```

Address characters I and J in the deep-boring cycle must be designated as length values.

Rotational axis parameters:

Axis name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	−Y

Note 3: *Shift amounts I, J, and K in the fine-boring (G76) and back-boring (G87) are designated for the Xp-, Yp-, and Zp-axis directions, respectively.*

(6) Coordinate Calculation Function

A coordinate calculation operation is executed in the cylinder side-surface coordinates.

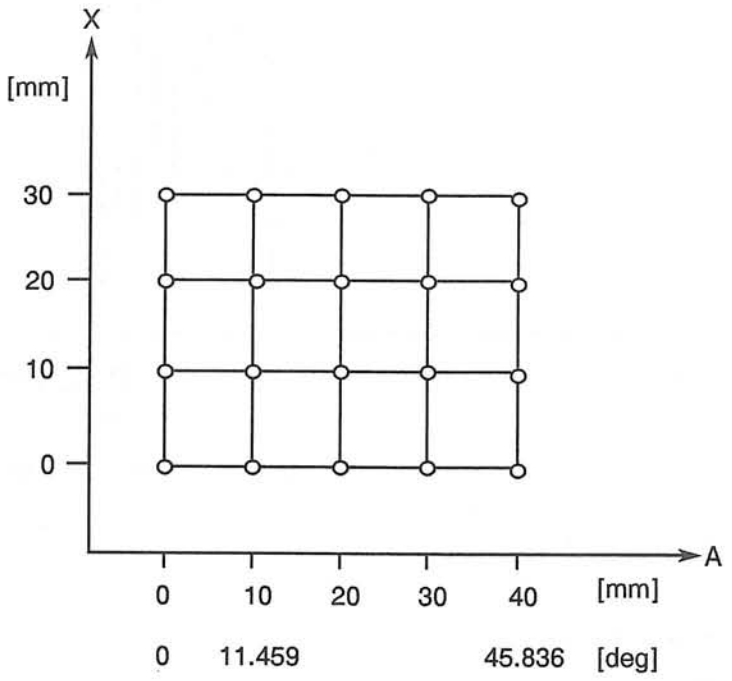
The following commands are designated as angle values (degrees):

Reference point coordinate, I and J of the grid, staggered grid, and square (interval in the horizontal and vertical directions), Q and R of the staggered grid (stagger interval in the horizontal and vertical directions)

For details, refer to the Programming Manual for OSP5020M/OSP500M-G.

Example: Grid X

When cylinder radius R = 50 mm




```
    :
G17     A0  X0
G175    R50
G83     G81 R0 Z-15 F100
A0      X0
GRDX    [A0 Z0] I11.459 J10  K4   P3
G174
    :
```

Address character "I" represents the shift amount in the A-axis (-Y axis) direction, and is therefore designated as an angle value (degrees).

Rotational axis parameters:

Axis Name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	−Y

(7) Area Machining

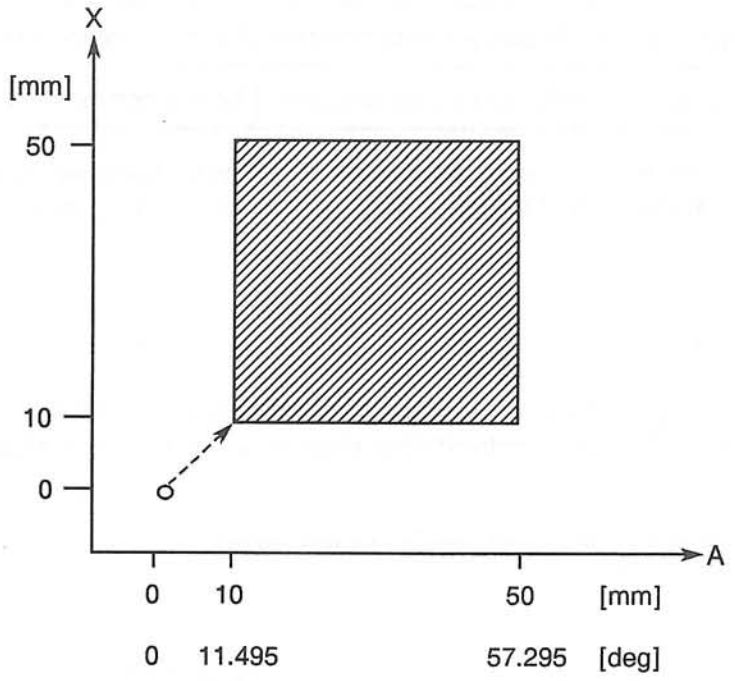
Area machining is executed in the cylinder side-face coordinates. A conventional linear axis is used as the cycle axis.

The coordinate values of the reference point, I (length in the horizontal-axis direction), and J (length in the vertical-axis direction) are designated as angle values (degrees) when corresponding rotational axes are present.

For details, refer to the Programming Manual for OSP5020M/OSP500M-G.

Example: Surface machining

When cylinder radius R = 50 mm




```
    :
G18     G0  A0  X0  Y0
G175    R50
FMILR   A11.459 X10 Y-50 I45.836   J40  K1   P70
Q10     R0  D1  F1
G174
    :
```

Address character "I" represents the length in the A-axis (Z axis) direction, and is therefore designated as an angle value.

Rotational axis parameters:

Axis name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	−Y

Note 4: In area machining, the I command is for the horizontal axis direction and the J command is for the vertical axis direction.

	I (Horizontal axis)	J (Vertical axis)
G17	Angle command in the A-axis (-Y-axis) direction	Length command in the X-axis direction
	Angle command in the B-axis (X-axis) direction	Length command in the Y-axis direction
G18	Angle command in the C-axis (-X-axis) direction	Length command in the Z-axis direction
	Angle command in the A-axis (Z-axis) direction	Length command in the X-axis direction
G19	Angle command in the B-axis (-Z-axis) direction	Length command in the Y-axis direction
	Angle command in the C-axis (Y-axis) direction	Length command in the Z-axis direction

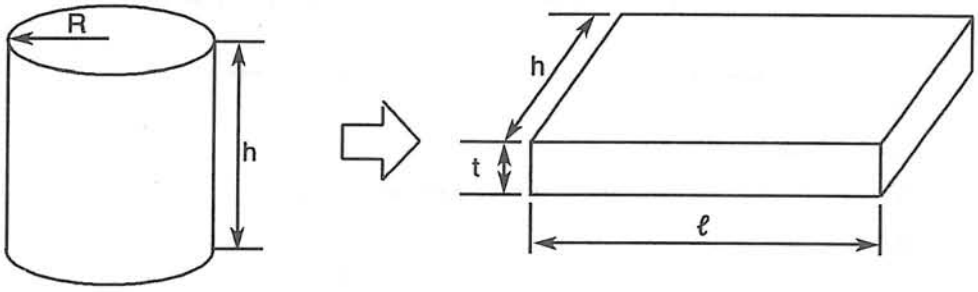
The above table is effective only when the rotation directions of rotational axes and the corresponding basic axes conform to the ISO Standard. The corresponding basic axis is shown in parentheses in the above table.

3. Animation Function

A graphic display of the cylinder side-surface machining operation can be executed. Note however only tool movement on the square pole which is obtained by developing the cylinder can be displayed.

- Blank Definition

  The blank is defined as a square pole, as shown below.



  h : Cylinder height

  ℓ : Circumferential length (range of rotational axis movement)

  t : Length equivalent to the radius (an arbitrary value can be designated because there is no display in the radial direction.)

- Screen Data

  Screen data for the cylinder side-surface machining is designated as shown below.

  ① 1st Page

  Rotational axis

  Designate a " * " setting.

② 3rd Page

Drawing center and drawing range

In addition to conventional length (mm) settings and adding operations, angle (degree) settings and adding operations are also possible.

[F1] (SET) "Setting value (deg)"; D **[WRITE]**

By designating ";D", the angle value is converted to a length value based on the conversion coefficient.

Calculation Coefficient (mm/deg)

Set the cylinder radius value followed by ";R".

The following calculation will be performed:

2π * "setting value"/360

This value serves as the coefficient for converting the angle value (deg) to a length value (mm).

If the coefficient determined here is different from that in the program (value for address "R" designated simultaneously with the G175 command), the animation function will be inaccurate.

③ 4th Page

Master axis designation

Designate the name of the rotational axis. The cursor direction will be fixed in the corresponding basic axis direction (designated by rotational axis parameters).

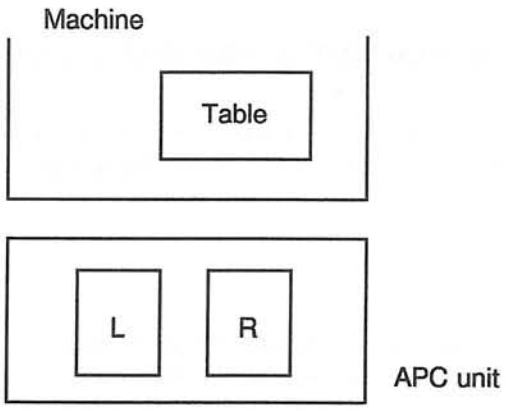
Note 1: *All settings other than those described above are executed in the usual manner. For details, refer to Section 4 "Animation Function" of the Special Functions Manual of OSP5020M/OSP500M-G (publication No. 3294-E).*

Note 2: *Even if 360° is exceeded during machining operation, coordinates will be displayed consecutively. However, the blank definition "L" basically must be within the following range: $0 \leqq \ell \leqq 360$ deg. Designate the movable range of the rotational axis taking this into consideration.*

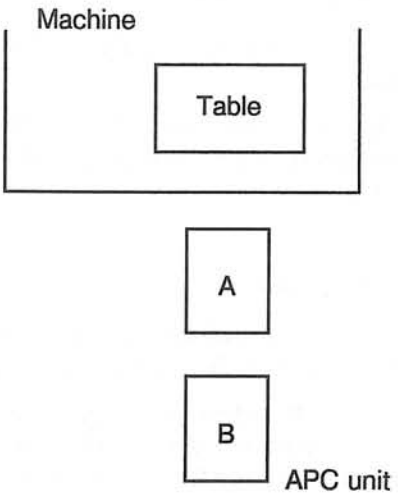
SECTION 23 2-PALLET APC PALLET IDENTIFICATION FUNCTION

1. Outline

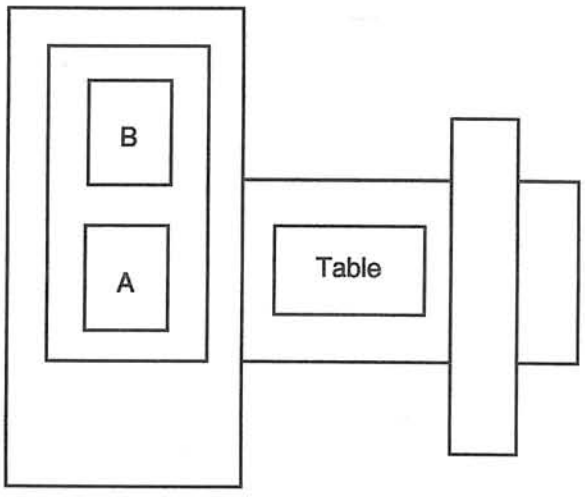
The 2-pallet APC pallet identification function is used with parallel 2-pallet APCs and rotary 2-pallet APCs to determine whether the pallet on the machine table is an "A" (or L) pallet or a "B" (or R) pallet. This identification function can be executed in the part program using system variables and the pallet identification command.



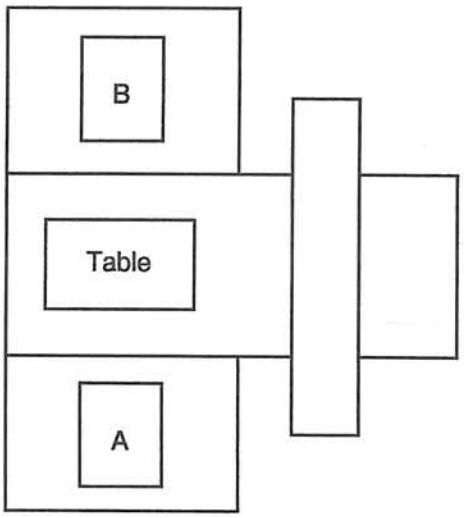
<Parallel 2-pallet APC>



<Rotary 2-pallet APC>



<Gate type vertical loading APC>



<Gate type horizontal loading APC>

2. System Variables for Pallet Identification

Name	VPLTK
Description	Type of pallet on the machine table
Attribute	READ only
Data	1: A (L) pallet 2: B (R) pallet (0: No pallet, or pallet ID not possible)

Program example:

```
N00 1   IF  [VPLTK EQ1] N100
        IF  [VPLTK EQ2] N200
        MSG [PALLET NG]  .......  Used only when the message function is provided
        GOTO    NEND
            :
N100    (part program for A-pallet)
            :
        GOTO    NEND
N200    (part program for B-pallet)
            :
NEND    M02
```

Program proceeds to "N100" for A-pallet, or to "N200" for B-pallet. Program changes according to the identified pallet.

3. Pallet Identification Command

This command enables the program to determine whether the pallet on the machine table is an "A" (L) or "B" (R) pallet.

3-1. Programming Format

[RP = *M289]

- M289 : Pallet identification command
- RP = * : Data of the pallet on the machine table
  - *:1 .... A (L) pallet
  - *:2 .... B (R) pallet

3-2. Application Example (Program Example)

```
        :
N100    M60
N200    RP=1 M289
        :
```

After a pallet change occurs, the pallet on the machine table is identified. If the pallet is an "A" pallet, program operation continues. If the pallet is not an "A" pallet, an alarm of level B will be activated and the program will be stopped at the N200 block.

3-3. Alarms

<Alarm B>

707 Pallet discrimination NG

The pallet ID operation executed at RP=*M289 indicates that the pallet type is not as desired.

708 Data word: 'RP'

The " * " value at the RP=*M289 command is other than 0, 1, and 2.

Code : Hexadecimal number of the command value

SECTION 24 REMOTE MONITORING SYSTEM/ COMMUNICATIONS UNIT

1. Overview

The remote monitoring system enables the communications between the NC machine and the personal computer through the telephone line to diagnose the trouble occurring on the NC machine with the personal computer from a distant place. This system can be installed easily by installing the communications unit on the NC and connecting the telephone line to the unit.

2. Consulting Okuma Representative

When any trouble has happened to the NC machine, call your local Okuma representative and explain the details of the trouble to the service engineer. If possible, leave the NC machine as it was when the trouble happened. The service engineer examines the customer's explanation and calls the required information from the NC machine to the personal computer in order to diagnose the trouble using the remote monitoring system.

3. Rmote Monitoring Operation

3-1. Starting the System

To establish the remote communication mode, press the **MODE** switch on the RMS (Remote Monitoring System) console mounted on the control box of the NC machine.

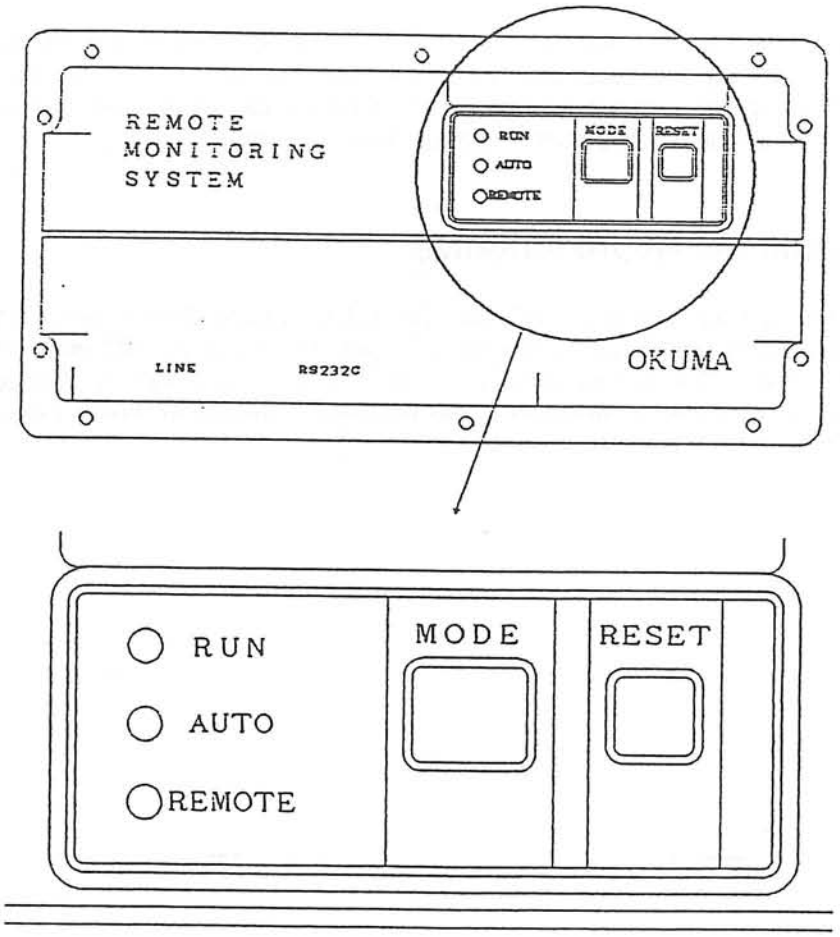


Fig. 24-1 Switches on the RMS Console

3-2. Messages for Starting the System

When the **MODE** switch is pressed, the messages for starting the system will be displayed on the CRT screen. (Fig. 24-2)

The message asking in which the connecting mode (manual or automatic) the telephone line should be connected will be displayed.

```
  Remote Monitoring System
Starting the Remote Monitoring
System
   Connecting in Manual Answer
mode?                Yes:Y, No:N

    To stop a transmisson, press
the MODE switch on the RMS
console.
```

Fig. 24-2 Messages for Starting the System

Manual Mode:

With this mode, the currently receiving telephone line which is wired to the communication unit is switched to the communication unit. (special mode)

Automatic Mode

With this mode, the communication unit automatically receives the telephone which is wired to the communication unit. Select this mode when switching the line to the communication unit from other telephone through the extension line.

3-2-1. Confirmation Messages for Starting the System (Automatic Mode)

When the manual mode is not selected (typing "N" on the screen (Fig. 24-2)), the confirmation messages for starting the system will be displayed on the CRT screen. (Fig. 24-3)

When the conditions confirmed by the messages are satisfied for starting the system, press the function key [F8]. The messages for send mode will be displayed on the screen. (Fig. 24-4)

```
  Remote Monitoring System
 With the machine not operating,
make sure the telephone line is
connected properly. Then press
the F8 key.

    To stop a transmisson, press
the MODE switch on the RMS
console.
```

Fig. 24-3 Confirmation Messages for Starting the System

```
  Remote Monitoring System
                        Send Mode
All NC operation panel keys be-
come inoperable during a trans-
mission.

    To stop a transmisson, press
the MODE switch on the RMS
console.
```

Fig. 24-4 Messages for Send Mode

3-2-2. Messages for Switching the Line (Manual Mode)

When the manual mode is selected (typing "Y" on the screen (Fig. 24-2)), the messages for switching the line will be displayed on the CRT screen. (Fig. 24-5)

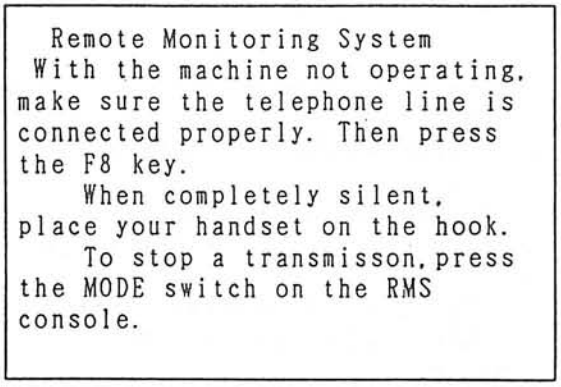


Fig. 24-5 Messages for Switching the Line

When the telephone line is disconnected at the personal computer, sound from the telephone at the NC machine will fade away. When no sound can be heard at all, press the function key [F8] and place the handset on the hook.
The telephone line can be switched to the communications unit.

3-3. Remote Operation Mode

When remote monitoring is started, the message for the remote operation mode will be displayed on the screen. (Fig. 24-6)

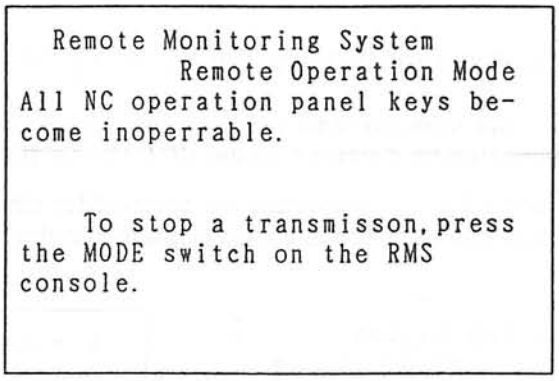


Fig. 24-6 Messages for Remote Operation Mode

In this mode, the machine cannot be operated using the keys on the operation panel.
If any machine operation is needed, press the **MODE** switch on the RMS console to stop communications. The operation panel will be operative. (Fig. 24-1)
Communications can also be stopped by turning off the control power.

3-4. Messages for Diagnostic Results

When the diagnosis of the trouble is completed on the personal computer, the diagnostic results will be displayed on the CRT screen. (Fig. 24-7)
When the color CRT is selected, the display color changes from sky-blue to yellow.

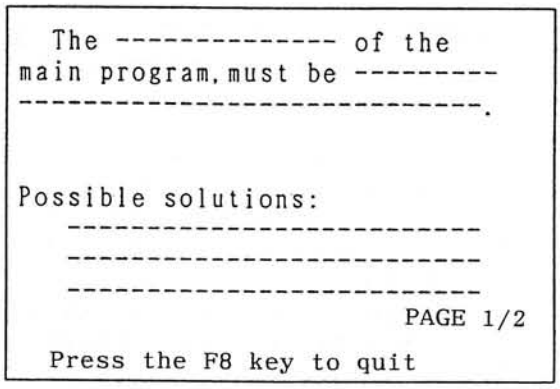


Fig. 24-7 Messages for Diagnostic Results

In the example in Fig. 24-7, the message indicates that the programming error caused the trouble and gives the measures to take.

At the lower right of the screen, the number of pages for the diagnostic result message and the currently displayed page number are displayed. Use cursor control keys [↑] and [↓] to see the previous page and next page. Press the function key [F8] to quit this display. When the function key [F8] is pressed, the message for quitting the remote monitoring mode will be displayed. (Fig. 24-8)

3-5. Quitting the Remote Monitoring Mode

Type "Y" to return to the normal NC mode. Program or operation mode might be changed during the remote monitoring. Check the program or the operation mode before starting normal NC operation.

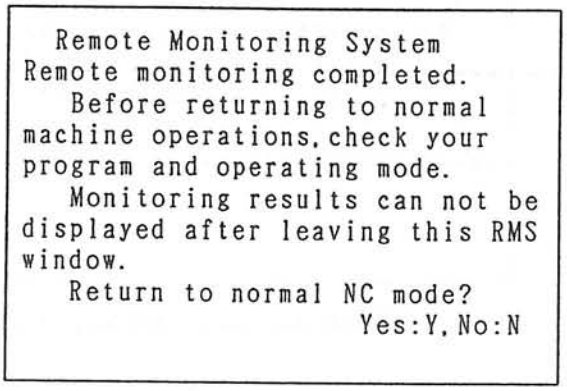


Fig. 24-8 Messages for Quitting the Remote Monitoring Mode

3-6. Re-displaying the Diagnostic Results

To display the diagnostic results screen again after returning to the normal NC mode, hole down the **MODE** switch on the RMS console for three seconds or longer. The AUTO and REMOTE indication lamps will start blinking and the diagnostic results will be displayed.

Note: If the control power is turned off after communications, the diagnostic results will be cleared.

4. Errors

4-1. Error Messages

The communications error message will be displayed when the telephone connected to the communications unit receives a call from other than the personal computer. (Fig. 24-9)
Wait for a while or press the **MODE** switch on the RMS console again. Pressing the **MODE** switch first time clears the message window and second time displays the message for starting the system (Fig. 24-2).

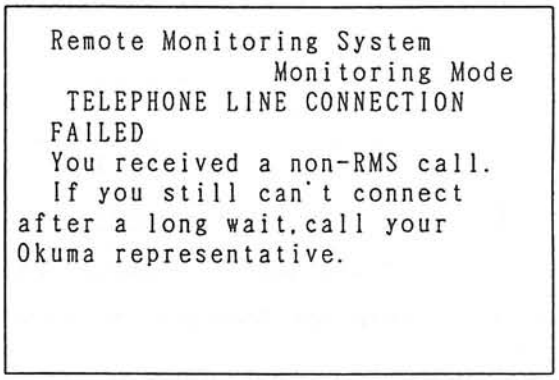


Fig. 24-9 Communications Error Message

The communications unit error message will be displayed when an error occurs in the communications unit. Press the **RESET** switch and the **MODE** switch to display the message for starting the system (Fig. 24-2). If this message is displayed repeatedly, consult your local Okuma representative. The communications unit might have any failure.

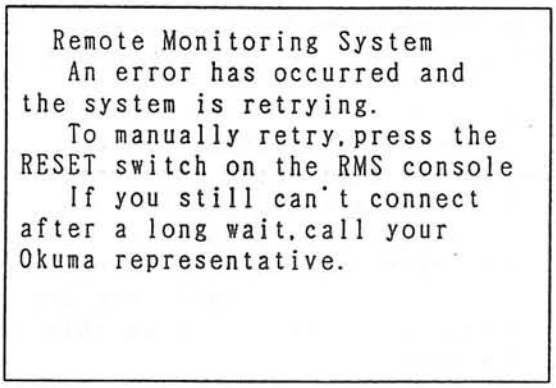


Fig. 24-10 Communications Unit Error Message

4-2. Other Errors

- The CRT screen does not display anything when the control power is turned on.
- The display is flickering or swaying, hard to figure out.
  In addition to the operation panel and the CRP board, the communications unit might be faulty.

SECTION 25 RELATIVE ACTUAL POSITION DISPLAY FUNCTION

1. Overview

This function displays relative actual position data of each axis with regard to the reference point on the RELATIVE ACT POSIT screen in the automatic, MDI, or manual operation mode individually.

The reference point can be set at any position for each basic and additional axis independently.

Similarly, relative actual position data can be displayed for each basic and additional axis independently.

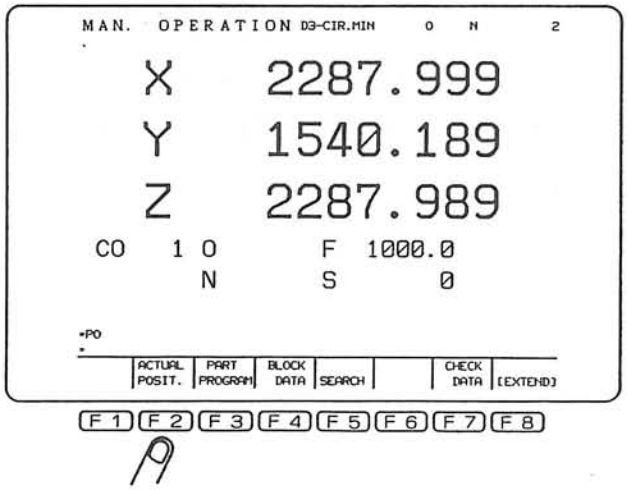
2. Operation

2-1. Selecting the RELATIVE ACT POSIT Screen

Follow the procedure below to display the RELATIVE ACT POSIT screen.

1) Press the function key [F2] (ACTUAL POSIT.) in the automatic, MDI, or manual operation mode.

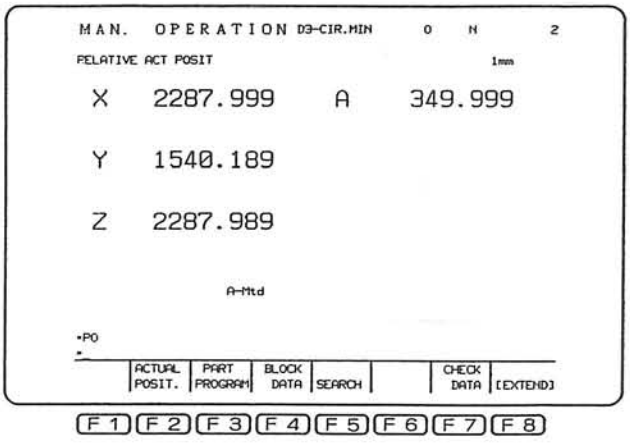
Operation Mode Screen



2) Press the page key (⇧P or ⇩P) repeatedly until the RELATIVE ACT POSIT screen is displayed.

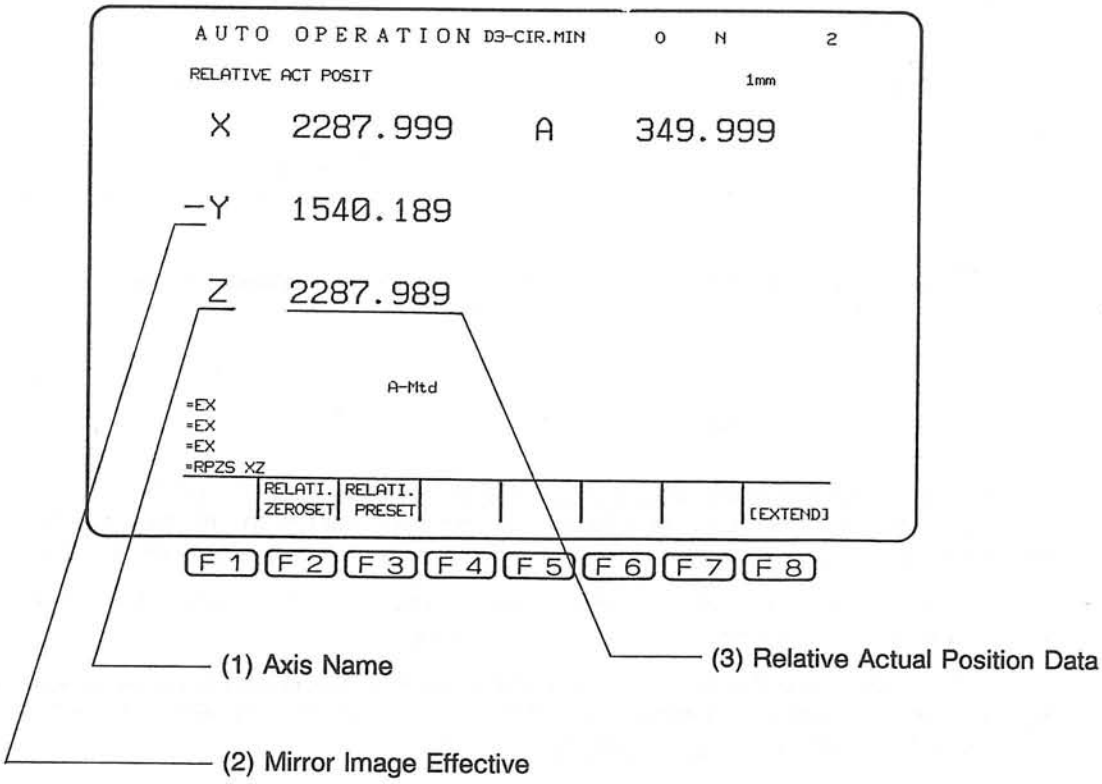
(⇧P or ⇩P)

RELATIVE ACT POSIT Screen



2-2. RELATIVE ACT POSIT Screen

The RELATIVE ACT POSIT screen looks like as follows.



(1) Axis Name

Indicates a basic axis name (X, Y, or Z) or an additional axis name (A, B, C, etc.).

An additional axis name is displayed only when an additional axis has been selected.

(2) Mirror Image Effective

When mirror image is effective, a minus sign "−" is placed preceding the axis name.

When mirror image is not effective, no sign is placed.

(3) Relative Actual Position Data

Relative actual position data calculated using the following equation is displayed in the selected unit system.

Relative actual position data of an additional axis is displayed only when an additional axis is selected.

Relative actual position data = (coordinate value output from the encoder*1) − (reference position*2) − (tool length offset value) − (machine zero point)

*1 . . . . . Whether or not the manual shift amount is added can be set at NC optional parameter (bit) No. 5, bit 7.

*2 . . . . . The reference position is the zero point (in the machine coordinate system) for the relative actual position. For the procedure to set the reference position, refer to 3. "Reference Position Setting".

(4) Reference Position

The reference position is the zero point in the machine coordinate system and is used to display the relative actual position, or, in their words, the zero point in the relative coordinate system. The reference position is calculated form the equation below and displayed for each axis in the selected unit system.
Relative actual position data of an additional axis is displayed only when an additional axis is selected.

Reference position = (output from position encoder *1) – (actual position value *2) – (tool length offset value) – (machine zero point)

*1 . . . . . Like the actual position display, it is possible to select whether or not the manual shift amount is included in the output form the position encoder by setting data at NC optional parameter (bit) No. 5, bit 7.

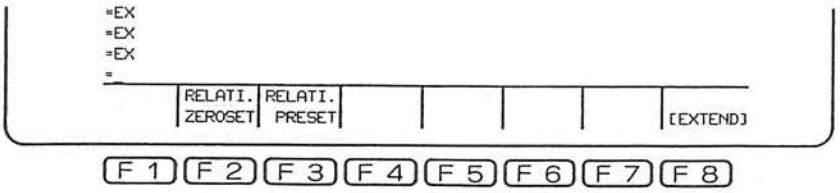
*2 . . . . . Where the actual position is set in the coordinate system is input.
Refer to 3, Reference Position setting.

2-3. Reference Position Setting

The reference position is the zero point (in the machine coordinate system) which is used to display the relative actual position, or, in other words, the zero point in the relative coordinate system. The reference position can be obtained by setting the coordinate value of the actual position.

The reference position can be set in two different manners : by setting the actual position at "0" and by setting the actual position at a desired position.

To set the reference position, press the function key [F8] (EXTEND) in the automatic, MDI, or manual operation mode repeatedly until functions "RELATI, ZEROSET" and "RELATI, PRESET" are assigned to function keys [F2] and [F3], respectively.

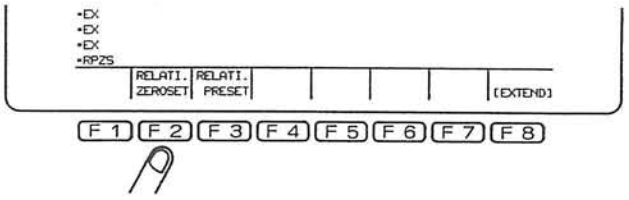


2-3-1. Setting the Actual Position at "0"

Follow the procedure below when setting the actual position at "0" in the relative coordinate system.

1) Press the function key [F2] (RELATI.ZEROSET) in the automatic, MDI, or manual operation mode.

   The prompt "=RPZS" will be displayed on the console line of the CRT screen.

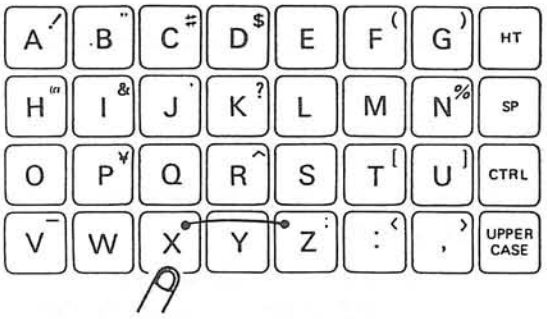


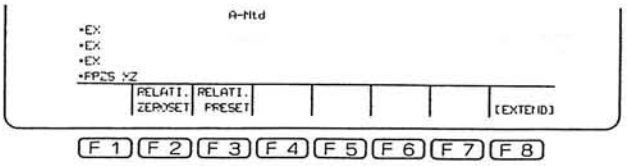
2) Key in axis address(es) for which "0" is set through the keyboard. When no axis address has been keyed in, "0" is set for all axes.

   Example: To set "0" for X and Z axes

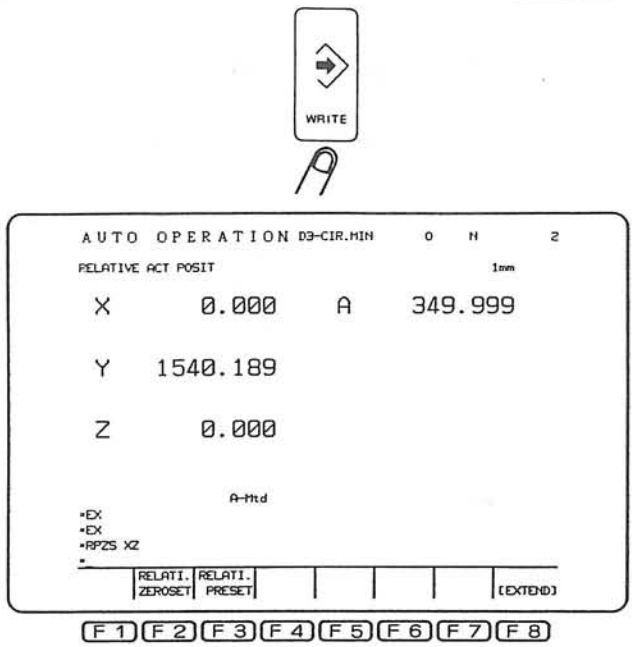
   =RPZS XZ

   Key in axis addresses





3) Press the **WRITE** key.



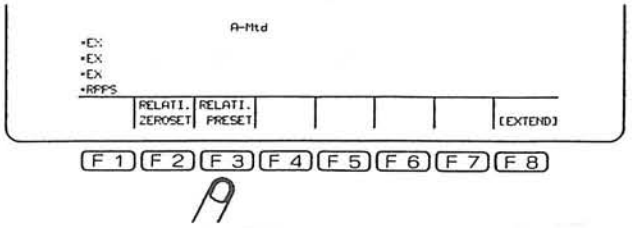
The reference position with which the actual position of the designated axis is "0" is obtained and relative actual position data of the designated axis will change to "0".

2-3-2. Setting the Actual Position at a Desired Position

Follow the procedure below when setting the actual position at a desired position in the relative coordinate system.

1) Press the function key [F3] (RELATI.PRESET) in the automatic, MDI, or manual operation mode.

   The prompt "=RPPS" will be displayed on the console line of the CRT screen.



⇩

2) Key in axis address(es) and numerical value for which the actual position is set at a desired position through the keyboard. When no axis address has been keyed in, the actual position is set at a desired position for each axis.

Example 1: To set the actual position of X and Z axes at 200 and 300, respectively

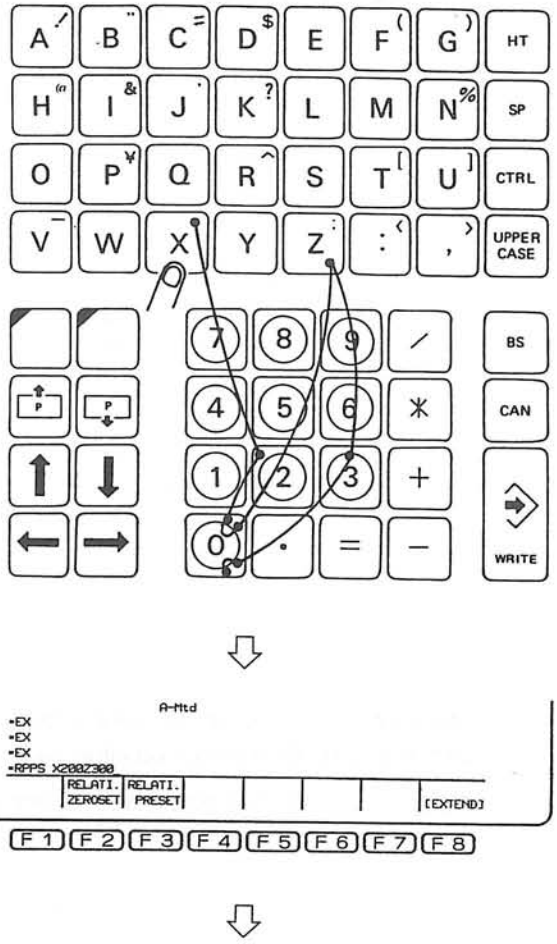
=RPPS X200Z300

Key in axis addresses and numerical values.

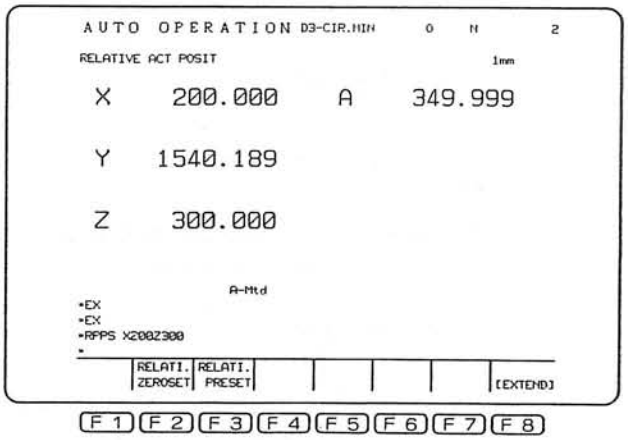
Example 2: To set the actual position of all axes at 100

=RPPS 100

Key in a numerical value without specifying axis addresses.



3) Press the **WRITE** key.



The reference position with which the actual position of the designated axis is a desired position is obtained and relative actual position data of the designated axis will change to a desired position.

Note: *Pressing the **WRITE** key without keying in address(es) and numerical value(s) does not set anything.*

2-3-3. Data Setting Range and Restrictions

(1) Data is input in the unit system (metric or inch) employed for machine operation and the decimal point position is fixed. For example, when "1" has been input while the 0.001 mm unit system is selected, it is recognized as 1 mm. The same rule also applies to the inch system.

(2) Data can be set within the following range.

- For linear axes: −99999.999 mm to +99999.999 mm

  (when the inch system is selected, the entered value is converted into a metric value and checked if it is within the above range.)

- For rotary axes: −99999.999° to +99999.999° (for the 0.001° unit system)

  −9999.9999° to +9999.9999° (for the 0.0001° unit system)

3. Precautions

(1) Data is input in the unit system (metric or inch) employed for machine operation and the decimal point position is fixed. (For example, when "1" has been input while the 0.001 mm unit system is selected, it is recognized as 1 mm.)

(2) When changing the reference position of all axes including rotary axes, the entered value is interpreted as length and degree.

(3) The reference position cannot be set for an indexable axis. In this case, the actual position data is displayed on the RELATIVE ACT POSIT screen.

However, when axis designation was not made with reference position setting, "0" is set at the indexable axis, causing no error.

(4) When the power is turned off, reference position data becomes "0" since it is not backed up by turning off of the power. (The machine zero point is employed as the reference position.)

However, when actual position data in the work coordinate system is rounded (parameter (bit) No. 2 bit 1 is ON) with the multi-turn rotary table specification, reference position data is calculated in reverse order. Therefore, when the work zero point is other than "0", a value other than "0" is set as reference position data.

(5) Work coordinate values do not change when the reference position has been changed.

(6) When the relative actual position value is smaller than -99999.999 mm (−9999.9999 inch for the inch system), "− OVERFLOW" will be displayed on the CRT screen.

When the relative actual position value is larger than +99999.999 mm (+9999.9999 inch for the inch system), "+ OVERFLOW" will be displayed on the CRT screen.

(7) The display of the relative actual position of a rotary axis (rotary table) varies depending on the rotary axis specification.

1) Rotary table and indexable axis specification

The relative actual position is displayed within 0° and 360°. The reference position is also displayed within 0° and 360°.

2) Rotary axis with limits and multi-turn rotary table

The relative actual position obtained using the equation on page 23 is displayed as it is. With the multi-turn rotary table, whether or not relative actual position data is expressed within 0° and 360° when the NC is reset can be set at NC optional parameter No. 2, bit 1.

Note: *When the additional axis is removed with the removable axis specification, "OVERFLOW" will be displayed as relative actual position data.*

4. Parameters

<NC Optional Parameter (Bit)>

(1) No. 2, bit 1

This parameter sets whether or not the work coordinate value is rounded when machine operation is reset with the multi-turn rotary table specification.

<Setting>

1 : Rounded

0 : Not rounded

(2) No. 5, bit 7

This parameter sets whether or not the shift amount due to manual interrupt or pulse handle overlap is added to actual position data and graphic display data.

<Setting>

1 : Added

0 : Not added

5. Error

2645 Relative actual position set axis name

During zero setting or presetting of the relative actual position, alphabetic character(s) other than axis addresses are used. Or an attempt is made to set an indexable axis.

Character-string	:	None
Code	:	Hexadecimal number of the ASCII code of the axis on which the error has been generated
Measures to Take	:	Specify correct axis address(es).

LIST OF PUBLICATIONS

Publication No.	Date	Edition
3295-E	November 1989	1st
3295-E-R1	September 1991	3rd
3295-E-R2	December 1991	4th
3295-E-R3	June 1992	5th
3295-E-R4	November 1992	6th B

This manual may be at variance with the actual product due to specification or design changes.
Please also note that specifications are subject to change without notice.
If you require clarification or further explanation of any point in this manual, please contact your OKUMA representative.