YASNAC i80M
INSTRUCTIONS

CNC SYSTEM FOR MACHINING CENTERS

Upon receipt of the product and prior to initial operation, read these instructions thoroughly, and retain for future reference.

YASKAWA

MANUAL NO. TOE-C843-11.30F

General Precautions

- Some drawings in this manual are shown with the protective cover or shields removed. in order to describe the detail with more clarity. Make sure all covers and shields are replaced before operating this product, and operate it in accordance with the directions in the manual.
- The figures and photographs in this manual show a representative product for reference purposes and may differ from the product actually delivered to you.
- This manual may be modified when necessary because of improvement of the product, modification, or changes in specifications.
  Such modification is made as a revision by renewing the manual No.
- To order a copy of this manual, if your copy has been damaged ro lost, contact your Yaskawa representative listed on the last page stating the manual No. on the front page.
- If any of the nameplates affixed to the product become damaged or illegible, please send these nameplates to your Yaskawa representative.
- Yaskawa is not responsible for any modification of the product made by the user since that will void our guarantee.

NOTES FOR SAFE OPERATION

Read this operator's manual thoroughly before installation, operation, maintenance or inspection of the YASNAC i80M. In this manual, the NOTES FOR SAFE OPERATION are classified as "WARNING" or "CAUTION".

⚠ WARNING	Indicates a potentially hazardous situation which, if not avoided, could result in death or serious injury to personnel.
⚠ CAUTION	Indicates a potentially hazardous situation which, if not avoided, may result in minor or moderate injury to personnel and damage to equipment. It may also be used to alert against unsafe practice.

Even items described in **⚠ CAUTION** may result in a vital accident in some situations. In either case, follow these important items.

Please note that symbol mark used to indicate caution differs between ISO and JIS.

ISO	JIS
⚠	◇

In this manual, symbol mark stipulated by ISO is used.

On products, caution symbol marks of ISO and JIS are used in labels.
Please follow the same safety instructions concerning caution.

STORAGE PRECAUTIONS

⚠ CAUTION

- **Do not store the product in locations subject to rain, water droplets, or harmful gases or liquids.**
  Failure to observe this caution may result in product failure.
- **Select a storage area indoors that is clean and meets the following temperature and humidity requirements.**
  Failure to observe this caution may result in product failure.
  Ambient temperature : −20℃ to 60℃
  Relative humidity : 10% to 90%

CAUTIONS ON USE AND OPERATION

⚠ WARNING

- **Do not touch any unit, terminals, etc., while the power is ON.**
  Failure to observe this warning may lead to electric shock or device malfunction.
- **Immediately after switching the power OFF, the product retains some electric charge. Do not touch any parts which are live when the power is ON for 5 minutes after switching the power OFF.**
  Failure to observe this warning may lead to electric shock or device malfunction.
- **Do not damage cables, subject them to excessive stress, or pinch them.**
  Excessive load on cables may cause electric shock.
- **When the unit is turned ON, never touch its rotating parts.**
  Failure to observe this warning may result in personal injury.
- **Never modify the product.**
  Failure to observe this warning may result in electric shock, fire, or product failure.

⚠ CAUTION

- **Before carrying out cutting operation with a new program, confirm safety by performing single block operation and dry run operation.**
  If this check operation is not performed, unexpected operation may be performed due to mis-setting of the amount of offset, leading to tool damage due to interference, and resulting accidents involving injuries to personnel.
- **The end user must not change parameters relating to machine accuracy, travel axis control and spindle axis control.**
  The NC parameters are set to the optimum values for each machine, and changing them could therefore result in unexpected operation. This could cause tool damage due to interference, and resulting accidents involving injuries to personnel.
  Reset the NC after any manual intervention.
- **Strictly observe the cautions in the user's manual when using programming functions.**
  Ignoring these cautions could lead to accidents involving injuries to personnel and malfunctions.
- **Use the product with the "System Number Switch" of the CPU set to "0" .**
  Use while set to another number may lead to malfunction.
- **Wait at least 2 seconds after turning the power OFF before turning it ON again.**
  Failure to observe this caution may lead to malfunction.

⚠ CAUTION

- **Use the product in an environment with the following characteristics.**

  Using it in an environment in which it is subject to high temperatures, high humidity, dust, corrosive gases, vibration or impacts may cause fire, electric shock or malfunction.

  - Free from gases or vapors that create a potentially explosive atmosphere.
  - Free from oil, organic solvents, etc.
  - Relative humidity in the range 10 to 90% RH, with no condensation.
  - Ambient temperature in the 0℃ to 45℃ with no freezing.
    (Installation site must not be exposed to direct sunlight, must be distanced from heat generating devices, and must be indoors.)
  - Vibration not exceeding 4.9m/S^2.

- **Do not let foreign matter such as electric wire scrap enter the unit.**

  Failure to observe this caution may result in fire, product failure or malfunction.

- **Do not restart automatic operation after stopping automatic operation and then performing "tool selection" in manual operation or "1 line MDI" operation.**

  The reason for this is that the "tool selection" operation may cause the coordinate system to be changed, leading to unexpected operation if automatic operation were restarted. This could cause tool damage due to interference, and resulting accidents involving injuries to personnel.

  Reset the NC after any manual intervention.

- **After stopping automatic operation and performing a manual intervention, do not restart automatic operation without resetting first.**

  If automatic operation is started with the "mirror image" or "manual absolute" function in effect, unexpected operation may be performed. This could cause tool damage due to interference, and resulting accidents involving injuries to personnel.

  Reset the NC after any manual intervention.

⚠ CAUTION

- **When an alarm occurs, eliminate its cause and confirm safety before resetting it.**
  Failure to observe this caution could result in malfunction. .
- **Be sure to check the following points on completing maintenance and inspection work.**
  - Check that all fastening bolts are tightened.
  - Check that no tools or other objects have been left inside the control panel.
  - Check that the control panel door is closed properly.

  Failure to carry out these checks may lead to electric shock, injuries, fire, and malfunction.
- **For details on trouble relating to the machine-related sequence, refer to the manual issued by the machine tool builder.**
- **Never attempt to disassemble or modify units or devices inside the NC unit.**
  Failure to observe this caution may lead to fire, product failure, or malfunction.
- **Do not change the set values of the devices, variable resistors, etc., in the control panel.**
  Failure to observe this caution may lead to fire, product failure, and malfunction.

PRECAUTIONS RELATING TO MAINTENANCE AND INSPECTION

⚠ WARNING

- **Always turn the power OFF (including the primary power supply) before carrying out daily inspection.**
  Carrying out the inspection with the power ON may lead to electric shock.
- **Wait 5 minutes after turning the power (including the primary power supply) OFF before removing or replacing any unit or part.**
  Failure to observe this warning may lead to electric shock and product failure.
- **Be sure to turn the power OFF before replacing the battery.**
  Failure to observe this warning may lead to electric shock and product failure.

⚠ CAUTION

- **To prevent personnel other than those involved in maintenance and inspection work from turning ON the power while maintenance and inspection is in progress, place signs stating "Do not turn the power on" or words to that effect at the primary power supplies of related control panels and other relevant locations.**
  Failure to observe this caution may lead to electric shock.
- **Replace fuses and batteries with the designated products.**
  Failure to observe this caution may result in fire or product failure.
- **Electronic devices such as C MOS ICs are rsed on the control boards. If you touch them with your bare fingers the static electrical charge in your body could destroy them; care must be taken when handling these devices. Before handling these devices for maintenance purposes, first discharge the static electricity in your body by touching a grounded metal device.**
  Failure to observe this caution could lead to injuries and product failure.
- **Do not install or remove boards, wiring, connectors, etc., while the power is ON.**
  Failure to observe this caution could lead to electric shock, product failure, and malfunction.

CONTENTS

CONTENTS

CONTENTS (Cont'd)

CONTENTS (Cont'd)

Page

CONTENTS (Cont'd)

INDEX

INDEX (Cont'd)

INDEX (Cont'd)

SECTION 1

PREFACE

This section describes the following item:
- Basic configuration of NC machine tool system
- Basic operation

CONTENTS PAGE

NOTE Items marked with ✣ indicate options.

1.1 BASIC CONFIGURATION OF NC MACHINE TOOL SYSTEM

YASNAC i80M allows parallel processing of multiple CPUs organically by applying digital servo drives and actuators, resulting in high speed and high precision processing.

Fig. 1.1 shows the basic configuration of the NC machine tool system. Each control part and control processing part is incorporated in one or several CPUs. These parts execute parallel processing except for requirements of synchronization with specified software and/or hardware.

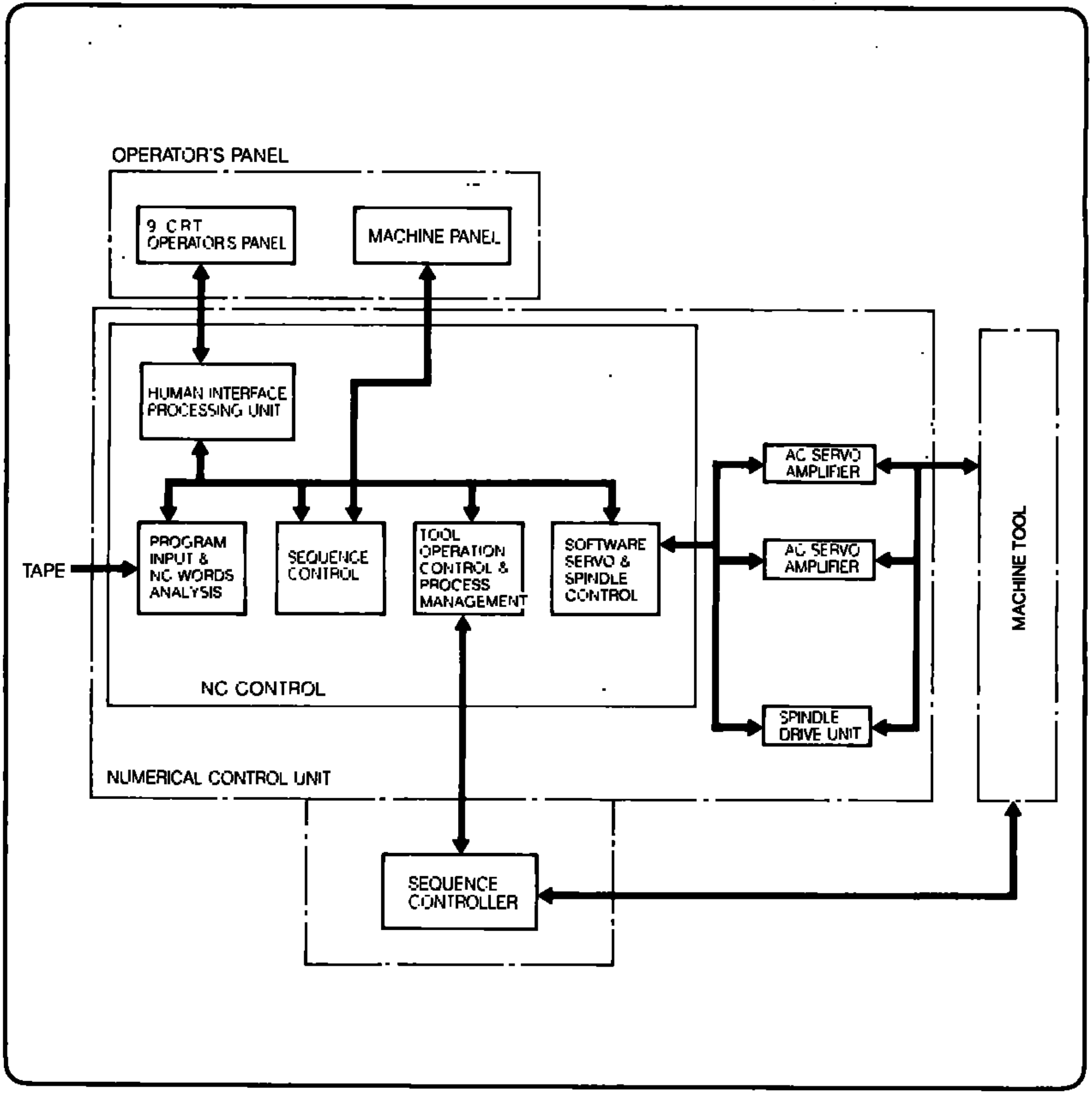


Fig. 1.1 Basic Configuration of NC Machine Tool System with YASNAC i80M

1.2 BASIC OPERATION

To process parts with NC machine tools, it will first be necessary to prepare a processing program, input this in the NC, set the work coordinates required to execute this program (run the machine), make the necessary preparations such as setting tool offset and editing, and then depressing the START button on the machine operation panel to proceed with the processing work.

A general flow of this system is shown in Fig. 1. 2.

A general configuration of this manual is shown on the right side of this flow. Explanation is given in SECTION 2, "PROGRAMMING" in relation to the functions of the control unit, explanation of the instruction words usable in the program and also the method of use.

Firm description is also given in SECTION 3, "DESCRIPTION OF OPERATION" on how to operate the machine tool system and the operating method of the operator panel with 9" CRT and the machine operating panel (example for explanation).

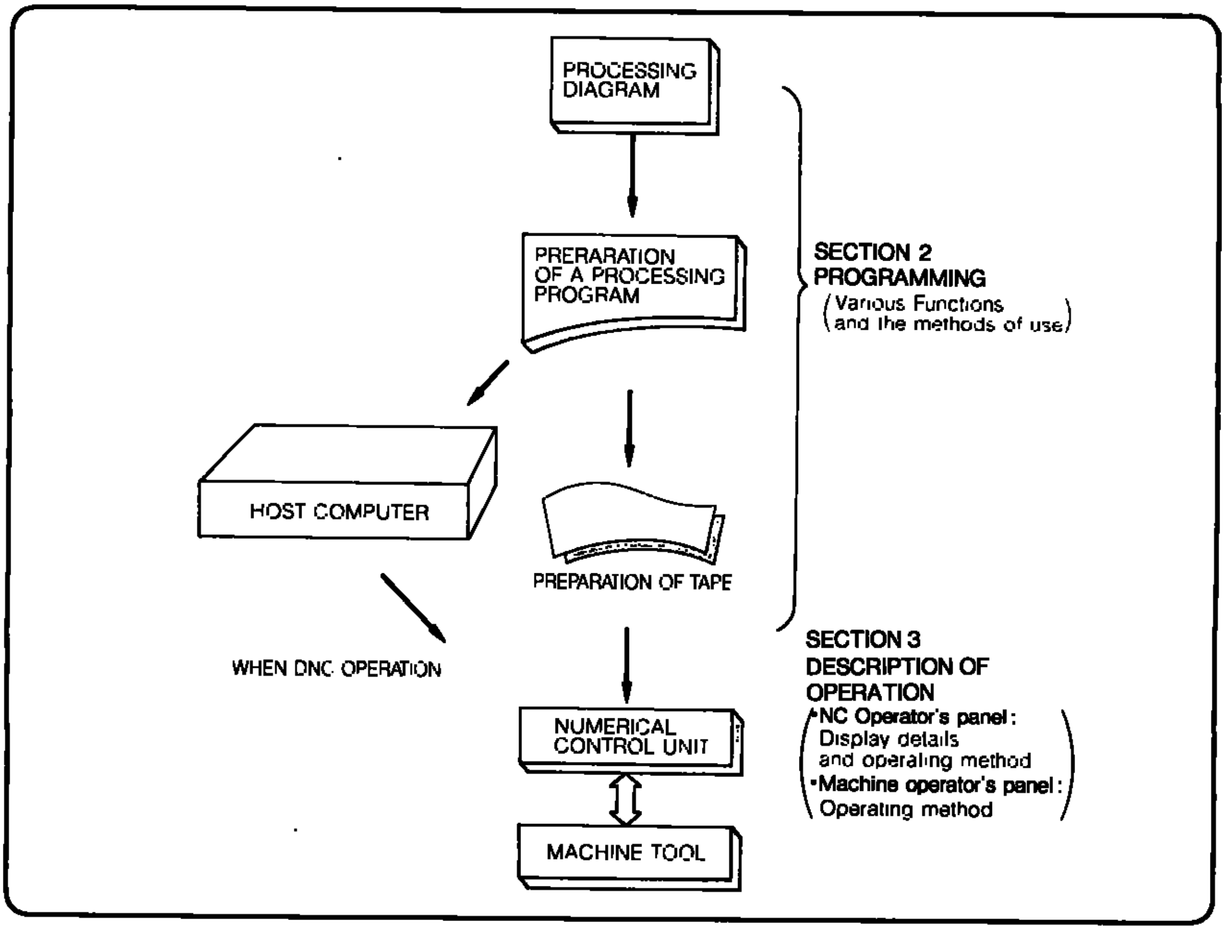


Fig. 1.2 General Flow of the System

SECTION 2

PROGRAMMING

This section describes programming and functions of the control axes.

CONTENTS

NOTE Items marked with* indicate options.

CONTENTS PAGE

2.1 CONTROL AXES

2.1.1 Control Axes Names

YASNAC can control axes shown in Table 2. 1. 1.

Table 2.1.1

Control Axes	Axes Name	Description	Reference
Main Axes	X, Y, Z	Position or distance in X, Y or Z coordinate direction.	Par. 2.1.2.1
4th and 5th axes*	A, B, C (for rotary motion) U, V, W (for parallel motion)	Commands in the directions of the 4th and 5th axes.	Par5. 2.1.2.2 and 2.1.2.3

- **ROTARY AXES (A, B or C Axis)**

Table 2.1.2

Rotary axes	Definition
A-axis	Rotary axis parallel to X-axis
B-axis	Rotary axis parallel to Y-axis
C-axis	Rotary axis parallel to Z-axis

NOTE **In this manual, rotary axes either A, B or C are indicated by B-axis.**

(1) The unit of output increment and input increment for rotary axes is "deg." instead of "mm" used with linear axes. In all other respects, the treatment is the same as those in mm. (Metric system)

(2) Even when inch system is selected by parameter, the values for the B-axis remain "deg." units.

- **LINEAR AXES (U, V, or W Axis)**

Table 2.1.3

Linear axis	Definition
U-axis	Linear axis parallel to X-axis
V-axis	Linear axis parallel to Y-axis
W-axis	Linear axis parallel to Z-axis

NOTE In this manual, linear axes either U, V or W are indicated by V-axis.

(1) The unit output increment and input increment for linear axis is the same as the other linear axes, X, Y and Z. No discrimination is necessary.

(2) When inch system is selected by parameter, input values must be in inches for V-axis.

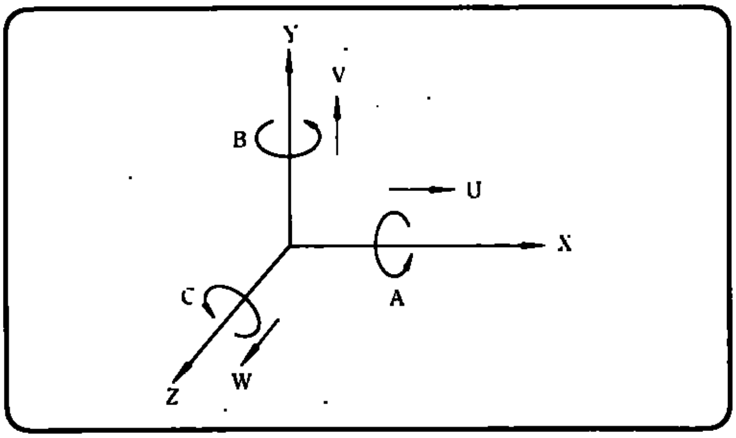


Fig. 2.1 Axes in Right-hand Coordinate System

2.1.2 Simultaneously Controllable Axes of Three-axis Control

Table 2.1.4 Simultaneously Controllable Axes of Three-axis Control

	Simultaneously controllable axes
Positioning G00	X, Y and Z axes
Linear interpolation G01	X, Y and Z axes
Circular interpolation G02, G03	Two axes: XY, YZ or ZX (see Note 1.)
*Circle cutting G12, G13	Two axes: X and Y
*Helical interpolation G02, G03	Circle in XY-plane and linear feed in Z-axis direction. Refer to par. 2.4.4 ; Helical Interpolation.
Manual control	Simultaneous control of X, Y and Z (see Note 2.)

NOTE

1. Circular arc plane is determined according to the currently effective G codes (G17 to G19) for plane designation.
   For details, refer to par. 2.4.3, "Circular Interpolation (G02, G03)," on page 41.
2. Manual pulse generator provides simultaneous one- or three-axis control.

2.1.2.2 Simultaneously Controllable Axes of Four-axis Control ※

- 4th (α) axis control

An additional 4th axis can be incorporated. In this manual, the 4th axis is referred to as α -axis, and represents any of the 6 axes, A, B, C, U, V or W.

Table 2.1.5 Simultaneously Controllable Axes of Four-axis Control

		Simultaneously controllable axes
Positioning	G00	X, Y, Z and α axes
Linear Interpolation	G01	X, Y, Z and α axes
Circular Interpolation	G02 G03	Two axes, XY, YZ, ZX, Xα, Yα, or Zα (See Note 1.)
※Circular Cutting	G12 G13	Two axes, X and Y
※Helical Interpolation	G02 G03	Three axes: Circle in XY plane and linear feed in Z-axis direction. Refer to par. 2.4.4Helical Interpolation
Manual Control		Four axes: X, Y, Z and α (See Note 2.)

NOTE

1. Circular interpolation is possible only when the 4th axis is a linear axis of U, W or V.
   Circular arc plane is determined according to the currently effective G codes for plane designation (G17 to G19). For details, refer to par. 2.4.3, "Circular Interpolation (G02, G03)."
2. Manual pulse generator is of 1-axis or 3-axis.

2.1.2.3 Simultaneously Controllable Axes of Five-axis Control ※

- **5th (β) axis control**

Additional 4th and 5th axis can be incorporated. In this manual, 4th axis is referred as α axis, 5th axis as β axis, and represents any of the six axes A, B, C, U, V or W.

Table 2.1.6 Simultaneously Controllable Axes of Five-axis Control

		Simultaneously controllable axes
Positioning	G00	X, Y, Z, α and β axes
Linear Interpolation	G01	X, Y, Z, α and β axes
Circular Interpolation	G02 G03	Two axes XY, XZ, ZX, $X\alpha$, $Y\alpha$, $Z\alpha$, $X\beta$, $Y\beta$, or $Z\beta$ (See Note 1.)
※Circular Cutting	G12 G13	Two axes, X and Y
※Helical Interpolation	G02 G03	Three axes: Circle in XY plane and linear feed in Z-axis direction. Refer to par. 2.4.4 Helical interpolation
Manual Control		Five axes: X, Y, Z, α and β axes (See Note 2)

NOTE

1. Circular interpolation is possible only when the 4th and 5th axes are linear axis. Circular arc plane is determined according to the currently effective G codes for plane designation (G17 to G19). For details, refer to par. 2.4.3 "Circular Interpolation (G02, G03).
2. Manual pulse generator is of 1-axis or 3-axis.

2.1.3 Least Input Increment and Least Output Increment

- **Least input increment**

The minimum input units that can be commanded by punched tape or MDI are shown in Tables 2.1.7, 2.1.8 and 2.1.9.

Table 2.1.7 Least Input Increment (Standard)

	Linear Axis	Rotary Axis*
Metric Input	0.001 mm	0.001 deg
Inch Input	0.0001 in.	0.001 deg

Table 2.1.8 Least Input Increment (Sub Microns)

	Linear Axis	Rotary Axis*
Metric Input	0.0001 mm	0.001 deg
Inch Input	0.00001 in	0.001 deg

Table 2.1.9 Least Input Increment (Sub Sub-microns)

	Linear Axis	Rotary Axis*
Metric Input	0.00001 mm	0.001 deg
Inch Input	0.000001 in	0.001 deg

NOTE Selection of mm-input or inch-input is set by Pm0007D0.

- **Least output increment**

Least output increment is the minimum unit of tool motion and mm-output or inch-output depends on an option.

Table 2.1.10 Least Output Increment (Standard)

	Linear Axis	Rotary Axis*
Metric Output	0.001 mm	0.001 deg
Inch Output	0.0001 in.	0.001 deg

Table 2.1.11 Least Output Increment (Sub Microns)

	Linear Axis	Rotary Axis※
Metric Output	0.0001 mm	0.001 deg
Inch Output	0.0001 in	0.001 deg

Table 2.1.12 Least Output Increment (Sub Sub-microns)

	Linear Axis	Rotary Axis※
Metric Output	0.00001 mm	0.001 deg
Inch Output	0.00001 in	0.001 deg

2.1.4 Maximum Move Command Values

Maximum move command values are shown below. The values in Tables 2.1.13, 2.1.14, and 2.1.15 apply not only to move command addresses X, Y, Z, α and β but also to distance command addresses I, J, K, R, Q.

Table 2.1.13 Maximum Move Command Values (Standard)

		Linear Axis	Rotary Axis*
Metric Output	Metric Input	±999999.999 mm	±999999.999 deg
	Inch Input	±39370.0787 in	±999999.999 deg
Inch Output	Metric Input	±999999.999 mm	±999999.999 deg
	Inch Input	±99999.9999 in	±999999.999 deg

Table 2.1.14 Maximum Move Command Values (Sub Microns)

		Linear Axis	Rotary Axis*
Metric Output	Metric input	±99999.9999 mm	±999999.999 deg
	Inch input	±3937.00787 in	±999999.999 deg

Table 2.1.15 Maximum Move Command Values (Sub Sub-microns)

		Linear Axis	Rotary Axis*
Metric Output	Metric input	±9999.99999 mm	±999999.999 deg
	Inch input	±393.700787 in	±999999.999 deg

NOTE If a greater value than the maximum is specified, proper operation is not guaranteed.

1. In incremental programming, input values must not exceed the maximum command value.
2. In absolute programming, move amount of each axis must not exceed the maximum command value.

The cumulative value must not exceed the values shown below.

Table 2.1.16 Maximum Cumulative Values (Standard)

	Linear Axis	Rotary Axis※
Metric input	±999999.999 mm	±999999.999 deg
Inch input	±99999.9999 in.	±999999.999 deg

Table 2.1.17 Maximum Cumulative Values (Sub Microns)

	Linear Axis	Rotary Axis※
Metric input	±99999.9999 mm	±999999.999 deg

Table 2.1.18 Maximum Cumulative Values (Sub Sub-microns)

	Linear Axis	Rotary Axis※
Metric input	±9999.99999 mm	±999999.999 deg

NOTE Listed input values do not depend on metric/inch output system.

2.2 PROGRAMMING

2.2.1 Process Sheet

Programs are first drafted on process sheets.

Process sheets should be easy to read and to make corrections, and should be designed and prepared by the user in conformity with the specifications of the NC.

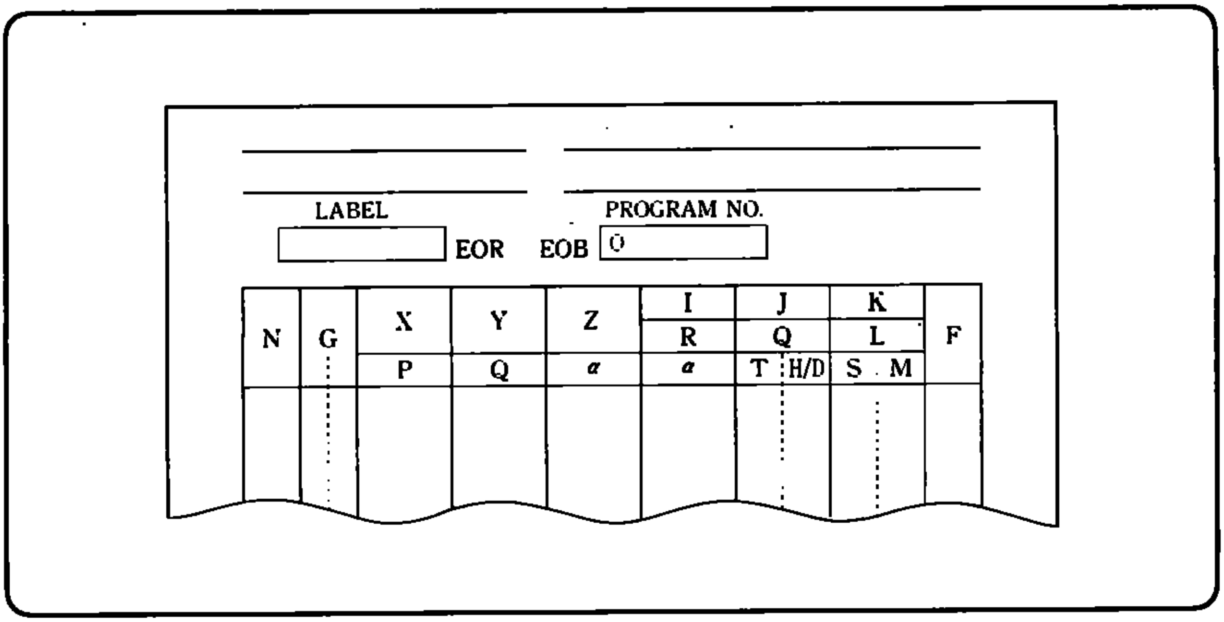


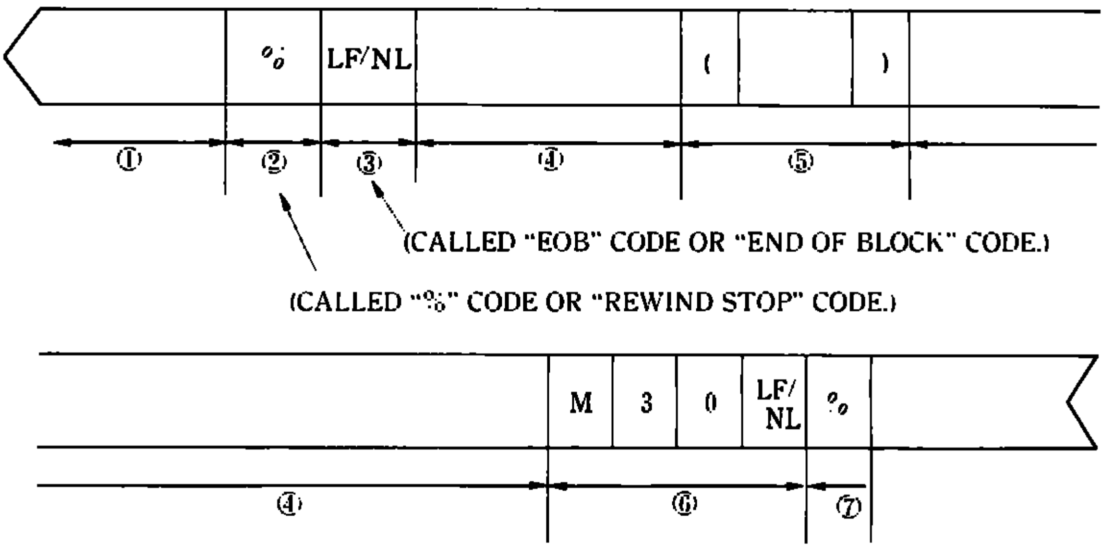
Fig. 2.2.1 Example of Process Sheet

2.2.2 Programming Format

Example of part program shows in Fig. 2.2.2 and 2.2.3

For details, see the following paragraphs.

①Label ——————————— 2.2.2.1
②Tape start ——————————— 2.2.2.2
③Program start ——————————— 2.2.2.3
④Program part ——————————— 2.2.2.4
⑤Comment part ——————————— 2.2.2.5
⑥Program end ——————————— 2.2.2.3
⑦Tape end ——————————— 2.2.2.2

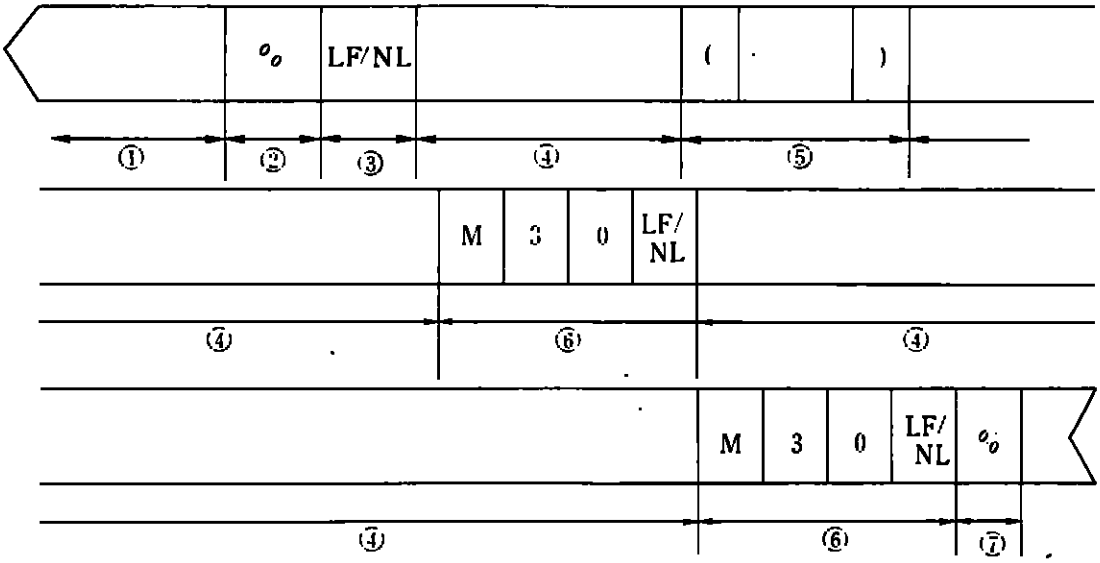


(CALLED "EOB" CODE OR "END OF BLOCK" CODE.)

(CALLED "%" CODE OR "REWIND STOP" CODE.)

NOTE M02 or M99 can be used for the program end instead of M30.
Whether the above-mentioned M code is to be the program end depends on parameter pm3005 D3 setting.

Fig. 2.2.2 Tape with a Main Program (ISO code)



NOTE Even if parameter pm3005 D3 is to determine whether an M code is to be the program end, it must be disregarded. An M code is not to be the program end at any time.

Fig. 2.2.3 Tape with a Multi-program (ISO code)

2.2.2.1 Label part

To facilitate tape classification or handling, an arbitrary "label" may be written in the first part of punched tape. The label skip function allows a section of data up to the first EOB code to be skipped, so that off-specification addresses or function codes can be used. In addition, parity-disregarded, modified code can also be used.

The label skip function is valid in the following cases in which case "LSK" is indicated on the CRT screen.

(1) When the power is turned ON
(2) When the equipment is reset

When label skip is valid, information up to the first EOB code on punched tape is disregarded.

2.2.2.2 Tape start／tape end

Punch the same code in tape start and tape end.

Table 2.2.1

EIA	ISO	Meaning
ER	%	Tape start/tape eno

(1) The ER code (rewind stop code) following the tape start label indicates the stop point when tape is rewound by a tape rewind command.

(2) The ER code for tape stop indicates the stop point when multiple part programs are stored in the NC memory.

2.2.2.3 Program start／program end

(1) Program start

The following code is punched to declare the beginning of the program section. This is required to clear the label skip.

Table 2.2.2

EIA	ISO	Meaning
CR	LF/NL	Program start

(2) Program end

The following code is punched to declare the end of the program section.

Table 2.2.3

EIA	ISO	Meaning
M02CR	M02LF/NL	Program end
M30CR	M30LF/NL	Program end & rewind
M99CR	M99LF/NL	Sub-program end

NOTE

1. When M02CR or M02LF/NL, or M30CR or M30LF/NL is executed, the equipment may or may not be reset or rewound depending on equipment specifications. Refer to the command manuals issued by your machine tool manufacturer.
2. When multiple part programs are stored in the NC memory, control may move to the next part program after reading the program end code shown above. This occurs when part programs are entered by total input.
3. If M02 or M30 is not present at the end of the program section and ER or LF/NL is immediately executed, the NC machine may be reset.

2.2.2.4 Program part

A punched section from the program start to the program end is called the program section. The program section consists of several blocks, and each block is comprised of several words. Each block is separated by the EOB code (;).

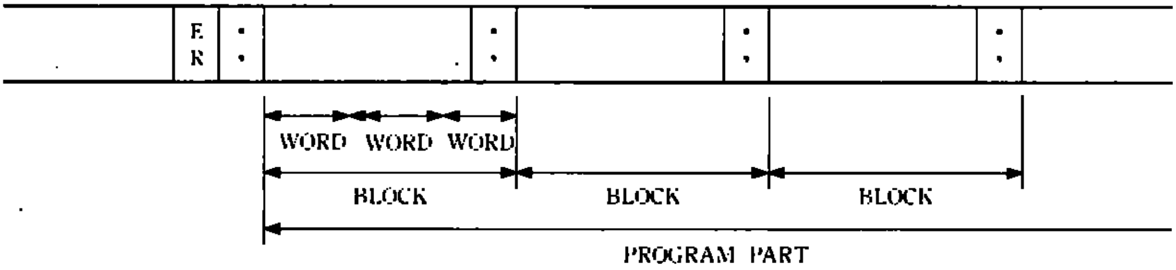


Fig. 2.2.4 Program Construction

- **Program number**

(1) By adding a program No. immediately after the program start code, it is possible to discriminate your program from other programs.

(2) A program No. can be specified by numbers of up to five digits following address O. Up to 99 program Nos. can be entered for the equipment. However, the maximum number of program Nos. can be increased to 299 or 999 by adding options.

- **Sequence number**

(1) Integers consisting of up to 5 digits may be written following an address character N as sequence numbers.

(2) Generally, sequential numbers are convenient as sequence numbers. Sequence numbers are reference numbers for blocks, and do not have any influence on the meaning or sequence of machining processes. Therefore, they may be sequential, non-sequential, or duplicated numbers. Not using any sequence number is also possible.

(3) When searching for sequence numbers, be sure to search or specify program numbers beforehand.

NOTE

1. When 6 or more digits are written as a sequence number, only the digits up to the 5th from the trailing end are effective.
2. When two or more blocks have the same sequence number, only one is retrieved and read, and no more searching is performed.
3. Blocks without sequence numbers can also be searched for with respect to the address data contained in the blocks.
4. Use four digits to specify a sequence number for G25 or M99.

- **Word**

A block is a collection of words. A word consists of an address (alphabet) and a numeral of several digits that follows the address.

G	02	Z	−10	(Includes a sign and decimal point)
Address^{*1}	Numeral^{*2}	Address^{*1}	Numeral^{*2}	
Word		Word		

^{*1} See (1). ^{*2} See (3).

(1) Address and function characters

Table 2.2.4 Address Characters

Address Characters	Meanings	B : Basic O : Option
A	Additional rotary axis parallel to X-axis	O
B	Additional rotary axis parallel to Y-axis	O
C	Additional rotary axis parallel to Z-axis	O
D	Tool radius offset number	B, O
E	User macro character, Feed of canned cycle	O
F	Feedrate	B
G	Preparatory function	B, O
H	Tool length offset number	B
I	X-coordinate of arc center, Radius for circle cutting	B O
J	Y-coordinate of arc center, Cutting depth for circle cutting	B, O
K	Z-coordinate arc center	B
L	Number of repetitions	B, O
M	Miscellaneous functions	B
N	Sequence number	B
O	Program number	B
P	Dwell time, Program No. and sequence No. designation in subprogram	B O
Q	Depth of cut, shift of canned cycles	O
R	Point R for canned cycles, Radius designation of a circular arc	B, O
S	Spindle-speed function	B
T	Tool function	B
U	Additional linear axis parallel to X-axis	O
V	Additional linear axis parallel to Y-axis	O
W	Additional linear axis parallel to Z-axis, new initial point of canned cycle	O
X	X-coordinate, dwell time	B
Y	Y-coordinate	B
Z	Z-coordinate	B

(2) Function characters

Table 2.2.5 Function Characters

EIA Code	ISO Code	Meanings	Remarks
Blank	NuL	Error in significant data area in EIA. Disregarded in ISO	
BS	BS	Disregarded	
Tab	HT	Disregarded	
CR	LF/NL	End of Block (EOB)	
	CR	Disregarded	
SP	SP	Space	
ER	%	Rewind stop	
UC		Upper shift	
LC		Lower shift	
2-4-5 bits	(	Control out (Comment start)	EIA : Special code
2-4-7 bits	)	Control in (Comment end)	
+	+	Disregarded. User macro operator	
−	−	Minus sign. User macro operator	
0 to 9	0 to 9	Numerals	
a to z	A to Z	Address characters	
		Optional block skip. User macro operator	
Del	DEL	Disregarded (Inlcuding All marks)	
.	.	Decimal point	
Parameter setting	#	Sharp (Variable)	EIA : Special code
*	*	Astrisk (Multiplication operator)	
=	=	Equal mark	
[	[	Left bracket	
]	]	Right bracket	
O	:	For comment in Macroprogram	
$	$		
@	@		
?	?		
,	,		

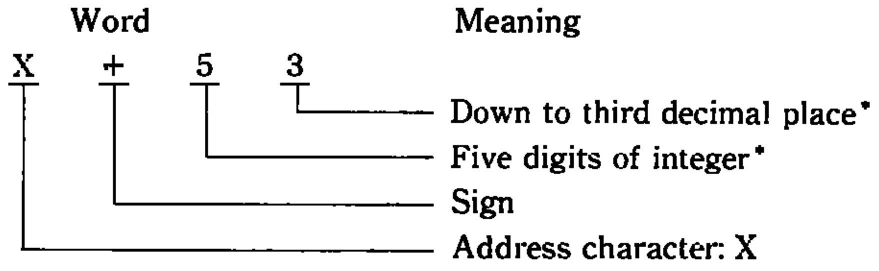
NOTE
1. Characters other than the above cause error in significant data area.
2. Information between Control Out and Control In is disregarded as insignificant data.
3. Tape code (EIA or ISO) can be designated by setting Pm0004 D0.

(3) Input format

A variable block format conforming to JIS (Japanese Industrial Standard) B6313 is used for YASNAC i80M.

Table 2.2.6 shows the input format. Numerals following the address characters in Table 2.2.6 indicate the programmable number of digits.

⟨EXAMPLE⟩



* Differs in metric and inch input format.

- Metric input format

  O5 N5 G3 a+53 F5 S2 T2 M3 D(H)2 B3;

- Inch input format

  O5 N5 G3 a+44 F31 S2 T2 M3 D(H)2 B3;

NOTE

1. "a" represents X, Y, Z, I, J or K.
2. P, Q, R and L are omitted in the above format because they are used for various meanings.

NOTE

1. A decimal point should be omitted in actual programming, when you make a program including decimal points. Refer to Par. 2.8.3 "Decimal Point Input"
2. The leading zeros can be suppressed for all address codes. Plus signs need not be programmed, but all minus signs must be programmed.
3. In the manual, EOB (end of block) code in a program example is represented by a semicolon (;). In actual programming, CR (EIA code) or LF/NL (ISO code) should be used instead of the semicolon (;).

Table 2.2.6 Standard Input Format

No.	Address		Metric Output		Inch Input		B : Basic O : Option
			Metric Input	Inch Input	Metric Input	Inch Input	
1	Program No.		O 5		O 5		B
2	Sequence No.		N 5		N 5		B
3	G function		G 3		G 3		B
4	Coordinate Word	Linear axis	a +63	a +54	a +63	a +54	B
		Rotary axis	b +63	b +63	b +63	b +63	O
5	Feed/min		F 60	F 41	F 60	F 51	B
6	Feed/min 1/10		F 61	F 42	F 61	F 52	B
9	S-function		S 5		S 5		O
10	T-function		T 2		T 2		B
			T 4		T 4		O
11	M-function		M 3		M 3		B
12	Tool Offset No.		H4 or D2		H4 or D2		B
13	B-function		B 3		B 3		O
14	Dwell		P 63		P 63		B
15	Program No. designation		P 5		P 5		B
16	Sequence No. designation		P 5		P 5		B
17	No. of repititions		L 9		L 9		B

Table 2.2.7 Sub Micron Input Format

No.	Address		Metric Output		B : Basic O : Option
			Metric input	Inch input	
4	Coordinate Word	Linear axis	a ±54	a ±45	O
		Rotary axis	b ±54	b ±54	O
5	Feed/min		F51	F32	O

Table 2.2.8 Sub Sub-micron Input Format

No.	Address		Metric Output		B : Basic O : Option
			Metric input	Inch input	
4	Coordinate Word	Linear axis	a ±45	a ±36	O
		Rotary axis	b ±45	b ±45	O
5	Feed/min		F42	F23	O

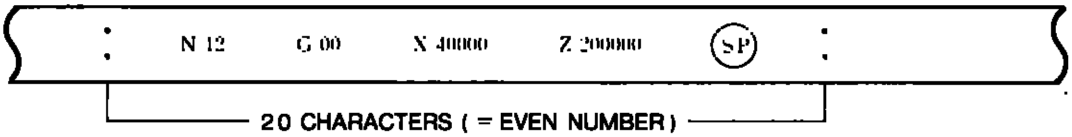
- **Block**

(1) One block is terminated with the end of block (EOB) code. The EOB code is represented by "CR" in EIA or "LF/NL" in ISO. However, the EOB code in programming examples in this manual is indicated by ";" for simplicity.

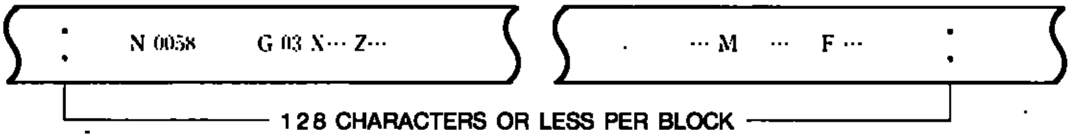
(2) Use only the address codes and function codes shown in Tables 2.2.4 and 2.2.5.

(3) Up to 128 characters can be written in one block.

Note however that the maximum number of characters defined above does not include Del and other invalid characters.



(a) Character number matching for TV check (if an odd number, error may result)



(b) Number of effective characters allowed in one block

Fig. 2.2.5

- **Optional block skip (／1) (／2 to／9) ※**

Those blocks in which " n"(n=1-9) is included are disregarded between n and the end of that block, when the external optional block skip switch for that number " n"is on.
With "／1," "1" can be omitted.

〈EXAMPLE〉

```
/2  N1234  G01  X100/3  Y200 :
```

When the switch for 2 is on, the entire block is disregarded, and when the switch for 3 is on, this block is read as if

```
N  1234  G01  X100 ;
```

NOTE

1. The optional block skipping process is executed while the blocks are read into the buffer register. If the blocks have been read, subsequent switching on is ineffective to skip the blocks.
2. While reading or punching out programs, this function is ineffective.

2.2.2.5 Comment part

- Message display by control-out/ control-in

(1) Control-out/control-in programming and display :

Any desired message can be programmed in a section enclosed with the control-out and control-in codes in a part program for display on the CRT screen. In this case, the information enclosed between the control-out and control-in codes is assumed to be meaningless information.

(2) Method of editing control-out and control-in codes :

The control-out and control-in section can be edited by normal edit operation.

① Depress SHIFT key, then depress U key to input character " (".

② Depress SHIFT key, then depress V key to input character ") ".

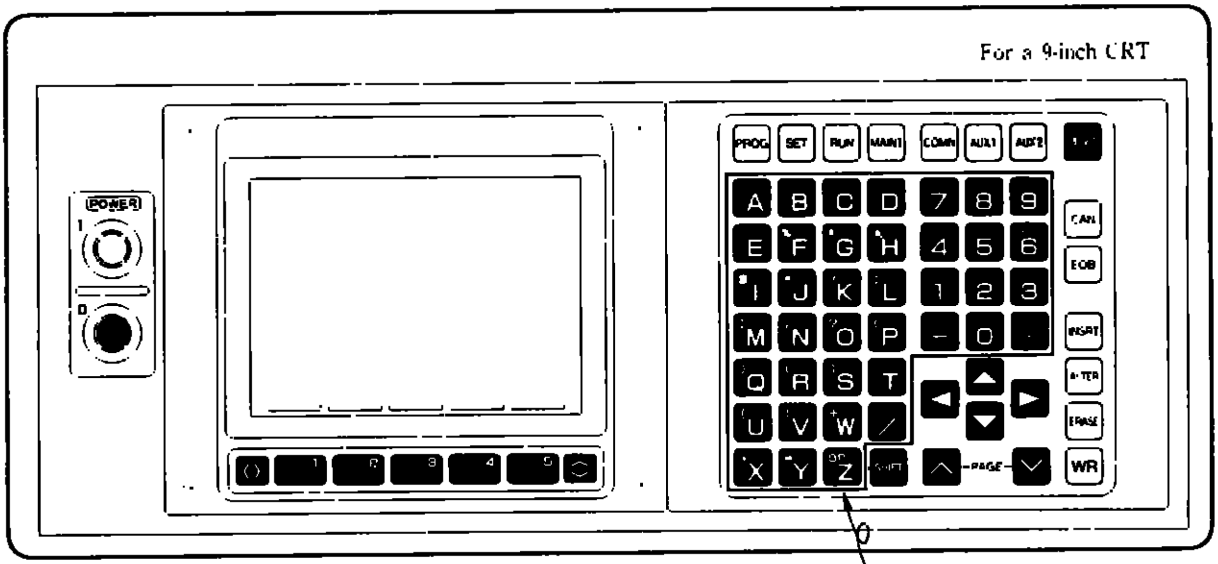
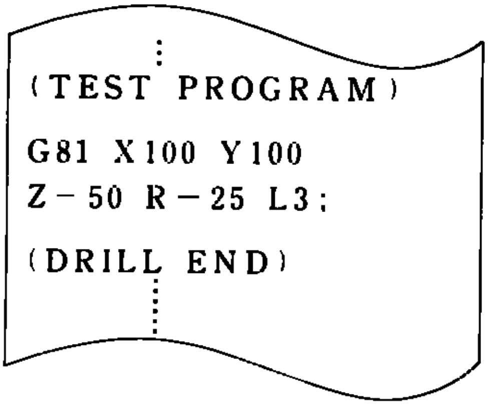

For a 9-inch CRT

Fig. 2.2.6 Characters Entered Between Control-out and Control-in (Keys Enclosed by Line)

NOTE

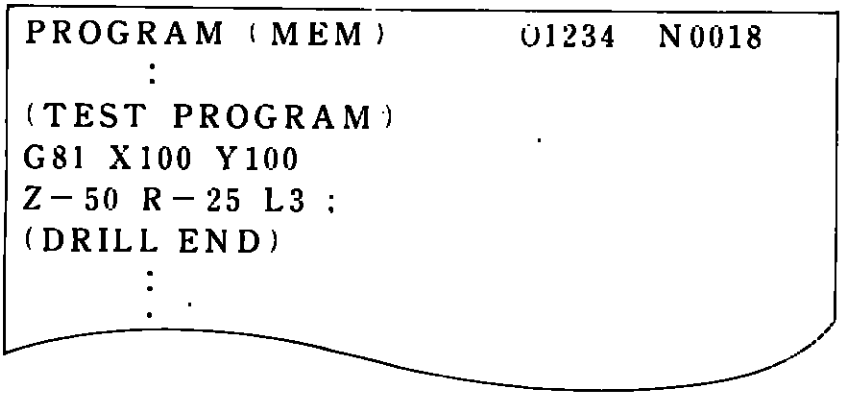
1. Characters can be entered between control-out and control-in by using the keys that are enclosed by line in Fig. 2.2.6.
2. Control-out and control-in cannot be entered in a section already enclosed by control-out and control-in. Only the characters in the frame in Fig. 2.2.6 can be entered in a section enclosed by control-in.

〈Example of programming〉



```
        :
(TEST PROGRAM)
G81 X100 Y100
Z-50 R-25 L3;
(DRILL END)
        :
```

〈Example of message display by control-out or -in〉



```
PROGRAM (MEM)          O1234  N0018
       :
(TEST PROGRAM)
G81 X100 Y100
Z-50 R-25 L3 ;
(DRILL END)
       :
       :
```

2.2.3 Buffer Register and Multi-active Register

- **Buffer register**

(1) During normal operation, two blocks of data are read in advance and offset computing is made for the follow-on operation.

(2) In the tool radius compensation C※mode, two blocks of data or up to 4 blocks of data are read in advance and compensation computing required for the next operation is executed.

(3) One block can contain up to 128 characters including EOB.

- **Multi-active Register※**

(1) For the portion of part programs sandwiched between M93 and M92, up to 7 blocks of data are read in advance.

Table 2.2.9

M code	Meaning
M92	Multi-active register OFF
M93	Multi-active register ON

(2) Inter-block stoppage can be eliminated when the program is so made that the automatic operation time of advance reading of 7 blocks is longer than processing time of advance reading of next 7 blocks of data.

NOTE Advance reading is not made for every 7 blocks but is always ready to be made up to 7 blocks in M93 mode.

2.2.4 Tape Code

- **Tape code**

With this control, both the EIA codes (EIA RS-244-A) and the ISO codes (ISO 840) can be used.

Table 2.2.10 shows the EIA and ISO punched tape formats.

Before starting to program any machining operation, a decision must be made as to the code to be used.

- **EIA/ISO auto-select**

(1) The equipment code must be switched over prior to operation by the completed NC tape.

(2) The code is automatically discriminated regardless of the contents of setting No. Pm0004 D0 (for the first RS 232C) or No. Pm0006 D0 (for the second RS 232C). This is automatically set so that the code is determined by the EOB code first read in a label skip state and the subsequent data are read by the determined code.

(3) For punching tapes.the code must be selected by the setting of Pm0004 D0 (for the first RS232C) or Pm0006 D0(for the second RS232C).
When Pm0004 D0 is set to "1" ······ EIA code
When Pm0004 D0 is set to "0" ······ ISO code

Table 2.2.10 Tape Code

EIA CODE									CHARACTERS		ISO CODE								
8	7	6	5	4	∘	3	2	1			8	7	6	5	4	∘	3	2	1
		○			∘				0				○	○		∘			
					∘			○	1		○		○	○		∘			○
					∘		○		2		○		○	○		∘		○	
			○		∘		○	○	3				○	○		∘		○	○
					∘	○			4		○		○	○		∘	○		
			○		∘	○		○	5				○	○		∘	○		○
			○		∘	○	○		6				○	○		∘	○	○	
					∘	○	○	○	7		○		○	○		∘	○	○	○
					∘				8		○		○	○	○	∘			
			○	○	∘			○	9				○	○	○	∘			○
	○	○			∘			○	a	A		○				∘			○
	○	○			∘		○		b	B		○				∘		○	
	○	○	○		∘		○	○	c	C	○	○				∘		○	○
	○	○			∘	○			d	D		○				∘	○		
	○	○	○		∘	○		○	e	E	○	○				∘	○		○
	○	○	○		∘	○	○		f	F	○	○				∘	○	○	
	○	○			∘	○	○	○	g	G		○				∘	○	○	○
	○	○		○	∘				h	H		○			○	∘			
	○	○	○	○	∘			○	i	I	○	○			○	∘			○
	○		○		∘			○	j	J	○	○			○	∘		○	
	○		○		∘		○		k	K		○			○	∘		○	○
	○				∘		○	○	l	L	○	○			○	∘	○		
	○		○		∘	○			m	M		○			○	∘	○		○
	○				∘	○		○	n	N		○			○	∘	○	○	
	○				∘	○	○		o	O	○	○			○	∘	○	○	○
	○		○		∘	○	○	○	p	P		○		○		∘			
	○		○	○	∘				q	Q	○	○		○		∘			○
	○			○	∘			○	r	R	○	○		○		∘		○	
		○	○		∘		○		s	S		○		○		∘		○	○
		○			∘		○	○	t	T	○	○		○		∘	○		
		○	○		∘	○			u	U		○		○		∘	○		○
		○			∘	○		○	v	V		○		○		∘	○	○	
		○			∘	○	○		w	W	○	○		○		∘	○	○	○
		○	○		∘	○	○	○	x	X	○	○		○	○	∘			
		○	○	○	∘				y	Y		○		○	○	∘			○
		○		○	∘			○	z	Z		○		○	○	∘		○	
					∘				Blank	NUL						∘			
		○		○	∘		○		BS		○				○	∘			
		○	○	○	∘	○	○		Tab	HT					○	∘			○
○					∘				CR	LF/NL					○	∘		○	
									—	CR	○				○	∘	○		○
			○		∘				SP		○		○			∘			
				○	∘		○	○	ER	%	○		○			∘	○		○
	○	○	○	○	∘	○			UC	—									
	○	○	○	○	∘		○		LC	—									
				○			○		—	(			○		○	∘			
	○			○			○		—	)	○		○		○	∘			○
	○	○	○		∘				+				○		○	∘		○	○
	○				∘				−				○		○	∘	○		○
	○				∘	○	○		○	:			○	○	○	∘		○	
		○	○		∘			○	/		○		○		○	∘	○	○	○
	○	○	○	○	∘	○	○	○	Del	DEL	○	○	○	○	○	∘	○	○	○
○	○	○	○	○	∘	○	○	○	All Mark		○	○	○	○	○	∘	○	○	○
See Note 2. • .									#		○		○			∘		○	○
○			○	○	∘				*		○		○		○	∘		○	
○				○	∘	○			=		○		○	○	○	∘	○		○
○		○	○		∘				[		○	○		○	○	∘		○	○
○		○			∘		○		]		○	○		○	○	∘	○		○
○			○		∘	○			$				○			∘	○		
○				○	∘	○	○	○	@		○	○				∘			
○			○	○	∘	○	○		?				○	○	○	∘	○	○	○
	○	○		○	∘		○	○	•				○		○	∘	○	○	
	○	○	○	○	∘	○			,		○		○		○	∘	○		
value of pm 4108									'				○			∘	○	○	○
value of pm 4109									"				○			∘		○	
value of pm 4144									<				○	○	○	∘	○		
value of pm 4145									>		○		○	○	○	∘	○	○	
value of pm 4146									;		○		○	○	○	∘		○	○

NOTE

1. For characters from # to ?, EIA codes have not been agreed upon. In the present system, for the time being, the left provisional codes are used.
2. EIA code of character # can be designated by the parameter Pm4100.

2.2.5 NC Tape

- **Paper tape**

  For part program tapes, eight-channel paper tapes for computers conforming to JIS C6243 (width: 25.4 ± 0.08 mm, thickness: 0.108 mm) are used.

  The color should be black or gray.

  Tapes with high transparency tend to cause reading errors, and should not be used.

- **NC tape punch**

  Part programs written on process sheets are punched in EIA or ISO codes in paper tape with a tape puncher.

  A part program tape should be provided with a proper length of feed holes at the leading and the trailing ends.

NOTE For a tape reader using 6" reels, the feed hole length should be at least 70 cm, and for a tape reader using 8" reels, it should be at least 1 m.

- **NC tape check**

  Punched part program tapes can be checked by an NC with the following functions.

  - Machine lock
  - M function lock
  - Dry run
  - Single-block operation

- **Splicing NC tape**

(1) To join part program tapes, the two ends should be placed end to end without overlapping and without a space.

(2) A proper length of splicing tape (approx. 0.08 mm in thickness) should be applied on one side.

(3) After splincing, the tape should be checked for correct alignment of the feed holes before use.

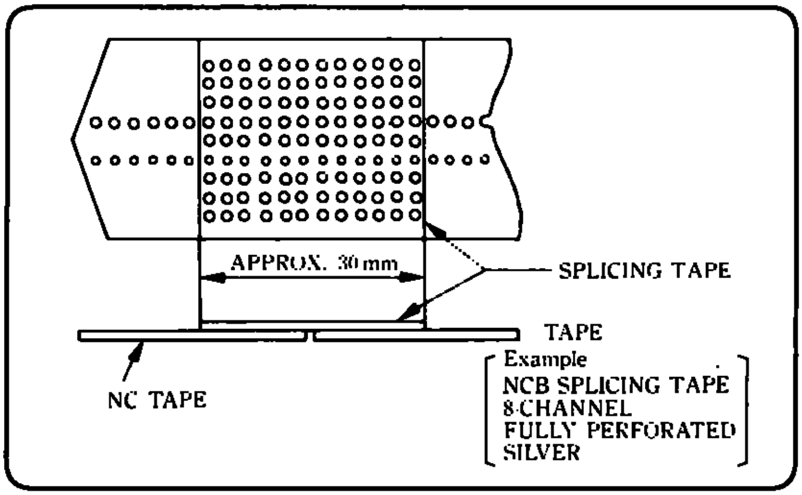


Fig. 2.2.7 Splicing Part Program Tape

NOTE
1. Tape splices are available in the fully perforated type and in the type in which only the feed holes are punched, but the former is more convenient.
2. Do not use rigid industrial adhesive, and do not make the joint too thick, as these conditions are conductive to jamming troubles.

- **Keeping NC tape**

Generally, properly maintained part program tapes can last at least 300 cycles, with one cycle consisting of one reading and one rewinding pass.

NOTE
1. Part program tapes should be stored in a clean area, free of contaminants and humidity.
2. Do not handle part program tapes wearing gloves contaminated with oil or cutting fluid.

2.3 PREPARATORY FUNCTION (G-FUNCTION)

An address character G and up to 3 digits following it specify the operation of the block.

(1) There are two types of G codes.

- Non-modal G code (marked with ∗): the codes are effective only for the commanded block.
- Modal G code (belonging to groups 01 to 23): the codes belonging to the division B are included in the basic specifications.

  Once they are commanded, they are effective until another G code in the same group will be commanded.

They can be programmed twice or more in the same block. When different G codes in the same group are programmed, the last appearing G code is effective.

(2) When a G code belonging to the 01 group is commanded during a canned cycle (G73, G74, G76, G77 and G81 through G89), the canned cycle is canceled and these codes in the group 09 becomes G80.

(3) When the RESET key is depressed during the execution of a tool radius compensation C (G41, or G42) or a canned cycle, they become G40 or G80, respectively, which cancels the programmed commands.

(4) G43, G44, G49 and G45 through G48 belonging to the ∗ group can be programmed together with the following G codes in the 01 group in the same block.

If ∗ group other than those G codes and G codes of 08 group can be programmed in the same block as G code of 01 group an alarm will be issued.

Table 2.3.1

	Combination G code
G43, G44, G49	G00, G01, G60
G45 to G48	G00, G01, G02, G03, G60

(5) G code shown in Table 2.3.2 can be selected by parameter setting. The G code specifies as to the state immediately after turning power supply on.

Table 2.3.2 Selection of G Code State at Power ON

Group	G Code to be applied	Parameter			
01	G00 or G01	Pm4000 D0		0	G00
				1	G01
03	G90 or G91	Pm4000 D1		0	G90
				1	G91
08	G43, G44 or G49	Pm4000 D3, D4	D 4	D 3	
			0	0	G49
			0	1	G43
			1	0	G44

(6) G code of groups 02 and 03 to be applied after resetting can be set by the parameter.

Table 2.3.3 Setting of G Code at Reset

Group	Timing	Parameter Pm4000 D5=1	Parameter Pm4000 D5=0
02	At power ON	G17	G17
	At reset	G code immediately preceding is stored.	G17
03		Parameter Pm4000 D6=1	Parameter Pm4000 D6=0
	At power ON	Pm4000 D1=0 . G90 Pm4000 D1=1 : G91	Pm4000 D1=0 : G90 Pm4000 D1=1 : G91
	At reset	G code immediately preceding is stored.	

Table 2.3.4 List of G Codes

G code	Group	Function	B : Basic O : Option
G00 ◥	01	Positioning (feedrate)	B
G01 ◥		Linear interpolation	B
G02		Circular interpolation CW, Helical interpolation CW	B, O
G03		Circular interpolation CCW, Helical interpolation CCW	B, O
G04	*	Dwell	B
G06		Positioning in error detect off mode	B
G09		Exact stop	B
G10		Tool offset value and work coordinate, Shift value modification	B, O
G12		Circle cutting CW	O
G13		Circle cutting CCW	O
G17 \	02	XY plane designation	B
G18		ZX plane designation	B
G19		YZ plane designation	B
G20	06	Inch input designation	B
G21		Metric input designation	B
G22	04	Stored stroke limit ON	O
G23		Stored stroke limit OFF	O
G25	*	Program copy	O
G27		Reference point returning check	B
G28		Automatic return to reference point	B
G29		Return from reference point	B
G30		Return to 2nd, 3rd, 4th reference points	B, O
G31		Skip function	O
G40 ◥	07	Tool radius compensation, cancel	O
G41		Tool radius compensation, left	O
G42		Tool radius compensation, right	O
G43 \	08	Tool length offset plus direction	B
G44 \		Tool length offset minus direction	B
G49 \		Tool length offset cancel	B
G45	*	Tool position offset, extension	B
G46		Tool position offset, retraction	B
G47		Tool position offset, double extension	B
G48		Tool position offset, double retraction	B
G50 ◥	15	Scaling OFF	O
G51		Scaling ON	O
G52 \	12	Return to base coordinate system	B, O
G53	*	Temporary shift to machine coordinate system	B

Table 2.3.4 List of G Codes (Cout'd)

G code	Group	Function	B : Basic O : Option
G54	12	Shift to work coordinate system 1	O
G55		Shift to work coordinate system 2	O
G56		Shift to work coordinate system 3	O
G57		Shift to work coordinate system 4	O
G58		Shift to work coordinate system 5	O
G59		Shift to work coordinate system 6	O
G60	01	Unidirectional approach	O
G61	13	Exact stop mode	B
G64		Exact stop mode cancel	B
G65	*	Non-modal call of user macro	O
G66	14	Modal call of user macro	O
G67 ◥		Modal call of user macro cancel	O
G68	18	Coordinate rotation ON	O
G69 ◥		Coordinate rotation OFF	O
G70	*	Bolt hole circle	O
G71		Arc	O
G72		Line-at-angle	O
G73	09	Canned cycle 10	O
G74		Canned cycle 11	O
G76		Canned cycle 12	O
G77		Canned cycle 13	O
G80 ◥		Canned cycle cancel	O
G81		Canned cycle 1. Output for external motion	O
G82		Canned cycle 2	O
G83		Canned cycle 3	O
G84	09	Canned cycle 4	O
G85		Canned cycle 5	O
G86		Canned cycle 6	O
G87		Canned cycle 7	O
G88		Canned cycle 8	O
G89		Canned cycle 9	O
G90	03	Absolute command designation	B
G91		Incremental command designation	B
G92	*	Coordinate system setting	B
G93	05	Solid tap mode ON	O
G94 ◥		Solid tap mode OFF	O
G98	10	Return to initial point for canned cycles	B
G99		Return to point R for canned cycles	B
G106	*	Automatic corner override	O
G125		Tool life management lifetime count magnification specified	O
G181	09	Canned cycle (two-stepped hole drilling)	O
G182		Canned cycle (two-stepped hole spot facing)	O
G185		Canned cycle (two-stepped hole boring)	O
G186		Canned cycle (two-stepped hole boring)	O
G187		Canned cycle (two-stepped hole boring)	O
G189		Canned cycle (two-stepped hole boring)	O

NOTE

1. The G codes in the * group are non-modal, and are effective only for the block in which they are commanded. They cannot be programmed two or more times in a block. They must be programmed only once in a block of their own.
2. The codes marked with ◥ are automatically selected at powerON or reset.
3. The codes marked with ◹ are automatically selected at powerON.

2.4 INTERPOLATION

2.4.1 Positioning

The positioning is commanded by G00, G06, G60 and Gnn.

The commanded axis moves at rapid speed to the position as follows.

- For absolute command (G90): to the position on work coordinate system.
- For incremental command(G91):from current position to the specified position.

2.4.1.1 Positioning (G00, G06)

- G00: Modal G code in 01 group

G00: Positioning is started in the error detection on mode.

In the error detection on mode, the number of stay pulses due to servo delay is checked after motion pulses were distributed, and the next block is started only when the number has reduced to a permissible level.

In this mode, sharp corner cutting is possible.

```
G00 X·········Y·········Z·········(※α·········β·········) ;
```

(where α and β =A, B, C, U, V, or W)

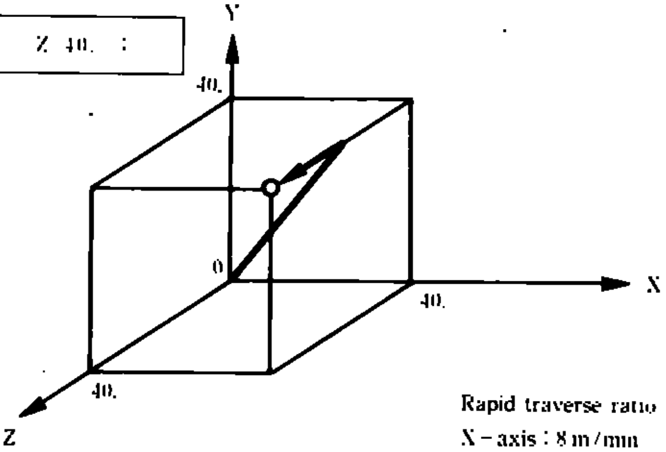
With this command, the tool is sent to the specified position in rapid traverse motions along the 3 axes (5 axes※) simultaneously. If any of the coordinate positions is not specified, the machine does not move along that coordinate axis.

NOTE

1. The rapid traverse rate for the respective axes are inherent to the machine tool. Refer to the machine tool builder's manual.
2. When programming tool positioning commands, take care to avoid the possibility of tool and workpiece interference.
   Motions in the respective axis directions are independent of each other, and therefore, the resultant tool path is not necessarily straight.

Example of Programming

```
G 00  X 40.  Y 40.  Z 40.  ;
```



Rapid traverse ratio

X – axis : 8 m / min
Y – axis : 8 m / min
Z – axis : 4 m / min

Fig. 2.4.1 Positioning of simultaneous 3-axis Rapid Traverse

2.4.1.2 G06 (Non-modal G code of group ✱)

G06 command performs positioning in the error detection OFF mode. This command is effective only in the specified block.
In this mode, the tool moves to the next block immediately after completion of pulse distribution. Therefore, the path at the corner section is rounded.
This command is given as shown below.

```
G06  X·········Y·········Z·········(※α·········β·········);
```

2.4.1.3 Unidirectional approach (G60) ✱

This function is effective to position the tool at high accuracy.

```
G60  X·········Y·········Z·········(※α·········β·········);
```

With this command, the tool moves and stops at the specified position.

NOTE Only for the direction specified by parameter Pm4014, the tool overtravels the stop position once by the amount specified by parameters (Pm4461 to Pm4465).
Then, it returns to the specified position to stop.

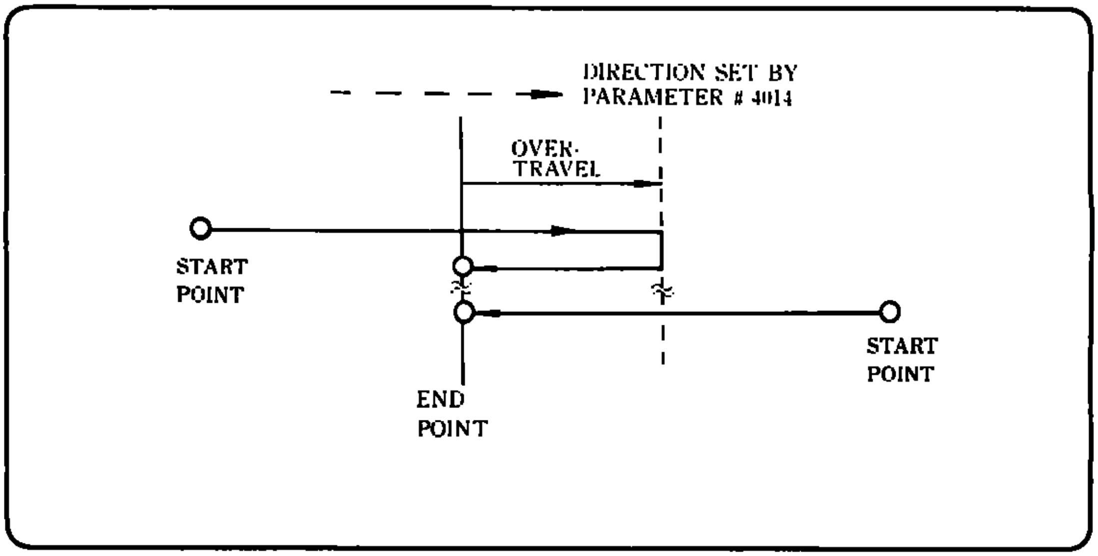


Fig. 2.4.2

2.4.2 Linear Interpolation (G01)

Use G01 for linear interpolation.

```
G01  X·········Y·········Z·········(※α·········β·········) F······ ;
```

where α and β = A, B, C, U, V, or W

With this command, the tool is moved simultaneously in the three (five ※) axial directions resulting in a linear motion.

If the command is omitted, axes are not moved.

(1) Feedrate

Specify by an F code. In this case, the feedrate is controlled so that the commanded hypothetical speed of all axes (tangential speed for component axial direction)becomes the specified value.

$$F = \sqrt{Fx^2 + Fy^2 + Fz^2 + (F\alpha^2 + F\beta^2)}$$

(where Fx, Fy...are feedrates in the X, Y... directions.)

NOTE If no F code is given in the block containing the G01 or in preceding blocks, the block constitutes an error "370".

(2) End point

The end point can be programmed either in absolute coordinates or in incremental values with G90 or G91 respectively. [Refer to par.2.8.1, "Absolute Incremental Specification (G90, G91)"].

Example of Programming

```
G 01  X 40.  Y 40.  Z 40.  F 100  ;
```

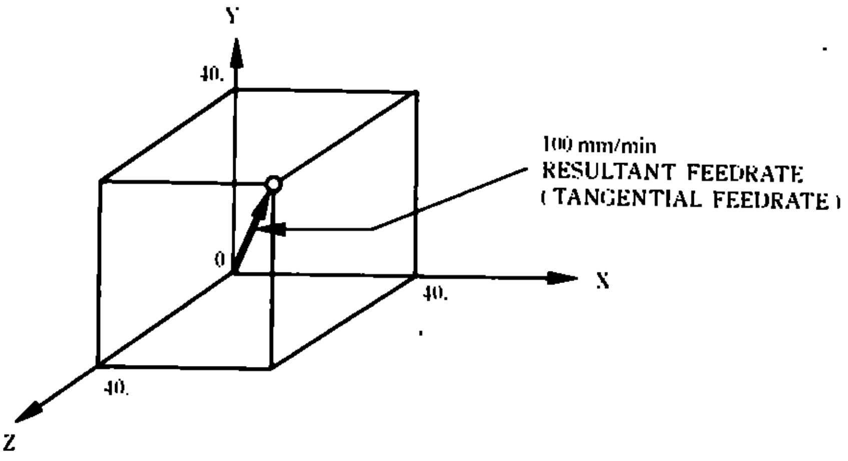


Fig. 2.4.3

(3) Where the optional 4th or 5th axis is rotary axis (A, B or C), for the same F code, the feedrates in the basic three axis directions (X, Y and Z), and the rotary axis feedrate are as indicated.

Table 2.4.1 Minimum F Command Unit

F-function			In minimum F command unit	
			Feedrate of basic three axes	Feedrate of rotary axes
Metric output	Metric input	F60	1 mm/min	1 deg/min
	Inch input	F41	0.1 in./min	1 deg/min
Inch output	Metric input	F60	1 mm/min	0.3937 deg/min
	Inch input	F51	0.1 in./min	1 deg/min

NOTE

1. Feedrate of linear 4th and 5th axes is the same as that of basic three axes.
2. Feedrate of rotary axis is the value where D7=0 at pm2004.
   For details, refer to Par. 2.5.2 "Feed".

2.4.3 Circular Interpolation (G02, G03)

Circular interpolation is instructed with G02 and G03. Commands in Table 2.4.2 are required for circular interpolation.

Table 2.4.2

1	Plane specification	G17	Circular arc on XY plane
		G18	Circular arc on ZX plane
		G19	Circular arc on YZ plane
2	Rotation direction	G02	Clockwise (CW)
		G03	Counterclockwise (CCW)
3	Position of end point	**G90 : Two axes among X, Y, and Z**	End point position at work coordinate system
		G91 : Two axes among X, Y, and Z	Distance from start point to end point
4	Distance from start point to center	Two axes among I, J and K	Signed distance from start point to center
	Radius of circular arc	R	Radius of circular arc
5	Feed rate	F	Speed along circular arc

(1) With the following commands, the tool is controlled along the specified circular paths on the XY, ZX, or YZ plane, at a tangential speed specified by the F code.

XY plane

$$\boxed{G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\cdots\cdots Y\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ I\cdots\cdots J\cdots\cdots \end{Bmatrix} Z(\alpha)\cdots\cdots F\cdots\cdots ;}$$

ZX plane

$$\boxed{G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Z\cdots\cdots X\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ K\cdots\cdots I\cdots\cdots \end{Bmatrix} Y(\alpha)\cdots\cdots F\cdots\cdots ;}$$

YZ plane

$$\boxed{G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\cdots\cdots Z\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ J\cdots\cdots K\cdots\cdots \end{Bmatrix} X(\alpha)\cdots\cdots F\cdots\cdots ;}$$

The moving direction of the tool along the circle is as follows.

G02: Clockwise (CW)
G03: Counter clockwise (CCW)

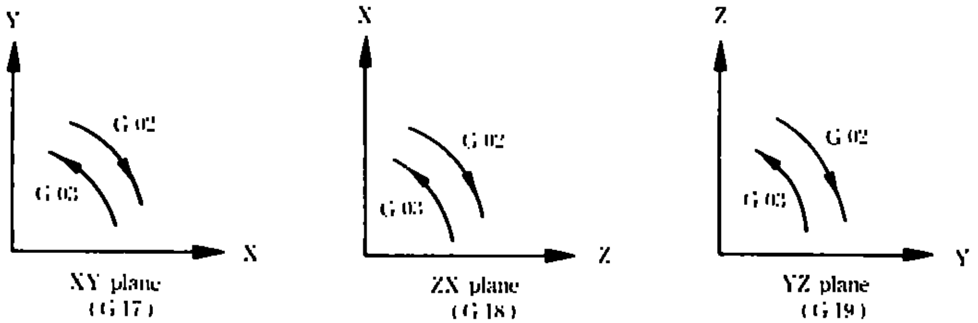


Fig. 2.4.4

(2) Normally, when circular interpolation (G02,G03) is to be programmed, the plane of interpolation should be specified in advance with G17,G18 or G19.

G17: XY plane or $X\alpha$,$X\beta$ plane
G18: ZX plane or $Z\alpha$,$Z\beta$ plane
G19: YZ plane or $Y\alpha$,$Y\beta$ plane

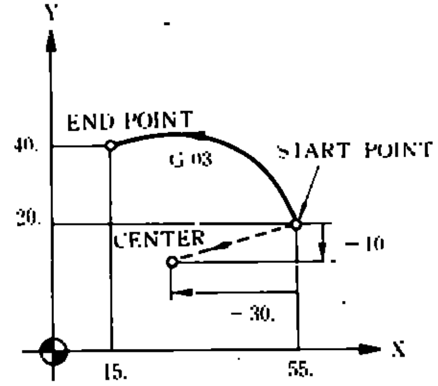
NOTE The 4th axis or 5th axis should be linear axis.

In addition to the plane of circular interpolation, these G codes specify planes for tool radius compensation (G41, G42). If no selection is made to the contrary, XY plane (G17) is selected automatically immediately after switching on the power supply.

(3) The end point of the circular arc may be specified by G90 or G91 respectively in absolute or incremental values. However, the center of the circle is always programmed in incremental values from the start point, irrespective of G90 or G91.

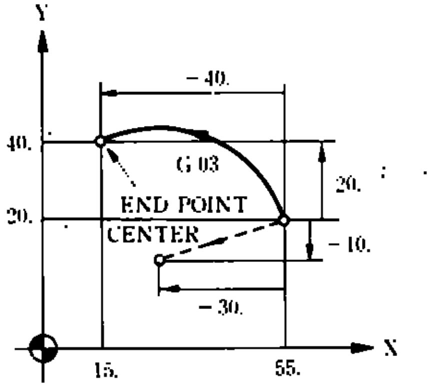
Example of Programming

```
G 17  G 90  G 03  X 15.  Y 40 I-30  J-10.  F 150 .
```



(a) Absolute command with G 90

```
G 17  G 91  G 03  X-40.  Y 20 I-30.  J-10  F 150  ;
```



(b) Incremental command with G 91

Fig. 2.4.5 End Point of Circular Arc

(4) Instead of the coordinates I,J, and K for the center of the circle, the radius can be directly specified with an R command. This is called cirular interpolation with R designation mode.

NOTE In this case,
- when R > 0, a circular arc with the center angle less than 180°, is specified, and
- when R < 0, a circular arc with the center angle larger than 180° is specified.

Example of Programming

```
G 17  G 02  X···  Y···  R ±···  F···  ;
```

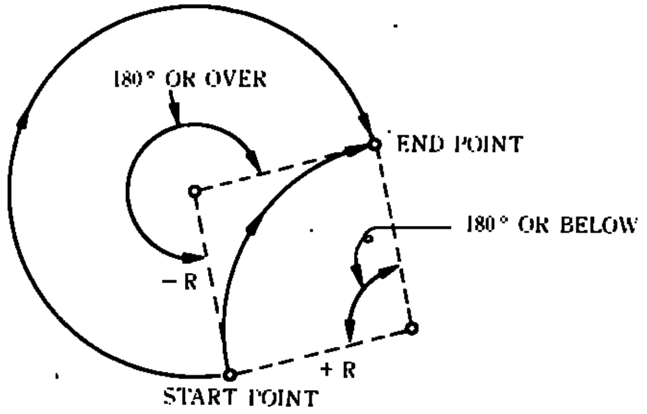


Fig. 2.4.6 Circular Interpolation with Radius R designation

(5)

```
G17  G02  (G03)  I······  J······  F······  Ln ;
```

With this command, complete circular interpolations are repeated n times. Without an L designation, the interpolation is executed only once.

NOTE If this command is executed at single block ON,the interpolation stops every circle.

(6) When a linear 4th axis option is used, circular interpolation is possible in the X α, Z α, and Y α planes in addition to the XY, ZX and YZ planes (where α = U, V, or W)

X α plane	$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\cdots\cdots \alpha\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ I\cdots\cdots J\cdots\cdots \end{Bmatrix} F\cdots\cdots :$
Z α plane	$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Z\cdots\cdots \alpha\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ K\cdots\cdots I\cdots\cdots \end{Bmatrix} F\cdots\cdots :$
Y α plane	$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\cdots\cdots \alpha\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ J\cdots\cdots K\cdots\cdots \end{Bmatrix} F\cdots\cdots :$

(7) When a linear 5th axis option is used, circular interpolation is possible in the X *b* , Z *b* , or Y*b* planes in addition to the XY, ZX, and YZ planes (where β = U, V, or W)

Xβ plane	$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\cdots\cdots \beta\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ I\cdots\cdots J\cdots\cdots \end{Bmatrix} F\cdots\cdots :$
Zβ plane	$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Z\cdots\cdots \beta\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ K\cdots\cdots I\cdots\cdots \end{Bmatrix} F\cdots\cdots :$
Yβ plane	$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\cdots\cdots \beta\cdots\cdots \begin{Bmatrix} R\cdots\cdots \\ J\cdots\cdots K\cdots\cdots \end{Bmatrix} F\cdots\cdots :$

NOTE

1. G17 G02 X…… $\left\{\begin{matrix} R\cdots\cdots \\ I\cdots\cdots\ J\cdots\cdots \end{matrix}\right\}$ F…… ;

   Where address characters for the 4th axis are missing as in the above command, the XY plane is automatically selected. Circular interpolation cannot be performed on the axes including rotary 4th axis.
2. Circular paths covering two or more quadrants can be programmed in a single block. A complete closed circle can also be programmed.

Example of Programming

```
G 00  X 0  Y 0 ;
G 02  X 0  Y 0  I 10.  J 0  F 100 ;
```

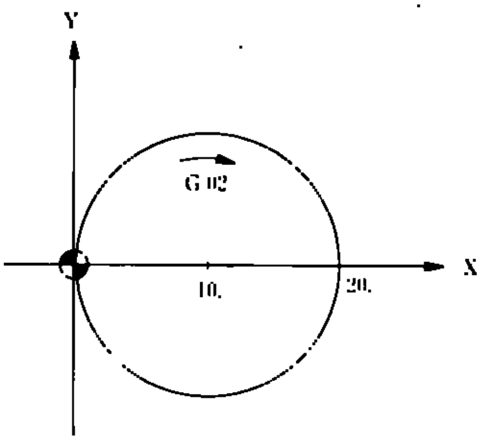
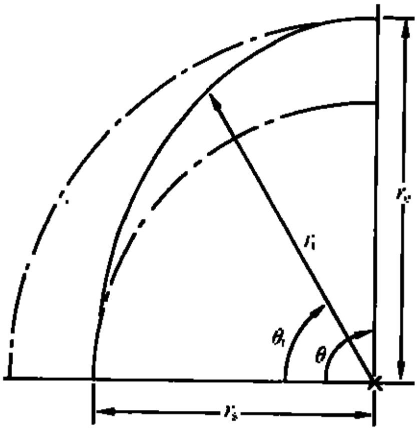


Fig. 2.4.7 One Circle

3. Figure 2.4.8 shows motions when the end point is not on the circumference:

If the specified end point is not on the specified arc, the radius is gradually varied from the start point to the end point, resulting in spiral interpolation.



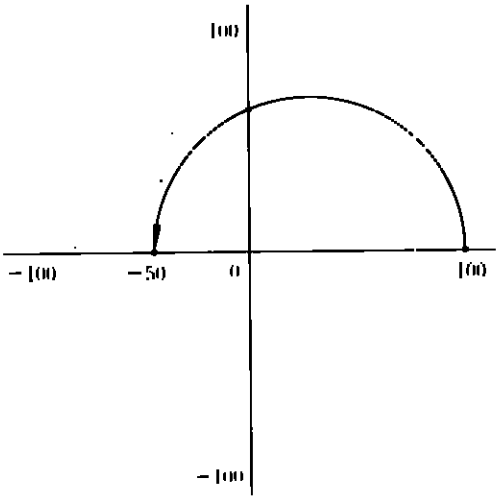
1. $r_i = r_s + (r_e - r_s)/\theta * \theta_i$

   Amount of correction of radius per unit angle

   $\Delta r = (r_e - r_s)/\theta$

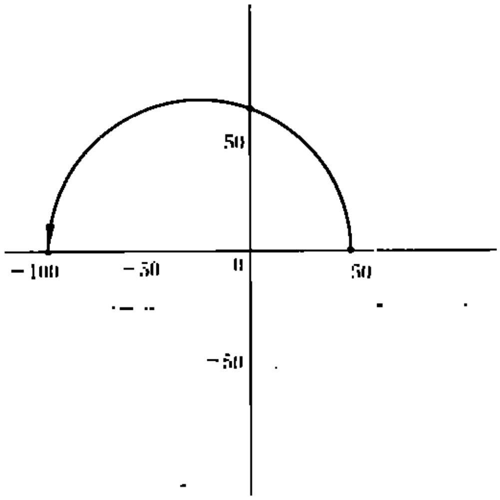
2. (a) Circular interpolation

```
G 01  X 100  Y 0  F 200 ;
G 03  X -50  I-100  ;
```



(a) Where end point is in circumuference

```
G 01  X 50  Y 0  :
G 03  X-100  I-50  :
```



(b) Where end point is out of circumuference

However, if the distance between the end point and the arc specified $(r_e - r_s)$ exceeds the pm4820 set values, an alarm will occur.

pm4015 D1=0 : Not check the arc radius allowable error.

D1=1 : Check the arc radius allowable error.

Fig 2.4.8

4. If the distance between the start and end points is smaller than the value set for parameter pm4450, these two points are connected by a straight line.

2.4.4 Helical Interpolation (G02, G03) ✻

A circular interpolation on a certain plane, and a linear interpolation along an axis not included in that plane can be executed in synchronization, and this combined interpolation is called helical interpolation.

⟨Command format⟩

(a) For XY plane

$$\boxed{G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\cdots\cdots Y\cdots\cdots\left\{\begin{matrix}R\cdots\cdots\\I\cdots\cdots J\cdots\cdots\end{matrix}\right\}Z\ (\alpha,\ \beta)\ \cdots\cdots F\cdots\cdots;}$$

(b) For ZX plane

$$\boxed{G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Z\cdots\cdots X\cdots\cdots\left\{\begin{matrix}R\cdots\cdots\\K\cdots\cdots I\cdots\cdots\end{matrix}\right\}Y\ (\alpha,\ \beta)\ \cdots\cdots F\cdots\cdots;}$$

(c) For YZ plane

$$\boxed{G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Y\cdots\cdots Z\cdots\cdots\left\{\begin{matrix}R\cdots\cdots\\J\cdots\cdots K\cdots\cdots\end{matrix}\right\}X\ (\alpha,\ \beta)\ \cdots\cdots F\cdots\cdots;}$$

(d) For X α plane

$$\boxed{G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\cdots\cdots \alpha\cdots\cdots\left\{\begin{matrix}R\cdots\cdots\\I\cdots\cdots J\cdots\cdots\end{matrix}\right\}Z\ (\beta)\ \cdots\cdots F\cdots\cdots;}$$

(e) For Z α plane

$$\boxed{G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Z\cdots\cdots \alpha\cdots\cdots\left\{\begin{matrix}R\cdots\cdots\\K\cdots\cdots I\cdots\cdots\end{matrix}\right\}Y\ (\beta)\ \cdots\cdots F\cdots\cdots;}$$

(f) For Y α plane

$$\boxed{G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Y\cdots\cdots \alpha\cdots\cdots\left\{\begin{matrix}R\cdots\cdots\\J\cdots\cdots K\cdots\cdots\end{matrix}\right\}X\ (\beta)\ \cdots\cdots F\cdots\cdots;}$$

(g) For Xβ plane

$$\boxed{G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\cdots\cdots \beta\cdots\cdots\left\{\begin{matrix}R\cdots\cdots\\I\cdots\cdots J\cdots\cdots\end{matrix}\right\}Z\ (\alpha)\cdots\cdots F\cdots\cdots;}$$

(h) For Zβ plane

$$G18 \left\{\begin{matrix}G02\\G03\end{matrix}\right\} Z\cdots\cdots\beta\cdots\cdots \left\{\begin{matrix}R\cdots\cdots\\K\cdots\cdots I\cdots\cdots\end{matrix}\right\} Y(\alpha)\cdots\cdots F\cdots\cdots ;$$

(i) For Yβ plane

$$G19 \left\{\begin{matrix}G02\\G03\end{matrix}\right\} Y\cdots\cdots\beta\cdots\cdots \left\{\begin{matrix}R\cdots\cdots\\J\cdots\cdots K\cdots\cdots\end{matrix}\right\} X(\alpha)\cdots\cdots F\cdots\cdots ;$$

Where α is one of the linear 4th axis, β is one of the linear 5th axis U, V, or W. If no 4th axis or 5th axis is programmed in (d) through (i), they are regarded as equal to (a), (b), and (c).

Example of Programming

```
G17  G03  X0  Y100. R100. Z90. F10 ;
```

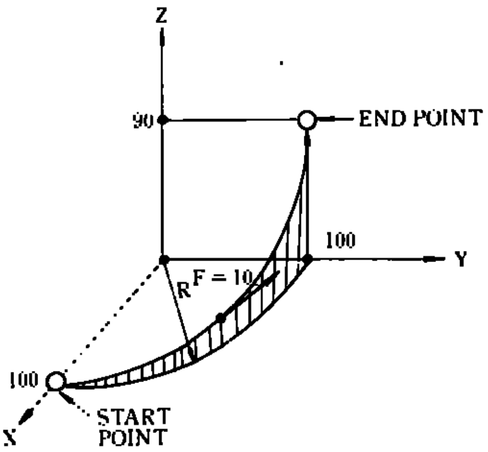


Fig. 2.4.9 Helical Interpolation

NOTE

1. The circular arc should be within 360°.
2. As long as above command format (a) is satisfied, the start and end points can be taken at any time.
3. The feedrate F means the 3-dimensional tangential speed on the plane of circular interpolation and z-axis.
4. Tool radius compensation C and be applied only to the circular path on the plane of circular interpolation.

2.5 FEED FUNCTIONS

2.5.1 Rapid Traverse Rate

The rapid traverse motion is used for the motion for Positioning (G00) and for the motion for Manual Rapid Traverse (RAPID).

This rapid traverse rate is specified by the machine tool builder, and set to each axis as a parameter.

(1) Setting range

For each axis, rapid traverse rates can be set at some suitable multiple of 1mm/ min (or deg/min).

The maximum programmable rapid traverse rate is 240,000mm/min.

NOTE Machine tools have their own optimum rapid traverse rates.
Refer to the manual provided by the machine tool builder.

(2) Motion path

Motion paths are normally not straight because the end points of each of the axes are reached at different times by independent axis motion.

(3) Override

For override rapid traverse rates, Fo, 25%, 50% and 100% of the basic rapid traverse rates, are available. Fo is a constant feedrate set by a parameter (Pm2447).

2.5.2 Feed

With six digits following an address character F, tool feedrates per minute (mm/min) are programmed at linear interpolation (G01) or circular interpolation (G02, G03).

(1) Programmable range of feedrates can be reduced to 1/10 by parameter.
See the following table.

Table 2.5.1

	Normal (pm 2004 D0 =0)		F 1/10 (pm 2004 D0 =1)	
Input	Format	F command range (Feedrates per minute)	Format	F command range (Feedrates per minute)
Micron	F60	F1 to F240000. mm/min	F61	F0.1 to F240000.0 mm/min
Submicron	F51	F0.1 to F24000.0 mm/min	F52	F0.01 to F24000.00mm/min
Sub-submicron	F42	F0.01 to F2400.00 mm/min	F43	F0.001 to F2400.000mm/min
Inch	F41	F0.1 to F9448.8 in/min	F42	F0.01 to F9448.81 in/min

1) F command 1/10 function does not affect F1 digit command speed.
2) Changing pm2004 D0 becomes effective at resetting.
3) F command 1/10 function does not affect F command at solid tapping.
4) F command 1/10 function does not affect E feeding at canned cycle.
The command format is the same as that of F command.
5) When F1/10 is used, the minimum unit of system variables for E and F command values are lowered by 1 digit.
Example) At command in millimeters

	Normal	F1/10
F command value system variable (1) =	1mm/min	0.1mm/min

6) When F1/10 is used, the format of E and F command value macro system variables and arguments (E, F) at macro calling add another digit in the decimal section.
Example) At command in millimeters
G65 P1 F1234 → #9 = 123.4

(2) Maximum feedrate

The maximum feedrate shown in Table 2.5.1 is subject to the performance of the servo system and the machine system. In this case, set the maximum limit by parameter Pm2800. Whenever a feedrate exceeding the maximum limit is commanded, the feedrate is clamped at the set maximum limit value.

(3) Simultaneous two-axis control

F command for linear and circular interpolations specifies feedrates in a direction tangential to the motion path.

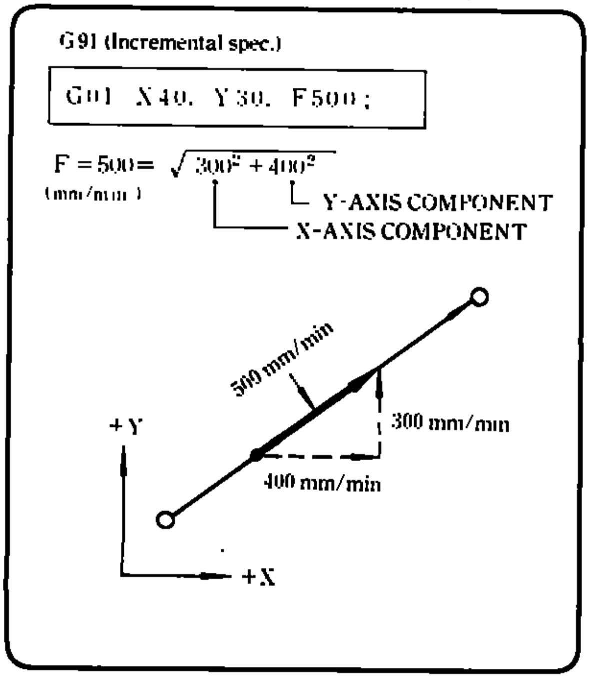


Fig. 2.5.1 For Linear Interpolation

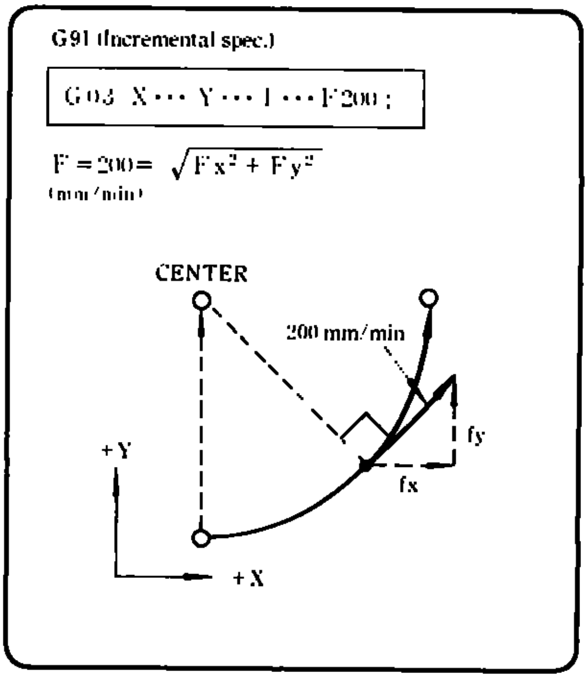


Fig. 2.5.2 For Circular Interpolation

(4) Simultaneous three-axis control

F command for linear interpolation specifies feedrates in a direction tangential to the motion path.

```
G01 X··· Y··· Z··· F400;
```

$$F = 400 = \sqrt{Fx^2 + Fy^2 + Fz^2}$$

(mm/min)

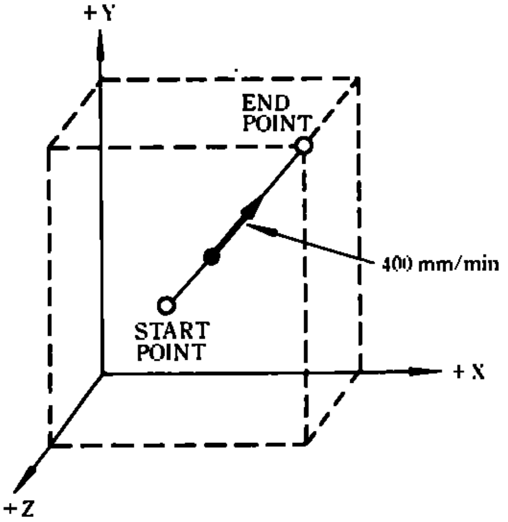


Fig. 2.5.3

※(5) Simultaneous four-axis control

F commands for linear interpolations specify feedrates also in a direction tangential to the motion path.

$$F\ (\text{mm/min}) = \sqrt{Fx^2 + Fy^2 + Fz^2 + F\alpha^2}$$

※(6) Simultaneous five-axis control

F commands for linear interpolations specify feedrates also in the direction tangential to the motion path.

$$F\ (\text{mm/min}) = \sqrt{Fx^2 + Fy^2 + Fz^2 + F\alpha^2 + F\beta^2}$$

(7) F command for interpolation on linear axis and rotary axis specifies feedrates in the direction tangential to the motion path.

Where G91;

G01 Z10. C60. F100.;

a) Metric input (F60)

Distance= $\sqrt{\underset{\substack{\uparrow\\ \text{Z-axis component}}}{10000^2}+\underset{\substack{\uparrow\\ \text{C-axis component}}}{60000^2}}$ =60827.625

Time=60827.625/100000=0.6082(min) =36.5(s)

b) Inch input (F52)

Distance= $\sqrt{\underset{\substack{\uparrow\\ \text{Z-axis component}}}{100000^2}+\underset{\substack{\uparrow\\ \text{C-axis component}}}{60000^2}}$ =1166190.0379

Time=1166190.0379/1000000=1.166(min) =69(s)

(8) The feedrate by rotary axis single command is determined in accordance with the unit of input system. In case of inch input, the feedrate is as shown below by parameter setting.

Table 2.5.2 Feedrate at Inch Input

		Format	pm2004 D7=0	pm2004 D7=1
Metric Output	Metric input	F60	1=1 deg/min	1=1 deg/min
	Inch input	F41	1=1 deg/min	1=0.1 deg/min
Inch Output	Metric input	F60	1=1 deg/min	1=1 deg/min
	Inch input	F51	1=1 deg/min	1=0.1 deg/min

pm2004 D7=0 : When the feedrate of rotary axis single command F41 at in input, 1=1 deg/min, F51, 1=1 deg/min

=1 : When the feedrate of rotary axis single command F41 at in input, 1=0.1 deg/min, F51, 1=0.1 deg/min

NOTE

1. If F0 is programmed, it is regarded as a data error. (alarm code "370")
2. Do not program F commands with minus numerals, otherwise correct operation is not guaranteed. (e. g. F-250 ← wrong)

2.5.3 F1-digit Feed *

(1) Specification of a value of 1 to 9 that follows F selects the corresponding preset feedrate.
Set the feedrate of each of F1 to F9 to the setting number shown in Table 2.5.3.

(2) By operating the manual pulse generator when F1−DIGIT switch is on, the feedrate of F1−digit command currently specified may be increased or decreased. Set the incremental or decreased value per pulse (F1−digit multiply) to the parameters listed in Table 2.5.4.
As a result of this operation, the contents of the setting number of the F1−digit feedrate are changed.

Table 2.5.3
F Command and Setting No.

F Command	Setting No. of F1-digit speed
F 1	Pm 0820
F 2	Pm 0821
F 3	Pm 0822
F 4	Pm 0823
F 5	Pm 0824
F 6	Pm 0825
F 7	Pm 0826
F 8	Pm 0827
F 9	Pm 0828

Setting "1" = 0.1 mm/min or
0.01 in./min

Table 2.5.4
F Command and Parameter No.

F Command	Setting No. of F1-digit multiply
F 1	Pm 2111
F 2	Pm 2112
F 3	Pm 2113
F 4	Pm 2114
F 5	Pm 2115
F 6	Pm 2116
F 7	Pm 2117
F 8	Pm 2118
F 9	Pm 2119

Setting "1" = 0.1 mm/min/pulse

(3) Upper limit of feedrate

Set the maximum feedrate of F1−digit designation to the following parameter. If a value greater than the usual maximum feedrate (the contents of Pm2800) is set, it is clamped by the contents of Pm2800.

Table 2.5.5 Parameter No. for Maximum Feedrate

Parameter No.	Meaning
Pm2865	Max speed of F1 to F4
Pm2866	Max speed of F5 to F9

NOTE

1. When the 1-digit numerals are set to the parameters pm0820 to pm0828, and pm2004 D0=0, the feedrate on the screen is displayed as "0". However, the machine moves in units of 0.1 to 0.9 mm/min or 0.01 to 0.09 in/min
2. If F0 is specified, alarm "370" will occur.
3. When DRY RUN switch is on, the rate of dry run is assumed.
4. For F1-digit specification, the feedrate override feature is invalid.
5. The feedrate stored in memory is retained after the power is turned off.
6. For the variable command of micro-program F1-digit command is possible.
7. F1-digit feed is specified in inches when inch is used by parameter setting, but the maximum feedrate is specified in millimeters only.

2.5.4 Feedrate per Minute

Feedrate per minute of feedrate per revolution can be selected by switching G93 and G94.

2.5.4.1 Feedrate per minute (G94)

G94 determines tool feedrates per minute (mm/min) for the digits following the address character F.

2.5.5 Solid Tap Mode (G93) *

The following G codes are specified to classify tapping by solid tap function or conventional tapping.

(1) G93: Solid Tap Mode

The tap cycle (G84/G74 commands) performs the solid tap. It is fed by command for feedrate per revolution. This mode is only for solid tapping.

(2) G94: Solid Tap Mode Cancel

This cancels the solid tap mode and changes to the conventional mode. The following tap cycle performs conventional tapping. It is fed by command for feedrate per minute.

NOTE G93, G94 is modal with G code of 05 group. G94 is selected at power ON/resetting.

2.5.6 Automatic Acceleration and Deceleration

Accel/decel for rapid traverse and cutting feed are automatically performed.

2.5.6.1 Accel/decel of rapid traverse and manual feed

In the following operation, the pattern of automatic accel/decel is linear.

- Positioning (G00)
- Manual rapid traverse (RAPID)
- Manual continuous feeding (JOG)
- Manual HANDLE feeding (HANDLE)

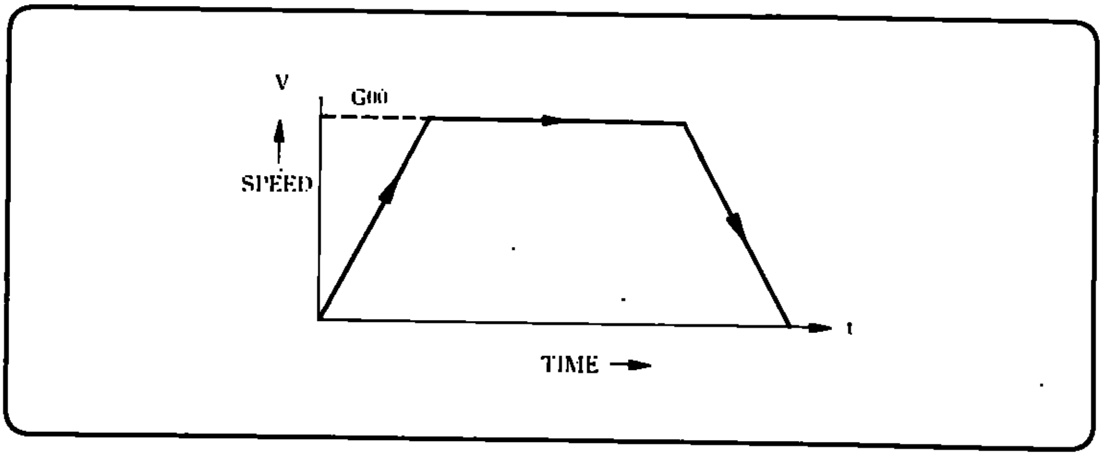


Fig. 2.5.4 Linear Accel/decel Speed

NOTE Rapid traverse rate and accel/decel constant of rapid traverse rate can be set by parameter. (Pm2461 –, Pm2801 –)

2.5.6.2 S-curve accel/decel ※

At positioning (G00), the above linear accel/decel can be changed to S-curve accel/decel.

Using S-curve accel/decel enables high-speed accel/decel positioning without causing machine vibration.

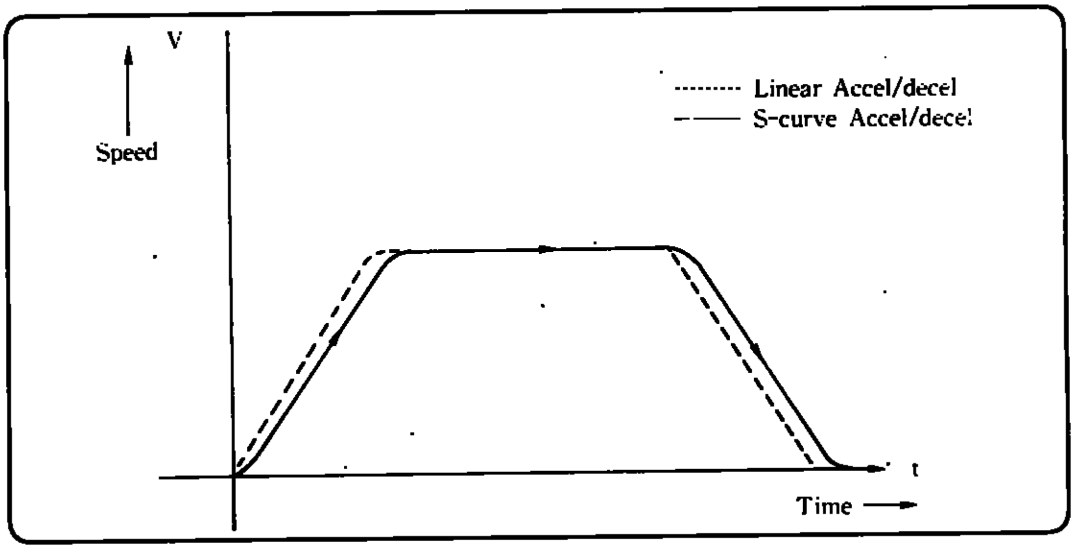


Fig.2.5.5 S-curve Accel/decel

Parameter

pm2591 : X-axis high-speed feeding accel/decel S-curve
(Setting range : 0 to 20 degrees)

pm2592 : Y-axis high-speed feeding accel/decel S-curve
(Setting range : 0 to 20 degrees)

pm2593 : Z-axis high-speed feeding accel/decel S-curve
(Setting range : 0 to 20 degrees)

pm2594 : 4th axis high-speed feeding accel/decel S-curve
(Setting range : 0 to 20 degrees)

pm2595 : 5th axis high-speed feeding accel/decel S-curve
(Setting range : 0 to 20 degrees)

S-curve accel/decel is provided with time constant (dwell time) in each axis and can be set from 0 to 20.

Set value 0 : Exactly the same speed change as linear accel/decel.
1 to 20(N) : S-curve accel/decel with 4 * N msec time constant can be obtained. Time constant is up to 60msec.

Minus setting : "0"
Setting exceeding 20 : "20"

2.5.6.3 Cutting feed accel/decel

(1) Automatic acceleration and deceleration of feed motion (G01 to G03) are in the exponential mode.

(2) Feedrate time constants and feedrate bias are set by parameters. During tapping, time constants and bias other than for normal feedrate can be set by parameters.
(Pm2501−, Pm2511−, Pm2821−, Pm2831−)

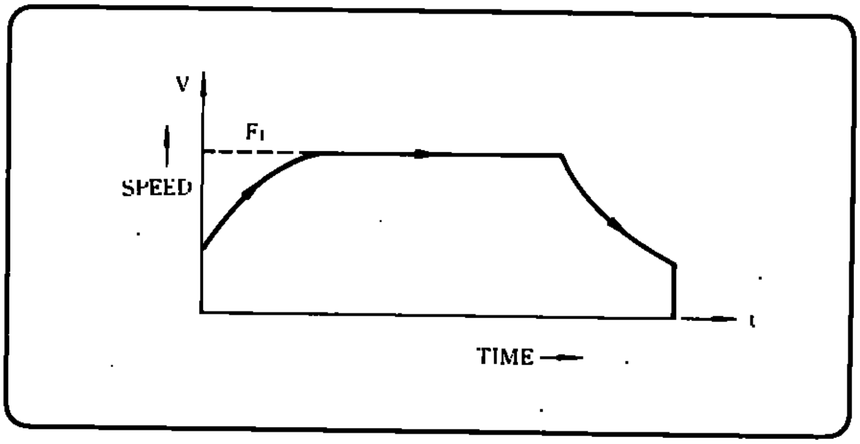


Fig. 2.5.6 Exponential Accel/decel Speed

NOTE Automatic accel decel parameters are set to the optimum values for respective machines. Do not change the setting unless required for special purposes.

2.5.7 Override

For details on override, refer to command or reference manuals issued by your machine tool builder.

2.5.7.1 Feed rate override

(1) In the automatic operation mode (TAPE, MEM, or MDI), the feedrate instructed by the F code can be overridden in 21 steps from 0 to 200 % in increments of 10%.

(2) However, the feed by tapping cycle G74 or G84 is as instructed by the F code, and cannot be overridden.

(3) If the OVERRIDE CANCEL switch is ON, the feed is not affected by this selection switch and runs as instructed by the F code.

Table 2.5.6 Feedrate Override

STEP	%	STEP	%
0	0	11	110
1	10	12	120
2	20	13	130
3	30	14	140
4	40	15	150
5	50	16	160
6	60	17	170
7	70	18	180
8	80	19	190
9	90	20	200
10	100	—	—

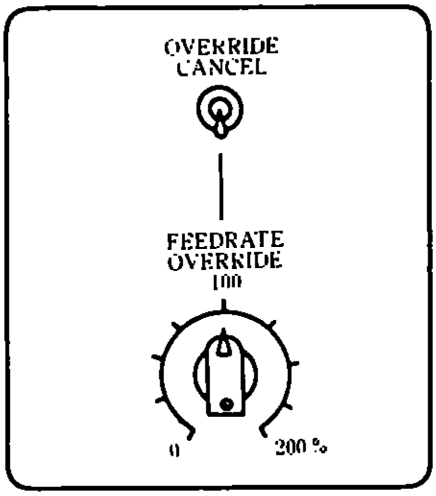


Fig. 2.5.7 Rapid Traverse Rate Override Switch

2.5.7.2 Rapid traverse rate override switch

(1) This switch is used to adjust the traverse rate by F_0, 25, 50 and 100%. 100% rate is the rapid traverse rate set by parameters Pm2801 and above.

(2) The switch is effective in both automatic operation including G00 command and in manual operation (RAPID mode).

(3) Fo is set by parameter Pm2447.

NOTE Fast forward speed can be overridden in six steps (100%, 50%, 25%, F_0, F_1, F_2).
F_1 and F_2 speeds are defined by parameters (Pm2448, Pm2449).
This function is optional. For details, refer to command or reference manuals issued by your machine tool builder.

2.5.8 Dwell (G04)

G04 P······ ;

(1) This command interrupts feed for the length of time designated by the address P. The same can be accomplished by commanding address X instead of address P.

(2) Dwell is programmed as an independent block.

(3) The maxinmum length of time which can be designated with address P is as follows.

Table 2.5.7

Format	Dwell time (P programmable range)
P63	0~999999.999 sec

The value is not dependent on metric inch input or metric · inch output.

Example of Programming

```
G04  P2500 ;
```

Dwell time : 2.5 sec.

(4) Two types of dwell can be selected by parameter:(Pm4015 D3).
Dwell when the specified value in the command block before the dwell block is identified by lag pulses of servo, or dwell on completion of pulse distribution.

2.5.9 Speed Control Reference

For speed control of interpolation, accel/decel control is executed by a time constant in start and stop.
Part program is created without regard to the control.
However, cutting in a corner may result in rounding by follow-up error between NC command and actual machine movement. To eliminate the rounding, add G04 (dwell) instruction in the corner for positive deceleration. Y-axis moves before X-axis has fully decelerated.

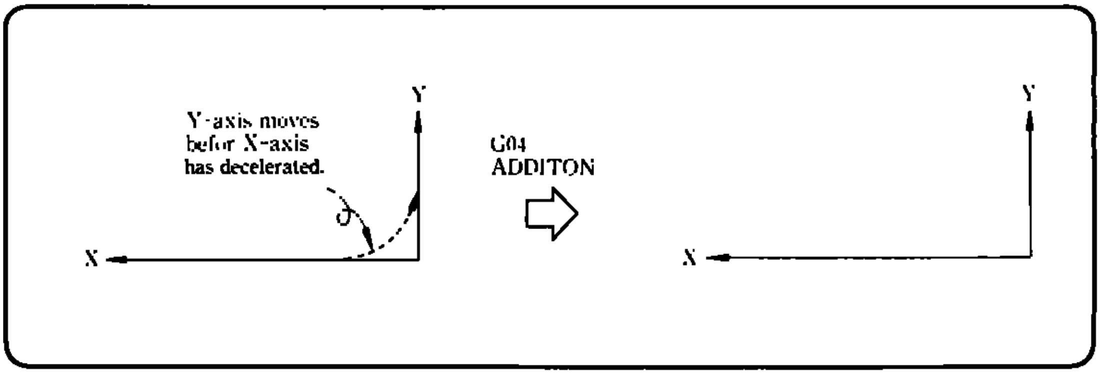


Fig. 2.5.8 Adding G04 Instruction in Corner

To control the follow-up error(rounding) in the program, use G09 (exact stop) in the program.

2.5.9.1 Exact stop (G09)

If a program contains code G09 in a cutting command, the next block is started in the error detection ON mode. (See NOTE 1 on next page.)
Use this function to obtain a sharp corner. G09 is a nonmodal code and its effect continues only in the block where it is specified.

2.5.9.2 Exact stop mode (G61, G64)

After G61, cutting command blocks are processed in the error detection ON mode until the code is canceled by G64.

NOTE

1. Error detection ON mode
   In the error detection ON mode, the number of lag pulses due to servo delay is checked after linear or circular interpolation command pulses were distributed, and the next block is started only when the number has reduced to a permissible level.
2. Error detection OFF mode.
   After a normal linear or circular interpolation command out of the G09 or G61 mode, the next block is started immediately after the pulses are distributed. Because of servo delay, corner trajectory is rounded. This is the error detection OFF mode.
3. Error detection ON/OFF for rapid traverse. The error detection ON/OFF mode for rapid traverse is controlled by G00 and G06 only, and G09, G61, and G64 are disregarded.

2.6 REFERENCE POINT RETURN

The reference point means a fixed position on the machine.

2.6.1 Automatic Reference Point Return (G28)

(1)

```
G28  X……Y……Z…… (※ α……β……) ;
```

This command allows the machine to be restored to the reference point. The machine is automatically restored to the reference point after being positioned at the specified position by rapid feed.

(2) This operation can be performed simultaneously up to three axes (※ up to five axes) in one system. However, any axis for which coordinate command is omitted does not move.

Example of Programming

```
(G90  G91) G28 X…… Y…… Z…… ;
```

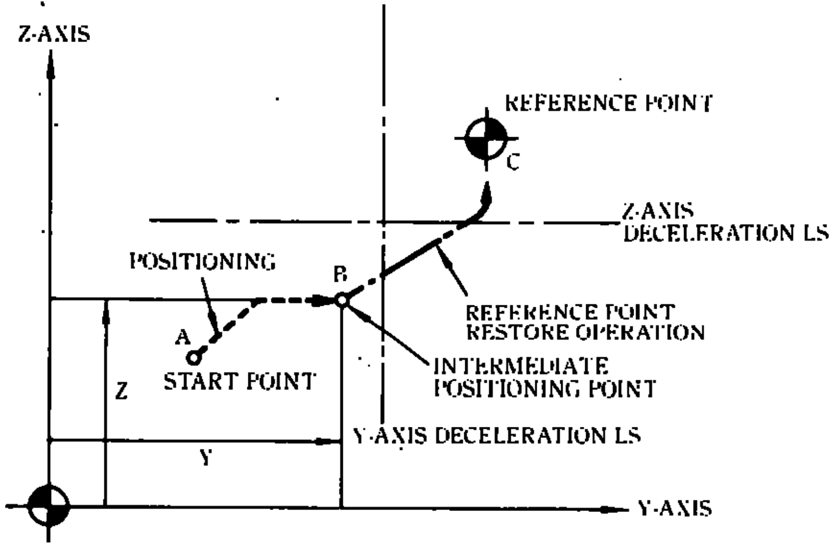


Fig. 2.6.1

(3) "Reference point return operation" means the same operation as a series of operations in which the movement is begun by manual reference point return operation and terminated when the reference point is restored.

(4) Reference point return is accomplished in two ways:
- Low-speed reference point return (a deceleration LS is used)
- Combination of low-speed reference point return the first time after power-on and high-speed reference point return the second and subsequent times (a deceleration LS is not used because the reference point is stored in the memory).

(5) **High-speed reference point return (Refer to parameter Pm4003 D6, D7.)** High-speed reference point return may be used in place of the automatic reference point return described above. In this case, movement is as shown below.

① After being positioned at intermediate positioning point B, the machine is positioned directly at the reference point by rapid feed.

② In this case, even if point B is located outside the reference point returnable area, the machine can be returned to the reference point.

③ However, high-speed reference point return is only possible for the axes for which normal reference point return operation has been completed after power-on by manual reference point return or G28 command.

④ If there is at least one axis in the G28 command for which normal reference point return operation has not been completed after power-on by manual reference point return or G28 command, normal reference point return operation is executed for all axes instructed by G28.

⑤ Automatic high-speed reference point return is only valid for G28 command, and does not affect the operation of manual reference point return.

(6) In the case of manual high-speed reference point return, axial movement in the RAPIDor JOG mode cannot be performed unless the reference point return switch is turned off after reference point return is completed.

(7) Rotation axis auto return to reference point

A rotation axis can also be returned to the reference point as well as a linear axis. After moving ±360.000° or further from the original reference point, the rotation axis returns to the reference position by rotating a minimum in the home positioning direction.

For example, from positions A and B, the axis returns to positions A' and B', respectively. (The home positioning direction is determined by pm4002 (D3, D4).

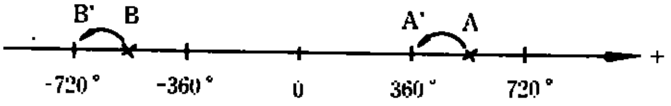


Fig.3.6.2 **Return of rotation axis to reference point (When reference point return direction is negative)**

NOTE

❶ If G28 is instructed during the tool radius compensation mode (G41, G42) or canned cycle, alarm "182" results. As a rule, G28 must be instructed in a tool radius compensation C or canned cycle canceled state.

❷ If G28 command is made during mirror image (M95), the following results.

Table 2.6.1

Parameter Pm4001 D2	Movement
0	Mirror image is applied at an intermediate point. Movement to the reference point is not affected by mirror image.
1	Alarm "127" results.

❸ For a G28-instructed block in tool position offset, the machine is moved to an offset-applied position by moving up to an intermediate point and to an absolute position (where no offset is applied) by moving up to the reference point.

However, movement by a move command by following parameters becomes as follows:

When parameter Pm4011 D1=0,

Pm4010 D6 = 0 Moved as instructed.

Pm4010 D6 = 1 : Moved with offset applied.

When parameter Pm4011 D1=1,

Pm4010 D7 = 0 . Moved as instructed.

Pm4010 D7 = 1 : Moved with offset applied.

④ Tool length offset can be canceled by G28 command. Whether or not it should be canceled is specified by a parameter. This specification is only effective when Z-axis command is present in the same block as G28.

Table 2.6.2 Selection of Cancel in Tool Length Correction

Parameter Pm4010 D7	Movement
0	Tool length offset is canceled during reference point return at reset. The H code is cleared to "0." However, tool length offset G code is retained.
1	Tool length offset is not canceled during reference point return at reset. The H code and tool length offset G code are retained. If an offset is already specified, be sure to cancel the offset before issuing the G28 command.

⑤ Reference point return valid/invalid can be selected for each axis. If reference point restore invalid is instructed for an axis in the G28 block, alarm "241" results. Refer to parameters Pm4002 D0 to D4.

⑥ When a cycle is started after power-on without performing all axis reference point restore, alarms "0411" to "0415" can be issued. Whether or not such alarms should be issued is selected by a parameter. For details, refer to parameter Pm4001 D6.
If set for "no alarm" in the above, whether or not reference point restore is required can be specified for each axis with a parameter. For details, refer to parameters Pm4018 D0 to D4.

⑦ When any move command other than G28 is executed after power-on without performing reference point return, alarms "0091"-"0099" can be issued. Whether or not such alarms should be issued is specified by a parameter. For details, refer to parameters Pm4004 D0-D7 and Pm4005 D0.

⑧ The absolute coordinate value of the axis instructed by the G28 block is stored in memory as an intermediate point. For axes not instructed by the G28 block, the intermediate point of the previously-instructed G28 is stored in memory as the intermediate point for that axis.

⑨ If M, S, T or B command is made in the same block as G28, the machine is moved to the reference point regardless of whether FIN processing is completed or not. Therefore, DEN output is made at the reference point.

⑩ Reference point return and machine lock intervention
There are two types of machine lock intervention operations: machine lock is turned ON after the machine is stopped by feed hold during movement or the machine lock is turned OFF after the machine is stopped by feed hold once again.
Table 2. 6. 3 shows how the machine is operated by machine lock intervention.

Table 2.6.3 Movement by Machine Lock Intervention

		Machine lock intervention up to an intermediate point	Machine lock intervention during movement to reference point
Machine lock OFF→ON	Low speed type	Stopped after being moved to an intermediate point, but not to reference point. No display is given.	Display keeps moving continually to detect the operation of the deceleration L.S. (The deceleration LS does not operate due to machine lock.)
	High-speed type		Axial movement does not occur at the point where machine lock is applied. After that, the display moves to the reference point of the current value display (universal). (Without axial movement)
Machine lock OFF→ON →OFF	Low-speed type	Machine moves to intermediate point on workpiece coordinate system and to reference point on machine coordinate system. Deviated reference point is displayed on the work coordinate system but it will never be deviated on the machine coordinate system.	Moved up to the reference point.
	High-speed type		The position is displaced by a quantity the machine lock applies. Therefore, although the universal of the current value display becomes the reference point, the machine is not at the reference point.

⑪ For the first reference point restore after power-on, attention should be paid to the position of the deceleration dog. Refer to NOTE in Par. 3. 6. 1. 1."Manual Reference Point Return".

2.6.2 Reference Point Return Check (G27)

(1) This function checks whether a part program which is created to return to the reference point after starting from the machine reference point, correctly returns to the reference point.

(2)

```
G27  X······Y······Z······ (※α······β······) ;
```

When this command is given, the machine is positioned at the instructed point simultaneously for three axes (※simultaneously for five axes) by rapid feed, then it is checked whether that position is the reference point. However, for axes for which coordinate command is omitted, positioning and check are not executed.

(3) If that position matches the reference point, the reference point return complete lamp lights. If all of the specified axes match the reference point, automatic operation is continued. If there is any axis for which the position is not matched, restore position error (alarm "2072" – "2079") results, and the automatic operation is interrupted. (The cycle start lamp goes out.)

(4) If G27 is instructed during tool offset, the machine is positioned at a point offset by the corrected quantity, so that the position does not match. Therefore, G27 can only be instructed after tool offset is canceled. Tool position offset and tool length offset cannot be canceled by G27 command.

(5) The reference point means a point inherent to the machine to which position the machine can be restored by "manual reference point return" or "G28 automatic reference point return." For details, refer to Par. 3. 4. 1. 1,"Manual Reference Point Return."

(6) Mirror image is valid in the movement direction by G27 command. To avoid an unmatch error, be sure to specify G27 in the M94 (mirror image OFF) mode.

(7) No check is made if G27 is executed with machine lock. If there is any one axis for which machine lock (e. g., disregarded the Z-axis) is ON, check is not made. For example, if X-axis is disregarded and G27 move command is made for X-axis, no check is made.

2.6.3 Return from Reference Point Return (G29)

(1) This function causes the machine, after being moved to the reference point by automatic reference point return (G28 or G30) to be returned to the original position by going back on the same path as reference point return.

G28 $\underbrace{\text{Y····· Z·····}}_{\text{B POINT}}$; A → B → C POINT (REFERENCE POINT)

⋮

G29 $\underbrace{\text{Y····· Z·····}}_{\text{D POINT}}$; C → B → D POINT

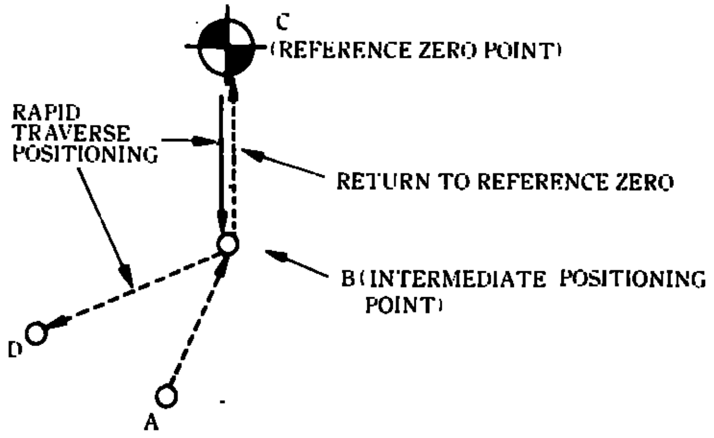


Fig. 2.6.2

(2) If G29 is instructed, the distance between B and C need not necessarily be considered in the program. Especially, if an incremental command is used, G29 is useful in returning to the original coordinate system after once being returned to the reference point.

(3) With G29, movement C — B and B → D is executed at rapid feed for three axes (※up to five axes) simultaneously. However, any axis for which coordinate command is omitted does not move.

(4) If G28 or G30 (refer to par. 2.6.4) is instructed a number of times, the coordinate value of point B for G29 movement becomes an intermediate point ultimately created by the latest G28 or G30.

(Example) Indicated by absolute command for easy understanding

```
                                       Intermediate point
                                       coordinate value
N21  G90  ;                              X   Y   Z
N22  G28  Z10.  Y20.           →      ( 0 ,  20. , 10. )
N23  G28  X30.  ;              →      (30. , 20. , 10. )
N24  G29  X-40.  Y-50.  ;

         G00  X30.  Y20.  ;
   =
         G00  X-40.  Y-50.  ;
```

Equivalent to these two blocks

Example of Programming

```
N31  G91 ;
N32  G28  Z······ ;
N33  G28  X20.  Y40. ;
N34  M06 ;
N35  G29  X40.  Y-40. ;
   ⋮
```

In the case of G91 command, valid for the command value of G29 (movement quantity from an intermediate point) and the intermediate point is not affected

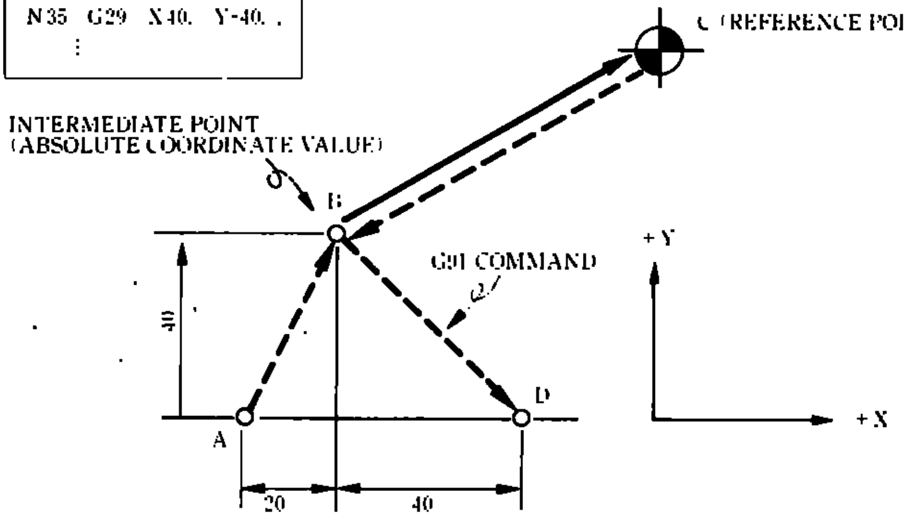


Fig. 2.6.3 B-Point Coordinate Value for G29 Movement

NOTE

❶ If G29 is instructed in tool offset mode (G41, G42) or canned cycle (G73, G74, G76, G77, G81 to G89), alarm "170" or "182" results.

❷ If G29 is instructed after power is turned ON without G28 or G30 being executed, alarm "0240" results.

❸ As a rule, the tool position offset must be canceled before G28 or G30 or G29 can be instructed. If instructed while the offset is applied, intermediate positioning point B is also offset and the tool passes point B'. Tool position offset and tool length offset cannot be canceled by G29 command.

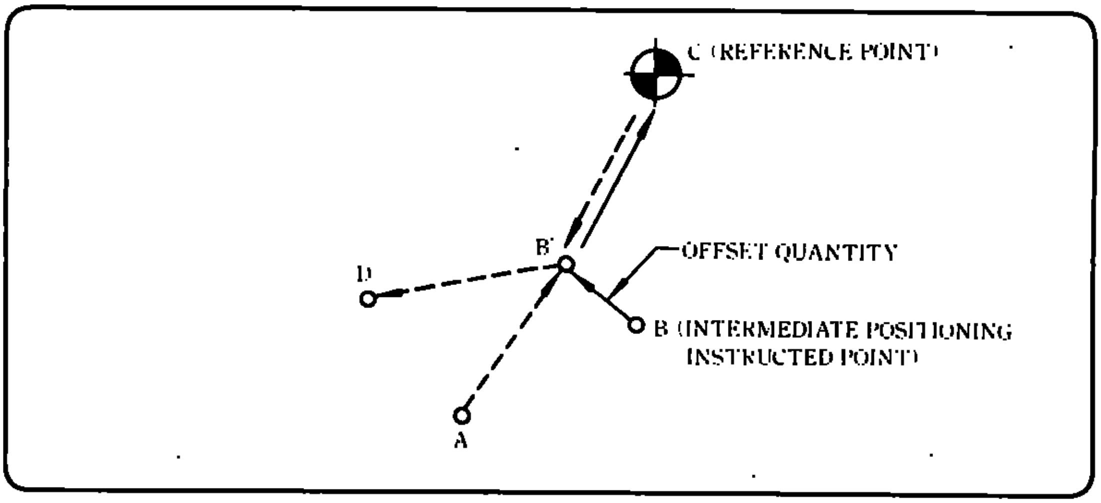


Fig. 2.6.4 Movement when Tool Position Offset is not canceled

❹ If G29 is instructed in mirror image (M95), the following results.

Table 2.6.4

Parameter Pm4001 D2	Movement
0	Mirror image is applied to intermediate point and command value.
1	Alarm "127" results.

❺ In the following cases, the intermediate point of G28 or G30 does not match the intermediate point of G29.
Therefore, avoid making commands or operations that cause such nonconformance.

- When the following is performed from the time G28 is completed to when G29 is instructed
  - Setting of coordinate system (G92, ZERO SET key operation)
  - Intervention of machine lock
  - Intervention of manual operation in manual absolute OFF
- When G28, G30, or G29 is instructed in a block after mirror image is canceled (M94) at a position different from the mirror image starting point.
- When G28, G30, or G29 is instructed after intervention of manual operation in manual absolute OFF.

2.6.4 Second to Fourth Reference Point Return (G30) ※

(1)

```
G30  Pn  X……Y……Z (※ α…… β……) ;
```

(Pn = P2, P3 ※ , P4 ※)

This command causes the machine to be positioned at the second, third, or fourth reference point after being positioned at the instructed intermediate point.

- P2: Second reference point
- P3: Third reference point ※
- P4: Fourth reference point ※

If Pn is omitted, the second reference point is selected by default.

(2) Any axis for which coordinate command is omitted does not move.

(3) The positions of each reference point are preset by parameters (listed below) as a distance from the first reference point.

P2: Pm6811 or above
P3: Pm6821 or above
P4: Pm6831 or above

⟨Program example⟩

```
G30  P3  X30.  Y50. ;
```

…… The X and Y axes are returned to the third reference point.

NOTE

- ❶ NOTE in Par. 2. 6. 1 "Automatic Reference Point Return(G28)" apply to the G30 command exactly as they are.
- ❷ If G29 is instructed after G30 command, the machine is positioned at the point specified by G29 after passing an intermediate point specified by G30. However, the intermediate point is only updated for the axes for which G30 is instructed.
- ❸ Before G30 command is executed, manual reference point return or normal reference point return by G28 command must have been completed after power-on. If there is any axis in G30 command for which reference point return is not completed, alarm "0240" may result.

2.7 COORDINATE SYSTEM

2.7.1 Types of Coordinate Systems (G54 to G59 ※, G92)

There are five types of coordinate systems as shown below.

(1) Reference coordinate system

This is a basic coordinate system set by G29, [ZERO SET] key, or automatic coordinate setting. Until any of these operations is performed after power-on, the power-on position is temporarily assumed to be the coordinate origin.

(2) Work coordinate system ※

This coordinate system is set by any command from G54 to G59. It is shifted from the reference coordinate system by a value set for that G code from work coordinate system software screen. Up to six work coordinate systems can be used.

(3) Local coordinate system ※

A coordinate system shifted from the given work coordinate system can be used by instructing G52 Q2. The local coordinate system is only valid when a work coordinate system is set.

(4) Machine coordinate system

This coordinate system is inherent to the machine determined by performing reference return, and is created on the reference point (0, 0, 0). If no coordinate setting is made after power-on, the power-on position is temporarily assumed to be the origin of the reference coordinate system.

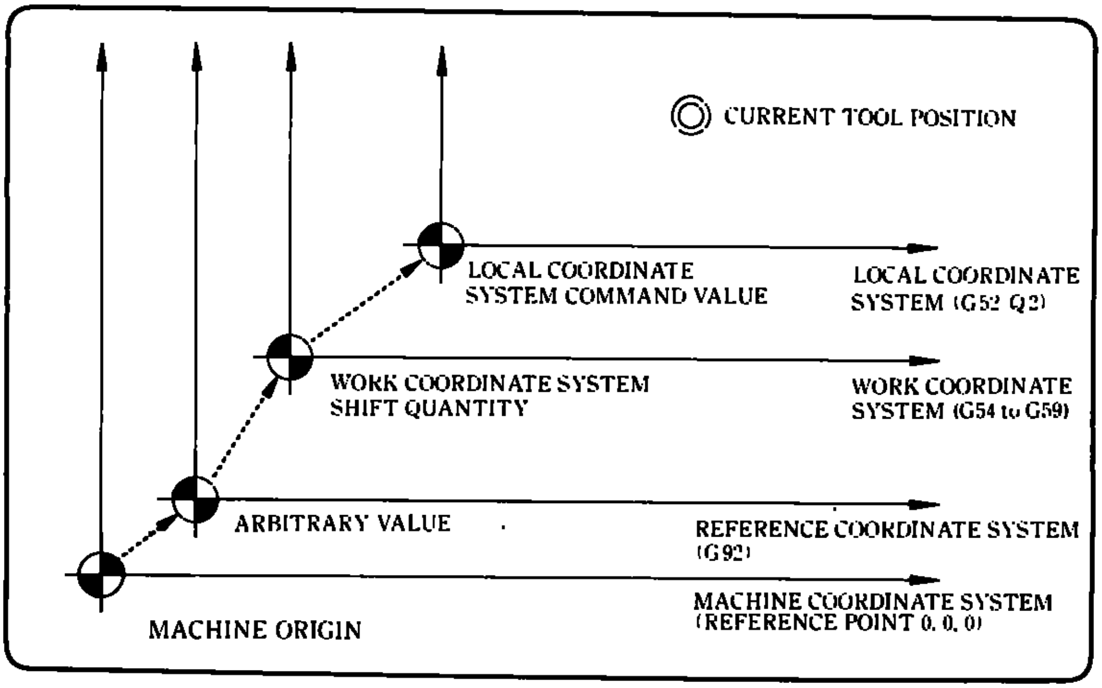


Fig. 2.7.1 Coordinate System

(5) Rotation of work coordinate system ※

The coordinate system can be rotated by a set angle around the origin (0, 0, 0) of the work coordinate system.

2.7.1.1 Setting reference coordinate system by G92

Coordinate systems must be set before move commands can be programmed. Once a coordinate system is set, one absolute coordinate system is determined, so that all subsequently issued absolute move commands are moved along the set coordinate system.

(1)
```
G92 X······Y······Z······ (※ α······ β······) ;
```

G92 is a command to specify the position of the "coordinate origin." With this command, the current tool position is set to the equipment as absolute coordinate points (X, Y, Z, ※ α, β). That is, it instructs the distance (a signed command) from the desired coordinate origin (0, 0, 0) on the program to the current tool position.

Example of Programming

```
G92 X500 Y300 Z400 ;
```

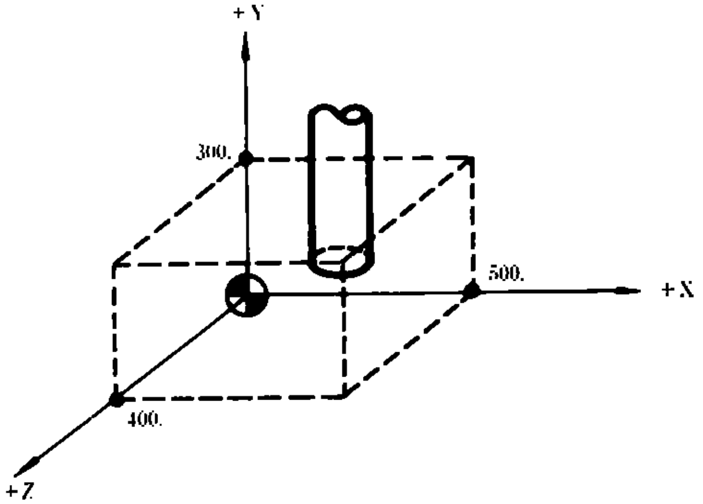


Fig. 2.7.2 Workpiece Coordinate System Setting by G92

(2) G92 is a G code of the non-modal group that is only effective in the instructed blocks. No other G codes nor the F,M,S,T, or ※B codes can be instructed in the same block.

NOTE

❶ As a rule, G92 must be instructed in states where each tool offset is canceled.

❷ When power is turned on, the current tool position is set at coordinates (0, 0, 0). Be sure to set coordinate systems before starting operation.

❸ The coordinate systems set are not affected by reset operation. To reset a coordinate system, perform any of the following:

- Operate the ZERO SET key. (Refer to Par. 3. 1. 3. 7. 1.)
- Write "G92 X0 Y0 Z0※α 0 β 0;" then execute it in the MDI mode.
- Momentarily turn the power OFF, then turn it on again.

2.7.1.2 Setting work coordinate systems by G54 to G59 ※

(1) Number of work coordinate system pairs can be added by options.

Table 2.7.1 Number of Pairs

Option 1	6 pairs
Option 2	54 pairs
Option 3	162 pairs

(2) Method of command

Table 2.7.2 Setting of WorK Coordinate Systems

Option 1	G54 J1 (P1)	G55 J1 (P1)	G56 J1 (P1)	G57 J1 (P1)	G58 J1 (P1)	G59 J1 (P1)
Option 2	G54 J1 (P1) to G54 J9 (P9)	G55 J1 (P1) to G55 J9 (P9)	G56 J1 (P1) to G56 J9 (P9)	G57 J1 (P1) to G57 J9 (P9)	G58 J1 (P1) to G58 J9 (P9)	G59 J1 (P1) to G59 J9 (P9)
Option 3	G54 J1 (P1) to G54 J27 (P27)	G55 J1 (P1) to G55 J27 (P27)	G56 J1 (P1) to G56 J27 (P27)	G57 J1 (P1) to G57 J27 (P27)	G58 J1 (P1) to G58 J27 (P27)	G59 J1 (P1) to G59 J27 (P27)

① Setting work coordinate systems (G54 to G59)

By G54 (G55, G56, G57, G58 or G59) J1 (J2 ··· J27) command, the subsequent programs move along the set work coordinate system. Address J can be replaced with address P. by switching parameters.

Pm4012 D7 = 0 : Address J

Pm4012 D7 = 1 : Address P

② G54 to G59 are modal commands.

③ G54 command is equivalent to G54 J1 or G54 J0 command.

④ Return to reference coordinate system (G52)

```
G52;
```

With the above command, it is possible to cancel the currently selected work coordinate system and return to the reference coordinate system.

⑤ The G52 command is a modal command.

⑥ If the work coordinate system shift quantity is changed after G54 to G59 is executed and a work coordinate system is selected, whether or not that shift quantity immediately becomes valid for operation (without G54 to G59 commands) is determined by a parameter (pm4012D0).

⑦ To change a coordinate system by G54 to G59 command, create a program so that a new coordinate system is entered in the G90 mode and then return to the reference cordinate system in the G90 mode.

⑧ Even if G54 to G59 is instructed while tool length offset or tool position offset is applied, the corrected quantity is not canceled. It is usually necessary to instruct G54 to G50 after canceling the tool length offset or tool position offset.

```
G43  Z0   H01 :
G54 :
G90  Z1000.
```

...... WORK COORDINATE SYSTEM POSITION Z100.

...... WORK COORDINATE SYSTEM POSITION Z1100.

Actual Z-axis movement quantity becomes 1400.

G54 shift quantity Z =-300.

Offset H01 = 100.

⑨ Do not normally use G92 in G54 to G59.
If G92 is instructed during execution in a coordinate system set by G54 to G59, G54 to G59 and reference coordinate system are shifted so that the current position becomes the position instructed by G92.

⑩ Be sure to instruct G54 to G59 in the G00 or G01 mode.
Command in any other mode results in alarm "322."

⑪ Temporary movement in machine coordinate system (G53)
A temporary movement on the machine coordinate system can be instructed by using G53 command. G53 is a non-modal G code.

(Program example)

```
G53 (G90) G00 X······Y······Z······(α······β······) :
```

NOTE 1. Commanding J2 or above in Option 1, J10 or above in Option 2, or J28 or above in Option 3 results in an alarm.

2. When the command opposite the address (J when P is selected, P when J is selected) selected by parameter pm4012 D7 is given by G54, the address is regarded as G54.
(Example) pm4012 D7=1 (at P selection) (Options 2 and 3)
Operates as G54 J2 X0 Y0; → G54 P1 X0 Y0;

(3) Program example using a work coordinate system

```
N1 G90 X100.  Y200. ;
N2 G54;
N3 X100.  Y300. ;
N4 X300.  Y200. ;
N5 G52;
N6 X0. Y0. ;
```

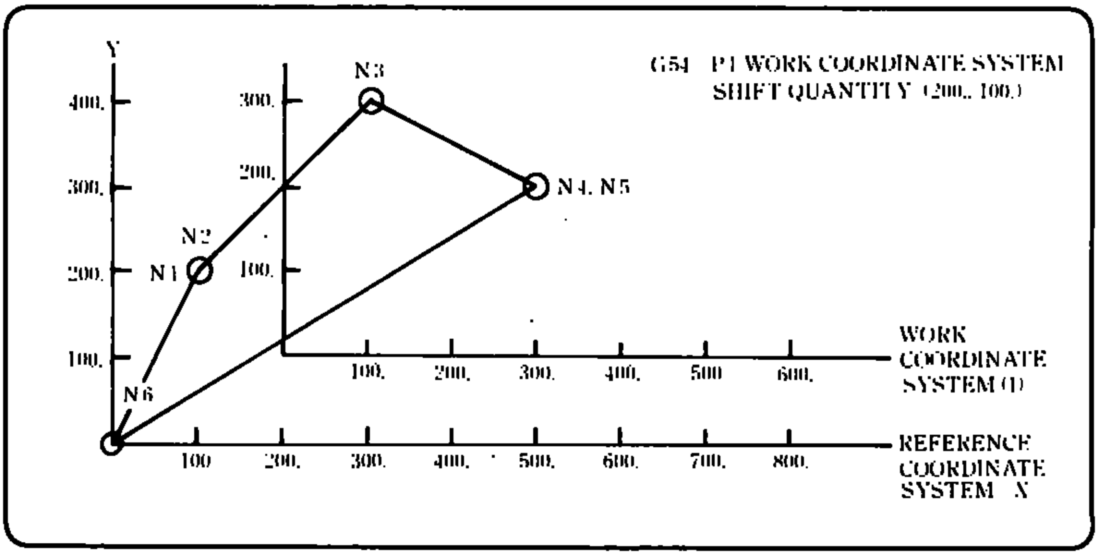


Fig. 2.7.3 Work Coordinate System Shift Quantity

(4) Changing the work coordinate system shift quantity with macroprograms

① The work coordinate system shift quantity (and external work coordinate system correction quantity) can be read by using system variables in the right side of the calculation formula.

② The values of the above can be changed by instructing the said system variable in the left side of the calculation formula.

③ For details on the relationship between system variables and work coordinate system shift quantities, refer to (C) Work coodinate system shift quantity in (3) System variables in Par. 2. 14. 2.

④ (Program example)

- `#116 = #2501 ;`

  G54 (P1) X-axis work coordinate system shift quantity is substituted for common variable #116.

- `#2511 = #4 ;`

  G54 (J2) X-axis work coordinate system shift quantity is erased, and the contents of local variable #4 are set.

(5) Changing the work coordinate system shift quantity by external data input.

① The work coordinate system shift quantity can be corrected by external data input signal.

② Upon input of an external correction quantity, all shift quantities of G54 (J1) to G59(J27)are changed to new shift quantities by adding the external correction quantity. In this case, the external work coordinate system correction quantity is corrected, without directly correcting the work coordinate system shift quantity.

〔Work coordinate system shift quantity on actual operation〕 =
〔External work coordinate system correction quantity〕
+ 〔Work coordinate system shift quantity〕

③ For the angle of rotation, work coordinate system shift by external data input must be corrected.

(6) Changing the work coordinate system shift quantity by programming (G10)

① The work coordinate system shift quantity can be corrected by G10 command.

```
G10  Q2  (or L2)  Pm  Jn  X……Y……Z (※: α……β……)
```

By instructing the above, the specified work coordinate system is corrected. The work coordinate system to be corrected can be specified by a combination of Pm and Jn.

- Pm …… is used to select G54 to G59.

  P1 = G54

  to

  P6 = G59

- Jn …… is used to select J1 to J27.

(Example)

P2 J3 → G55 J3

P4 J2 → G57 J2

② Omission of J or specification of J0 is assumed to be equivalent to J1.

③ If an incorrect value is specified for m or n, alarm "202" results.

NOTE

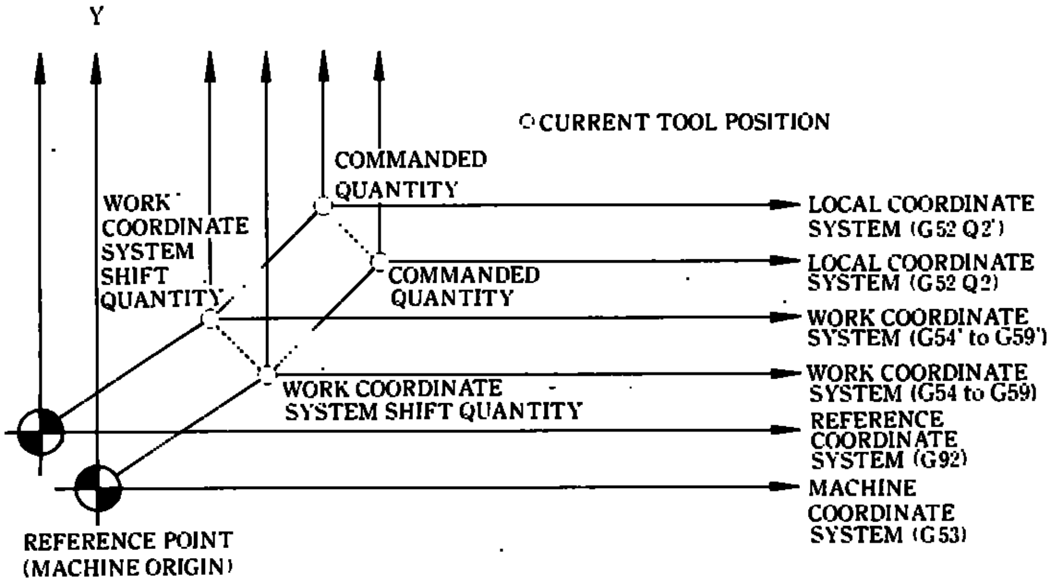
❶ • Without absolute position detector
When power is turned ON, the current tool position is set to coordinates 0(0, 0, 0). In other words, the current tool position is temporarily assumed to be the coordinate origin until the appropriate coordinate system is set.

❷ In a state where G92 (reference coordinate system) is not set, a work coordinate system is shifted from the origin of the machine coordinate system.

❸ In a state where G92 (reference coordinate system) is set, a work coordinate system is shifted from the origin of the reference coordinate system.

❹ If G92 is instructed in a state where a work coordinate system is set, the reference coordinate system is set so that the current tool position becomes the coordinate values instructed by G92, and at the same time the work coordinate system is shifted from the origin of that reference coordinate system.

❺ If work coordinate system cancel (G52) is instructed, the work coordinate system returns to the reference coordinate or machine coordinate system. The same applies when the local coordinate system is canceled.



NOTE The reference point and machine origin are located at the same position.

Fig. 2.7.4 Change of Work Coordinate System Shift Quantity

❻ The machine coordinate system, reference coordinate system, and work coordinate systems are not affected by reset.

❼ The reference coordinate system means a coordinate system that is automatically set by G92 or ZERO SET key.

❽ A move command can be executed in the same block as a work coordinate system setting command. In this case, movement is made in the work coordinate system.

2.7.1.3 Local coordinate system (G52Q2) ※

(1)

```
G52  Q2  X……Y……Z……(※α……β);
```

(α, β respectively indicate the 4th and 5th axes.) with this command, a coordinate system shifted from the work coordinate system by an instructed quantity is created.

This coordinate system is called the local coordinate system, and the tool is hereafter moved in this specified local coordinate system.

(2) Program example

```
N1 G90 G01 X100 Y200 F100;
N2 G54;
N3 X100. Y300.;
N4 G52 Q2 X300 Y200.;
N5 X200. Y100.;
N6 G52 Q2 X0 Y0;
N7 X0 Y0;
N8 G52;
N9 X0 Y0;
```

- Work coordinate system shift quantity

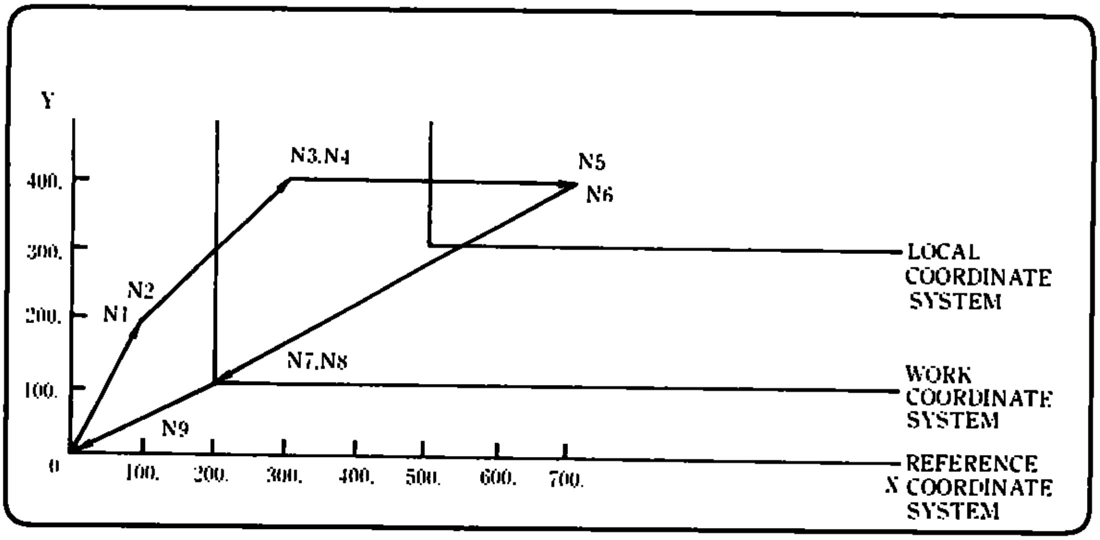


Fig. 2.7.5 Local Coordinate System
(WORK COORDINATE SYSTEM SHIFT QUANTITY (200. 100.))

(3) G52 Q2 X0 Y0 Z0 (*α0 β0) ;

By this command, the local coordinate system is canceled and returned to the work coordinate system.

(4) G52 ;

Returned to the reference coordinate system by this single-block command.

NOTE

❶ The G52 Q2 command is only effective when there is at least one work coordinate system set. If this command is issued when there is no work coordinate system set, alarm "321" results.

❷ Do not set a coordinate system by using the G92 command or ZERO SET key in a work coordinate system and local coordinate system state.

❸ Also see NOTE of Par. 2.7.1.1, "Setting reference coordinate system by G92".

❹ If this option is not added, G52 cancels work coordinate systems (G54 to G59).

❺ The local coordinate system is canceled by reset, and whether or not it returns to the work coordinate system is specified by parameter setting.

2.7.1.4 Machine coordinate system (G53)

(1) (G90) G53 X……Y……Z…… ;

With this command, only this block can be temporarily moved to a position (X, Y, Z) on the machine coordinate system.

G53 is a non-modal G code.

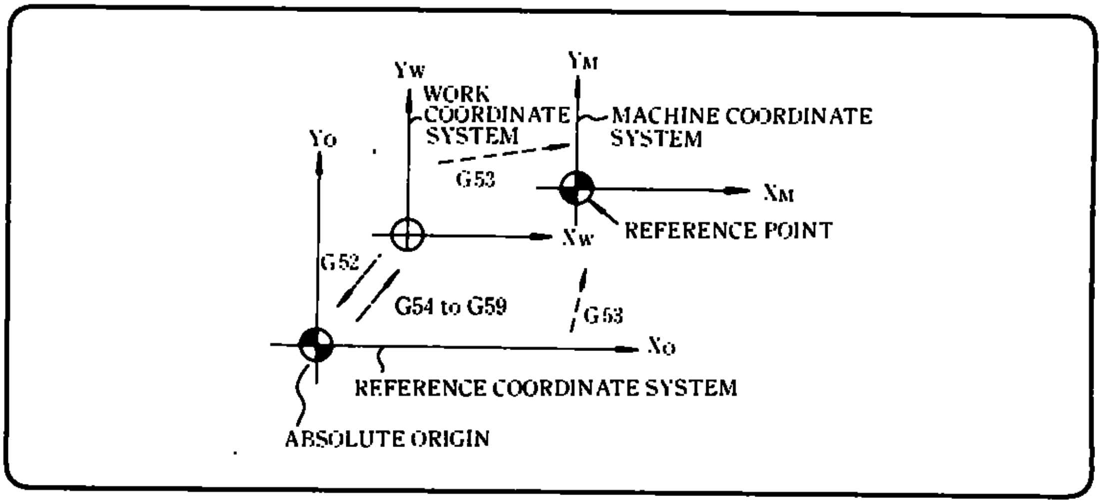


Fig. 2.7.6 Machine Coordinate System

(2) Program example

(Reference point restore operation)

```
N1 G92 X200. Y100. ;
N2 G54 G90 X100. Y200. ;
N3 G53 X300. Y100. ;
N4 X300. Y0 ;
N5 G52 ;
N6 X0 Y0 ;
```

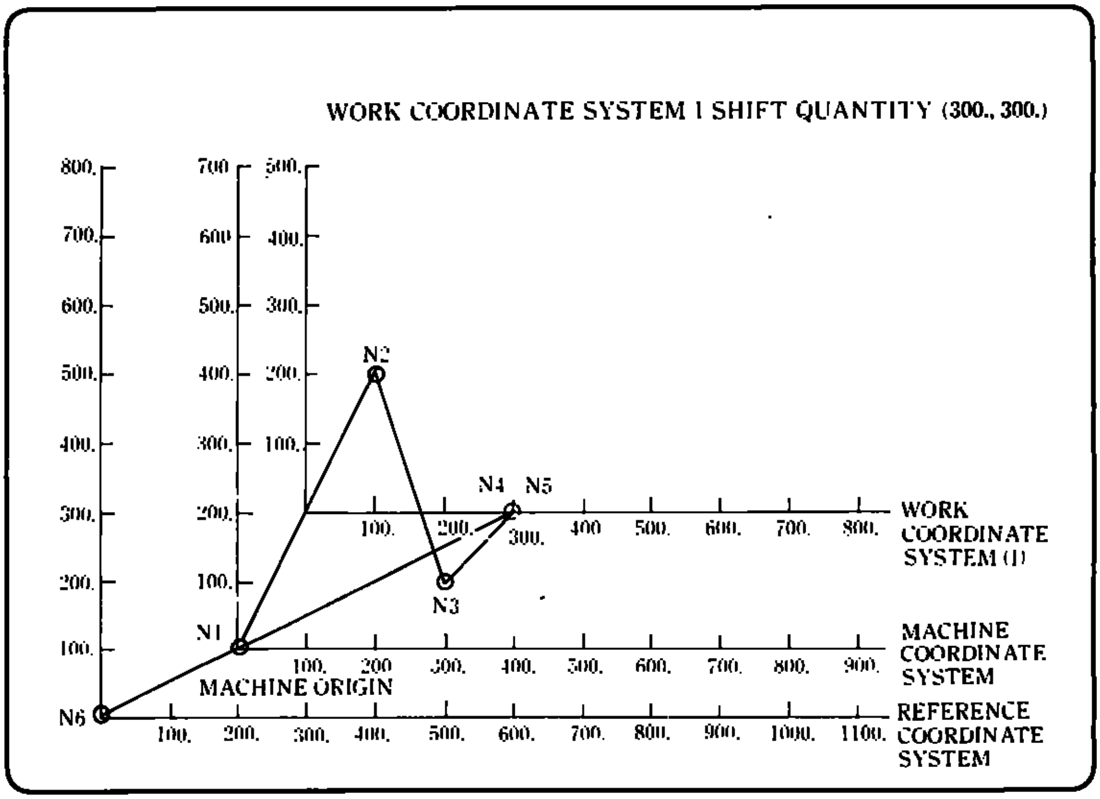


Fig. 2.7.7 Example of Machine Coordinate System (G53)

NOTE

❶ The G53 command must always be issued under the following conditions (otherwise, alarm "322" may result):

- Not in a mirror image
- Not in a canned cycle or tool radius compensation C
- The immediately preceding 01-group G code is any of G00, G01, or G60.
- G53 and G00 are not instructed in the same block.

❷ If the G53 command is executed when machine lock is ON, the current value display moves until a command value by which value must otherwise be moved if machine lock is OFF, is reached before completion. If machine lock ON/OFF is switched over in the middle of the G53 block, the machine will not be correctly positioned. However, if one whole block of G 53 is executed in a machine lock OFF state, the machine is correctly positioned as instructed regardless of whether there was machine lock intervention previously.

❸ Be sure to use the G53 command, in the G90 mode. If G53 is instructed while still in the G91 mode, the coordinate value is handled as the value of G90 mode.

❹ If G53 is instructed in a state where tool length offset or tool position offset is applied, movement is made with the corrected quantity temporarily canceled. It is generally recommended to instruct G53 after canceling the tool length offset or tool position offset.

2.7.1.5 Rotation of work coordinate system ✻

(1) By setting an angle of rotation when a work coordinate system is set by G54 to G59 command, it is possible to rotate the coordinate system simultaneously with its movement.

(2) Program example

`G92 X0 Y0 Z0 ;`

⋮ Nothing changes in G92 state.

⋮

⋮

`G54 ;` (G68 X0 Y0 R···)

⋮ At the same time the work coordinate system is shifted by G54, the coordinate system is rotated by a quantity of R around the point (0, 0) of that work coordinate system.

⋮

⋮

`G55 ;` (G69)

⋮ (G68 X0 Y0 R···)

⋮ A work coordinate system shifted by G54 and then rotated is canceled and a new coordinate system is created by using the G55 work coordinate system shift quantity and angle of rotation.

⋮

⋮

`G52 ;` (G69)

⋮ Return to the coordinate system of G92 by G52. At this time, rotation is also canceled.

⋮

⋮

`M30 ;`

NOTE ( ) means the contents of execution performed by coordinate rotating function.

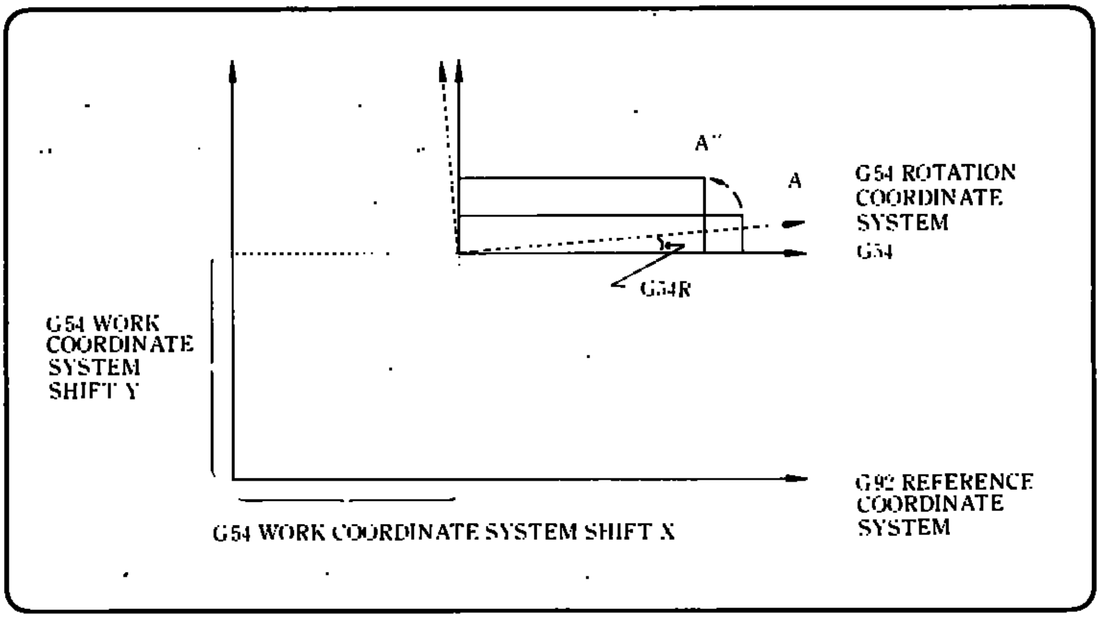


Fig. 2.7.8 Rotation of Work Coordinate System

NOTE

❶ When coordinate rotation is performed by G54 command, the G28 G30 command performs rotation at an intermediate positioning point, but does not perform coordinate rotation at the reference point. G53 does not perform coordinate rotation either.

❷ The plane of coordinate rotation for this specification is fixed to the G17 plane.

❸ For cancellation after coordinate rotation by the G54 command, clear the shift quantity of the work coordinate system being instructed or that of the G52 command to 0.

❹ Combination with the mirror image, scaling, and coordinate rotation functions
The above G codes must be arranged in the specified order. Otherwise, alarm 285 occurs. Work coordinate rotation (G54) ➤ Mirror image (M95) ➤ Scaling (G51) ➤ Coordinate rotation (G68)

❺ Reset during coordinate rotation by G54
Intervention to coordinate rotation by resetting affects the programmed trajectory thereafter. This is because of the following two specifications:

1) By resetting, the current position is fetched into NC as an NC instruction position, to set it again.
2) In coordinate rotation, an axis motion length is calculated from the NC instruction position to the center of rotation.

❻ For specifying rotating coordinate system, execute both X-and Y-axis specifications as the first move command of the new coordinate.
When only one axis is specified by ❺ 2), inadvertant movement is sometimes performed.

2.7.2 Plane Selection (G17/G18/G19)

(1) The planes to perform circular interpolation, tool radius compensation, and coordinate rotation are specified by G codes: G17, G18, and G19.

G17: XY plane
G18: ZX plane
G19: YZ plane

(2) When the 4th linear axis ※ is selected, the following planes are newly added:

G17: XY plane or Xα plane
G18: ZX plane or Zα plane
G19: YZ plane or Yα plane

NOTE 'α' indicates any one of U, V, or W.

(3) When the 5th linear axis(*) is selected, the following new planes are added:

G17: XY plane or Xβ plane
G18: ZX plane or Zβ plane
G19: YZ plane or Yβ plane

NOTE 'β' indicates any of U, V, or W.

(4) A move command for a single axis can be programmed irrespective of plane specification by G17, G18, or G19.
For example, if programmed as shown below, the Z-axis is moved.

```
G17  Z······ ;
```

(5) The plane to perform tool radius compensation by using G41 or G42 is univocally determined by G17, G18, or G19. Note however that no correction plane can be specified for a plane that includes a 4th rotation axis.

(6) When power is turned ON, the XY plane (G17) is selected.

(7) A canned cycle is fixed to the G17 plane.

G17: Z-axis hole opening axis

2.8 ENTERING COORDINATE VALUES

2.8.1 Absolute/incremental Specification (G90, G91)

These G codes specify whether the move data following the axis address is an absolute value or incremental value.

(1) G90 ······ Absolute specification

In subsequent blocks after a G90-containing block, the move data following addresses X, Y, Z (※ α, β) are handled as absolute values.

```
G90  G00  X······Y······Z······ ;
```

(2) G91 ······ Incremental specification

In subsequent blocks after a G91-containing block, the move data are handled as incremental values.

```
G91  G01  X······Y······F······ ;
```

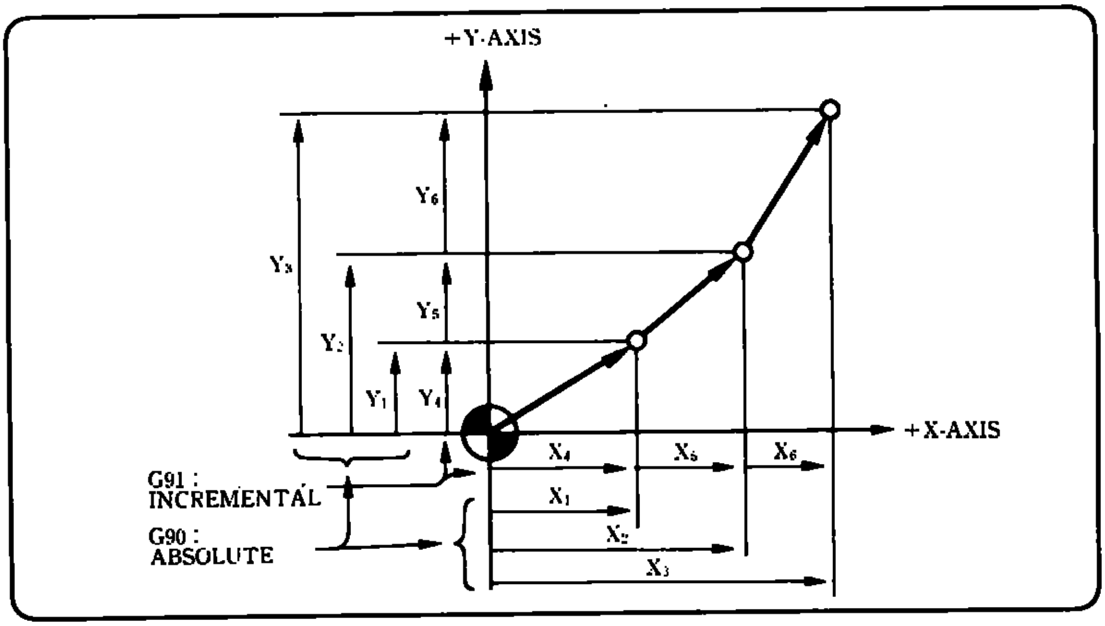


Fig. 2.8.1 Absolute/Incremental Command (G90, G91)

(3) G90 and G91 are model G codes in group 03.

(4) If G90 and G91 are instructed in the same block, the G code instructed later is effective.

NOTE The initial state of these G codes when power on can be specified by parameter Pm4000 D1.

Table 2.8.1 Initial State

Parameter Pm4000 D1	Initial State
"0"	G90
"1"	G91

Post-reset status can be set by parameter Pm4000 D6:

Table 2.8.2 Initial status after reset

Parameter Pm4000 D6:	Group 03 G code to be selected after reset
"0"	Set by Pm4000 D1.
"1"	Preceding G code is retained.

2.8.2 Inch/metric Input Specification (G20, G21)

(1) The G codes shown below specify the unit of input between metric or inch.

Table 2.8.3 Specification for Unit of Input

G Code	Unit of Input
G20	Input in inches
G21	Input in mm

(2) These G codes are specified at the beginning of the program in an independent block.

When these G codes are executed, the following are matched to the altered unit of input:

① Subsequent programs
② Offset quantity
③ Setting and part of parameters
④ Part of manual operations
⑤ Various display

NOTE

- The inch/metric selection setting is rewritten by G20 and G21. Therefore, the states of G20 and G21 for turning power on are determined by the setting parameter.

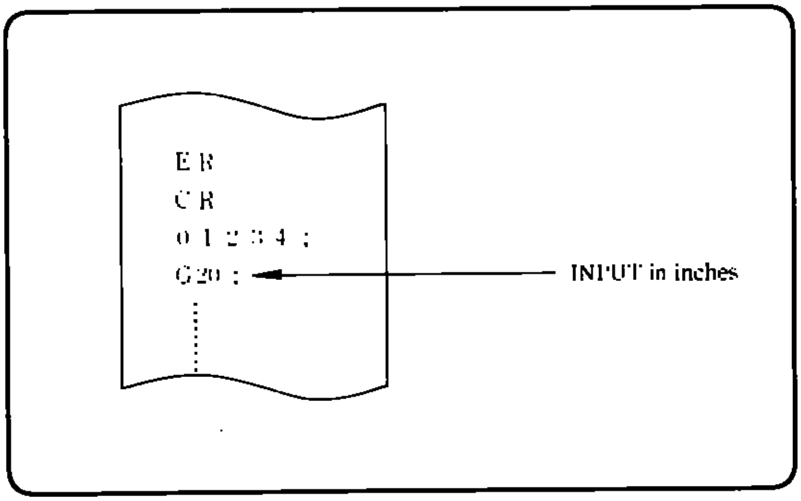


Fig. 2.8.2. Example of Programming

❷ When switching between G20 and G21 can be instructed in the middle of the program, the following processing must be accomplished in advance:
- If any work coordinate system (G54 to G59) is being used, return it to the reference coordinate system.
- Cancel all tool offsets (G41 to G48).

❸ After switching between G20 and G21 is instructed, perform the following processing:
- Set coordinate systems (G92) for all axes before instructing a move command.
- If work coordinate system display and external coordinate value display are to be used, clear the current value to zero.

❹ The tool offset quantities stored in memory are handled differently in the G20 and G21 modes. Issue the G20/G21 command after correcting the offset quantities.

Table 2.8.4 Tool Offset Quantities in G20 and G21 Commands

Stored offset quantity	G20 (inch) mode	G21 (mm) mode
150000	5.9055 in	150.000 mm

2.8.3 Decimal Point Input

Numeric values with decimal points can be used for addresses concerning coordinate word (distance), time, and speed.

① The addresses in which decimal points can be used are as follows:

Coordinate word: X, Y, Z, I, J, K, A, B, C, U, V, W, Q, R

Time: P

Feedrate: F

(Example)

[mm] [inch]

X15. ——→ X15.000mm or X15.0000in

Y20.5 ——→ Y20.500mm or Y20.5000in

(G94)F25.6 ——→ F25.0mm/ min or F25.6in/ min

(For F6.0) (For F4.1)

G04P1. ——→ Dwell 1.000s

② When numeric values without decimal points are input, they are normally handled by the equipment as 1 = 0.001 mm (or 0.0001 inch, or 0.001 deg).

2.8.4 Scaling (G50, G51) ※

Forms instructed by a part program can be expanded or contracted to any desired rate of magnification.

(1) The following G codes are used for this purpose.

Table 2.8.5

G Code	Group	Meaning
G50	15	Scaling OFF
G51	15	Scaling ON

(2) G51 I……J……K……P…… ;

With the above command, the form is expanded or contracted around the coordinate value specified by I, J, and K as the center of scaling by the rate of magnification specified by P.

(3) G50 ;

The above command cancels the scaling mode.

(4) The magnification rates of expansion and contraction are 0.000001 to 99.999999 times.

(5) The command unit of P is 1=0.000001.

When P is instructed with a decimal point appended, it is assumed for the G51 block only that there are six digits after the decimal point.

(Example)

P 0.999999 → 0.999999 times
P 2.0 → 2 times
P 2 → 0.000002 times

(6) If the magnification rate specified by P is omitted, a rate of magnification determined by setting Pm0803 and Pm0804 is assumed.

Rate of magnification = Pm0803/Pm0804

(Example) Pm0803 = 3, Pm0804 = 100
Rate of magnification = 3/100 = 0.03 times

NOTE The rate of magnification determined by setting must not exceed the range of expansion and contraction stipulated above.

(7) When I, J, and K are specified in a G51 block, scaling is only applied to the specified axes.
Scaling cannot be applied to any additional axis (4th or 5th axis).

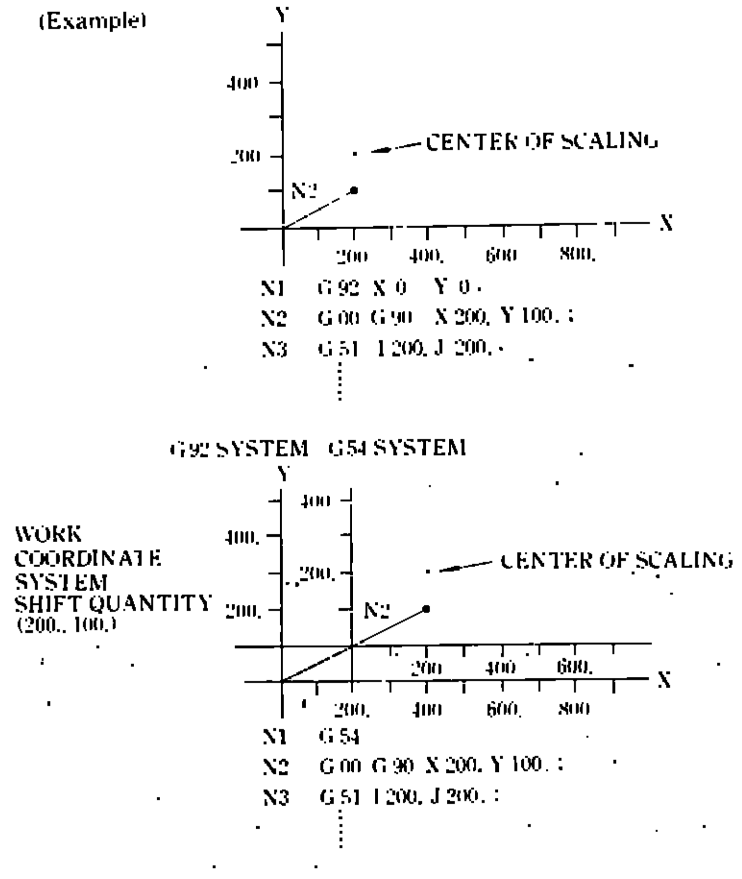
I······X-axis, J······Y-axis, K······Z-axis

NOTE Scaling is not applied to axes for which I, J, and K are not specified.

(Example) G51 I100 J0 P 0.8 ;

In the above case, scaling is applied to X-axis and Y-axis, but not applied to Z-axis.

(8) Specify the distance from the origin of that work coordinate system to the center of scaling when work coordinate system is specified to I, J or K of G51 block.



```
N1   G 92  X 0   Y 0 ;
N2   G 00  G 90   X 200. Y 100. ;
N3   G 51  I 200. J 200. ;
       :
```

```
N1   G 54
N2   G 00  G 90  X 200. Y 100. ;
N3   G 51  I 200. J 200. ;
       :
```

Fig. 2.8.3 Scaling command for Specified Work Coordinate System

(9) By the command G51 IOJOP…;, the current position can be specified as the center of scaling.

NOTE

❶ Scaling normally turns on at machining approach, and turns off after machining is completed. If scaling is turned on in the middle of machining, the workpiece may not be correctly shaped.

❷ Scaling is normally applied to the two axes of the machining plane. If scaling is applied to only one axis, circular command cannot be correctly scaled, so alarm "281" results.

❸ G51 I……J……K……P…… ; , G50 ;

These blocks must be instructed with a single block. If X, Y, and Z are instructed in the same block, alarm "281" results.

❹ When a magnification rate of scaling equal to or greater than X1 is specified, make sure the magnified command value does not exceed the maximum command value.

❺ 0 cannot be instructed for the magnification rate of scaling. Specification of 0 results in alarm "281".

❻ Scaling is not applied to offset quantities.

❼ No canned cycle can be executed while scaling is applied to Z-axis (otherwise, alarm "280" results)

❽ When reset (by depressing the RESET key or by M02 or M30 command), scaling is turned off (G 50).

❾ Command value and current value display during scaling become the values after scaling.

❿ The following G codes cannot be instructed during scaling (otherwise, alarm "280" results): G27, G28, G29, G30, G31, G52Q2, G53, G92

⓫ Scaling (G51) cannot be instructed in tool radius compensation C (otherwise, alarm "280" results).

⓬ The following shows alarm codes related to scaling:

Table 2.8.6 Alarm Code

Alarm No.	Contents
280 Unusable G code in scaling	Unusable G code is instructed during scaling mode.
281 Scaling format error	· More than one G50 or G51 is specified. · The magnification rate is 0. · An unusable G code is instructed in scaling.

⓭ For scaling, G51 and G50 must always be used as a pair. Once the scaling mode is entered, new G51 commands are disregarded.

⓮ Combination with work coordinate rotation, mirror image, or coordinate rotation function

The above G codes must be arranged in the specified order. Otherwise, alarm "285" occurs.

Work coordinate rotation (G54) > Mirror image (M95) > Scaling (G51) > Coordinate rotation (G68)

⓯ Scaling command in mirror image

When the scaling function is used in a mirror image operation, the mirror image function changes the center of scaling.

⓰ Coordinate rotation command during scaling

When the coordinate rotation function is used during scaling, the scaling function changes the center of rotation.

The angle of rotation remains unchanged.

2.8.5 Coordinate Rotation (G68, G69) ✽

(1)

$$\left\{\begin{matrix} \text{G17} \\ \text{G18} \\ \text{G19} \end{matrix}\right\} \text{G68} \quad \text{a}\cdots\cdots\text{b}\cdots\cdots\text{R}\cdots\cdots ;$$

a, b: Coordinate values at the center of rotation

R: Angle of rotation (counterclockwise direction specified by + with an absolute value)

When the above command is entered, the coordinate is rotated around the point specified by 'a' and 'b' by the angle specified by R. The angle of rotation is specified in units of 0.001 deg.

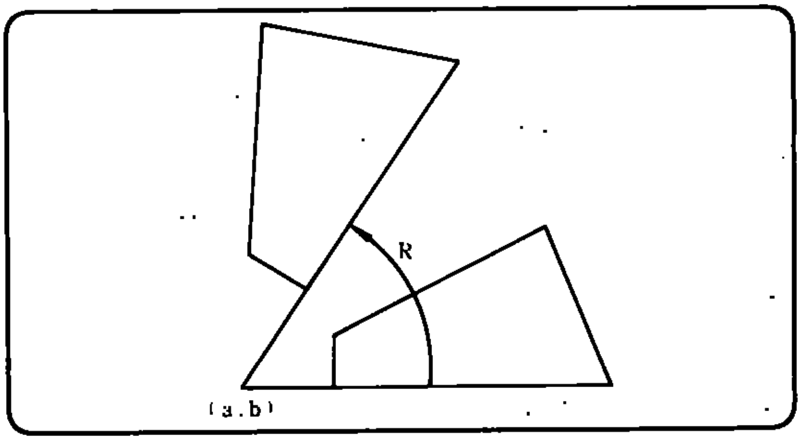


Fig. 2.8.4

(2) The rotation plane is determined by the plane (G17, G18, or G19) that had been selected when the command was made.

G17: XY plane or $X\alpha$,$X\beta$ plane
G18: ZX plane or $Z\alpha$,$Z\beta$ plane
G19: YZ plane or $Y\alpha$,$Y\beta$ plane

NOTE The 4th axis or 5th axis must be a linear axis.

(3) `G69 ;`

By the above command, the coordinate rotation mode is canceled.

NOTE

① If 'a' and 'b' are omitted, the position at which G68 was instructed is assumed to be the center of rotation. R cannot be omitted.

② `G68 a……b……R…… ; , G69 ;`

These blocks must be instructed as independent blocks. Also note that for coordinate rotation, G68 and G69 must be used as a pair.

③ The position display indicates the position where coordinate rotation is applied.

④ Combination with work coordinate rotation, mirror image, or coordinate rotation function

The above G codes must be arranged in the specified order. Otherwise, alarm "285" occurs.

Work coordinate rotation (G54) 〉 Mirror image (M94) 〉 Scaling (G51) 〉 Coordinate rotation (G68)

⑤ Coordinate rotation command in mirror image

When the coordinate rotation function is used in a mirror image operation, the mirror image function changes both the center of rotation and the direction of rotation.

⑥ Coordinate rotation during scaling

When the coordinate rotation function is used during scaling, the scaling function changes the center of rotation.

The angle of rotation remains unchanged.

⑦ G68 and G69 are modal G codes in group 18.

⑧ G69 is automatically selected when the power is turned on or the equipment is reset.

⑨ Coordinate rotation (G68) cannot be instructed in tool radius compensation C (otherwise, alarm "310" results).

⑩ Coordinate rotation normally turns on at machining approach and turns off after machining is completed. If coordinate rotation is turned on in the middle of machining, the work may not be correctly machined.

⑪ The table below shows alarm codes related to coordinate rotation.

Table 2.8.7 Alarm Code

Alarm No.	Contents
310 Coordinate Rotation Unusable G command	Unusable G codes are instructed in G68 mode. G68 is instructed in tool radius compensation C.
311 Coordinate Rotation Format Error	There is a format error in the G68 or G69 command block.

2.9 SPINDLE FUNCTION (S FUNCTION)

2.9.1 Spindle Command

(1) The spindle speed (in units of r/min) can be directly specified by entering a 5-digit number following address S (S□□□□□).

(2) The instructed S becomes effective from the moment the S-command complete input signal (SFIN) turns on.

(3) If S-specification is instructed in the same block as M03 (spindle forward rotation) or M04 (spindle reverse rotation), the program normally advances to the next block after the spindle reaches the speed specified by S. For further information, refer to command or reference manuals issued by your machine tool manufacturer.

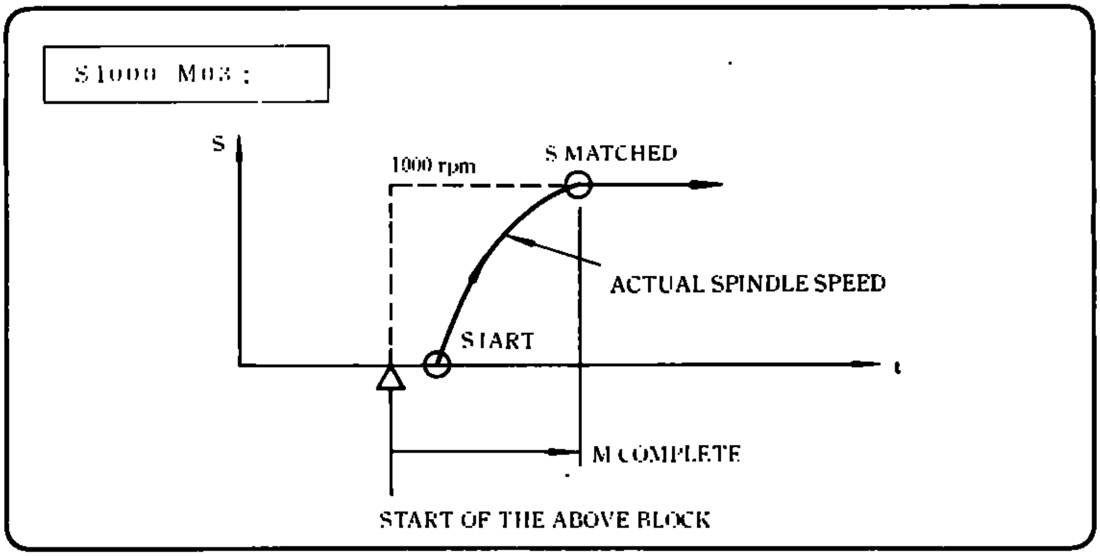


Fig. 2.9.1 Spindle Command

(4) The S-command is modal, so that once instructed, it remains valid until another S-command is given.
Even if M05 is instructed and the spindle actually stops, the S-command is retained. Consequently, the previously instructed S-command can be started by instructing M03 (or M04) again.

(5) If you change an S-command after the spindle is started by M03 (or M04), pay attention to the selected range of spindle speed.

NOTE
1. The lower limit of S-command (S0 and its vicinity) is determined by the main axis motor and varies with each machine. For details, refer to command or reference manuals issued by your machine tool builder
2. Do not instruct a negative S-command.

2.10 TOOL FUNCTION (T FUNCTION)

2.10.1 Tool Select Command

2.10.1.1 T2-digit specification

(1) The tool No. is instructed by a 2-digit number following address T (T □□ : tool selection specified).
The leading zeros may be omitted.

(2) The numbers that can be used for tool select specification varies with each machine.
Also note that when the T-code is instructed in the same block as a move command, the following is determined depending on each machine.

① whether or not the T-command is executed simultaneously with the move command.

② whether or not the T-command is executed after the move command is completed.

Therefore, refer to command or reference manuals issued by your machine tool builder.

(3) T-code command is modal, so that once instructed, it remains valid until another T-command is given.

(4) Because the T-code command is generally used to have the automatic tool changer (ATC) select the tool No. to be used next, it can be specified independent of G codes (for correcting the length or radius of the currently used tool) or H or D codes.

2.10.1.2 T4-digit specification *

(1) The tool No. and tool offset No. are instructed by a 4-digit number following address T (T □□□□ : tool number).
The leading zeros may be omitted.

(2) The function and operation is same as T2-digit specification.

2.11 AUXILIARY FUNCTION (M FUNCTION)

2.11.1 M Function

The auxiliary function is instructed by a number of up to three digits following M. Except in specific M codes, the definitions of M00 to M89 depend on the specifications of machine tool builders. For details, refer to command or reference manuals issued by your machine tool builder.

The following describes the specific M codes relating to the NC equipment.

2.11.1.1 M codes related to stopping (M00, M01, M02, M30)

(1) M00 (program stop)

When M00 is instructed during automatic operation, the automatic operation is interrupted after the operation of that block is excuted and the. M00R signal is selected. The operation can be restarted by depressing the cycle start switch.

(2) M01 (optional stop)

When M01 is instructed when the OPTIONAL STOP switch is on, the same operation as with M00 is performed. If the OPTIONAL STOP switch is off, M01 is disregarded.

(3) M02 (end of program)

M02 is used to terminate the program. When M02 is instructed during automatic operation, the automatic operation is terminated after the operation of that block is executed and then stopped. At this time, the NC equipment is generally reset. How the equipment actually responds varies with machine type. Therefore, refer to the manual issued by your machine tool builder.

(4) M30 (end of tape)

M30 is normally used to terminate the tape. When M30 is instructed during automatic operation, the automatic operation is terminated after the operation of that block is executed and then stopped. At this time, the NC equipment is generally reset and, simultaneously, memory rewind is executed. How the equipment actually responds varies with machine type. Therefore, refer to the manual issued by your machine tool builder.

NOTE ❶ When M00, M01, M02, or M30 is instructed, the NC equipment stops advance reading. For these M codes, M2-digit BCD code and individual decode signal are output.

❷ Whether or not the spindle or coolant is turned off by M00, M01, M02, or M30 command depends on the manufacturer's machine specifications. For details, refer to command or reference manuals issued by your machine tool manufacturer.

2.11.1.2 Internal processing M codes

M codes from M90 to M99 and M190 to M199 are internally processed by the NC equipment. Even when these M codes are instructed, no external output signals (BIN code and decode outputs) are sent from the equipment.

Table 2.11.1

M Code	Meaning
✣ M90 ◥	Program interrupt function OFF
✣ M91	Program interrupt function ON
✣ M92 ◥	Multiactive register OFF
✣ M93	Multiactive register ON
✣ M94 ◥	Mirror image OFF
✣ M95	Mirror image ON
✣ M96 ◥	Tool radius compensation C circular turnaround mode
✣ M97	Tool radius compensation C intersection calculation mode
M98	Subprogram call
M98	Subprogram termination
✣ M191	Comment statement output function
M192	Tool life control count

M190 to M199 are used for extension codes.

NOTE The ◥ mark indicates M codes that are selected when power is turned ON. M codes are not affected by resetting.

2.11.1.3 Other M codes

(1) All M codes other than those listed as specific M codes shown in Pars. 2.11.1.1 and 2.11.1.2 may be used for a variety of different purposes depending on the machine builder's specifications.
The following shows a typical usage example.

Table 2.11.2

M Code	Meaning	Remarks
M03	Spindle forward rotation	Normally, M03 and M04 cannot be switched. Be sure to switch them by using M05.
M04	Spindle reverse rotation	
M05	Spindle stop	
M08	Coolant ON	
M09	Coolant OFF	

(2) When an M-command is instructed in the same block as a move command, the following is determined depending on each machine.

① whether or not the M-command is executed simultaneously with the move command.

② whether or not the M-command is executed after the move command is completed.

Therefore, refer to command or reference manuals issued by your machine tool builder.

2.11.2 Secondary Auxiliary Function (B Function) ※

(1) Positioning of dividing tables is specified by a 3-digit number following address B.

(2) The relationship between individual B codes and dividing positions is determined by the machine manufacturer's specifications.
Also, when a B-command is instructed in the same block as a move command, the following is determined depending on each machine.
① whether or not the B-command is executed simultaneously with the move command.
② whether or not the B-command is executed after the move command is completed.
Therefore, refer to command or reference manuals issued by your machine tool builder.

(3) B-code command is modal, so that once instructed, it remains valid until another B-command is given.

(4) The address range of B-code is 0 to ±999.
The address with decimal point can be specified, however, the degits below the decimal point is ignored.
(Example) B30.6; is equivalent to B30;.

NOTE

1. The standard B-function interface is a "3-digit BIN" output which is output to the machine side.
2. "B" is used for B-code address specification. Therefore, if B-function is installed, the B-axis in the fourth axis or fifth axis control cannot be added.

There are three kinds of tool offsets as follows:

(1) Tool length offset (used to offset the tool length)
This function is valid for Z-axis. It becomes effective from a block in which G43 or G44 and H code are commanded. It is canceled by the H00 or G49 command.

(2) Tool position offset (used for simple offset of tool radius)
This function is valid for X, Y, and Z axes (※ 4th-and 5th axes). This function is only valid in blocks where any G code from G45 to G48 is commanded.

(3) Tool radius offset C※ (used to offset the tool radius of a complex form)
This function is valid for an arbitrary machining form. It is effective for the XY plane and YZ or ZX plane. This function becomes effective from when G41 or G42 and D code are commanded. It is canceled by the G40 command.

NOTE For details on each of the above corrections, refer to Pars. 2.12.2 to 2.12.4.

2.12 TOOL OFFSET

2.12.1 Tool Offset Amount

2.12.1.1 Tool offset value memory

(1) For any correction described above to be valid, correction amount must be written to the tool offset value memory in advance.

(2) The number of tool offset value memories are as follows:

Table 2.12.1

		Number of Tool Offset Value Memories
Basic Specifications		99
Additional Tool Offset Value Memories	Option 1	299
	Option 2	999
	Option 3	1199

(3) The setting range of tool offset amount is as shown in Table 2.12.2.

Table 2.12.2

	Linear Axes	※Rotary Axes
Input in mm	0 to ± 999.999 mm	0 to ± 999.999 deg
Input in inches	0 to ± 999.999 in	0 to ± 999.999 deg

NOTE These values are not affected by the unit of output.

(4) For write-in operation of tool offset value, refer to Par. 3.4.

2.12.1.2 H and D functions (H, D codes)

The tool offset No. is specified by a 2-digit number (※ 3-or 4-digit number) following address H or D.

H□□ or D□□
Tool offset No.

Two types of H, D function methods are available, which can be selected by parameter.

pm4029 D0 = 0 : H/D shared method
pm4029 D0 = 1 : H/D separated method

(1) H/D shared method (Pm4029 D0=0)

① Tool offset Nos. [01] to [99] directly correspond to 99 tool offset value memory Nos. That is, the offset amount stored in the tool offset value memory of the specified No. is used.

② Tool offset No. [00] (H00, D00) have different meanings depending on each correction function. For details, refer to the description of each G-function.

③ Discrimination of H and D code usage

❶ The H and D codes (used to specify a tool offset No.) must be used differently depending on the offset function as follows:

Table 2.12.3 Usage of H and G Codes

Code	Usage
H Code	Tool length offset
D Code	Tool position offset or tool radius compensation

❷ However, the tool offset Nos. "01" to "99" and the H or D codes may be used in any desired combination. For easy programming, however, it is recommended that the H code be selected from H01 to H30, and the D code be selected from D31 to D99.

Table 2.12.4 H and G Codes and Offset No.

Offset method	G code	H, D code	Tool offset value memory
Tool length offset	G43 G44 G49	H $\boxed{0}\boxed{2}$	
Tool position offset	G45 G46 G47 G48	D $\boxed{9}\boxed{9}$	
Tool radius compensation C (Intersecting point calculation method)	G40 G41 G42		

No	Offset amount
01	
02	
03	
·	
:	
97	
98	
99	

(2) H.·´D separated method (Pm4029 D0=1)

① Two kinds of codes, H and D, are used for the same correction No. correction No.

Therefore, the correction value is different when specified as H10 and when specified as D10.

② In this method, the number of correction pairs is halved, because two memories are used for H and D of the same No.

Table 2.12.5

		Number of Tool Offset Value Memories
Basic Specifications		H 49 D 49
Additional Tool Offset Value Memories	Option 1	H 149 D 149
	Option 2	H 499 D 499
	Option 3	H 599 D 599

2.12.2 Tool Length Offset (G43/G44/G49)

Tool length offset adds or subtracts the contents of tool offset value memory for the coordinate command value of Z-axis, and is used to correct the tool length.

(1) G codes for tool length correction

Table 2.12.6

G Code	Group	Meaning
G43	08	Positive (+) direction
G44	08	Negative (−) direction
G49	08	Cancel

(2) G43 and G44 are modal commands so that once commanded, they remain effective until they are canceled by G49.

(3) The G49 command cancels tool length offset.

(4) The H00 command also has the function to cancel tool length offset.

(5) Tool length offset is commanded in the following format:

①

```
  (G01)
G43  (G44)    Z……H…… ;
```

With this command, the tool position is moved by the tool offset amount specified by the H code in the add or subtract direction toward the Z-axis command position. In other words, the position is compensated for by the tool offset amount relative to the Z-axis command target point.

②

```
  (G01)  Z……;
G43 (G44)  H……;
```

With this command, the position is moved by the tool offset amount specified by the H code.

③

```
G43 (G44)   Z……H……;
H……;
```

With this command, the position is moved by the difference between the previous tool offset amount and the new tool offset amount.

NOTE The G codes in group 01 must be G00, G01, or G60 when G43, G44, or G49 is commanded. Command in G02 or G03 modes results in error.

(6) Direction of offset

The direction of offset is determined by the sign of tool offset amount (specified by H code) and the G code.

Table 2.12.7

	Sign of tool offset amount	
	Positive	Negative
G43	Offset in the positive direction	Offset in the negative direction
G44	Offset in the negative direction	Offset in the positive direction

Example of Programming

```
H10 ········OFFSET AMOUNT   -3.0
H11 ········OFFSET AMOUNT    4.0    CRT DISPLAY WITH
                                    OFFSET AMOUNT
                                    ADDED
                                    (Z-AXIS ONLY)
N101  G92  Z0;                              0.000
N102  G90  G00  X1.0  Y2.0;                 0.000
N103  G43  Z-20.  H10;                    -23.000
N104  G01  Z-30.  F1000;                  -33.000
N105  G00  Z0  H00;                         0.000
           ·
           ·
           ·
N201  G00  X-2.0  Y-2.0;
N202  G44  Z-30.  H11;                    -34.000
N203  G01  Z-40.  F1000;                  -44.000
N204  G00  Z0  H00;                         0.000
```

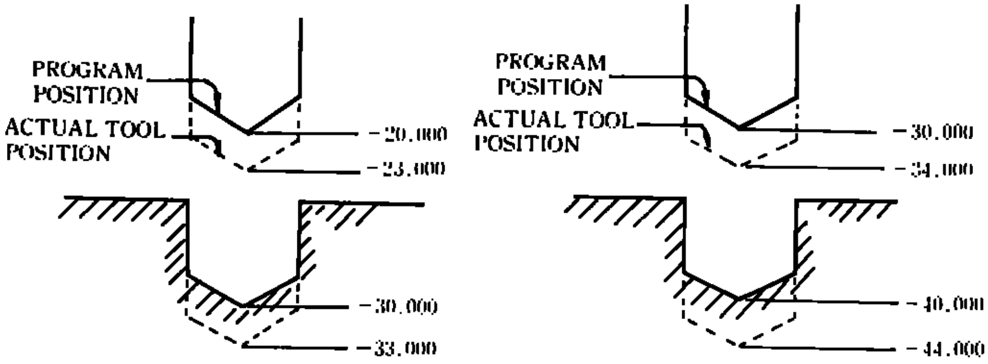


Fig. 2.12.1 Tool Position Offset

NOTE

1. If the offset amount is altered by MDI during the offset mode, H becomes effective from the specified block.
2. Tool position offset or tool radius compensation-C can be added to tool length offset.
3. G43, G44, or G49 cannot be commanded in the canned cycle mode (otherwise, an alarm results).
4. If a G92 command inclusive of Z-axis is commanded in tool length offset, the length correction is canceled. Although this is the case, it is recommended that tool length offset be canceled before G92 is commanded.
5. The tool offset No. (H-code No.) that is made effective by tool length offset can be displayed during automatic operation.
6. G43, G44, and G49 must always be commanded during G00 or G01. If G43 is commanded while still in G02 or G03, for example, no alarm may be issued.
7. After return to reference point of Z and other axes or reset operation is commanded when tool length compensation by Pm4010 D7 is effective, whether to cancel or retain H code and the tool offset amount can be specified.

   Also, when the equipment is reset, whether G code (G49) in group 08 should be canceled or retained can be specified.

2.12.3 Tool Position Offset (G45 to G48)

Tool position offset extends or contracts a program-specified movement amount by the contents of tool offset value memory, and is used primarily to correct the tool radius of a square shape. Therefore, this function need not be used for equipment provided with G40, G41, and G42 (tool radius compensation-C) as options.

(1) G codes for tool position offset

Table 2.12.8

Code	Group	Meaning
G45	*	Extension
G46	*	Contraction
G47	*	Twofold extension
G48	*	Twofold contraction

(2) **G45 to G48 expands or contracts the movement amount of an axis commanded in that block in the direction of movement. Extension or contraction is only executed in the block in which G45 to G48 is commanded, and length of movement in all other blocks are performed as programmed.**

Consequently, if an offset amount from the program diagram by extension or contraction is to be returned to the original, extension or contraction in the reverse direction must be commanded.

(3) To clarify the above operation, create program commands using incremental specification (G91).

Conversely, if commanded with absolute specification (G90), extension or contraction may be performed along the direction of movement for a movement quantity (i.e., incremental quantity) from the end point of the previous block (offset not applied) to the command target point.

(4) When commanding G45 to G48, be sure to specify the tool correction No. by D code simultaneously with axis specification. However, it may be omitted if the same D code is used because the D code is a modal code. Tool radius values must have been set in tool offset amount memory.

Example of Programming

```
    G91
① G00  G46  X·····Y·····D01;          ··········CONTRACTION
② G01  G47  Y·····  (D01)  F·····:         ····TWOFOLD EXTENSION
③      G47  X·····  (D01)  :          ············TWOFOLD EXTENSION
④      G47  Y·····  (D01)  :          ············TWOFOLD EXTENSION
⑤      G47  X·····  (D01)  :          ············TWOFOLD EXTENSION
⑥ G00  G46  X·····Y·····(D01)  :           ····CONTRACTION
```

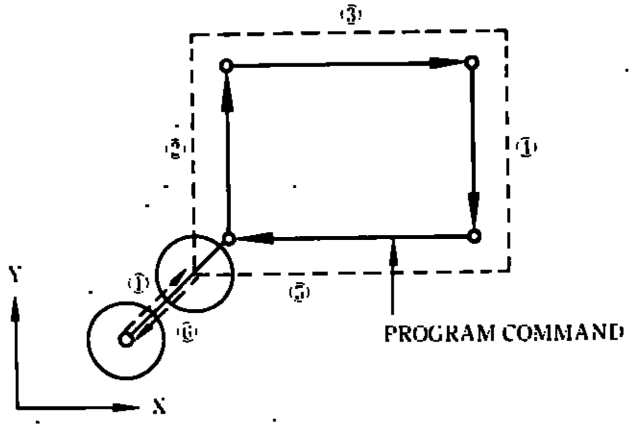


Fig. 2.12.2· Tool Position Offset

(5) Extension/contraction

Extension or contraction is determined by the sign of tool offset amount (specified by D code) and the G code.

Table 2.12.9 Sign of Tool Offset Amount

	Sign of tool offset amount	
	Positive	Negative
G45	Extension	Contraction
G46	Contraction	Extension
G47	Twofold extension	Twofold contraction
G48	Twofold contraction	Twofold extension

NOTE Normally, "positive" sign be used for tool offset amount.

(6) Extension/contraction quantity

① The tool position is extended or contracted relative to the commanded incremental movement quantity by one or two times the specified tool offset amount.

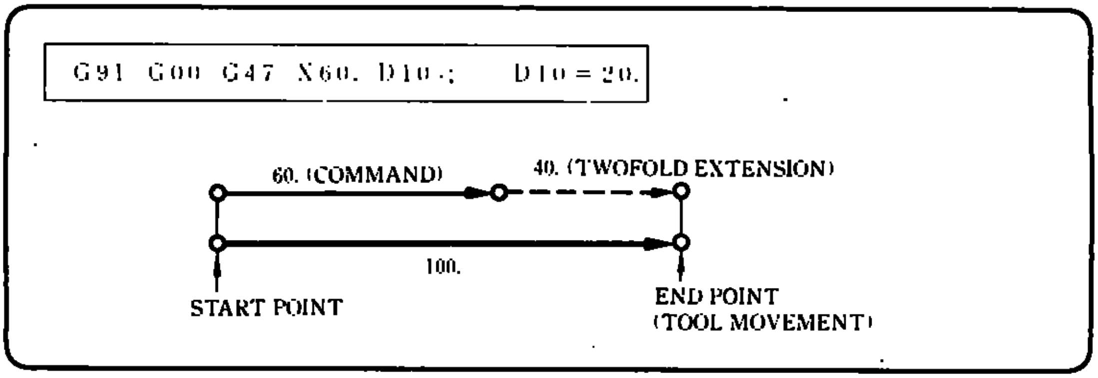


Fig. 2.12.3

② If that particular axis was extended or contracted in a previous block and the start point has already been offset, the total movement amount becomes equal to the one described above, but the moved distance is from the offset start point.

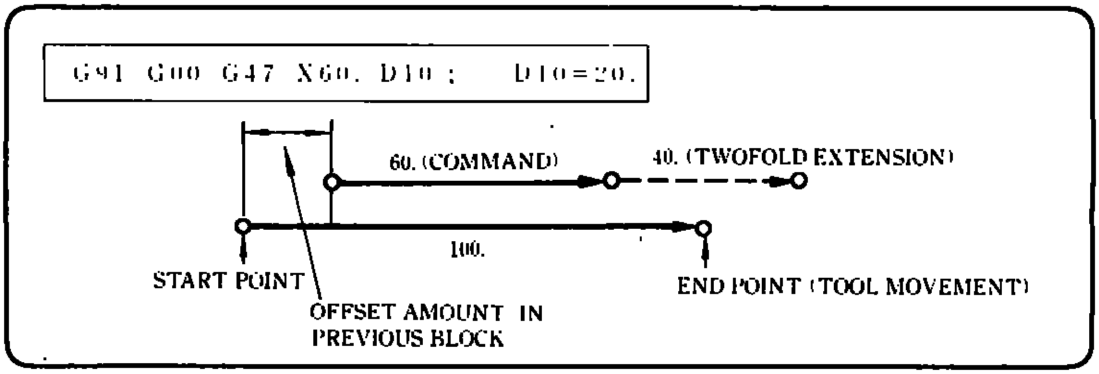
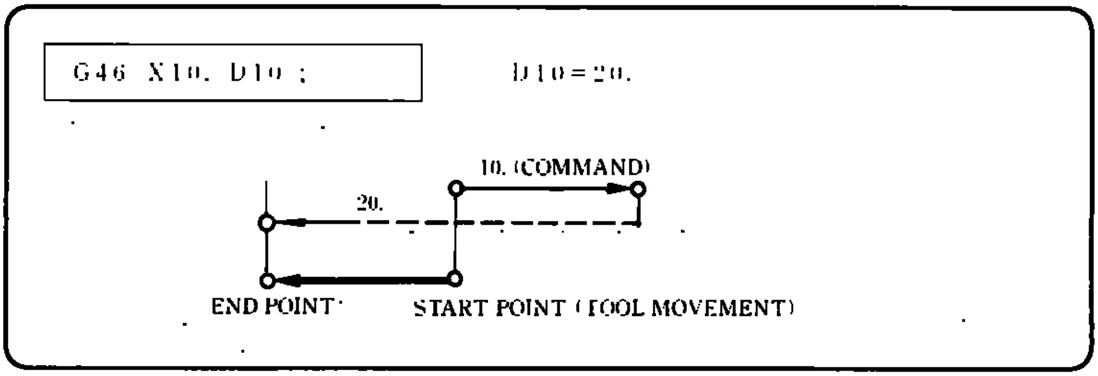


Fig. 2.12.4

NOTE If the tool offset amount is greater than movement quantity in the program, the direction of movement may be reversed as a result of extension or contraction applied. In such a case, the tool moves in the direction opposite to that of the program.



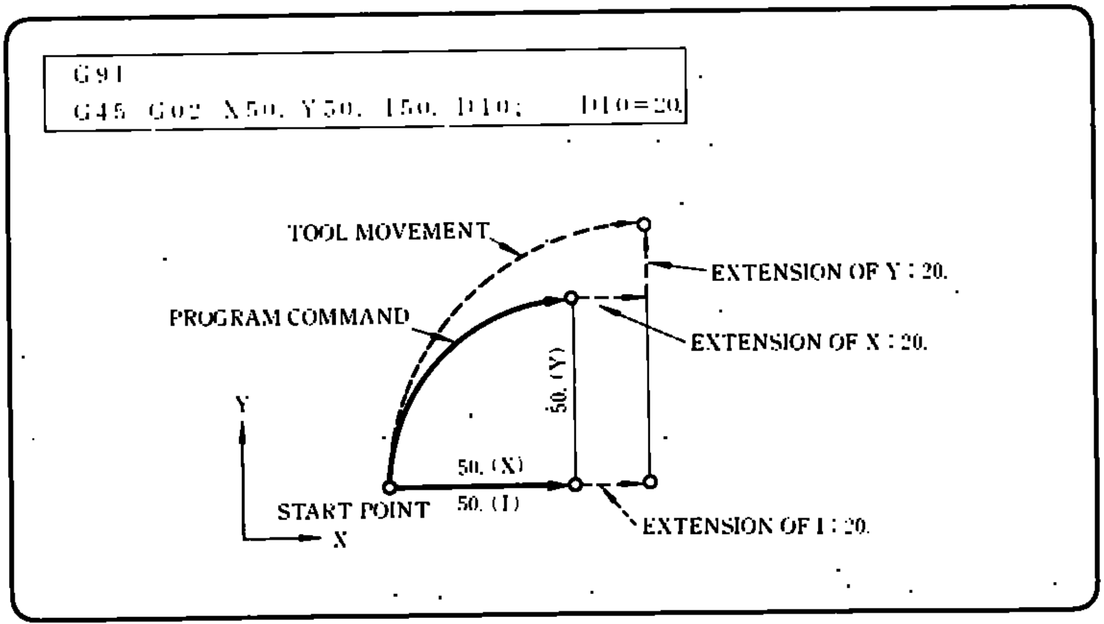
```
G46 X10. D10 ;            D10=20.
```

Fig. 2.12.5

(7) Although descriptions above have been made for X and Y axes, G45 to G48 can be commanded in the same way for Z-axis, too.

(8) Application to circular interpolation
If I, J, or K are commanded in a G45 to G48 block, extension or contraction is performed in the same direction as X, Y, and Z. Tool radius compention can be accomplished by one command for 1/4, 3/4, or 4/4 circles only.

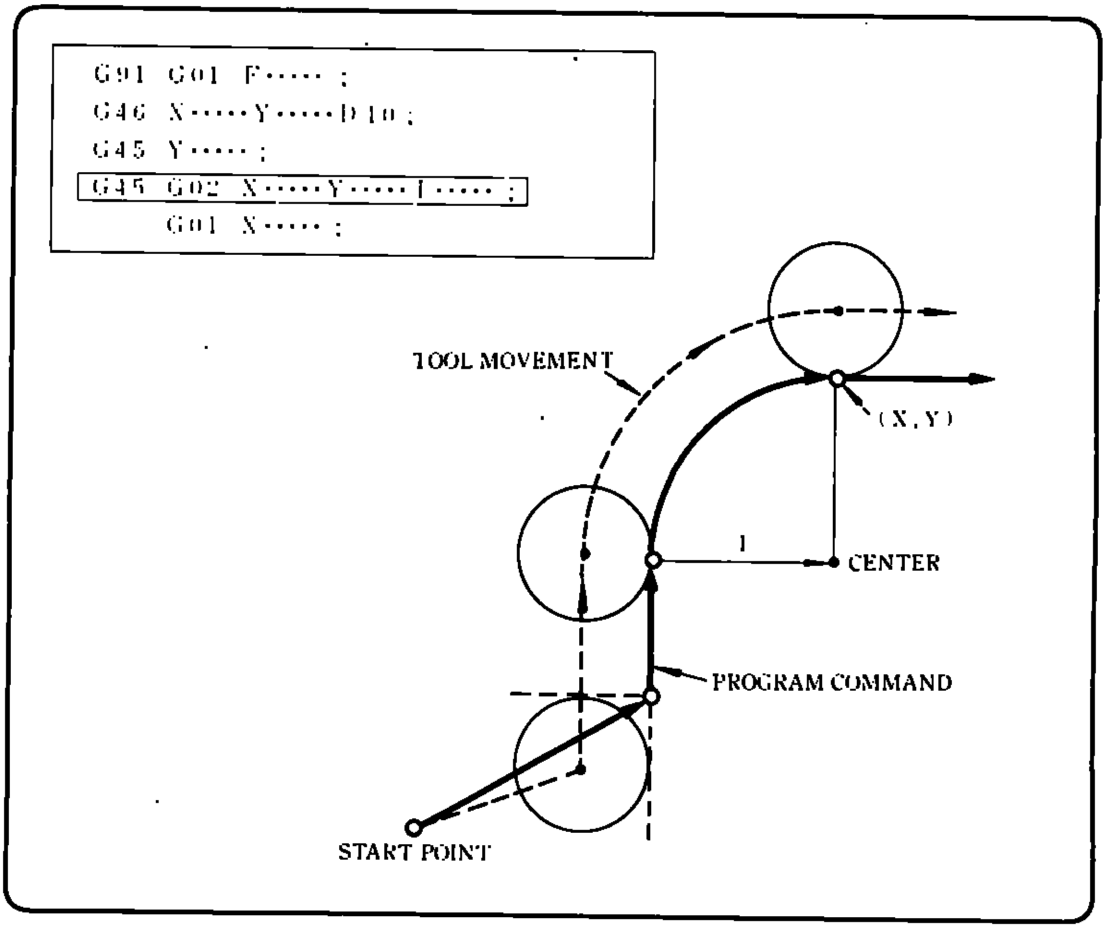
NOTE For 1. 2 circle command, compose it by using 1. 4 circle commands.



```
G91
G45 G02 X50. Y50. I50. D10;      D10=20.
```

Fig. 2.12.6 Tool Radius Compensation in 1/4, 3/4 or 4/4 Circle

Actually, if offset is applied from the previous block, as shown in Fig. 2.12.7, the circular arc can be the correct tool radius offset.



```
G91 G01 F····· ;
G46 X·····Y·····D10 ;
G45 Y····· ;
G45 G02 X·····Y·····I····· ;
    G01 X····· ;
```

Fig. 2.12.7 For Applying Offset from the Block

(9) G45 to G48 can only be commanded with G codes in group 01 (G00, G 01, G02, G03,or G60). If commanded with other G codes, alarm "365" results.

(10) If move offset only is desired in the incremental specification (G91) mode, command "0" for the axial movement amount.

```
G91 G01 G45 X0 Y0 D10 F······ ;
```

——Both X and Y axes move in the positive direction by the axis offset amount specified by D10.

```
G91 G00 G46 X0 D11 ;
```

——X-axis moves in the negative direction by the offset amount specified by D11.

NOTE Adding a sign to "0" cannot reverse the moving direction.

(11) Command for the correction No. of tool position offset (G45 to G48) can be selected between H-code and D-code by parameter setting.

Parameter setting

Pm4011 D1
= 1: H-code valid
= 0: D-code valid

NOTE

1. If G45 to G48 is commanded for simultaneous two-axis move command, both of the two axes are extended or contracted. Use of such commands in cutting operation may result in either excessive cut-in or insufficient cut.

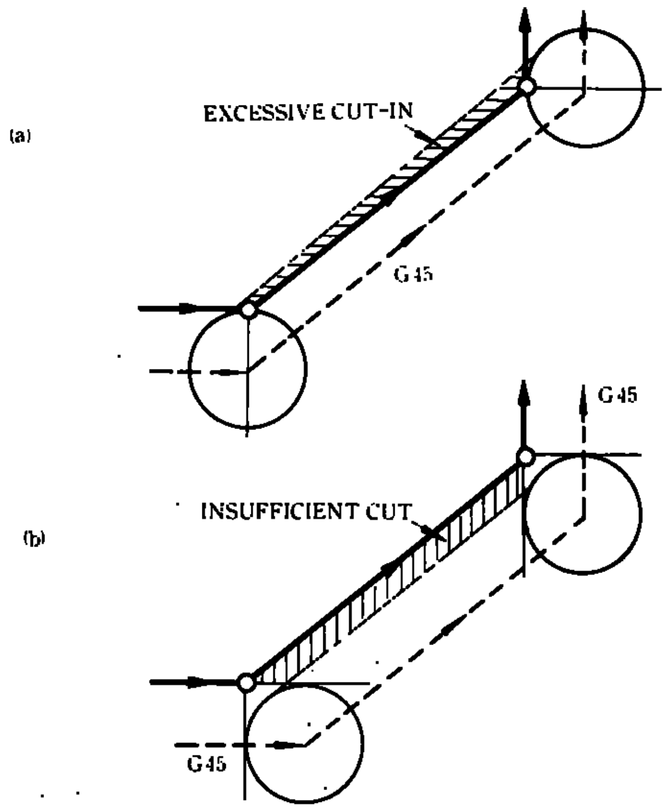
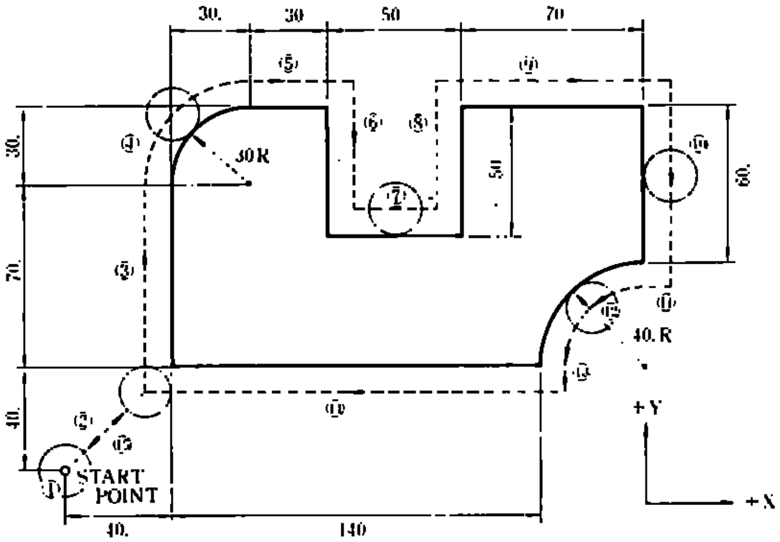


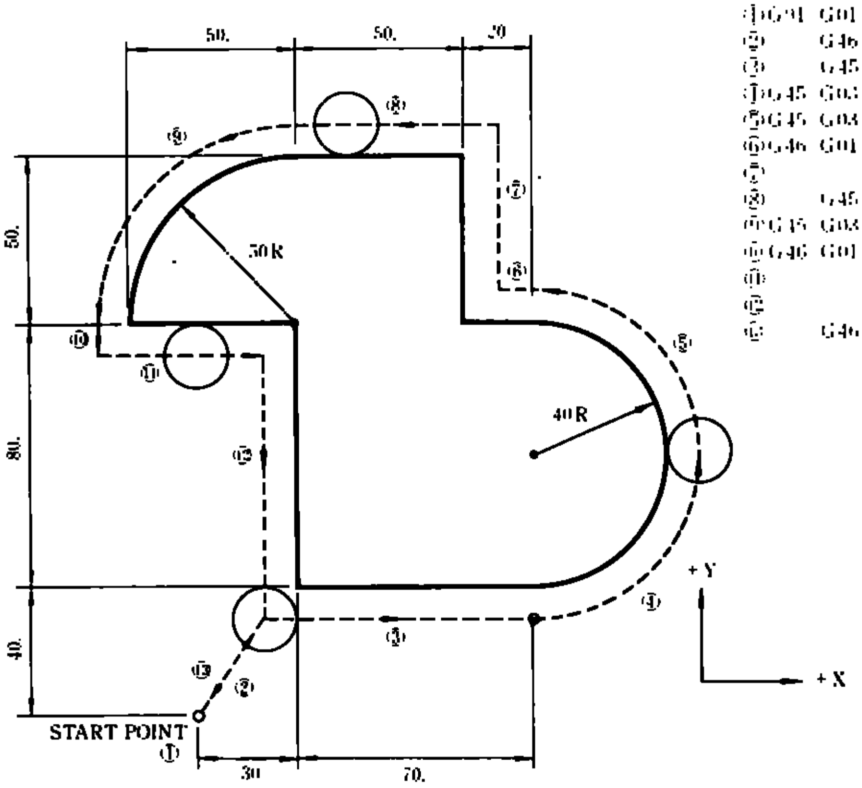
Fig. 2.12.8 When G45 to G48 Command is added to Simultaneous Two-Axis Move Command

2. If offset amount was changed by MDI, the preset offset command is not influenced. The new offset amount becomes effective when G45 to G48 is commanded.
3. Tool position offset can be added to the tool length offset.
4. Mirror image may be applied to tool position offset. That is, symmetrical machining with this correction applied is possible.
5. Tool position offset is irrelevant to plane-specifying G codes (G17/G18/G19).
6. G45-G48 cannot be commanded in the fixed cycle mode. Alarm "170" may result.
7. If G92 is commanded in offset, the coordinate system is set after offset is canceled for the commanded axes only. Therefore, G92 must, as a rule, be commanded after the offset quantity is restored by commanding extension or contraction in the reverse direction. However, the following move commands move differently depending on the parameter.
   - When parameter Pm4011 D1 = 0,
     Pm4010 D6=0 : Moved as commanded
     =1 : Moved with offset applied
   - When parameter Pm4011 D1 = 1,
     Pm4010 D7=0 : Moved as commanded
     =1 : Moved with offset applied



```
①G91  G01  Z−25.  F150;
②     G46  X40.  Y40.  D10  F300;
③     G45  Y70. ;
④G45  G02  X30.  Y30.  I30.;
⑤G45  G01  X30.;
⑥          Y −50. ;
⑦     G48  X50. ;
⑧          Y50. ;
⑨     G47  X70. ;
⑩     G47  Y −60. ;
⑪     G46  X 0;
⑫G46  G03  X −40.  Y −40.  J −40. ;
⑬G46  G01  Y 0 ;
⑭     G47  X −140. ;
⑮     G46  X −40.  Y −40. ;
⑯          Z25. ;
```

Fig. 2.12.9 Example of Programming A



```
①G91  G01  F300;
②     G46  X30.  Y40.  D15;
③     G45  X70. ;
④G45  G03  X40.  Y40.  J40.,
⑤G45  G03  X −40.  Y40.  I −40. ;
⑥G46  G01  X −20. ;
⑦          Y50  ;
⑧     G45  X −50. ;
⑨G45  G03  X −50.  Y −50.  J −50;
⑩G46  G01  Y 0,
⑪          X50. ;
⑫          Y −80. ;
⑬     G46  X −30.  Y −40.;
```

Fig. 2.12.10 Example of Programming B

8. G45 to G48 offset cancel by G28 (G30)

(Example) (G91 mode)

```
N0 G92  X0  Y0 ;
N1 G46  G00  X50.  Y50.  D5 ;D5=5
N2 G47  Y50. ;
N3 G47  X50. ;
N4 G28  X30.  Y-30. ;
N5 X50,  Y10. ;
```

The middle point in N4 is changed by offset, while the mechanical reference point is not.

In N5, move to P1 if offset cancel is selected by Pm4010.

In N5, move to P2 if no offset cancel is selected by Pm4010.

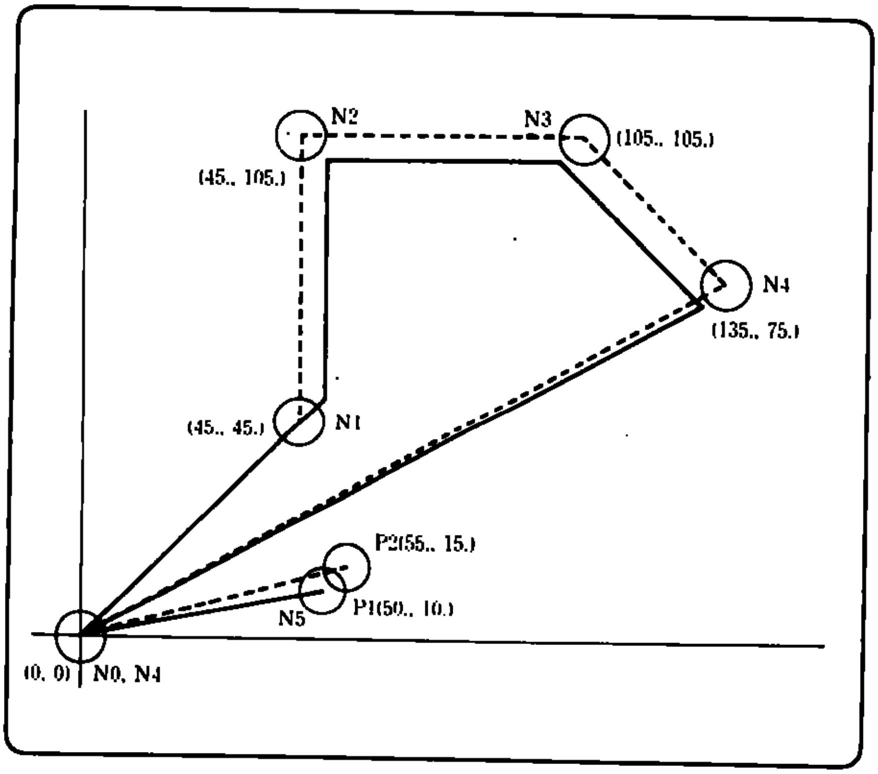


Fig. 2.12.11 G45 to G48 offset cancel by G28 (G30)

9. G45 to G48 offset cancel by G92

(Example) (G91 mode)

```
N0 G92  X0  Y0 ;
N1 G46  G00  X50.  Y50.  D5 ; D5=5
N2 G47  Y50. ;
N3 G47  X50. ;
N4 G92  X0.  Y0 ;
N5 X-50.  Y-50. ;
```

In N5, move to P1 if offset cancel is selected by Pm4010.

In N5, move to P2 if no.offset cancel is selected by Pm4010.

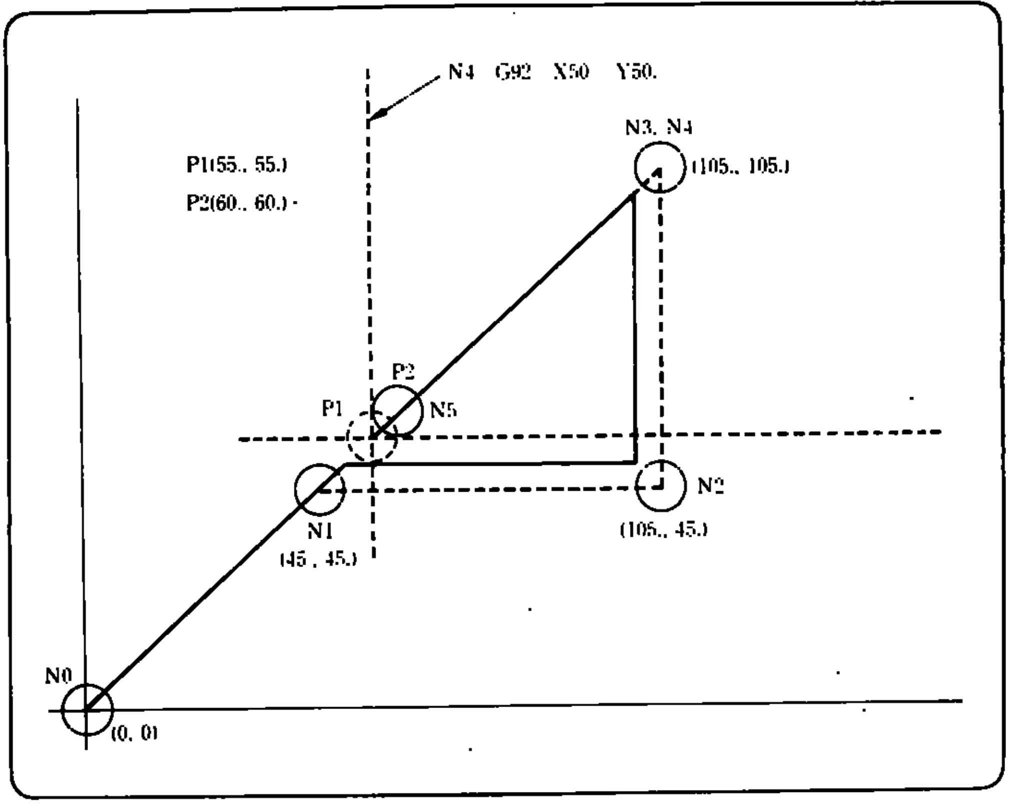


Fig. 2.12.12 G45 to G48 offset cancel by G92

2.12.4 Tool Radius Offset-C (G40, G41, G42) ✻

By giving the radius value of the tool used, it is possible to automatically offset the tool path by that quantity. Offset quantities (tool radius values) are stored in tool offset value memory in advance by using MDI. In a program, the desired tool offset value memory No. can be specified with the D code.

(1) Specification of offset direction and tool radius offset-C specified by D code are initiated by command of G41 or G42, and canceled by G40. G41 and G42 indicate the direction in which the tool is offset relative to the direction of advance.

Table 2.12.10 G Code of Tool Rudius Compensation C

Code	Group	Meaning
G40	07	Tool radius offset-C cancel
G41	07	Tool radius offset-C (offset on left side)
G42	07	Tool radius offset-C (offset on right side)

NOTE G40 is set when the power is turned ON.

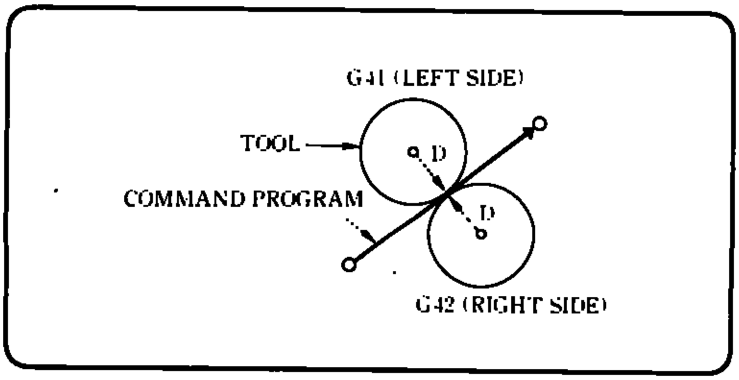


Fig. 2.12.13 Tool Radius Offset-C

NOTE However if the contents of the tool offset value memory specified by D code are negative, the direction of offset (right or left) in Fig. 2.12.11 is reversed. The D code must always be specified in the same block as G41 or G42 or in the preceding block. If D00 is specified, the nose radius is assumed to be "0."

G41 and G42 can be switched over from one to another during the offset mode. Details are described in Item (6) on page 138.

(2) Specification of offset plane

The plane to which tool radius compensation is applied is specified by G17, G18, or G19. These are G codes in group 02, so that when the power is turned on, the XY plane (G17) is specified by default.

Table 2.12.11 G Code of Plane Specification

Code	Group	Meaning
G17	02	XY plane specified
G18	02	ZX plane specified
G19	02	YZ plane specified

NOTE G17 is specified when the power is turned ON.
The G code for plane specification must always be specified in the same block as G41 (or G42) or a block preceding it. If the offset plane is switched in offset mode, alarm "183" will occur. Tool radius compensation-C cannot be applied for the plane that includes a fourth axis or a fifth axis (option).

(3) Method of entering the offset mode

(a) There are two types which can be switched by parameter.

pm4012 D6 = 0 : Type A
pm4012 D6 = 1 : Type B

① Type A : Does not move if move command is not provided in G41 (G42) command block.

If a move command is not given in the G41 (G42) command block, no axes move. In other words, offset is not applied until a move command is encountered. If movement in the offset plane is not commanded in a block next to G41 (or G42) ,offset operation is initiated for that block by prereading one block ahead. For such blocks (with no move commands given), two continuous blocks can be programmed. If no motion in the offset plane is contained in three or more consecutive blocks, the offset operation is aborted and no offset block is generated.

② Type B : Moves even if move command is not provided in G41 (G42) command block.

When G41 (G42) is commanded, the tool moves to a position offset only by radius value. The offset direction is on the normal line of the starting point of the block next to G41 or G42, and on the left or right of the advancing direction according to G41 (G42).
When the coordinate command is not provided in the block of G41

(G42), the tool moves only by offset value.

When a move within the compensation plane is not commanded in the block next to G41 (G42), the data in one more block ahead is read in advance and compensation is performed in that block. In this way, for blocks without move command, continuous two blocks can be programmed.

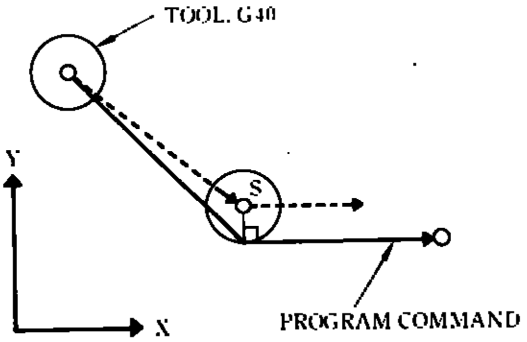
If a move command within the compensation plane is not provided in three blocks or more, tool diameter compensation starts from the next block.

(b) For offset startup, movement is made by considering offset, so that the G code in group 01 must be G00 or G01. If G codes other than G00 or G01 are commanded, alarm "180" results. When offset is entered in the G00 mode, offset points move with each axis independently positioned, so attention should be paid to possible interference with the workpiece.

(c) Offset startup is divided into two types depending on the shape.

- Startup of inside corner (180°or less)
  Moved onto the normal line of a block's direction (vector) next to startup.

```
G17  G01  F······ ;
G41  D······X······Y······ ;
X······ ;
```



NOTE In this and the following figures, "S" indicates the single block stop point.

Fig. 2.12.14 Start up ① of Inside Corner

```
G17  G10  F······ ;
G41  D······X······Y······ ;
G02  X······Y······J······ ;
```

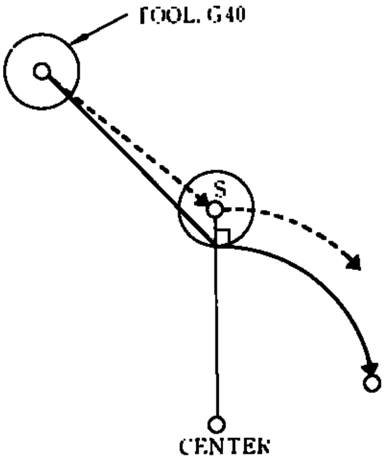


Fig. 2.12.15 Start up ② of Inside Corner

- Startup of outside corner (180° or more)

  There are two types that can be switched over by a parameter.

  Parameter Pm4013 D0 = 1: Type A

  Parameter Pm4013 D0 = 0: Type B

❶ Type A:

```
G17   G01   F······;
G42   D······X······Y······;
X······;
```

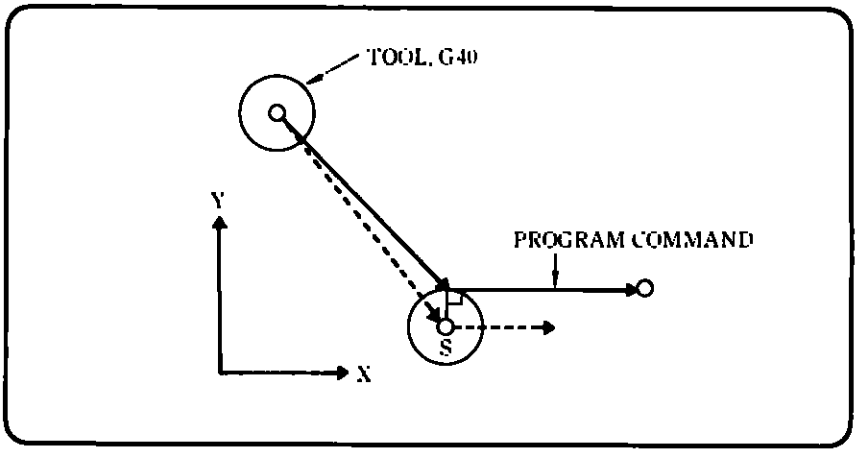


Fig. 2.12.16 Start up ① of Outside Corner

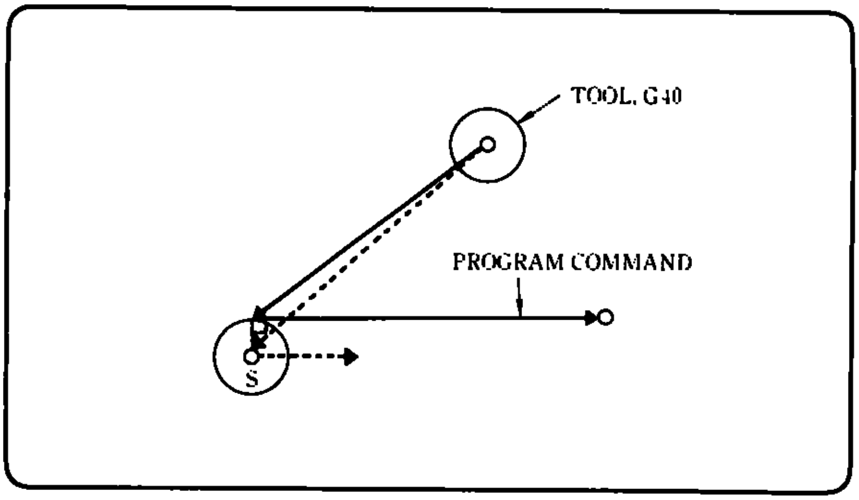


Fig. 2.12.17 Start up ② of Outside Corner

```
G17  G01  F······ ;
G42  D······X······Y······ ;
G02  X······Y······J······ ;
```

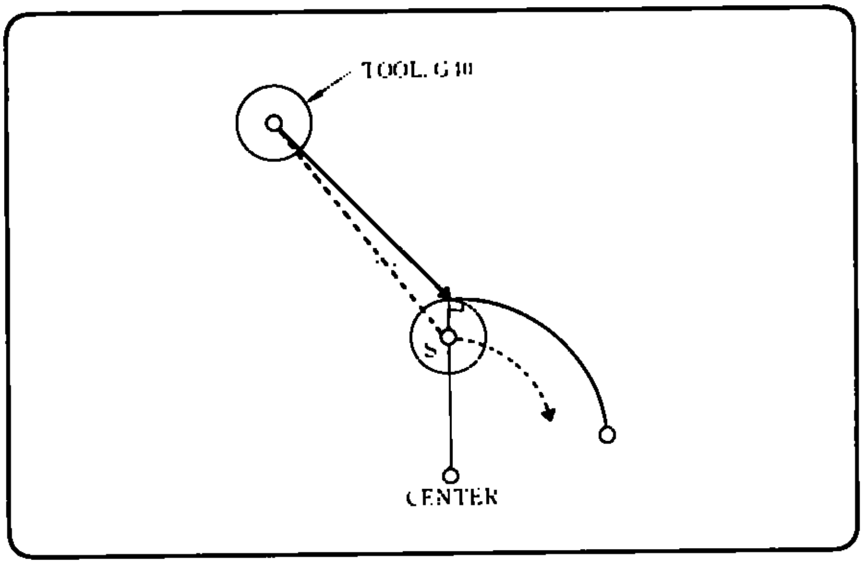


Fig. 2.12.18 Start up ③ of Outside Corner

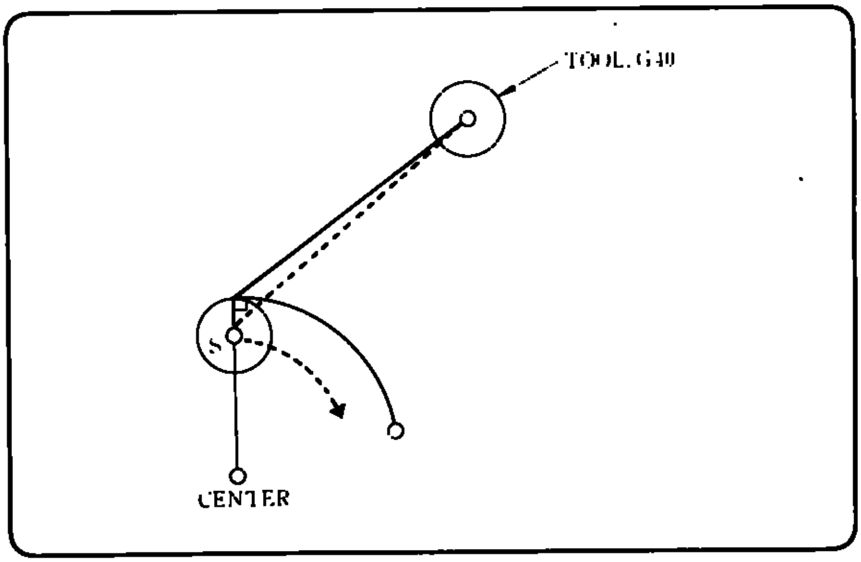


Fig. 2.12.19 Start up ④ of Outside Corner

❷ Type B:

```
G17  G01  F······ ;
G42  D······X······Y······ ;
X······ ;
```

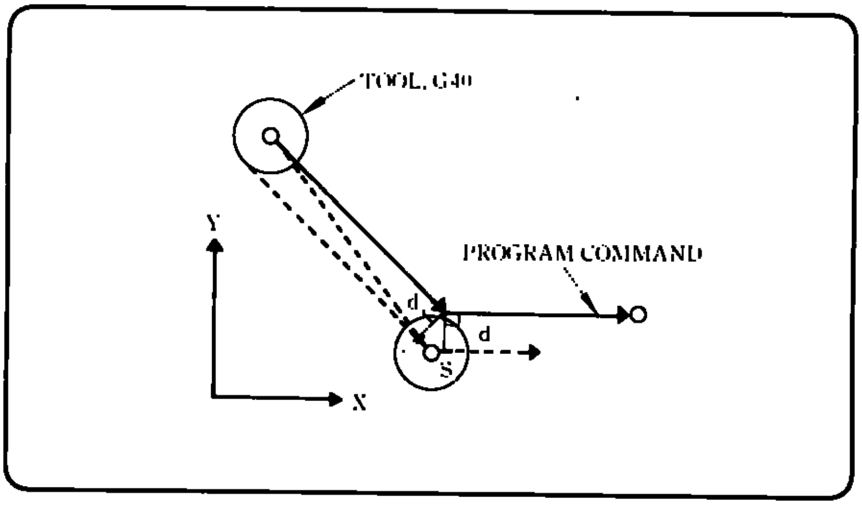


Fig. 2.12.20 **Start up ⑤ of Outside Corner**

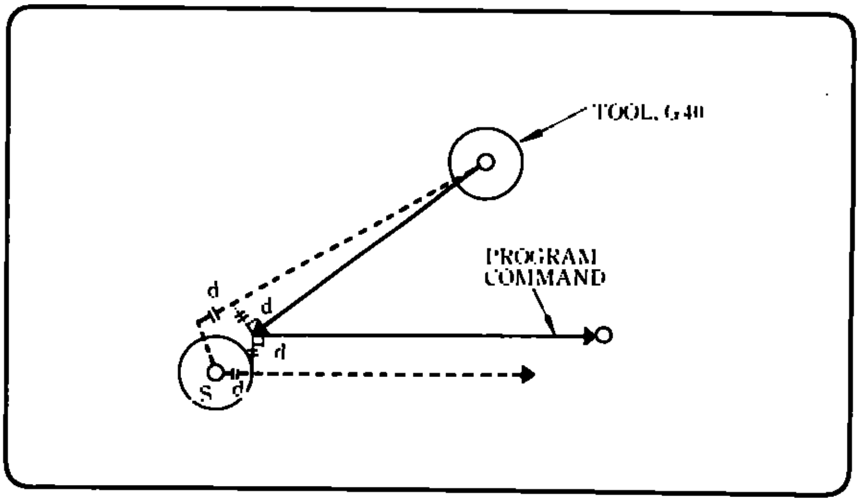


Fig. 2.12.21 **Start up ⑥ of Outside Corner**

```
G17   G01   F······ ;
G42   D······X······Y······ ;
G02   X······Y······J······ ;
```

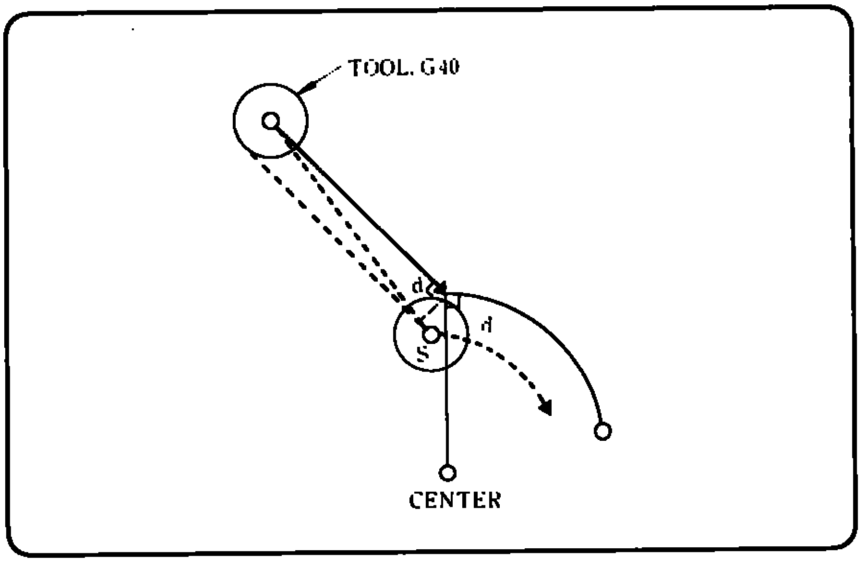


Fig. 2.12.22 Start up ⑦ of Outside Corner

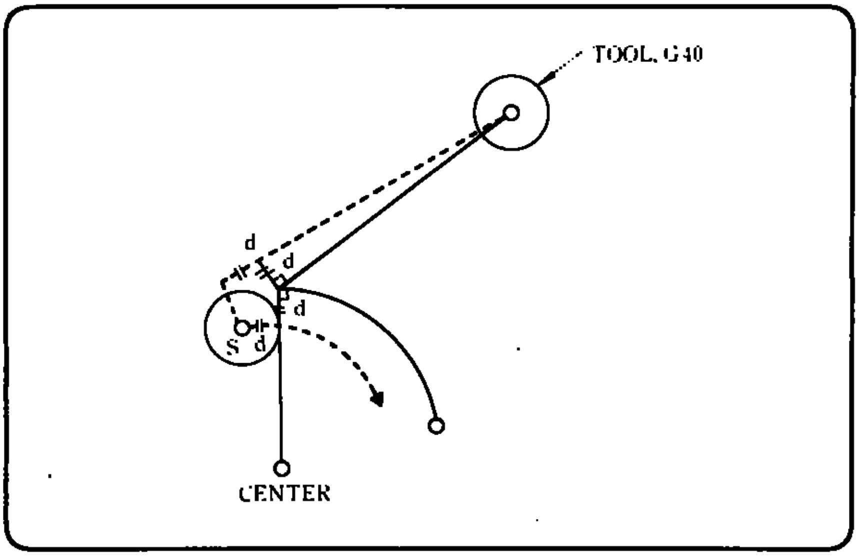


Fig. 2.12.23 Start up ⑧ of Outside Corner

(4) Movement during offset mode

When tool radius compensation mode is entered by G41 (G42), the tool moves along the offset path until G40 is commanded after that. The path is automatically calculated by the NC equipment, so that the diagram shape may only be specified in the program. The tool path is controlled depending on the angle between each block as follows:

(a) Inside corner (180° or less): Intersection calculation method

- Line and line

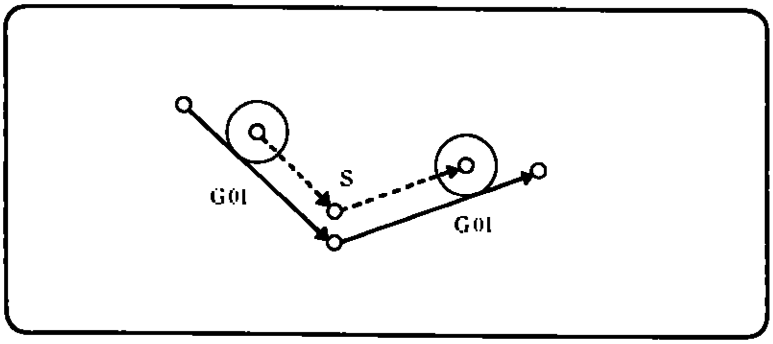


Fig. 2.12.24 Line and line

- Line and arc

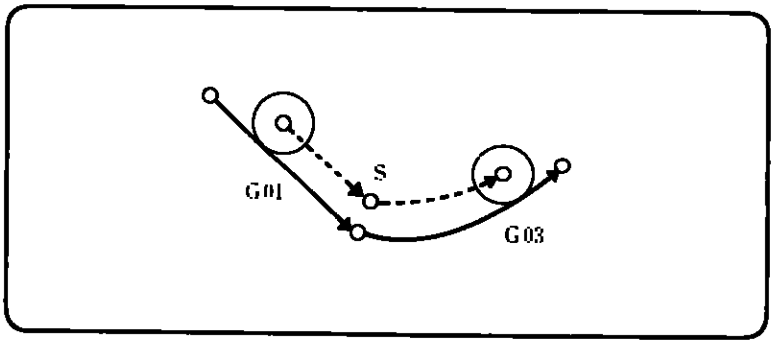


Fig. 2.12.25 Line and Arc

- Arc and arc

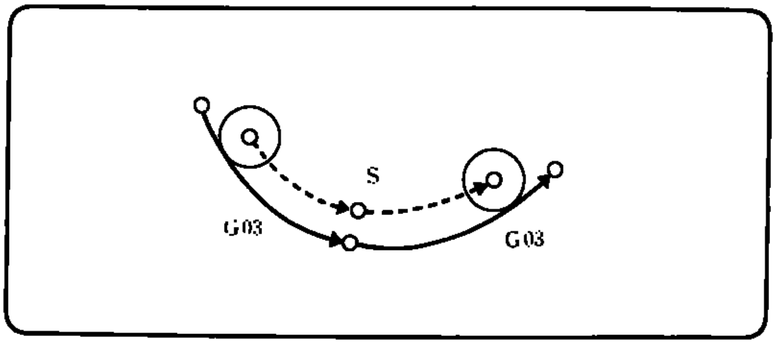


Fig. 2.12.26 Arc and Arc

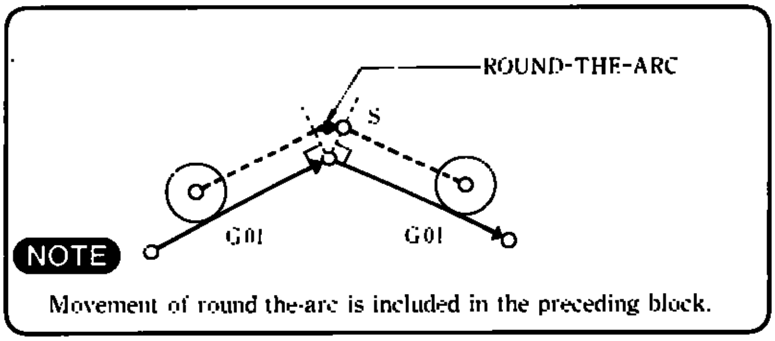
(b) Outside corner (180° or more)

There are two types that can be switched over by the circular section M code.

M96 : Tool radius compensation round-the-arc ON
M97 : Tool radius compensation round-the-arc OFF
(Intersection calculation is executed.)

(i) Round-the-arc ON

- Line and line



NOTE Movement of round the-arc is included in the preceding block.

Fig. 2.12.27 Line and Line

- Line and arc

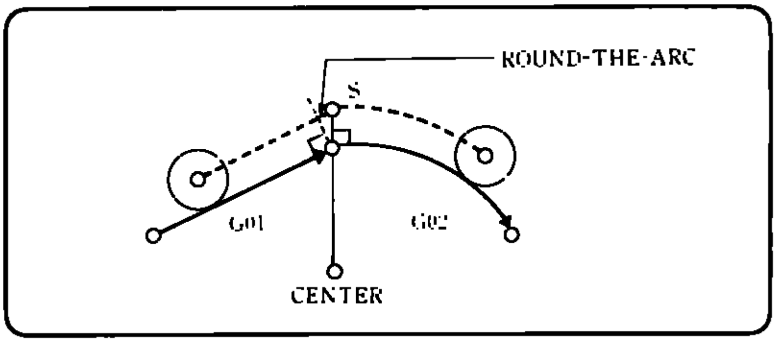


Fig. 2.12.28 Line and Arc

- Arc and arc

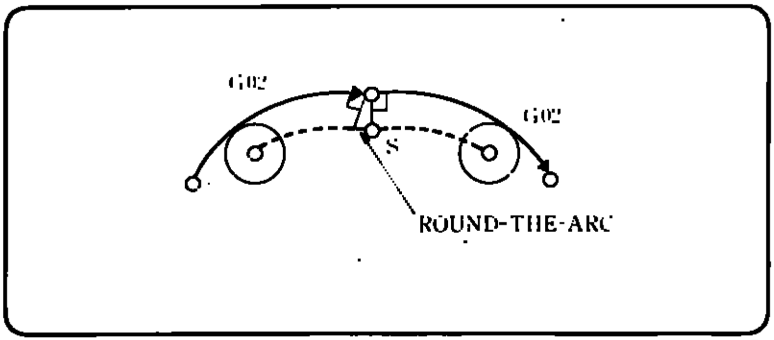


Fig. 2.12.29 Arc and Arc

(ii) Round-the-arc OFF

❶ When material angle α is 90° ≦ α < 180°

- Line and line

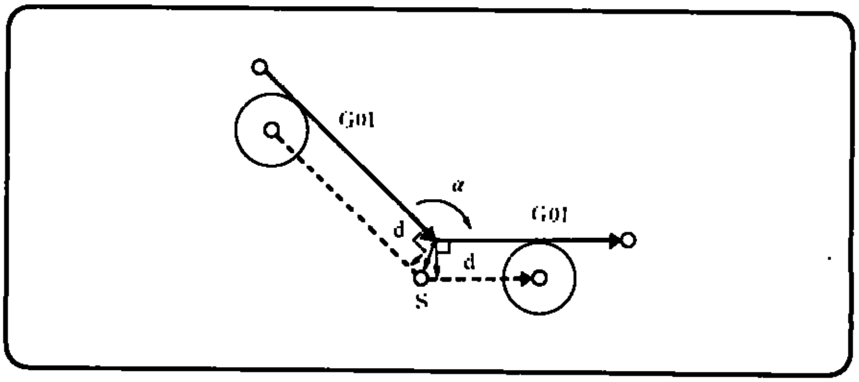


Fig. 2.12.30 Line and Line

- Line and arc

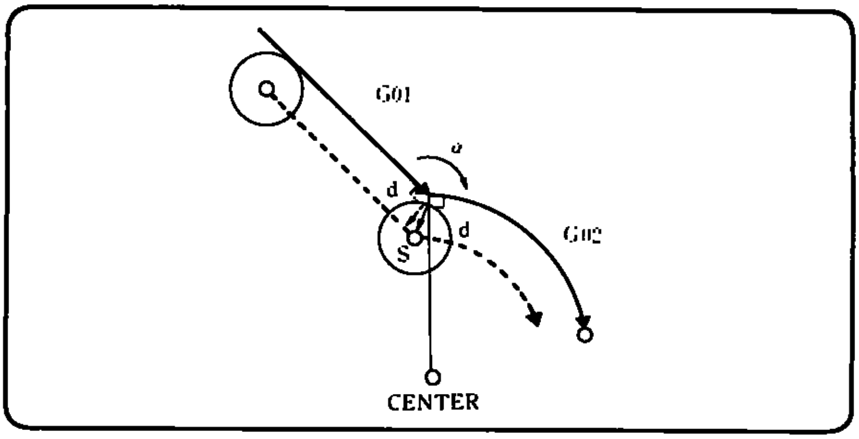


Fig. 2.12.31 Line and Arc

- Arc and arc

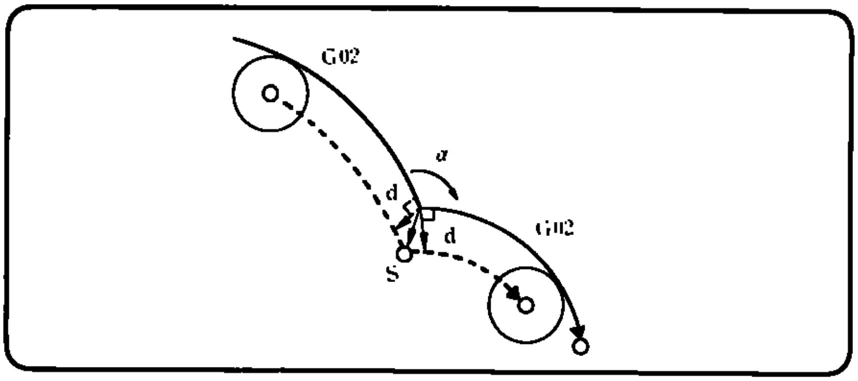


Fig. 2.12.32 Arc and Arc

❷ When material angle α is $\alpha < 90°$

- Line and line

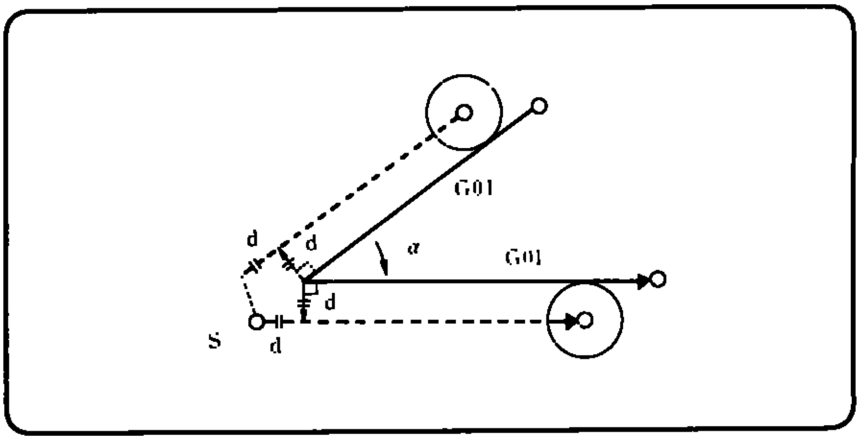


Fig. 2.12.33 Line and Line

- Line and arc

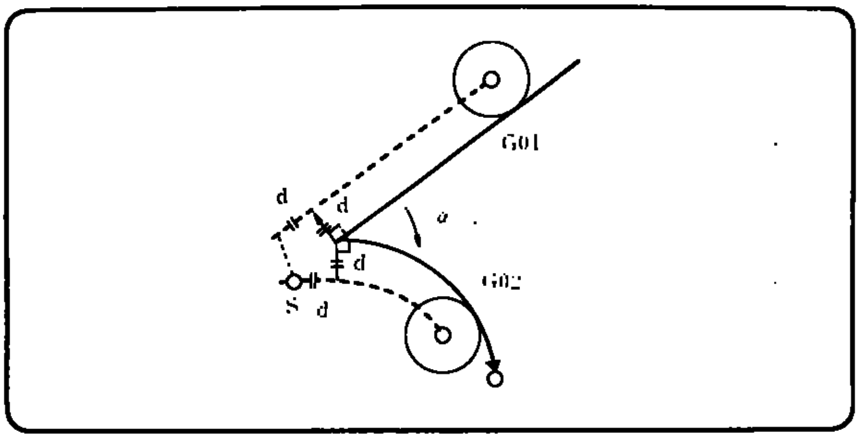


Fig. 2.12.34 Line and Arc

- Arc and arc

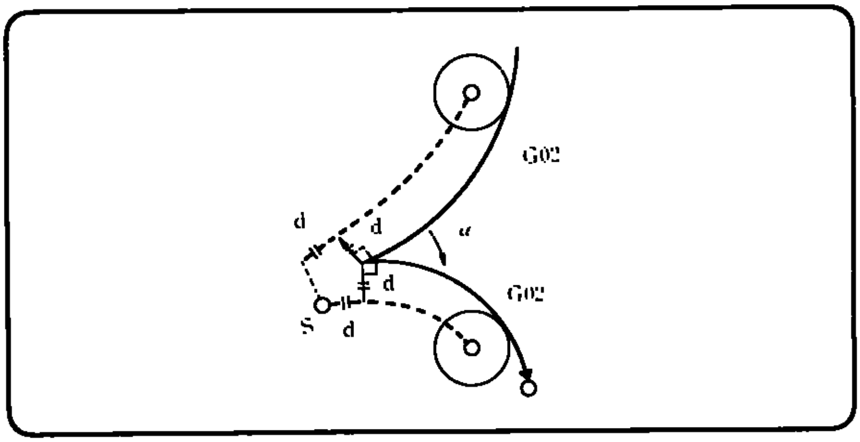


Fig. 2.12.35 Arc and Arc

(c) Movement in G00 mode

A G00-specified block ultimately positioned independently for each axis toward the offset position. Attention should be paid to the tool locus.

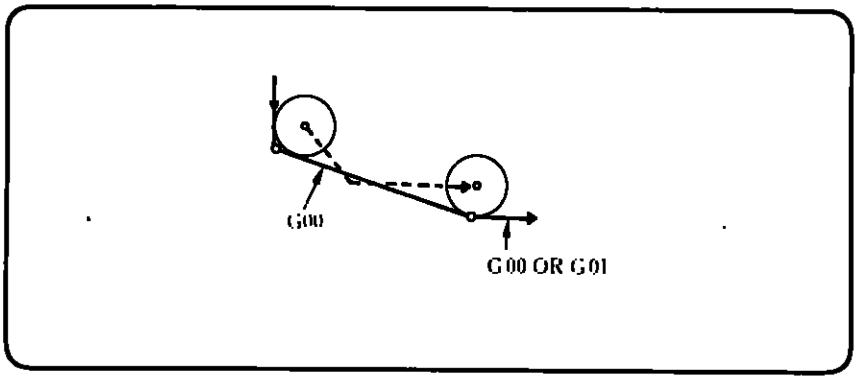


Fig. 2.12.36 Movement ① by G00 Mode

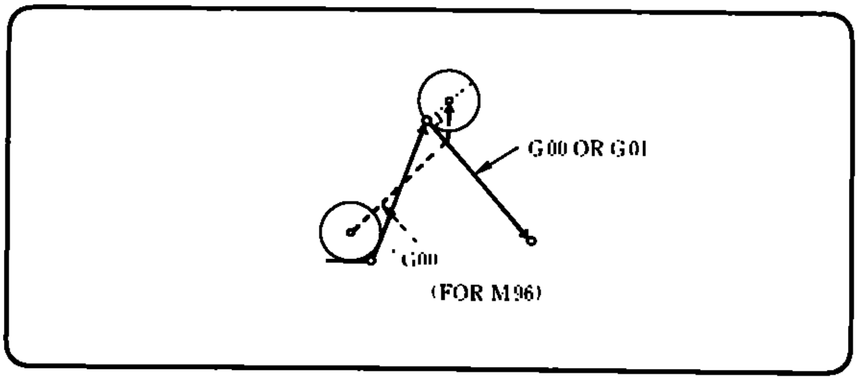


Fig. 2.12.37 Movement ② by G00 Mode

(5) Commands involving no movement in offset mode

(a) During tool radius compensation, the NC equipment normally calculates the tool path by prereading data for two blocks ahead. If a block without coordinate command, e.g., G04 (dwell), is included in these preread blocks, calculation is made by prereading one block further ahead. Such non-coordinate-commanded blocks can be processed for up to two consecutive blocks. If there is no coordinate command in three or more blocks, calculations are made for tool radius to obtain the correct path. Therefore, when commanding G41(G42), care must be taken in the subsequent shape-indicating programs so that move commands in the offset plane are not discontinued in three or more blocks.

Example of Programming

```
      G17  G01  G41  X······Y······D······F······ ;
      X······Y······ ;
               ⋮
      X······Y······ ;
G04   P1000 ;
      X······Y······ ;
               ⋮
      X······Y······ ;
      Z······ ;
      Z······ ;
      X······Y······ ;
               ⋮
      X······Y······ ;
      G40  X······Y······ ;
```

Block involving no movement in offset plane (If the number of such a block is less than three, commands may be smoothly connected.)

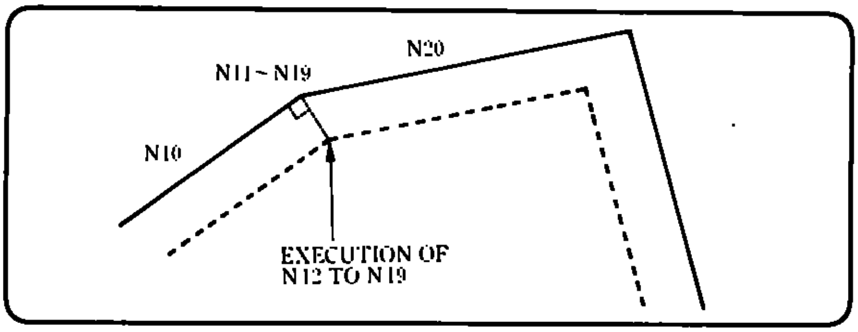


Fig. 2.12.38 Positioning if there is No Coordinate Command in Four or more Blocks

(b) If there is no move command in four consecutive blocks, the tool is positioned on the end point normal line by the offset quantity in the immediately preceding block. In such a case, if movement in the offset plane cannot be commanded in three or more consecutive blocks for an unavoidable reason (e.g., third axis escape operation) and, in addition, positioning on the normal line is undesirable, one or more I, J, or K-based dummy blocks may be inserted.

Example of Programming

```
N001  G17  G01  G41  X·····  Y·····  D·····  F····· ;
N002  X·····  Y····· ;         |
      :                        | XY PLANE
N010  X·····  Y····· ;         |
N011  I·····  J····· ;  ·····DUMMY BLOCK
N012  Z·····  ;                | Z AXIS
      :                        | (4 BLOCKS
N019  Z·····  ;                |  MIN.)
N020  X·····  Y····· ;         |
      :                        | XY PLANE
N029  X·····  Y····· ;         |
N030  G40  X·····  Y····· ;
```

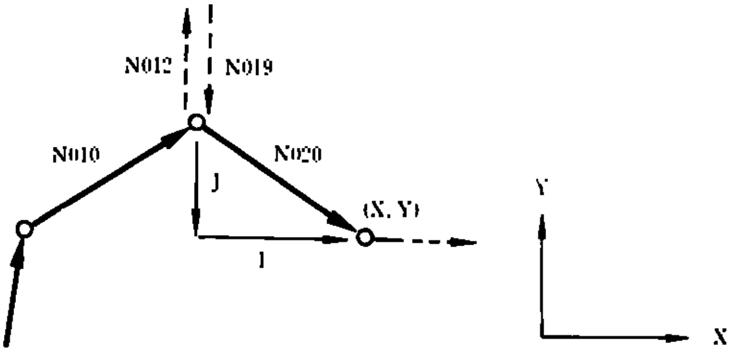


Fig. 2.12.39 Dummy Block Insertion

Dummy blocks do not involve actual movement, and only give the necessary data for the calculation of tool radius compensation. In the above example, the same command as the first block (N020) of movement in the XY plane that is restarted after Z-axis movement is commanded by using I and J as dummy commands. I, J, or K are used for the addresses of this dummy command, and correspond to X, Y, and Z axes. Use them according to plane specifications.

I: Dummy for X-axis command
J: Dummy for Y-axis command
K: Dummy for Z-axis command

To be commanded with incremental values.
If N020 X······Y······in the above example is an absolute specification, the command must be made after converting it to an equivalent incremental value.

NOTE If a dummy block is to be created for circular interpolation, create it as follows:

Example of Programming

```
N050  G10  X······Y······ ;
N051  G10  I  (b)  J  (−a)  ;  —— Dummy block
N052       Z······ ;  |
N052       Z······ ;  }  Z-axis
N059       Z······ ;  |
N060  G03  X······Y······  I (a)  J (b);  —— Circular
                                              interpolation
N061  G01  X······Y······ ;
```

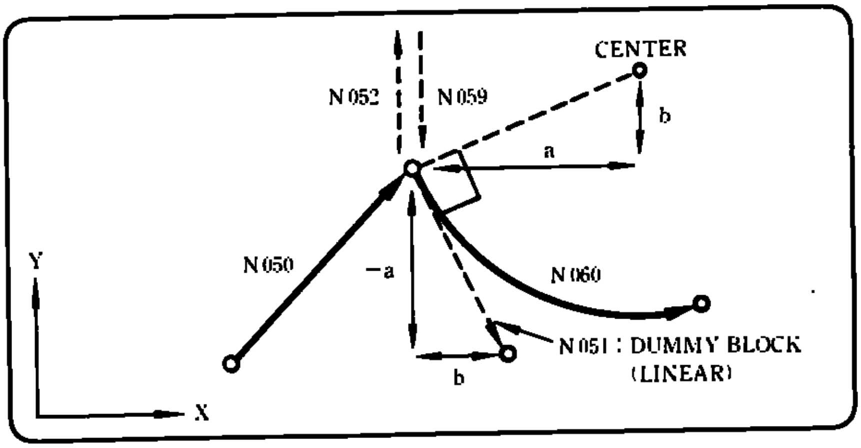


Fig. 2.12.40 Dummy Block Insertion

In this case, insert a linear dummy block to specify the tangential direction at the start point of the commanded circular arc as shown in Fig. 2.12.40. Pay attention to the sign of dummy block data depending on the shape of circular arc. The tool is stopped at point A by the dummy block, to be prepared for the next circular command.

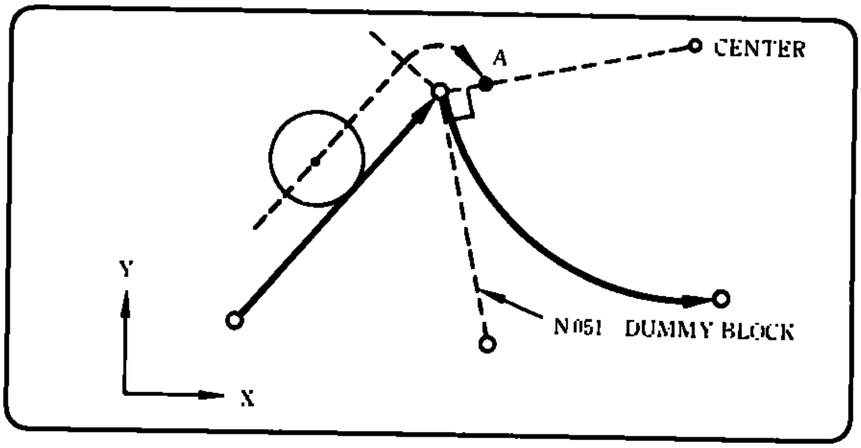


Fig. 2.12.41 Stop at point A by Dummy Block

NOTE If an I-, J-, or K-based dummy block is commanded along with a move command in the offset mode, I, J, or K are disregarded. However, if I, J, or K is commanded when offset is canceled, the offset position is corrected from point 1 to point 2 by direction specification by I or J.

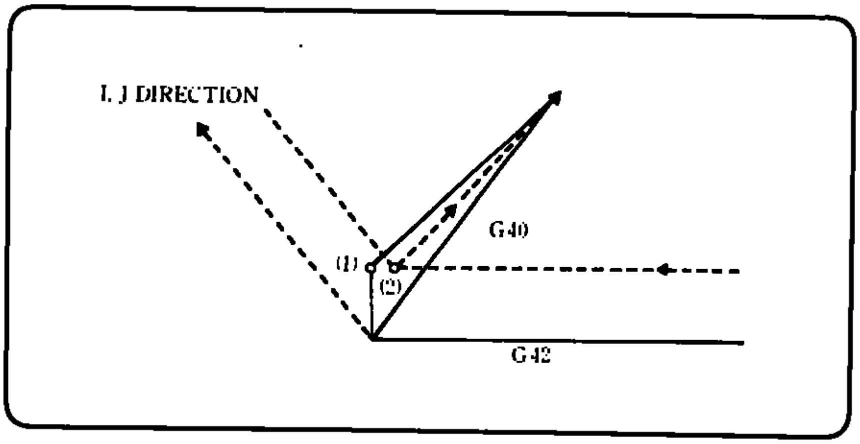


Fig. 2.12.42 Correction of Offset Position

(6) G41/G42 switching in offset mode

Tool radius compensation can be switched directly between G41 and G42 during the offset mode without being canceled by G40.

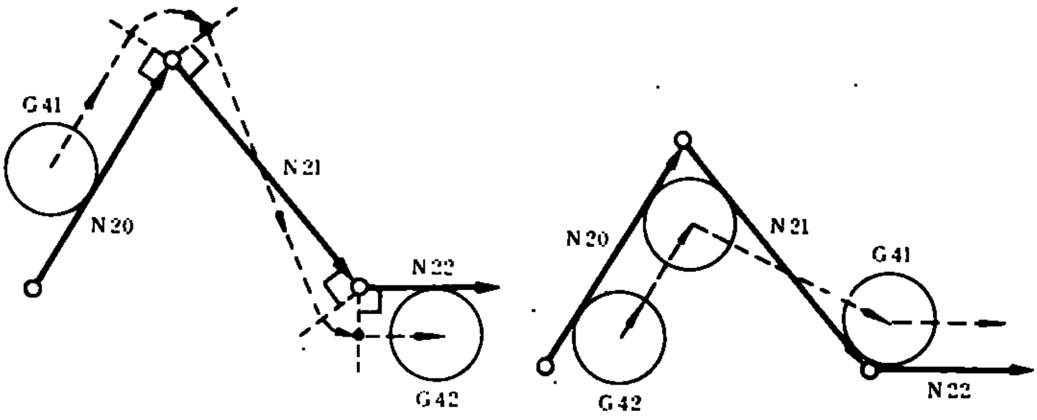
There are two methods of switching which can be selected by a parameter.

Pm4013 D1 = 1 : Type A
Pm4013 D1 = 0 : Type B

(a) Type A: Switching is accomplished by using the start and end of the direction switching block

Example of Programming

```
N10  G17  G01  F····· ;
N11  G41  (G42)  D····· ;
            :
N20  G01  X·····  Y·····  F····· ;
N21  G42  (G41)  X·····  Y····· ;   —— SWITCHING BLOCK
N22            X····· ;
```



(a) G41 → G42 (M96 MODE)

(b) G42 → G41

NOTE Even when commanded by dividing the N21 block into two blocks as shown below,

```
G42 (G41) ;
     X·····  Y····· ;
```

the same results as above are obtained.

Fig. 2.12.43 Switching of Start and End in Direction Switching Block

(b) Type B: Switched at the intersection of tool center path (offset-applied path)

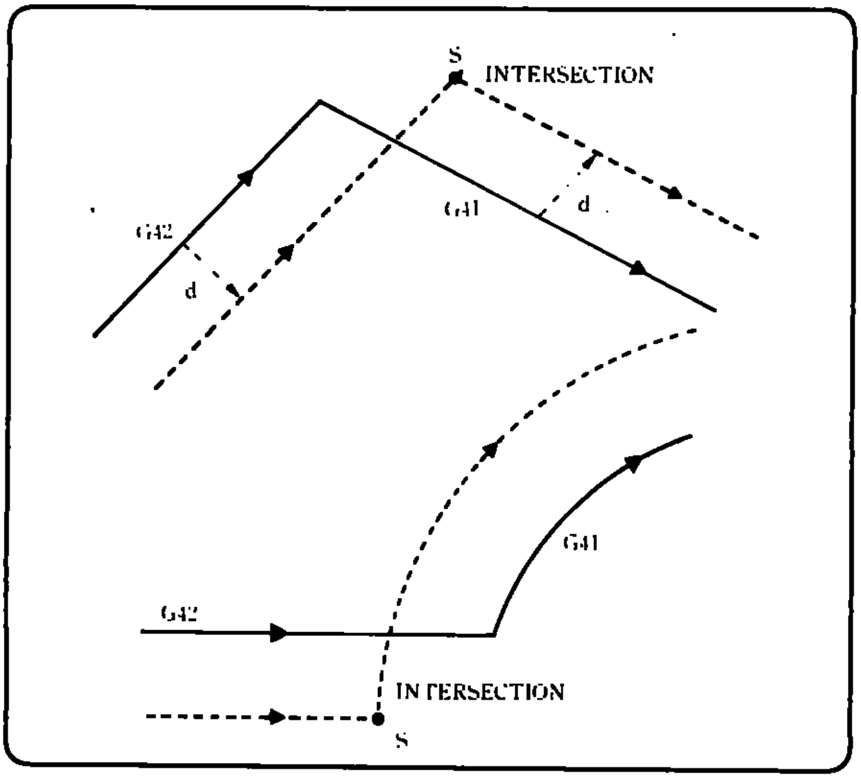


Fig. 2.12.44 Switching at Intersection of Tool Center Path

NOTE If there is no intersection, switching is accomplished by the Type A method.

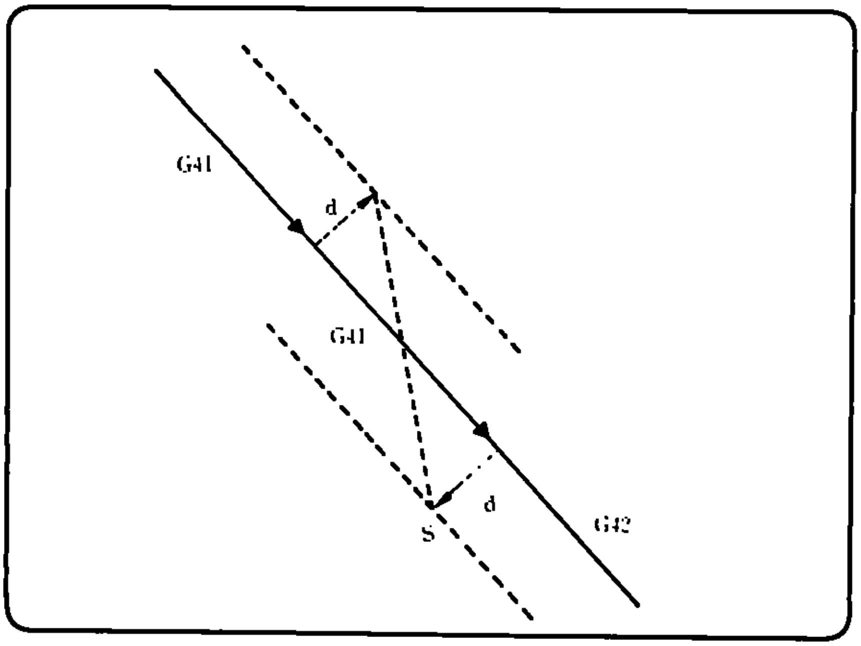


Fig. 2.12.45 Switching for No Intersection

(7) Changing tool offset quantity in offset mode
There are two methods which can be selected by a parameter.

Pm4013 D2=1 : Type A
Pm4013 D2=0 : Type B

Type A: A new offset quantity is made effective by offset calculation of the block in which D code is commanded and the next block.

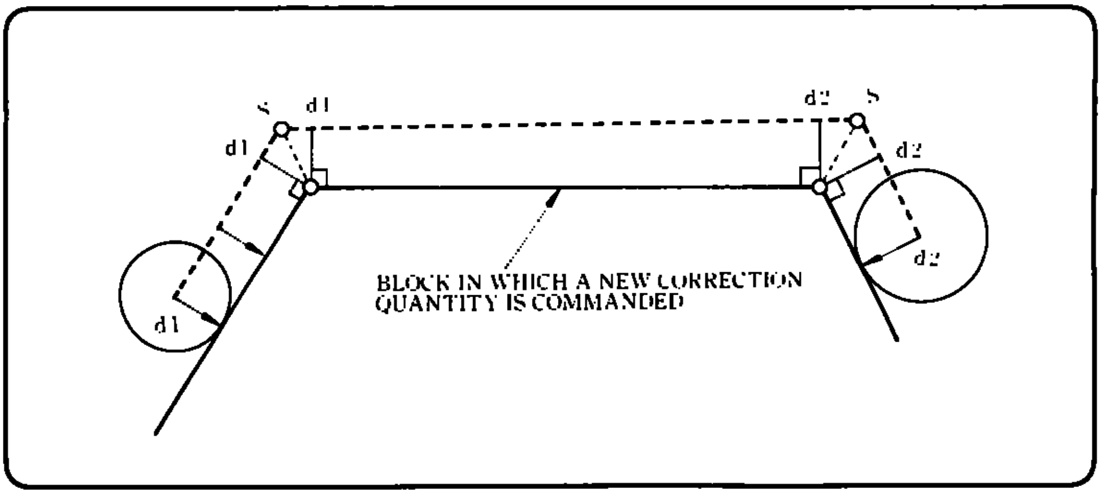


Fig. 2.12.46

Type B: A new offset quantity is made effective by offset calculation of the block in which D code is commanded and the block that precedes D-code block.

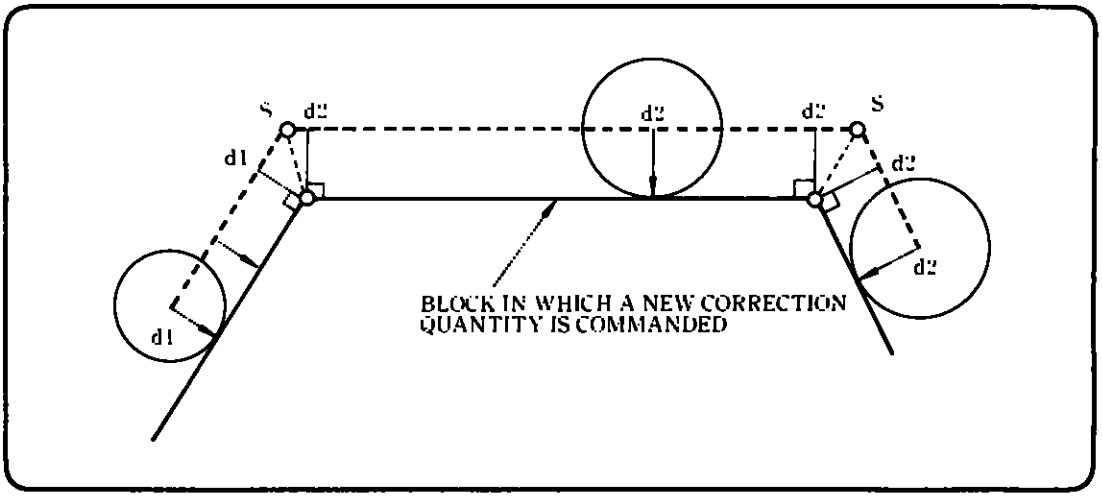


Fig. 2.12.47

(8) Method of canceling offset

(a) There are two types which can be switched by parameter.

```
Pm 4012 D6 = 0 : Type A
Pm 4012 D6 = 1 : Type B
```

① Type A : Does not move when move command is not provided in G40 command block.
The offset mode is canceled by the first move command given in a block specified following the G40 block. In principal, move command for the offset plane should be specified with G40 in the same block.

② Type B : Moves even if move command is not provided in G40 command block.
G40 cancels tool radius compensation C and performs positioning at the end point as programmed.
In this case, the tool moves to the normal line of the end point of the block immediately before G40.

(b) Because G40 involves move operation for cancellation, if it is other than G00 or G01, alarm "181" results as in the case of G41 (G42).

(c) Offset cancellation is divided into two types depending on the shape.

- Cancellation of inside corner (180° or less)

```
G41
  :
G01  X ······ F ······ ;
G40  X ······ Y ······ ;
```

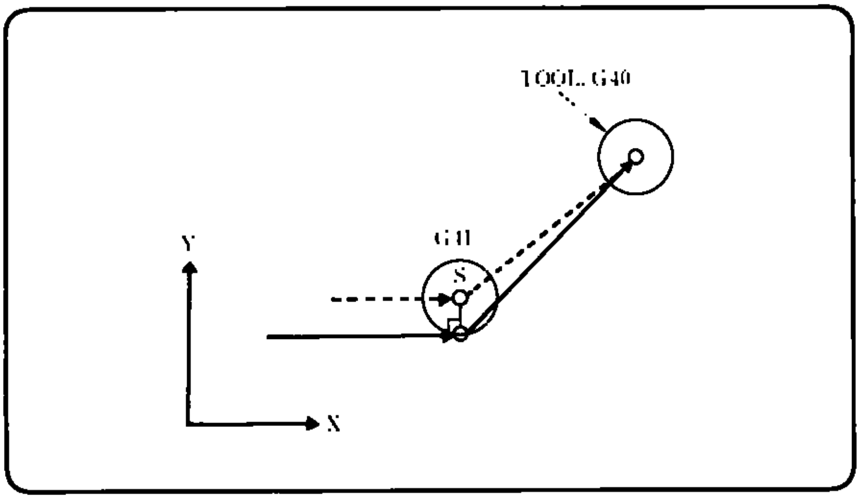


Fig. 2.12.48 Cancellation (Line) of Inside Corner (180° or less)

```
G41
  ⋮
G02  X ······ Y ······ I ······ J ······ ;
G01  G40  X ······ Y ······ ;
```

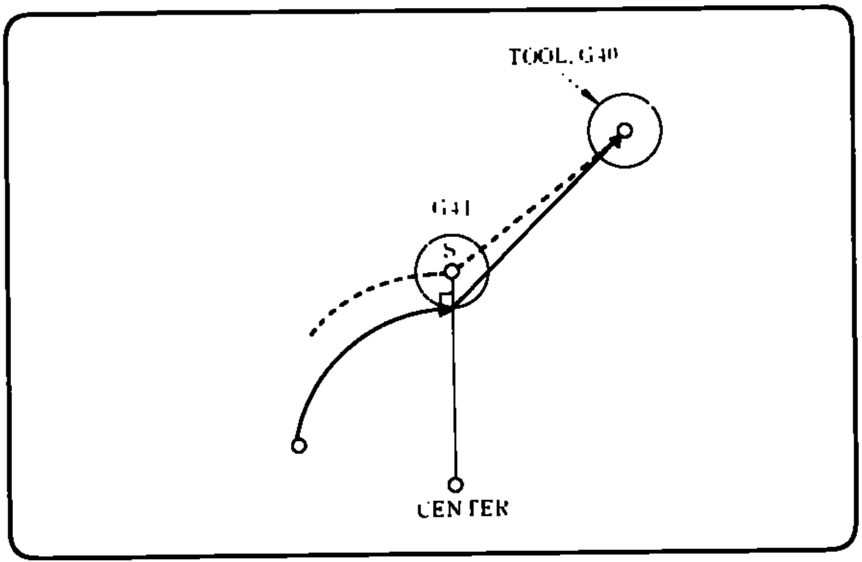


Fig. 2.12.49 Cancellation (Arc) of Inside Corner (180° or less)

- Cancellation of outside corner (180° or more)

There are two types which can be selected by a parameter.

Pm4013 D0=1 : Type A
Pm4013 D0=0 : Type B

This parameter is the same as that used for startup.

❶ Type A: End point is reached as programmed via the offset position on the end point normal line of the block immediately before cancellation.

```
G42
 :
G01  X …… F …… ;
G40  X …… Y …… ;
```

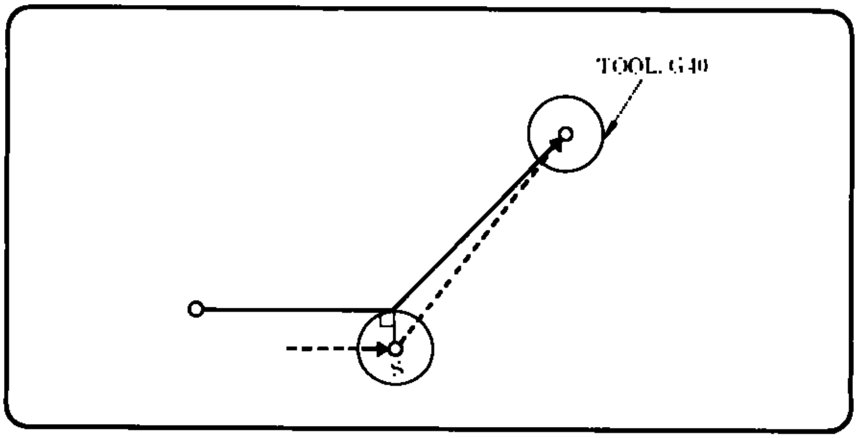


Fig. 2.12.50 Cancellation ① of Outside Corner (180° or more)

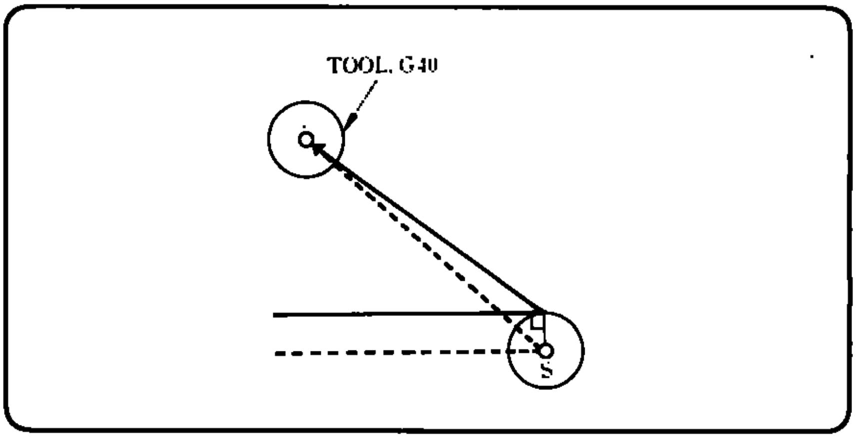


Fig. 2.12.51 Cancellation ② of Outside Corner (180° or more)

```
  G42
    :
 G02  X …… Y …… I …… J …… ;
 G01  G40  X …… Y …… ;
```

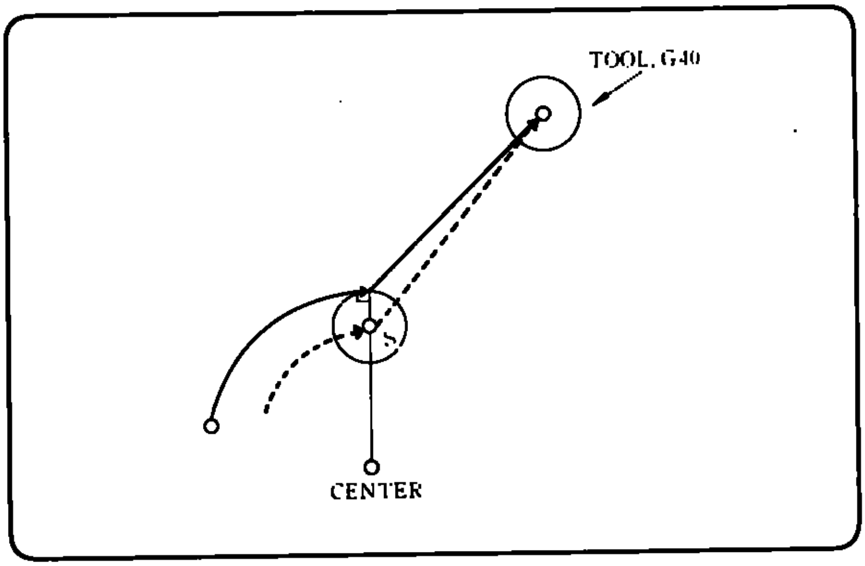


Fig. 2.12.52 Cancellation ③ of Outside Corner (180° or more)

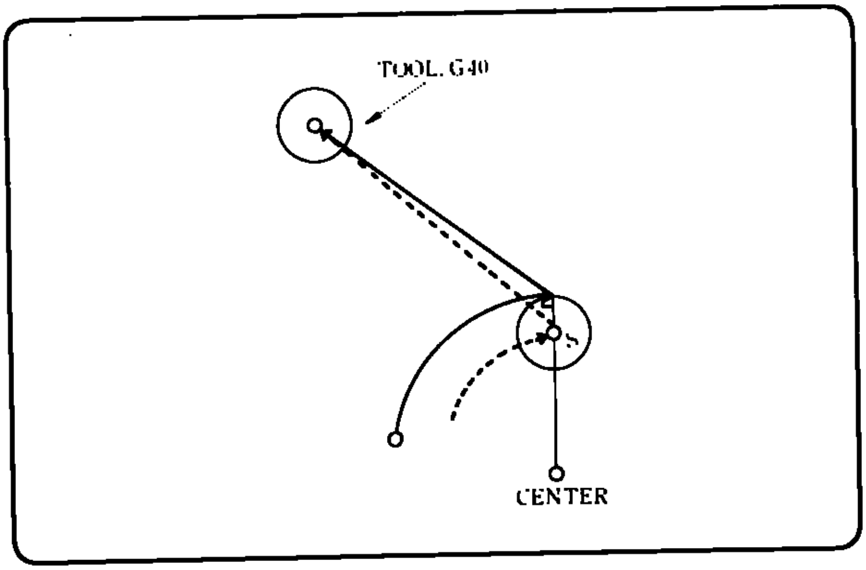


Fig. 2.12.53 Cancellation ④ of Outside Corner (180° or more)

❷ Type B:

```
  G42
    ⋮
G01  X ······ F ······;
G40  X ······ Y ······;
```

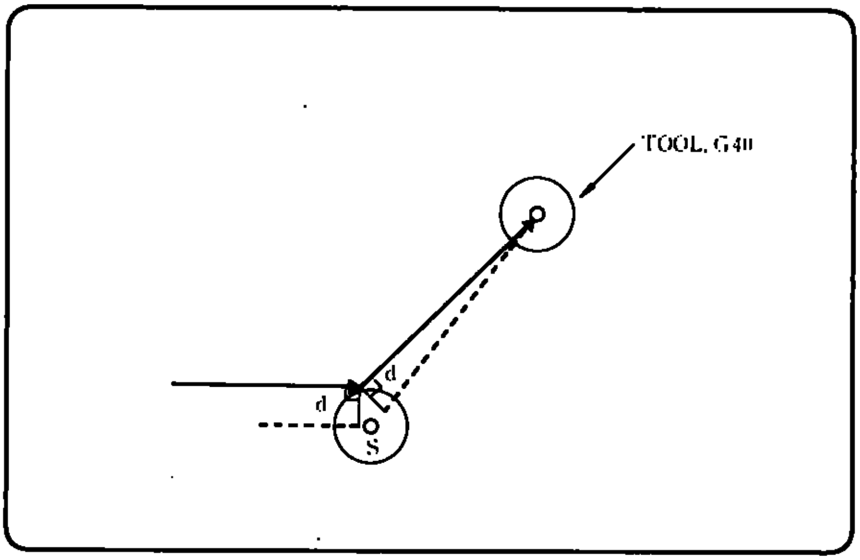


Fig. 2.12.54 **Cancellation ⑤ of Outside Corner (180° or more)**

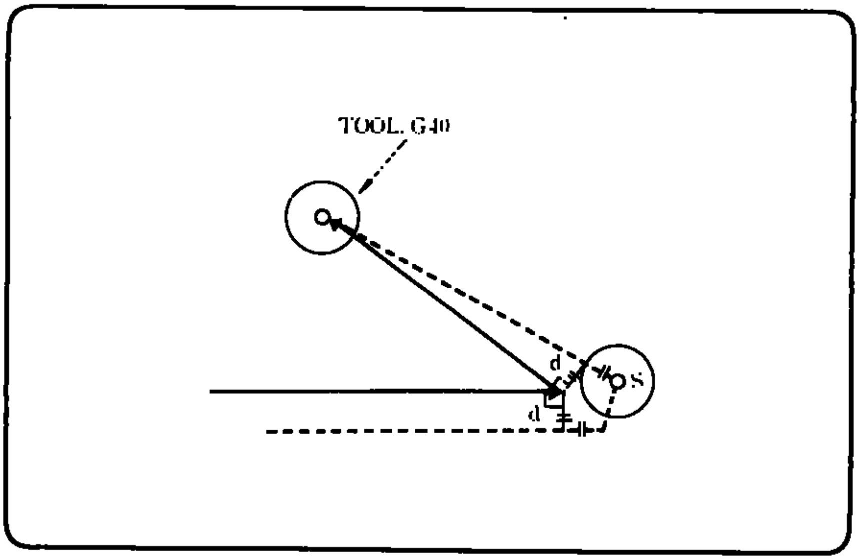


Fig. 2.12.55 **Cancellation ⑥ of Outside Corner (180° or more)**

```
 G42
  ⋮
G02  X ······ Y ······ I ······ J ······ ;
G01  G40  X ······ Y ······ ;
```

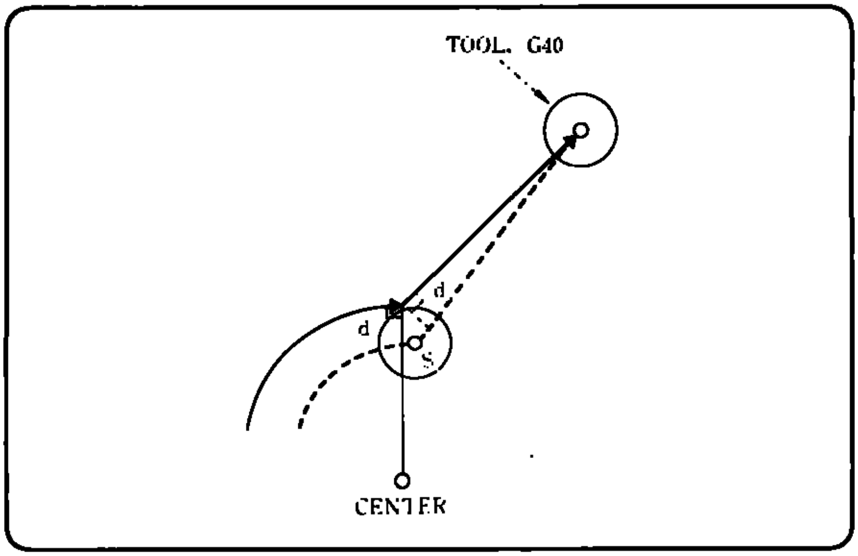


Fig. 2.12.56 Cancellation ⑦ of Outside Corner (180° or more)

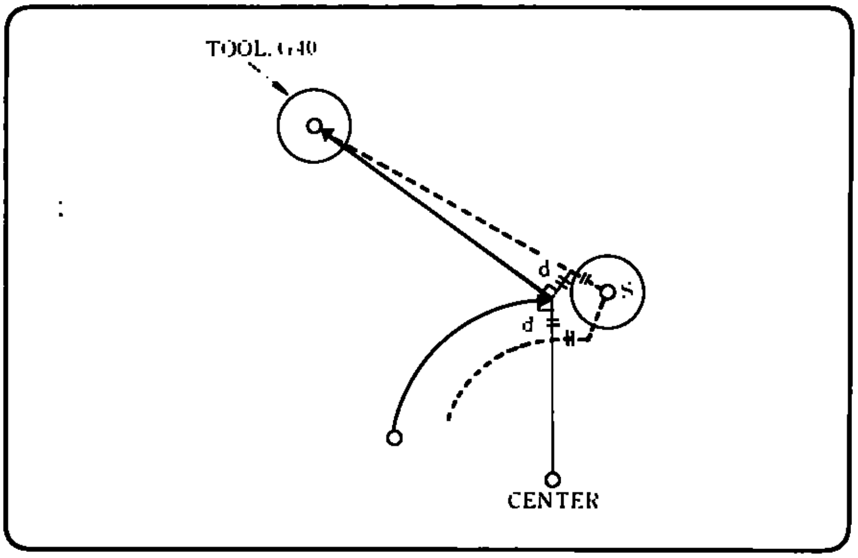


Fig. 2.12.57 Cancellation ⑧ of Outside Corner (180° or more)

(9) Interference check

This function prevents the tool from accidentally cutting into material other than the intended workpiece (interference). If there is a possibility of interference, an alarm may be generated (type A) or the tool center path is corrected (type B).

Type A and type B can be selected by a parameter.

Pm4013 D4=1 : Type A
Pm4013 D4=0 : Type B

Interference check can be changed to valid/invalid by the following parameter.

Pm 4013 D6 = 0 : With interference check
Pm 4013 D6 = 1 : Without interference check

(a) Interference definition

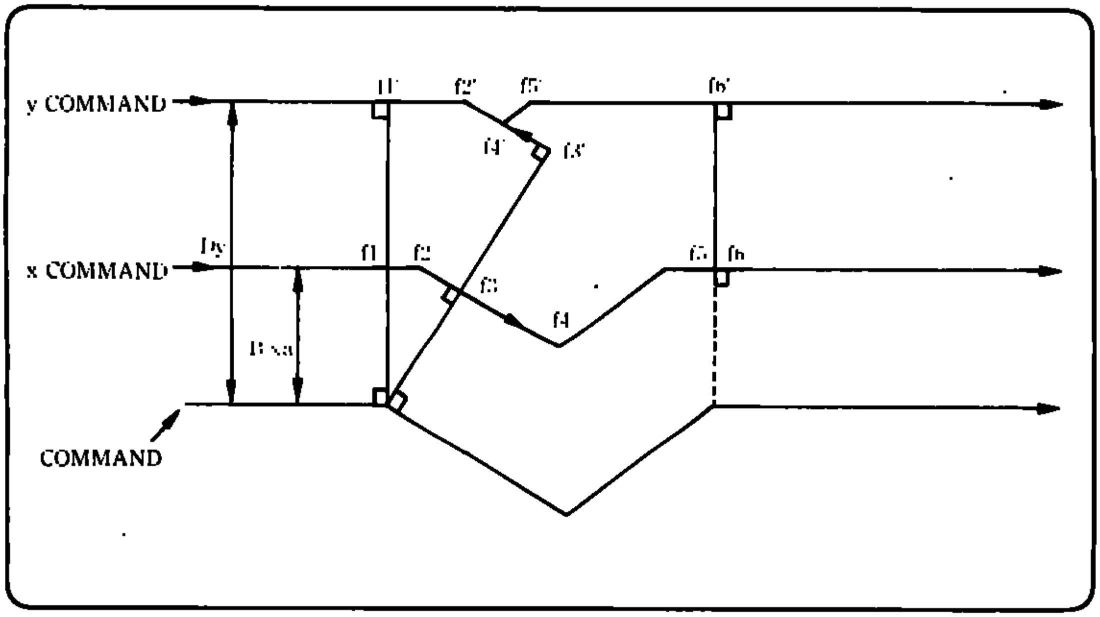


Fig. 2.12.58 Interference Definition

For the above command, the tool radius path for tool radius compensation quantity Dy is f1 → f2 → f3 → f4 → f5 → f6; for Dx, f3 → f4 is 180° different from the correct command direction f3′ → f4′ in f1′ → f2′ → f3′ → f4′ → f5′ → f6′. This instance is assumed to be an interference, and an alarm is generated.

(b) Type A: Alarm generation

(i) Because cut-in occurs at end point (2) of N1, alarm "187" results and the tool is stopped at start point (1) of N0.

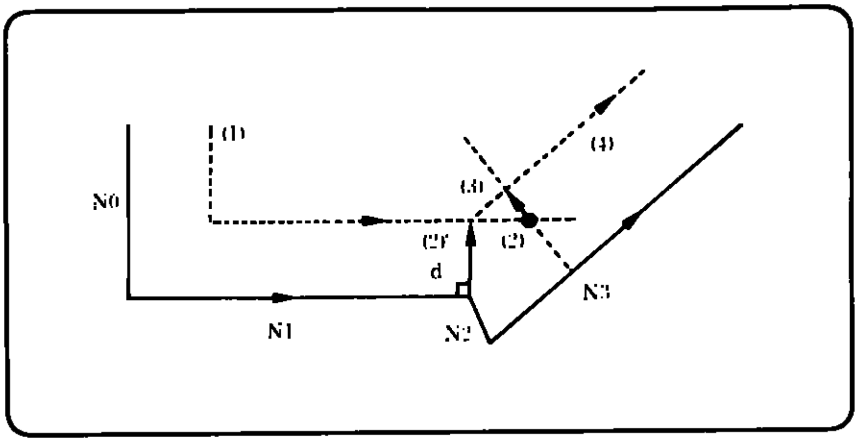


Fig. 2.12.59 Alarm Occurrence ①

(ii) Because cut-in occurs at end point (2) of N1, alarm "187" results and the tool is stopped at start point (1) of N0.

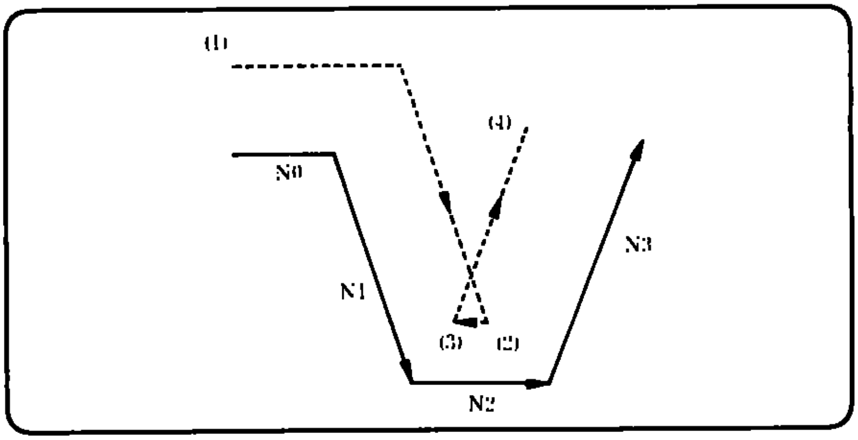


Fig. 2.12.60 Alarm Occurrence ②

If the programmed block move direction differs greatly from the offset path move direction as shown above, it is assumed that cut-in (interference) occurs.

NOTE Interference check is not performed at offset startup.

(c) Type B: Automatic offset at occurrence of interference

If it is assumed that interference occurs as a result of tool radius compensation calculation, interference-generating movement is erased and a non-interfering path is created.

(i)

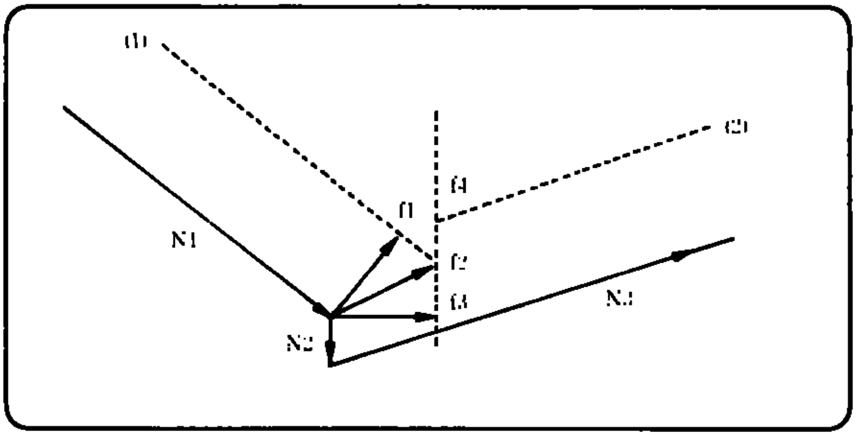


Fig. 2.12.61 Automatic Correction ① at Occurrence of Interference

In (i), three points f1, f2, f3 are created at the N1-N2 connection by radius compensation calculation. Also, f4 is created at N2-N3. Here, interference check is executed for four points from f1 to f4, and interference-generating points are sequentially erased and the tool center path is created by connecting the remaining points.

Checked for f3 — f4 : Interfering f3 erased
Checked for f2 — f4 : Interfering f2 erased
Checked for f1 — f4 : No interference
Moved to (1) — f1 — f4 — (2)

(ii)

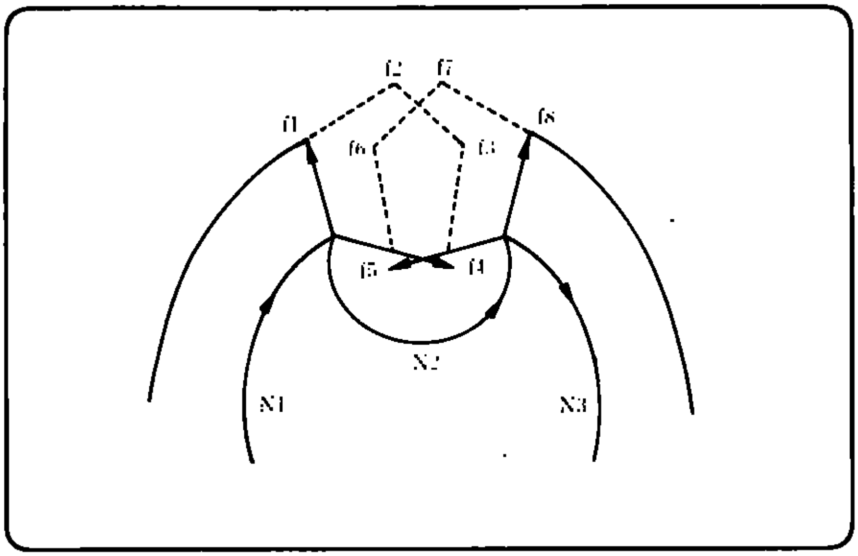


Fig. 2.12.62 Automatic Correction (2) at Occurrence of Interference

Checked for f4 — f5: Interfering f4, f5 erased
Checked for f3 — f6: Interfering f3, f6 erased
Checked for f2 — f7: No interference
Moved in order of f1 → f2 → f7 → f8

(iii)

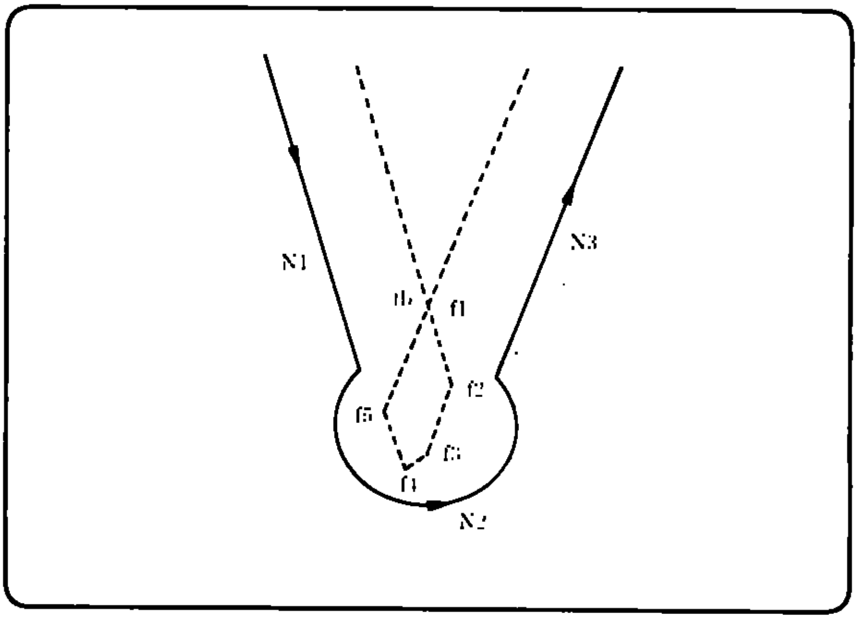


Fig. 2.12.63 Automatic Correction ③ at Occurrence of Interference

f1, f2, f3 at N1 – N2
f4, f5, f6 at N2 – N3

Checked for f3 — f4: Interfering f3, f4 erased
Checked for f2 — f5: Interfering f2, f5 erased
Checked for f1 — f6: Interfering f1, f6 erased
Alarm generated and tool stopped at start point of N1 block

When connecting points are erased by interference check, if all connecting points are erased as in the case of Fig.2.12.63, alarm "188" is generated and the tool is stopped unconditionally.

NOTE

(Precautions on tool radius compensation-C)

1. Limits on the maximum command values of coordinate words (Tables 2.1.12, 2.1.13 and 2.1.14) are applied even in tool radius compensation-C.
2. Command shapes causing alarm

   The following command shapes result in alarm "184."

(a) When an arc

[radius of command circle $r+5 \leqq$ tool radius d]

is commanded for inside offset of arc command.

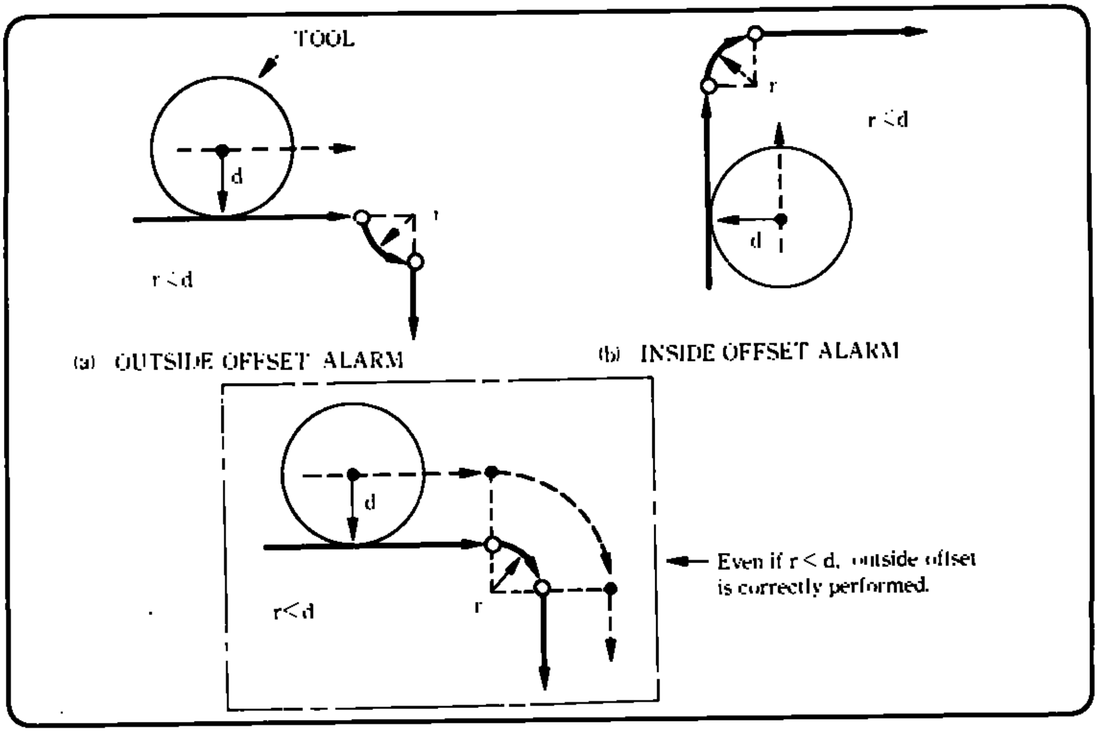


Fig. 2.12.64 Command ① Occurred Alarm

(b) When no intersection resides on the offset tool center locus.

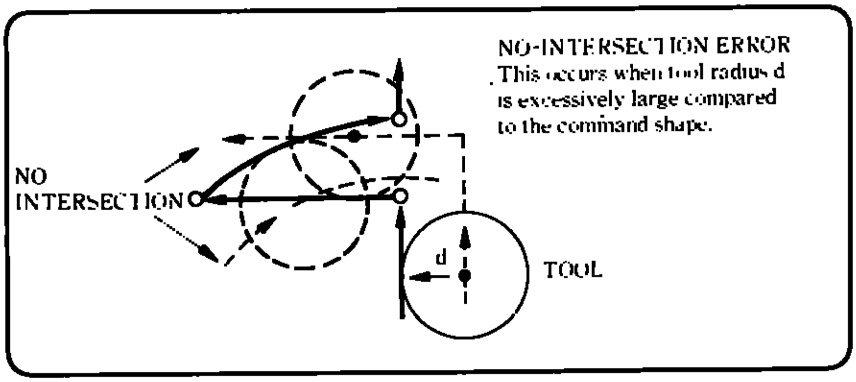


Fig. 2.12.65 Command ② Occurred Alarm

3. When any of the following G codes is commanded during the offset mode, alarm "182" results.

Table 2.12.12 Prohibited G codes

	Prohibited G codes
G codes that cause alarm	G12, G13 (G17 to G19) G28, G29 G73, G74, G76, G77, G81 to G89 G92

NOTE When reset in the offset mode, offset is canceled and G40 results.

4. Tool radius compensation-C is applied to the movement shape that is offset by tool length offset and tool position offset. As a rule, avoid applying tool radius compensation-C after using tool position offset to correct the tool radius. This is because movement-to-data correspondence becomes difficult to understand.
5. Before G41 or G42 and G40 can be commanded, G00 or G01 and F code must be commanded in the same block or preceding block.
6. If plane-specifing G codes G17 to G19 are commanded in offset to change the offset plane, alarm "183" results.
7. Exact round cutting (G12, G13) and canned cycle (G73, G74, G76, G77, G80 to G89)are commanded in the tool radius compensation cancel mode. Exact round cutting itself has the tool radius compensation function. Commanding these codes in the offset mode results in alarm "182."
8. Tool radius compensation-C is also possible for circular interpolation by radius specification.
9. Subprograms (M98, M99) may be commanded during the offset mode without any problem.
10. If simultaneous three-axis movement (* up to five axes) is commanded in the offset mode, offset is applied to the projected diagram onto the offset plane specified by G17, G18, or G19.

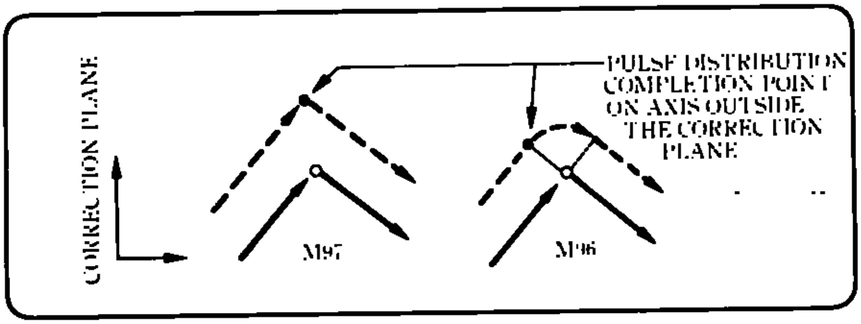
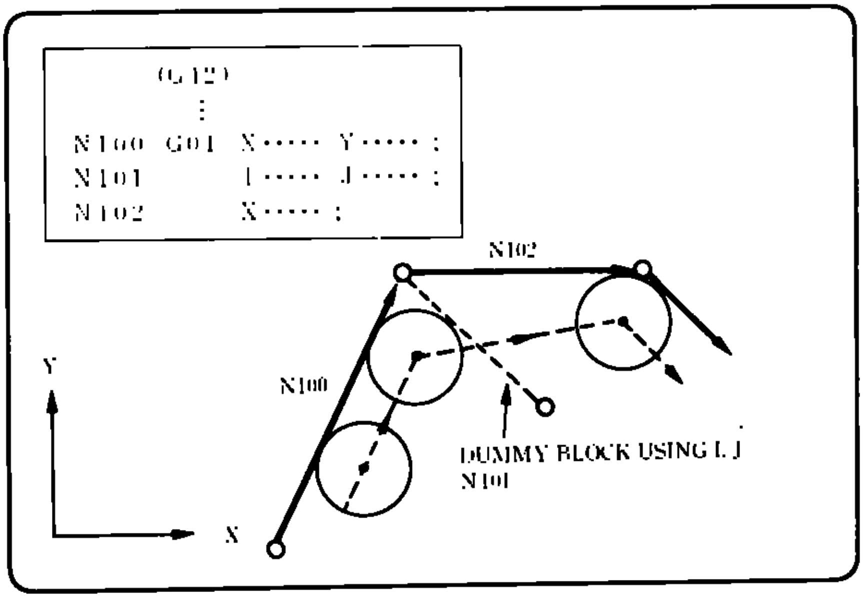


Fig. 2.12.66

11. If circular interpolation outside the offset plane specified by G17, G18, or G19 is commanded in the offset mode, alarm "183" results.

12. The offset position can be temporarily corrected by commanding a dummy block using addresses I, J, or K.



```
      (G12)
        ⋮
N100  G01  X·····  Y····· ;
N101       I·····  J····· ;
N102       X····· ;
```

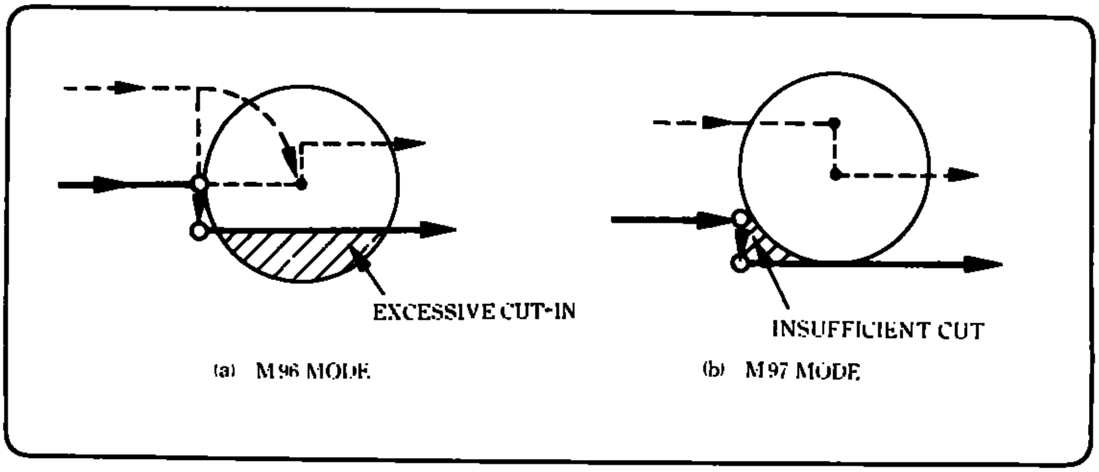
Fig. 2.12.67

13. When a M00 or M01 (M02, M30) command is received, block preread is disabled, so that normal precise correct offset is stopped prior to completion. To avoid this, insert a dummy block using I, J, or K into a block immediately before the M00, M01 command. Precise offset can be continued in this way.

```
      (G41)
N200  G01  X······ Y······ ;
N201       I······ J······ ;
N202       M00     ;
N203       X······ Y······ ;
```

N203 move data has been commanded by I, J, K.

14. The number of tool offset value memories available differs between basic and optional specifications. The maximum value of tool radius value is ±999.999mm (or ±99.9999 inches).
15. If correction of a step difference smaller than the tool radius is executed in the M96 mode, excessive cut may occur. If performed in the M97 mode, insufficient cut may result (which is preferable to the above, however).

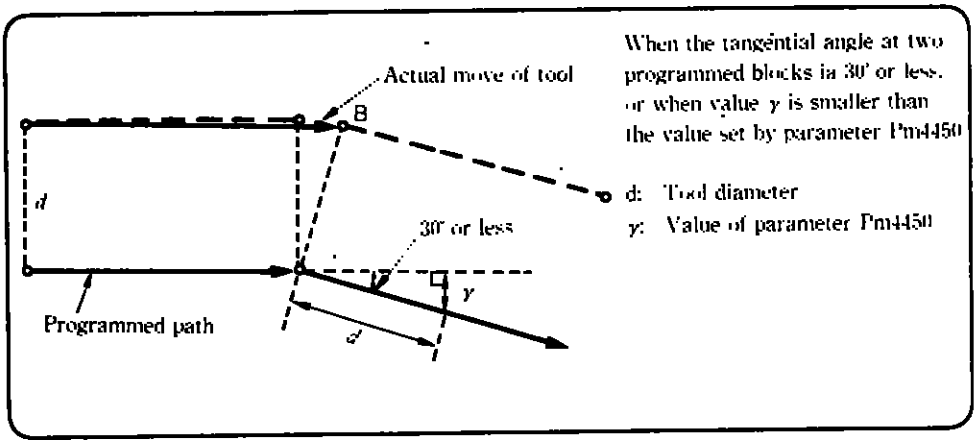


(a) M96 MODE (b) M97 MODE

Fig. 2.12.68 Correction of a Step Difference Smaller than the Tool Radius

16. Even in the M96 mode, if the tangential angle is 30° or less or value $\Delta\gamma$ is smaller than a certain value set by Pm4450, the tool moves directly to point B without performing round-the-corner arc operation.

    Note that since the value set for pm4450 is also used as the minimum distance for the execution of circular interpolation, the setting for this parameter has influence on normal circular interpolation.



When the tangential angle at two programmed blocks is 30° or less, or when value γ is smaller than the value set by parameter Pm4450

d: Tool diameter

γ: Value of parameter Pm4450

Fig. 2.12.69 Tool motion with small programmed tangential angle

17. Type A cancellation of offset

When move command for offset plane is not provided in G40 command block, the program such as examples 1 and 2 may not be executed as commanded.

(Example 1) Where concellation of offset and work coordinate system setting are commanded simultaneously.

```
......
G40;
G0 Z-10.;
G90 G55 J02 X0 Y0;
```

(Example 2) Where cancellation of offset and offset startup are commanded simultaneously.

```
......
G40;
G42 G01 X0. Y0.;
```

⑽ G39 (compensation around circular arc)

By commanding G39 during tool diameter compensation C command (G41/G42), the tool can move in the corner in circular arc.

G 39 is a non-modal G code.

● **G39 commanding method**

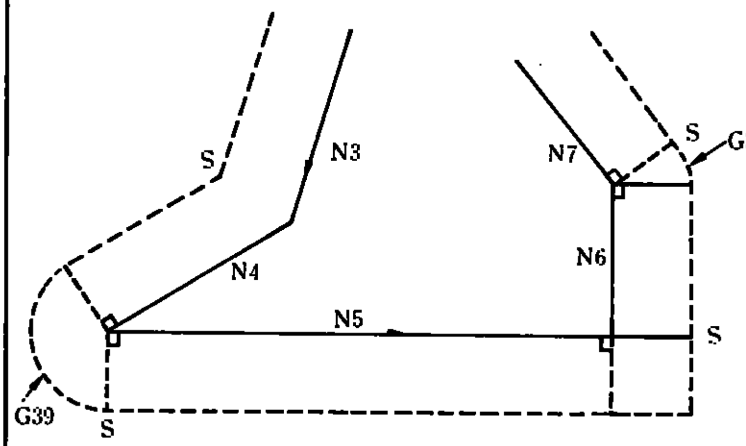
① G39 is commanded alone.

② All codes after G39 are disregarded.

G39 I_ J_ S_ F_ ;

As described above, I, J, S and F codes are disregarded even if they are commanded.

(Example)



```
(G91, G42)
N3    X-10. Y-30. ;
N4    X-30. Y-20. ;
      G39 ;
N5    X80. ;
N6    Y30. ;
      G39 ;
N7    X-20. Y30. ;
        :
```

Fig. 2.12.70 Compensation around Circular Arc

NOTE

1. After completion of G39 block, the mode (M96/M97) before the block is entered.
2. At single-block execution, the cursor does not stop at the G39 block but at the next block.
3. If an error occurs at G39 command in the interruption check, the G39 is disregarded and automatic correction is performed.

(II) Intervention by MDI operation during offset mode.
The offset mode cannot be intervened by MDI operation.

NOTE One line MDI is available. (See NOTE 3 in Par.3.5.2.7)

Example of Programming (A)

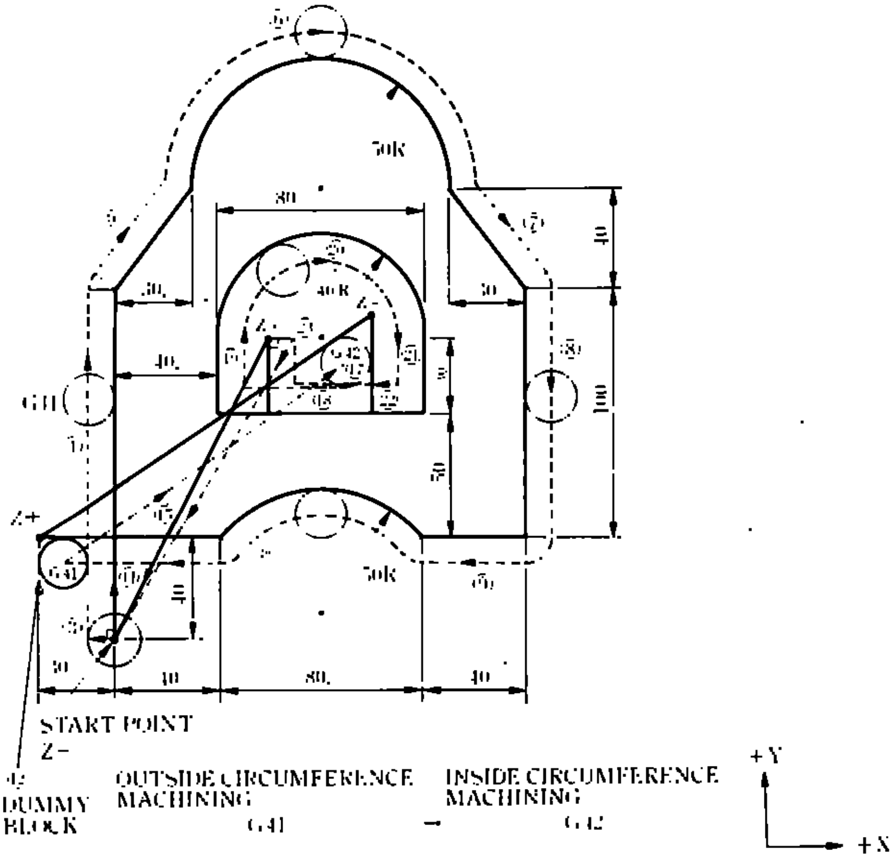


Fig. 2.12.71

①	G91 G01 Z−25. F150:	Incremental, Z-axis lowered
②	G17 (G01) F300:	XY plane specification,feed command
③	G41 Y10. D21:	Radius compensation starts at offset No. 21
④	Y130. :	Offset onto the vector line at the start point of this block
⑤	X30. Y40. :	
⑥	G02 X100. I50. :	
⑦	G01 X30. Y-50. :	Outside circumference machining
⑧	Y-100. :	
⑨	X-40. :	
⑩	G03 X-80. R50. :	R-specified circular arc
⑪	G01 X-70. :	

⑫	J-20. ;	Dummy block (used to correct offset position)	
⑬	Z25. ;	Z-axis raised	two blocks without movement
⑭	M01. ;	Optional stop	
⑮	G42 (G01) X130. Y90. F2000 ;	Offset direction switched	
⑯	Z-25. F150. ;	Z-axis lowered	(from left to right)
⑰	Y-40. F300. ;	Inside circumference machining	
⑱	X-60. ;		
⑲	Y30. ;		
⑳	G02 X80. I 40. ;		
㉑	G01 Y-30. ;		
㉒	X-60. ;		
㉓	Y30. ;	Offset onto the normal line at the end point of this line	
㉔	Z25. ;	Z-axis raised	
㉕	G40 (G01) X-60. Y-120. F2000 ;	Offset cancel command	

NOTE Commands in ( ) are written here for easy understanding.

Example of Programming (B)

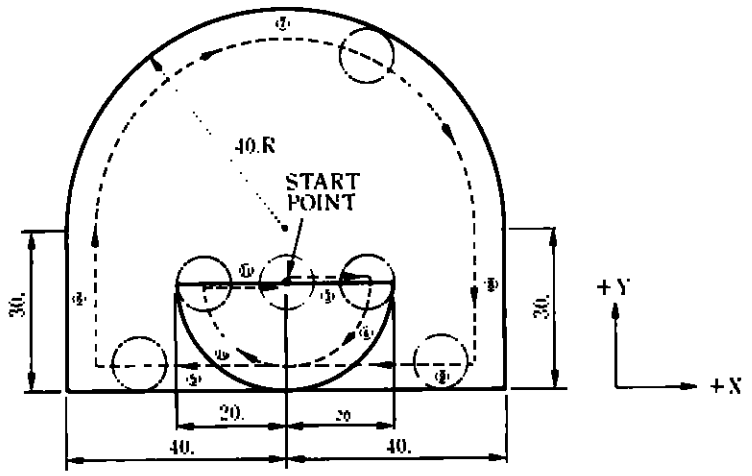


Fig. 2.12.72

```
   (G40)
①  G91  G01   Z-25.  F150;
②  G17  F300;
③       G42   D20.  X20;
④       G42   X-20.  Y-20.  I-20. ;
⑤       G01   X-40.  ;
⑥             Y30. ;
⑦       G02   X80.    Y40. ;
⑧       G01   Y-30. ;
⑨             X-40. ;
⑩       G02   X-20.  Y20.  J20. ;
⑪  G40  G01   X20. ;
⑫             Z25. ;
```

For inside circumference machining in program example (A), the double cutting allowances at the start and end of machining are varied by the tool radius. In the example (B), the double cutting allowances indicate an example in which inside circumference machining is zero irrespective of the tool radius.

2.12.5 Multiactive Registers (M92/M93) ※

(1) The following M codes are used.

Table 2.12.13 Multiactive Register

Code	Meaning
M92	Multiactive register OFF
M93	Multiactive register ON

(2) M93 ;

When M93 is commanded, a seven-block preread mode is entered until M92 is commanded subsequently. In other words, data are preread for up to seven blocks to be prepared for subsequent operation. If the automatic operation time for the seven preread blocks is longer than the preread and calculation time for the next seven blocks, interblock stoppage can be avoided.

(3) M92 ;

This command cancels the seven-block preread mode.

2.12.6 Internal M Codes for Round-the-arc Discrimination (M96/M97)

(1) The following M codes are used.

Table 2.12.14

Code	Meaning
M96	Tool radius compensation round-the arc ON
M97	Tool radius compensation round-the-arc OFF (intersection calculation is executed)

NOTE By parameter, the M96 or M97 can be selected when the power is supplied or when resetting.
(Pm4001 D1 = 0 : M96)
(Pm4001 D1 = 1 : M97)

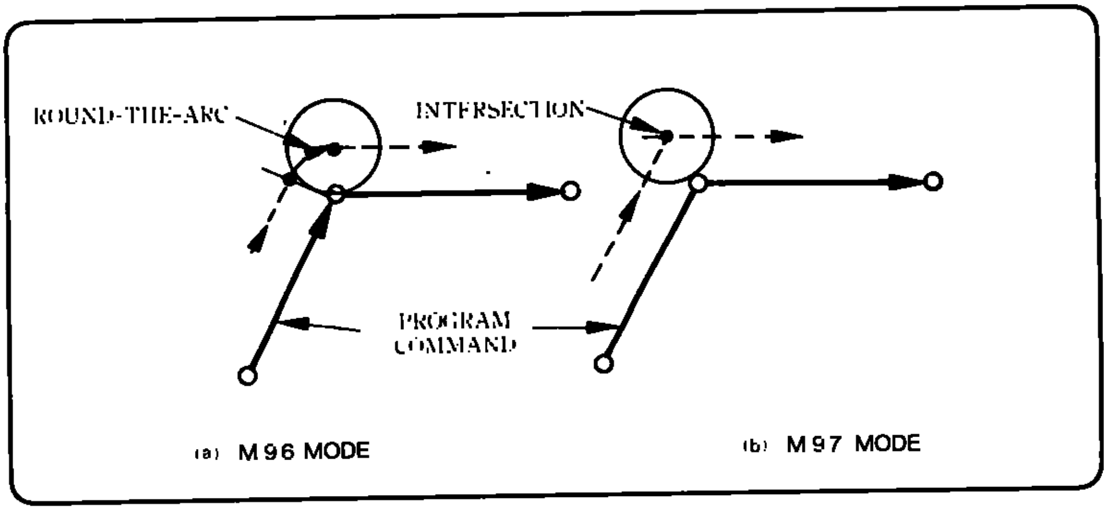


Fig. 2.12.73

(2) For a command shape having a corner with 180° or larger tangential angle in the tool radius compensation mode entered by G41 (or G42), the tool moves around that corner in a circular arc in the M96 mode. In the M97 mode, however, the tool locus is not a circular arc, and an intersection shifted by the tool radius is calculated so that the offset movement passes that intersection.

(3) The move command blocks in which M96 and M97 commands are effective are as shown below.

```
 G01   X······ Y······;
(G01)  X······ Y······ M96;
 G01   X······ Y······ F······;
       M96 ( or  M97)
(G01)  X······ Y······;
```

Effective from corner movement in these two blocks

Effective from corner in these two blocks

2.13 PROGRAM SUPPORT FUNCTIONS

2.13.1 Canned Cycles ※

2.13.1.1 Canned cycles

Canned cycles (G73, G74, G76, G77, G80 to G89, G181, G182, G185, G186, G187, G189) represent a simplified program in which specific movement over several blocks is commanded by a single-block command. There are 19 kinds of canned cycles. G80 is used to cancel these canned cycles.

(1) Table 2.13.1 shows the relationship between canned-cycle G codes and their movements.

(a) Normal hole opening canned cycles

Table 2.13.1 List of Normal Hole Opening Canned Cycles

Code	Feeding	Bottom	Escape	Use
G73	Intermittent feed (each pitch can be dwelled)	–	Rapid feed	High-speed deep hole drilling
G74	Cutting feed	Spindle forward run after dwell	Cutting feed→dwell →spindle reverse run	Reverse tapping
G76	Cutting feed	Spindle dividing →shift	Rapid feed→shift, spindle start	Boring
G77	Spindle dividing→shift →rapid feed→shift→ Spindle start→cutting feed	Dwell	Rapid feed→spindle dividing→shift→ rapid feed→shift→ spindle start	Back boring
G80	–	–	–	Cancel
G81	Cutting feed	–	Rapid feed	Drilling
G82	Cutting feed	Dwell	Rapid feed	Spot facing
G83	Intermittent feed	–	Rapid feed	Deep hole drilling
G84	Cutting feed	Spindle reverse run after dwell	Cutting feed→dwell→ Spindle forward run	Tapping
G85	Cutting feed	–	Cutting feed	Boring
G86	Cutting feed	Spindle stop	Rapid feed→shift, spindle start	Boring
G87	Cutting feed	Spindle stop	Manual return→ spindle start	Boring
G88	Cutting feed	Spindle stop after dwell	Manual return→ spindle start	Boring
G89	Cutting feed	Dwell	Cutting feed	Boring

(b) Two-step hole-opening canned cycles

Table 2.13.2 List of Two-step Hole-Opening Canned Cycles

Code	Feeding	Bottom	Escape	Use
G181	Cutting feed→rapid feed→cutting feed	–	Rapid feed	Two-step hole drilling
G182	Cutting feed→rapid feed→cutting feed	Dwell	Rapid feed	Two-step spot facing
G185	Cutting feed→rapid feed→cutting feed	–	Cutting feed 2→rapid feed→cutting feed 2	Two-step hole boring
G186	Cutting feed→rapid feed→cutting feed	Spindle stop	Rapid feed→spindle start	Two-step hole boring
G187	Cutting feed 1→cutting feed 2→cutting feed 1	Spindle stop after dwell	Cutting feed 2	Two-step hole boring
G189	Cutting feed 1→rapid feed→cutting feed 1	Dwell	Cutting feed 2→rapid feed→cutting feed 2	Two-step hole boring

(2) Method of command

- For normal hole opening canned cycles

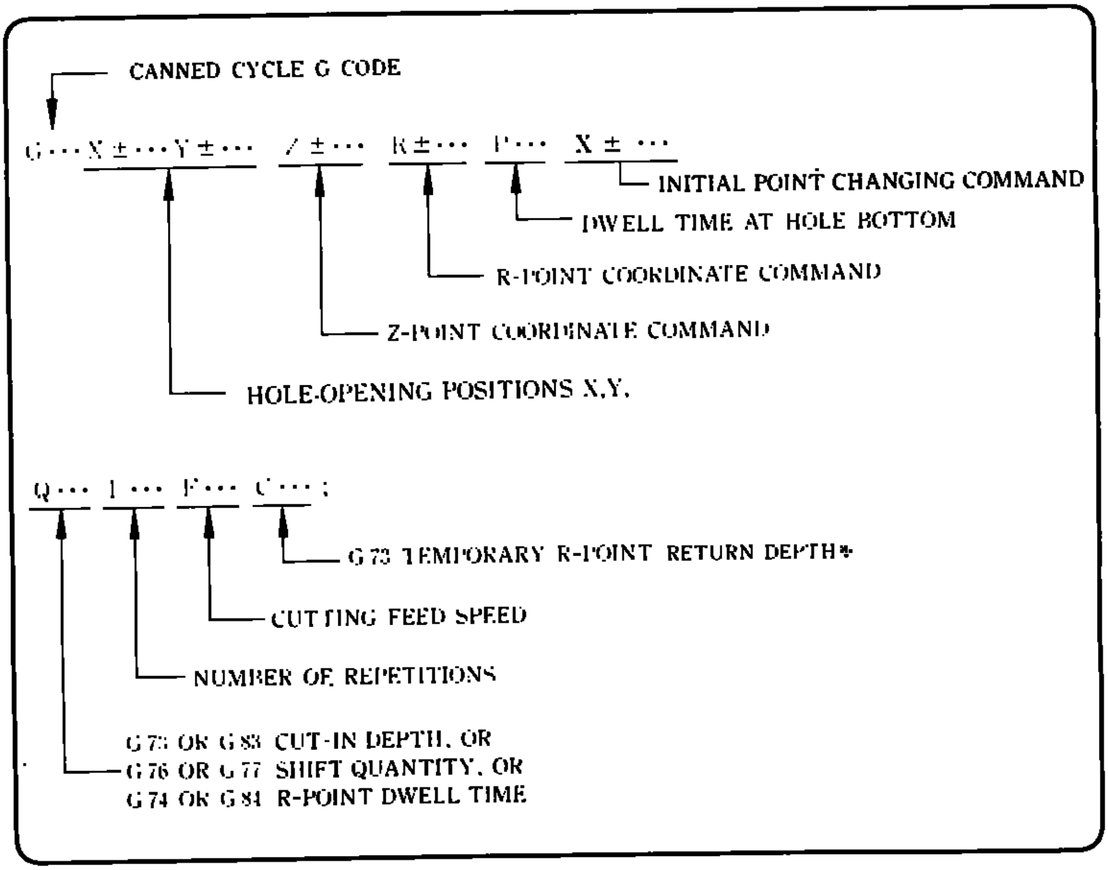


Fig. 2.13.1 Normal Hole-Opening Canned Cycles

Operations ① to ④ are executed as one cycle by the above command.

① Positioning of hole-opening position
② Rapid feed up to R-point
③ Hole drilling (boring) down to Z-point
④ Return to R-point or initial point

(a) Initial point

The initial point is the absolute position of the hole opening axis when the canned cycle mode has been entered from a canned cycle cancel state. The initial point does not change even when a canned cycle is performed in the G99 (R-point level return) mode.

(b) Description of each address

① Positioning axes (X, Y): The hole opening position is commanded with an incremental value or absolute value. The positioning movement can be selected from G00 and G01 by parameter setting. For G01, the speed is commanded by F code.

② Hole opening axis (Z): The position of the hole bottom is commanded with an absolute value or an incremental value from R-point. Movement from R-point to hole bottom is made at the speed commanded by F code in G01. G00 movement may be included depending on the type of canned cycle (e.g., intermittent feed). Return from hole bottom to R-point is made by G00 or G01 movement depending on the type of canned cycle.

③ R: The position of R-point is commanded with an absolute value or an incremental value from the initial point. The operating axis is the hole-opening axis. Return from R-point to the initial point is made by G00 movement.

④ L: The number of repetitions is commanded by address L. If L is not commanded, 1 is assumed. Also, if commanded as L = 0, positioning to (X, Y) only is performed.

- For optional canned cycle *

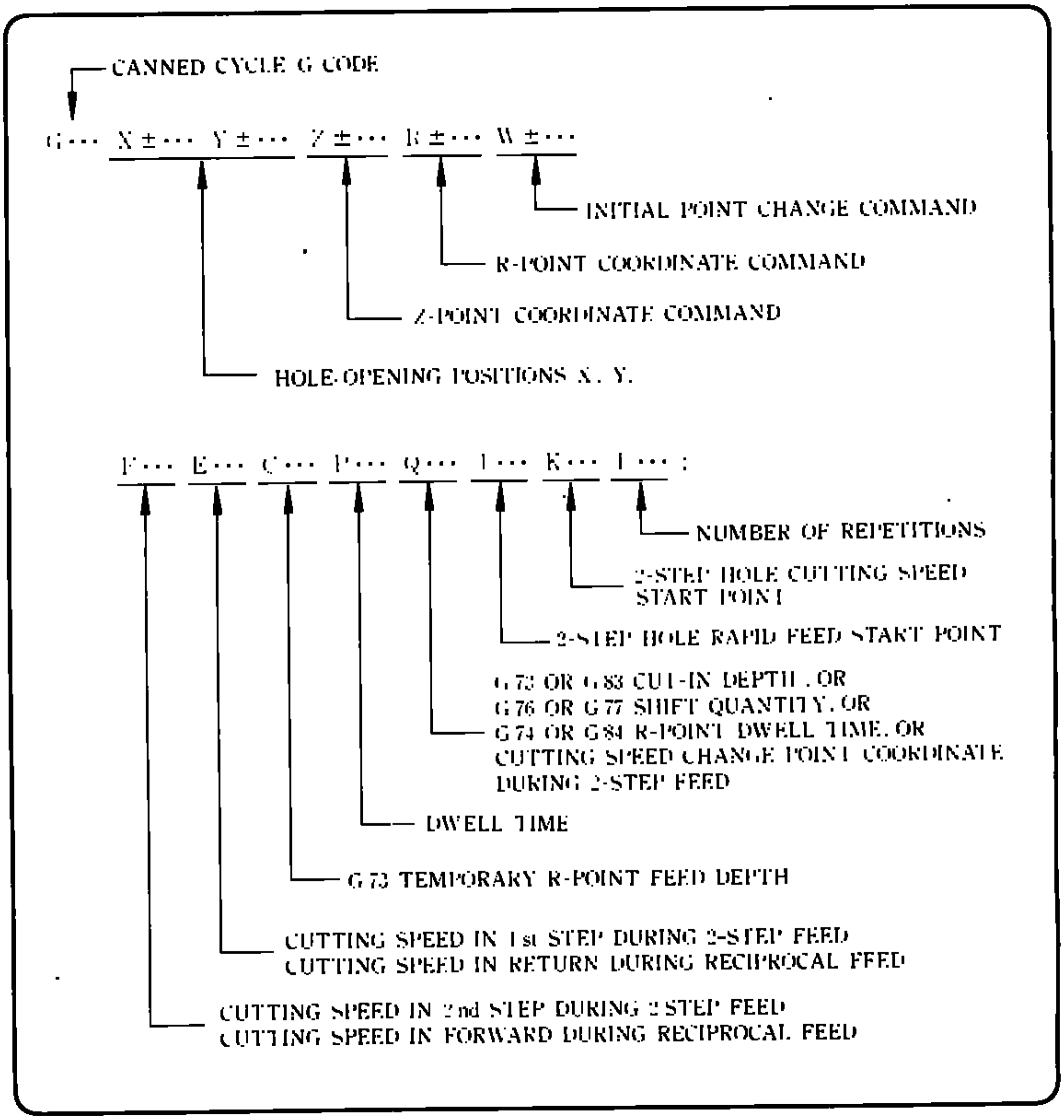


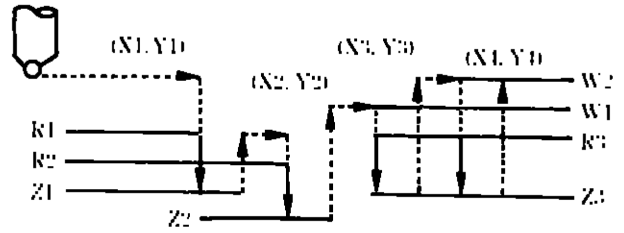
Fig. 2.13.2 Optional Canned Cycle

(a) Initial point change

When the initial point is to be changed in a canned cycle, there is no need to specify it after canceling the canned cycle.

W is used for address. Hereafter, this is called W-point return.

```
N1   G99  G81  X1···  Y1···  R1···  Z1···  F··· ;
N2   G98       X2···  Y2···  R2···  Z2···  W1··· ;
N3             X3···  Y3···  R3···  Z3···  W2··· ;
N4             X4···  Y4··· ;
```



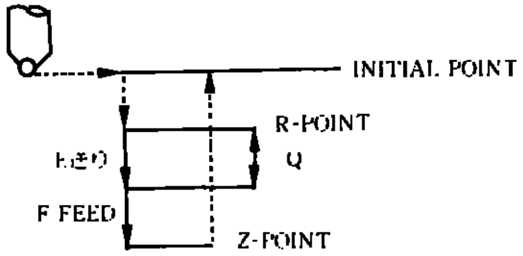
NOTE

1. Point W here is created in reference to the initial point in the preceding block, separate from point W in the G91 mode (incremental command).
2. In the G98 mode (initial level return), a single block stops at point W or initial point, not at point R.
3. If an optional W axis is used, this command has no effect. If address W is commanded, it is used as an axial command.

Fig.2.13.3

(b) Two-step feed*

In drilling operation, if the cutting speed in the vicinity of drill start is slowed down, the accuracy of hole position can be increased without performing center hole machining.



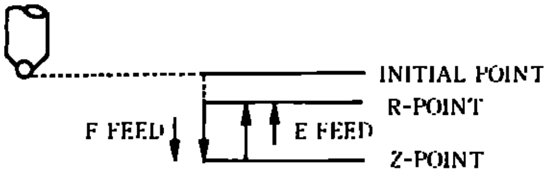
Address E: Speed in 1st step
Address Q: Distance for 1st step speed(commanded without a sign)

Fig.2.13.4

(c) Reciprocal feed*

Cutting speeds in forward and return can be separately commanded such as in tapping or boring operations. In tapping operations, machining can be completed before the tap is fully extended.

Furthermore, the machining time in boring operations can be reduced. In reaming operations, possible damage to the machined surface in the return process can be prevented.



Address E: Return speed

Fig.2.13.5

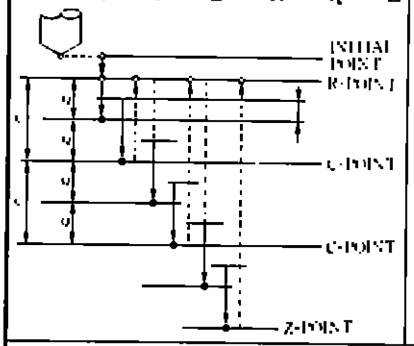
(3) Operation list

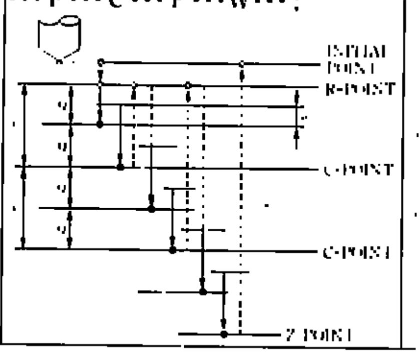
(a) Normal hole-opening canned cycles (including new initial point return)

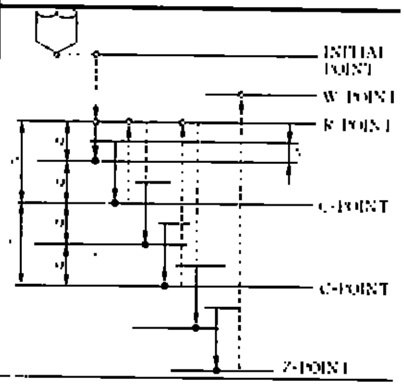
● Dwell
○ Single-block stop

Table 2.13.4

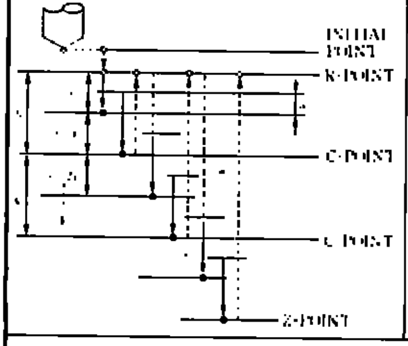
<table>
<tr><th></th><th>G99 (R-Point Return)</th><th>G98 (Initial Point Return)</th><th>G98 (W-Point Return)</th></tr>
<tr><td>G73
(Fixed Pitch)

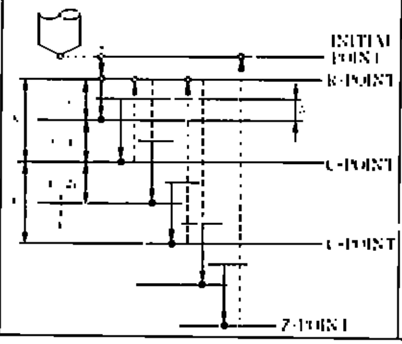
High-Speed Deep Hole Drilling</td><td colspan="3">G73 X･･･Y･･･Z･･･R･･･Q･･･L･･･F･･･C●･･･P･･･W･･･;</td></tr>
<tr><td></td><td>


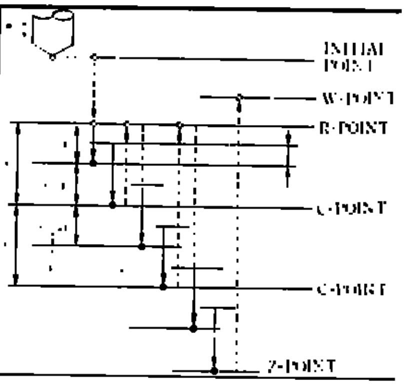
</td><td>


</td><td>


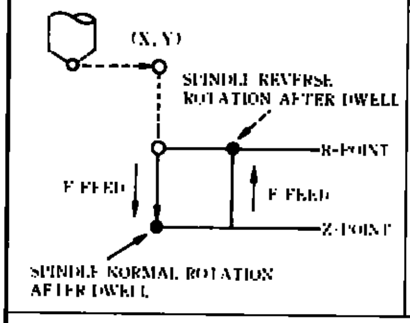
</td></tr>
<tr><td></td><td colspan="3">The dwell time depends on P or setting Pm401
Pm4017 D4=0: Pm401 invalid; 1: Pm401 valid
δ: Setting Pm670
Q: Unsigned incremental value
C: Unsigned incremental value</td></tr>
<tr><td>G73
(Variable Pitch)

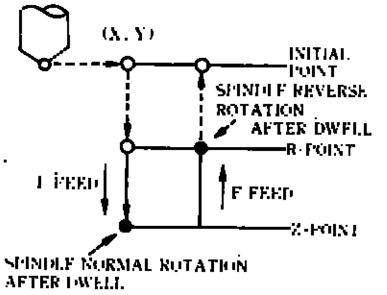
High-Speed Deep Hole Drilling</td><td colspan="3">G73 X･･･Y･･･Z･･･R･･･I･･･J･･･K･･･L･･･F･･･C●･･･P･･･W･･･;</td></tr>
<tr><td></td><td>


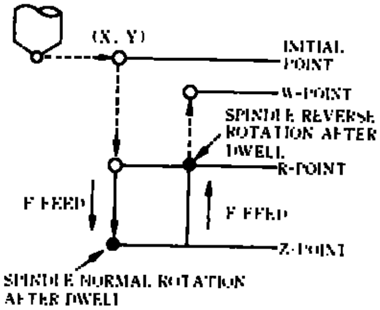
</td><td>


</td><td>


</td></tr>
<tr><td></td><td colspan="3">The dwell time depends on P or setting Pm401
Pm4017 D4=0: Pm401 invalid; 1: Pm401 valid
δ: Setting Pm670
Q: Unsigned incremental value
C: Unsigned incremental value</td></tr>
<tr><td>G74

Reverse Tap</td><td colspan="3">G74 X･･･Y･･･Z･･･R･･･P･･･L･･･F･･･Q●･･･W･･･;</td></tr>
<tr><td></td><td>


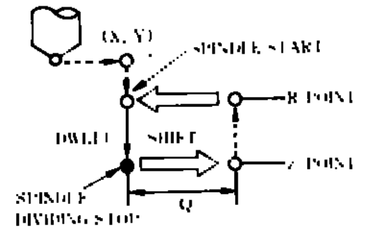
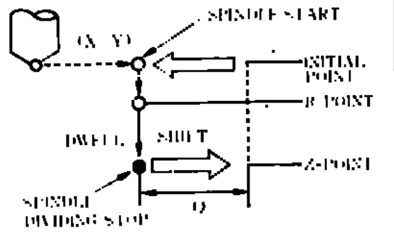
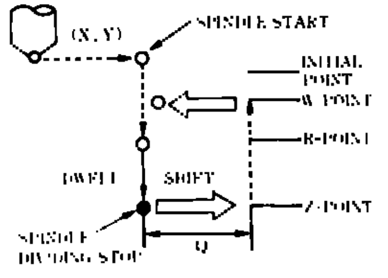
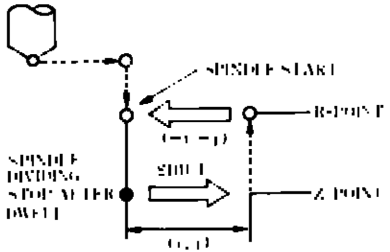
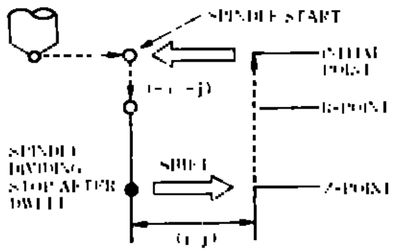
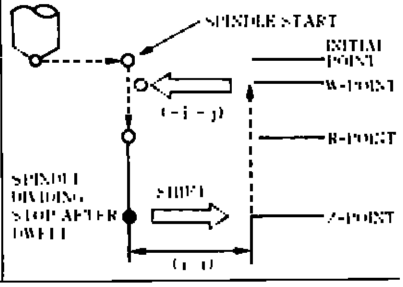
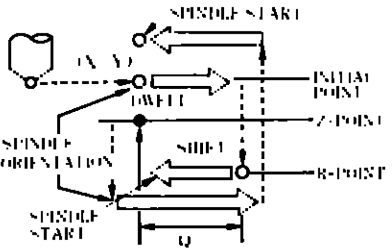
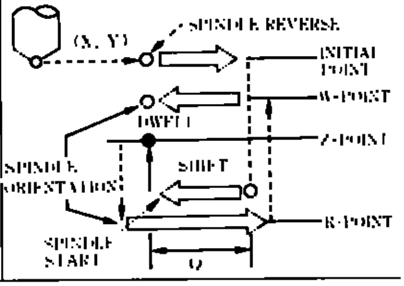
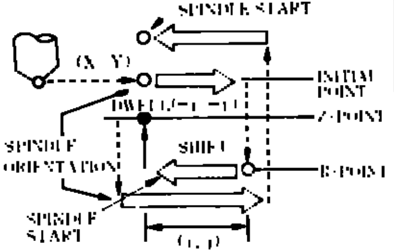
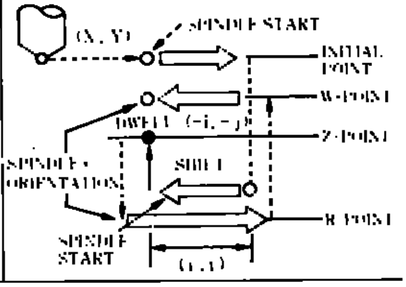
</td><td>


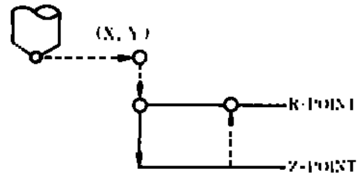
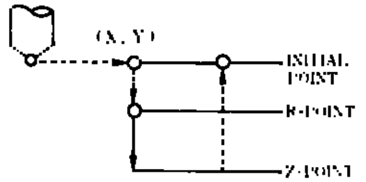
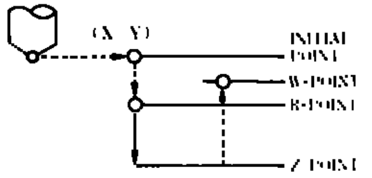
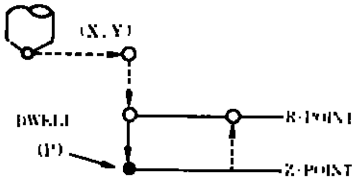
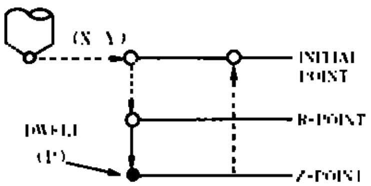
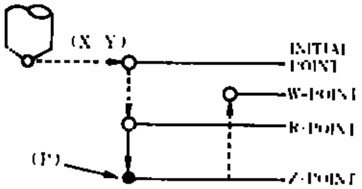
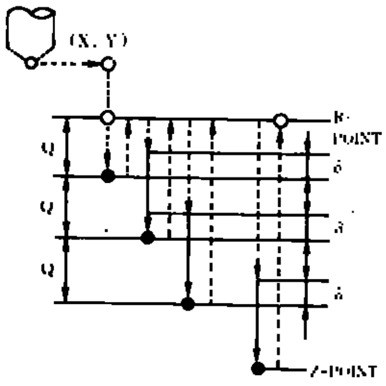
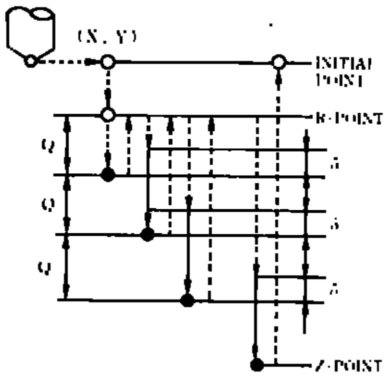
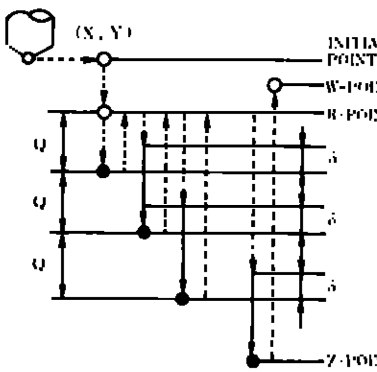
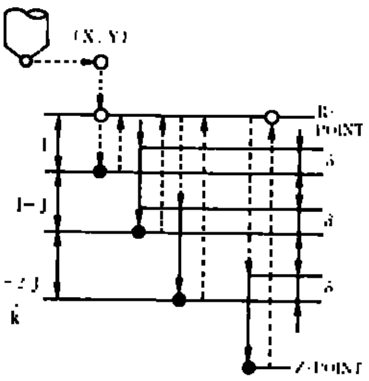
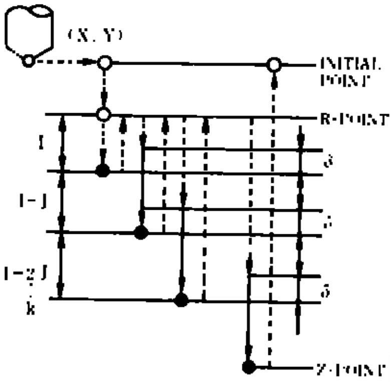
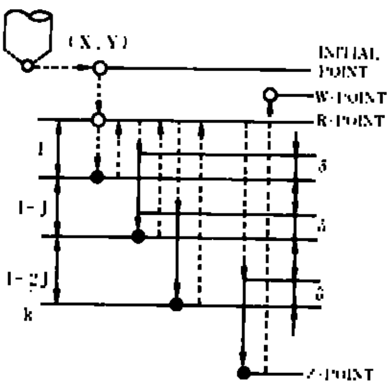
</td><td>


</td></tr>
<tr><td></td><td colspan="3">P: Z-point dwell time
Q: R-point dwell time</td></tr>
</table>

● Dwell
○ Single-block stop

Table 2.13.4 (Cont'd)

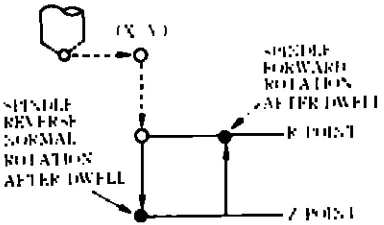
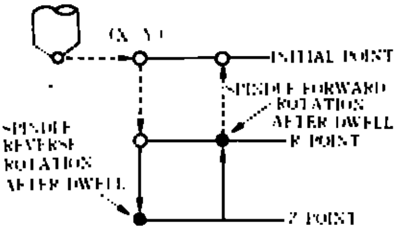
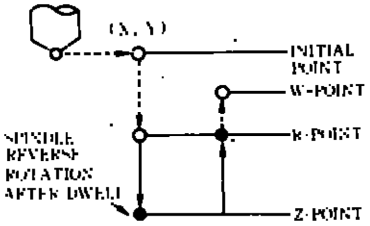
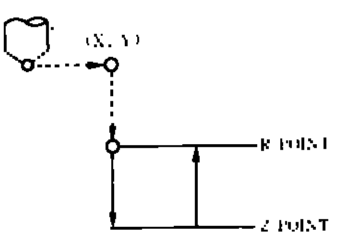
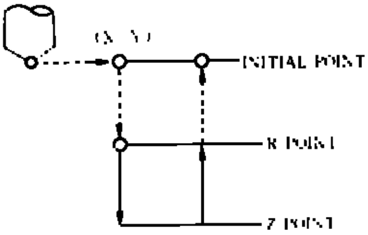
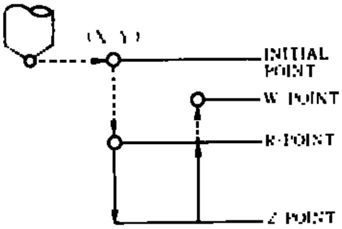
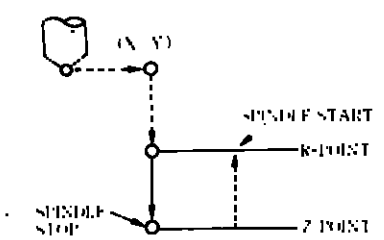
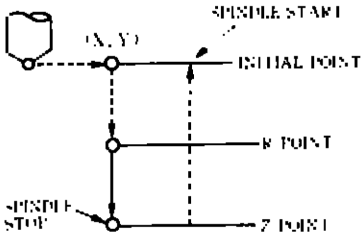
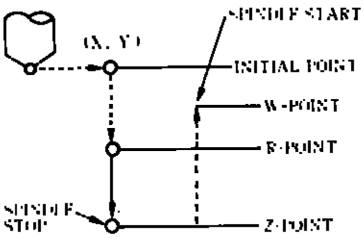
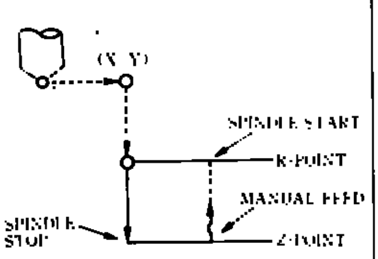
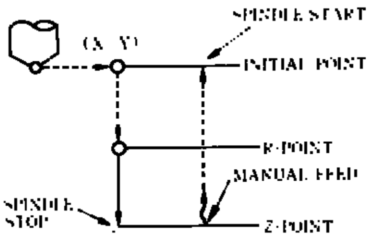
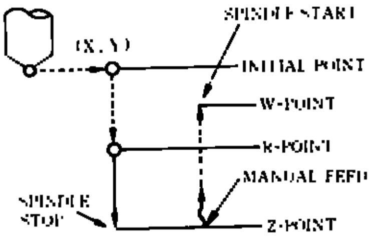
	G99 (R-point Return)	G98 (Initial Point Return)	G98 (W-Point Return)
G76 (Canned Shift) Boring	G76 X···Y···Z···R···Q··· L··· F··· P ··W ··;		
			
	Q: Shift quantity (signed incremental value) Shift speed depends on setting Pm2864 The dwell time depends on P or setting Pm4000 Pm4017 D6=0: Pm4000 invalid, 1: Pm4000 valid		
G76 (Variable Shift) Boring	G76 X···Y···Z···R··· I··· J··· L··· F··· P ··W ··;		
			
	i: X-axis shift quantity (signed incremental value) j: Y-axis shift quantity (signed incremental value) Shift speed depends on setting Pm2864 The dwell time depends on P or setting Pm4000 Pm4017 D6=0: Pm4000 invalid, 1: Pm4000 valid		
G77 (Canned Shift) Back Boring	G77 X···Y···Z···R···Q··· L··· F··· P ··W ··;		
	Not used		
		Q: Shift quantity (signed incremental value) Shift speed depends on setting Pm2864 The dwell time depends on P or setting Pm4000 Pm4017 D6=0: Pm4000 invalid, 1: Pm4000 valid	
G77 (Variable Shift) Back Boring	G77 X···Y···Z···R··· I··· J··· L··· F··· P ··W ··;		
	Not used		
		i: X-axis shift quantity (signed incremental value) j: Y-axis shift quantity (signed incremental value) Shift speed depends on setting Pm2864 The dwell time depends on P or setting Pm4000 Pm4017 D6=0: Pm4000 invalid, 1: Pm4000 valid	

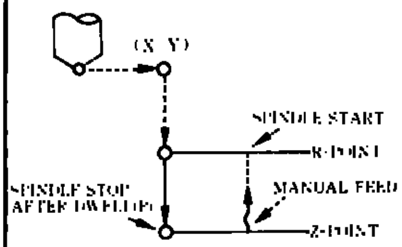
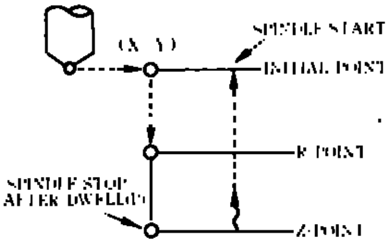
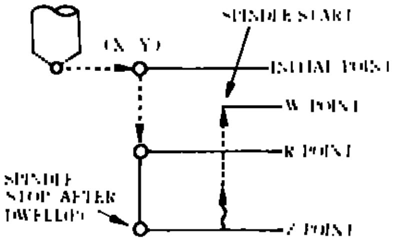
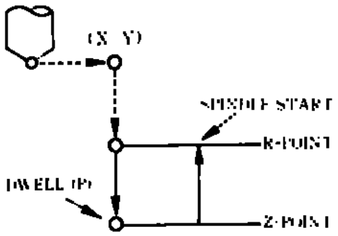
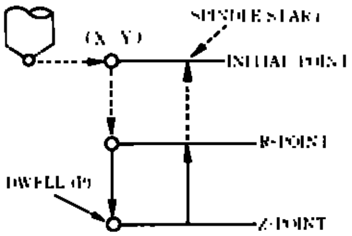
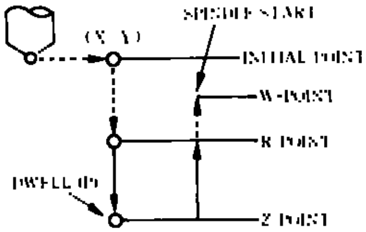
	G99 (R-Point Return)	G98 (Initial Point Return)	G98 (W-Point Return)
G80 Cancel	G80 ;		
G81 Drilling	G81 X···Y···Z···R···L···F···W ··; 		
G82	G82 X···Y···Z···R···P···L···F···W ··; 		
G83 (Canned Pitch) Deep Hole Drilling	G83 X···Y···Z···R···Q···L···F···P ··W ··; 		
	δ: Setting data (Pm87) The dwell time depends on P or setting Pm0l. Pm0l7 D1=0: Pm0l invalid, 1: Pm0l valid		
G83 (Variable Pitch) Deep Hole Drilling	G83 X···Y···Z···R···I··· J···L···F···P ··W ··; 		
	I: Initial value J: Decreased value K: Final value (incremental value) The dwell time depends on P or setting Pm0l Pm0l7 D1=0: Pm0l invalid 1: Pm0l valid		

● Dwell

○ Single-block stop

Table 2.13.4 (Cont'd)

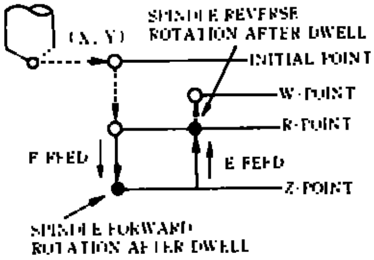
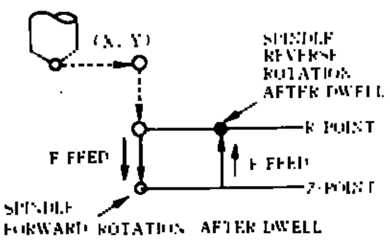
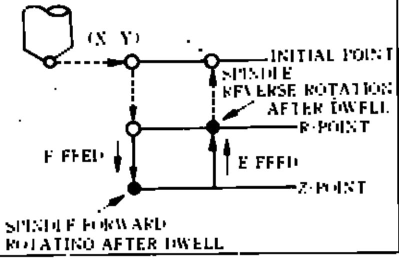
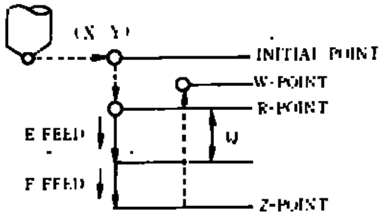
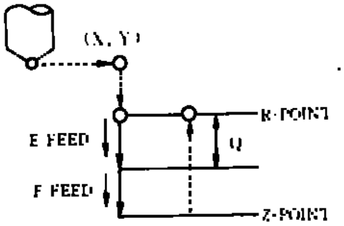
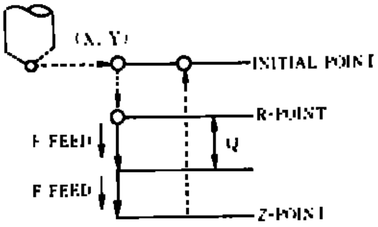
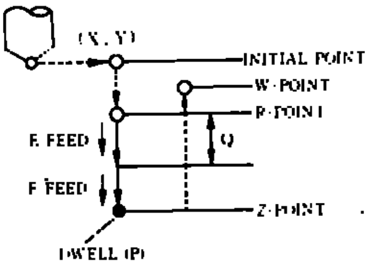
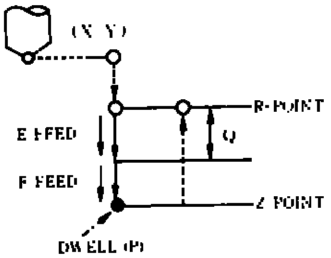
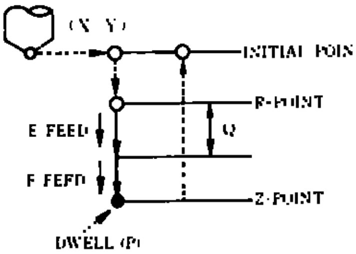
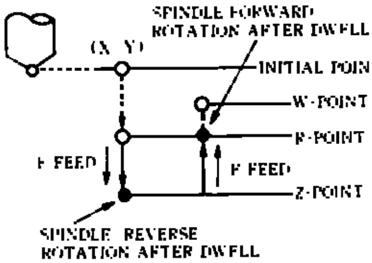
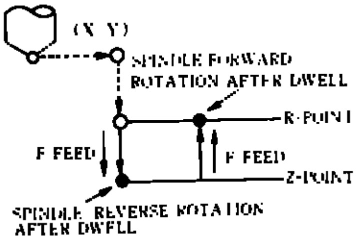
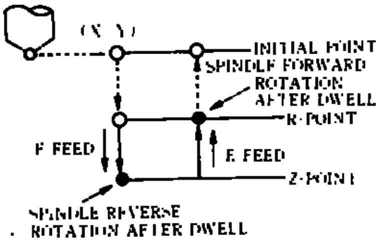
	G99 (R-Point Return)	G98 (Initial Point Return)	G98 (W-Point Return)
G81	G81 X•••Y•••Z•••R•••P••• L	•••F•••Q*••W ••;	
Tapping	 P : Z-point dwell time Q : R-point dwell time		
G85	G85 X•••Y•••Z•••R•••L•••F	•••W ••;	
Boring			
G86	G86 X•••Y•••Z•••R•••L•••F	•••W ••;	
Boring			
G87*	G87 X•••Y•••Z•••R•••Q••• L	•••F•••W ••;	
Boring			

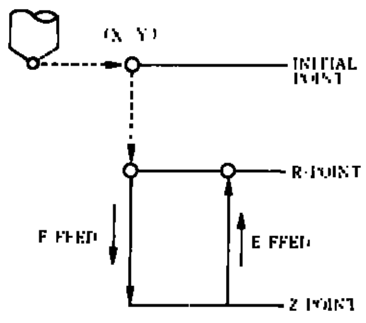
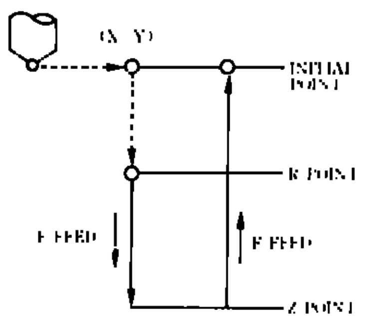
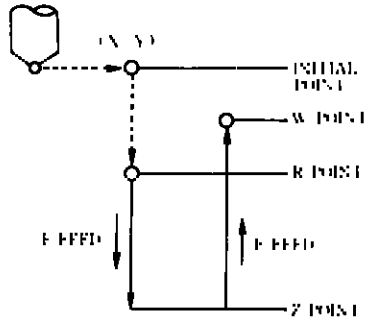
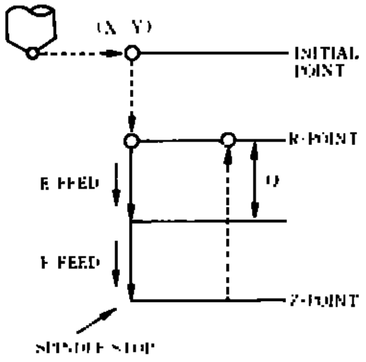
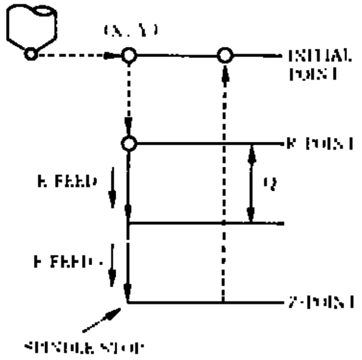
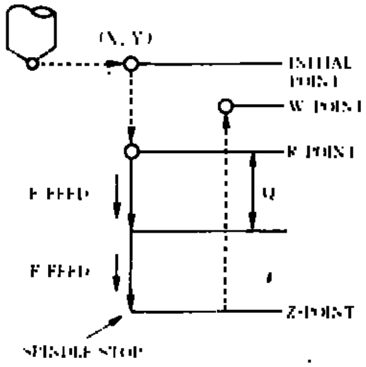
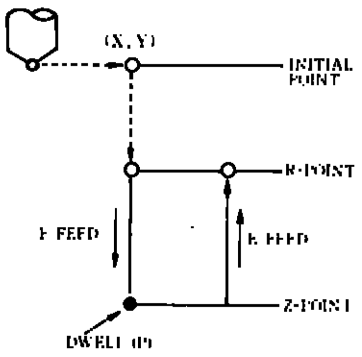
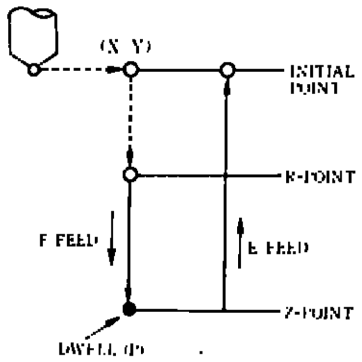
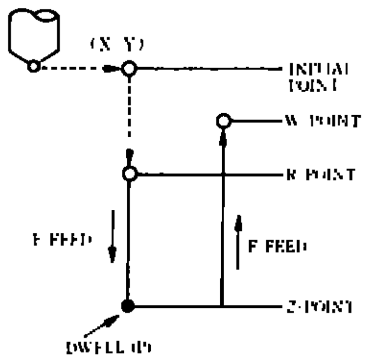
	G99 (R-Point Return)	G98 (Initial Point Return)	G98 (W-Point Return)
G88 Boring	G88 X···Y···Z···R···P···L···F···W··; 		
G89 Boring	G89 X···Y···Z···R···P···L···F···W··; 		

(b) Special 2-step hole-opening canned cycles and 2-step hole opening canned cycles✣ (including new initial point return)

● Dwell
○ Single-block stop

Table 2.13.5

<table>
<tr><th></th><th>G99 (R-Point Return)</th><th>G98 (Initial Point Return)</th><th>G98 (W-Point Return)</th></tr>
<tr><td rowspan="3">G74
Reverse Tapping</td><td colspan="2">G74 X···Y···Z···R···P···L···F···E···Q···W··· ;</td><td rowspan="2"> </td></tr>
<tr><td> </td><td> </td></tr>
<tr><td colspan="3">E: Z point dwell time
Q: R-point dwell time
F: Return speed in reciprocal feed (Z point→R-point)</td></tr>
<tr><td rowspan="3">G81
Drilling</td><td colspan="2">G81 X···Y···Z···R···L···F···E···Q···W··· ;</td><td rowspan="2"> </td></tr>
<tr><td> </td><td> </td></tr>
<tr><td colspan="3">E: 2-step feedrate (1st step: E, 2nd step: F)
Q: Speed change point (unsigned incremental value)</td></tr>
<tr><td rowspan="3">G82
Spot Facing</td><td colspan="2">G82 X···Y···Z···R···P···L···F···E···Q···W··· ;</td><td rowspan="2"> </td></tr>
<tr><td> </td><td> </td></tr>
<tr><td colspan="3">E: 2-step feedrate (1st step: E, 2nd step: F)
Q: Speed change point (unsigned incremental value)</td></tr>
<tr><td rowspan="3">G84
Tapping</td><td colspan="2">G84 X···Y···Z···R···P···L···F···E···Q···W··· ;</td><td rowspan="2"> </td></tr>
<tr><td> </td><td> </td></tr>
<tr><td colspan="3">E: Z-point dwell time.
Q: R-point dwell time.
F: Return speed in reciprocal feed (Z-point→R point)</td></tr>
</table>

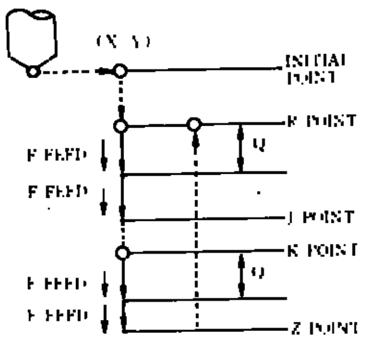
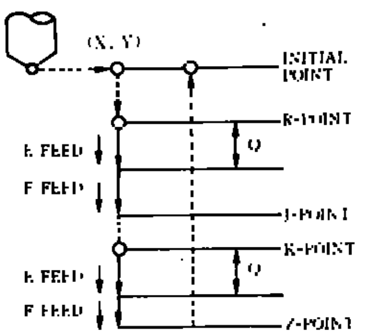
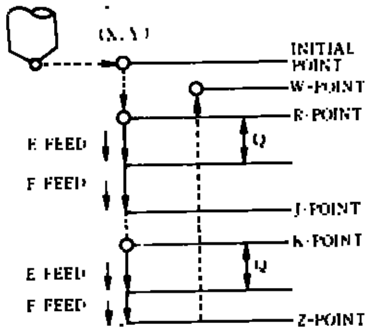
	G99 (R-Point Return)	G98 (Initial Point Return)	G98 (W-Point Return)
G85	G85 X･･･Y･･･Z･･･R･･･L･･･F	･･･E･･･W･･･ ;	
Boring			
	F: Return feed speed in reciprocal feed (Z-point → R-point)		
G86	G86 X･･･Y･･･Z･･･R･･･L･･･F	･･･E･･･Q･･･W･･･ ;	
Boring			
	F: 2 step feedrate Q: Speed change point (unsigned incremental value)		
G89	G89 X･･･Y･･･Z･･･R･･･P･･･L	･･･F･･･E･･･W･･･ ;	
Boring			
	E: Return speed in reciprocal feed (Z-point→R-point)		

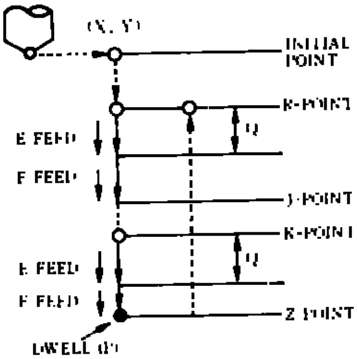
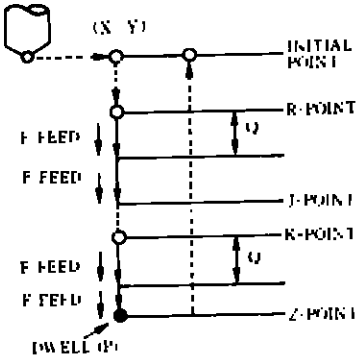
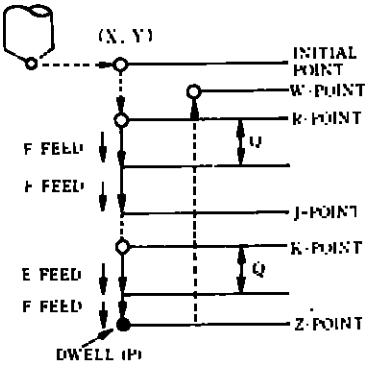
2.13 PROGRAM SUPPORT FUNCTIONS (Cont'd)

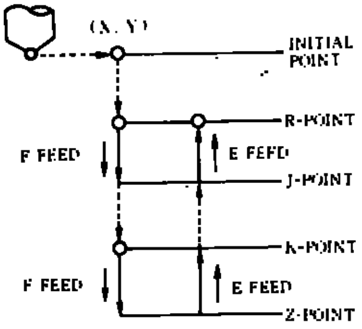
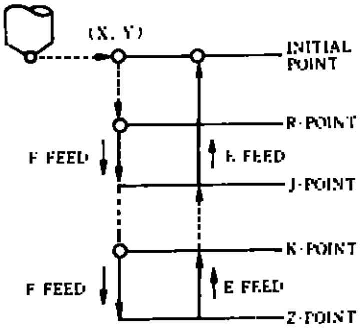
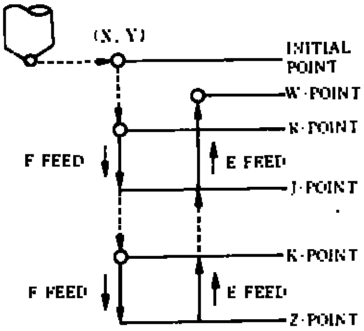
● Dwell
○ Single-block stop

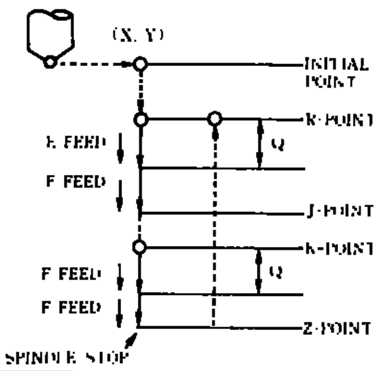
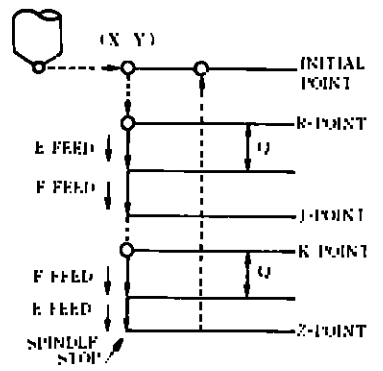
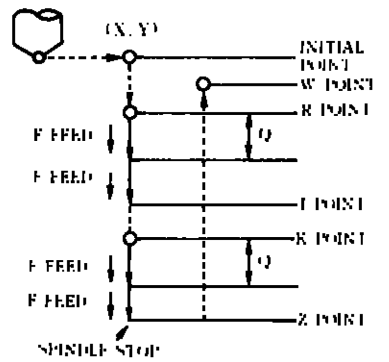
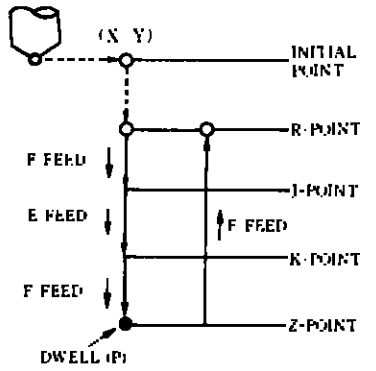
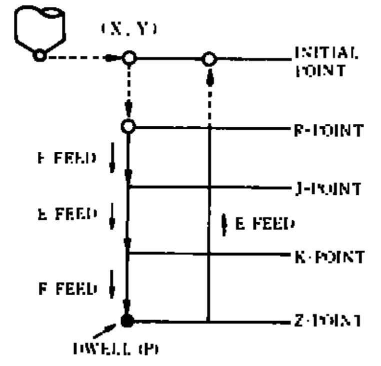
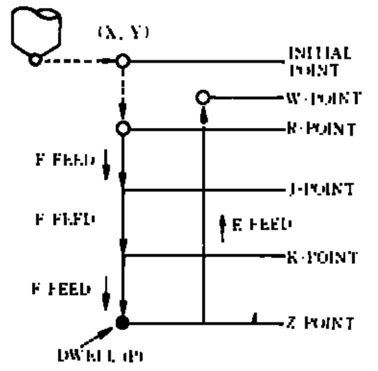
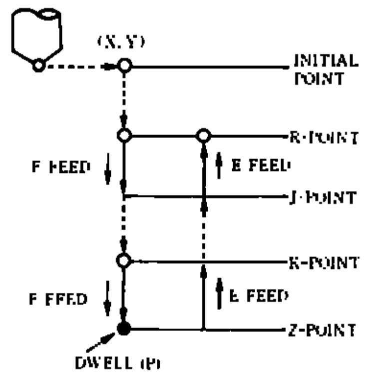
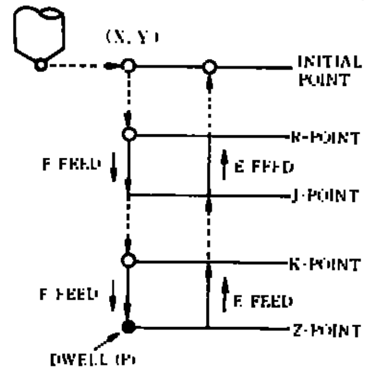
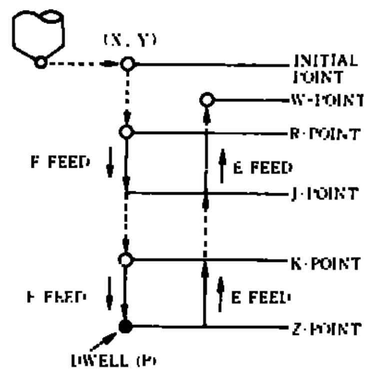
Tadle 2.13.5 (Cont'd)

<table>
<tr><th></th><th>G99 (R-Point Return)</th><th>G98 (Initial Point Return)</th><th>G98 (W-Point Return)</th></tr>
<tr><td rowspan="3">G181

2-Step
Hole
Drilling</td><td colspan="3">G181 X･･･Y･･･Z･･･R･･･L･･･F･･･J･･･K･･･E･･･Q･･･W･･･ ;</td></tr>
<tr><td> </td><td> </td><td> </td></tr>
<tr><td colspan="3">J: 2-step hole rapid feedrate start point
(signed absolute value/incremental value)
K: 2-step hole cutting feedrate start point
(signed absolute value/incremental value)
F: 2-step feedrate (1st step: E, 2nd step: F)
Q: Speed change point (unsigned incremental value)</td></tr>
<tr><td rowspan="3">G182

2-Step
Spot
Facing</td><td colspan="3">G182 X･･･Y･･･Z･･･R･･･P･･･L･･･F･･･J･･･K･･･E･･･Q･･･W･･･ ;</td></tr>
<tr><td> </td><td> </td><td> </td></tr>
<tr><td colspan="3">J: 2-step hole rapid feedrate start point
(signed absolute value/incremental value)
K: 2-step hole cutting feedrate start point
(signed absolute value/incremental value)
E: 2-step feedrate (1st step: E, 2nd step: F)
Q: Speed change point (unsigned incremental value)</td></tr>
<tr><td rowspan="3">G185

2-Step
Boring</td><td colspan="3">G185 X･･･Y･･･Z･･･R･･･L･･･F･･･J･･･K･･･E･･･W･･･ ;</td></tr>
<tr><td> </td><td> </td><td> </td></tr>
<tr><td colspan="3">J: 2-step hole rapid feedrate start point
(signed absolute value/incremental value)
K: 2-step hole cutting feedrate start point
(signed absolute value/incremental value)
F: Return feedrate in reciprocal feed
(Z-point→K-point, J-point→R-point)</td></tr>
</table>

	G99 (R-Point Return)	G98 (Initial Point Return)	G98 (W-point Return)
G186	G186 X···Y···Z···R···L···	F···J···K···E···Q···W··· ;	
2-Step Hole Boring			
	J: 2-step hole rapid feedrate start point (signed absolute value/incremental value) K: 2-step hole cutting feedrate start point (signed absolute value/incremental value) E: 2-step feedrate (1st step: E, 2nd step: F) Q: Speed change point (unsigned incremental value)		
G187	G187 X···Y···Z···R···L···	F···P···J···K···E···W··· ;	
2-Step Hole Boring			
	J: 2-step hole rapid feedrate start point (signed absolute value/incremental value) K: 2-step hole cutting feedrate start point (signed absolute value/incremental value) E: 2-step feedrate (1st step: E, 2nd step: F) Q: Speed change point (unsigned incremental value)		
G189	G189 X···Y···Z···R···P···	L···F···J···K···E···W··· ;	
2-Step Hole Boring			
	J: 2-step hole rapid feedrate start point (signed absolute value/incremental value) K: 2-step hole cutting feedrate start point (signed absolute value/incremental value) F: Return feedrate in reciprocal feed (Z point→K-point, J point→R-point)		

(c) Programing Example in G90/91 mode

Example of Programming

Ⓐ G98 G90 G81 X··· Y··· Z-70. R-40. F··· ;

→ INITIAL POINT RETURN. ABSOLUTE

Ⓑ G99 G91 G81 X··· Y··· Z-70. R-40. F··· ;

→ R-POINT RETURN. INCREMENTAL

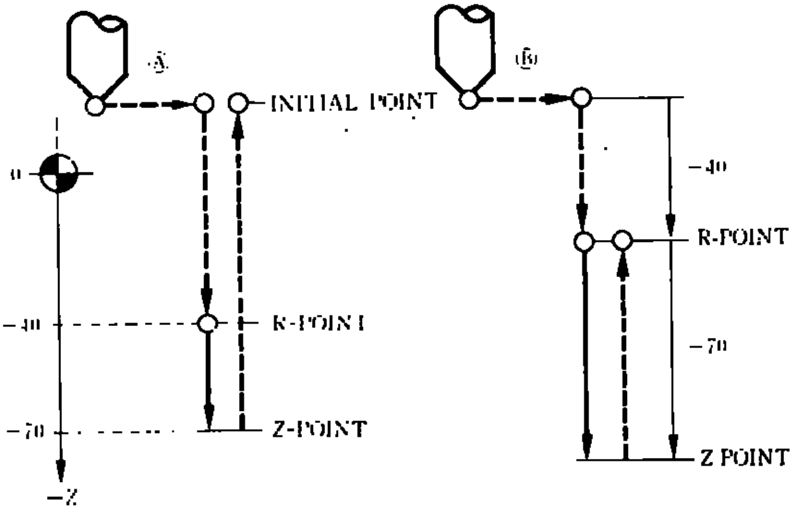


Fig.2.13.6

```
N1  G92  X0   Y0   Z0 ;
N2  G98  G90  G81  X10.  Y10.  Z-50.  R-20.  F100 ; ······①
N3  G91  X20.  R-30. ;  ······②
N4        Z-50. ;       ······③
N5  G99  Z-40.  R-50. ; ······④
N6  G80  G00  Z50. ;    ······⑤
```

Table 2.13.6 Absolute Position

Absolute position		Remarks
Z =	R =	
① -50.	-20.	Canned cycle start
② -50.	-30.	Change or X, R : Z-point at the position of the previous block
③ -80.	-30.	Change of Z : R-point at the position of the previous block
④ -90.	-50.	Change of R, Z
⑤ Tool moves to Z=0 (absolute position).		Canned cycle cancel

Only the newly commanded addresses are changed including switching G90 → G91 as in N2 → N3 in the above example. For uncommanded addresses, the positions commanded by the previous block are retained. Changes of R and Z in this program example are summarized in Table 2.13.6.

(d) R-point, Z-point switching in the G91 mode

If R is commanded without commanding Z when a canned cycle is performed in the G91 mode, Z-point is recalculated to obtain an increment quantity from new R-point.

```
  G92  X0  Y0  Z0
G98 G91 G81 X······ Y······ R-5.0 F······;

        X -R -7.0;
        X -Z -3.0;
        R -4.0 Z -11.0;
```

Table 2.13.7 Increment Quantity at R and Z Points

R Point	Z Point
−5.0	−15.0
−7.0	−17.0 ①
② −7.0	−10.0
−4.0	−15.0

① Because the Z-point position is commanded to be − 10.0 in the G91 mode in advance.

② Because the R-point position is commanded to be − 7.0 in the G91 mode in advance.

(4) Method of commanding G73 and G83 ※

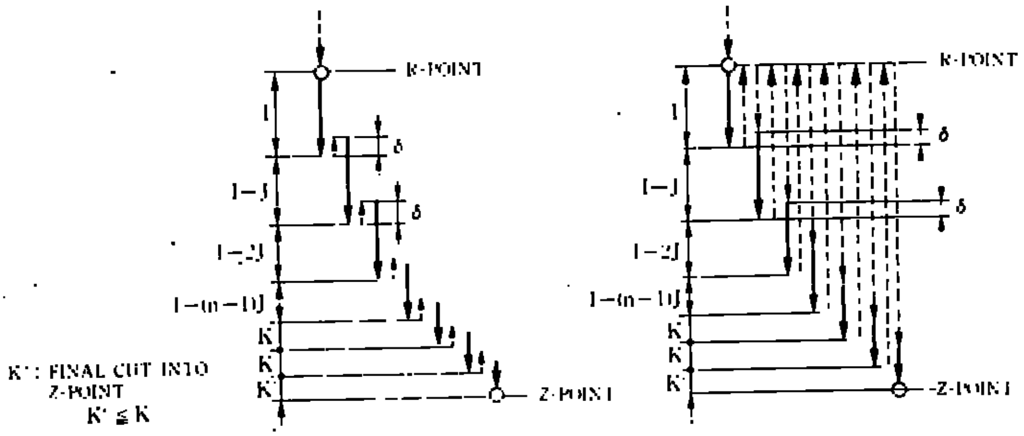
(a) Variable pitch command

In the G73 and G83 deep hole drilling cycles, variable pitch cut can be commanded by using address, I, J, or K in place of address Q that is used to command a certain cut-in quantity.

I : Initial value of cut-in quantity
J : Decreased value for 2nd and subsequent cuts
K : Final value of cut-in quantity

} To be commanded without a sign

NOTE Addresses P, Q, I, J, and K are modal in a canned cycle, so that once commanded, they remain valid until the canned cycle is canceled.



(a) FOR G73 (b) FOR G83

NOTE The value of δ is given by parameter setting (Pm870 for G73, Pm871 for G83).

Fig. 2.13.7 Command of G73 or G83

NOTE

❶ Q, I, J, and K are modal in a canned cycle. They are commanded without a sign.

❷ A variable pitch can be commanded by using Q in place of address I. If Q, I, J, and K are simultaneously commanded, variable pitch cut is performed with Q taken as the initial value of cut-in quantity.
Because Q is modal, if there is a Q command before variable pitch is commanded with I, J, or K, Q 0 must be commanded.

❸ Alarm "174" occurs unless Q and I, J, or K are commanded.

❹ If J=0 or no J command is specified in a G73/G83 variable pitch command, uniform pitch drilling will be performed.

Example of Programming

```
G91 G73 X··· Y··· R-30. Z-55. I10. J1. K4. F··· ;
```

CUT-IN QUANTITY

1st	10 mm	← I10.
2nd	9 mm	
3rd	8 mm	
4th	7 mm	
5th	6 mm	
6th	5 mm	
7th	4 mm	← K4.
8th	4 mm	
9th	2 mm	← K'
TOTAL	55.00mm	← -Z -55.

(b) Each pitch dwell command

In deep hole drilling G73 and G83, drill load can be reduced by inserting a small dwell at the hole bottom of each pitch. The dwell time is commanded by setting Pm401 or address P.

NOTE

- ❶ If each pitch dwell is commanded by the program, setting Pm401 is overridden. If not commanded by the program, dwell is performed according to the value of setting Pm401 (1=1ms). P0 is also matched to Pm401.
- ❷ The default value of hole bottom dwell can be set with #401.
- ❸ Pm401 can be nullified by parameter Pm4017 D4. If there is no P command at this time, pitch dwell is not performed.

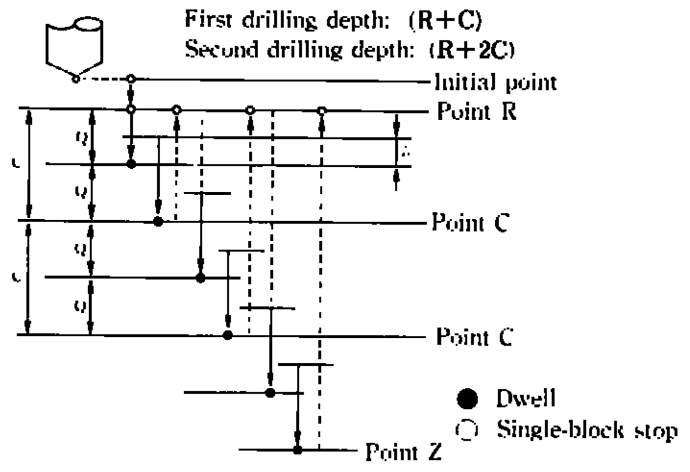
Pm4017 D4=0 : Pm401 invalid
Pm4017 D4=1 : Pm401 valid

(c) Lift in the middle command (C-point command)

It is possible to lift the tool to R-point in the middle of high-speed deep hole drilling G73; in addition, the tool can be single-block stopped. As a result, it is possible to remove chips during the cycle. This is commanded with address C.

The C-point is specified by an unsigned incremental value. The cutting tool retracts to the R-point after each drilling of distance C.

NOTE For C-axis set as the additional axis, this command is invalid. This is axis command with address C.



NOTE Both C and Q are unsigned incremental values. Specification of negative value is meaningless.

Fig.2.13.8 Lift in the Middle Command (C-point Command)

(5) Method of commanding boring (G76) and back boring (G77)

(a) The direction of shift for G76 and G77 can not only be commanded by [Q command + angle (setting Pm805)], but can also be commanded by the program.

(b) The dwell time before shift can also be commanded by setting Pm400 or by program.

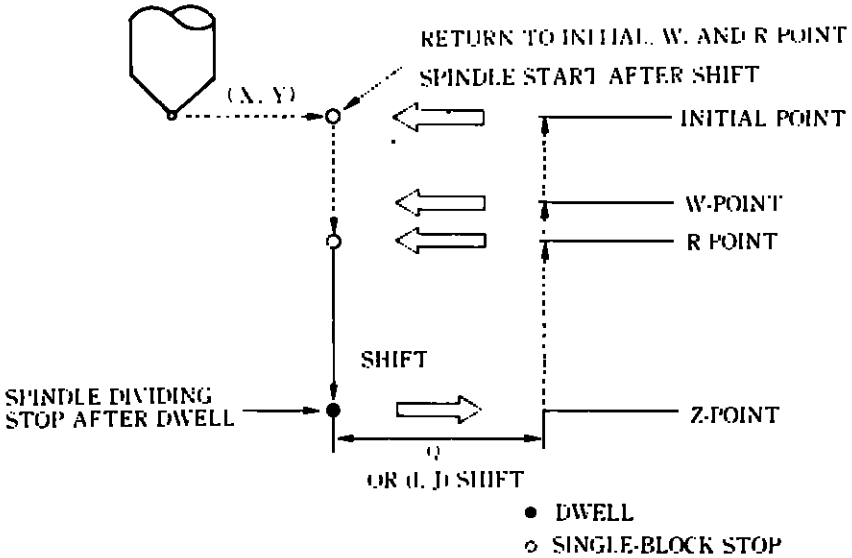
G76 X······Y······Z······R······ I ······ J ······ P······ ;

└Dwell time

└Specification of shift direction

- Boring (G76) command

```
G76 X_ Y_ Z_ R_ Q_ L_ F_ P_ W_ ;
                                (CANNED SHIFT)
G76 X_ Y_ Z_ R_ I_ J_ L_ F_ P_ W_ ;
                              (VARIABLE SHIFT)
```



NOTE

❶ If address Q and address I or J are simultaneously commanded, address I or J is disregarded.

❷ For the hole bottom dwell, setting Pm 400 is overridden by program command.

❸ Setting Pm 400 is only valid when bit D3 of Pm 4017 is ON.

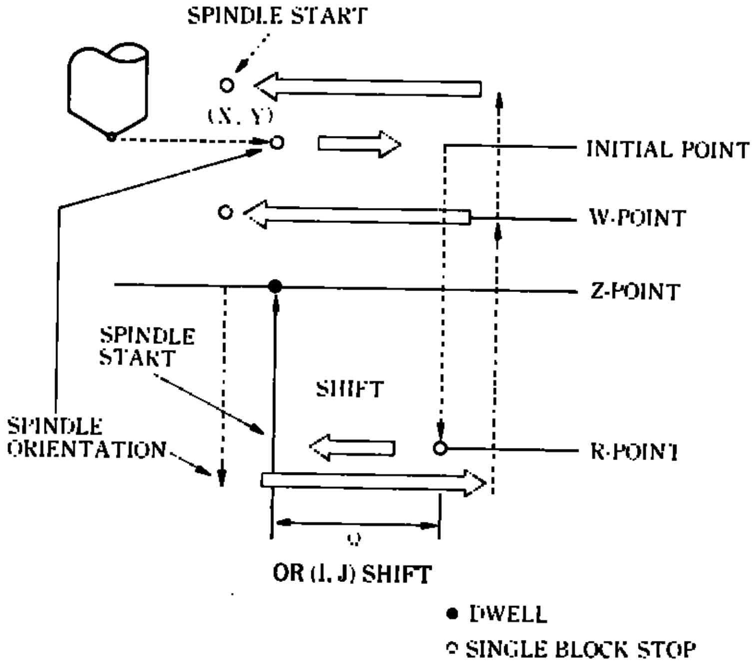
❹ The shift speed is set by Pm 2864. However, if Pm 2864=0, F command is followed.

❺ Because address Q is modal, if there is Q command before I or J command, I or J is disregarded. I or J can be made valid by commanding it after canceling the canned cycle with G80.

Fig.2.13.9 Boring (G76) Command

- Back boring (G77) command

```
G77 X_ Y_ Z_ R_ Q_ L_ F_ P_* W_*;
                              (CANNED SHIFT)
G77 X_ Y_ Z_ R_ I_ J_ L_ F_ P_* W_*;
                              (VARIABLE SHIFT)
```



NOTE

1. If address Q and address I or J are simultaneously commanded, address I or J is disregarded.
2. For the hole bottom dwell, setting Pm 400 is overridden by program command.
3. Setting Pm 400 is only valid when bit D3 of Pm4017=1.
4. The shift speed is set by Pm2864. However, if Pm2864=0, F command is followed.
5. Because address Q is modal, if there is Q command before I or J command, I or J is disregarded. I or J can be made valid by commanding it after canceling the canned cycle with G80.

Fig.2.13.10 Back Boring (G77) Command

(6) Method of commanding reverse tap (G74) and tap (G84) ✻

(a) In a tap cycle, dwell can be commanded at R-point return and initial point return independent of hole bottom dwell. Tap elongation or contraction can be offset by this command.

(b) Although G74 and G84 are defined as reverse tap and tap, there is no difference. That is to say, if entered with M03, M04 is output at the hole bottom; if entered with M04, M03 is output at the hole bottom. If G84 is commanded without rotating the spindle, M04 is output at the bottom of the hole. If G74 is commanded, M03 is output.

- **Reverse tap (G74) command**

```
G74 X_ Y_ Z_ R_ P_ L_ F_ E_ Q_ W_ ;
```

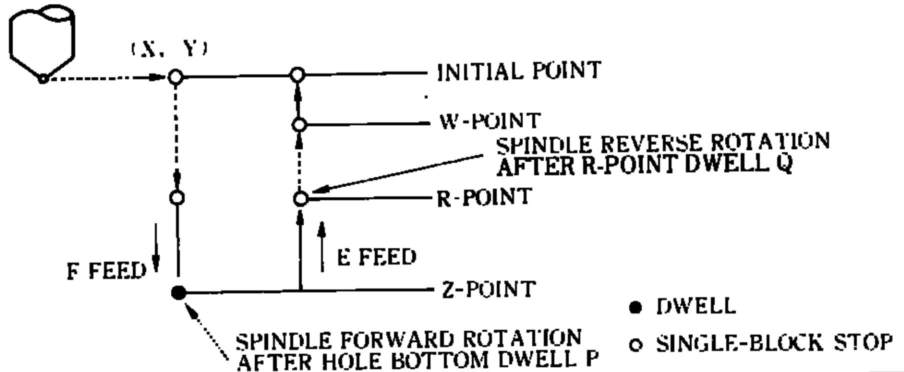


Fig.2.13.11 Reverse Tap (G 74) Command.

- **Tap (G84) command**

```
G84 X_ Y_ Z_ R_ P_ L_ F_ E_ Q_ W_ ;
```

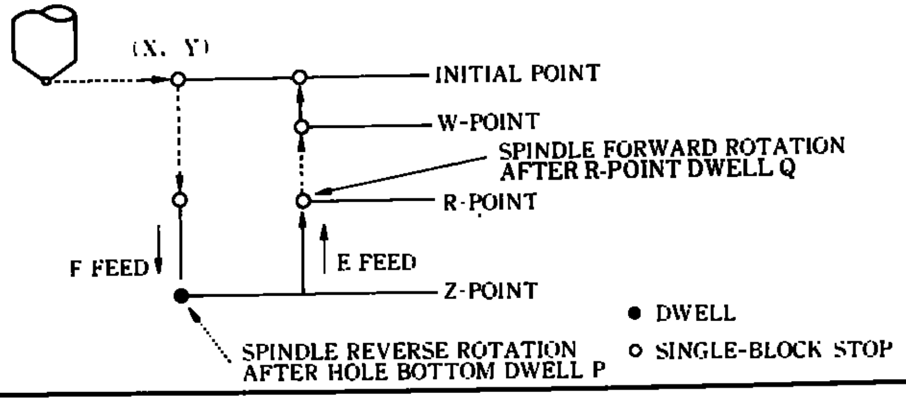


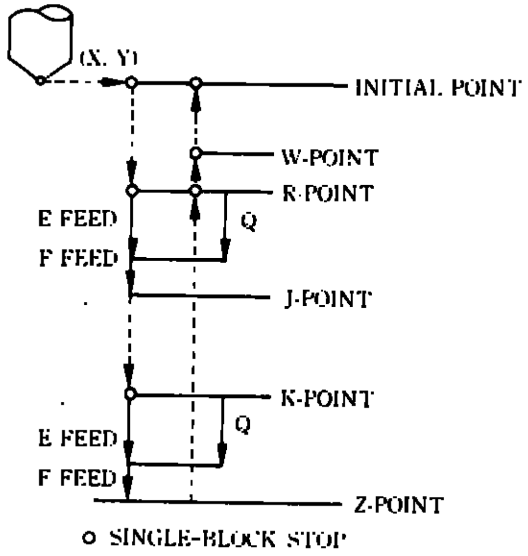
Fig.2.13.12 Tap (G 84) Command

(7) Method of commanding 2-step hole canned cycle

Two-step cycles are provided to enable opening holes in hollow workpieces such as castings. These cycles correspond to G80 to G 89 canned cycles. For two-step hole commands, 100 is added to each G code to produce G181, G182, G185, G186, G187, and G189, commands.

- 2-step hole drilling (G181) command

```
G181 X_ Y_ Z_ R_ L_ F_ J_ K_ E_ Q_ W_ ;
```



J : 2-STEP HOLE RAPID FEEDRATE START POINT (absolute/incremental)
K : 2-STEP HOLE CUTTING FEED START POINT (absolute/incremental)
E : 2-STEP FEEDRATE (1st step : E, 2nd step : F)
Q : SPEED CHANGE POINT (incremental)
W : NEW INITIAL POINT (absolute/incremental)

NOTE To use J or K in the G91 (incremental) mode, use an incremental value measured from point R.

Fig.2.13.13 2-step Hole Drill (G181) Command

NOTE

❶ When a canned cycle is executed by turning the single-block switch on, the tool temporarily stops at the following halfway points, and the feed hold lamp lights.

(a) After positioning to (X, Y) point
(b) After positioning to R-point
(c) After termination of each cycle if L command given

The single-block stop at completion of a canned cycle works as usual, and the feed hold lamp does not light.

❷ Before entering the canned cycle mode, be sure to command R and Z to newly define R-point and Z-point. R-point and Z-point are erased when the canned cycle is canceled.

❸ If a canned cycle is executed after changing address data, any of the following address commands must be given in that block; otherwise, the canned cycle will not be executed.

X, Y, Z, α, β, R command

❹ When M, S, T,or ※ B code is commanded in a canned cycle block, M, S, T or ※ B is sent out when the first positioning is accomplished. This applies even when an L-time command is given.

M, S, T and ※ B must generally be commanded independent of other codes.

❺ If any of the following codes is commanded during the canned cycle mode, alarm "170" results.

- G codes in group ＊ except G04, G70, G71, G72
- G codes in group 07 (G41, G42)

Therefore, if G92, G27, or G28 is to be commanded, the canned cycle must always be canceled before that command.

❻ An independent dwell (G04) block can be inserted in the middle of the canned cycle mode. The dwell is executed correctly.

❼ If a canned cycle is commanded in the tool radius compensation-C (G41, G42) mode, alarm "182" results.

❽ Before a canned cycle can be entered, spindle forward rotation or reverse rotation (M03 or M04) must always be started by automatic operation command. Do not enter a canned cycle by switching between spindle forward rotation and reverse rotation by manual operation.

❾ Calling subprogram (M98) in the canned cycle mode a subprogram can be called up by commanding M98 P……L……; in the canned cycle mode. The canned cycle can be continuously executed by the called subprogram. In this case, although the value of P-specification (dwell time) in the canned cycle is temporarily destroyed by P-specification (program No. at jump destination) in the M98 command, the original value is automatically restored after control jumps to the subprogram. Limits on maximum quadruple levels for M98, command from punched tape, etc. are the same as in cases other than the canned cycle mode.

T0 command canned cycle and M98 in the same block causes an alarm.

⓾ L-specification for the number of canned cycle repetitions is non-modal. However, it may be temporarily saved in some special cases as in the following example.

(Example)

```
G91  G81  X10. R-20. Z-30. F100 ;
```

Even when block processing has advanced to this point,

L3; ……Because X, Y, Z, α, and R are not specified, the canned cycle is not executed. L3 is saved.

X20.; ……G81 is executed three times by the saved L3 specification. L3 is erased after completion.

Specification in a canned cycle is saved until it is actually executed as shown in the above example.

⓫ Cancellation of a canned cycle occurs when G80 or a G code in group 01 is commanded. If a G code in group 01 and the canned cycle G code are commanded in the same block, alarm "170" results. However, this does not apply if the canned cycle G code is G80.

⓬ When a canned cycle is commanded from a canned cycle cancel state, the modal G codes in up to the immediately preceding block are saved in the same state in which the canned cycle was canceled.

⓭ F commanded in a canned cycle remains intact even when the canned cycle is canceled.

⓮ As for address search in a canned cycle, note that address search in the middle of one cycle (i.e., when the block is stopped) results in an alarm. However, address search in a block in a stopped state performed by program command is possible. If multiple cycles are performed (according to L-times specification) in one block by program command, address search at completion of each cycle results in an alarm as in the above case.

⓯ Another canned cycle can commanded in a given canned cycle. If address data are omitted in the newly commanded canned cycle block, modal information in up to the immediately preceding block is assumed.

- Example of use of canned cycle

```
N10 G92 X0 Z0;
N11 G90 G98;                    ——→ Absolute, initial point return
N12 G81 X30. Y40. R-20. Z-30. F200;  ——→ Drilling cycle
N13      M98 P400;              ——→ Subprogram call
N14 G00 X0 Y0;
N15      T05;                   ——→ Tap selection
N16      M06;                   ——→ Tool change
N17 G84 X30. Y40. R-20. Z-30. F2000;  ——→ Tapping cycle
N18      M98 P400;              ——→ Subprogram call
N19 G00 X0 Y0;
      ⋮
O400
N100 G91 X40. L3;  ⎫
N101      Y40. ;   ⎪
N102      X-40. L3;⎪
N103      Y30. ;   ⎬  Hole position pattern subprogram
N104      X-40. L3;⎪
N105 G90 G80;      ⎪
N106      M99;     ⎭
```

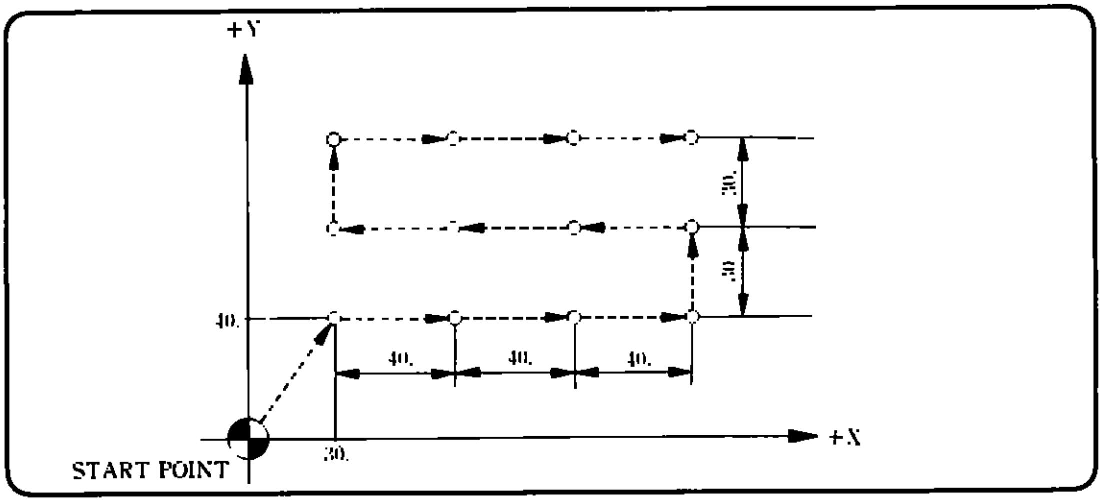


Fig. 2.13.14 Example of Canned Cycle Use

2.13.1.2 Solid tap function ❖

This function enables tapping to be completely synchronized between spindle rotation and Z-axis feed. It eliminates the need for a floating chuck, and thus enables high-speed, high-accuracy tapping. Conventional tapping can also be commanded by the program.

(1) Conditions for the addition of solid tap function

The solid tap function can only be added under the following conditions:

(a) S5-digit analog output option is installed.

(b) The spindle drive is performed by the VS-626MT III or VS-626VM3 made by Yaskawa.

(c) The spindle or spindle motor is equipped with PG, and the gear ratio used for solid tap is fixed (for equipment where the spindle and motor are directly coupled, the best performance can be obtained).

(2) Description of programming

(a) G code for commanding the solid mode

In order for tapping using the solid tap function and tapping of the conventional type to be discriminated, the following G codes must be commanded.

- **G93—Solid tap mode command (ON)**

  Solid tap is executed in the subsequent tap cycles (G84/G74 commands) (per-rotation feed command is used). No machining other than solid tap can be executed in this mode.

- **G94—Solid tap mode cancel command (OFF)**

  The solid tap mode is canceled, and the conventional mode is entered. Conventional tap is executed in the subsequent tap cycles (per-minute feed command is used).

NOTE G93/G94 are G codes in group 05, and are modal. When the power is turned ON or the equipment is reset, G94 is set by default.

(b) Commanding a solid tap cycle

Solid tap can be executed by commanding the following after issuing the G93 command.

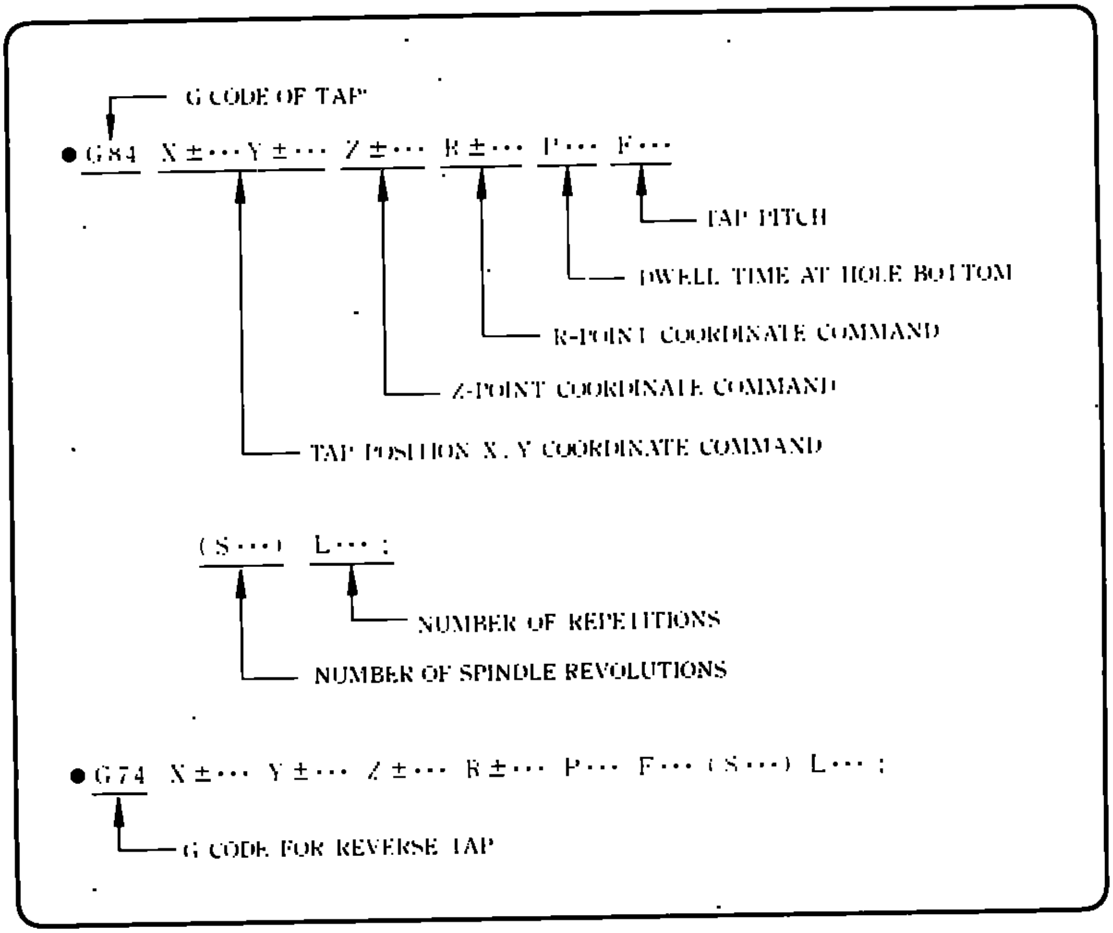


Fig.2.13.15 Solid Tap Cycle Command

The format and contents other than F are the same as for conventional tap. F in solid tap is commanded by Z-axis pitch per revolution of spindle (mm/rev, in/rev).

Unit of F command and command range

When input in mm 1 = 0.0001 mm/rev
Range 0.0001 to 200.0000 mm/rev
When input in inch 1 = 0.000001 in/rev
Range 0.000001 to 7.874015 in/rev
Range of S-command 1 = 1 rev
Range 1 to 4500 rev

NOTE No commands can be issued in which F × S exceeds 24000.0mm/min or 944.8 in/min.

(c) Operation

(i) M＊＊ command

Gear to perform solid tapping is selected.
When it is not commanded, only gear A is used.

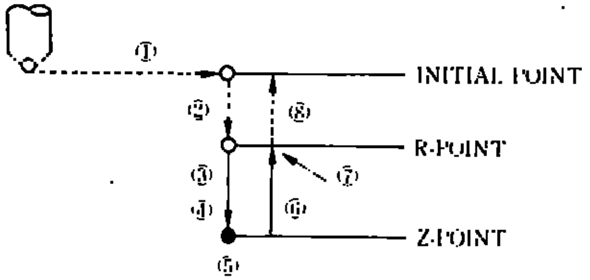
(ii) G93 command

When the G93 command is executed, the spindle stops and a position control loop relative to the spindle is created for the solid tap mode to be entered. Depending on parameter setting (Pm1053 D2 = 1), the solid tap mode can be entered after the spindle division is performed and the spindle is positioned at the designated position after it stops. The spindle dividing is performed at the proper position only when the rotation numbers of the spindle and spindle PG rotation is 1 to 1. This mode is canceled by G94.

(iii) G84/G74 commands

- G84 command

(1) Positioned as commanded by X, Y
(2) Positioned at R-point
(3) Spindle rotated in normal direction, and cutting fed up to Z-point
(It is checked at the beginning of this block whether the deviation pulse of spindle and Z-axis is within the designated quantity. Error detection ON)
(4) Spindle stopped (deviation pulse of spindle is checked here)
(5) Dwell (if P commanded)
(6) Spindle rotation reversed, and cutting fed up to R-point
(7) Spindle stopped
(8) Positioned at the initial point (in the case of G98)



NOTE Linear accel/decel (constants set by Pm2471) is automatically applied to the spindle movement.

Fig.2.13.16

- G74 command

Operates in the same way as G84 except that spindle forward in (3) for G84 is reverse rotation for G74, and than spindle reverse rotation in (6) for G84 is forward rotation for G74.

(iv) G94 command

When the G94 command is executed, the G93 solid tap mode is canceled. If G84/G74 is commanded in this mode, tap movement is changed to the conventional type.

(d) Program example

```
N1 G92 X0 Y0 Z0;
(M **);                                       ..................................①
N2 G93;                                       ..................................②
N3 G98 G90;                                   ..................................③
N4 G84 X100. Y100. Z-50 R-20. F1. S3000;     ......................④
N5     X200. Y200. ;                          ..................................⑤
N6     X10. Y10. ;                            ..................................⑥
N7 G80 ;                                      ..................................⑦
N8 G94 ;                                      ..................................⑧
```

Table 2.13.8 Description of Movement

	Description of movement
①	Spindle gear A/B change (Only gear A used when this command is not provided.)
②	Solid tap mode ON
③	Canned cycle modal information
④	Solid tapped at (100., 100.) with 1mm pitch
⑤	Solid tapped at (200., 200.) with 1mm pitch
⑥	Solid tapped at (10., 10.) with 1mm pitch
⑦	Canned cycle canceled
⑧	Solid tap mode OFF

NOTE

❶ Notes related to G93

(a) Only G93, S, F, and N codes can be commanded in a G93 block. If any other code is commanded, alarm "250" results.

(b) When commanding G93, make sure the G codes in group 01 are in a G00 or G 01 state. If commanded in any other mode, alarm "250" results.

(c) S-code in the G93 (solid tape) mode is handled as S-command for solid tap.

(d) No G codes other than those listed below can be commanded during the G93 mode (otherwise, alarm "250" results).

- Commandable G codes: G00, 01, 04, 70, 71, 72, 74, 80, 84, 90, 91, 98, 99

In the case of G01, however, move commands cannot be given although the G code may be commanded.

(e) G93 spindle dividing places the spindle at a fixed position according to the origin pulses of the spindle PG.

(f) After once starting the G93 mode, usually the spindle will not rotate even if the mode is canceled by G94, unless M03 or M04 is commanded again. (Exception may be programmed by mechanical sequence processing.)

Although S commands are left, gear selection signal is not output in the G93 mode. Specify necessary S commands together with M03 or M04.

If more than one gear is provided, select the gear to be used for solid tapping before G93 is commanded and executed.

❷ Notes related to G84/G74, G94 cancellation of solid tap

(a) Concerning precautions on the method of command for R-point, Z-point, G98 /G99, G90/G91, and number of repetitions (L), those on the conventional tap program are applied as is.

(b) Signs of the output voltages for spindle forward rotation and reverse rotation by G84 and G74 are changed by a parameter. When Pm1001 D0=0, forward rotation is output with positive (+) sign; when Pm1001 D0=1, forward rotation is output with negative (−) sign. For spindle reverse rotation, SINV input is not used and the rotation is automatically reversed by the controller.

(c) When commanding G94 after completion of solid tap, be sure to cancel the canned cycle with G80.

(d) When G94 is commanded after completion of solid tap, F-command is cleared to 0. So be sure to command F when commanding the cutting feed program after commanding G94.

❸ Relationship between solid tap and various operations

(a) Dry run

Whether the solid tap mode or the conventional tap mode should be executed when G93 is executed with the dry run switch ON can be selected by the following setting for a parameter.

pm4016 D6=0	Conventional tap mode
pm4016 D6=1	Solid tap mode

If a tap mode is called up with the setting of "pm4016 D6=1" and G93 is executed with dry run switch ON'G93 becomes invalid, and the G84/G74 commands in the solid tap mode are handled as conventional type G84/G74.

The feed/min of G84/G74 is determined by the setting of JOG switch.

Once the dry run switch is turned on and the solid tap mode is started, tapping feed by G84/G74 is processed the same as that of conventional G84/G74 even after the dry run switch is turned off. Therefore, the spindle is not activated when G93 is executed. Whether dry run in the G93 block is ON, is determined when the G93 code is read. This determination normally occurs during execution of a block immediately preceding the G93 block. Therefore, when G93 is operated with dry run ON for program check, etc., be sure that the dry run switch is ON from the beginning and do not change it before completion.

(b) Auxiliary function lock

If G93 is executed with the auxiliary function lock switch ON, G93 is nullified, and the G84/G74 commands in the solid tap mode are handled as conventional type G84/G74.

Positioning control on the spindle is not exercised either.

Whether the auxiliary function in the G93 block is locked, is determined when the G93 code is read. Therefore, when performing program check, etc. with auxiliary function lock switch ON, be sure the dry run switch is also ON from the beginning and do not change it before completion.

(c) Machine lock, Z-axis disregard.

When solid tap is executed with machinelock.Z axis disregard ON, the spindle rotates, but Z-axis is only displayed.

(d) Override, spindle override

Feed override during solid tap is entirely fixed. However, rapid feedrate override is effective. Spindle override input in the solid tap mode is nullifed, and entirely fixed.

(e) Feed hold

Feed hold during cutting by solid tap G84/G74 is invalid.

(f) Change of mode

Change of mode during cutting by solid tap G84/G74 is invalid.

(g) Program restart

When the program is restarted for any block in the solid tap mode, G93 is not executed. G93 must therefore be executed by commanding it with MDI.

(h) Although E command for reciprocal feed does not cause an alarm, it is disregarded. The return feedrate is set by parameter pm1252.

(3) Solid tap-related functions

By adding the solid tap functions (optional), the following functions are added or modified.

(a) Display of servo position deviation

The servo position deviation display screen during the solid tap mode indicates the spindle servo position deviation value on X-axis, and the number of synchronization error pulses of spindle and Z-axis on Y-axis. (Z-axis servo position deviation value is indicated on Z-axis.)

However, the pulse display on Y-axis is not exact when in the continuous mode (single-block OFF).

Also note that when parameter Pm4015 D6=1, the peak values of synchronization error pulse can be displayed on X-and Z-axis. (X-axis: positive peak value; Z−axis: negative peak value)

(b) Error detection during solid tap

By setting parameter Pm4015 D5 to "1," it is possible to place rapid feedrate commands during solid tap (X, Y positioning, Z-axis positioning) into the error detect OFF mode. When this parameter is turned ON, the cycle time can be reduced.

NOTE In this case, however, control advances to the Z-axis block immediately after X, Y positioning pulse distribution is completed, so care should be taken for the program.

(c) Alarm codes

The following alarm codes are added.

(i) Alarm "250" solid tap function unusable command

- The G93 block contains an invalid command other than G93, S, F, and N.
- The G code in group 01 when G93 is commanded is not 00 or 01.
- An invalid G code is commanded during the G93 mode.
- The pitch F of solid tap is greater than the allowable range (200. mm/rev).

(ii) Alarm "2191" solid tap I/O error

- When solid tap is executed, the spindle positioning control loop is not incorporated.
- When dividing is performed for G93, SLPC is turned off before dividing is completed.

(iii) Solid tap return speed increase function

By setting a value to parameter pm1252, it is possible to control the solid tap cutting speed so that return speed (Z-point → R-point cutting) is 'n' ($0.1 \leqq n \leqq 25.5$) times the forward (R-point → Z-point cutting). However, the number of spindle revolutions is clamped (fixed) to the commanded value of S magnification rate which is the maximum number of solid tap revolutions of Pm1416. Furthermore, when Pm1252=0, n=1.

Pm1252 Specification range: 0 to 255; Unit: 1=0.1 time
Specification 0 is assumed to be 1 time.

(4) Solid tapping related parameter and I/O signals

(i) Parameter list

① Spindle allocation stop verification r/min

pm1225 : 1st spindle gear A   Setting 1 = 1 r/min
pm1226 : 1st spindle gear B   Range 1 to 255

② Error detection ON area at spindle loop

pm1331 : 1st spindle gear A   Setting 1 = Least detection unit
pm1332 : 1st spindle gear B   Range 1 to 255

③ Servo error area at spindle loop ON

pm1351 : 1st spindle gear A   Setting 1 = 1 %
pm1352 : 1st spindle gear B   Range 0 to 200

④ Spindle r/min for reference 10V in gear used in solid tapping
pm1415 : 1st spindle gear A Setting 1 = 1 r/min
pm1435 : 1st spindle gear B Range 1 to 32767

⑤ Solid tapping spindle maximum r/min
pm1416 : 1st spindle gear A Setting 1 = 1 r/min
pm1436 : 1st spindle gear B Range 1 to 32767

⑥ Spindle position loop gain at solid tapping
pm1417 : 1st spindle gear A Setting 1 = 0.01 (1/s)
pm1437 : 1st spindle gear B Range 1 to 32767

⑦ Solid tapping servo axis pull-in in-position width
pm1500 : Solid tapping gear A Setting 1 = 1 pulse
pm1501 : Solid tapping gear B Range 1 to 32767

⑧ Solid tapping synchronous compensation parameter (k1)
pm1502 : Solid tapping gear A Range-32767 to 32727
pm1504 : Solid tapping gear B

⑨ Solid tapping synchronous compensation parameter (k2)
pm1503 : Solid tapping gear A Range-32767 to 32727
pm1505 : Solid tapping gear B

⑩ Spindle and motor gear ratio at solid tapping
pm1510 : Number of 1st spindle gear A spindle side teeth
pm1511 : Number of 1st spindle gear A spindle intermediate teeth
pm1512 : Number of 1st spindle gear A motor intermediate teeth
pm1513 : Number of 1st spindle gear A motor side teeth
pm1514 : Number of 1st spindle gear B spindle side teeth
pm1515 : Number of 1st spindle gear B spindle intermediate teeth
pm1516 : Number of 1st spindle gear B motor intermediate teeth
pm1517 : Number of 1st spindle gear B motor side teeth
Setting 1 = 1 tooth
Range 0 to 32768

⑪ Solid tapping spindle linear accel/decel time constant
pm2471 : 1st spindle gear A Setting 1 = 1 ms
pm2472 : 1st spindle gear B Range 0 to 32767

⑫ Spindle allocation starting speed
pm2541 : 1st spindle gear A Setting 1 = 1 r/min
pm2542 : 1st spindle gear B Range 0 to 32767
Spindle allocation creep speed
pm2546 : 1st spindle gear A Setting 1 = 1 r/min
pm2547 : 1st spindle gear B Range 0 to 32767

(ii). I/O Signal List

① Solid tapping spindle selection input (STGR1)

# 31155 : 1st spindle	0 : Gear A selected
	1 : Gear B selected

2.13.2 Hole-opening Pattern Cycle (G70, G71, G72) ※

When a radius and angle are specified, this function performs positioning by converting the data into rectangular coordinates.
This function is used with canned cycles (G81 to G89, G73, G74, G76, G 77). "Bolt hole circle," "arc,"and "line at angle"can be implemented. The tool is moved to the coordinate position calculated from the radius and angle at rapid feedrate (G00).

(1) Bolt hole circle (G70) command

The tool is positioned at a point on a circle drawn around coordinates specified by X and Y at radius I, equally divided by L, starting from a point at angle J against X-axis on that circle.

```
G70 X······Y······I······J······L······ ;
```

X, Y : Used to define the center position of the bolt hole circle. The center is defined according to G90/G91.

I : Used to set the radius of the bolt hole circle. It must be set with a positive number. The unit can be set with the minimum unit of setting.

J : Used to set the angle of the point that is positioned first. It is given in units of 0.001 degree, with the counterclockwise direction assumed to be positive.

L : Used to set the number by which the circle is divided. If positive, positioned counterclockwise; if negative, positioned clockwise.

Example of Programming

```
G81 G98 G90 Z-50. R-20. F20 L0 ;
G70 X90. Y30. I40. J20. L6 ;
G80 G00 X0 Y0 ;
```

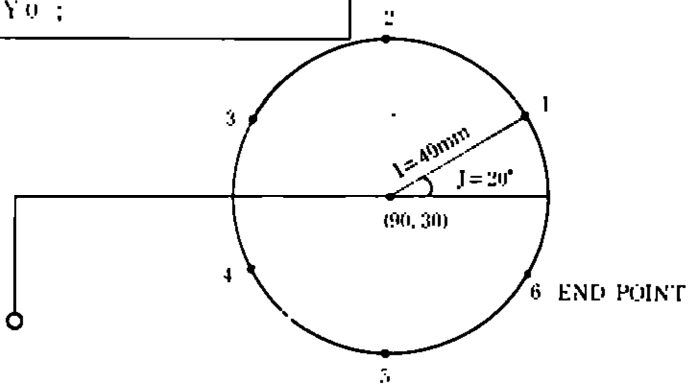


Fig. 2.13.17 Bolt Hole Circle Command (G 70)

(2) Arc (G71) command

The tool is positioned respectively at L points at angle intervals K on a circle drawn around coordinates specified by X and Y at radius I starting from a point at angle J against X-axis on that circle.

```
G71 X······Y······I······J······K······L······ ;
```

X, Y : Used to define the center position of the arc. The center is defined according to G90/G91.

I : Used to set the radius of the arc. It must be set with a positive number. The unit can be set with the minimum unit of setting.

J : Used to set the angle of the point that is positioned first. It is given in units of 0.001 degree, with the counterclockwise direction assumed to be positive.

K : Used to set the angle interval. It is given in units of 0.001 degree. If positive, positioned counterclockwise.

L : Used to set the number of positioning times. It must be set with a positive number.

Example of Programming

```
G81  G98  G90  Z-50.  R-20.  F20  L0 ;
G71  X70.  Y30.  I110.  J20.  K15.2  L7 ;
G80  G00  X0   Y0 ;
```

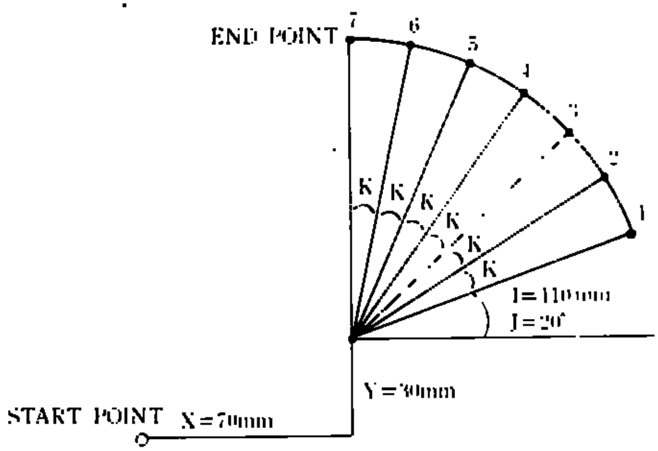


Fig. 2.13.18 Arc (G71) Command

(3) Line at angle (G72) command

The tool is positioned at L points at equal intervals I in the direction at angle J against X-axis starting from the coordinates specified by X and Y.

```
G72 X······Y······I······J······L······ ;
```

X, Y : Used to set the coordinates of the start point. The center is defined according to G90/G91.

I : Used to set intervals. If I is negative, positioned in the direction symmetrical with a point centered on the start point. The unit can be set with the minimum unit of setting.

J : Used to set the angle. It is given in units of 0.001 degree, with the counterclockwise direction assumed to be positive.

L : Used to set the number of positioning times. It must be set with a positive number.

Example of Programming

```
G81 G98 G90 Z-50. R-20. F20 L0 ;
G72 X70. Y30. I25. J15.5 L6 ;
G80 G00 X0 Y0 ;
```

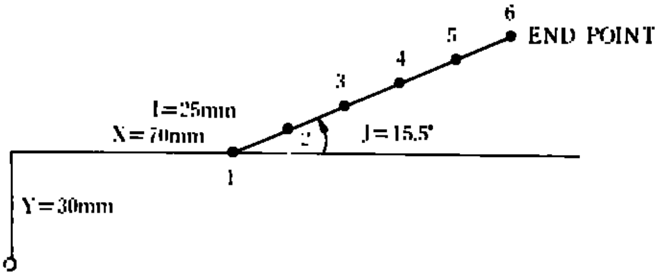


Fig. 2.13.19 Line at Angle Command (G72)

NOTE

❶ For a hole-opening cycle to be executed using G70, G71, or G72, a canned cycle where L = 0 (G73, G74, G76, G77, G81 to G89) must, as a rule, be commanded in the preceding block as shown in the program example above. By definition of L = 0, the canned cycle movement is not executed at that position and hole machining data are entered.

❷ G70, G71, or G72 and the canned cycle G code can be commanded in the same block. However, a canned cycle (G73, G83) containing I, J, or K commands cannot be commanded in the same block. When using G73 or G83, and 2-step hole canned cycle (G181, G182, G185, G186, G187 or G189) be sure to use Q or command I,J,or K in the previous block in advance.

❸ Because G70, G71, or G72 machining is completed on the last hole, if the tool is to be moved to the next position, it is necessary for the G91 (incremental) mode to calculate that position. G90 (absolute) mode is more convenient because it is not a necessary calculation.

❹ When G70, G71, or G72 machining is completed, the system remains in the canned cycle mode. Attention must be paid to the program in the next block. Generally, it is necessary to cancel the mode with G80.

❺ If G70, G71, or G72 is commanded in nose radius compensation (G41, G42), alarm "182" results.

❻ G70, G71, and G72 are non-modal G codes.

❼ G70, G71, and G72 must always be commanded in a canned cycle mode (otherwise, alarm "171" results).

2.13.3 Circle Cutting (G12, G13) *

This function provides a canned cycle in which a series of operations to cut a circle can be commanded with a single block. Various commands are provided as shown below.

(1) Circle cutting (G12, G13) command

G12 (G13) I······D······F······ ;

With this command, a circle can be cut.

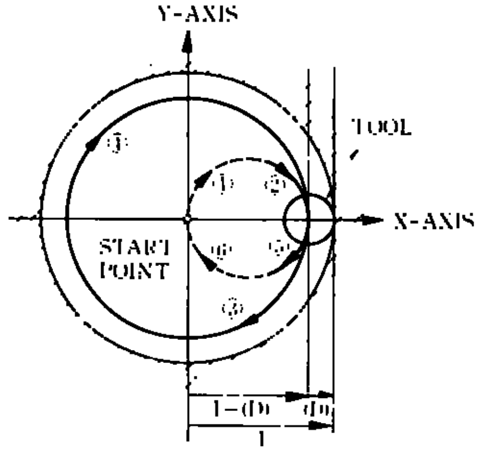
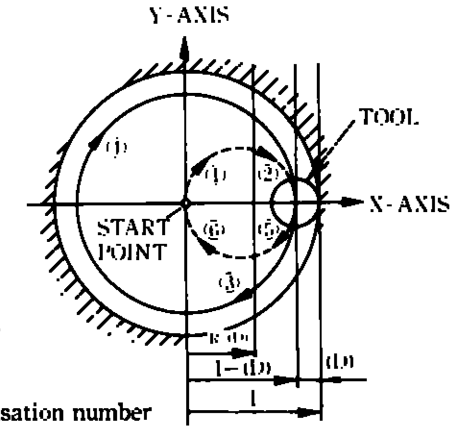


Fig. 2.13.20 Circle Cutting Command (G12, G13)

(2) High-speed feed interval specification R

G12 (G13) I······R······D······F······ ;

With this command, circle cutting is executed as shown in Fig.2.13.21. R specifies the interval in which the tool is fed at high speed.



TOOL-PATH
G12 : ①→②→③→④→⑤→⑥
G13 : ⑥→⑤→④→③→②→①
① and ⑥ are high speed feed interval.

NOTE

(D) Shows that tool diamefer compensation number of tool offset amount is stored.

Fig. 2.13.21 High-speed Feed Interval Specification R

· G12: Clockwise (CW)
G13: Counterclockwise (CCW)
I : Radius of finished circle (signed incremental value)
R : High-speed feedrate interval (signed incremental value)
D : Tool radius compensation No.
F : Cutting feedrate

(3) Circle section repeat specification L

G12 (G13) I······D······L······F······ ;

With this command, circle cutting is executed by repeating the circle section a number of times as specified by L.

(4) Spiral circle cutting specification Q, K

G12 (G13) I······D······K······Q······F······ ;

With this command, it is possible to command a cycle in which the circle is cut while moving in a spiral form as shown in Fig.2.13.22. For easy understanding, the diagram below shows an example in which tool radius (D) is set to 0.

NOTE Q is commanded without a sign.

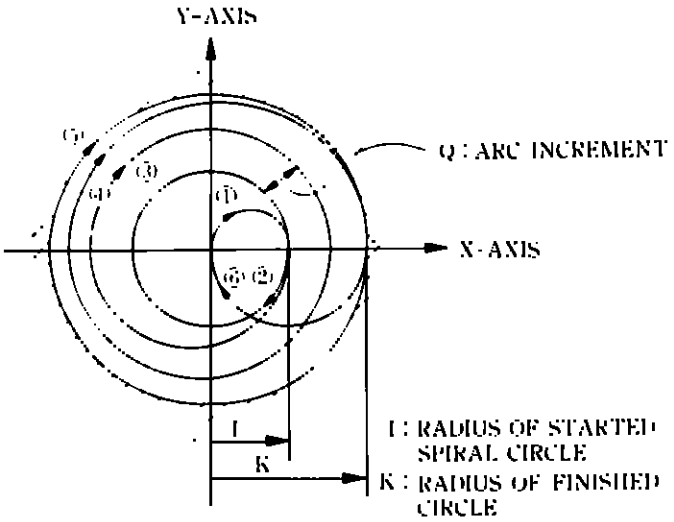


Fig. 2.13.22 Spiral Circle Cutting Specification Q , K

(5) Combined use of rapid feedrate interval, repeat command, and spiral Commands (2) to (4) described above can combined as in

```
G12 (G13)  I……R……(or J……) K……Q……L……F…… ;
```

J : Allowable cutting

(6) Automatic operation specification J of high-speed feed interval

```
G12 (G13)  I……J……D……F……  ;
```

With this command, the high-speed feed interval is operated automatically, as shown is Fig.2.13.23 (b).

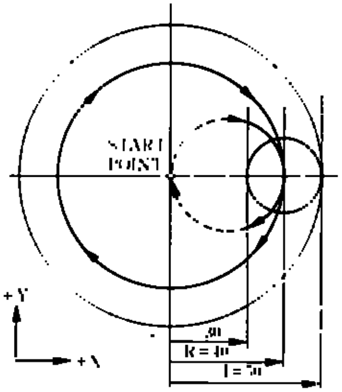
J : Cutting allowance

NOTE However, it should be commanded in the range J≦D, otherwise, high-speed feeding can not be executed.

Example of Programming

(a)

```
G00  Z-40. .
G12  I 50. R40. D10  F300 :
G00  Z40. .                    D10=10.0mm
```



(b)

```
G00  Z-40. .
G13  I-50. J 7. D15  F300 .
G00  Z40. .                    D15=-8.0mm
```

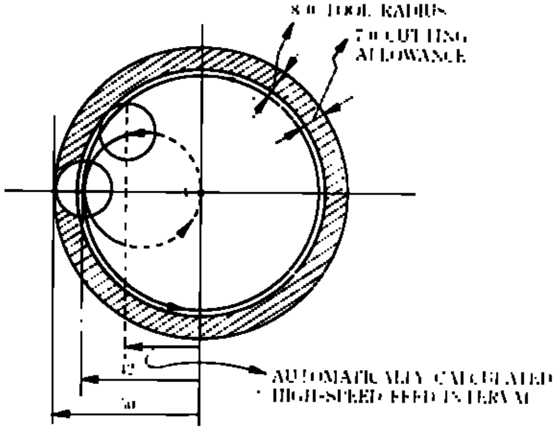
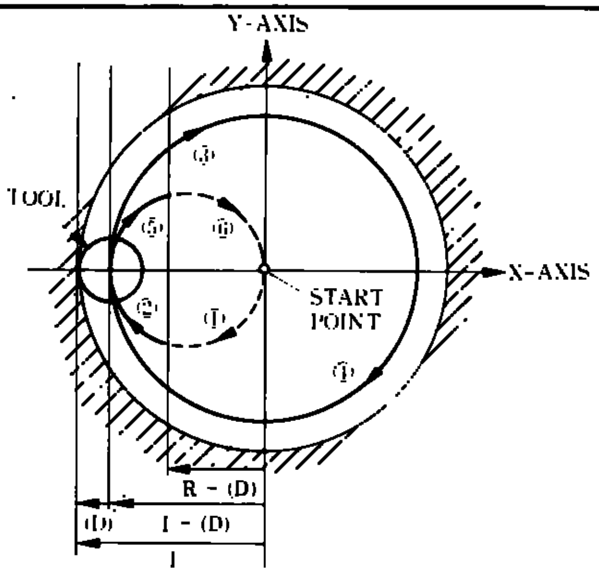


Fig. 2.13.23 Tool Radius Correction No. D is Commanded

NOTE

1. Circle cutting is only performed in the XY plane. If commanded for cutting in other planes, alarm "361" results.
2. The speed in the high-speed feed interval is set by parameter #2862. Override can be applied to the speed in the high-speed feed interval. However, if the dry run switch is ON, the speed is matched with the dry run speed.
3. In circle cutting (G12, G13), the tool radius is compensated for irrespective of G41 or G42 (tool offset) specification.
   G12 and G13 must be commanded in the G40 (tool radius compensation cancel) mode. If commanded in any other mode, alarm "182" results.
4. Although movements in "X+" direction are explained here, movements in "X−" direction (symmetrical to Y-axis) can also be made by changing the signs of I, J, K, and D accordingly. In the example in Fig. 2.13.24, all I, R, and (D) are negative. Note however that cutting in the direction toward Y-axis is impossible.
5. I, J, K, R, Q, and L in circle cutting are effective in one block only. Although J and L can be omitted, "I" must always be commanded. If all are omitted, alarm "362" results.
6. Radius I of finished circle and high-speed feed interval R are subject to the follwing limits (if the limits are exceeded, the tool is not fed at high speed):
   $$|R - d| < |I - d|$$
   Also, if the signs of R − d and I − d are different, high-speed feed is not executed.
7. When commanding G12 and G13, be sure to command tool radius compensation number D. If D is omitted, the tool is moved in the following way depending on the setting of the parameter pm4010.
   pm4010 D1 = 0 : Offset is applied with previously specified D number.
   D1 = 1 : Offset is not applied.



TOOL PATH

G12 : ①→②→③→④→⑤→⑥

G13 : ⑥→⑤→④→③→②→①

Note : (D) indicates the set value of tool offset amount.

Fig. 2.13.24 X-Direction Movement (Object to Y-Axis)

2.13.4 Mirror Image ON/OFF (M95/M94) ※

(1) The following M codes are used.

Table 2.13.9 Mirror Image ON/OFF

M Code	Meaning
M94	Mirror image OFF
M95	Mirror image ON

(2) Mirror images can be applied (ON) or removed (OFF) from an arbitrary point in the program. These codes must always be commanded with an independent block.

(3) The M94 and M95 commands are modal. When the power is turned on, M94 (OFF) is set by default.

(4) The axis in which a mirror image is applied, is specified with settings Pm0002 D0 to Pm0003 D0 or using a "mirror image axis specifying" switch. For details on this operation, see Par. 3.5.1.28.

(5) Mirror image is applied to the specified axis from a block next to the one in which M95 is commanded. That is, movement is made in the direction opposite to the program-commanded movement direction.

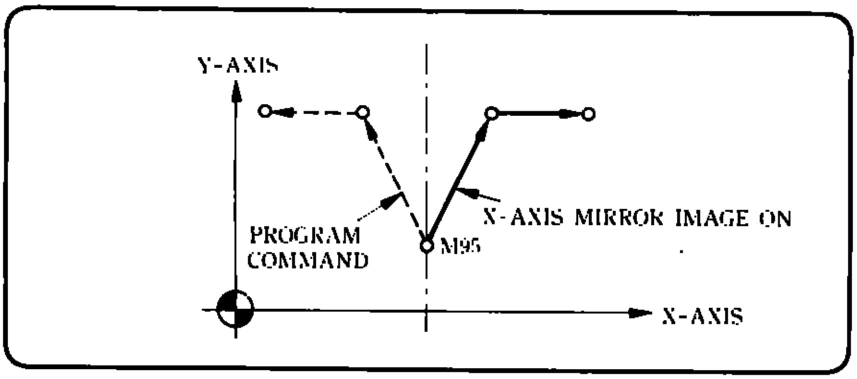


Fig. 2.13.25 Mirror Image ON (M95)

The same mirror effect is obtained regardless of whether the move command is made by an absolute or incremental command. Remember that the M95-commanded block is the mirror point.

(6) Mirror images are canceled in blocks after the M94-commanded block. The position at which mirror image is begun to be applied and the cancel position must be the same.

NOTE

1. When commanding G28 or G29 due to a change of tools or completion of machining, be sure to turn the mirror image off with M94. If G28 or G29 is commanded with the mirror image on, alarm "127" results.
2. Mirror image is invalid for the corrective movement by tool length offset function.
3. Do not change the mirror image axis specification while operating in the M95 (image ON) mode.
4. The current value display (POS) while mirror image is ON indicates the actual tool movement.
5. Be sure to program so that the position at which mirror image is begun to be applied and the position where it is terminated are at the same position. If not matched, movement after mirror image cancellation is displaced by the difference between the mirror image start and end positions (see the diagram below).

```
N01 G92 X0 Y0 ;
N02     M95 .      ......... MIRROR IMAGE ON
N03 G90 G01 X40. Y40  F300 ;
N04     X120.  ;
N05     Y100   ;
N06     X40. ;
N07     Y40. ;   <--- IF M94 . IS INSERTED HERE.THE DOTTED LINE
N08     X0 Y0 ;       MOVEMENT SHOWN BELOW RESULTS.
N09     M94 ;      .........CORRECT MIRROR IMAGE OFF
N10     X20  Y-40. ;
N11     X60. ;
```

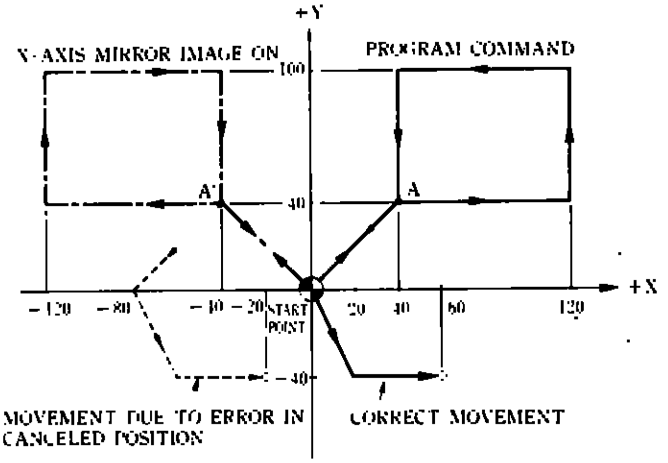


Fig. 2.13.26 Movement when Mirror Image Start and End Positions Differ

6. When the equipment is reset, M94 is set by default.
7. Combination with work coordinate rotation, scaling, or coordinate rotation function.

   The above G codes must be arranged in the specified order.

   Otherwise, alarm "285" occurs.

   Work coordinate rotation (G54) > Mirror image (M94) > Scaling (G51)> Coordinate rotation (G68).
8. Scaling and coordinate rotation in mirror image.

   When the scaling function is used in a mirror image operation, the mirror image function changes the center of scaling.

   When the coordinate rotation function is used in a mirror image operation, the mirror image function changes both the center of rotation and the direction of rotation.

---

(7) Mirror image external input function

(a) Outline

In addition to the conventional mirror image function, a new function is provided that permits setting the mirror image execution mode by parameter setting when the system is turned on or reset. If the mirror image execution mode is on when the system is turned on or reset, it is also possible to specify by parameter setting whether or not the mirror image should be applied at an intermediate point of G28.

(b) Method of usage

- When the system is turned on or reset

  Pm4001 D0 = 0: M94 mode (mirror image OFF)

  Pm4001 D0 = 1: M95 mode (mirror image ON)
- If the M95 mode is on when the system is turned on (i.e., Pm4001 D0=1)

  Pm4001 D2 = 0: Mirror image applied at intermediate point of G28

  Pm4001 D2 = 1: Mirror image not applied at intermediate point of G28

NOTE If #4001 D0 = 0, the conventional specifications are assumed. Therefore, if G28 or G29 is commanded in this state, the mirror image must be turned off with M94.
If G28 or G29 is commanded with mirror image on, alarm "127" results.

(c) Program example

Example of Programming (Mirror image ON only for X-axis)

```
G90 G00 X0 Y0 ; ·····1
G00 X100. Y50. ; ·····2
G28 X40. Y100. ; ·····3
  :
```

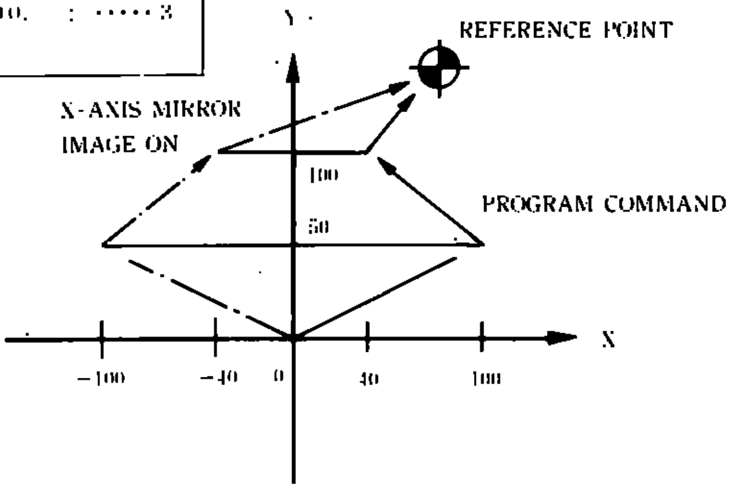


Fig. 2.13.27 Mirror Image ON in Intermediate Point of G28

NOTE If axis specification in the mirror image (M95) mode is commanded with M code, the M codes used to set and reset it must be prohibited from being preread by parameter setting (Pm4400 to Pm4409).

2.13.5 Programmable Data Input (G10) ※

The tool offst quantity and work coordinate system can be adjusted by using the G10 command as shown below.

(1) Setting tool offset quantity (G10)

Tool offset quantities are normally set by writing with MDI. When set by program, the following methods are followed.

(a) H/D common method

```
G10  L10  P……R…… ;
```

(P: tool offset No.; R: tool offset quantity)

With this command, an arbitrary tool offset quantity can be altered to the commanded value.

- When G10 is executed in the G90 mode, value R is saved as is.
- When G10 is executed in the G91 mode, value R is added to the value of the previous offset quantity.

NOTE L10 may be omitted.

(b) H/D separate method

Writing H code and writing D code varies in program format.

(i) For H code

```
G10  L10  P……R…… ;
```

NOTE L10 may be omitted.

(ii) For D code

```
G10  L12  P……R…… ;
```

or

```
G10  Q1  P……R…… ;
```

NOTE Either L12 or Q1 may be used.

(2) Changing work coordinate system shift quantities ※
The respective work coordinate systems corresponding to G54 to G59 are preset as setting data.

(a) By commanding from the program as

```
G10  Q2  Pm  Jn  X……Y……Z…… (※α……β……β……);
```

or

```
G10  L2  Pm  Jn  X……Y……Z…… (※α……β……β……);
```

the work coordinate system shift quantities can be altered.

- Q2 and L2 are used to discriminate the setting of work coordinate system shift quantities from those of tool offset quantities.
- Pm (m: 1 to 6) corresponds to the work coordinate system "m" to be set.

No P or P1	G54
P2	G55
P3	G56
P4	G57
P5	G58
P6	G59

- Jn (n: 1 to 27) corresponds to the expansion quantity of the work coordinate system to be set.

No J specification or J0 or J1 means G54 J1	G54 J1
J2	G54 J2
⋮	⋮
J27	G54 J27

The above meanings are summarized in the table below.

Table 2.13.10 Programmable Data Input Format

Type			Format
Offset	H/D Common		G10 L10 P······ R······ ; G10 P······ R······ ;
	H/D Separate	H	G10 L10 P······ R······ ; G10 P······ R······ ;
		D	G10 L12 P······ R······ ; G10 Q1 P······ R······ ;
Work Coordinate System Shift quantity	6 Pairs		G10 Q2 P1 ~ P6 (No J,J0,J1) Axis shift quantity ; G10 L2 P1 ~ P6 (No J,J0,J1) Axis shift quantity ;
	54 Pairs		G10 Q2 P1 ~ P6 J1 ~ J9 Axis shift quantity ; G10 L2 P1 ~ P6 J1 ~ J9 Axis shift quantity ;
	162 Pairs		G10 Q2 P1 ~ P6 J1 ~ J27 Axis shift quantity ; G10 L2 P1 ~ P6 J1 ~ J27 Axis shift quantity ;

NOTE · When the work coordinate system shift amount is changed, then commanded value is stored as it is regardless of the G90 or G91 mode.

2.13.6 Program Call Functions

2.13.6.1 Subprogram call (M98／M99)

This function permits calling up a subprogram that has been stored in memory after appending the appropriate program No. and executing it any number of times.

(1) The following M codes are used.

Table 2.13.11 Subprogram

M Code	Meaning
M98	Subprogram call-up
M99	Subprogram End

(2) Subprogram call (M98)

```
M98P·········L······ ;
```

With this command, it is possible to call a subprogram of the program No. specified by P and execute it a number of times specified by L. If L is omitted, the subprogram is executed one time. It is also possible to call another subprogram from the called subprogram. Such multiplex calls can be nested up to quadruple calls.

(3) Subprogram format (M99)

Subprograms must be created in the following format and must be saved in part program memory before they can be called.

```
O··········· ;              ←— PROGRAM NO.
················· ;
················· ;
················· ;
M99 ;                       ←— SUBPROGRAM END
```

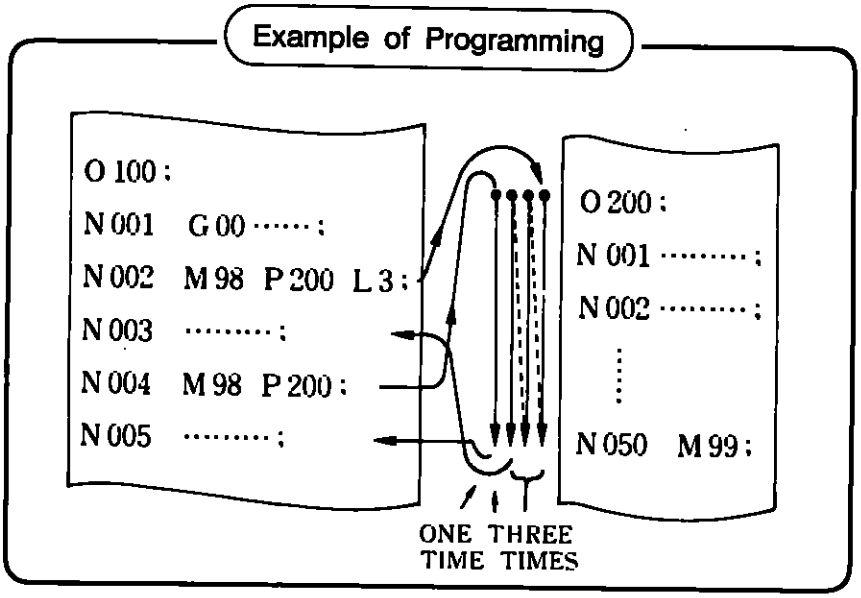
Fig. 2.13.28 Subprogram Format (M99)

(4) Automatic return from subprogram (M99)

```
M99 ;
```

Command M99 in an independent block at the end of the subprogram. If M99 is commanded in a program (subprogram) at the jump destination specified by M98 above, control automatically returns to the block next to the main program from which the subprogram has been called.

Example of Programming



```
O 100 ;
N 001   G 00 ······ ;
N 002   M 98  P 200  L 3 ;
N 003   ········· ;
N 004   M 98  P 200 ;
N 005   ········· ;
```

```
O 200 ;
N 001 ········· ;
N 002 ········· ;
   ⋮
N 050   M 99 ;
```

Fig. 2.13.29 Subprogram Execution Process

Special use of M98

```
M98P...L...Q...;
```

Enter the above command to start the called subprogram from the sequence number specified by Q.

NOTE Use up to four digits for the sequence number.

(5) Special use of M99

M99P…… ;

With this command, when the subprogram is terminated, control returns to the block of the sequence No: specified by P without returning to the block next to the main program.

NOTE

1. If the program No. specified by address P cannot be found, alarm "390" results.
2. This function can only be used when the subprogram is stored in part program memory. The main program can be commanded from NC tape or part program memory.
3. If the subprogram multiplicity exceeds a quadruple nest, an error results.
4. If M99 ; is commanded for the main program, control returns to the beginning of the main program and the system operates in an endless cycle.

2.13.6.2 Program copy (G25) ※

(1) $$\boxed{G25\ \ P\underbrace{\cdots}_{p1}\underbrace{\cdots}_{p2}Q\underbrace{\cdots}_{q1}\underbrace{\cdots}_{q2}L\cdots\ ;}$$

With this command, it is possible to execute a section of programs from sequence No.q2 in arbitrary program No. pl to sequence No.q2 in program No.ql a number of times as specified by L.

G25 : G code for program copy
P : pl; start O number
: p2; start sequence No.
Q : ql; end O number (pl = ql, however)
: q2; end sequence No.
L : Number of repetitions (if omitted, one time)
Enter 5-digit O-number and 4-digit N-number. Leading zeros in p2 and q2 cannot be omitted.

(2) Although G25 can be commanded from tape, MDI, or memory operation, the programs (pl, ql) to be copied must have been stored in memory. The sequence Nos. specified by addresses p2 and q2 are searched from the beginning of each program No. Therefore, sequence Nos. must not be duplicated. If duplicated, the first encountered sequence No. is valid.

(3) G25 can even be commanded in the program to be copied.
This is called a multiplexed program copy, and can be set in up to four levels. However, if it exceeds a quadruple, alarm "261" results.

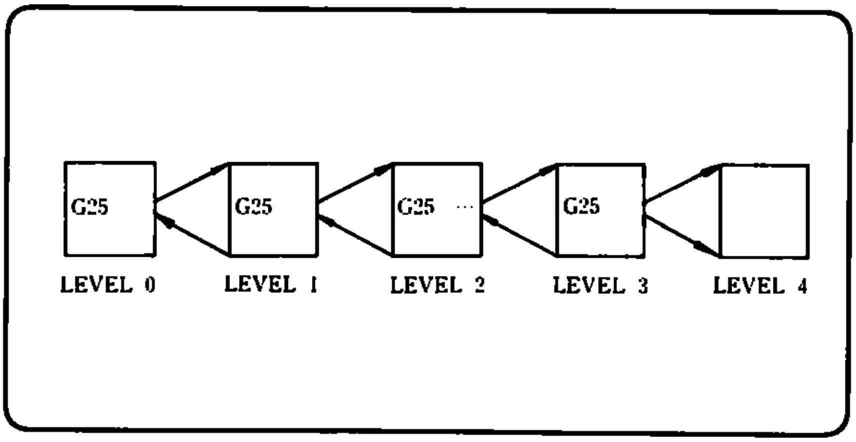


Fig. 2.13.30 Multiplexed Program Copy

(4) Program copy can be commanded in canned cycles (G70 to G72, G73, G76, G81 to G89).

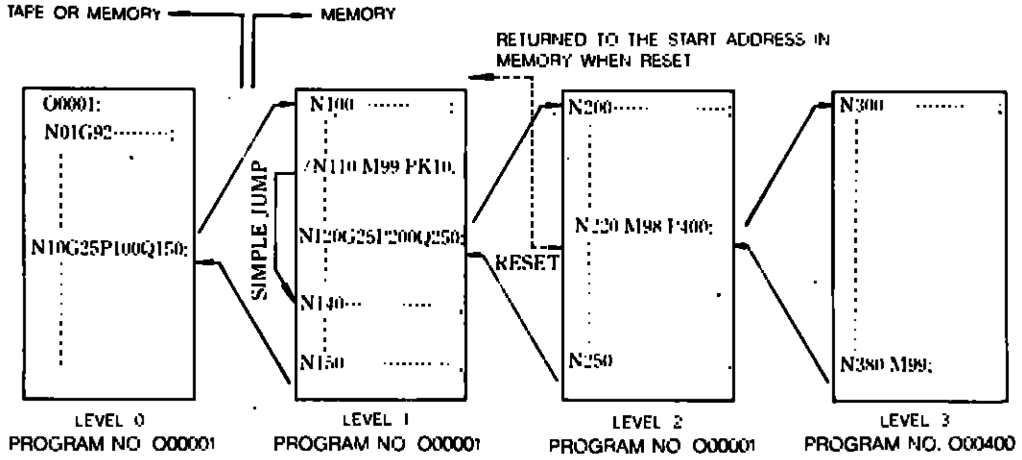
NOTE

1. When used with a subprogram, if a total of G25 and M98 call levels exceeds a quadruple, alarm "261" results.
2. If G25 is used in programmable call (G65, G66), multiplexed call is possible up to a quadruple separately from macroprogram calls.
3. Sequence Nos. specified by addresses p2 and q2 are searched from the beginning of each program No. Therefore, the sequence Nos. must not be overlapped.
4. In the command below

   | G25P (p1) (p2) Q (q1) (q2) L ···; |

   (1) If p1 cannot be found, alarm "390" results.
   (2) If p2 and q2 cannot be found, alarm "393" results.
   (3) If p1, p2, q2 cannot be found, alarm "260" results.
5. If the G25 command is reset during execution due to an error or M30/M02 is reset, control returns to the beginning of level-0 calling program.
6. If P or Q is commanded in less than four digits, sequence Nos. alone are considered, and the sequence Nos. in the current program No. are searched.
7. For both P and Q commands, the leading zeros can be omitted. However, if there are more than five digits, the four lower digits are assumed to be the sequence No.
8. If M98 or M99 is commanded in the G25 block, alarm "260" results.
9. If H99 is commanded between blocks indicated by G25 p2 and q2, alarm "264" results.
10. The 0 number (q1) at the end of Q command may be omitted. When omitted, this number is automatically assumed to be the same as the start 0 number in P command.
11. Program copy command in a canned cycle cannot be commanded in the same block as the canned cycle command. Otherwise, G code alarm "170" results.

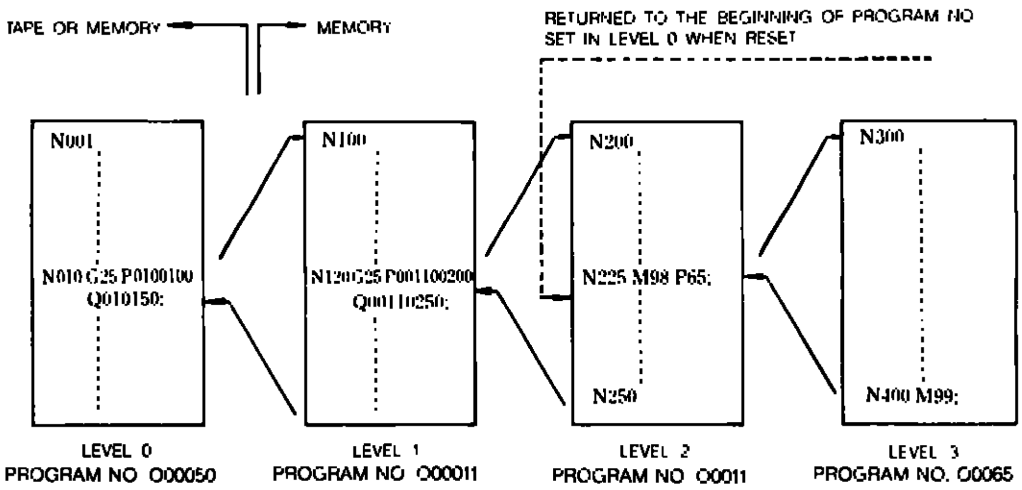
- Program example 1 (When copied within the same program No.)



NOTE If commanded from tape, the program No. must always be indicated in the P command.
(Example) N10 G25 P0100100 Q150;
└─ Program No.

Fig. 2.13.31 Copy in Same Program No.

- Program example 2 (when other program Nos. are copied)



NOTE 1. M98 can be used in the program copied by G25. Even when G25 and M98 are used together as in this case, the quadruple level cannot be exceeded.
2. When jumped to any other level by M99, it enters an endless loop.

Fig. 2.13.32 Copy other Program No.

2.13.6.3 Macroprogram call (G65, G66, G67) ※

(1) Simple call

G65 P……L…… 〈argument specification〉 ;

With this command, it is possible to call a macroprogram of the program No. specified by P and execute it a number of times as specified by L.

(2) Modal call

G66 P……L…… 〈argument specification〉 ;

With this command, the mode is entered from which a macroprogram of the specified program is called. After that, the specified macro is executed a number of times as specified by L each time move command is excuted.

NOTE G67 ;

This command cancels the modal call mode.

2.13.7 Auto Corner Override (G106)※

Cutting inside of a corner increases load on the tool as the contacting area of the tool and the workpiece increases.

The auto corner override function varies cutting speed by automatically overriding at the point where load to the tool varies in a single block. The point of change of the load is calculated from the programmed cutting margin and the current tool diameter. The override rate is set by the parameter.

(1) Command format

The auto corner override is a non-modal G code:

```
G106  G01 X······Y······I······ (or J···) F···;
```

I: Cutting margin in X-axis direction

J: Cutting margin in Y-axis direction

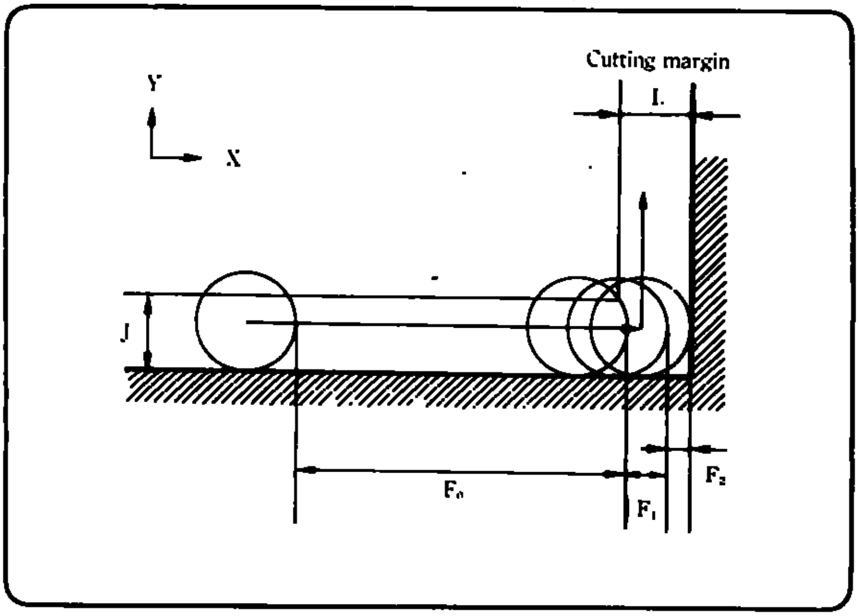


Fig. 2.13.33 Command format

(2) Parameters

Preset override rates at the first and second steps:

Pm2120...F1: Percentage of overriding of first-step F feedrate (1 to 100)

Pm2121...F2: Percentage of overriding of second-step F feedrate (1 to 100)

NOTE If the cutting margin is greater than the tool diameter, Pm2120 is fixed to 100%.

(3) Operation

This function automatically calculates the position where load to the tool increases inside a corner from the tool diameter and the cutting margin. Load is applied to different points depending on the tool diameter and the cutting margin. There are the following three cases:

① Tool diameter is not greater than 1/2 of cutting margin (D ≦ I/2)

② Tool diameter is greater than cutting margin (D > I)

③ Tool diameter is not greater than cutting margin and is greater than 1/2 of cutting margin (I/2 < D ≦ I)

In the above cases, cutting speed is determined differently as follows:

① Tool diameter is not greater than 1/2 of cutting margin.

- No corner override is applied regardless of the parameter value. Cutting is performed at uniform speed set by F0.

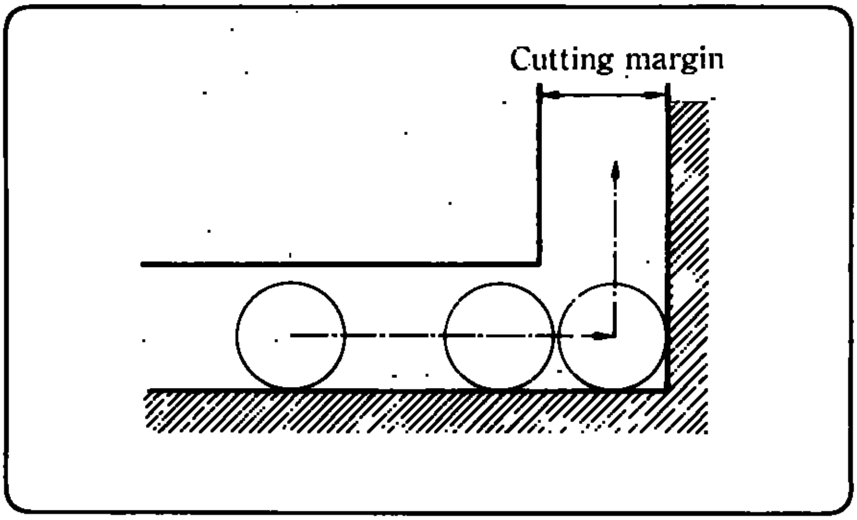


Fig. 2.13.34 Tool diameter is not greater than 1/2 of cutting margin.

② **Tool diameter is greater than cutting margin.**

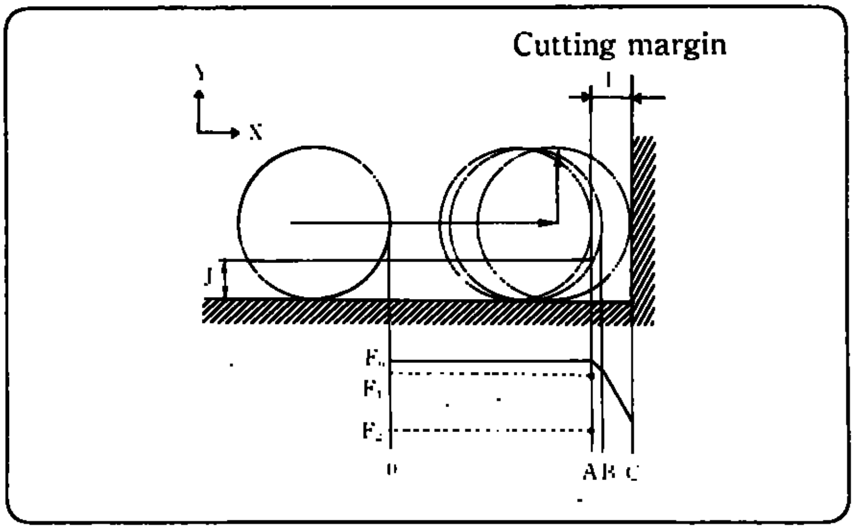


Fig. 2.13.35 Tool diameter is greater than cutting margin.

Cutting speed

F_0 = (commanded feedrate) × (override rate%)

$F_1 = F_0$ × (pm2120)

$F_2 = F_0$ × (pm2121)

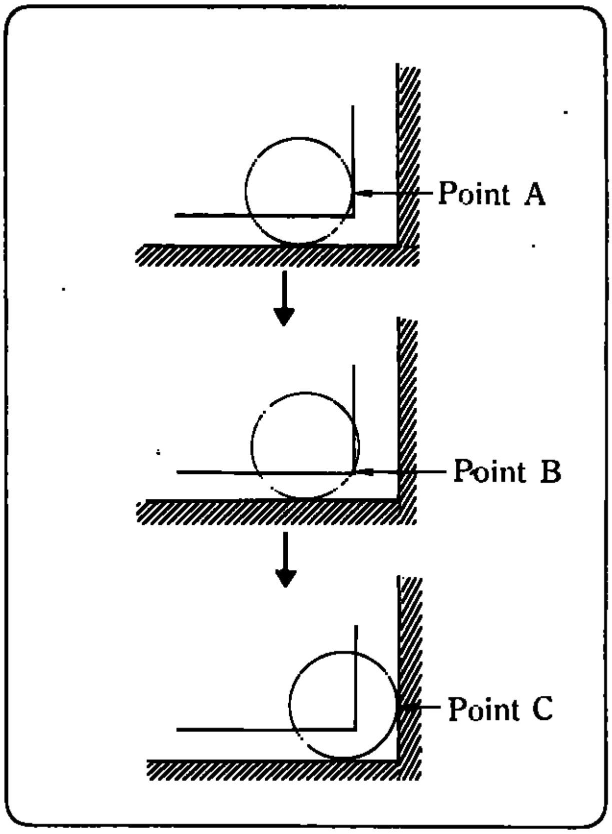


Fig. 2.13.36 Speed change points

③ Tool diameter is not greater than cutting margin and is greater than 1/2 of cutting margin.

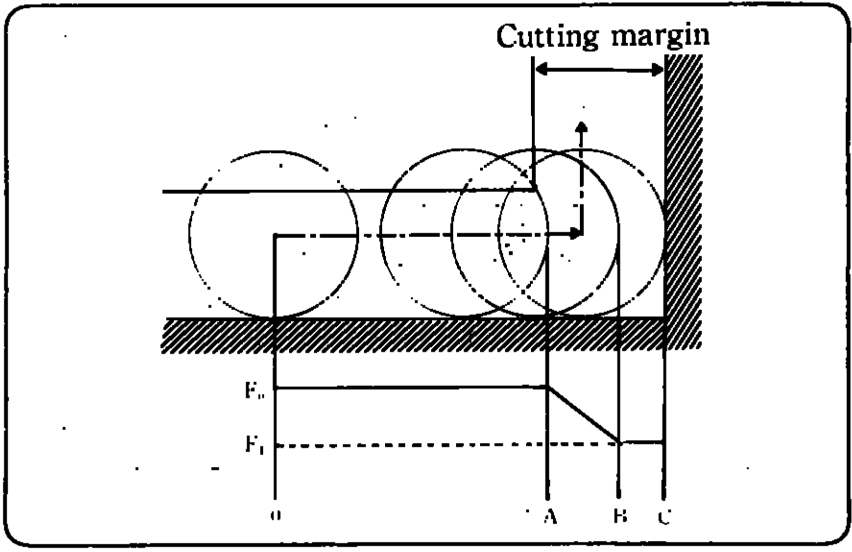


Fig. 2.13.37 Tool diameter is not greater than cutting margin and is greater than 1/2 of cutting margin

Cutting speed

F_0 = (commanded feedrate) × (override rate%)

$F_1 = F_0$

Speed change point

$F_2 = F_0 \times$ (pm2121)

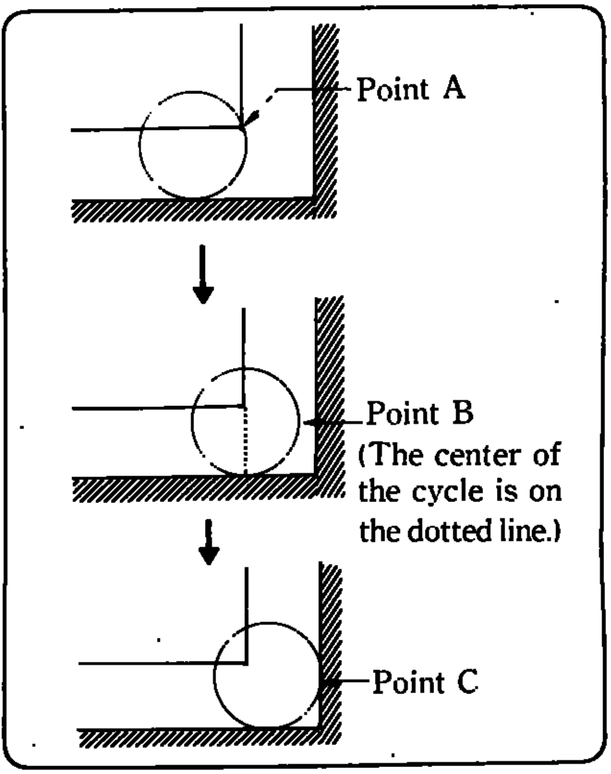


Fig. 2.13.38 Speed change points

NOTE

❶ **Group 01 G code with G106**

If G106 is to be commanded, group 01 G code must be G01.

Otherwise, an alarm occurs.

❷ **G106 is unavailable in canned cycle**

G106 cannot be used in a canned cycle. Cancel the canned cycle mode before commanding G106.

❸ **Axial command in G106 block**

Axial commands cannot be used in a G106 block except X and Y-axis commands. If an axial command other than X or Y command is used, or if neither X nor Y command is used, an alarm occurs.

❹ **X and Y in the same block**

Corner override can be applied only in single axis direction.

If both X and Y commands are used in the same block, corner override is applied only in a single axis according to cutting margin command I or J.

❺ **Cutting command (I or J) is essential in G106 block**

Cutting margin must be commanded (with either I or J) in a G106 block. If both the commands are omitted or used at the same time, an alarm occurs.

❻ **Override parameters**

Set the override parameters so as to satisfy the following conditions:

(1) Override rate must be from 1(%) to 100(%). If a value greater than 100 is set, 100 is used. If 0 is set, no corner override is applied.

(2) F_0, F_1, and F_2 must be in the following relation:

$F_0 < F_1 < F_2$

If parameter setting makes F_2 greater than F_1, the F_1 override rate is appropriated for actual F_2 override.

2.13.8 Stored Stroke Limit check

2.13.8.1 Stored stroke limit *

This function checks to see that the current values of each axis during manual or automatic operation are not in the prohibited area specified by a parameter or the G22 command.

When entered in the prohibited area, operation is stopped and an error is displayed.

(1) The outside of the boundary specified by a parameter for No.1 prohibited area (stored stroke limit-A) is made the prohibited area. Generally, this is used in place of overtravel. The parameter specifies upper limit value A1 and lower limit value B1.

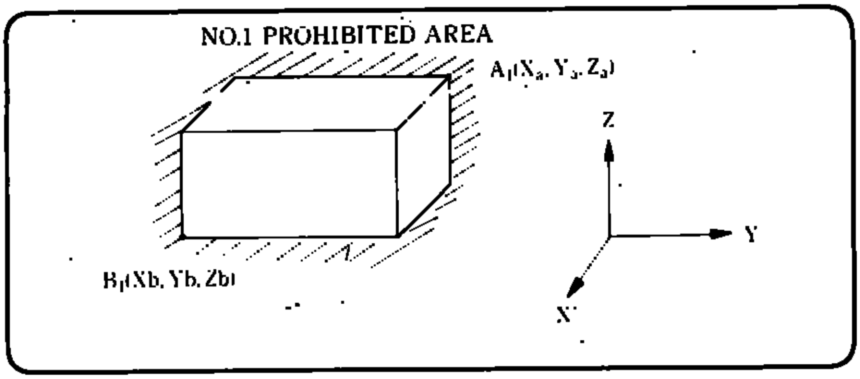


Fig. 2.13.39 Stored Stroke Limit A

(2) Whether or not stroke A (No.1 prohibited area · stored · stroke limit check) is to be performed can be set for each axis by a parameter.

Table 2.13.12 No.1 Prohibited Area Stored Stroke Limit Check Valid/Invalid

Axis	X	Y	Z	4	5
Parameter	Pm6002 D0	Pm6002 D1	Pm6002 D2	Pm6002 D3	Pm6002 D4
Meaning of Bit	0 : Performs limit check.				
	1 : Not performs limit check.				

(3) Area specifying parameters and setting are as follows:

Table 2.13.13 Parameter and Setting of No.1 Prohibited Area Specification

Axis	X	Y	Z	4	5
Boundary Value (+) : point A1	Pm6901	Pm6902	Pm6903	Pm6904	Pm6905
Boundary Value (−) : point B1	Pm6911	Pm6912	Pm6913	Pm6914	Pm6915

2.13.8.2 Stroke Limits B, C ✲

(1) The boundary can be specified by setting or program G22. Whether the inside or outside of that boundary must be a prohibited area is set by a parameter.

No.2 prohibited area is referred to as stroke limit B. No.3 to No.5 prohibited areas are referred to as stroke limit C.

```
G22  X……Y……Z……  I……J……K……  P…… ;
            ↑           ↑          ↑
            ①           ②          ③
```

① A2-A5 points upper limit value

② B2-B5 points lower limit value

③ Area specification: No.2 to No.5

When the above command is instructed, the program begins to check the prohibited area specified by P.

```
G23 P…… ;
```

with this command, the check function is cleared.

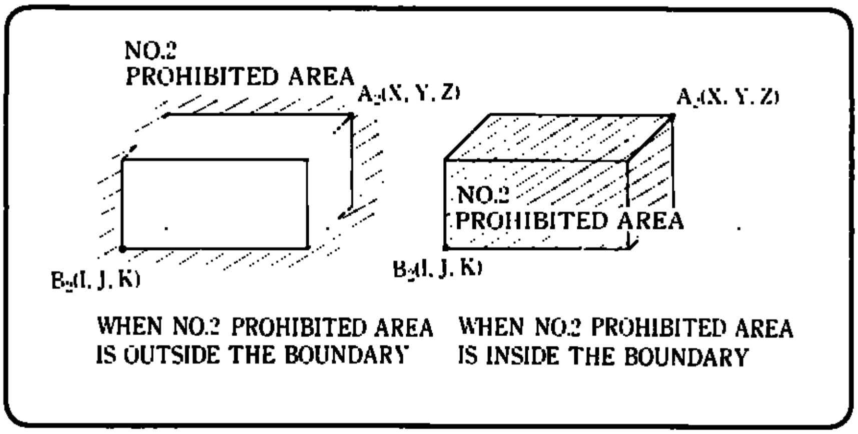


Fig. 2.13.40 Stored Stroke Limit B and C

NOTE If P is omitted, P2 is used.

(2) The setting Nos. of area specification are as follows:

Table 2.13.14

Type of prohibited area boundary value	X	Y	Z
No.2 prohibited area (+) : A2 point	Pm0831	Pm0832	Pm0833
No.2 prohibited area (−) : B2 point	Pm0834	Pm0835	Pm0836
No.3 prohibited area (+) : A3 point	Pm0837	Pm0838	Pm0839
No.3 prohibited area (−) : B3 point	Pm0840	Pm0841	Pm0842
No.4 prohibited area (+) : A4 point	Pm0843	Pm0844	Pm0845
No.4 prohibited area (−) : B4 point	Pm0846	Pm0847	Pm0848
No.5 prohibited area (+) : A5 point	Pm0849	Pm0850	Pm0851
No.5 prohibited area (−) : B5 point	Pm0852	Pm0853	Pm0854

NOTE **Point-A sets the positive (+) side boundary value and point-B sets the negative (−) side boundary value on machine coordinate systems.**

(3) Specifying check axis
Specify the check axis of which stored stroke limit is to be checked in No.2 to No.5 prohibited areas. (Up to three axes can be specified.)

Table 2.13.15

Check area	2. Stroke limit check axis No.		
	No.1	No.2	No.3
No.2 prohibited area	pm6111	pm6112	pm6113
No.3 prohibited area	pm6114	pm6115	pm6116
No.4 prohibited area	pm6117	pm6118	pm6119
No.5 prohibited area	pm6120	pm6121	pm6122

1 = X-axis
2 = Y-axis
3 = Z-axis

(4) The setting Nos. for outside/inside specification of No.2 to No.5 prohibited areas are as follows:

Table 2.13.16

Check area	Setting parameter No.	Meaning
No. 2 prohibited area	Pm0008 D4	(0) Inside prohibited area
No. 3 prohibited area	Pm0008 D5	
No. 4 prohibited area	Pm0008 D6	(1) Outside prohibited area
No. 5 prohibited area	Pm0008 D7	

(5) Whether the Nos.2 to 5 prohibited area check must be ON (valid) or OFF (invalid) can be specified by setting parameter or input signal.

(i) Setting parameter

ON/OFF of Nos.2 to 5 prohibited area check can ce specified by setting parameter. However, by setting parameter, checks for up to two prohibited areas of No.2 to No.5 prohibited areas can be turned ON simultaneously.

Table 2.13.17

Check area	Setting parameter No.	Meaning
No.2 prohibited area	Pm0008 D0	(0) Invalid (1) Valid
No.3 prohibited area	Pm0008 D1	
No.4 prohibited area	Pm0008 D2	
No.5 prohibited area	Pm0008 D3	

(ii) Input signal

ON/OFF of Nos.3 to 5 prohibited area check can be specified by input signal.

Table 2.13.18

Addres	Bit		Prohibited Area
	D1	D0	
# 3012	0	0	Nos. 3 to 5 prohibited area check invalid
	0	1	No. 3 prohibited area check valid
	1	0	No. 4 prohibited area check valid
	1	1	No. 5 prohibited area check valid

NOTE 1. Whether the stored stroke limit function is ON (valid) is checked when the axis has returned to the reference point. If valid, the function is immediately executed. If the reference point is in the prohibited area, an alarm occurs immediately. To prevent this, turn on the function when all the axes have returned to the reference point after turning on power.

2. Nos. 3 to 5 prohibited area check is valid by the OR condition of the setting parameter and input signal.

NOTE

1. The No. 1 and two prohibited areas of No. 2 to No. 5 can be overlapped.
2. **Motion in prohibited area**

   Each prohibited area is valid after once returning to reference point by G28 or return to manual reference point is accomplished.
3. **Error handling**

   If the axis is in a prohibited area when the stored stroke limit is turned on, an error occurs immediately.

   If this error occurs, turn off the prohibited area of No. 2 to No. 5 by setting parameters, then rewrite data or move the axis out of the prohibited area by manual operation.
4. If an axis enters into a prohibited area causing an alarm, retraction is possible only in the reverse direction of entry.
5. **Stored stroke limit A**

   Stored stroke limit A is checked both on linear and rotary axes.

   For rotary axes, a parameter is provided to determine whether to check the limit. Normally, the limit check is not performed.

Table 2.13.19 Servo axis parameters

Axis name	X	Y	Z
Parameter	pm6030 D6	pm6031 D6	pm6032 D6
Axis name	4	5	
Parameter	pm6033 D6	pm6034 D6	

D6=0:Linear axis
1:Rotary axis

Table 2.13.20 Stored stroke limit on/off parameters for rotary servo axes

Axis name	X	Y	Z
Parameter	pm6004 D0	pm6004 D1	pm6032 D2
Axis name	4	5	
Parameter	pm6004 D3	pm6004 D4	

Each bit determines:

0: To check the limit.

1: Not to check the limit.

6. Stored stroke limits cannot be checked when the machine is locked.
7. **Limit check with axis omitted by G22**

   For the axes omitted in command G22, parameter values are used for limit check.
8. Fig.2.14.41 shows definitions of whether a boundary is to be included in the prohibited area:

Prohibited area	Boundary value	Included/not included
Outside prohibited area	Upper limit	Included
	Lower limit	Not included
Inside prohibited area	Lower limit	Not included
	Upper limit	Included

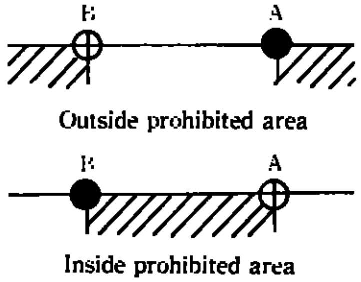


Fig. 2.13.41 Definition of whether boundary is included in prohibited areas

9. **Restriction on stored stroke limit B and C functions**

   The stored stroke limits B and C allows the designation of two prohibited areas of No. 2 to No. 5 at the same time. If more than two areas are set valid at the same time, an alarm occurs.

10. When setting the coordinate values for defining a stored stroke limit area width, set the values which are greater than the value obtained by the following calculation according to the axis feedrate. If the stored stroke limit area width is set smaller than the calculated value, there is a case that an alarm detection can not be performed.

    Feedrate (mm/min) $\div 60$(sec) $\div 10^3 = \underline{6}$ (msec)

    ↑

    Coefficient for detection

(Example)

Feedrate (m/min)	Stored Stroke Limit Area Width (mm)
4	0.4
24	2.4
36	3.6

2.13.9 Comment Statement Output (M191) ※

Comment statements in a part program are convenient to monitor which part of the program is being executed on the screen.

`M191/*YASNAC i80M*/;`

Enter the above command."YASNAC i80M" is displayed on the warning message output line on the screen. Enclose a comment statement between "/*" and "/*". If an invalid command is entered, ALM115 "M191 FORMAT ERROR" is displayed.

NOTE

1. **Up to 16 characters can be used for a comment statement.**
   If more characters are entered, only 16 characters are displayed.
2. **Output of warning**
   If a warning occurs while a comment statement is displayed, the comment disappears and "WARNING" is displayed.
3. **When M191 is commanded in the same block with motion command**
   The comment statement is displayed after the motion is completed.
4. **A comment statement in control-in or control-out will not be displayed.**
5. **A comment statement out of a M191 block will not be displayed.**

2.13.10 Break Point

When a block of the sequence number specified by the parameter has been executed in programmed operation, a warnig message of "BREAK POINT!" is issued and operation is stopped by that block.
The warning can be cleared by cycle start or resetting.
Set the sequence numbers by the following parameters:

pm801: Break point sequence number 1
pm802: Break point sequence number 2

2.13.11 High-speed Cutting (Available only when expansion CPU option is provided)*

Conventional high-speed cutting was such that the host computer decoded the part program and converted it into binary data to enable the NC equipment to move at high speed, and in this way DNC operation was performed via a communications module. For the i80M, the part program stored in part program memory is converted into binary data at high speed, making it possible to execute the program at high speed.

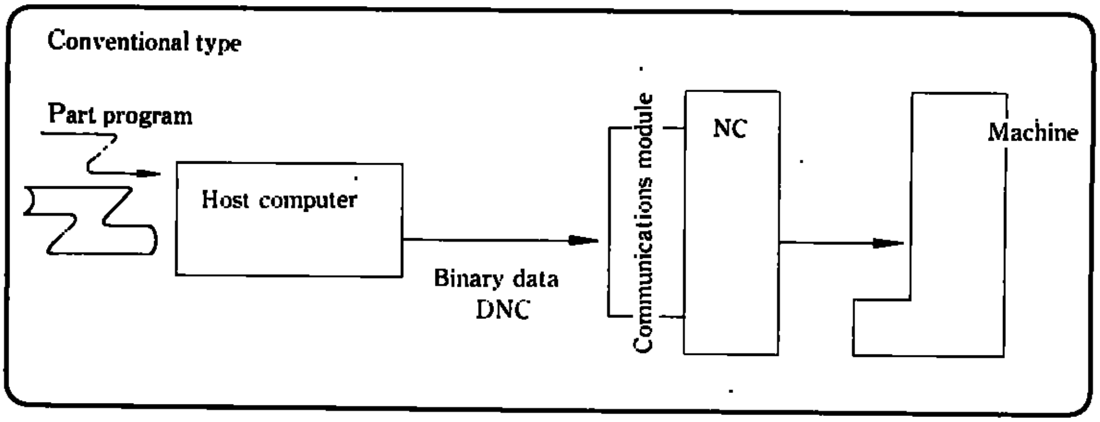


Fig. 2.13.42 Conventional High-speed Cutting

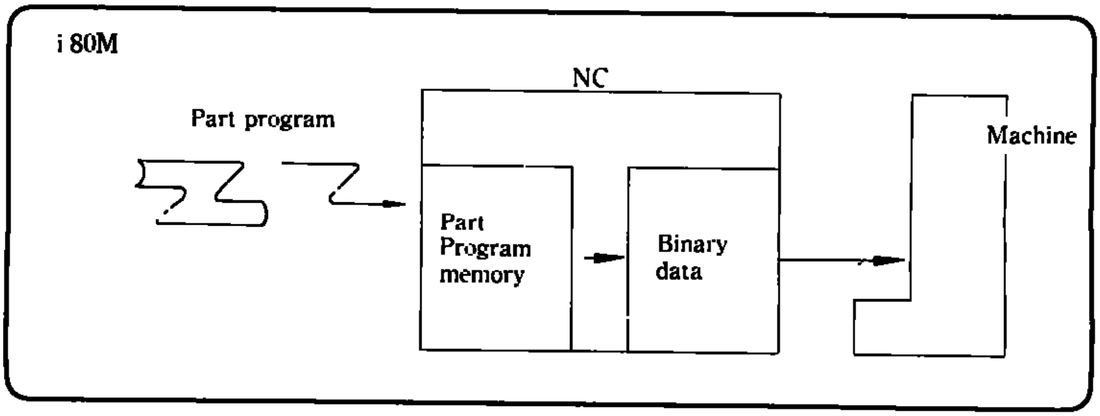


Fig. 2.13.43 YASNAC i80M High-speed Cutting

(1) Conversion mode

Although there is an operation mode in which linear interpolation (G01) blocks in part program memory are converted into movement quantities per unit time (hereafter called segment data), the following three modes are provided for conversion modes.

(i) Non-conversion mode

Same as in normal operation. The part program is decoded block by block as the machine is operated.

(ii) Specified section conversion mode

If there is a block enclosed with "HON" and "HOF" in the part program, the linear interpolation blocks that satisfy the following conditions are converted into segments for high-speed cutting operation to be executed.

- The block is an incremental (G91) block.
- The block is not accompanied by MSTB command.
- Movement quantities in each axis cannot be specified with the decimal point.

(iii) Automatic conversion mode

All linear interpolation blocks that fall under the conditions (ii) are converted into segments.

(2) Description of programming

Enclose a section that falls under (1)(ii) conditions with "HON" and "HOF."

```
(G91F10000 ; )
 ......
 HON ;
 X100 ;
 Y100 ;
 X50 ;
 X50Y50 ;
 ......
 HOF ;
 ......
```

(3) Parameters

(i) The basic axis addresses are fixed to X, Y, and Z.

Additional axis addresses can be set to A, B, C, U, V, or W as selected (parameters pm1201, pm1202).

(ii) High-speed conversion start and end specification codes

These codes are referred to as "HON"..."HOF" in this manual, and can be specified within the following range as desired.

- Three alphanumeric characters (alphabets at the beginning)
- Those which are not reserved for use in the NC language (e.g., HON, HOF, G05) can be specified with ASCII code.

Start specification code	Parameters Pm4161, Pm4162, Pm4163
End specification code	Parameters Pm4164, Pm4165, Pm4166

(iii) Segment data type

Select the type with parameter Pm4167.

(iv) Selecting the conversion mode

Select the mode with parameter Pm4160.

(v) Number of high-speed cutting axes

Set the number of these axes with parameter Pm1200.

(4) Unit of input

Select the unit of input between 0.001 mm and 0.0001 inch with parmeter Pm0007. Note that the unit of output is fixed to 0.001 mm.

NOTE

❶ Feed value(F command)-only blocks are not conditions for the end of segment conversion.

❷ Message "HSC" is displayed during high-speed cutting operation.

❸ Single blocks are invalid in a section enclosed with "HON"and"HOF".

❹ Millimeter, inch systems cannot be changed over with G20/G21.

❺ Length offset, scaling, and coordinate rotation are impossible.

❻ Dry run is invalid in a section enclosed with "HON"and"HOF".

❼ External deceleration cannot be executed in a section enclosed with "HON"and"HOF".

❽ Reset key, external reset, and feed hold are valid in a section enclosed with "HON" and"HOF".

❾ Subprogram M98 can be used.
Between"HON"and"HOF", no feedrate is displayed on the program operation screen due to special internal processing of NC.

❿ Between"HON"and"HOF", processing is performed with the basic axis (X, Y, and Z)and the axis specified parameters pm 1202. Improper parameter setting disrupts programmed operations.

Commands for axes that are not set by the parameters are disregarded, and no alarm is issued.

2.14 MACROPROGRAM (G65, G66, G67)※

A set of instructions is provided for this system. By using these instructions, the tool machine manufacturers or the users of the system can produce a program to implement their original functions. This program is called a macroprogram, which can be called and executed by a single-block command using G65 or G66.
A macroprogram makes the following possible:

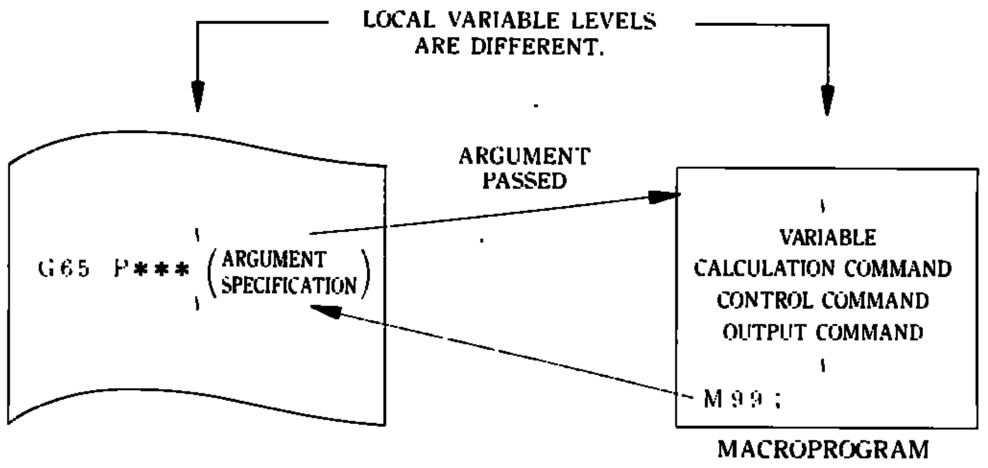
① Variables can be used.
② Arithmetic operations using variables and constants are possible.
③ Branch or repeat control is possible.
④ Messages and dates can be output.
⑤ Argument specification is possible.

Therefore, macroprogram permits creating a program to execute complex arithmetic operations or operations requiring conditional judgment. The differences between a macroprogram and subprogram are as follows:

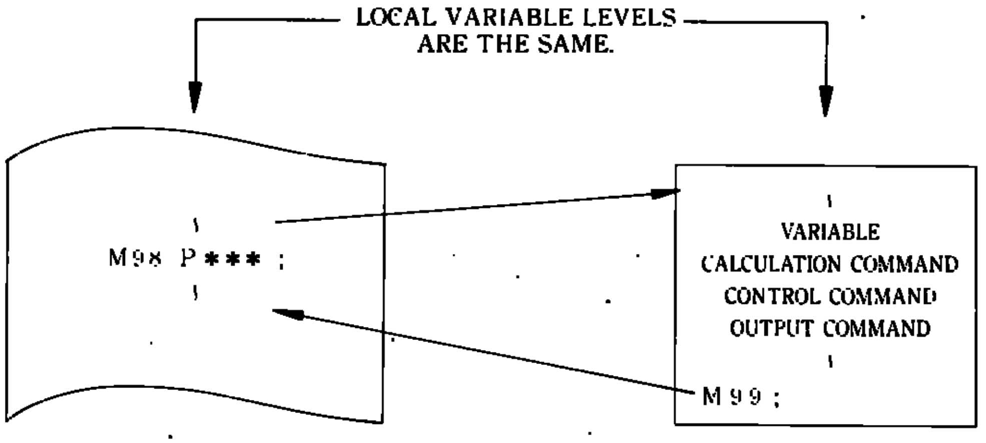
(1) Although macroprogram call (G65, G66) can specify an argument, subprogram call (M98) cannot specify an argument.

(2) When a M98 block contains any command other than P, Q, or L, control jumps to a subprogram after it is executed. On the other hand, any command other than P or L in G65 and G66 is assumed to be argument specification, so that control immediately jumps to a macroprogram.

(3) Local variables used in a macroprogram have levels corresponding to the level of that macroprogram. Local variables used in a subprogram have levels which are unchanged. That is, local variables in a macroprogram are different before and after the macroprogram is called; local variables in a subprogram are the same.



(a) MACROPROGRAM



(b) SUBPROGRAM

Fig. 2.14.1 Difference Between Macroprógram and Subprogram

2.14.1 Macroprogram Call

A Macroprogram is normally executed in its called format.
In this case, a macroprogram is called in the following ways:

Table 2.14.1 Calling up of Macro Program

Method of call	Command code	Remarks
Simple call	G65	
Modal call (a)	G66	G67 to cancel
G code call	G＊＊＊	G command in 3 digits
M code call	M＊＊＊	M command in 2 or 4 digits
S code call	S＊＊＊	S command in 6
T code call	T＊＊＊	T command in 4
B code call	B＊＊＊	B command in 4 digits

(1) Simple call (G65)

```
G65  P······L······  〈Argument specification〉  ;
```

With this command, a macroprogram of the program No. specified by P is called and executed a number of times as specified by L.
If it is necessary to pass argument to the macroprogram, the argument can be specified in this block.

Table 2.14.2

Address	Meaning	Number of digits
P	Program No.	5 digits
L	Number of repetitions	9 digits

Argument specification: A real number is assigned to the local variable that corresponds to the level of the called macroprogram. (For details, refer to the description of argument specification.)

NOTE When specifying arguments, be sure to place G65 before all arguments. Any commands before G65 are processed as normal commands, and move to macro-program after completion of processing.

(2) Modal call (G66, G67)

A modal call command sets a mode from which a macroprogram is called. When given conditoins are met, the macroprogram is actually called and executed.

NOTE G67 is used to cancel the call mode.
If arguments are specified, G66 must always be placed before all arguments.
When G66 is commanded, G67 corresponding to it must be commanded in the same block.

Table 2.14.3 Model Commanf Calling-up Conditions

Call condition	Mode setting code	Mode cancel code
After move command execution	G66	G67

(a) Call after move command execution (G66)

```
G66  P……L……  〈argument specification〉 ;
```

This command sets a mode from which a macroprogram is called. In each block after this command where a move command is given, a macroprogram of the program No. specified by P is called after the movement is completed and executed a number of times as specified by L. If there is an argument specification, the argument is passed each time a macroprogram is called as in the case of the simple call command. In this case, the relationship between argument address and local variable is the same as in simple call (G65).

(3) Macro call by G code

```
G＊＊＊ 〈argument specification〉 ;
```

With this command, a macro/subprogram of the program No. corresponding to the specified G code is called and executed.

The G＊＊＊code to call a macro/subprogram can be set by selecting from a maximum of 24 pairs of G codes (each in up to three digits) that are not used in the NC equipment. Any desired program No. to be called can be set corresponding to the G code thus set.

Table 2.14.4 Setting Parameter

Number of paris	Calling G code	Program No. to be called
1.	Pm4480 3-digit Max.	Pm4840 5-digit Max.
2. to 23.	to	to
24.	Pm4503	Pm4863

└ 24 pairs max.

(4) Macro call by M code.

```
M***  ;
```

With this command, a macro/subprogram of the program No. corresponding to the specified M code is called and executed. In this case, there are two methods to call a program.

The M***code to call a macro/subprogram can be set by selecting from a maximum of 24 pairs of M codes (each in up to four digits) except the M00, M01, M02, M30, and internal processing M codes. Any desired program No. to be called can be set corresponding to the M code thus set. In addition, the method of call shown above can be set for each M code.

Table 2.14.5 Setting Parameter

Number of pairs	Calling M code	Program No. to be called
1.	Pm4504 4-digit Max.	Pm4864 5 digit Max.
2. to 23.	to	to
24.	Pm4527	Pm4887

└ 24 pairs max.

An augment can be specified for macro program code by M code. In this block, motion commands must not be specified.

pm4020 D5 = 0 : Call without augment

pm4020 D5 = 1 : Call with augment

If more than one M code is used in a single block, the first M code is checked whether it is for macro program call. If Pm4020 (D5) is set for call with augment, the second M code is determined whether it is a normal or augment M code by the following parameter:

Pm4020 D6 = 0: Normal M code

Pm4020 D6 = 1: Argument specification

NOTE **Unlike normal M code commands, while the macro/subprogram calling M code is being executed, no M code or MF output is made.**

(5) Macro call by S code

The given S command can be made a normal S command or a macro/subprogram calling command. When the program No. to be called (Pm4888) is "0," normal S command is assumed.

When used as a macro/subprogram calling command, there are two methods to call the program.

One pair of program Nos. to be called can be set as desired.

The method of call can be selected from the two methods listed above.

When called, the command value of S is made the argument of common variable #147. No other argument specification can be made.

Table 2.14.6 Setting Parameter

Command selection	Program No. to be called
Normal S command	Pm 4888 = 0
Macro Calling S command	Pm4888 5-digit max.

When the S command is made the calling command by command selection, if two or more S codes are commanded in one block, the first encountered S code is checked to see if it is the calling S command. The second and subsequent S codes are assumed to be normal S codes or disregarded (or alarmed). Consequently, if two or more S codes need to be included, the S code desired to be the calling code must be placed before all other S codes.

Pm4020 D7 = 0: Normal S code

Pm4020 D7 = 1: Disregarded

NOTE While the calling S code is being executed, no S code or SF output is made.
The S command is a 6-digit command.

(6) Macro call by T code

The given T command can be made a normal T command or a macro/subprogram calling command. When the program No. to be called (Pm4889) is "0," normal T command is assumed.

When used as a macro/subprogram calling command, there are two methods to call the program.

One pair of program Nos. to be called can be set as desired.

The method of call can be selected from the two methods listed above.

When called, the command value of T is made the argument of common variable #149. No other argument specification can be made.

Table 2.14.7 Setting Parameter

Command selection	Program No. to be called
Normal T command	Pm 4889=0
Calling T command	Pm 4889 5 digit max.

NOTE While the caling T code is being executed, no T code or TF output is made.
The T command is a 2 or 4-digit command.

(7) Macro call by B code

The given B command can be made a normal B command or a macro/subprogram calling command. When the program No. to be called (Pm4890) is "0", normal B command is assumed. When used as a macro/subprogram calling command, there are two methods to call the program.

One pair of program Nos. to be called can be set as desired.

The method of call can be selected from the two methods listed above.

When called, the command value of B is made the argument of common variable # 146. No other argument specification can be made.

Table 2.14.8 Setting Parameters

Command selection	Program No. to be called
Normal B command	Pm 4890 = 0
Calling B command	Pm 4890 5-digit max.

NOTE

1. While the calling B code is being executed, no B code or BF output is made. The B command is a 5-digit command.
2. The value for common variable #146 is determined according to the B command function. When B command is second auxiliary function, the value is of 3 digits, when it is of additional axis, the value is read-in in the same format as that of coordinate command address.

NOTE

When macro calls of G, M, T, S and B are commanded in the same block, the priority shows below.

G > M > T > S > B

(Example)

| G······M······T······S······B······ ; | (No alarm)

Valid Invalid

| B······S······T······M······G······ ; | (No alarm)

Invalid Valid

Only G command is valid without regard to the commanded order.

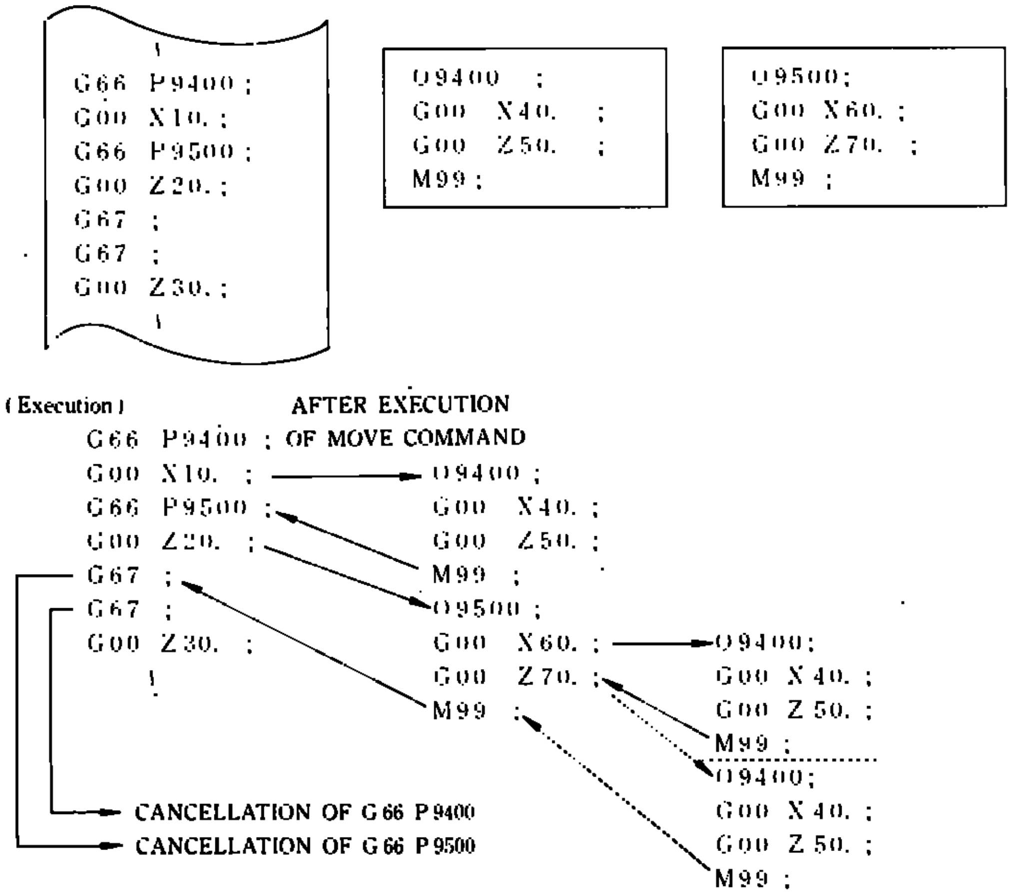
2.14.1.1 Multiplexed macro call

(1) As multiplexed calls are possible for subprograms, they are also possible for macroprograms. Another macroprogram can be called from a macroprogram. In this case; When one call is executed using G65, G66, G, M, S, T, or B code, the multiplier is increased by one.

The multiplier for multiplexed calls using the macro-calling codes described above, is possible up to a quadruple including the original call.

(2) The multiplier for multiplexed calls using G, M, S, T, and codes is one. That is, a macroprogram using G, M, S, T, or B code from a macroprogram that was called by using G, M, S, T, or B code cannot be called. If a macroprogram is called using G, M, S, T, or B code from a macroprogram that was called by using G, M, S, T, or B code, an alarm is generated for G code; and M, S, T, and B codes, are executed as normal M, S, T, and B commands.

(3) Although the modal call by G66 works in such a way that the specified macroprogram is called and executed each time a move command is executed, if two or more G66s are commanded in the same program, the commands are made valid sequentially relative to the move command in the called macroprogram. In other words, the macroprograms are executed sequentially starting with the last commanded one.



```
G66 P9400 ;
G00 X10. ;
G66 P9500 ;
G00 Z20. ;
G67 ;
G67 ;
G00 Z30. ;
```

```
O9400  ;
G00  X40.  ;
G00  Z50.  ;
M99 ;
```

```
O9500;
G00 X60. ;
G00 Z70.  ;
M99  ;
```

(Execution)

```
G66  P9400 ;
G00  X10.  ;
G66  P9500 ;
G00  Z20.  ;
G67  ;
G67  ;
G00  Z30.  ;
```

```
O9400 ;
G00  X40. ;
G00  Z50. ;
M99  ;
O9500 ;
G00  X60. ;
G00  Z70. ;
M99  ;
```

```
O9400;
G00 X40. ;
G00 Z 50. ;
M99 ;
O9400;
G00 X 40. ;
G00 Z 50. ;
M99 ;
```

NOTE

When G66 is multiplex commanded, use G67 to cancel the multiplexed commands, and be sure to cancel them sequentially beginning with the last-commanded G66.

G66 must not be commanded in a macroprogram called by G66.

Fig. 2.14.2 Macro Multiplexed Call

2.14.1.2 Argument specification

Argument specification refers to assigning real numbers to local variables used in macroprograms. There are two types of argument specifications: type I and type II. These types can be used as desired. including a combined use of two types.

(1) Relationship between address and local variable in argument specification-1

Table 2.14.9

RELATIONSHIP BETWEEN ADDRESS AND VARIABLE		CALLING COMMAND AND USABLE ADDRESS
ADDRESS FOR ARGUMENT SPECIFICATION - I	LOCAL NUMBER	G65, G66
A	# 1	○
B	# 2	○
C	# 3	○
D	# 7	○
E	# 8	○
F	# 9	○
G	#10	×
H	#11	○
I	# 4	○
J	# 5	○
K	# 6	○
L	#12	×
M	#13	○
N	#14	×
O	#15	×
P	#16	×
Q	#17	○
R	#18	○
S	#19	○
T	#20	○
U	#21	○
V	#22	○
W	#23	○
X	#24	○
Y	#25	○
Z	#26	○

NOTE ○ : Can be used
× : Cannot be used

(2) Relationship between address and local variable in argument specification- II

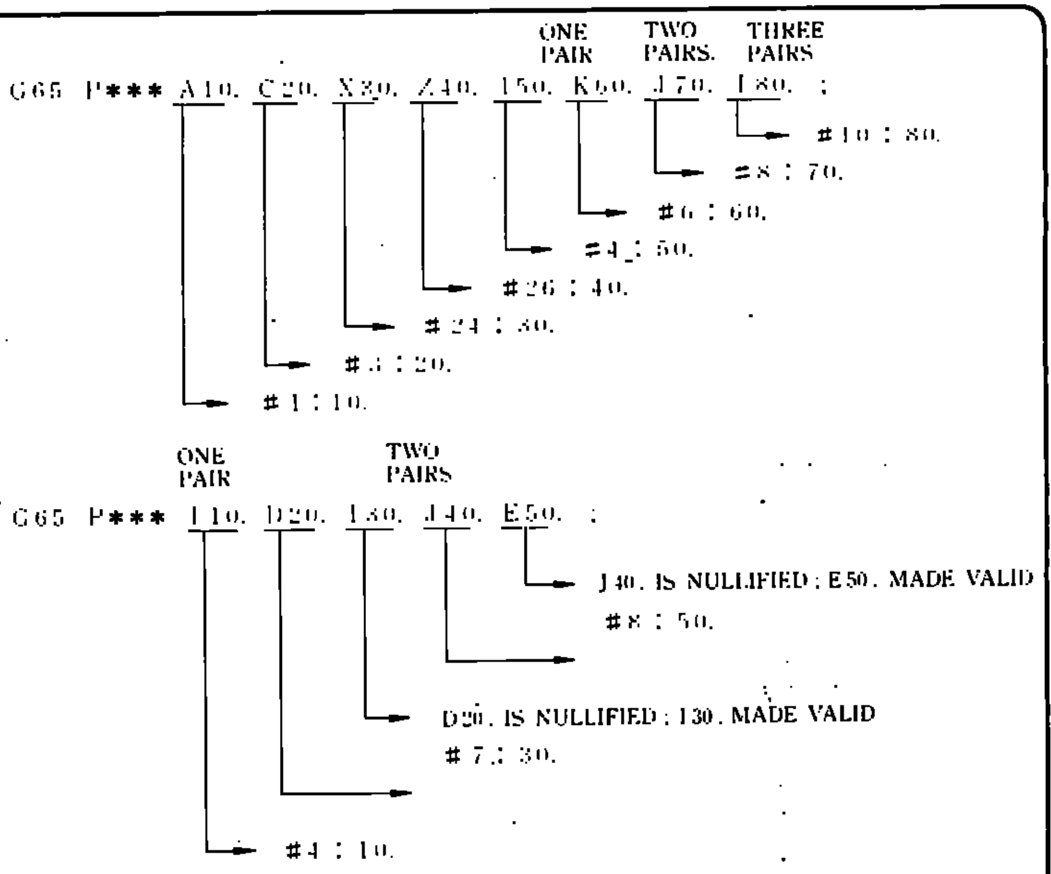
Table 2.14.10

RELATIONSHIP BETWEEN ADDRESS AND VARIABLE		CALLING COMMAND AND USABLE ADDRESS
ADDRESS FOR ARGUMENT SPECIFICATION-II	LOCAL NUMBER	G65, G66
A	# 1	○
B	# 2	○
C	# 3	○
I^1	# 4	○
J^1	# 5	○
K^1	# 6	○
I^2	# 7	○
J^2	# 8	○
K^2	# 9	○
I^3	#10	○
J^3	#11	○
K^3	#12	○
I^4	#13	○
J^4	#14	○
K^4	#15	○
I^5	#16	○
J^5	#17	○
K^5	#18	○
I^6	#19	○
J^6	#20	○
K^6	#21	○
I^7	#22	○
J^7	#23	○
K^7	#24	○
I^8	#25	○
J^8	#26	○
K^8	#27	○
I^9	#28	○
J^9	#29	○
K^9	#30	○
I^{10}	#31	○
J^{10}	#32	○
K^{10}	#33	○

See NOTE on next page.

NOTE

- ❶ ○ : Can be used
- ❷ If I, J, or K is commanded, be sure to command them in order of I, J, and K. In addition, for argument specification-Ⅱ, suffixes 1 – 10 to I, J, and K indicate the order of each pair of I, J, and K, and need not be written in actual commands.
- ❸ For addresses in which argument specification is not required, the commands may be omitted. In this case, local variables corresponding to addresses without command are "blank".
- ❹ If multiple addresses are instructed for one variable No., the one instructed later is valid.
- ❺ If multiple I, J, and K are commanded, the order of pairs is determined for each I/J/K pair, so that variable No's. are determined corresponding to that order.



(Example)

When argument specification is made, the code used to call a macroprogram must always be placed before the argument specification. The value of argument specification may include a sign and decimal point irrespective of the address. If no decimal point is used, the value is stored in variable as a decimal-point value according to normal number of address digits type.

Fig. 2.14.3 Example of Argument Specification

(3) Decimal point position of argument

Arguments are normally specified as a signed, decimal-point value. If no decimal point is used, the position of the decimal point becomes as shown in Table 2.14.12 below.

Fig. 2.14.11 Decimal Point Position of Argument

Address for argument specification	Input in mm	Input in inches
A, C	3	3
B (Second auxiliary function)	3	3
B (With second auxiliary function)	0	0
D, H	0	0
E, F	0 (1)	1 (2)
I, J, K	3	4
M, S, T	0	0
Q, R	3	4
U, V, W	3	4
X, Y, Z	3	4

NOTE

❶ The value indicates the position of the decimal point as reckoned from the lowest digit.

❷ Numbers in ( ) indicate the number of digits before the decimal point for addresses E and F when parameter Pm2004 D0 = 1

❸ The address of second auxiliary function can be specified other than B by setting the parameter pm4112.

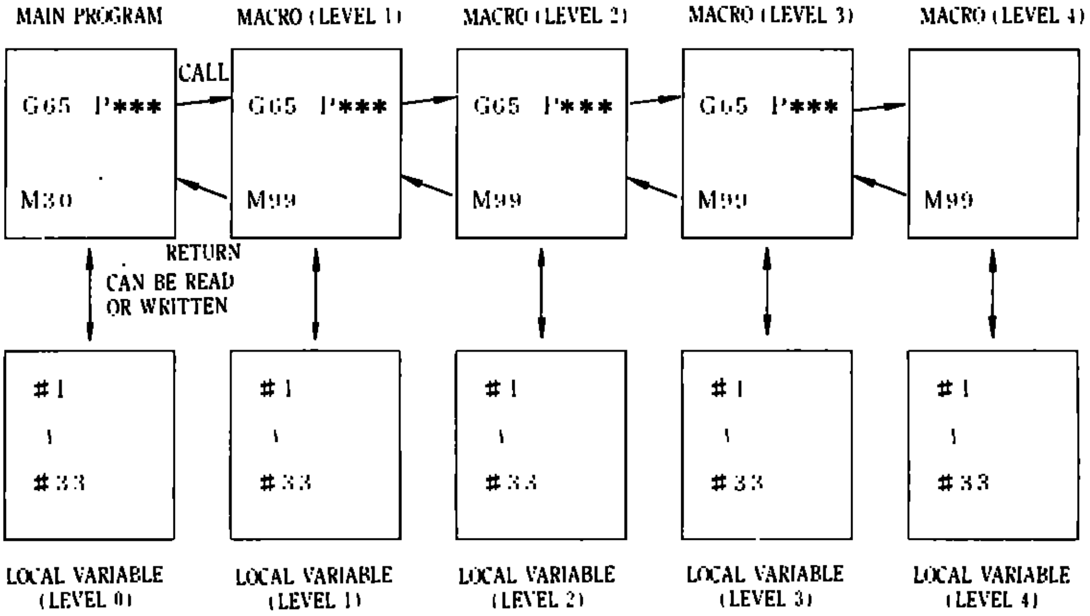
2.14.2 Variables

There are three types of variables: local variables, common variables, and system variables.

(1) Local variables (# 1 to # 33)

Local variables are used locally for each macro. This type of variable is such that each time a macro is called, new local variables(# 1 to # 33) are independently secured for that macro. In this case, when an argument is specified, the value of that argument is stored in the local variable; or the results of calculation in the macro are stored in the local variable. The contents of the local variable are empty if no argument is passed to the macro.

When a macro returns by instruction of M99, the local variable for that macro becomes blank. Also, When the equipment is turned on or reset, all local variables become blank.



NOTE

1 Local variable of level 0 is secured for the main program; local variables(levels 1 to 4) corresponding to the called macro level are secured for others.

2 When a macro is called by G65, for example, the local variable of the level of the calling side is saved, and a local variable of the level of the newly called side is secured. In this case, argument can be passed to the macro. Consequently, even for the same macro, if the called timing is different, the local variable of that macro level has different values.

3 When a macro returns to one level lower than that by instruction of M99, the local variable of that macro level is cleared to ⟨blank⟩. The local variable of the returned macro level becomes the local variable that was saved when called.

4 MDI write of local variable :

Do not write in the local variable by MDI during macro program execution Interrupt the operation by single-block stop and make sure that variable can be rewritten.

5 Local variables can be used in a subprogram, too. In this case, the local variables that can be used are those of the current macro level. However, when a subprogram is called, argument cannot be specified. When a subprogram returns by instruction of M99, the local variable of the current macro level is not ⟨blank⟩ either.

Fig. 2.14.4 Local Variable

(2) Common variables (# 100 to # 299, # 500 to # 999)
Common variables can be used in common by the main program, subprograms, and macros, or through a multiplexed state of all of these. That is, a common variable resulting from calculations made by a certain macro can be used in common by other macros. However no argument can be specified for common variables.

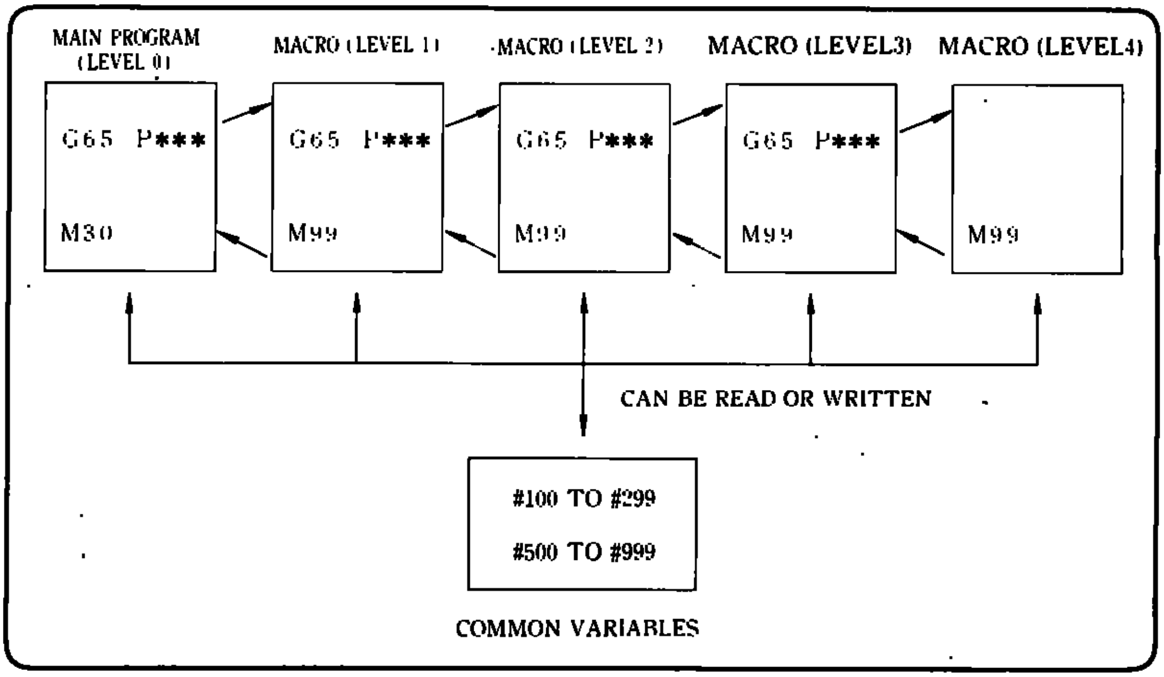


Fig. 2.14.5 Common Variable

① Common variables come in the following two types depending on the state when reset.

Table 2.14.12 Common Variable

# 100 to # 299	Become blank when the equipment is turned on or reset. However, variables cannot be made 'blank' when reset by specifying parameter # 4009 D1 = 1
# 500 to # 999	Saved without being blanked when the equipment is turned on or reset.

② A number of pairs can be added for common variables by using options.

Table 2.14.13

Optional Type	Number of Pairs
a	# 100 to # 149 (50 pairs) # 500 to # 559 (60 pairs)
b	# 100 to # 199 (100 pairs) # 500 to # 599 (100 pairs)
c	# 100 to # 199 (100 pairs) # 500 to # 699 (200 pairs)
d	# 100 to # 299 (200 pairs) # 500 to # 999 (500 pairs)

(3) System variables

These are predetermined variables according to the application, and come in the following types:

Table 2.14.14 System Variable S

Type of system variable	System variable No.
Interface input signal	# 1000 to # 1031, # 1032
Interface output signal	# 1100 to # 1131, # 1132
Tool offset quantity, work coordinate shift quantity	# 2001 to # 2299 # 12001 to # 13199
Alarm message display	# 3000
Clock	# 3001, # 3002
Single-block stop and auxilliary function complete wait control	# 3003
Feed hold, feedrate override, and exact stop control	# 3004
RS-232C data output	# 3100
Modal information	# 4000 to # 4999
Position information	# 5000 to # 5999

(a) Interface input signal :

- By instructing system variables # 1000 to # 1031 on the right side of an operation formula, it is possible to read the ON/OFF states of each of 32-point input signals exclusively used for a macroprogram. Table 2.14.16 shows the relationship between input signals and system variables.

Table 2.14.15 Interface Input Signal and System Variables

System Variables	# 1007	# 1006	# 1005	# 1004	# 1003	# 1002	# 1001	# 1000
Input Signal	UI 7 2^7	UI 6 2^6	UI 5 2^5	UI 4 2^4	UI 3 2^3	UI 2 2^2	UI 1 2^1	UI 0 2^0
System Variables	# 1015	# 1014	# 1013	# 1012	# 1011	# 1010	# 1009	# 1008
Input Signal	UI 15 2^{15}	UI 14 2^{14}	UI 13 2^{13}	UI 12 2^{12}	UI 11 2^{11}	UI 10 2^{10}	UI 9 2^9	UI 8 2^8
System Variables	# 1023	# 1022	# 1021	# 1020	# 1019	# 1018	# 1017	# 1016
Input Signal	UI 23 2^{23}	UI 22 2^{22}	UI 21 2^{21}	UI 20 2^{20}	UI 19 2^{19}	UI 18 2^{18}	UI 17 2^{17}	UI 16 2^{16}
System Variables	# 1031	# 1031	# 1029	# 1028	# 1027	# 1026	# 1025	# 1024
Input Signal	UI 31 2^{31}	UI 30 2^{30}	UI 29 2^{29}	UI 28 2^{28}	UI 27 2^{27}	UI 26 2^{26}	UI 25 2^{25}	UI 24 2^{24}

The value read by the above system variable becomes 1.0 or 0.0 depending on whether the corresponding input signal was ON or OFF.

Table 2.14.16

Input Signal	Variable Value
ON	1.0
OFF	0.0

- By instructing system variable #1032 on the right side of an operation formula, it is possible to read all of the above 32 input signals (U10 to U131) collectively as a positive decimal value.

$$\#1032=\sum_{i=0}^{31}\#\,[1000+i]\ \times 2^{i}$$

- It is impossible to place a value by instructing system variables #1000 to #1032 on the left side of an operation formula.

(b) Interface output signal

- By instructing system variables #1100 to #1131 on the left side of an operation formula, it is possible to output ON 'OFF signals to each of 32 point output signals exclusively used for a macroprogram. Table 2.14.17 shows the relationship between output signals and system variables:

Table 2.14.17 Interface Output Signal and System Variables

System Variables	# 1107	# 1106	# 1105	# 1104	# 1103	# 1102	# 1101	# 1100
Output Signal	UO 7 2^{7}	UO 6 2^{6}	UO 5 2^{5}	UO 4 2^{4}	UO 3 2^{3}	UO 2 2^{2}	UO 1 2^{1}	UO 0 2^{0}
System Variables	# 1115	# 1114	# 1113	# 1112	# 1111	# 1110	# 1109	# 1108
Output Signal	UO 15 2^{15}	UO 14 2^{14}	UO 13 2^{13}	UO 12 2^{12}	UO 11 2^{11}	UO 10 2^{10}	UO 9 2^{9}	UO 8 2^{8}
System Variables	# 1123	# 1122	# 1121	# 1120	# 1119	# 1118	# 1117	# 1116
Output Signal	UO 23 2^{23}	UO 22 2^{22}	UO 21 2^{21}	UO 20 2^{20}	UO 19 2^{19}	UO 18 2^{18}	UO 17 2^{17}	UO 16 2^{16}
System Variables	# 1131	# 1130	# 1129	# 1128	# 1127	# 1126	# 1125	# 1124
Output Signal	UO 31 2^{31}	UO 30 2^{30}	UO 29 2^{29}	UO 28 2^{28}	UO 27 2^{27}	UO 26 2^{26}	UO 25 2^{25}	UO 24 2^{24}

When the above system variables are substituted for by 1.0 or 0.0, the corresponding output signals are output in an ON or OFF state.

Table 2.14.18

Input Signal	Variable Value
ON	1.0
OFF	0.0

Note however that when variables # 1100 to # 1131are substituted for by any value other than 1.0 or 0.0, they are handled as follows:

‹Blank› and a value less than 0.5	0.0
Those other than the above	1.0

- By instructing system variable #1132 on the left hand side of an operation formula, it is possible to output all of the above 32 point output signals (UO0 to UO31) collectively. In this case, a positive decimal value placed into #1132 is output after being converted into a binary 32-bit value.

$$\#1132=\sum_{i=0}^{31} \# \left[1100+i\right] \times 2^{i}$$

- By instructing system variables # 1100 − # 1131 on the right side of an operation formula, it is possible to read the last output ON/OFF states of each (1.0, 0.0, positive decimal values).

(c) Offset quantity, work coordinate shift quantity

- By instructing system variables #12001 to #13199 on the right side of an operation formula,it is possible to read the tool offset quantity.
- By instructing system variables # 5200 to # 5327, # 7001 to # 7947, or # 17821 to # 19987 on the right side of an operation formula, it is possible to read the work coordinate system shift quantity (and external work coordinate system correction quantity).
- By instructing the above system variables on the left side of a formula, it is possible to change the values of those variables.

(Program example)

(A)

```
#116 = #12016
```

The contents of tool offset No. 16 are placed into common variable #116.

(B)

```
#5321 = #4
```

The work coordinate system shift quantity of G59 X-axis is erased, and the contents of local variable #4 are set.

(d) Tables 2.14.19 to 2.14.22 show the relationship between tool offset Nos. and work coordinate system shift quantites.

Table 2.14.19

	Basic			299-Pair option			999-Pair option		1199-Pair option	
	offset No.	System variable		offset No.	System variable		offset No.	System variable	offset No.	System variable
		i series	X3 series		i series	X3 series		i series		i series
H/D common type	H(D) 01	# 12001	# 2001	H(D) 01	# 12001	# 2001	H(D) 01	# 12001	H(D) 01	# 12001
	H(D) 02	# 12002	# 2002	H(D) 02	# 12002	# 2002	H(D) 02	# 12002	H(D) 02	# 12002
	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮
	H(D) 99	# 12099	# 2099	H(D)299	# 12299	# 2299	H(D) 999	# 12999	H(D)1199	# 13199
H/D separated type	H 01	# 12001	# 2001	H01	# 12001	# 2001	H01	# 12001	H01	# 12001
	H 02	# 12002	# 2002	H02	# 12002	# 2002	H02	# 12002	H02	# 12002
	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮
	H 49	# 12049	# 2049	H149	# 12149	# 2149	H499	# 12499	H599	# 12599
	D 01	# 12051	# 2051	D01	# 12151	# 2151	D01	# 12501	D01	# 12601
	D 02	# 12052	# 2052	D02	# 12152	# 2152	D02	# 12502	D02	# 12602
	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮
	D 49	# 12099	# 2099	D149	# 12299	# 2299	D999	# 12999	D1199	# 13199

NOTE # 2001 to # 2299system variables are provided to maintain compatibility with the X series.

12001 to # 13199system variables can only be used with the i series.

2001 to 12001, either can be used, however, use the system variables for " i "series if possible.

Table 2.14.20 Work Coordinate System Shift Quantity System Variables (6-pair Basic)

System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series
Shift quantity	X	# 5201	# 2500		X	# 5321	# 2506
	Y	# 5202	# 2600	G59	Y	# 5322	# 2606
	Z	# 5203	# 2700		Z	# 5323	# 2706
	4	# 5204	# 2800		4	# 5324	# 2806
	5	# 5205	# 2900	P 1	5	# 5325	# 2906
	R	# 5207	# 2950		R	# 5327	# 2956
	X	# 5221	# 2501				
	Y	# 5222	# 2601				
G54	Z	# 5223	# 2701				
J 1	4	# 5224	# 2801				
	5	# 5225	# 2901				
	R	# 5227	# 2951				
	X	# 5241	# 2502				
	Y	# 5242	# 2602				
G55	Z	# 5243	# 2702				
J 1	4	# 5244	# 2802				
	5	# 5245	# 2902				
	R	# 5247	# 2952				
	X	# 5261	# 2503				
	Y	# 5262	# 2603				
G56	Z	# 5263	# 2703				
J 1	4	# 5264	# 2803				
	5	# 5265	# 2903				
	R	# 5267	# 2953				
	X	# 5281	# 2504				
	Y	# 5282	# 2604				
G57	Z	# 5283	# 2704				
J 1	4	# 5284	# 2804				
	5	# 5285	# 2904				
	R	# 5287	# 2954				
	X	# 5301	# 2505				
	Y	# 5302	# 2605				
G58	Z	# 5303	# 2705				
J 1	4	# 5304	# 2805				
	5	# 5305	# 2905				
	R	# 5307	# 2955				

NOTE # 2500 to# 2996system variables are provided to maintain compatibility with the X series.
Since # 2500 is equal to # 5201, either can be used, however, use the system variables for " i " series if possible.

Table 2.14.21 Work Coordinate System Shift Quantity System Variables
〔54-pair Option (6-pair Basic + the following)〕

System Variable				System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series			i Series	X3 Series
	X	# 7001	# 2511		X	# 7121	# 2521		X	# 7241	# 2531
	Y	# 7002	# 2611		Y	# 7122	# 2621		Y	# 7242	# 2631
G54	Z	# 7003	# 2711	G54	Z	# 7123	# 2721	G54	Z	# 7243	# 2731
J 2	4	# 7004	# 2811	J 3	4	# 7124	# 2821	J 4	4	# 7244	# 2831
	5	# 7005	# 2911		5	# 7125	# 2921		5	# 7245	# 2931
	R	# 7007	# 2961		R	# 7127	# 2971		R	# 7247	# 2981
	X	# 7021	# 2512		X	# 7141	# 2522		X	# 7261	# 2532
	Y	# 7022	# 2612		Y	# 7142	# 2622		Y	# 7262	# 2632
G55	Z	# 7023	# 2712	G55	Z	# 7143	# 2722	G55	Z	# 7263	# 2732
J 2	4	# 7024	# 2812	J 3	4	# 7144	# 2822	J 4	4	# 7264	# 2832
	5	# 7025	# 2912		5	# 7145	# 2922		5	# 7265	# 2932
	R	# 7027	# 2962		R	# 7147	# 2972		R	# 7267	# 2982
	X	# 7041	# 2513		X	# 7161	# 2523		X	# 7281	# 2533
	Y	# 7042	# 2613		Y	# 7162	# 2623		Y	# 7282	# 2633
G56	Z	# 7043	# 2713	G56	Z	# 7163	# 2723	G54	Z	# 7283	# 2733
J 2	4	# 7044	# 2813	J 3	4	# 7164	# 2823	J 4	4	# 7284	# 2833
	5	# 7045	# 2913		5	# 7165	# 2923		5	# 7285	# 2933
	R	# 7047	# 2963		R	# 7167	# 2973		R	# 7287	# 2983
	X	# 7061	# 2514		X	# 7181	# 2524		X	# 7301	# 2534
	Y	# 7062	# 2614		Y	# 7182	# 2624		Y	# 7302	# 2634
G57	Z	# 7063	# 2714	G57	Z	# 7183	# 2724	G57	Z	# 7303	# 2734
J 2	4	# 7064	# 2814	J 3	4	# 7184	# 2824	J 4	4	# 7304	# 2834
	5	# 7065	# 2914		5	# 7185	# 2824		5	# 7305	# 2934
	R	# 7067	# 2964		R	# 7187	# 2974		R	# 7307	# 2984
	X	# 7081	# 2515		X	# 7201	# 2525		X	# 7321	# 2535
	Y	# 7082	# 2615		Y	# 7202	# 2625		Y	# 7322	# 2635
G58	Z	# 7083	# 2715	G58	Z	# 7203	# 2725	G58	Z	# 7323	# 2735
J 2	4	# 7084	# 2815	J 3	4	# 7204	# 2825	J 4	4	# 7324	# 2835
	5	# 7085	# 2915		5	# 7205	# 2825		5	# 7325	# 2935
	R	# 7087	# 2965		R	# 7207	# 2975		R	# 7327	# 2985
	X	# 7101	# 2516		X	# 7221	# 2526		X	# 7341	# 2536
	Y	# 7102	# 2616		Y	# 7222	# 2626		Y	# 7342	# 2636
G59	Z	# 7103	# 2716	G59	Z	# 7223	# 2726	G59	Z	# 7343	# 2736
J 2	4	# 7104	# 2816	J 3	4	# 7224	# 2826	J 4	4	# 7344	# 2836
	5	# 7105	# 2916		5	# 7225	# 2926		5	# 7345	# 2936
	R	# 7107	# 2966		R	# 7227	# 2976		R	# 7347	# 2986

Table 2.14.21 Work Coordinate System Shift Quantity System Variables 〔54-pair Option (6-pair Basic + the following)〕 (Cont'd)

System Variable				System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series			i Series	X3 Series
	X	# 7361	# 2541		X	# 7481			X	# 7601	
	Y	# 7362	# 2641	G54	Y	# 7482		G54	Y	# 7602	
G54	Z	# 7363	# 2741		Z	# 7483			Z	# 7603	
J 5	4	# 7364	# 2841		4	# 7484			4	# 7604	
	5	# 7365	# 2941	J 6	5	# 7485		J 7	5	# 7605	
	R	# 7367	# 2991		R	# 7487			R	# 7607	
	X	# 7381	# 2542		X	# 7501			X	# 7621	
	Y	# 7382	# 2642	G55	Y	# 7502		G55	Y	# 7622	
G55	Z	# 7383	# 2742		Z	# 7503			Z	# 7623	
J 5	4	# 7384	# 2842		4	# 7504			4	# 7624	
	5	# 7385	# 2942	J 6	5	# 7505		J 7	5	# 7625	
	R	# 7387	# 2992		R	# 7507			R	# 7627	
	X	# 7401	# 2543		X	# 7521			X	# 7641	
	Y	# 7402	# 2643	G56	Y	# 7522		G56	Y	# 7642	
G56	Z	# 7403	# 2743		Z	# 7523			Z	# 7643	
J 5	4	# 7404	# 2843		4	# 7524			4	# 7644	
	5	# 7405	# 2943	J 6	5	# 7525		J 7	5	# 7645	
	R	# 7407	# 2993		R	# 7527			R	# 7647	
	X	# 7421	# 2544		X	# 7541			X	# 7661	
	Y	# 7422	# 2644	G57	Y	# 7542		G57	Y	# 7662	
G57	Z	# 7423	# 2744		Z	# 7543			Z	# 7663	
J 5	4	# 7424	# 2844		4	# 7544			4	# 7664	
	5	# 7425	# 2944	J 6	5	# 7545		J 7	5	# 7665	
	R	# 7427	# 2994		R	# 7547			R	# 7667	
	X	# 7441	# 2545		X	# 7561			X	# 7681	
	Y	# 7442	# 2645	G58	Y	# 7562		G58	Y	# 7682	
G58	Z	# 7443	# 2745		Z	# 7563			Z	# 7683	
J 5	4	# 7444	# 2845		4	# 7564			4	# 7684	
	5	# 7445	# 2945	J 6	5	# 7565		J 7	5	# 7685	
	R	# 7447	# 2995		R	# 7567			R	# 7687	
	X	# 7461	# 2546		X	# 7581			X	# 7701	
	Y	# 7462	# 2646	G59	Y	# 7582		G59	Y	# 7702	
G59	Z	# 7463	# 2746		Z	# 7583			Z	# 7703	
J 5	4	# 7364	# 2846		4	# 7584			4	# 7704	
	5	# 7465	# 2946	J 6	5	# 7585		J 7	5	# 7705	
	R	# 7467	# 2996		R	# 7587			R	# 7707	

Table 2.14.21 Work Coordinate System Shift Quantity System Variables
〔54-pair Option (6-pair Basic+the following)〕 (Cont'd)

System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series
G54 J 8	X	# 7721		G54 J 9	X	# 7841	
	Y	# 7722			Y	# 7842	
	Z	# 7723			Z	# 7843	
	4	# 7724			4	# 7844	
	5	# 7725			5	# 7845	
	R	# 7727			R	# 7847	
G55 J 8	X	# 7741		G55 J 9	X	# 7861	
	Y	# 7742			Y	# 7862	
	Z	# 7743			Z	# 7863	
	4	# 7744			4	# 7864	
	5	# 7745			5	# 7865	
	R	# 7747			R	# 7867	
G56 J 8	X	# 7761		G56 J 9	X	# 7881	
	Y	# 7762			Y	# 7882	
	Z	# 7763			Z	# 7883	
	4	# 7764			4	# 7884	
	5	# 7765			5	# 7885	
	R	# 7767			R	# 7887	
G57 J 8	X	# 7781		G57 J 9	X	# 7901	
	Y	# 7782			Y	# 7902	
	Z	# 7783			Z	# 7903	
	4	# 7784			4	# 7904	
	5	# 7785			5	# 7905	
	R	# 7787			R	# 7907	
G58 J 8	X	# 7801		G58 J 9	X	# 7921	
	Y	# 7802			Y	# 7922	
	Z	# 7803			Z	# 7923	
	4	# 7804			4	# 7924	
	5	# 7805			5	# 7925	
	R	# 7807			R	# 7927	
G59 J 8	X	# 7821		G59 J 9	X	# 7941	
	Y	# 7822			Y	# 7942	
	Z	# 7823			Z	# 7943	
	4	# 7824			4	# 7944	
	5	# 7825			5	# 7945	
	R	# 7827			R	# 7947	

Table 2.14.22 Work Coordinate System Shift Quantity System Variables
〔162-pair Option (54-pair Basic + the following)〕

System Variable				System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series			i Series	X3 Series
G54 J10	X	# 17821		G54 J11	X	# 17941		G54 J12	X	# 18061	
	Y	# 17822			Y	# 17942			Y	# 18062	
	Z	# 17823			Z	# 17943			Z	# 18063	
	4	# 17824			4	# 17944			4	# 18064	
	5	# 17825			5	# 17945			5	# 18065	
	R	# 17827			R	# 17947			R	# 18067	
G55 J10	X	# 17841		G55 J11	X	# 17961		G55 J12	X	# 18081	
	Y	# 17842			Y	# 17962			Y	# 18082	
	Z	# 17843			Z	# 17963			Z	# 18083	
	4	# 17844			4	# 17964			4	# 18084	
	5	# 17845			5	# 17965			5	# 18085	
	R	# 17847			R	# 17967			R	# 18087	
G56 J10	X	# 17861		G56 J11	X	# 17981		G56 J12	X	# 18101	
	Y	# 17862			Y	# 17982			Y	# 18102	
	Z	# 17863			Z	# 17983			Z	# 18103	
	4	# 17864			4	# 17984			4	# 18104	
	5	# 17865			5	# 17985			5	# 18105	
	R	# 17867			R	# 17987			R	# 18107	
G57 J10	X	# 17881		G57 J11	X	# 18001		G57 J12	X	# 18121	
	Y	# 17882			Y	# 18002			Y	# 18122	
	Z	# 17883			Z	# 18003			Z	# 18123	
	4	# 17884			4	# 18004			4	# 18124	
	5	# 17885			5	# 18005			5	# 18125	
	R	# 17887			R	# 18007			R	# 18127	
G58 J10	X	# 17901		G58 J11	X	# 18021		G58 J12	X	# 18141	
	Y	# 17902			Y	# 18022			Y	# 18142	
	Z	# 17903			Z	# 18023			Z	# 18143	
	4	# 17904			4	# 18024			4	# 18144	
	5	# 17905			5	# 18025			5	# 18145	
	R	# 17907			R	# 18027			R	# 18147	
G59 J10	X	# 17921		G59 J11	X	# 18041		G59 J12	X	# 18161	
	Y	# 17922			Y	# 18042			Y	# 18162	
	Z	# 17923			Z	# 18043			Z	# 18163	
	4	# 17924			4	# 18044			4	# 18164	
	5	# 17925			5	# 18045			5	# 18165	
	R	# 17927			R	# 18047			R	# 18167	

Table 2.14.22 Work Coordinate System Shift Quantity System Variables
〔162-pair Option (54-pair Basic + the following)〕 (Cont'd)

System Variable				System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series			i Series	X3 Series
	X	# 18181			X	# 18301			X	# 18421	
	Y	# 18182			Y	# 18302			Y	# 18422	
G54	Z	# 18183		G54	Z	# 18303		G54	Z	# 18423	
J13	4	# 18184		J14	4	# 18304		J15	4	# 18424	
	5	# 18185			5	# 18305			5	# 18425	
	R	# 18187			R	# 18307			R	# 18427	
	X	# 18201			X	# 18321			X	# 18441	
	Y	# 18202			Y	# 18322			Y	# 18442	
G55	Z	# 18203		G55	Z	# 18323		G55	Z	# 18443	
J13	4	# 18204		J14	4	# 18324		J15	4	# 18444	
	5	# 18205			5	# 18325			5	# 18445	
	R	# 18207			R	# 18327			R	# 18447	
	X	# 18221			X	# 18341			X	# 18461	
	Y	# 18222			Y	# 18342			Y	# 18462	
G56	Z	# 18223		G56	Z	# 18343		G56	Z	# 18463	
J13	4	# 18224		J14	4	# 18344		J15	4	# 18464	
	5	# 18225			5	# 18345			5	# 18465	
	R	# 18227			R	# 18347			R	# 18467	
	X	# 18241			X	# 18361			X	# 18481	
	Y	# 18242			Y	# 18362			Y	# 18482	
G57	Z	# 18243		G57	Z	# 18363		G57	Z	# 18483	
J13	4	# 18244		J14	4	# 18364		J15	4	# 18484	
	5	# 18245			5	# 18365			5	# 18485	
	R	# 18247			R	# 18367			R	# 18487	
	X	# 18261			X	# 18381			X	# 18501	
	Y	# 18262			Y	# 18382			Y	# 18502	
G58	Z	# 18263		G58	Z	# 18383		G58	Z	# 18503	
J13	4	# 18264		J14	4	# 18384		J15	4	# 18504	
	5	# 18265			5	# 18385			5	# 18505	
	R	# 18267			R	# 18387			R	# 18507	
	X	# 18281			X	# 18401			X	# 18521	
	Y	# 18282			Y	# 18402			Y	# 18522	
G59	Z	# 18283		G59	Z	# 18403		G59	Z	# 18523	
J13	4	# 18284		J14	4	# 18404		J15	4	# 18524	
	5	# 18285			5	# 18405			5	# 18525	
	R	# 18287			R	# 18407			R	# 18527	

Table 2.14.22 **Work Coordinate System Shift Quantity System Variables (162-pair Option (54-pair Basic + the following)) (Cont'd)**

System Variable				System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series			i Series	X3 Series
G54 J16	X	# 18541		G54 J17	X	# 18661		G54 J18	X	# 18781	
	Y	# 18542			Y	# 18662			Y	# 18782	
	Z	# 18543			Z	# 18663			Z	# 18783	
	4	# 18544			4	# 18664			4	# 18784	
	5	# 18545			5	# 18665			5	# 18785	
	R	# 18547			R	# 18667			R	# 18787	
G55 J16	X	# 18561		G55 J17	X	# 18681		G55 J18	X	# 18801	
	Y	# 18562			Y	# 18682			Y	# 18802	
	Z	# 18563			Z	# 18683			Z	# 18803	
	4	# 18564			4	# 18684			4	# 18804	
	5	# 18565			5	# 18685			5	# 18805	
	R	# 18567			R	# 18687			R	# 18807	
G56 J16	X	# 18581		G56 J17	X	# 18701		G56 J18	X	# 18821	
	Y	# 18582			Y	# 18702			Y	# 18822	
	Z	# 18583			Z	# 18703			Z	# 18823	
	4	# 18584			4	# 18704			4	# 18824	
	5	# 18585			5	# 18705			5	# 18825	
	R	# 18587			R	# 18707			R	# 18827	
G57 J16	X	# 18601		G57 J17	X	# 18721		G57 J18	X	# 18841	
	Y	# 18602			Y	# 18722			Y	# 18842	
	Z	# 18603			Z	# 18723			Z	# 18843	
	4	# 18604			4	# 18724			4	# 18844	
	5	# 18605			5	# 18725			5	# 18845	
	R	# 18607			R	# 18727			R	# 18847	
G58 J16	X	# 18621		G58 J17	X	# 18741		G58 J18	X	# 18861	
	Y	# 18622			Y	# 18742			Y	# 18862	
	Z	# 18623			Z	# 18743			Z	# 18863	
	4	# 18624			4	# 18744			4	# 18864	
	5	# 18625			5	# 18745			5	# 18865	
	R	# 18627			5	# 18747			5	# 18867	
G59 J16	X	# 18641		G59 J17	X	# 18761		G59 J18	X	# 18881	
	Y	# 18642			Y	# 18762			Y	# 18882	
	Z	# 18643			Z	# 18763			Z	# 18883	
	3	# 18644			4	# 18764			4	# 18884	
	5	# 18645			5	# 18765			5	# 18885	
	R	# 18647			R	# 18767			R	# 18887	

Table 2.14.22 Work Coordinate System Shift Quantity System Variables
〔162-pair Option (54-pair Basic + the following)〕 (Cont'd)

System Variable				System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series			i Series	X3 Series
G54 J19	X	# 18921		G54 J20	X	# 19041		G54 J21	X	# 19161	
	Y	# 18922			Y	# 19042			Y	# 19162	
	Z	# 18923			Z	# 19043			Z	# 19163	
	4	# 18924			4	# 19044			4	# 19164	
	5	# 18925			5	# 19045			5	# 19165	
	R	# 18927			R	# 19047			R	# 19167	
G55 J19	X	# 18941		G55 J20	X	# 19061		G55 J21	X	# 19181	
	Y	# 18942			Y	# 19062			Y	# 19182	
	Z	# 18943			Z	# 19063			Z	# 19183	
	4	# 18944			4	# 19064			4	# 19184	
	5	# 18945			5	# 19065			5	# 19185	
	R	# 18947			R	# 19067			R	# 19187	
G56 J19	X	# 18961		G56 J20	X	# 19081		G56 J21	X	# 19201	
	Y	# 18962			Y	# 19082			Y	# 19202	
	Z	# 18963			Z	# 19083			Z	# 19203	
	4	# 18964			4	# 19084			4	# 19204	
	5	# 18965			5	# 19085			5	# 19205	
	R	# 18967			R	# 19087			R	# 19207	
G57 J19	X	# 18981		G57 J20	X	# 19101		G57 J21	X	# 19221	
	Y	# 18982			Y	# 19102			Y	# 19222	
	Z	# 18983			Z	# 19103			Z	# 19223	
	4	# 18984			4	# 19104			4	# 19224	
	5	# 18985			5	# 19105			5	# 19225	
	R	# 18987			R	# 19107			R	# 19227	
G58 J19	X	# 19001		G58 J20	X	# 19121		G58 J21	X	# 19241	
	Y	# 19002			Y	# 19122			Y	# 19242	
	Z	# 19003			Z	# 19123			Z	# 19243	
	4	# 19004			4	# 19124			4	# 19244	
	5	# 19005			5	# 19125			5	# 19245	
	R	# 19007			R	# 19127			R	# 19247	
G59 J19	X	# 19021		G59 J20	X	# 19141		G59 J21	X	# 19261	
	Y	# 19022			Y	# 19142			Y	# 19262	
	Z	# 19023			Z	# 19143			Z	# 19263	
	4	# 19024			4	# 19144			4	# 19264	
	5	# 19025			5	# 19145			5	# 19265	
	R	# 19027			R	# 19147			R	# 19267	

Table 2.14.22 Work Coordinate System Shift Quantity System Variables
〔162-pair Option (54-pair Basic+the following)〕 (Cont'd)

System Variable				System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series			i Series	X3 Series
G54 J22	X	# 19281		G54 J23	X	# 19401		G54 J24	X	# 19521	
	Y	# 19282			Y	# 19402			Y	# 19522	
	Z	# 19283			Z	# 19403			Z	# 19523	
	4	# 19284			4	# 19404			4	# 19524	
	5	# 19285			5	# 19405			5	# 19525	
	R	# 19287			R	# 19407			R	# 19527	
G55 J22	X	# 19301		G55 J23	X	# 19421		G55 J24	X	# 19541	
	Y	# 19302			Y	# 19422			Y	# 19542	
	Z	# 19303			Z	# 19423			Z	# 19543	
	4	# 19304			4	# 19424			4	# 19544	
	5	# 19305			5	# 19425			5	# 19545	
	R	# 19307			R	# 19427			R	# 19547	
G56 J22	X	# 19321		G56 J23	X	# 19441		G56 J24	X	# 19561	
	Y	# 19322			Y	# 19442			Y	# 19562	
	Z	# 19323			Z	# 19443			Z	# 19563	
	4	# 19324			4	# 19444			4	# 19564	
	5	# 19325			5	# 19445			5	# 19565	
	R	# 19327			R	# 19447			R	# 19567	
G57 J22	X	# 19341		G57 J23	X	# 19461		G57 J24	X	# 19581	
	Y	# 19342			Y	# 19462			Y	# 19582	
	Z	# 19343			Z	# 19463			Z	# 19583	
	4	# 19344			4	# 19464			4	# 19584	
	5	# 19345			5	# 19465			5	# 19485	
	R	# 19347			R	# 19467			R	# 19587	
G58 J22	X	# 19361		G58 J23	X	# 19481		G58 J24	X	# 19601	
	Y	# 19362			Y	# 19482			Y	# 19602	
	Z	# 19363			Z	# 19483			Z	# 19603	
	4	# 19364			4	# 19484			4	# 19604	
	5	# 19365			5	# 19485			5	# 19605	
	R	# 19367			R	# 19487			R	# 19607	
G59 J22	X	# 19381		G59 J23	X	# 19501		G59 J24	X	# 19621	
	Y	# 19382			Y	# 19502			Y	# 19622	
	Z	# 19383			Z	# 19503			Z	# 19623	
	4	# 19384			4	# 19504			4	# 19624	
	5	# 19385			5	# 19505			5	# 19625	
	R	# 19387			R	# 19507			R	# 19627	

Table 2.14.22 Work Coordinate System Shift Quantity System Variables
(162-pair Option (54-pair Basic + the following)) (Cont'd)

System Variable				System Variable				System Variable			
		i Series	X3 Series			i Series	X3 Series			i Series	X3 Series
G54 J25	X	# 19641		G54 J26	X	# 19761		G54 J27	X	# 19881	
	Y	# 19642			Y	# 19762			Y	# 19882	
	Z	# 19643			Z	# 19763			Z	# 19883	
	4	# 19644			4	# 19764			4	# 19884	
	5	# 19645			5	# 19765			5	# 19885	
	R	# 19647			R	# 19767			R	# 19887	
G55 J25	X	# 19661		G55 J26	X	# 19781		G55 J27	X	# 19901	
	Y	# 19662			Y	# 19782			Y	# 19902	
	Z	# 19663			Z	# 19783			Z	# 19903	
	4	# 19664			4	# 19784			4	# 19904	
	5	# 19665			5	# 19785			5	# 19905	
	R	# 19667			R	# 19787			R	# 19907	
G56 J25	X	# 19681		G56 J26	X	# 19801		G56 J27	X	# 19921	
	Y	# 19682			Y	# 19802			Y	# 19922	
	Z	# 19683			Z	# 19803			Z	# 19923	
	4	# 19684			4	# 19804			4	# 19924	
	5	# 19685			5	# 19805			5	# 19925	
	R	# 19687			R	# 19807			R	# 19927	
G57 J25	X	# 19701		G57 J26	X	# 19821		G57 J27	X	# 19941	
	Y	# 19702			Y	# 19822			Y	# 19942	
	Z	# 19703			Z	# 19823			Z	# 19943	
	4	# 19704			4	# 19824			4	# 19944	
	5	# 19705			5	# 19825			5	# 19945	
	R	# 19707			R	# 19827			R	# 19947	
G58 J25	X	# 19721		G58 J26	X	# 19841		G58 J27	X	# 19961	
	Y	# 19722			Y	# 19842			Y	# 19962	
	Z	# 19723			Z	# 19843			Z	# 19963	
	4	# 19724			4	# 19844			4	# 19964	
	5	# 19724			5	# 19845			5	# 19965	
	R	# 19727			R	# 19847			R	# 19967	
G59 J25	X	# 19741		G59 J26	X	# 19861		G59 J27	X	# 19981	
	Y	# 19742			Y	# 19862			Y	# 19982	
	Z	# 19743			Z	# 19863			Z	# 19983	
	4	# 19744			4	# 19864			4	# 19984	
	5	# 19745			5	# 19865			5	# 19985	
	R	# 19747			R	# 19867			R	# 19987	

(e) Alarm message display

#3000 = ⟨alarm No.⟩ (⟨alarm message⟩) ;

⟨alarm No.⟩ : 4-digit alarm No. that is not used with the equipment. (5000 to 5999)

Data can be converted to variable value

⟨alarm message⟩: ASCII character stringing up to 32 characters (numbers and special characters)

By instructing the above, it is possible to place the equipment in an alarm state. However, alarm states can only be entered after the immediately preceding block has been executed.

(f) Clock

By instructing the system variable shown below on the right side of an operation formula, it is possible to read time. By instructing it on the left side of an operation formula, it is possible to preset time.

Table 2.14.23 Clock

Type	System variable	Unit	When powered on	Count condition
Clock 1	# 3001	1 ms	Preset to 0	Normal
Clock 2	# 3002	1 s	Same as when power is OFF	When STL signal ON

(g) Single-block stop and auxiliary function complete wait control.

By assigning a given value to system variable #3003, it is possible to nullify the single-block switch for the subsequent blocks or advance to the next block without waiting for auxiliary functions (M,S,T,B) complete signal (FIN) to be checked.

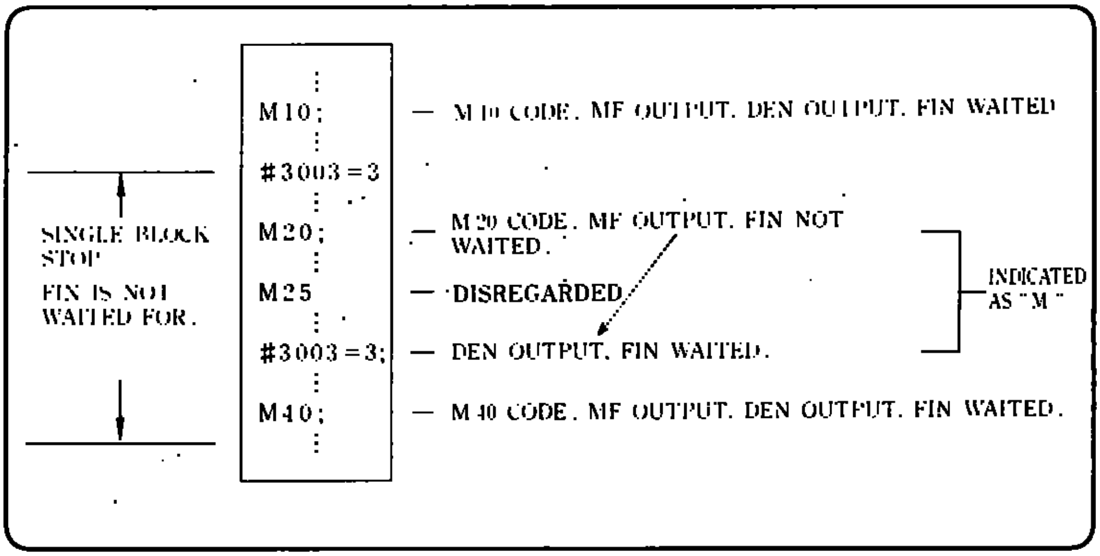
If an auxiliary function is commanded in a state where check for auxiliary function complete signal (FIN) is not waited for, distribution complete signal (DEN) is not output and FIN signal is not checked although M, S, T and B code output and M, S, T and B read output are made as usual. When a block in which a value is assigned for # 3003 to wait for check of auxiliary function complete signal as usual is executed after that, DEN is output and, at the same time, FIN is waited for.

When the auxiliary function complete wait signal is not waited for, commands M, S, T, and B are disregarded once after they are commanded separately, until the parameter setting is changed to wait for the complete signal.

In the above state, even if a processing stop M code (M00, M01, M02, or M30) is commanded, processing goes on to the next block without waiting for the complete signal.
Note that #3003 is cleared to 0 when reset.

Table 2.14.24 Single-Block Stop and Auxiliary Function Complete Wait Control

# 3003	Single-block switch	Auxiliary function complete signal (FIN)
0	Valid	Waited
1	Invalid	Waited
2	Valid	Not waited
3	Invalid	Not waited



(h) Feed hold, feedrate override, and exact stop control
By assigning a given value to system variable #3004, the control as shown in Table 2.14.25 is executed. For the subsequent blocks or not to perform exact stop check.
When reset, #3004 is cleared to 0.

Table 2.14.25

# 3004	Feed hold	Feed rate override	Exact stop check
0	Valid	Valid	Valid
1	Invalid	Valid	Valid
2	Valid	Invalid	Valid
3	Invalid	Invalid	Valid
4	Valid	Valid	Invalid
5	Invalid	Valid	Invalid
6	Valid	Invalid	Invalid
7	Invalid	Invalid	Invalid

- For feed hold

  Feed hold is nullified from a block in which #3004 = 1, 3, 5, 7 is commanded until #3004 = 0, 2, 4, 6 is commanded. If the feed hold button on the operator panel is depressed while a block in which feed hold is nullified is being executed, feed hold is not accepted and the feed hold lamp will not light.
- For feedrate override

  Feedrate override is nullified from a block in which #3004 = 2, 3, 6, 7 is commanded until #3004 = 0, 1, 4, 5 is commanded.
- For exact stop

  Exact stop is not checked from a block in which #3004 = 4, 5, 6, 7 is commanded until #3004 = 0, 1, 2, 3 is commanded.

〈Example of Programming〉

Tapping Cycle in Incremental command

(A) Main program

```
G91 ;
G65  P9601  L···  R···  Z··· ;
                    |     └─Argument Specification of #26
                    └─Argument Specification of #18
```

(B) Macroprogram

```
O9601 ;
#3003  = 1 ;  ─Invalid of single-block SW─┐
G00  Z#18 ;                    ┌ Feed hold │
#3004  = 7    ─Invalid of     ┤ Feedrate  │
G01  Z#26 ;                    │   overriole│
M05 ;                          └ Exact stop│
M04 ;                               ↑      │
#3004  = 0 ;  ──────────────────────┘      │
G00  Z  -#18                               │
M05 ;                                      │
M03 ;                                      │
#3003  = 0 ;  ─────────────────────────────┘
M99 ;
```

(i) RS-232C data output 1 (#3100)

By instructing system variable#3100, it is possible to output messages or variable data to external equipment via the RS-232C data I/O interface.

- #3100 = (〈message〉) ;

When this command is instructed, a message enclosed by control-out and control-in is output. CR and LF (carriage return, line feed) codes are automatically appended to the last of the message.

〈message〉 : ASCII character string of up to 256 characters (alphanumeric and special characters)

#3100 = ( ) ;

When this command is instructed,only CR and LF (carriage return, line feed) codes are output.

- #3100 = [⟨variable⟩] ;

When this command is instructed, the value of⟨variable⟩is output as 9-digit signed decimal data (four digits before the decimal point, five digits after the decimal point).

⟨variable⟩ : local variable, common variable, system variable.

NOTE
1. The fifth digit before the decimal point in the above case is rounded off to the nearest whole number.
2. If the number of digits after the decimal point exceeds six digits, *(asterisk) is output.

- Special codes that can be used with the macroprogram proper.

❶ Special codes listed in Table 2.14.26 can be used.

❷ EIA code hole patterns of characters marked by ★ conform to Table 2.14.26 as standard. However, other hole patterns can be specified by using the following parameters:
When the values of these parameters are "0," the hole patterns in Table 2.14.26 are assumed.

Pm 4100	#
Pm 4101	[
Pm 4102	]
Pm 4103	*
Pm 4104	=
Pm 4105	(
Pm 4106	)
Pm 4107	,

pm4108	'
pm4109	"
pm4144	<
pm4145	>
pm4146	;

NOTE 1. For these parameters, read the desired hole pattern with a binary number, then set a value of it converted into a decimal number.

(Example)

8	7	6	5	4	○	3	2	1
○			○	○	○			

Set "152" for this hole pattern.

2. " ; " does not mean EOB but a semi-colon in the above table.
If a parameter in the above list is set to 0, the standard hole pattern listed in Table 2.14.26 is used.

Table 2.14.26 Special Codes

	Meaning of code	Application	EIA Code									ISO Code								
			8	7	6	5	4	○	3	2	1	8	7	6	5	4	○	3	2	1
	SP	Comments				○		○				○		○			○			
★	(	Alarm message and comments					○	○		○							○			
★	)			○			○	○		○							○			○
	+	Addition		○	○	○		○									○		○	○
	−	Subtraction		○				○									○	○		○
	:	Comments		○				○	○	○							○		○	
		Division			○	○		○			○	○					○	○	○	○
★	#	Variable		Parameter Specification								○					○		○	○
★	*	Multiplication	○			○	○	○				○					○		○	
★	=	Equal sign	○				○	○	○			○					○	○		○
★	[	Brackets	○		○	○		○				○	○				○		○	○
★	]		○		○			○		○		○	○				○	○		○
	$	Comments	○			○		○	○					○			○	○		
	@		○				○	○	○	○	○	○	○				○			
	?		○			○	○	○	○	○				○	○	○	○	○	○	○
	.	Decimal point		○	○		○	○		○	○			○		○	○	○	○	
★	,	Comma		○	○	○	○	○				○		○		○	○			

(j) Modal information

By instructing the following system variables on the right side of an operation formula, it is possible to read the modal commands instructed in blocks up to the immediately preceding one. These system variables cannot be instructed on the left side of an operation formula.

Table 2.14.27

Modal commands	Macro system variable
G code (group 01) to (group ＊)	# 4001 to # 4024
J (P) code (Expansion work coordinate system)	# 4080
B code	# 4102
D code	# 4107
E code	# 4108
F code	# 4109
H code	# 4111
Sequence No.	# 4114
Program No.	# 4115
S code (1)	# 4119
T code	# 4120

(k) Position information

By instructing the following system variables, it is possible to read various position information. These system variables cannot be instructed on the left side of an operation formula.
formula.

Table 2.14.28 Position Information

Position information	Macro system variable	Read during startup
X-axis block end position (ABSIO) to 5th axis block end position (ABSIO)	# 5001 to # 5005	Possible
X-axis machine coordinate system position (ABSMT) to 5th axis machine coordinate system position (ABSMT)	# 5021 to # 5025	Possible (*)
X-axis POS. ABS position (ABSOT) to 5th axis POS. ABS position (ABSOT)	# 5041 to # 5045	Possible (*)
X-axis skip signal position (ABSKP) to 5th axis skip signal position (ABSKP)	# 5061 to # 5065	Possible
Z-axis offset quantity	# 5803	Possible
X-axis servo position deviation quantity to 5th axis servo position deviation quantity	# 5101 to # 5105	Possible (*)

NOTE If system variables marked by * are instructed, completion of the immediately preceding block is waited for and read is performed after completion.

Table 2.14.29

Abreviation	ABSIO	ABSMT	ABSOT	ABSKP
Meaning	End position of immediately preceding block	Command current postion	Command current postion	Position at which skip signal turned on in G31 block
Coordinate system	Work coordinate system	Machine coordinate system	Work coordinate system	Work coordinate system
Tool offset quantity	Not included	–	Included	Included

NOTE The unit of input is mm or inch as specified:

Table 2.14.30

	Unit	
	Micron	Submicron
Input in mm	0.001	0.0001
Input in inch	0.001	0.00001
Input in deg	0.001	0.0001

NOTE ❶ If the skip signal in the G31 block does not turn on, the skip signal position is the end position of the G31 block.

❷ If the skip signal in the G31 block turns on, the end position of the G31 block becomes the skip signal position.

(1) NC parameter, keep memory

NC parameter or keep memory can be read.

Table 2.14.31

System Variable Type	System Variable No.
NC parameter	# 50000 to # 56949
Keep memory	# 57100 to # 57999

NOTE The number obtained by adding 50000 to the NC parameter No. or keep memory No. becomes system variable No.

To read parameter pm6000, see the following.

```
# 100 = # 56000
```

2.14.3 Expression of Variables

Variables are expressed by variable No. or alphanumeric following #.

(1) Method of specifying variable No. directly

#i (i = variable No.)

(Example) #1, #101, #501, #2001

(2) Method of specifying an expression as variable No.

〔〈expression〉〕

(Example) # 〔#100〕, # 〔#501 + 1〕, # 〔#1/2〕

2.14.4 Quoting Variables

A numeric value following the address can be substituted for with a variable.

〈address〉 #i or 〈address〉 − #i

By instructing the above, it is possible to assign the value of a variable or its negative number (complement) as the command value of that address.

(Example)

Command with variable	Equivalent command
G # 30, # 30 = 1.0	G01
X # 101, # 101 = 100.	X100.
Z # 103, # 103 = 300.	Z300.
F # 140, # 140 = 0.3	F0.3

In other words, G #30, X #101, Z #103, and F #140 ; commands are equivalent to G01, X100., Z300., F0.3 ; commands.

(1) Addresses ⁄, O, N cannot quote variables.

(Example)

#5	n(n= 1 to 9) in " n" cannot be replaced with a variable.
O # 100	O number (program No.) cannot be replaced with a variable.
N # 200	N number (sequence No.) cannot be replaced with a variable.

(2) No variable No. can be replaced with a variable.

(Example)

When 10 in "# 10" is to be replaced with #20, instruct it as # [#20]; # #20 cannot be instructed.

(3) When a variable is address data, those data become a value in which numbers less than the minimum setting unit of that address are rounded off to the nearest thousandth decimal.

(Example)

Variable for address data	Real setting data
X # 1 (# 1=45.2346)	X45.235 (input in 0.001mm)
F # 2 (# 2=0.255)	F0.26mm/rev (F32 format)
G04P # 3 (# 3=5.37672)	G04P5.377s
M # 4 (# 4=2.7236)	M03
G # 4 (# 4=2.7236)	G03

NOTE If the G code with decimal point function is on, the value becomes G02.7.

(4) A value following the address can be replaced with ⟨expression⟩ . By instructing

⟨address⟩ [⟨expression⟩] or ⟨address⟩ − [⟨expression⟩] ,

it is possible to assign the value of ⟨expression⟩ or its negative number (complement) as the command value of that address.

(5) Constants without the decimal point used in [ ] are handled as constants that have the decimal point at the end.

2.14.5 Undefined Variables

Variables which are not yet defined are called undefined variables. The variable values of undefined variables are ⟨blank⟩ .

The following falls under the category of undefined variables:

(1) Local variables and common variables (#100 – #299) when the equipment has power on or reset.

(2) Local variables for which no arguments are specified when a macroprogram is called.

(3) Local variables corresponding to the level of a macroprogram when the macroprogram returns by instruction of M99.

(4) Local variables and common variables in which no values have been stored in a macroprogram.

(5) Common variables in which no values have been written by MDI.

NOTE Variable #0 is used a variable (undefined variable) whose variable value is always ⟨blank⟩. Variable #0 cannot be instructed on the left side of an operation formula.

Meaning of ⟨blank⟩.

① When an underfined variable is quoted, the address itself is also disregarded.

#2= ⟨blank⟩	G00X #2 ; is equivalent to G00 ;.
#2=0	G00X #2 ; is equivalent to G00 X 0 ;.

② When used in an operation formula, undefined variables are handled as variable value 0 except when they are replaced with ⟨blank⟩.

● For #2= ⟨blank⟩

#3=#2 ;	#3= ⟨blank⟩
#3=# [#2+#2]	#3=#0= ⟨blank⟩
#3=#1 *#2 ;	#3=0
#3=#2+#2 ;	
#3=#2/#2 ;	
#3=5 *#2 ;	
#3=2−#2 ;	#3=2
#3=5/#2	Division error

③ If used in a conditional expression, undefined variables are handled as variable value 0 except for EQ and NE.

● For #3=0 and #2= ⟨blank⟩

#3EQ	#2 is not satisfied.
#3NE	#2 is satisfied.
#3GE	
#3LT	#2 is not satisfied.

2.14.6 Arithmetic Commands

By performing general arithmetic operations in which local variables, common variables, system variables, and constants are combined with operations or functions, it is possible to substitute the results of operation for a given variable. Using variables in these arithmetic operations means reading the required data from the internal variable data area, and substituting the results of operation means writing the operation result data into the internal variable data area. The write cycle is completed when the execution of one block is completed. The basic formula of arithmetic operation is $\boxed{\#i = \langle \text{expression} \rangle}$: The following operations and functions can be used:

(1) Definition or replacement of variable

#i=#j	Definition or replacement
#i=# [#j=#k]	Indirect specification

(2) Add calculations

#i=#j+#k	Sum
#i=#j−#k	Difference
#i=#jOR#k	Logical sum (for each bit in binary 32 bits)
#i=#jXOR#k	Exclusive logical sum (for each bit in binary 32 bits)

(3) Multiply calculations

#i=#j*#k	Product
#i=#j #k	Quotient
#i=#jAND#k	Logical product (for each bit in binary 32 bits)
#i=#jMOD#k	Residue (For #j and #k, the residue is obtained after being rounded to an integer. If #j is negative, #i also becomes negative.)

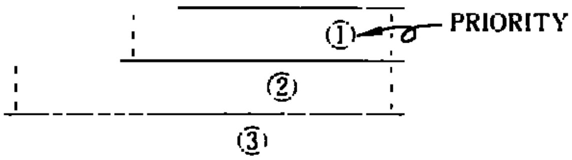
(4) Functions

#i=SIN [#j]	Sine (in units of degrees)
#i=COS [#j]	Cosine (in units of degrees)
#i=TAN [#j]	Tangent (in units of degrees)
#i=ATAN [#j]or #i=ATAN[#j/#k]	Reverse tangent
#i=SQRT [#j]	Square root
#i=ABS [#j]	Absolute value
#i=BIN [#j]	Conversion from BCD to binary
#i=BCD [#j]	Conversion from binary to BCD
#i=ROUND [#j]	Conversion into integer by rounding off
#i=FIX [#j]	Digits after the decimal point discarded
#i=FUP [#j]	Digits after the decimal point raised to a unit
#i=ASIN [#j]	Reverse sine
#i=ACOS [#j]	Reverse cosine
#i=LN [#j]	Natural logarithm
#i=EXP [#j]	Exponent based on e (=2.718)

(5) Combination of operations

Operations and functions in (1) to (4) above can be combined. In this case, the priority of operation is in order of the function, multiplication calculation, and addition calculation.

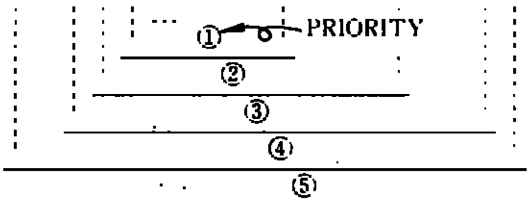
(Example) #i = #j + #k*SIN [#1]



(6) Altering the sequence of operations by [ ]

By enclosing a part with [ ], it is possible to perform the calculation of that part over the other parts. [ ] can be used for up to five duplications including the function [ ].

(Example)#i =SIN [[[#j+#k] *#1+#m] *#n]



NOTE

❶ Constants without the decimal point used in ⟨expression⟩ are assumed to have the decimal point at the end of the constant. The constant values that can be used in ⟨expression⟩ are ±$\underbrace{99999999}.\underbrace{9999999}$
(8 digits befor the decimal point) (7 digits after the decimal point)

❷ Although function ROUND converts numbers into integers by rounding them to the nearest whole number, the digits that are rounded off are as follows:

(a) If used in an operation command, conditional expression IF or WHILE, the digits after the decimal point are rounded off.

(b) If used in address data, the digits less than the minimum setting unit of that address are rounded off.

(Example 1) When #10 = 12.3758
#1 = ROUND [#10] becomes #1 = 12.0.
ROUND [#10] in IF [#10 GT ROUND [#10]] is 12.0.

(Example 2) When #10 = 12.3758
G00 X [ROUND [#10]] : (assuming that the minimum setting unit is 0.001 mm) is equivalent to G00 X12.376.

❸ The numeric data handled in macroprograms are in a floating point format.
$\underline{M*2^E}$ M: one sign bit + 52-bit binary data
E: one sign bit + 10-bit binary data

When single block input is on

When the single block input (SBK) is on, calculation commands are controlled by pm0007 D1 as follows: pm0007 D1 = 0: Do not stop in single-block.
1: Stop in single-block.

2.14.7 Control Commands

The following two commands control the macroprogram flow.

- Branch command

```
IF [<conditional expression>] GOTO <sequence No.> ;
```

- Repetition command

```
WHILE [<conditional expression>] DO <No.> ;
        |
END <No.> ;
DO  <No.> ;
        |
END <No.> ;
```

(1) Branch command

```
IF [<conditional expression>] GOTO <sequence No.> ;
```

When this command is issued, control jumps to the block of the specified sequence No. in the same program when ⟨conditional expression⟩ is satisfied.

If ⟨conditional expression⟩ is not satisfied, control advances to the next block.

⟨sequence No.⟩ : 5-digit integer, variable, or [⟨expression⟩]

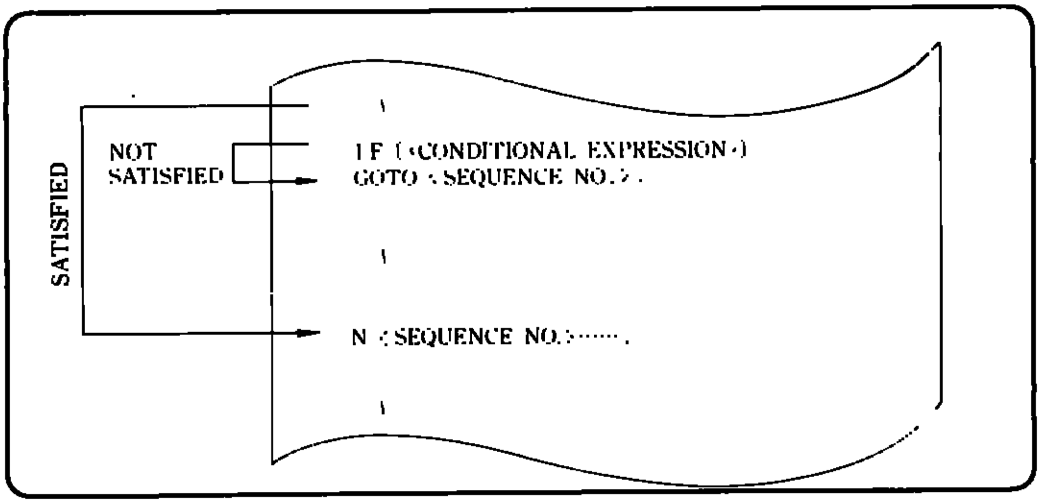


Fig. 2.14.6 Branch Command

When IF [⟨conditional expression⟩] is omitted, the command becomes a simple jump command.

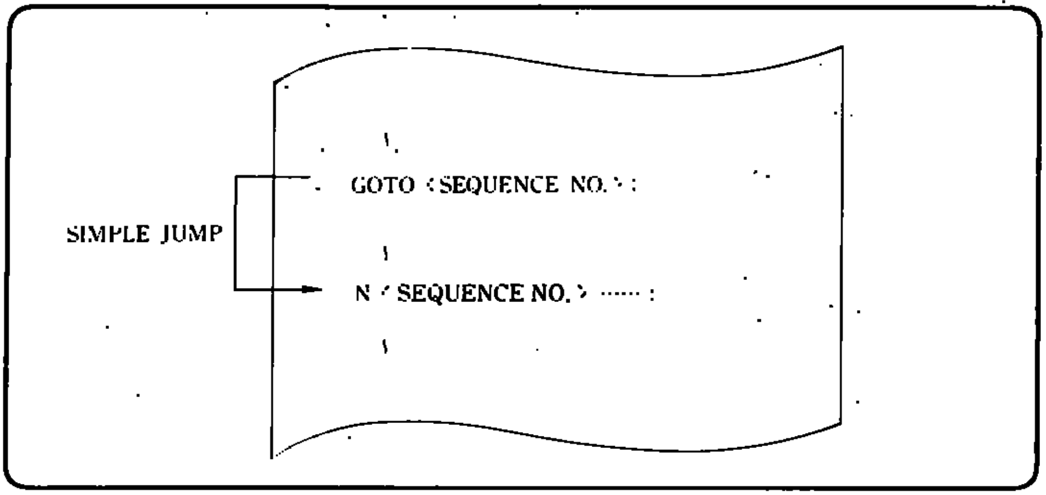


Fig. 2.14.7 Branch Command (Simple Jump Command)

NOTE ⟨sequence No.⟩ must as a rule be placed at the beginning of that block. Even if it is not positioned at the beginning of a block, data are executed from the beginnig of that block. If branch occurs, branching in the reverse direction of the program takes more execution time than branching in the forward direction.

In place of GOTO ⟨sequence No.⟩, an NC statement or macro statement can be instructed in one block. However, the following macro statements cannot be used because of limits on macro statements:

Control command, RS-232C data output2, status monitoring command

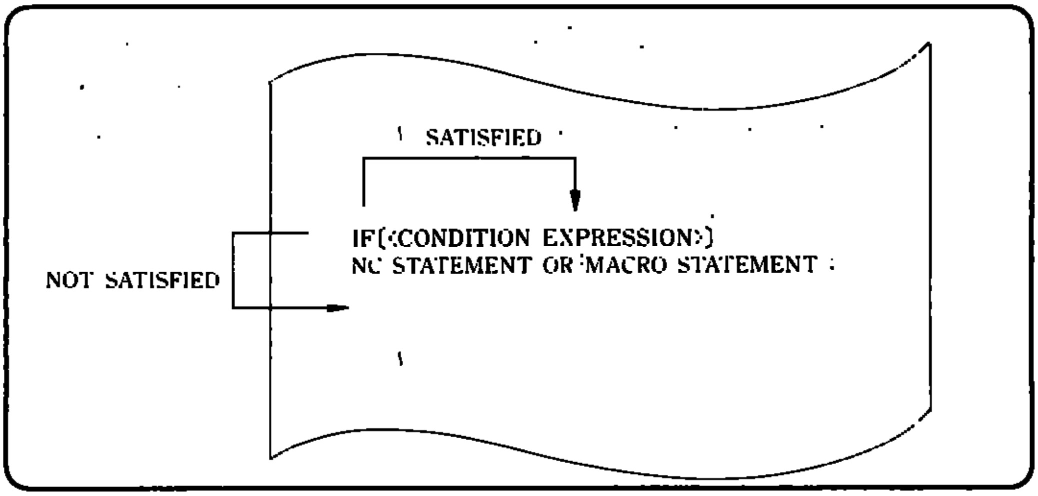


Fig. 2.14.8 NC Statement or Macro Statement Command

There are the following types of ⟨conditional expression⟩ :

Table 2.14.32

Conditional expression	Meaning
#iEQ#j	#i=#j
#iNE#j	#i≠#j
#iGT#j	#i>#j
#iLT#j	#i<#j
#iGE#j	#i≧#j
#iLE#j	#i≦#j
A OR B	Logical sum of A and B
A AND B	Logical product of A and B
A XOR B	Exclusive logical sum of A and B

NOTE In place of #i and #j, constants or ⟨expression⟩ can be used.

(2) Repetition command

```
WHILE [〈conditional expression〉] DO 〈No.〉 ;
      |
END 〈No.〉 ;
```

〈No.〉 = 1, 2, 3

When this command is issued, operation is repeated from a block next to DO to the block of END while〈conditional expression〉is satisfied. If〈conditional expression〉is not satisfied, control jumps to the block next to END.

When WHILE [〈conditional expression〉] is omitted, blocks from DO to END are infinitely repeated.

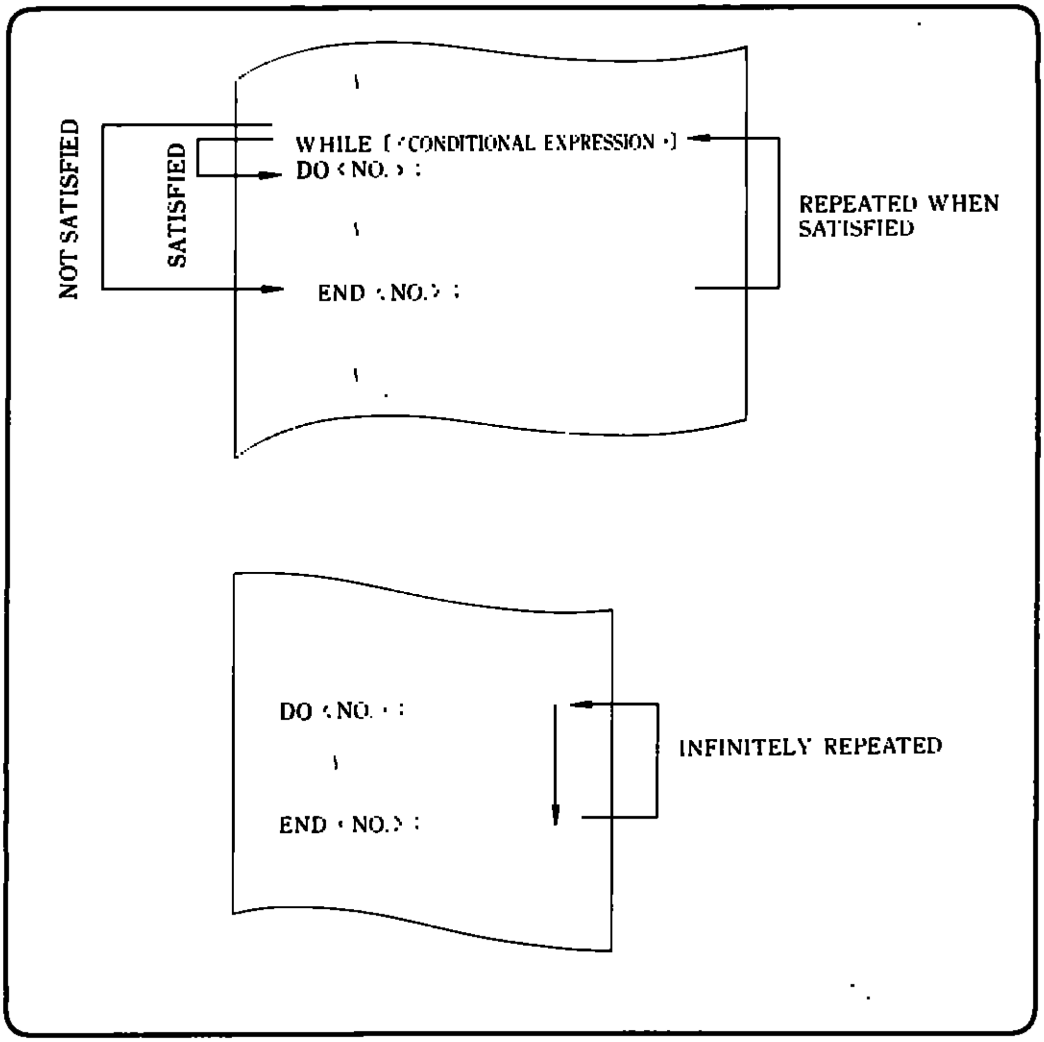


Fig. 2.14.9 Infinitely Repeated

(a) DO must precede END.

DO1	END 1
≀	≀
END 1	DO1
Correct	Wrong

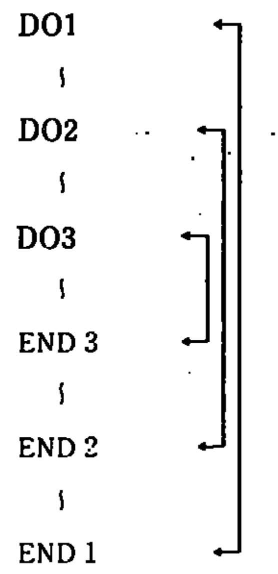
(b) 〈No.〉 in DO 〈No.〉 and END 〈No.〉 must be the same value, and must be specified in 1 to 1 pair.

DO1	DO1	DO1	DO1
≀	≀	≀	≀
END 1	END 2	DO1	END 1
⋮		≀	≀
DO2		END 1	END 1
≀			
END 2			
Correct	Wrong	Wrong	Wrong

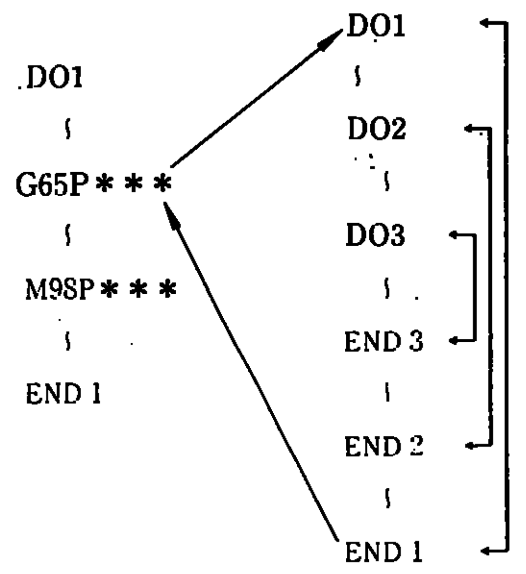
(c) The same 〈No.〉 can be used any number of times. However , the range of repetition must not overlap.

DO1	DO1	DO1	DO1
≀	≀	≀	≀
END 1	DO1	DO1	DO2
⋮	≀	≀	≀
DO1	END 1	END 1	END 1
≀	≀	≀	≀
END 1	END 1	END 1	END 2
Correct	Wrong	Wrong	Wrong

(d) Up to triple multiplexing in the DO to END loop is allowed in one macroprogram or subprogram.
It is also possible to call a macroprogram or subprogram within the range of DO to END. In this case, the DO to END loop in the called program can also be used in up to triple multiplexing.

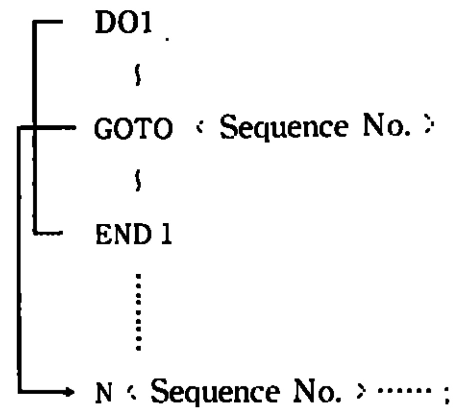


Correct

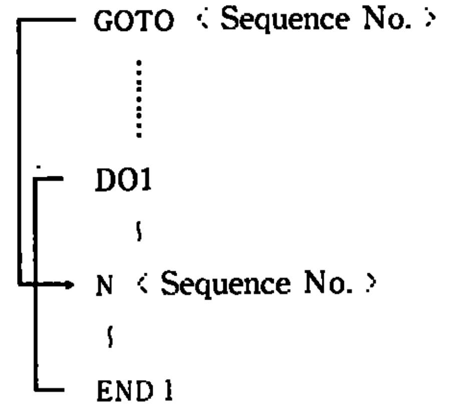


Correct

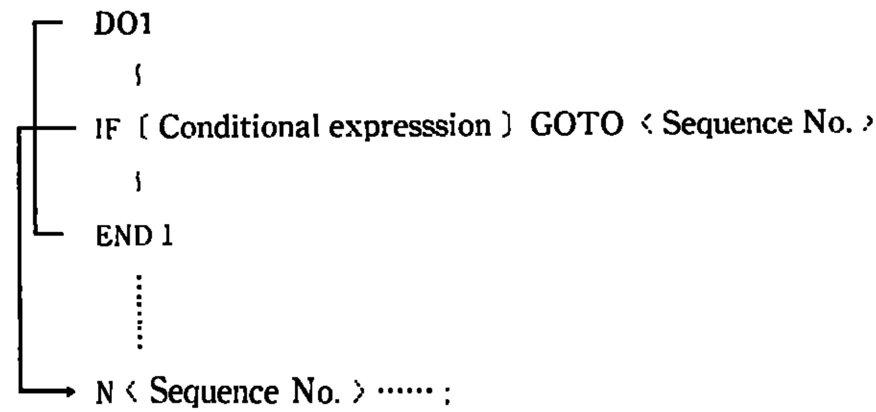
(e) It is possible to jump off from inside the DO to END loop by using GOTO ⟨sequence No.⟩. However, it is impossible to jump into the DO to END loop by using GOTO ⟨sequence No.⟩ .



Correct



Wrong



Correct

NOTE When the single block input (SBK) is on, control commands are controlled by pm0007 D7 as follows: pm0007 D1 = 0: Do not stop after executing a single block.
1: Stop after executing a single block.

2.14.8 Entering Macroprograms

(1) The method of operation to enter and edit macroprograms is entirely the same as the method used for normal NC machining programs and subprograms. There are no limits on the volume of macroprograms, and a total of NC machining programs and subprograms and macroprograms can be stored up to the limit of memory capacity.

(2) The macroprogram entry Nos. are subject to the following usage classification:

There are two cases: the O number of NC machining programs is classified and is not classified. Set it by parameter.

Table 2.14.33

Program No.	Usage classification	Protect
O1 to O7999	Can be freely entered, erased, and edited.	
O8000 to O8999	Edit and display protect can be independently applied by setting.	Protect 1
O900 to O9999	Edit and display protect can be collectively applied by using a parameter.	Protect 2
O10000 to O99999	Can be freely entered, erased, and edited.	

2.14.9 RS-232C Data Output-2 (BPRNT, DPRNT)

In addition to RS-232C data output-1 described in par. 2.14.2, the following macro commands can be executed:

Open command	POPEN
Data output command	BPRNT or DPRNT
Close command	PCLOS

These commands are used to output variables and characters through external equipment having the RS-232C interface.

(1) Open command (POPEN)

By instructing as

```
POPEN 〔a〕 ;
```

a — No. of RS-232C

DC2 control code is output from the NC equipment.

This must be instructed before a series of data output commands.

1 or 2 (optional) can be specified for the No. of the RS-232C; if omitted, the first RS-232C is selected.

(Example)

POPEN;	The first RS-232C is opened.
POPEN 〔2〕 ;	The second RS-232C is opened.

(2) Data output command (BPRNT or DPRNT)

(a) BPRNT

By instructing as

```
BPRNT 〔a#b 〔c〕 …〕 ;
```

c — Number of effective digits after the decimal point

b — Variable

a — Character

the following functions can be implemented:

- For characters, the specified character is output in the ISO code as is. The following characters can be specified:

- Alphabets (A to Z)
- Numbers
- Special characters (*, /, +, − , . (period)

NOTE However, '*' is output with space code.

- The value of the variable is handled as 2-word (32-bit) data by considering the digits after the decimal point, and are output as is beginning with the upper byte as binary data. All variables are stored in memory with the decimal point added. Therefore, instruct the number of effective digits after the decimal point by enclosing it with parentheses following the variable command.
- The EOB code is output with the ISO code after the command data are output.
- Variables which are 〈blank〉 are assumed to be 0.

(Example) BPRNT [POS ** X : 100 [2] Y : 101 [1]] ;

Variables : 100 = 1.2096

: 101 = 2.623

For the above example, data will be output as follows:

Output data

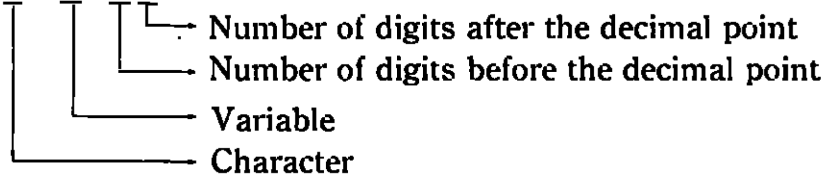
50	CF	53	A0	A0	D8	00 00 00 79	59	00 00 00 1A	OA
'P'	'O'	'S'	Space	Space	'X'	121	'Y'	26	EOB

Output is independent from the following parameters:

1. Pm0004 and Pm0006 parameters except D4 and D6, and Pm0009
2. To use the BPRNT function, set parameters to turn off the control code control and turn on the RTS control.
   Otherwise, output data are not guaranteed.

(b) DPRNT

```
DPRNT [a #b [c d] ……] ;
```



The following functions can be implemented by instructing as above.

- For characters, the specified character is output with the ISO code as is as in the case of (a) BPRNT command.
- The value of the variable is output for the specified number of digits with the ISO code digit-by-digit beginning with the highest-order digit.

  The decimal point is also output with the ISO code.

  To output the value of a variable, instruct variable No. following " #," then instruct the number of digits before the decimal point and the number of digits after the decimal point by enclosing them with parentheses.

  It is assumed that the value of a variable consists of up to eight digits (c + d ≦ 8).

  If the highest-order digits are 0, the parameter is processed as below.

Parameter	Pm4009 D2=0	Space code is output.
	Pm4009 D2=1	Nothing is output.

  If the instructed number of digits after the decimal point is other than 0, the value after the decimal point is always output for the number of digits present. If the instructed number of digits after the decimal point is 0, the decimal point is not output.

  For the + code when the sign is positive, the parameter is processed as below.

Parameter	Pm4009 D2=0	Space code is output.
	Pm4009 D2=1	Nothing is output.

- The EOB code is output with the ISO code after the command data are output.

2.14 MACROPROGRAM (G65, G66, G67)※(Cont'd)

- Variables which are 〈blank〉 are assumed to be 0.

(Example) DPRNT [POS X : 100 [43] Y # 101 [43]] ;

Variables : 100=12.479
: 100=1.568

For the above example, data will be output as follows:

- Output data

(1) When pm4009 D2=0

50	CF	53	D8	A0	A0	B1	B2	2E	B4	37	39	59
'P'	'O'	'S'	'X'	Space	Space	1	2	.	4	7	9	'Y'

A0	A0	A0	B1	2E	35	36	B8	OA
Space	Space	Space	1	.	5	6	8	EOB

(2) When pm4009 D2=1

50	CF	53	D8	B1	B2	2E	B4	37	39	59	B1	2E
'P'	'O'	'S'	'X'	1	2	.	4	7	9	'Y'	1	.

35	36	B8	OA
5	6	8	EOB

(3) Close command (PCLOS)

(Format) By instructing as $\boxed{\text{PCLOS 〔a〕 ;}}$

└ No.of RS-232C

DC4 control code is output from the NC equipment.

This must be instructed when all data output commands are finished.

The No. of RS-232C is the same as in the case of (1) POPEN.

(Example)

PCLOS;	The first RS-232C is closed.
PCLOS 〔2〕 ;	The second RS-232C is closed.

NOTE

1. Set the parameter so that the output interface is the RS-232C.
2. Set various RS-232C data (baud rate, etc.) by using parameters.
3. When outputting data with the DPRNT command, set whether or not the leading zeros should be output with space.

Pm4009 D2 = 0	The leading zeros when data are output by the DPRNT command are output with space.
Pm4009 D2 = 1	Nothing is output for the leading zeros.

4. Open command (POPEN) and close command (PCLOS) need not be instructed in succession.
   Once the open command is issued, there is no need to instruct the open command until the close command is issued after that.
5. The command being output is reset and stopped by the data output command, and the subsequent data are lost.
   Therefore, if reset by M30, for example, is applied at the end of the program that is outputting data, instruct the close command at the end of the program so that processing for M30 is not executed until all data are output.
6. The open command and close command must always be used in a pair. Do not instruct the close command when there is no open command instructed.

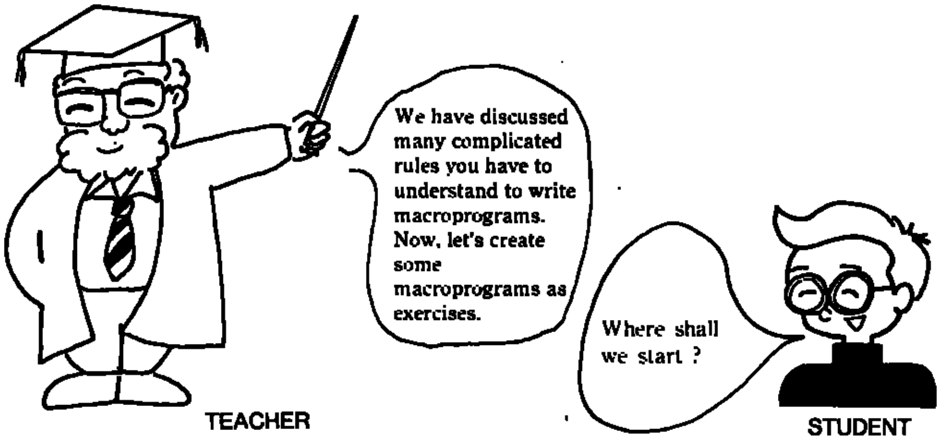
2.14.10 Alarm Nos. of Macroprogram

Table 2.14.34 lists the alarm Nos. and the causes of alarms related to macroprograms.

Table 2.14.34

Alarm No.	Contents	Alarm No.	Contents
Pm 0210	CONSTANT DATA OUT OF RANGE	Pm 0220	GOTO NO. FORMAT ERROR
	Macro constant overflow :		GOTO NO. format error :
	Constant in the macroprogram exceeds the limited range.		'n' in GOTO n exceeds the instructable range ; or 'n' cannot be found.
Pm 0211	UNMATCH G67 COMMAND	Pm 0221	O DIVIDE IN MACRO
	Macro command cancel error :		Macro O divide error :
	There are too many G67 cancel codes.		Division was made with O in a macro.
Pm 0212	MACRO FORMAT ERROR	Pm 0222	ROOT VALUE NEGATIVE
	Macro format error :		Square root negative number error :
	There is an error in the format		The number in $\sqrt{\ }$ is negative.
Pm 0213	UNEF INED # NO.	Pm 0223	FLOATING DATA OUT OF RANGE
	Undefined variable No. used :		Floating point data overfolow :
	A value not defined as variable No. is used. :		Floating point data exceeded the allowable range.
Pm 0214	ILL LEFT SIDE # NO.	Pm 0224	G66 M99 PROG ERROR
	Variable unusable on left :		G66 M99 command error :
	The variable in assign statement is a variable prohibited from assignment.		Axial movement is instructed when M99 returns in modal call (G66).
Pm 0215	[ ] LIMIT OVER	Pm 0225	MACRO SYSTEM ERROR
	[ ] Multiplexity over :		Macro system error :
	The multiplexity of [ ] exceeds the upper limit.		The operation stack overflowed.
Pm 0216	MACRO CALL LIMIT OVER	Pm 0226	ASIN, ACOS, LN, SQRT ERROR
	Macro call multiplexity over :		ASIN, ACOS, LN, SQRT error :
	The muliplexity of macro calls exceeds the upper limit.		Go out of range in ASIN, ACOS, LN, or SQRT function.
Pm 0217	DO-END FORMAT ERROR	Pm 0227	EXCHANGE OVERFLOW
	DO-EMD format error :		Overflow during integer conversion :
	DO-END is not corresponded 1 to 1.		An overflow occurred during conversion into integer.
Pm 0218	[ ] UNMATCH	Pm 0228	BCD INPUT DATA OVERFLOW
	[ ] command formet error :		BCD input data overflow :
	The number of [ ] do not corresponded to each other.		Input data in BCD function are overflowing.
Pm 0219	DO-END NO. OUT OF RANGE	Pm 0229	BIN FORMAT ERROR
	DO-END NO range error :		BIN format error :
	For DO m, not in the range of 1≦m≦3.		There is a format error in BIN function.
		Pm 0230	EXP OUTPUT DATA OVERFOLW
			EXP output data overflow :
			An overflow occurred in EXP function.

2.14.11 Macroprogram Examples



(1) Canned cycle (G82)

: An example of normal G82 command takes the following format:

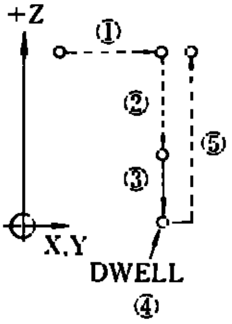
(P1)

```
G91 ; (incremental  command)
G82X100.  Y50.  R−80.  Z−40.  P3.0   F250 ;
```

……This command is divided into the following and executed within the NC unit:

(P2)

```
  G91 ;
①G00X100, Y50 ;
②G00Z−80. ;
③G01Z−40  F250 ;
④G04P3. ; (Dwell)
⑤G00Z−((−80) +(−40)) ;
```



First, these moving distances may all be converted into variables.

: You mean # 1 to # 33 local variables ? But there are type I and type Ⅱ local variables.

: When variables are few, type I in which X, Y, and Z are used is better, because argument specification is easy to understand.

: We will use it. When type I is used, the table of variables becomes like this, doesn't it?

(P3)

X100.	Y50.	R−80.	Z−40.	P3.0	F250
↓	↓	↓	↓	↓	↓
#24	#25	#18	#26	?	#9

Address P of dwell time cannot be used for argument specification, can it?

: No. We use some other addresses for it. Because dwell is a timer, we change address to U instead of P.

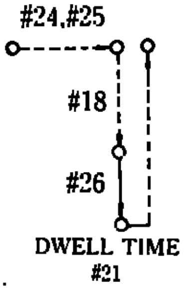
```
P3.0→U3.0
       ↓
      #21
```

Rewrite the previous program (P2) by using this variable.

: Yes, I will do it.

(P4)

```
 G91 ;
①G00X#24, Y#25 ;
②G00Z#18. ;
③G01Z#26  F#9 ;
④G04P#21 ;
⑤G00Z− [#18+#26] ;
```



Is this all right?

: Not good. You have forgotten to specify something in block ⑤, haven't you ?.

: Oh yes, "ROUND" is required.

```
⑤G00Z- [ROUND [#18] +ROUND [#26] ] ;
```

So I should change it like this ?

: Right. After that, add `⑥M99 ;`
Now, the macroprogram proper is completed.

: Completed already ? It's very simple.

: Now, create this macro call and the macroprogram proper in a complete form by using G65.

: Yes. Uhhhm... A macro call command is:

(P5)

```
G91 ;
G65P9082X100. Y50. R-80. Z-40.
        U3.0  F250. ;
```

The macroprogram proper is:

(P6)

```
O 9082 ;
G00X#24 Y#25 ;
G00Z#18 ;
G01Z#26 F#9 ;
G04P#21 ;
G00Z - [ROUND[#18] +ROUND[#26]];
M99 ;
```

I arbitrarily assigned program No. "O 9082" for the macroprogram.

: That looks OK.

* *

: Teacher, that is strange. With this program, point-R and point-Z must be instructed each time.

: That's right. Good question. In an ordinary canned cycle, once point-R and point-Z are instructed, those values are saved.

: Isn't there any good idea?

: Yes, there is. In such a case, "common variables" are useful. By using common variables, you first produce a macroprogram that specifies the positions of point-R and point-Z. By way of it, add U and F.

: I've got it ! Now, I divide the macroprogram proper into two parts as follows.

```
0   9000 ;
#100=#18 ;
#101=#26 ;
#102=#21 ;
#103=#9  ;
M99 ;
```

```
0   9082 ;
G00X#24  Y#25 ;
G00Z#100 ;
G01Z#101  F103 ;
G04P#102 ;
G01Z- [ROUND [#100] +ROUND [#101]] ;
M99 ;
```

For macro call, I make it as follows:

```
G91 ;
G65  P  9000  R-80.  Z-40.  U3.0  F250. ;
G65  P  9082  X 100.  Y50. ;
G65  P  9082  X……  Y…… ;
  ⋮
  ⋮
```

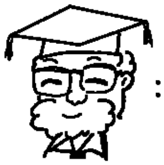
: Very good.

* *

: But this canned cycle always returns to the initial point.

: That's right. Initial point return (G98) and R-point return (G99) are invalid. That's a problem. To overcome this problem, you must check in which state G98 and G99 are now by using a system variable called "current value of model information command," then change the command for the return destination of the tool.

: Because G98 and G99 are G codes in group "10," I should use system variable #4010. Am I right?

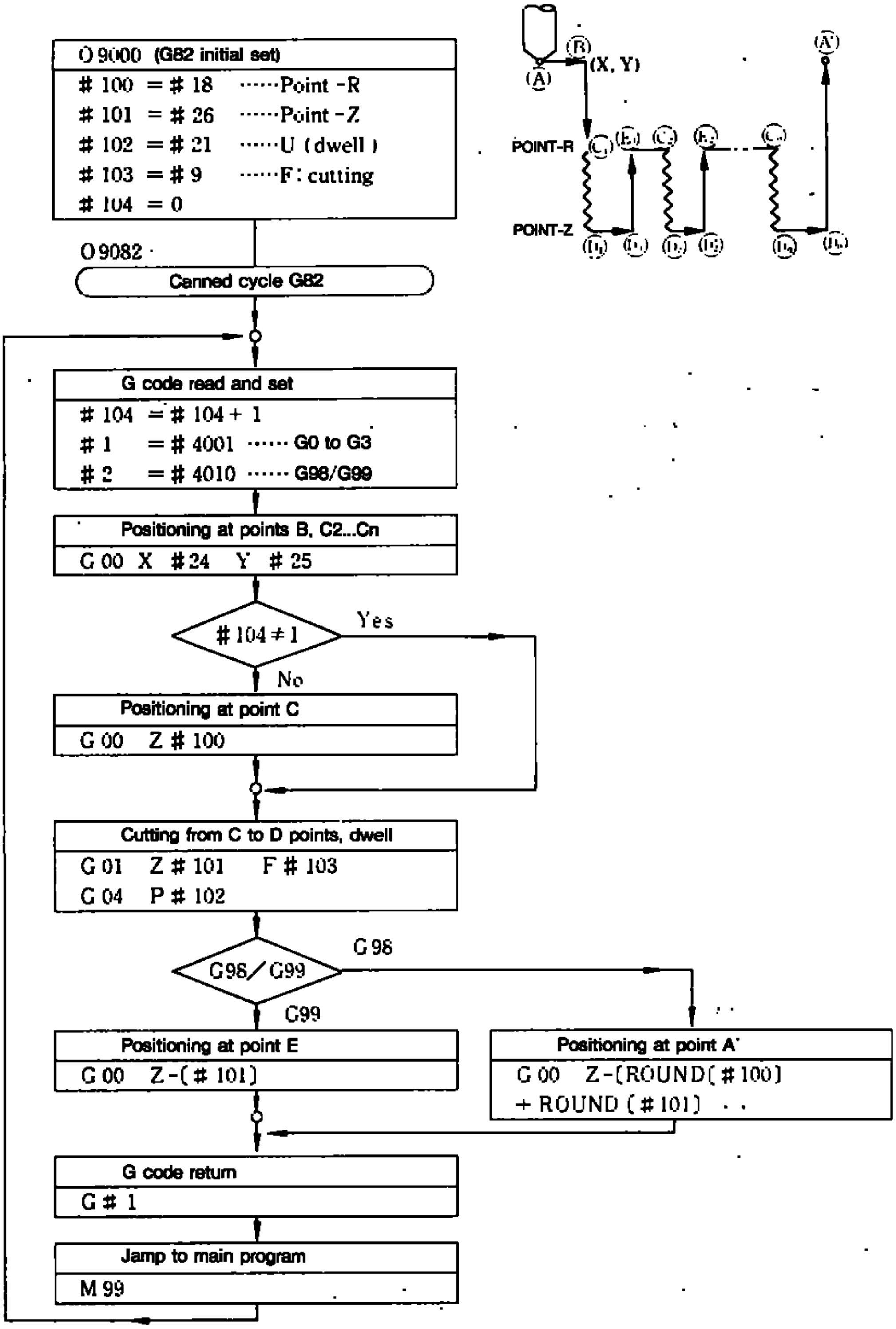


: Yes, you're right. And you change the command for the return destination of the tool by using command "IF... GOTO..." In that way, let us make one more improvement. That is to say, we make sure that even after execution of this macro, the G codes in group "01" before execution are saved. The G codes in group "01" include G00 to G03 and so on.

: This is rather difficult... Is the next program good?

```
G91  G99 ;
G65  P  9000  R−80.  Z−40.  U3.0  F250. ;
G65  P  9082  X−100.  Y50. ;
G65  P  9082  X……  Y…… ;
  ⋮
  ⋮
G98 ;
G65  P  9082  X……  Y…… ;
```

```
     O  9000 ;
     #100=#18 ;
     #101=#26 ;
     #102=#21 ;
     #103=#9  ;
     #104=0 ;
----------------------------------------------
     O  9082 ;
     #104=#104+1 ;
     #1  =#4001 ; ……G0 to G3
     #2  =#4010 ; ……G98/G99
     G00  X#24  Y#25 ;
     1F (#104  NE1) GOTO1 ;
     G00  Z#100 ;
N1   G01  Z#101  F#103;
     G04  P#102 ;
     1F (#2  EQ98) GOTO2 ;
     G00  Z− (#101)
     GOTO3 ;
N2   G00  Z− (ROUND (#100) +ROUND (#101)) ;
N3   G#1 ; …… G CODE RETURN

     M99 ;
```



```
O 9000 (G82 initial set)
# 100  = # 18   ······Point -R
# 101  = # 26   ······Point -Z
# 102  = # 21   ······U (dwell)
# 103  = # 9    ······F: cutting
# 104  = 0

G code read and set
# 104  = # 104 + 1
# 1    = # 4001 ······ G0 to G3
# 2    = # 4010 ······ G98/G99

Positioning at points B, C2...Cn
G 00  X  # 24   Y  # 25

# 104 ≠ 1

Positioning at point C
G 00   Z # 100

Cutting from C to D points, dwell
G 01   Z # 101    F # 103
G 04   P # 102

G98/G99

Positioning at point E
G 00   Z-(# 101)

Positioning at point A'
G 00   Z-[ROUND[#100]
+ ROUND (#101)

G code return
G # 1

Jamp to main program
M 99
```

: Very good.

(2) Pocket mill

: We will produce a macroprogram for a cycle that performs pocket machining as shown in Fig. 2.14.10.

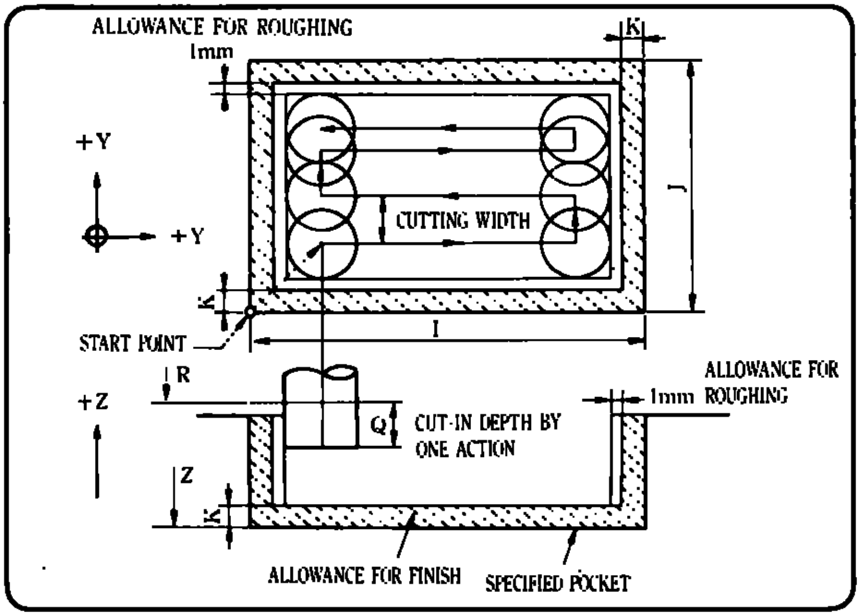


Fig. 2.14.10 Pocket

: Let's try the macro call instruction.

```
G65  P9061  X···Y···Z···R···I···J···K···
            T···Q···D···F···E··· ;
```

where

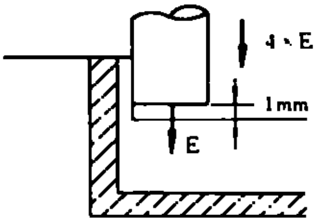
X, Y : Absolute coordinate values of start point
(left lowermost position of pocket)

Z : Absolute position of pocket bottom

R : Absolute position for rapid feedrate tool return

I, J : Length in X and Y-axis directions of pocket (unsigned)

K : Allowance for finish (remained allowance, unsigned)
If not specified, 0 is assumed.
T : Percentage of cutting width (specified by % value)
Cutting width = tool diameter × T/100
Q : Cut-in depth per action of Z-axis (unsigned)
D : Tool offset No.
F : Feedrate on X-Y plane
E : Feedrate during Z-axis cut-in
(Fed at a speed four times that of E up to a position 1mm before the previous cutting bottom.)




: The allowance for roughing (1mm width) inside the allowance for finish is shaved at a stroke in the last process as shown in Fig. 2.14.11 and the cycle is completed after return to the start point.

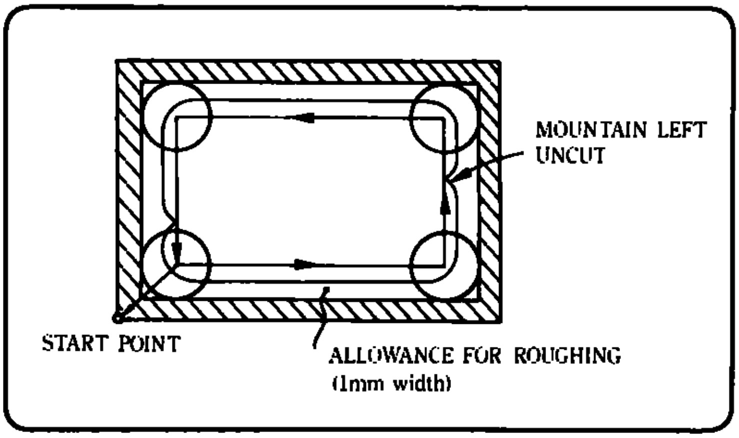


Fig. 2.14.11


: Next, I will explain the macroprogram proper.

```
  O  9061 ;
#10=# [2000+#7]  —— Tool radius
#11=#6+1.0+#10 ;
#12=#5-2*#10 ;
#13=2*#10*#20/100 —— Cutting width
#14=FUP [#12/#13]  —— Number of cuts in X-axis
                      direction minus 1
#27=#24+#11 ;}
#28=#25+#11 ;} X,Y coordinates of machining start point
#29=#26+#6 ; —— Z-axis coordinate of cutting bottom
#30=#24+#4-#11 ;
#15=#4003 —— Reading of G90/G91
G90 ; ———— Absolute command
G00  X#27  Y#28 ;
G00Z  #18 ;
#32=#18 ; —— #    : Cutting bottom under execution
DO1 ;
#32=#32-#17 ;
1F [#32  GT#29] GOTO1 ;
#32=#29 ;
N1  G01  Z#32  F#8 ;
G01X#30  F#9 ;
#33=1 ;
WHILE [#33  LE  #14] DO2 ;
1F [#33  EQ  #14] GOTO2 ;
G01  Y [#28+#33*#13] F#9 ;
GOTO3 ;
N2  G01  Y [#25+#5-#11] ;
N3  1F [#33  AND  1  EQ  0] GOTO4 ;    Cutting loop
G01  X  #27 ;
GOTO5 ;
N4  G01  X  #30 ;
N5  #33=#33+1 ;
END2 ;
```

```
G00  Z  #18 ;
1F 〔#32  LE#29〕 GOTO6 ;
G00  X  #27  Y  #28 ;
G01  Z 〔#32+1.0〕 F 〔4*#8〕 ;
END1 ;
N6  #11=#11-1.0 ;             ─┐
#27=#27-1.0 ;                  │
#28=#28-1.0 ;                  │
#30=#30-1.0 ;                  │
#31=#25+#5-#11                 │
G00  X  #27  Y  #28 ;          │  Roughing cycle
G01  Z  #32  F  #8 ;           │
G01  X  #30  Y  #9 ;           │
     Y  #31 ;                  │
     X  #27 ;                  │
     Y  #28 ;                 ─┘
G00  Z  #18 ;
G00  X  #24  Y  #25 ; ── Return to start point
G  #15 ; ──────── Restoration of G90/G91
M99 ;
```


: Thank you very much.

2.15 AUTOMATIC MEASURING FUNCTION

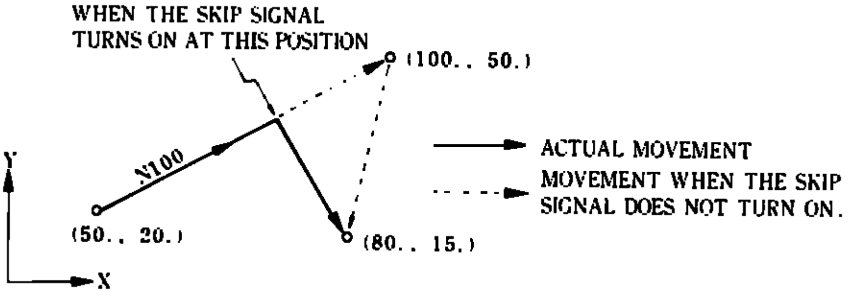
2.15.1 Skip Function (G31) ✻

(1) G31X······Y······Z······(※α······β······) F······ ;

When the skip signal input turns on during the movement of this interpolation, control advances to the next block by leaving the rest of the move command unexecuted. A delay time from when the skip signal turns on to when the equipment starts processing in response,is 0.5ms or less. Therefore, processing is performed at very high speed. G31 is a non-modal code.

Example of Programming

```
N100 G90 G31 X100. Y50. ;
N200 G01 X80. Y15. ;
```



NOTE Even when the G 31 block is executed, if the skip signal does not turn on, control stops at the end of that block and alarm " 491 " results.

Fig. 2.15.1 Skip Function

(2) The block feedrate for G31 can be set in the following two ways depending on parameter (Pm2001 D0):

① Specified by F as in the case of ordinary programs

② Preset by parameter (Pm2440)

(3) When the skip signal turns on, that coordinate value is automatically stored in memory as parameter data.

Pm 811	Coordinate value of X-axis is stored.
Pm 812	Coordinate value of Y-axis is stored.
Pm 813	Coordinate value of Z-axis is stored.
Pm 814	Coordinate value of the 4th axis is stored.
Pm 815	Coordinate value of the 5th axis is stored.

- These coordinate values are not the positions where feed was stopped when the skip signal turned on.
- These data can be handled as coordinate data in the macroprogram.

(4) If the skip signal does not turn on even when G31 is executed due to setting (Pm0007 D2), control can be automatically advanced to the next block.

pm0007 D2 = 0 : Issues an alarm when skip is not turned ON before G31 Block is completed.

pm0007 D2 = 1 : Does not issue alarm before G31 block is completed.

NOTE

1. Skip signal polarity specification

   Specify the polarity of the skip signal by parameter 5011 d_0 .

   Pm5011 D0 = 0 : Enables skip signal when 1 is changed to 0.(Negative logic)

   D0 = 1 : Enables skip signal when 0 is changed to 1.(Positive logic)

2. Execution of skip

   If the input skip signal does not match the skip signal polarity specification, operation is skipped from the beginning of the G31 block to the next block.

   If Pm5011 D0 = 0 (negative logic.) skips to the next block when the skip signal is 0 (low.)

   If Pm5011 D0 = 1 (positive logic.) skips to the next block when the skip signal is 1 (high.)

3. **G codes in which skip function can not be used**

   In the following states, skip function can not be used. If used, an alarm "061" occurs.

   - In canned cycle mode (G12, G13, G81 to G89 and G181 to G189)
   - In solid tap mode (G93)
   - In tool radius offset mode (G45 to G48)
   - In coordinate rotation mode (G68)
   - Commanded in the same block as the scaling (G51)

2.15.2 Program Interrupt function ON/OFF (M91, M90)*

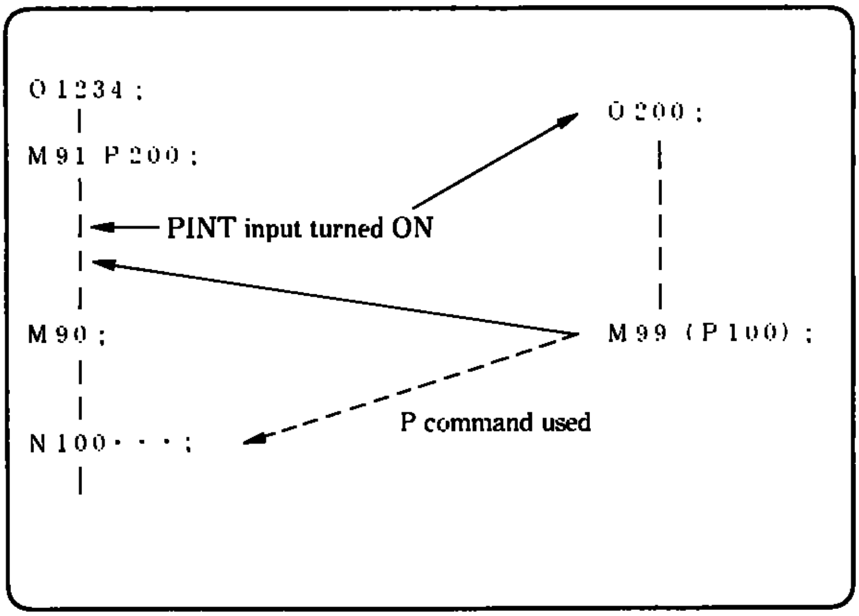
(1) The following M codes are used.

Table 2.15.1

M Code	Meaning
M90	Progam interrupt function OFF
M91	Progam interrupt function ON

(2) M91P…… ;

When this command is instructed and the program interrupt signal input turns on during program execution, the program currently being executed is interrupted (stopped after deceleration if interrupt occurs during movement) and control jumps to the program No. specified by P.



(3) M90 ; is a command for program interrupt function OFF.

NOTE

❶ A single block must be allocated to M90 or M91P. If a motion command is in the same block, ALM112 "M90/M91 FORMAT ERROR" occurs.

❷ Interrupt signal while interrupt program is executed

If an interrupt signal is input in the M91 Mode, it is disregarded while the interrupt program is being executed.

❸ ALM114 "M90/M91 COMMAND ERROR"

Neither M90 nor M91 can be used in a program started by programmed interrupt. If the command is used, ALM114 "M90/M91 COMMAND ERROR" occurs.

❹ Macro local variable level

M91 can be used in a sub-program or macro program.

An interrupt program is not counted for the program nesting level, therefore it does not vary the macro local variable level.

❺ Specifying destination

The M99 command in an interrupting program returns to the block next to the interrupted block. To specify returning to other block, use M99P.

When returning to the first program by M99, the modal infomation before interrupt is restored.

When returning to the first program by M99P, the modal information updated in the interrupting program is retained.

❻ When interrupt input occurs during block stop

If interrupt signal is input during block stop, interrupting program is started when operation is restarted by cycle start.

❼ Programmed interrupt input during high-speed cutting

Programmed interrupt is disregarded during high-speed cutting.

❽ Specifying interrupting program start sequence number

Use command Q in a M91P block to specify the sequence number at which the interrupting program is to be started.

❾ Modal information at interrupt

Modal information is retained even if a programmed interrupt occurs and the interrupting program is started. When the interrupting program is stopped and the first program is restarted by M99, the modal information at the time of interrupt is restored.

The current coordinate system of the first program is not restored.

When the interrupting program is stopped and the first program is restarted by M99P,the modal information in the interrupting program is retained.

- Disregarded during low speed G28

  Programmed interrupt is disregarded while low-speed return to reference point is being executed. High-speed return to reference point can be interruptd.

- Programmed interrupt in G31 (skip) block

  If programmed interrupt is input in a G31 (skip) block, the skip mode is canceled and programmed interrupt is performed.

- Programmed interrupt in M, S, T, or B command block

  If programmed interrupt is input while a block containing M, S, T, or B command is being executed, moving axes are slowed down and stopped, and the interrupting program is started. However, the block is not executed until the MT function complete signal is received.

- Program interrupt during solid tapping

  If programmed interrupt is input during solid tapping, the interrupting program is started after the tapping block is completed.

2.15.3 Tool Life Control ※

Tools are classified into groups and life is specified for each tool. Individual tools are identified in each group.

When a tool comes to the end of its life, the next predetermined tool in the same group is used.

(1) Life control specifications

(a) Life control object tool number

- Maximum number of object tools: 256
- Number of groups and tools

Table 2.15.2

	Number of groups	Number of tools
①	128	2
②	64	4
③	32	8
④	16	16

Select from ① to ④ by parameters pm0009 D6 and D7.

(b) Life control data

Name	Explanation
Tool Number(T)	Number assigned to each tool whose life is to be controlled
Tool Length Offset Number(H)	Length offset number assigned to a tool specified by the tool number
Tool Diameter Offset Number(D)	Diameter offset number assigned to a tool specified by the tool number
Life(LIFE)	Life of a tool specified by the tool number
Count of Operations (USED)	Count of operations of a tool specified by the tool number
Status(STS)	Life status of tool specified by the tool number : Over, skipped, or life controlled
Life type	Life control type of the group : Time, number of times of use, length of operation, or number of holes the tool made

(c) Life type

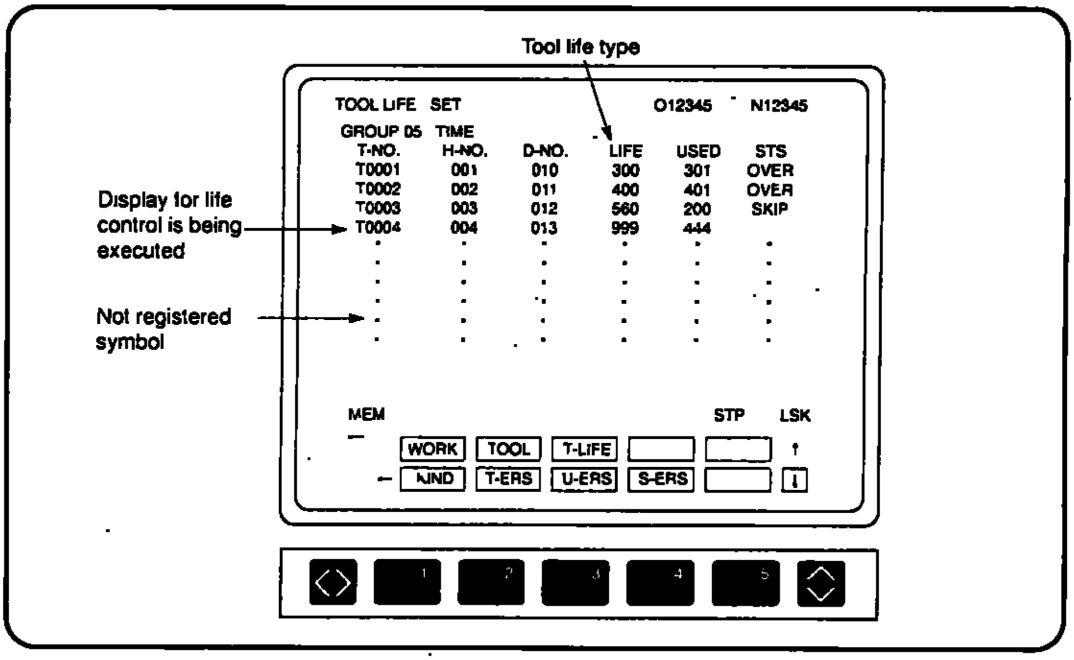
- Control by time (1-9999 minutes)
- Control by number of times of use (1-9999 times)
- Control by length of operation (1-9999 m or ft)
- Control by number of holes(1-9999)

Life of a tool is counted in the specified units while the life-controlled tool is serving in actual cutting.

(2) Setting of tool life control data

Tools are classified groups according to its type and tool life is specified for each group. The tool group can be specified on the part program and when the tool comes to the end of its life, it is replaced with another tool in the same tool group according to the preset tool selection order.

(3) Display of tool life control data



```
TOOL LIFE  SET                              O12345   N12345
 GROUP 05  TIME
  T-NO.    H-NO.    D-NO.    LIFE    USED    STS
  T0001     001      010      300     301    OVER
  T0002     002      011      400     401    OVER
  T0003     003      012      560     200    SKIP
  T0004     004      013      999     444
    .        .        .        .       .       .
    .        .        .        .       .       .
    .        .        .        .       .       .
    .        .        .        .       .       .
    .        .        .        .       .       .
    .        .        .        .       .       .

MEM                                             STP   LSK
```

Fig. 2.15.2 Tool Life Control Data Display Screen

(a) Display

The screen shown in Fig.2.15.2 can be displayed by the tool life control job of SET process.

The data of up to 10 tools in one group can be displayed on one page. The tool life control data display screens for the number of groups are provided and the group to be displayed can be selected or switched by the page key. Also, the screen of the group to be displayed can be called by the cursor △ or ▽ key after inputting the group No.

(b) Search of tool No.

By entering the tool No. started with T and pressing the cursor △ or ▽ key, the desired tool No. is searched in the data of all groups and the tool group is displayed.

NOTE When 16 tools are registered in one group, its screen is composed of two pages. In this case, use the cursor key to display the second page since the page key can switched the display only in group unit.

(4) Setting the tool life control data

(a) Setting by key input

By using the keys on NC operator panel, the tool life control data can be written-in or edited.

(i) Selection of group

The group can be selected by pressing the page key or pressing the cursor key after the group No. is entered.

(ii) Setting of tool Nos.

The tool Nos. for life control should be written-in.

Move the cursor to the item for the tool to be registered in group and enter the tool No. (1 to 9998), then press WR key.

After the tool No. is entered, the other data for the tool are initialized; 9999 in LIFE, 0 in H-NO, D-NO and USED, and blank is STS.

NOTE During execution of life control function, the tool is selected to be in use in the displayed order.

When the already registered tool No. is re-entered, a warning message REPETITION ERR ! is displayed.

(iii) Write-in of tool length and diameter offset numbers (H-NO and D-NO)

The length and diameter offset numbers for the tool should be written-in.

When H999 or D999 command is executed by the part program, the written-in offset number is used.

Move the cursor to the item for H-NO or D-NO and enter the offset number, then press WR key.

The setting range differs in option.

The default value of H-NO and D-NO is 0.

(iv) Write-in of life (LIFE)

The life of tool should be set.

Move the cursor to the item for LIFE and enter the tool life (1 to 9999), then press WR key.

(v) Write-in of number of times of use (USED)

The count of operations of tool should be set.

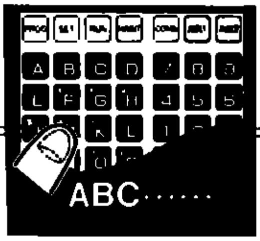
Move the cursor to the item USED and enter the count of operations (0 to 9999), then press WR key.

NOTE Enter "0" for the unused tool.

When the count of operations exceeds the set life value, "over" is displayed automatically in the item "STS".

(b) RS-232C input/output

The tool life control data can be input/output by following the instructions displayed on the I/O screen.



SECTION 3
DESCRIPTION OF OPERATION

This section describes the operation for NC operator panel with 9" CRT display, key functions and display operation.

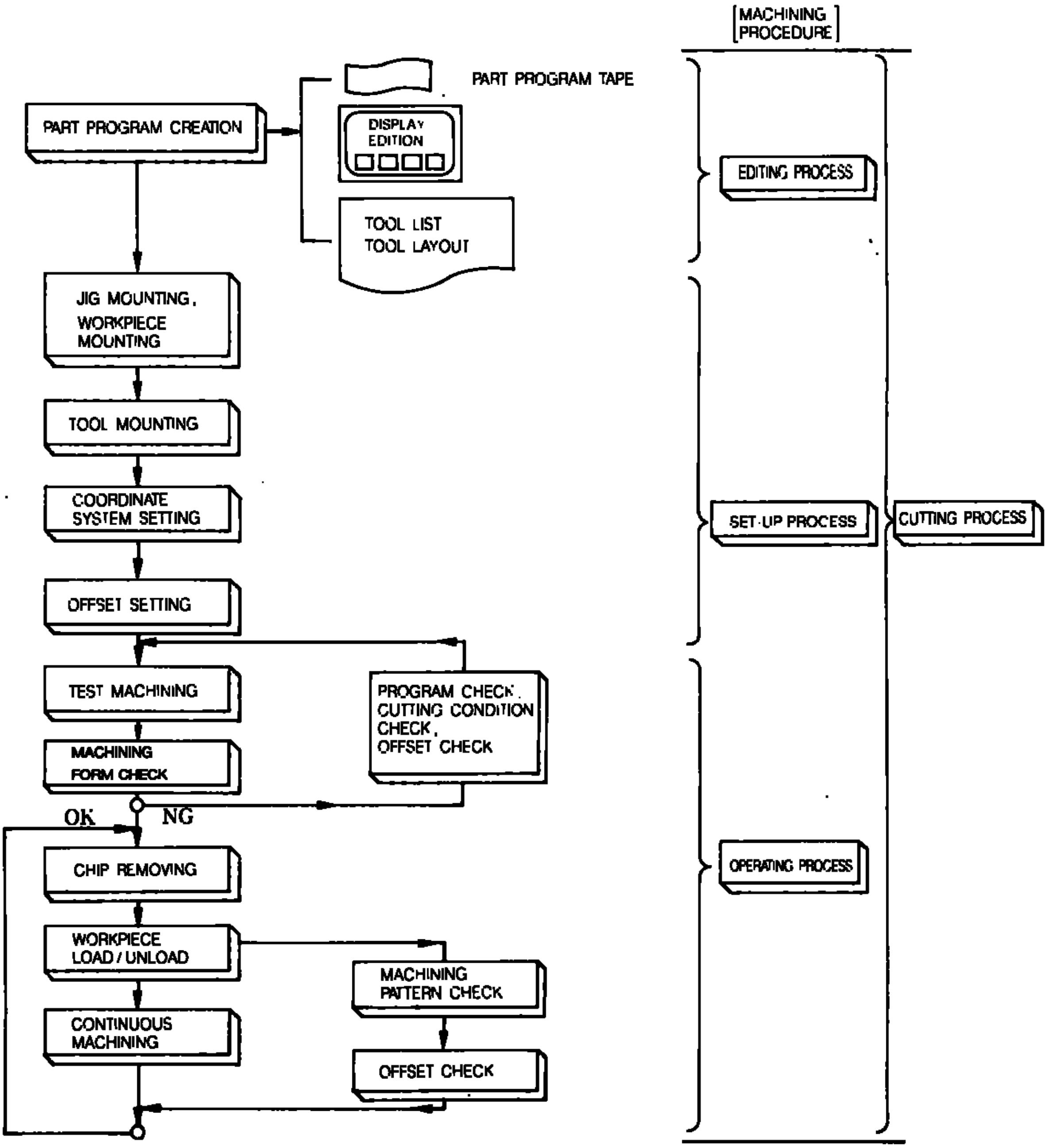
CONTENTS PAGE

CONTENTS

PAGE

3 DESCRIPTION OF OPERATION

The YASNAC i series starts the machining process by giving the question "How is it being cut ?". The interface is the operator panel (keyboard) and display unit.



Machining Procedure

3.1 DISPLAY

The display screen is structured by the 6 blocks below.

EDITING PROCESS
SET-UP PROCESS
OPERATING PROCESS
AUXI PROCESS
} Machining process screens for machining the workpiece.

MAINTENANCE PROCESS : Screens used for maintenance of the NC.

COMMON PROCESS : Screen needed for all the processes above. Monitors another screen by interrupting an operation.

Keys are assigned to the screens to control the display (four machining process keys, maintenance process key,common key).
The keys allow each machining process screen to be displayed immediately, featuring a "human-oriented NC."

3.2 STRUCTURE OF PROCESS, JOB AND FUNCTION

Each process has jobs, and each of the jobs has functions.
Up to five jobs can be assigned to a single process.

NOTE The number of functions of the jobs is not limited.
The jobs and functions are controlled by the soft key. Fig.3.2.1 shows an example of the structure.

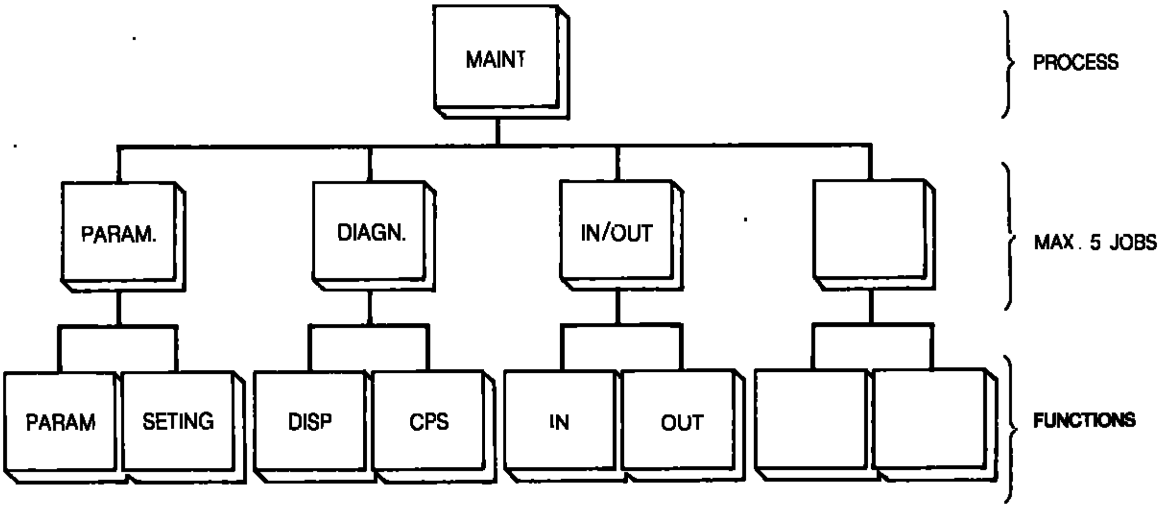
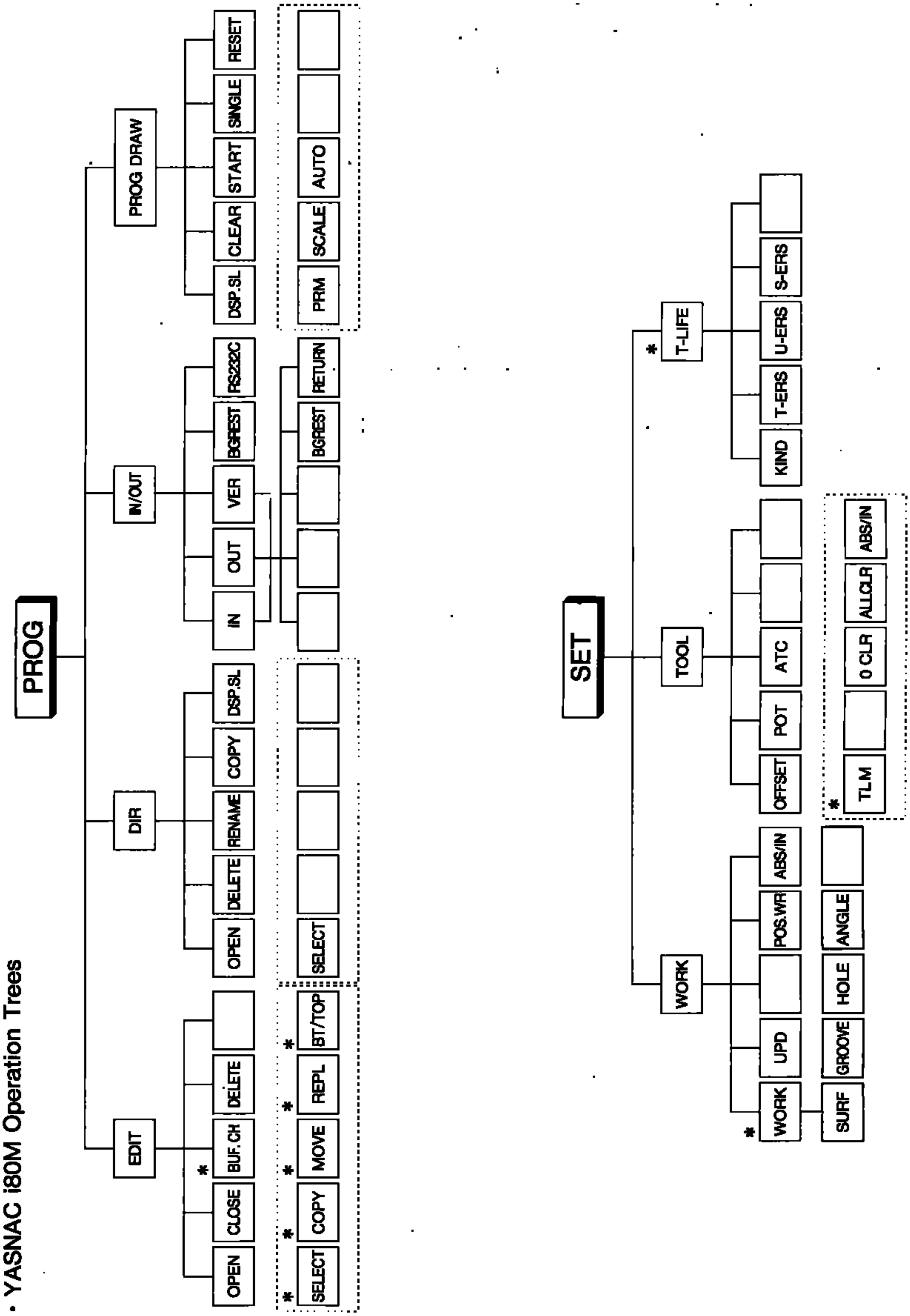
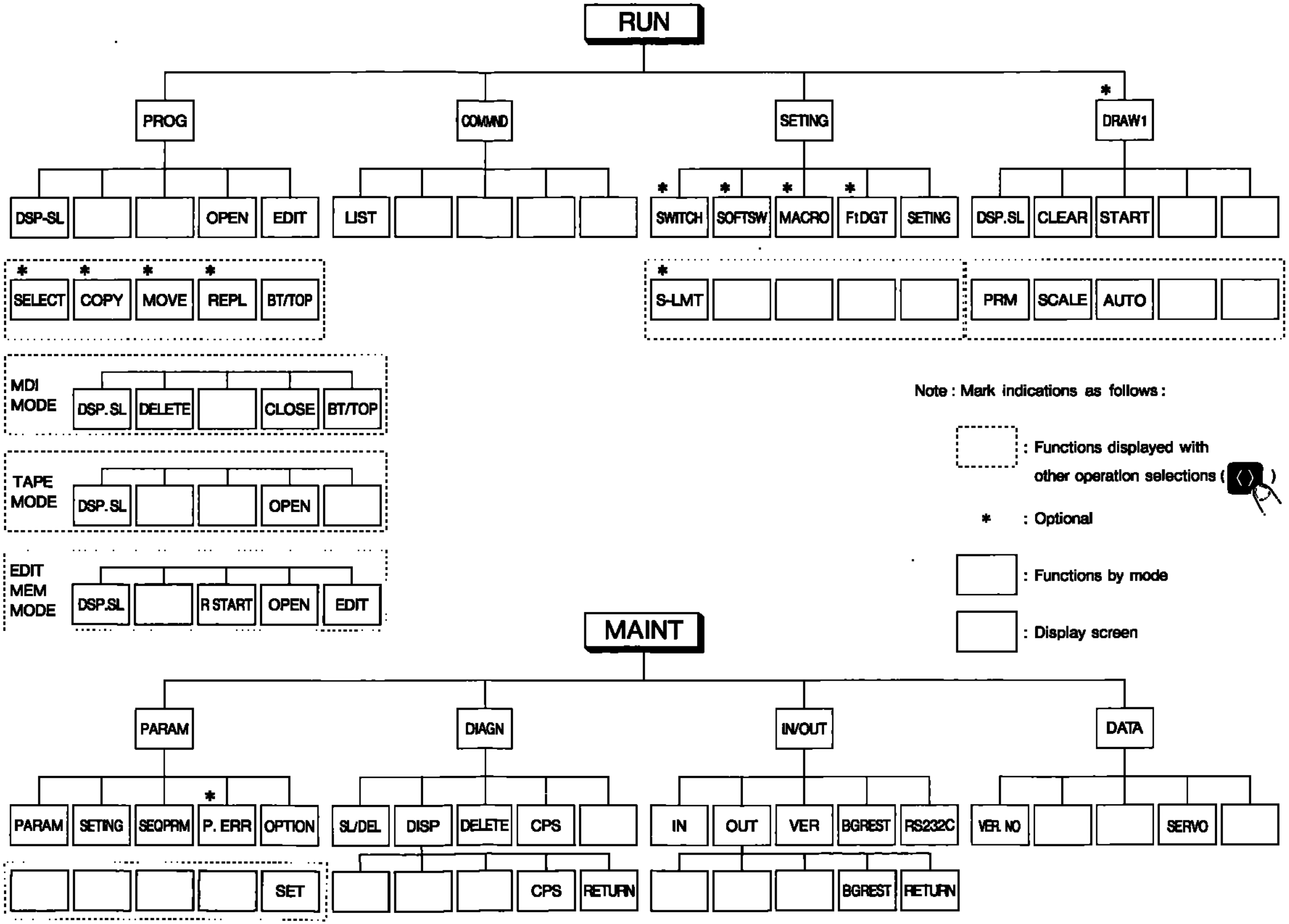


Fig. 3.2.1 Example of Structure

- YASNAC i80M Operation Trees





Note : Mark indications as follows :

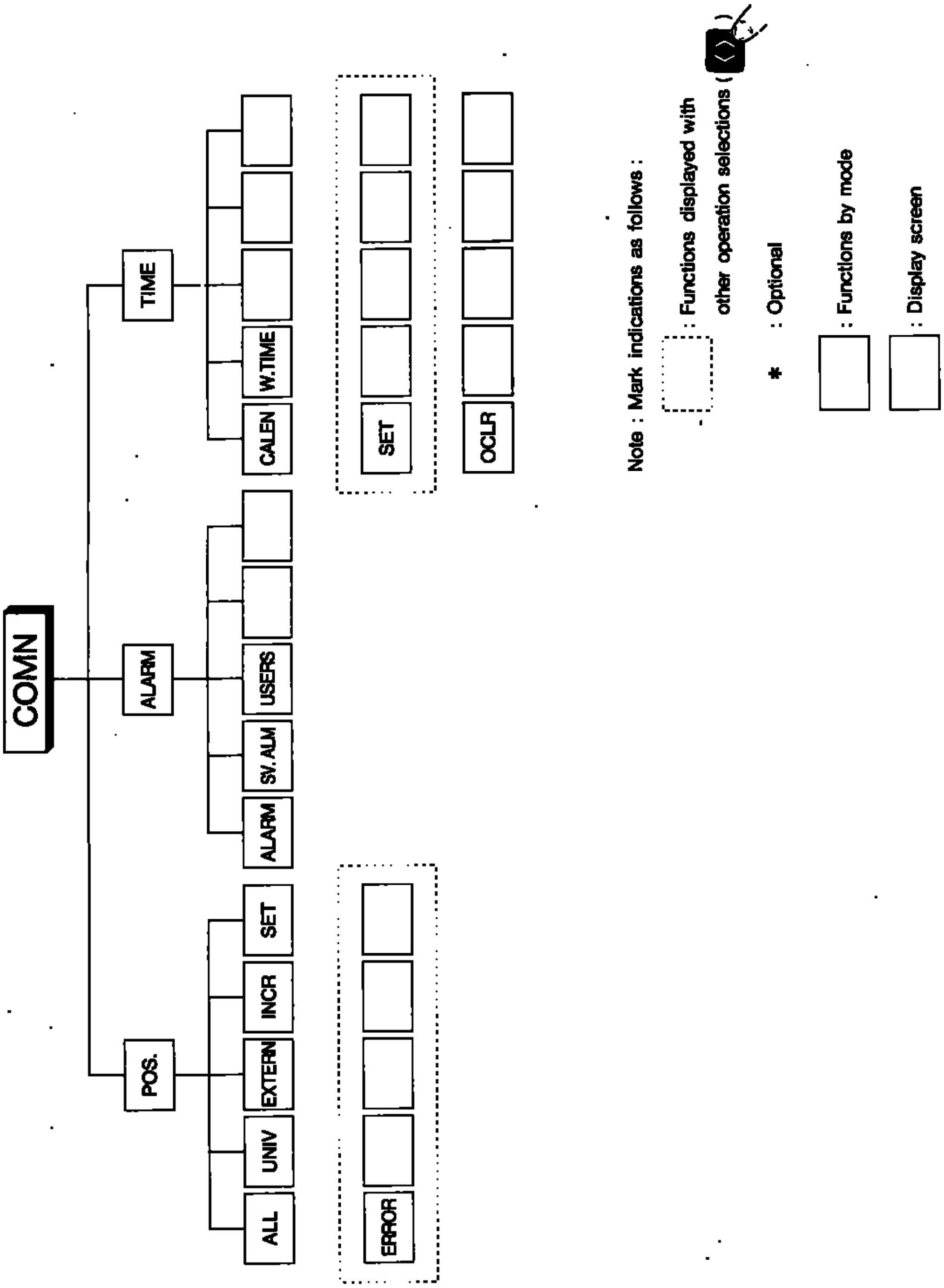
⬚ : Functions displayed with other operation selections (⟨⟩)

* : Optional

☐ : Functions by mode

☐ : Display screen

3.2 STRUCTURE OF PROCESS. JOB AND FUNCTION (Cont'd)



Note : Mark indications as follows :

: Functions displayed with other operation selections ( )

* : Optional

: Functions by mode

: Display screen

3.3 NC OPERATOR PANEL AND DISPLAY SCREEN

3.3.1 Functions of NC Operator Panel Keys

Fig.3.3.1 is the YASNAC i series panel.

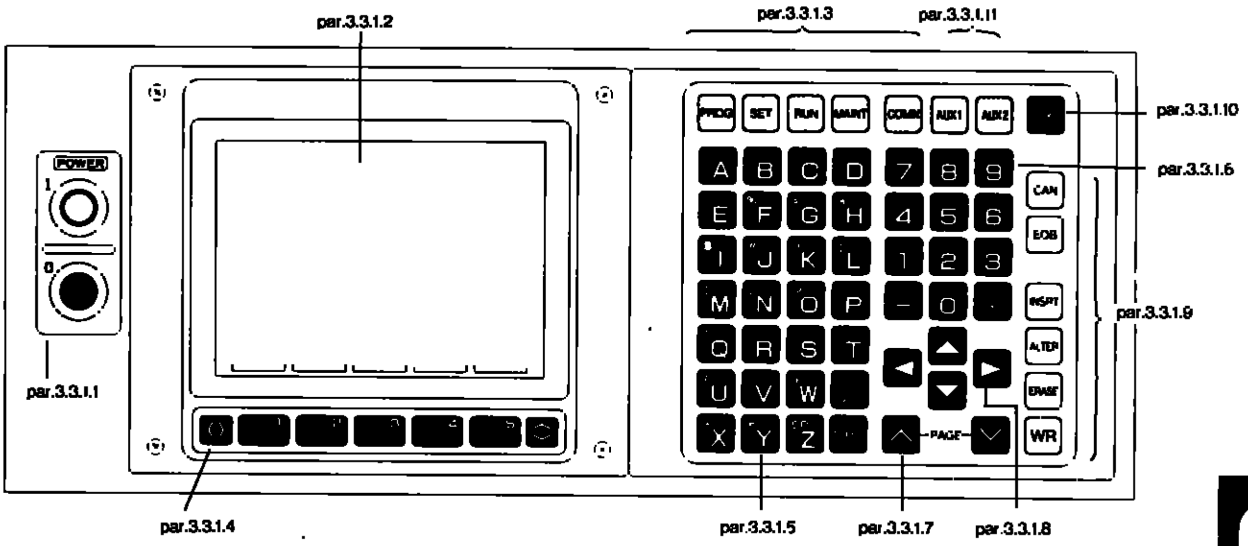
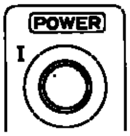


Fig. 3. 3. 1 NC Operator Panel

3.3.1.1 Power ON/OFF buttons

(1)



Power ON button:

Supplies power to the NC unit.
The power is supplied to the control unit by the first push, and to the servo by the second push.

NOTE

The machine may be supplied the power to control and servo units by one push. Refer to the manual published by the machine tool builder.

Also depress this button to supply power to the servo unit after recovering from emergency stop.

Depressing this button turns OFF power to both the controller and the servo unit of the NC unit.

(2)

Power OFF button:

Disconnects the power to the NC unit. The power supplied to the control unit and servo unit is completely disconnected.

3.3.1.2 9-inch graphic display

The data are displayed in alphanumeric characters of 1 × 1 to 3 × 3 magnification.

Maximum number of characters	40 characters · 20 lines (or 25 lines) =800 characters (or 1000 characters) [In 1-time characters]
Displayed characters	Numerals (0 to 9, −, .) Alphabets (A to Z) Special codes [/(slash), EOB, +, #, SP, =, etc.]
Graphic display	640×400 dots (fine type)

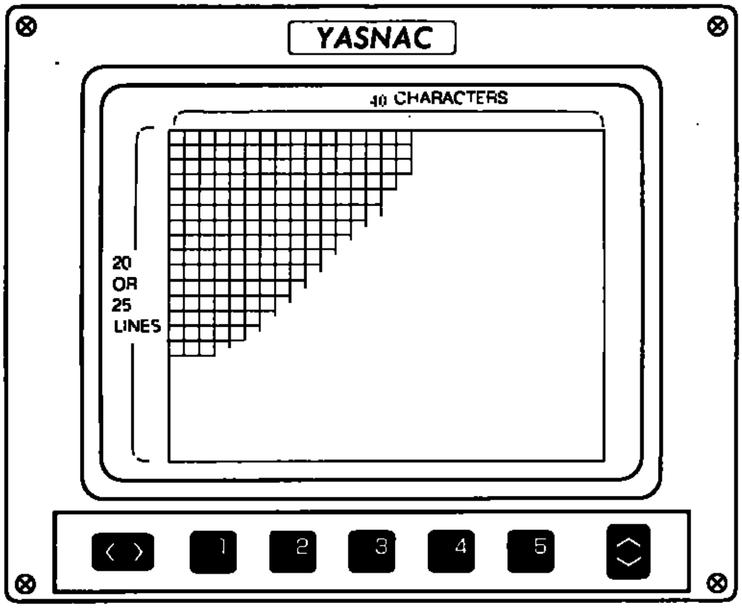


Fig. 3.3.2 9-inch Graphic Display Unit

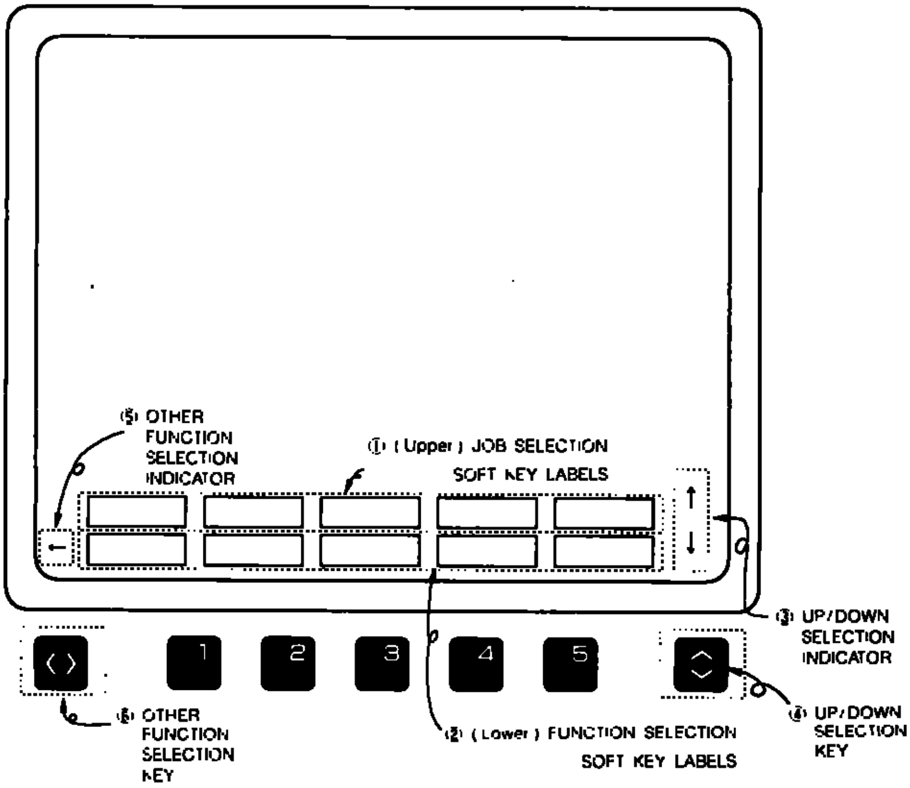
3.3.1.3 Process keys

PROG	Editing key:	Selects the editing process.
SET	Set-up key:	Selects the set-up process.
RUN	Operation key:	Selects the operating process.
MAINT	Maintenance key:	Selects the maintenance process.
COMM	Common key:	Selects the common display or returns to each process.

3.3.1.4 Soft keys

( ) 1 2 3 4 5 ⌃⌄ : Selection keys for display and writing.

NOTE Software key labels corresponding to the software keys are displayed. Depressing keys 1 through 5 calls up the corresponding functions.
The soft key labels are divided into the up and down parts as shown in Fig.3.3.3



(Description)

① Valid when up/down selection indicator is ↑ .

② Valid when up/down selection indicator is ↓ .

③ ↑ : means job selection mode.
↓ : means function selection mode.

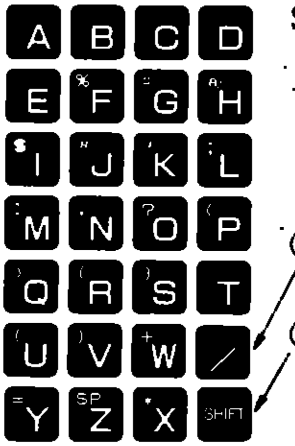
④ Switches job selection mode or function selection mode by depressing this key.

⑤ Displayed when there are more than 6 functions.

⑥ Depress this key to display the other functions.

Fig. 3. 3. 3 Soft Keys

3.3.1.5 Address keys



Specified by these keys when writing various data.

⟨Meaning of special characters⟩

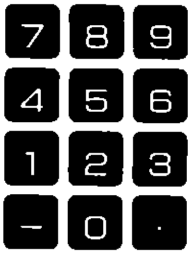
① SLASH KEY: Used to command optional block skip.

② SHIFT KEY: Used to change the input the special character on the upper left corner of the key. Depress this key and F to Z key.

NOTE

The above special characters are used for the operator in the macro program.

3.3.1.6 Data keys



Used to write numerical values as the MDI command value, tool offset amount, setting and parameters.

9 to 0 keys: Used to enter numerals.

– Minus key: Used to enter the sign.

. Decimal key: Used to enter the decimal point.

3.3.1.7 Page keys

∧ PAGE ∨ : Used to display the previous or next pages.

(1) Depress ∧ key to display the previous page.

(2) Depress ∨ key to display the next page.

(3) Continuously depressing ∧ or ∨ keys can update the page continuously to the lower or higher page.

3.3.1.8 Cursor keys

▲ ◀ ▶ ▼ : Used to move the cursor on display. The cursor on display is shown in inverse state.

- Up/down cursor

(1) Depress the ▲ key to move the cursor forward.

(2) Depress the ▼ key to move the cursor backward.

(3) Continuously depressing the ▲ or ▼ key can move the cursor continuously to the forward or reverse direction.

(4) To move the cursor to the desired position, enter the desired number and depress ▲ or ▼ key.

- Left/right cursor

(1) Depress the ◀ key to move the cursor to the left.

(2) Depress the ▶ key to move the cursor to the right.

3.3.1.9 Action keys

CAN
EOB
INSRT
ALTER
ERASE
WR

Used for various applications. Depressing these keys causes the display to change (the screen changes, data are input, etc.).

CAN **Cancel key:**

Used to cancel type error of numerals or address data.

EOB **End of block key:**

Used to command the end of a block. " ; " (semicolon) is displayed on the screen, instead of EOB.

INSRT **Insert key:**

Used to insert data in the memory.

ALTER **Alter key:**

Used to correct data in the memory.

ERASE **Erase key:**

Used to erase data in the memory.

WR **Write key:**

Used to store the address data (= word) typed via the address keys and data keys.

3.3.1.10 Reset key

: Resets the internal status of the NC unit.

(1) Depress the key to execute the following.

- Cancel the move command
- Clear the buffer
- Reset the alarm after removing the cause of alarm
- Cancel tool offset
- Cancel auxiliary functions
- Turn on the label skip function
- Rewind the memory

Differs according to parameter setting

pm4008 D0 = 0 : Rewinds memory
pm4008 D0 = 1 : Does not rewind memory

NOTE When pm 4008 D0=1, memory is rewound by softkey SRCH

- Send the reset signal
- Reset the G codes
- Clear the key buffer

(2) Note that the following are not affected by the reset.

- Current position of each axis
- F command
- S and T commands
- Tool offset amount, setting data, and parameter data

3.3.1.11 Auxilliary key

AUX1 AUX2 : Used for expanding the function, etc.

3.4 DISPLAY AND WRITING

3.4.1 Constant Display

In the top and the 6 lines of the bottom of the screen, the following display appears as shown in Fig. 3.4.1, regardless of the process, job or function selections. For the details of each constant display, see Tables 3.4.1 to 3.1.4.

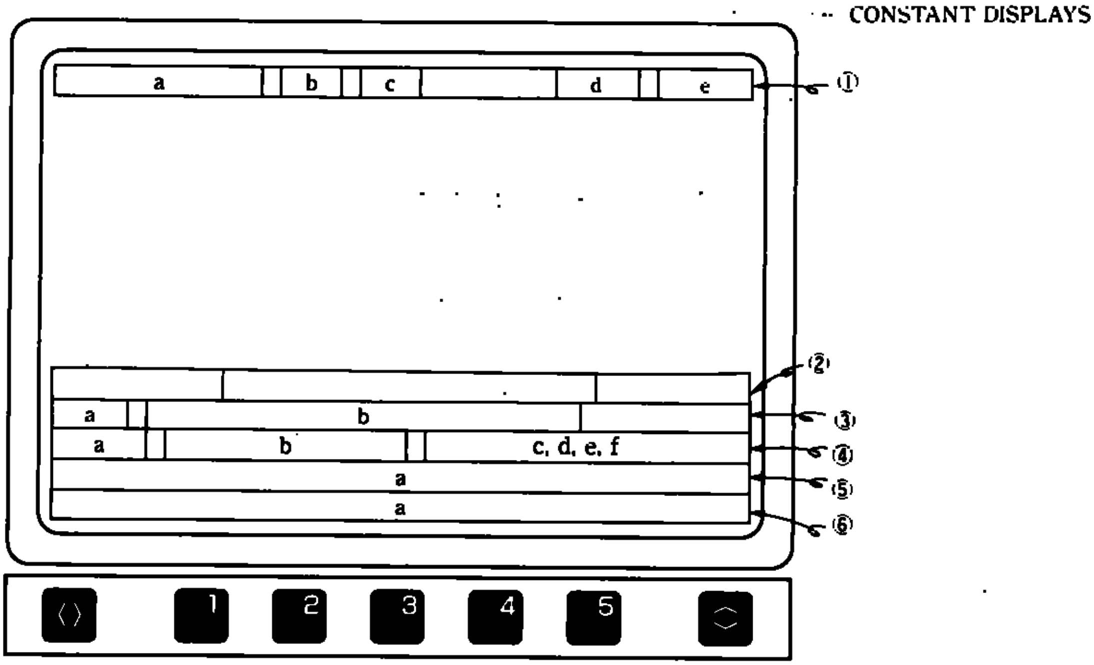


Fig. 3.4.1 Constant Display

Table 3.4.1 Meaning of Constant Display

Constant Display	Message	Meaning
❶ Screen title display	Alarm, offset, etc.	The job name of the selected screen is the title.
Selected process display	Editing, Set-up, Operation, Maintenance, Common, AUX1, AUX2	The selected process name.
Operating program number display	O***** O 0 0 0 0 1 to O 9 9 9 9 9	The number of the program operating in the selected series and line. O***** shows that program number is not selected.
Executed sequence number display	N 0 0 0 0 0 to N 9 9 9 9 9	The current sequence number of the selected series and line.
❷ Message display of operation result	INPUT? (Y/N) INPUTTING INPUT STOPPED INPUT COMPLETE OUTPUT? (Y/N) OUTPUTTING OUTPUT STOPPED OUTPUT COMPLETE VERIFY? (Y/N) VERIFYING VERIFICATION STOPPED VERIFICATION COMPLT CREATE? (Y/N) CREATION COMPLETE COPY? (Y/N) COPY COMPLETE RENAME? (Y/N) RENAMING COMPLETE DELETE? (Y/N) DELETION COMPLETE SELECT OTHER JOB PUSH JOB KEY TO RETRY PUSH EXEC TO CONTINUE EXECUTING STOP? (Y/N) SEARCH COMPLETE INPUT O NO. INPUT COMMENT SAVING COMPLETE TRANSFER PARM? (Y/N) TRANSFERRING PARM TRANSFER COMPLT INPUT O NO. SAVING SEARCHING INPUT? (PUSH Y) OUTPUT? (PUSH Y) VERIFY? (Y/N) OK? (Y/N) SET POINT 1 SET POINT 2 SET POINT 3 MODE IS UNSUITABLE SEARCH STOPPED INPUT O NO. (DELETE) DELETE ALL? (Y/N) INPUT OLD O NO. INPUT ORIGINAL O NO. INPUT NEW O NO. INPUT O NO. (COPY) RANGE IS UNSUITABLE DRAWING COMPLETE CLR ALL OFFSETS (Y/N) SEARCH COMPLETE INPUT OLD STRING INPUT NEW STRING (Yes/No/All)?	Result of key operation, or unit executed status.

Table 3.4.1 Meaning of Constant Display (Cont'd)

Constant Display	Message	Meaning
● Alarm number	0 0 0 1 to 9 9 9 9	The top priority alarm occurring in the selected series line. If there is no alarm in the selected series line, the top priority alarm occurring in the other series line is displayed.
Alarm message	TH error, Character cannot be used, External data input error, etc.	
● Operation mode	Edit, Memory, MDI, Tape, Step, Handle, Jog, Rapid feed	The mode of the selected operation.
Warning	INPUT ERROR! O NO. NOT FOUND! NOT FOUND! ALREADY IN! OVER MEM CAP! TOO MANY PROGES! VERIFY ERROR! MACRO LOCK! LINE LOCK! RUNNING PROGRAM! NC RUNNING! FORMAT ERROR! EDIT LOCK! NOT FOUND! ALREADY EDIT! SELECT MODE ERR! COPY MODE ERROR! MOVE MODE ERROR! INPUT ERROR! PRM SETTING ERR! CAN'T SET COORD! OVER MDI BUFFER! PRM SETTING ERR! PROGRAM READING! READING PROGRAM! EDITING PROGRAM! IMPOSS COLLECT! TOO MANY CHARS! SETTING UNREADY! BIAS ERROR! BOTTOM ERROR! OFFSET ERROR! H D ERROR! OVER MEM CAP! RUNNING PROGRAM! MEASURMENT ERR! TOO LARGE AREA! ALREADY IN! NC RUNNING! BREAKE POINT! PLAYBACK LOCK! SYSTEM NO.1! FORMAT ERROR! PROG NESTING ER! ADDRESSING ER! AXIS MOVING! REPETITION ERR! CAN'T WRITE! RUN MODE ERROR! AXIS ZR POS CMP! AXIS MOVING! NOT SET FOR ABS! IN/OUT VER LOCK! NO ANSWER INTEX! NO ANSWER MMON! EDIT INVALIDITY! NO APPLICATION! RUNNING PROGRAM! RESTARTING PROG! FSED MODE INVLD! CAN'T STORE FS!	Lower priority error. The operation does not stop. Reset the error by key input, mode selection or mode change, or screen switching.

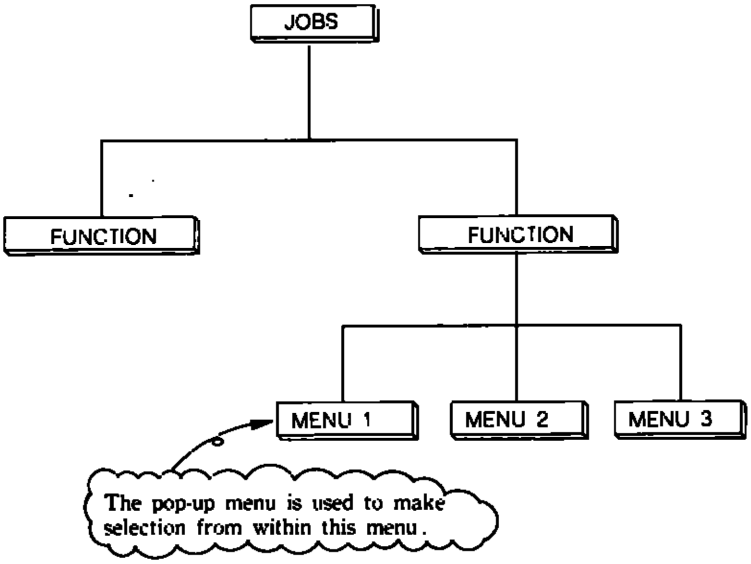
Table 3.4.1 Meaning of Constant Display (Cont'd)

Constant Display	Message	Meaning
	CAN'T CHANGE FS! CAN'T CLEAR FS! FS STORE EXCEED!	
Operation status	M/S/T/F/R/B DWELL	Indicates that it is waiting for the M code. S code. T code. to be completed. F indicates that it is during cutting; R during rapid feed ; B during execution of B function; DWELL during dwell.
		Indicates the status of the selected series.
	HSC	Indicates tool is cutting at high speed.
	STP	Indicates tool is stopped.
	RST	Indicates tool is in the reset position.
	BUFn	Indicates tool is in n-block advance reading status.
● Alarm Status		Indicates the status of the selected series line.
	ALM	Indicates an alarm has occurred.
	BAT	Indicates a battery alarm has occurred.
	A/B	Indicates both an alarm and a battery alarm have occurred.
	BGA	Indicates a background alarm has occurred.
	B/B	Indicates both a background alarm and a battery alarm have occurred.
	EBA	Indicates an absolute battery alarm has occurred.
	E/B	Indicates both absolute battery alarm and background alarm have occurred.
	B/E	Indicates both battery alarm and absolute battery alarm have occurred.
	A/E	Indicates both an alarm and absolute battery alarm have occurred.
Label skip	LSK	Displayed when label skip is ON.
● Key buffer		Key input echo back display (up to 40 characters can be input)
Soft key		Selection key for display and writing (upper and lower cases)

3.4.2 Pop-up Menu

The pop-up menu is used to select the menu within the function. There is normally a single menu under a function, but when there are multiple menus, the pop-up menu is used to make the selection.

Soft keys that support pop-up menu can be distinguished with the special soft key frame of ▯.



The pop-up menu is used to make selection from within this menu.

(Example)
Depress the DELETE soft key to display the pop-up menu, allowing selection from two menus. Select the desired menu by using ▲ or ▼ and WR, to delete the menu.

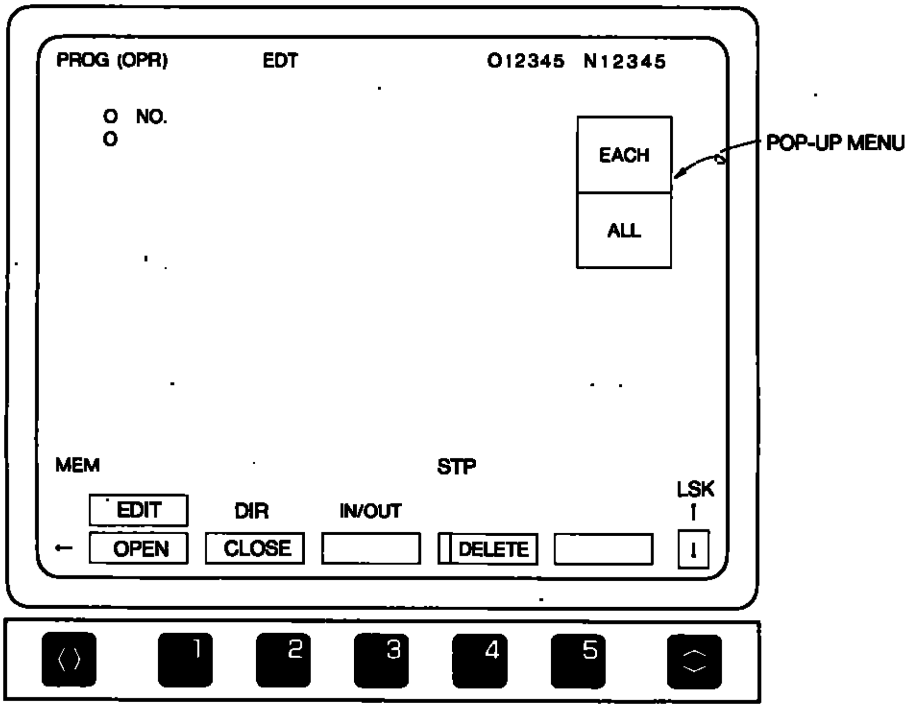


Fig. 3.4.2 Deletion in Pop-up Menu

3.4.3 Key Buffer Edit Function

Characters can be added to or deleted from data entered and displayed in the key buffer display area on the CRT screen.

This function is helpful to correct entry errors when

- the user finds an error immediately after typing, or
- the user depressed the WR key but the "INPUT ERROR!" warning message appeared and the data could not be entered.

(1) The key buffer cursor blinks at the beginning of the key buffer display area. (The cursor is indicated by anunderline in the following explanation.) As the user enters characters, the cursor is shifted to the right and displayed after the last character.

(2) If ◀ or ▶ is depressed when characters are displayed on the key buffer area, the key buffer cursor moves accordingly. In other words, the cursor keys take effect in the key buffer area only.

NOTE The cursor keys move the cursor over the whole screen when no data have been input in the key buffer area.

(3) Examples of key buffer editing are explained in the following:

(a) **Erase**

G00 X100. Z100. ; _

Suppose the above data have been entered and are displayed on the key buffer area. Erase X100.

1 Depress ◀ to move the cursor to "Z".

G00 X100. $\underline{Z}$100. ;

2 Depress the CAN key five times to erase "X100."

G00 Z100. ;

Each time the CAN key is depressed, the character immediately before the key buffer cursor is erased.

NOTE If the key buffer cursor is positioned at the first column of the key buffer area, nothing can be erased.

(b) Insert

X100. Z100. ; _

Suppose the above data have been entered and are displayed on the key buffer area. Insert "G00" before "X100."

1 Depress ◀ to move the cursor to "X".

X100. Z100. ;

2 Depress G , O , O . The characters are inserted before the key buffer cursor.

G00 X100. Z100. ;

3.4.4 Buzzer Function

The buzzer can be sounded when data are entered from the NC operator panel. Whether to sound the buzzer can be set with the parameter as follows:

pm0007 D7 0 : Not sound the operator panel buzzer.
1 : Sound the operator panel buzzer.

3.4.5 Program Editing Process. PROG

Part programs are created in the editing process.
The following operations are accomplished in the process : part program editing, program list display, and I/O with external equipment. These operations can be started anytime regardless of the controller operation mode. The operations can also be started during automatic operation.
The operations are classified into the following three jobs : part program editing, part program list display, and part program I/O verification.
Procedure for the operations are explained in the following :

1. Depress the PROG key. Any of the following jobs is displayed.

● Part program editing job	See Par.3.4.5.1 (page 348.)
● Part program list job	See Par.3.4.5.2 (page 378.)
● Part program I/O verification job	See Par.3.4.5.3 (page 394.)

2. To display a necessary job, depress the corresponding job soft key (EDIT , DIR , or IN/OUT) Otherwise, depress the PROG key to switch the displayed job.

3.4.5.1 Part program editing (job soft key EDIT)

Depress the EDIT job soft key. The following functions are available in this job :

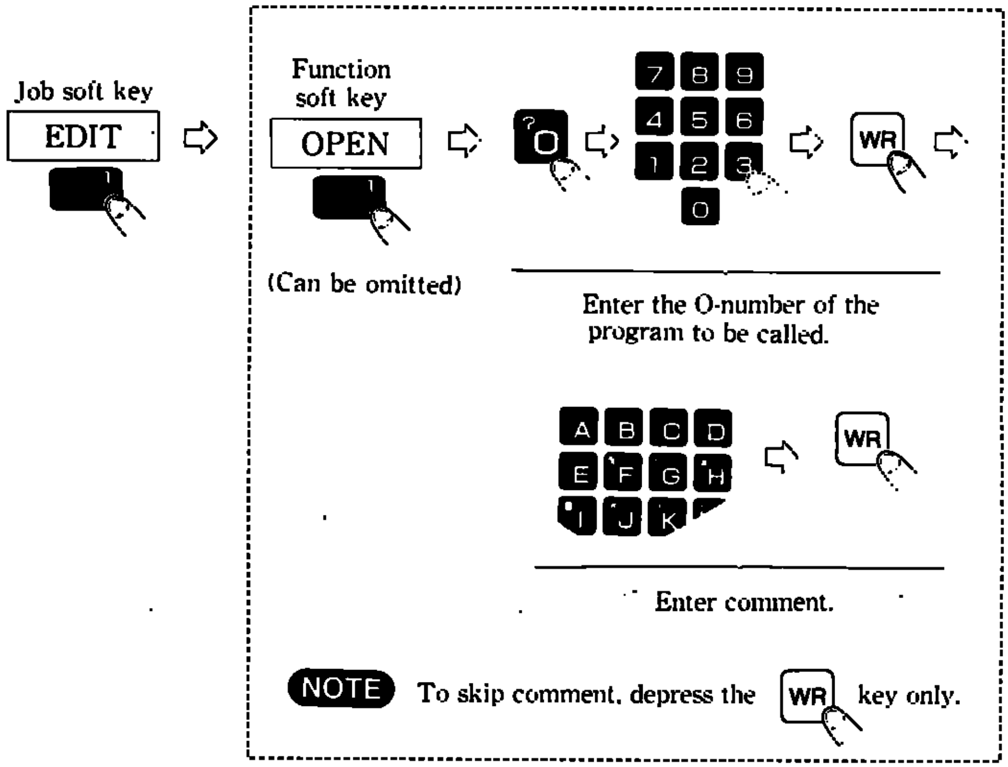
- **Part program call** : Calls a part program to be edited on the screen.
- **Part program edit** : Edits the part program called on the screen.
- **Part program save** : Saves the part program that was called to edit on the screen.
- **Part program erase** : Erases a part program.

Details of the above functions are described in the following. Up to three programs can be edited at the same time using an option.

(1) Calling part program to edit

Call a part program to be edited on the screen. If the specified O-number is not in memory, a new program is created.

KEYPOINT



(Can be omitted)

Enter the O-number of the program to be called.

Enter comment.

NOTE To skip comment, depress the WR key only.

➪ With the use of an option, up to three part programs can be called at the same time by repeating the operation in the above box. The OPEN function soft key need not be depressed.

NOTE
1. In the editing prohibited status, new program numbers cannot be created. Clear the editing prohibited status in advance.
2. A currently running program can also be called, but cannot be edited.

1. Depress the EDIT job soft key.

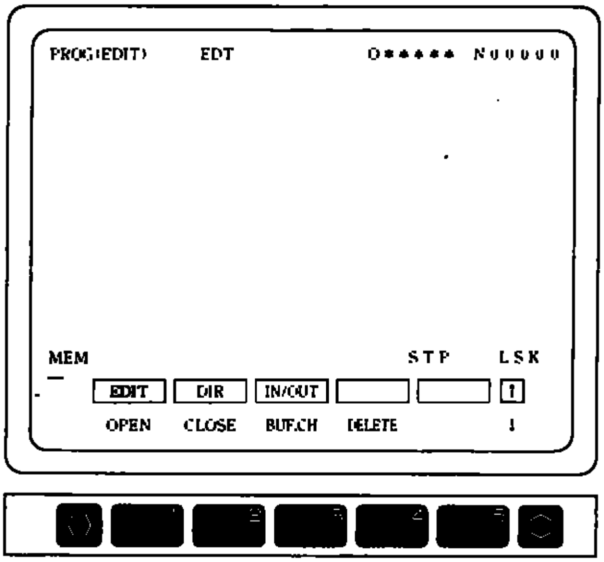


Fig. 3.4.2 Part Program Call Screen

2. Depress the OPEN function soft key. "INPUT O NO." is displayed.

   This step may be skipped.

3. Enter the O-number of the program to call.

   (For example, O 1 1 1 WR)

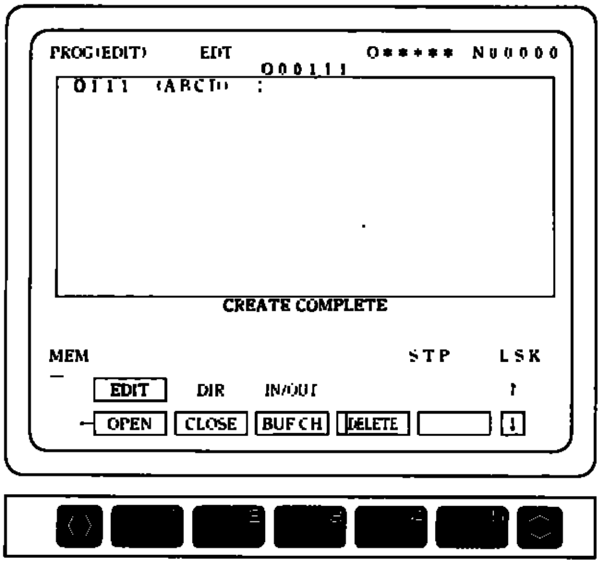
4. If the input O-number is in memory, the contents of the part program are displayed. If the number was not found, "INPUT COMMENT" is displayed.

5. Enter comment.

   (For example, A B C D WR)

   To skip comment, depress the WR key only. If the specified O-number is not found, the new number is entered in memory and displayed on the screen.

The contents of the called part program are displayed as shown in Fig. 3.4.3 :



PROG(EDIT) EDT O****** N00000

O00111

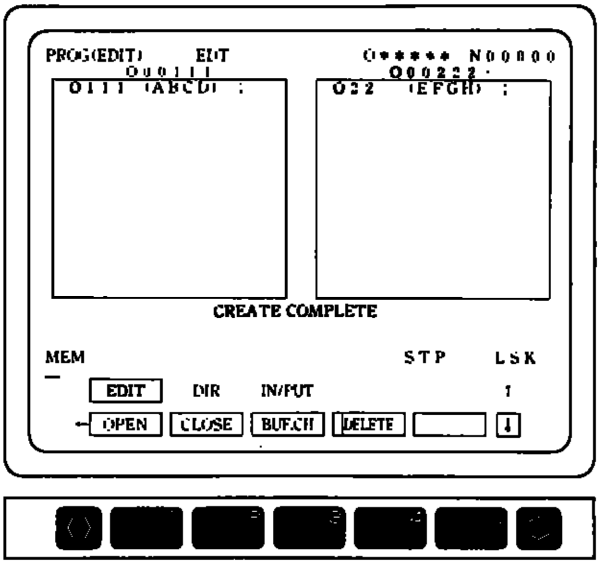
O111 (ABCD) ;

CREATE COMPLETE

MEM STP LSK

Fig. 3.4.3 Contents of (one) Called Part Program

6 If another program is to be called, depress the OPEN function soft key. "INPUT O NO." is displayed. Repeat steps **3**, **4**, and **5**. Up to three part programs can be called at the same time using an option. In Fig.3.4.4, contents of two part programs are displayed on the screen; in Fig.3.4.5, three.



PROG(EDIT) EDT O****** N00000

O00111 O00222

O111 (ABCD) ; O222 (EFGH) ;

CREATE COMPLETE

MEM STP LSK

Fig. 3.4.4 Contents of (two) Called Part Programs

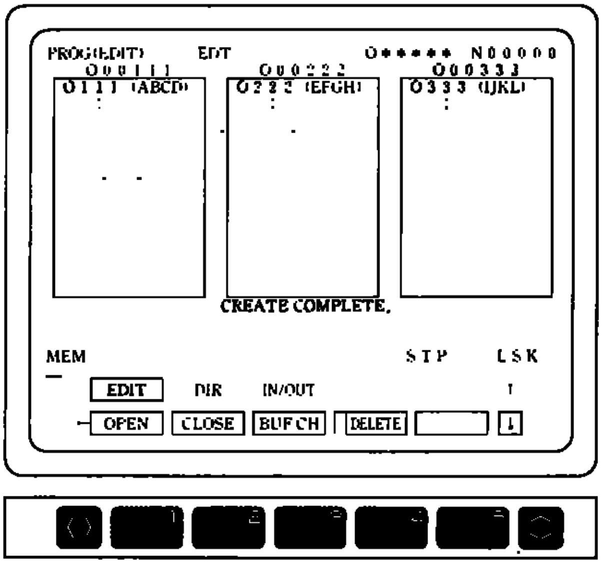


Fig. 3.4.5 Contents of (three) Called Part Programs

NOTE

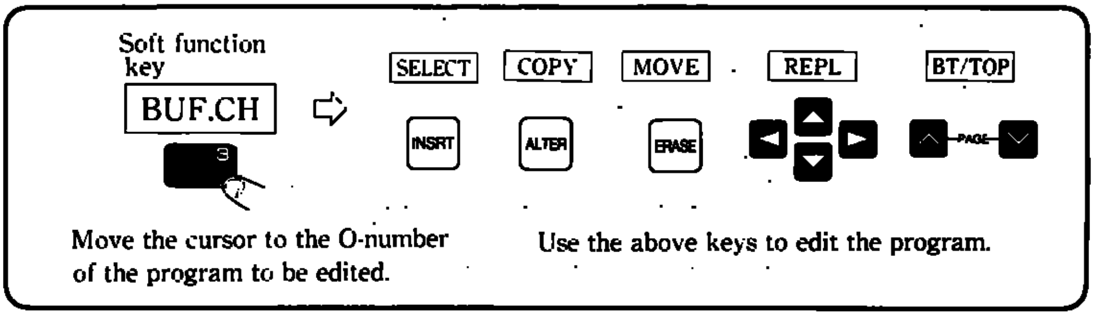
1. If a program is called although there has already been a maximum number of programs that can be edited at the same time, the new program is opened only after the one selected by the [BUF.CH] function soft key has been saved automatically.
2. A program can be called without using the [OPEN] function soft key :
   - Enter the O-number of the program to call, then depress the [WR] key.
   - If the O-number is in memory, the program is called.
   - If the O-number is not in memory, "INPUT COMMENT" is displayed, indicating that the program is new.
   - Enter comment and depress the [WR] key.

(2) Editing part program

Edit the part programs called on the screen.

Expansion editing functions (copy, move, alter, erase with range specified) can be available with an option.

KEYPOINT



Move the cursor to the O-number of the program to be edited.

Use the above keys to edit the program.

1 If more than one program has been called, depress the BUF.CH function soft key the necessary number of times to move the cursor to the O-number of the program to be edited.

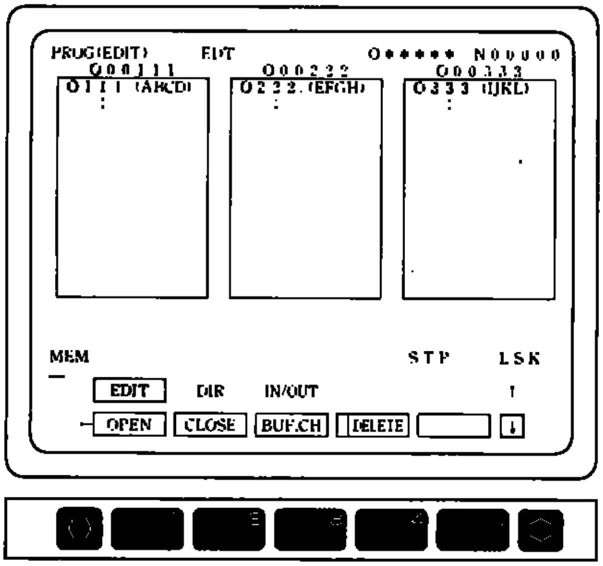


Fig. 3.4.6 Part Program Editing Screen

2 For editing programs, use the following keys : SELECT, COPY, MOVE, REPL, and BT/TOP function soft keys, INSRT ALTER and ERASE editing keys, cursory keys ▲ ▼ ◀ ▶ and ∧ PAGE ∨ Functions of these keys are explained in the following.

KEYPOINT

INSRT : Inserts data immediately after the word indicated by the cursor. [See (a).]

ALTER : Replaces the word indicated by the cursor with new data. [See (b).]

ERASE : Erases the word indicated by the cursor. [See (c).]

◀ ▲ ▼ ▶ : Move the cursor to the preceding or next block. Enables address search. [See (e) and (f).]

▲ PAGE ▼ : Display the preceding or next screen.

Function soft keys

SELECT : Specifies the beginning of a character string for COPY or MOVE.

COPY : Copies data. [See (g).] (Inserts a copy of an already registered character string to another place. Maximum character string length : 1024 characters)

MOVE : Moves data. [See (h).] (Moves an already registered character string to another location. Maximum character string length : 1024 characters)

REPL : Replaces data. [See (i).] (Searches for a specified character string and replaces it with a new one.)

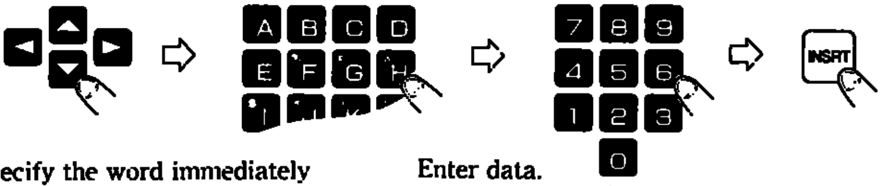
BT/TOP : Moves the cursor to the last address of the program displayed on the screen. If depressed again, moves the cursor to the first address of the program displayed on the screen.
The destinations are switched every time the key is depressed.

NOTE

1. In the editing prohibited status, new program numbers cannot be created. Clear the editing prohibited status in advance.
2. A currently running program cannot be edited. Stop the program before editing.

(a) Inserting word INSRT

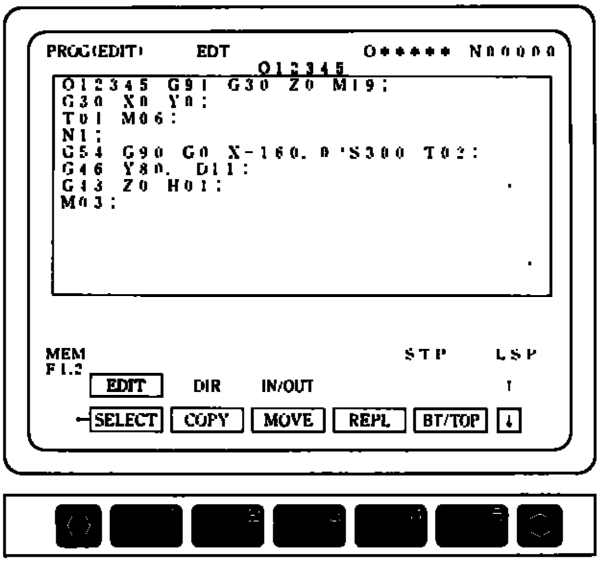
KEYPOINT



Specify the word immediately before the insertion location.

Enter data.

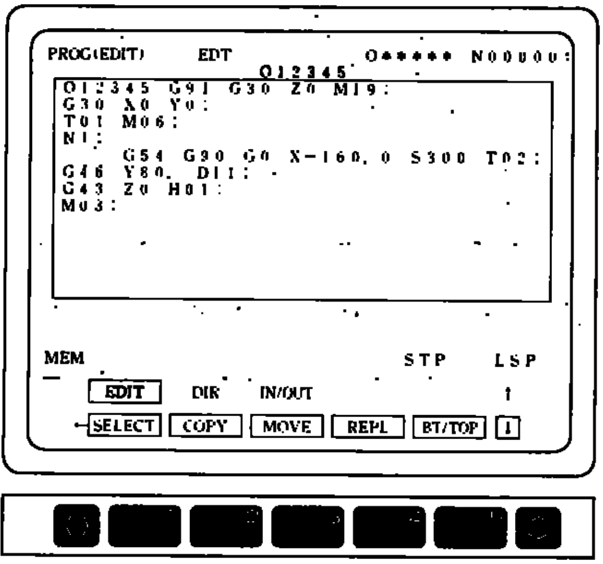
Place the cursor at the word immediately before the location where new data are to be inserted. Enter the data, then depress the INSRT key. The data are inserted immediately after the word indicated by the cursor.



```
O12345 G91 G30 Z0 M19;
G30 X0 Y0;
T01 M06;
N1;
G54 G90 G0 X-160.0'S300 T02;
G46 Y80. D11;
G43 Z0 H01;
M03;
```

Fig. 3.4.7 Screen before Insertion

For example, enter ⇩ F 1 . 2 INSRT The screen changes as follows :



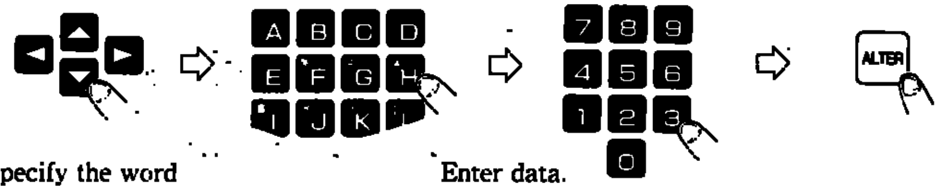
```
PROG(EDIT)      EDT               O****** N00000:
                         O12345
O12345 G91 G30 Z0 M19:
G30 X0 Y0:
T01 M06:
N1:
      G54 G90 G0 X-160.0 S300 T02:
G46 Y80. D11:
G43 Z0 H01:
M03:

MEM                                    STP   LSP
    EDIT    DIR   IN/OUT                      ↑
 ←SELECT  COPY   MOVE   REPL   BT/TOP  ↓
```

Fig. 3.4.8 Screen after Insertion

NOTE After insertion, the cursor is at the last input word. Up to 40 words can be inserted at one time by this operation.

(b) Altering word ALTER

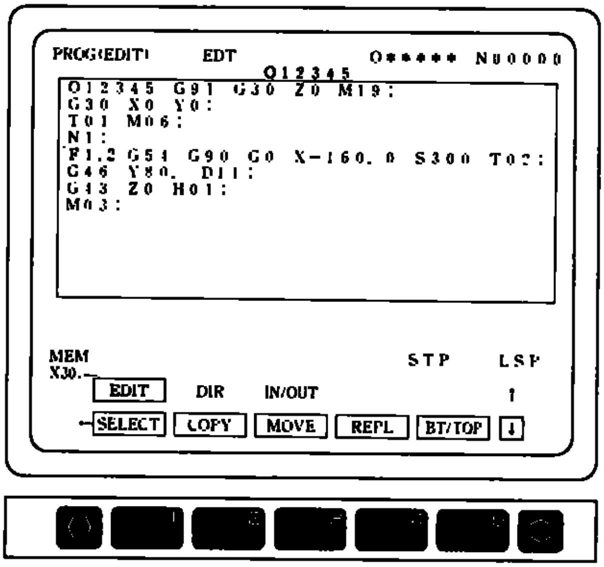
KEYPOINT



Specify the word to be changed.

Enter data.

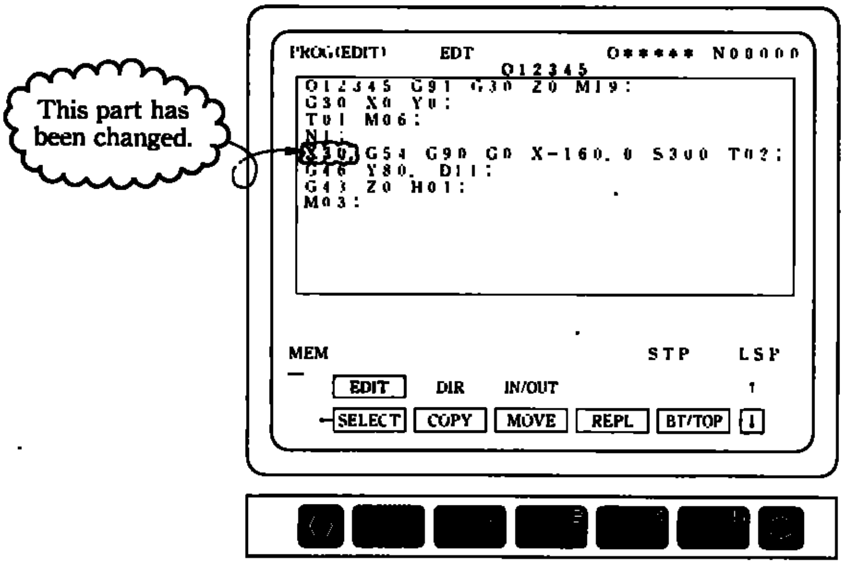
Place the cursor at the word to be changed, enter new data, then depress the ALTER key. The word at the cursor location is erased and replaced by the new entry.



```
PROG(EDIT)      EDT                O****** N00000
                      O12345
O12345 G91 G30 Z0 M19:
G30 X0 Y0:
T01 M06:
N1:
F1.2 G54 G90 G0 X-160.0 S300 T02:
G46 Y80. D11:
G43 Z0 H01:
M03:

MEM                                  STP   LSP
X30._
```

Fig. 3.4.9 Screen before Change

For example, enter ⇩ X 3 0 . ALTER The screen changes as follows :



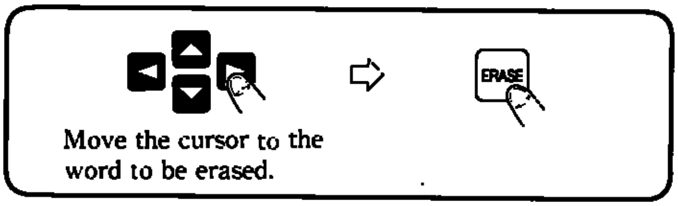
```
PROG(EDIT)      EDT                O****** N00000
                      O12345
O12345 G91 G30 Z0 M19:
G30 X0 Y0:
T01 M06:
N1:
X30. G54 G90 G0 X-160.0 S300 T02:
G46 Y80. D11:
G43 Z0 H01:
M03:

MEM                                  STP   LSP
_
```

Fig. 3.4.10 Screen after Change

NOTE If more than one word (up to 40 characters) is entered, the new data replace only one word at the cursor location.

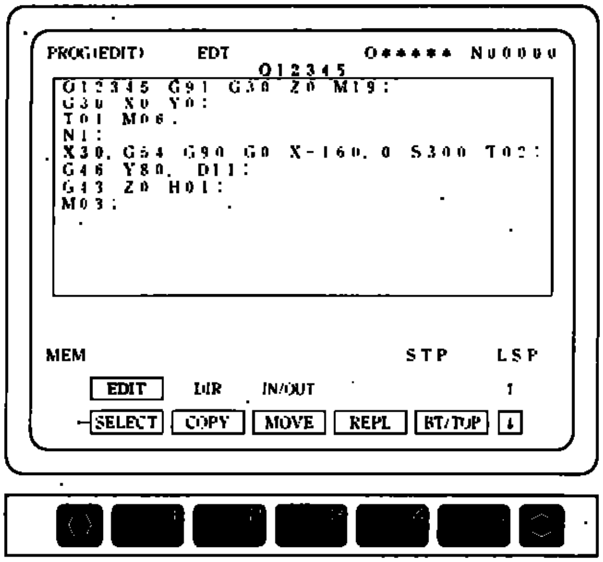
(c) Erasing word ERASE

KEYPOINT



Move the cursor to the word to be erased.

Place the cursor at the word to be erased, and depress the ERASE key. Only that word is erased.

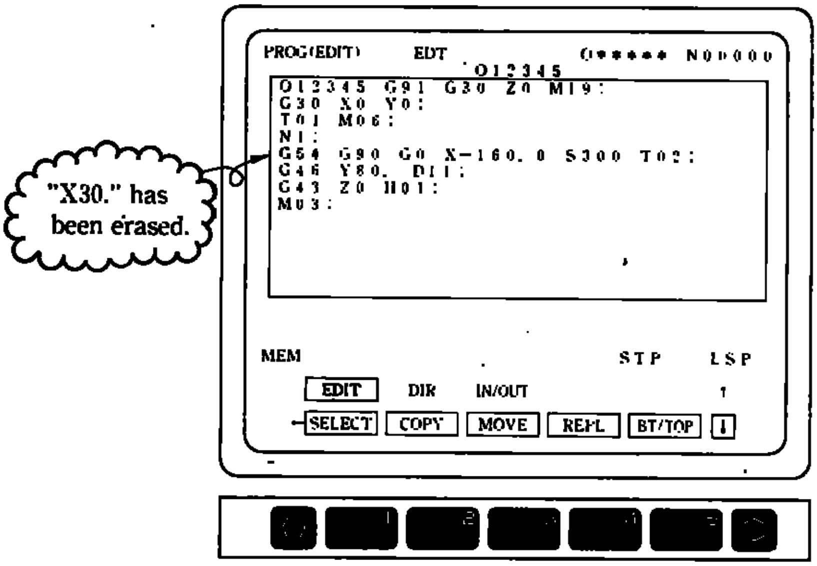


```
PROG(EDIT)      EDT              O****** N00000
                      O12345
O12345 G91 G30 Z0 M19:
G30 X0 Y0:
T01 M06.
N1:
X30. G54 G90 G0 X-160.0 S300 T02:
G46 Y80. D11:
G43 Z0 H01:
M03:

MEM                                STP   LSP
```

Fig. 3.4.11 Screen before Erasure

⇩

Depress the ERASE key. The screen changes as follows :



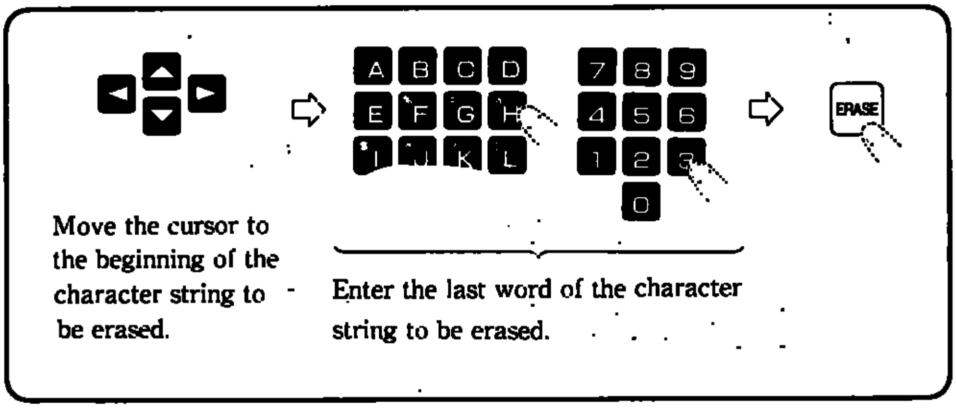
```
PROG(EDIT)        EDT                    O******  N00000
                            O12345
O12345 G91 G30 Z0 M19;
G30 X0 Y0;
T01 M06;
N1;
G54 G90 G0 X-160.0 S300 T02;
G46 Y80. D11;
G43 Z0 H01;
M03;
```

Fig. 3.4.12 Screen after Erasure

(d) **Search and erase**

The search and erase function erases a block from the current cursor location to a specified character string.

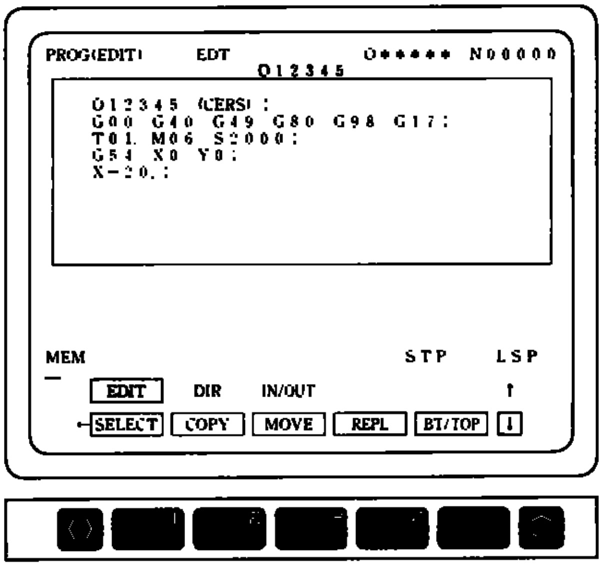
KEYPOINT



Move the cursor to the beginning of the character string to be erased.

Enter the last word of the character string to be erased.

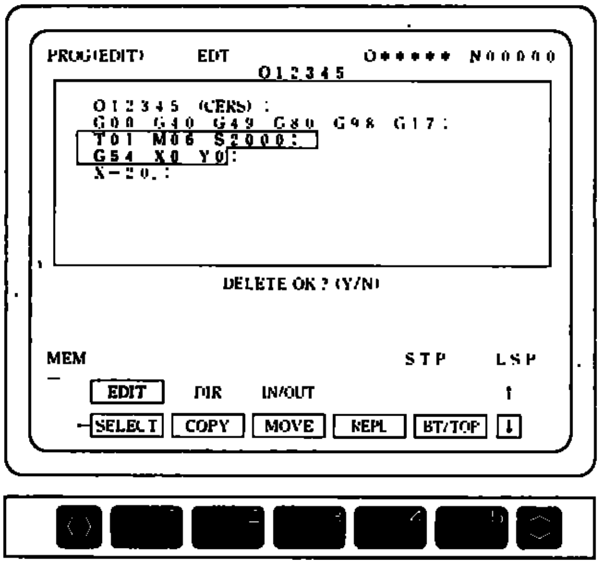
❶ Place the cursor at the beginning of the character string to be erased.



```
PROG(EDIT)      EDT                O****** N00000
                     O12345
    O12345 (CERS) ;
    G00 G40 G49 G80 G98 G17;
    T01 M06 S2000;
    G54 X0 Y0;
    X-20.;

MEM                                  STP    LSP
```

Fig. 3.4.13 Selecting the first Character of Character String to be Erased

❷ Enter the last word of the block to be erased, then depress the ERASE key. For example, to erase up to G54 X0 Y0, enter Y, O, ERASE.

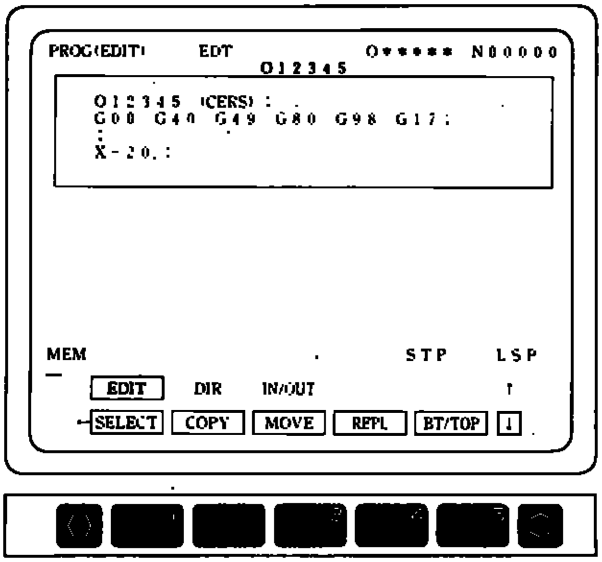


```
PROG(EDIT)        EDT                    O****** N00000
                           O12345
   O12345 (CERS) ;
   G00 G40 G49 G80 G98 G17;
   T01 M06 S2000;
   G54 X0 Y0;
   X-20.;

                    DELETE OK ? (Y/N)

MEM                                         STP    LSP
_
     EDIT      DIR     IN/OUT                        ↑
   SELECT    COPY    MOVE    REPL    BT/TOP    ↓
```

Fig. 3.4.14 Screen for Specifying the block to be Erased

- The block from the current cursor location to the specified character string is highlighted. Also, "DELETE OK? (Y/N)" is displayed highlighted, for prompting.
- If the specified character string was not found in the program, "NOT FOUND!" is displayed for warning.
- As long as "DELETE OK? (Y/N)" is displayed, INSRT, ALTER, and ERASE are disabled. Search and erase are canceled if any soft key is depressed.
- To cancel search and erase, depress any of [illegible], CAN, or N. After the [illegible] or CAN key is depressed, the cursor returns to where it was before the block was specified. After N is depressed, the cursor remains where it is.

❸ Confirm the block to be erased and depress Y. The highlighted character string is erased.



```
O12345 (CERS) :
G00 G40 G49 G80 G98 G17;
:
X-20.:
```

Fig. 3.4.15 Specifying Characters to be Erased (highlighted)

NOTE

1. If a character string beginning with an O is specified, an O-number will be erased. This is not a part of the search and erase.
2. Character strings are searched for by binary search. (Leading zeros need not be specified.)

(e) Moving cursor

❶ Depress the up and down cursor keys ▲ or ▼ to move the cursor to the preceding or next block in a part program.

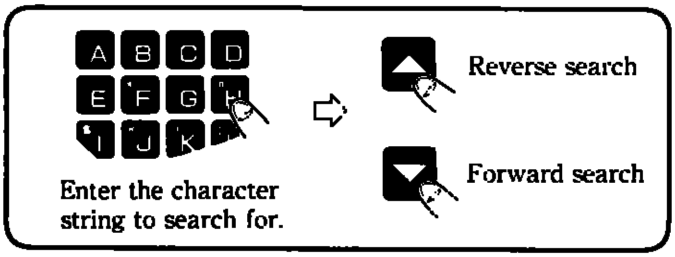
. The cursor is always at the beginning of a block.

❷ Depress the left and right cursor keys ◀ or ▶ to move the cursor to the preceding or next word.

❸ Depress the page keys ▲—PAGE—▼ to turn pages of a part program.

(f) Address search

KEYPOINT



❶ Enter the character string to search for.

❷ Depress cursor key ▼ to search from the current cursor location to the end of the program. During a search, "SEARCHING" is displayed.

- Depress cursor key ▲ to search from the current cursor location toward the beginning of the program.

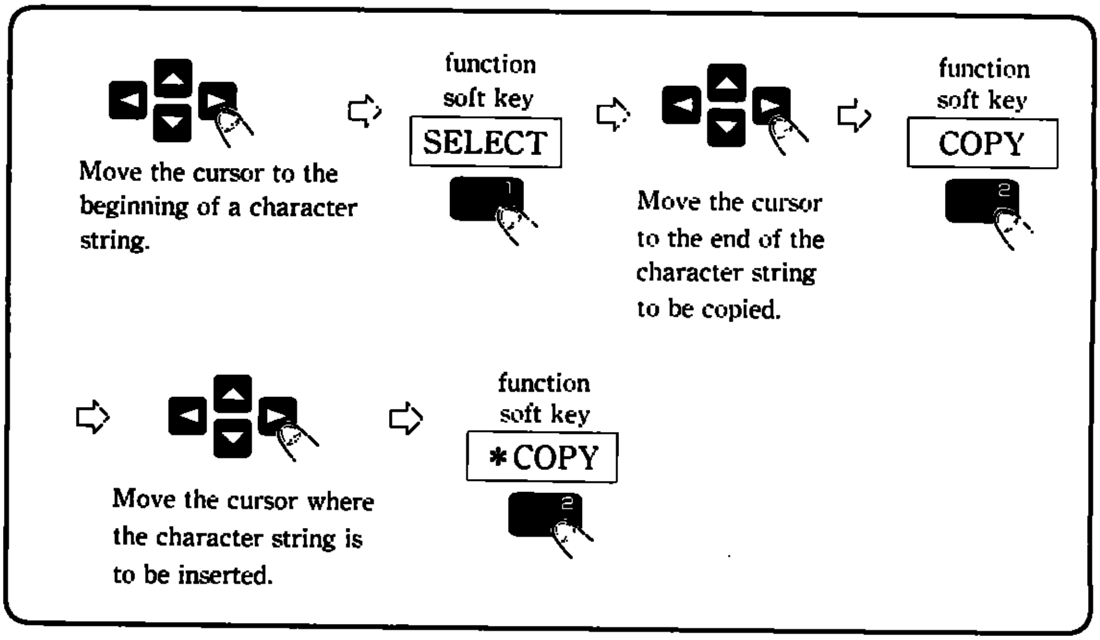
❸ If the target word is found, the message disappears and the cursor is at the word.

❹ If the target word is not found, a warning message is issued.

search for the same character string toward the beginning of the program, depress  keys at the same time. Search operation can be repeated an unlimited number of times until another character string is searched for.

(g) Copy function soft key COPY

KEYPOINT



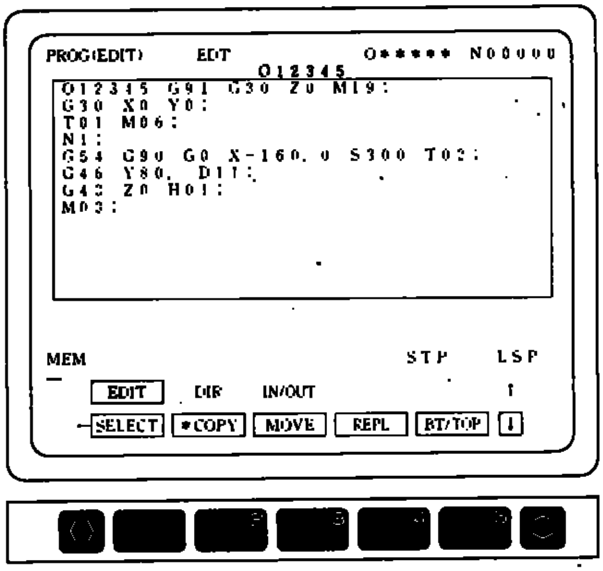
❶ Move the cursor to the beginning of a character string.

❷ Depress the SELECT function soft key. The key indication ("SELECT") is highlighted and the select mode is started.

NOTE If the SELECT function soft key is depressed when it is highlighted, the select mode is canceled.

❸ Move the cursor to the end of the character string to be copied. The specified character string is highlighted.

❹ Depress the COPY function soft key. The selected character string is saved in memory. At the same time, an asterisk is displayed at the side of the COPY function soft key. The character string that has been highlighted returns to normal display.



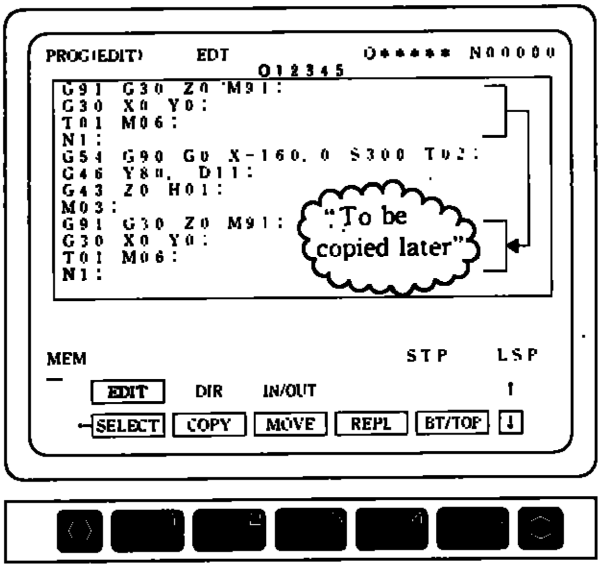
```
PROG(EDIT)      EDT                   O****** N00000
                          O12345
O12345 G91 G30 Z0 M19;
G30 X0 Y0;
T01 M06;
N1;
G54 G90 G0 X-160.0 S300 T02;
G46 Y80. D11;
G43 Z0 H01;
M03;

MEM                                    STP    LSP
```

Fig. 3.4.16 Specifying Character string to Copy

❺ Move the cursor where the character string is to be inserted.

❻ Depress the ***COPY** function soft key. A copy of the selected character string is inserted at the cursor location.

❼ As long as the asterisk is displayed at the **COPY** function soft key (***COPY**) the same character string can be copied repeatedly each time the function soft key is depressed.



```
PROG(EDIT)      EDT                   O****** N00000
                          O12345
G91 G30 Z0 M91;
G30 X0 Y0;
T01 M06;
N1;
G54 G90 G0 X-160.0 S300 T02;
G46 Y80. D11;
G43 Z0 H01;
M03;
G91 G30 Z0 M91;
G30 X0 Y0;
T01 M06;
N1;

MEM                                    STP    LSP
```

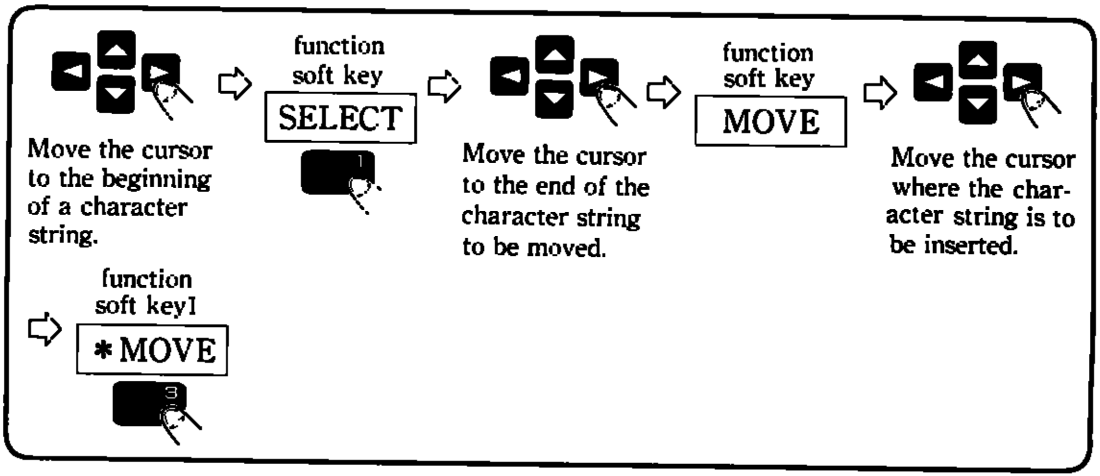
Fig. 3.4.17 Copy Screen

NOTE

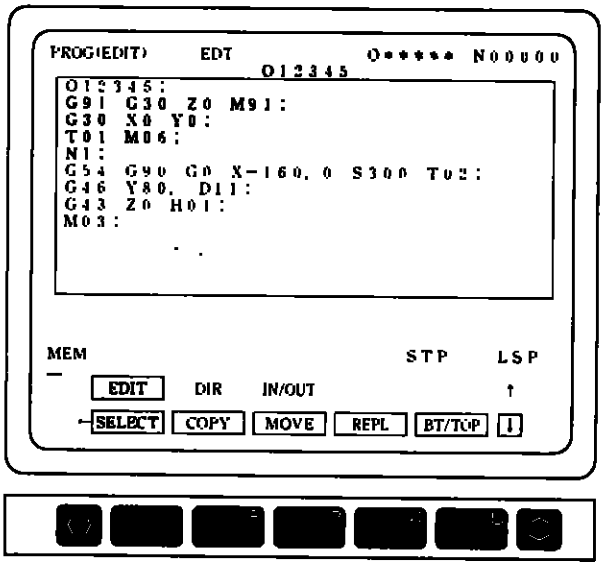
1. If power is turned off, the selected character string is cleared and the asterisk disappears.
2. The same character string can be copied in two or more programs without selecting it in each program.
3. If MOVE is used after COPY, the character string selected to copy is cleared.
4. If more than 1024 characters are selected to copy, "MEM: AREA OVER" is displayed for warning.

(h) Move function soft key MOVE

KEYPOINT



❶ Specify a character string to move by depressing SELECT, in the same way as in specifying to copy. The selected character string is highlighted.



```
PROG(EDIT)      EDT               O*****  N00000
                        O12345
O12345;
G91 G30 Z0 M91;
G30 X0 Y0;
T01 M06;
N1;
G54 G90 G0 X-160.0 S300 T02;
G46 Y80. D11;
G43 Z0 H01;
M03;

MEM                                    STP    LSP
```

Fig. 3.4.18 Specifying Character String to be Moved

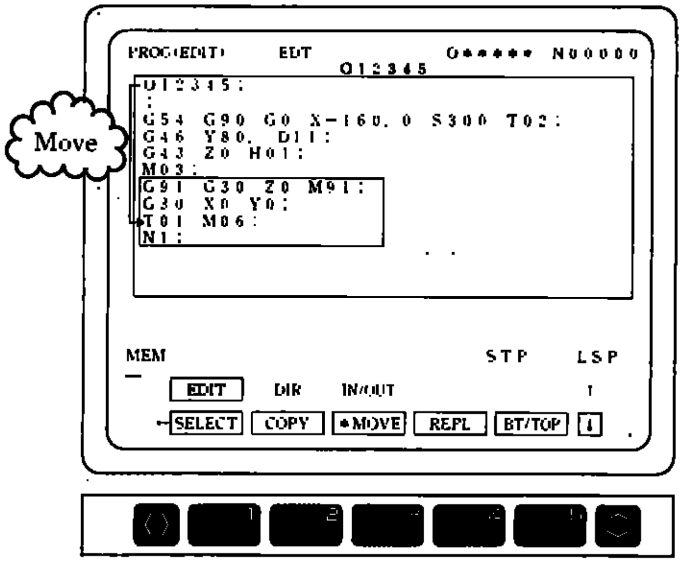
❷ Depress the MOVE function soft key. The selected character string is saved in memory. At the same time, an asterisk is displayed on the side of the function soft key.

The character string that has been highlighted is erased.

❸ Move the cursor where the character string is to be inserted.

❹ Depress the [*MOVE] function soft key. The selected character string is inserted at the cursor location.

❺ As long as the asterisk is displayed at the [*MOVE] function soft key, the same character string can be moved over and over by pressing the [*MOVE], function soft key.



```
PROG(EDIT)      EDT                  O****** N00000
                          O12345
O12345;
 :
G54 G90 G0 X-160.0 S300 T02;
G46 Y80. D11;
G43 Z0 H01;
M03;
G91 G30 Z0 M91;
G30 X0 Y0;
T01 M06;
N1;

MEM                                   STP    LSP
```

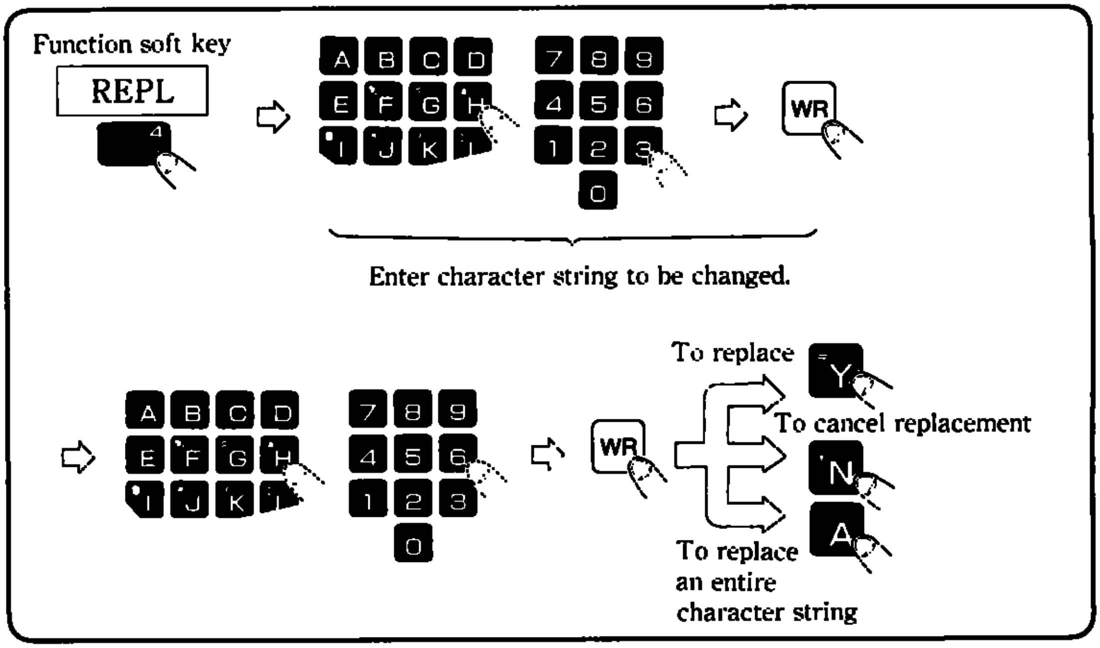
Fig. 3.4.19 Screen after Move

NOTE

1. If power is turned OFF, the selected character string is cleared and the asterisk disappears.
2. The same character string can be moved through two or more programs.
3. If COPY is used after MOVE, the character string selected to move is cleared.
4. If more than 1024 characters are selected to copy, "MEM. AREA OVER" is displayed for warning.
5. If an O-number at the beginning of the program is included, the block cannot be moved. If such a block is selected, "MOVE MODE ERROR!" is displayed for warning.

(i) Replace function soft key REPL

KEYPOINT



● Depress the REPL function soft key. "INPUT OLD STRING" is

displayed. Enter the character string to be changed.

See the following examples :

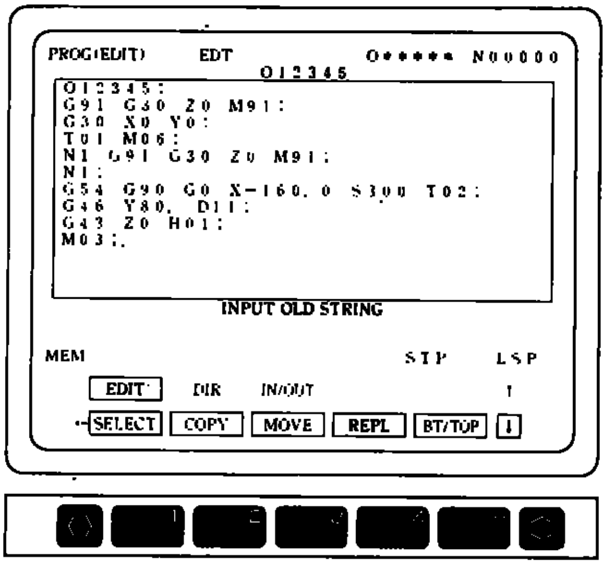
- To change "G00", enter

- To change the address block of address Y, enter

- To change the data block of data -10.5, enter

(a) Only a single word can be specified for the old character string. If two or more words are specified, "INPUT ERROR!" is displayed.

(b) The wild card ("＊") can be used for the old character string. Use the wild card as a substitute for an unknown character in the address block or a number in the data block.



```
PROG(EDIT)      EDT                O****** N00000
                      O12345
O12345;
G91 G30 Z0 M91;
G30 X0 Y0;
T01 M06;
N1 G91 G30 Z0 M91;
N1;
G54 G90 G0 X-160.0 S300 T02;
G46 Y80. D11;
G43 Z0 H01;
M03;.

              INPUT OLD STRING

MEM                                STP    LSP
```

Fig. 3.4.20 Replace Screen

NOTE

1. Only one wild card can be used. If two or more are used, "INPUT ERROR!" is displayed for warning.
2. When the wild card is used, that part is disregarded and only the rest of the character string is searched for.

● "INPUT. NEW STRING" is displayed.

(a) More than one word can be entered for the new character string.

NOTE The new character string must begin with an address. Otherwise, "INPUT ERROR" is displayed for warning.

(b) If the wild card is used in the old character string, it can be also used in the new character string.

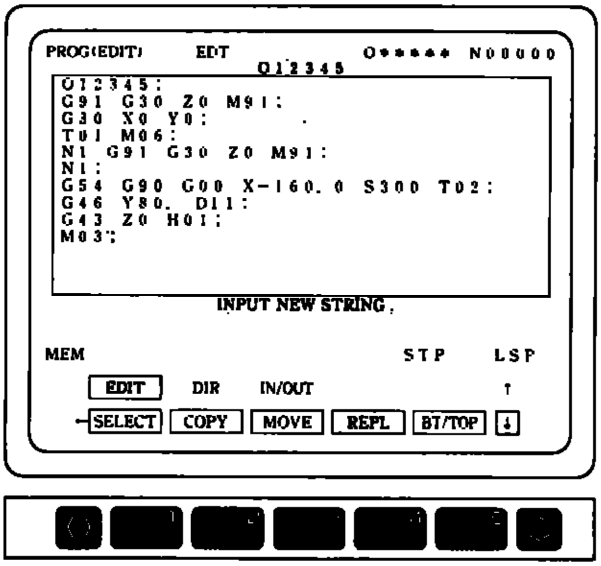
NOTE If the wild card was used in the address block of the old character string, it must not be used in the data block of the new character string. If the wild card was used in the data block of the old character string, it must not be used in the address block of the new character string.

Examples	Old string	New string
	∗10.0	∗-10.0
	Y∗	Z∗
	∗10.0	Z∗ → "INPUT ERROR!"

Enter the new character string.

Examples

- To change "G00" to "G01", enter G 0 1 WR
- To change the address block of "Y∗" to "Z", enter Z SHIFT X WR
- To change the data block of "∗-10.5" to "10.5", enter SHIFT X 1 0 . 5 WR



```
PROG(EDIT)      EDT             O*****  N00000
                     O12345
O12345;
G91 G30 Z0 M91;
G30 X0 Y0;
T01 M06;
N1 G91 G30 Z0 M91;
N1;
G54 G90 G00 X-160.0 S300 T02;
G46 Y80. D11;
G43 Z0 H01;
M03;
                INPUT NEW STRING.
MEM                               STP   LSP
```

Fig. 3.4.21 Entering New Character String

NOTE If the wild card is used in the old character string, only a single word can be specified for the new character string. If two or more words are specified, "INPUT ERROR!" is displayed.

❸ The cursor moves to the character string specified in ❶, and "(Yes/No/All)?" is displayed.

(a) Depress Y to replace the character string specified in ❶ by the one specified in ❷.
If the character string specified in ❶ is at another location, the cursor moves to it.
If the character string specified in ❶ was not found, then the replacement operation ends.

(b) Depress N not to replace the character string.
If the character string specified in ❶ is at another location, the cursor moves to it.
If the character string specified in ❶ was not found, then the replacement operation ends.

(c) Depress A to replace every character string specified in ❶ by the one specified in ❷, throughout the program displayed on the screen, without asking for approval for each replacement.

(d) Depress the REPL 4 function soft key again, or depress the [illegible] key to exit from the replacement mode.

NOTE
1. If the character string specified in ❶ could not be found at all, "NOT FOUND!" is displayed.
2. As long as "Yes/No/All)?" is displayed, SELECT, COPY, MOVE, BT/TOP, and other editing operations are disabled.

(3) Saving part program after editing

Called part programs must be saved after being edited. Part program memory in the controller is rewritten only after the program is stored. After one program is saved, another part program can be called.

KEYPOINT

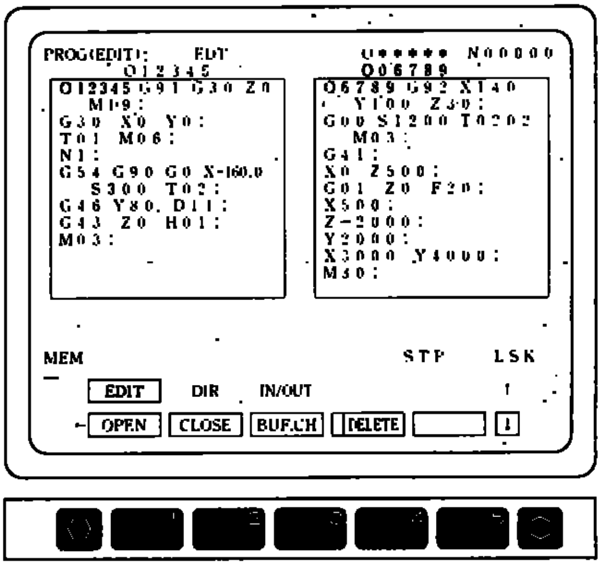
Function soft key

BUF.CH ⇨ CLOSE

Move the cursor to the O-number to save.

1. Assume that O-numbers of programs that have been called are displayed on the screen as shown in Fig 3.4.23.
2. Depress the BUF.CH function soft key to select the O-number of the program to save. The cursor moves each time the key is depressed.
3. Depress the CLOSE function soft key.

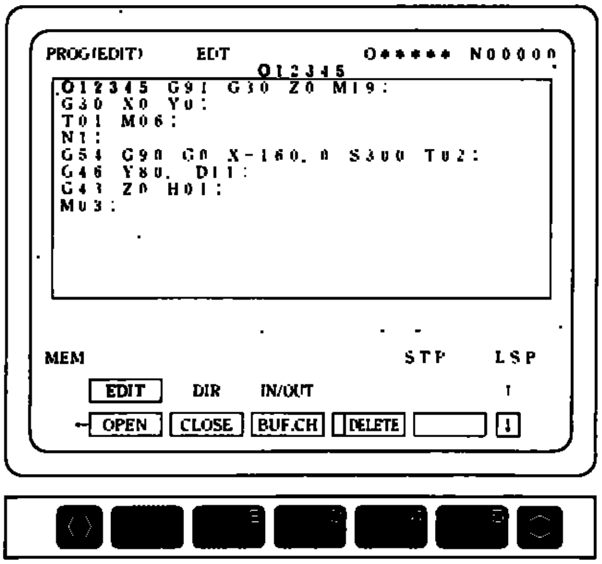
NOTE If any process key except the editing process keys is depressed while a part program is being edited, the program is automatically saved temporarily. Depress the editing process key again to continue editing. Do not turn OFF power during editing. Save part programs before turning OFF power.



```
PROG(EDIT):      EDT                    O****** N00000
          O12345                        O06789
O12345 G91 G30 Z0               O6789 G92 X140
   M19:                             Y100 Z30:
G30 X0 Y0:                      G00 S1200 T0202
T01 M06:                           M03:
N1:                             G41:
G54 G90 G0 X-160.0              X0 Z500:
   S300 T02:                    G01 Z0 F20:
G46 Y80. D11:                   X500:
G43 Z0 H01:                     Z-2000:
M03:                            Y2000:
                                X3000 Y4000:
                                M30:

MEM                                       STP   LSK
```

Fig. 3.4.22 Screen before Part Program is Saved

Depress the CLOSE function soft key. The screen changes as follows :



```
PROG(EDIT)       EDT                    O****** N00000
                          O12345
O12345 G91 G30 Z0 M19:
G30 X0 Y0:
T01 M06:
N1:
G54 G90 G0 X-160.0 S300 T02:
G46 Y80. D11:
G43 Z0 H01:
M03:

MEM                                       STP   LSP
```

Fig. 3.4.23 Screen after Part Program is Saved

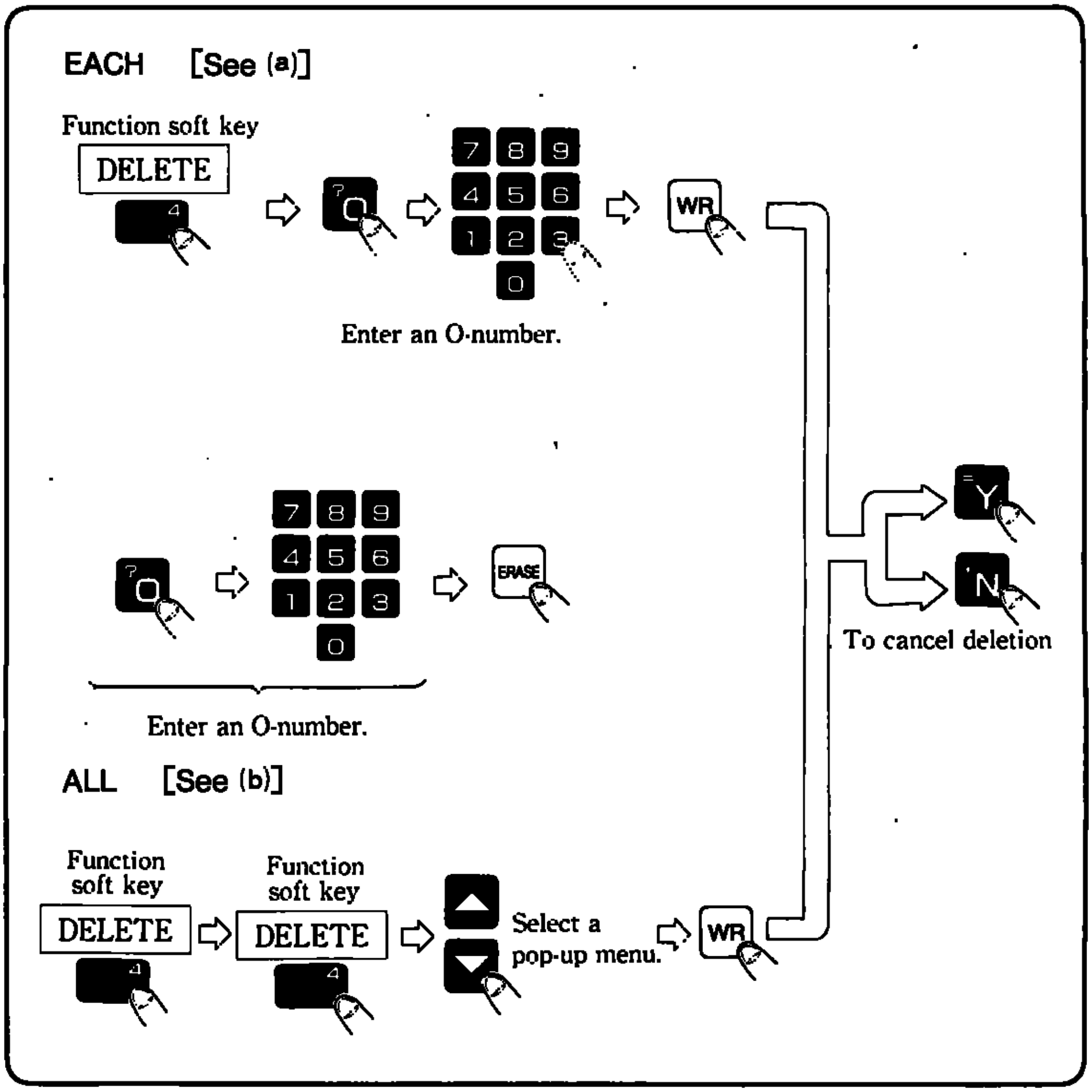
(4) Deleting part program

There are two deleting menus:

EACH : Deletes a single part program.

ALL : Delete all part programs.

KEYPOINT



NOTE

1. In the editing prohibited status, part programs cannot be deleted. Clear the editing prohibited status in advance.
2. A currently running program cannot be deleted. Stop the program before deleting it.

(a) EACH

There are two procedures :

(i) Using DELETE function soft key

1 Depress the DELETE function soft key. "INPUT DEL O NO." is displayed.

2 Enter the O-number of the program to delete.

(For example, 0 1 2 3 WR)

(ii)Starting with O-number

1 Enter the O-number of the program to delete.

2 Depress the ERASE key.

3 After (i) or (ii), "DELETE OK? (Y/N)" is displayed.

- Depress Y to delete the program. Then "DELETE COMPLETE" is displayed.
- Depress N not to delete the program.

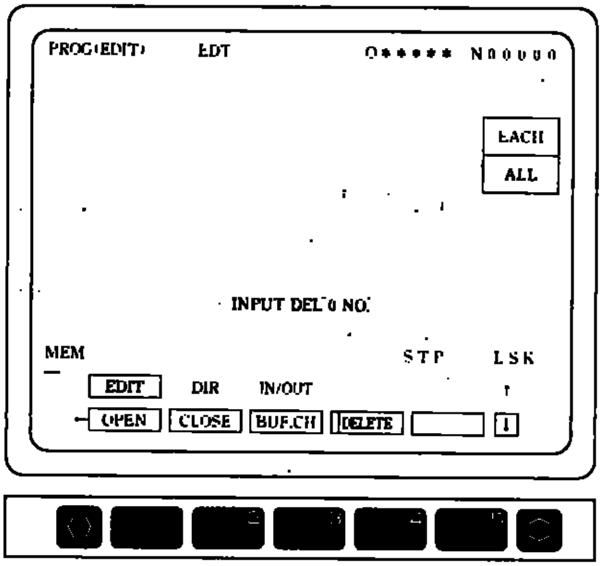
NOTE If the specified O-number is not found, "NOT FOUND O NO!" is displayed.

(b) ALL

1 Depress the DELETE function soft key.

2 Depress the DELETE function soft key again. The pop-up menu appears.

3 Depress the cursor key ▼ to select an O-number.



INPUT DEL O NO.

Fig. 3.4.24 Specified Deleting Menu Screen

4 Depress the WR key. The pop-up menu for selecting the deleting menu disappears. Then "DELETE ALL? (Y/N)" is displayed.

- Depress Y to delete. Then "DELETE COMPLETE" is displayed.
- Depress N not to delete the program.

NOTE
1. If the specified O-number is called for editing, the program is automatically saved and then deleted.
2. If the specified O-number is called for running, "O* * * * *" is displayed for the running program, and the status before any program is deleted is restored.

3.4.5.2 Part program list job (job soft key DIR)

Depress the DIR function soft key. The following functions are available:

- **Display part program entry list** : Displays a list of part program numbers that are stored in NC memory.
- **Call part program** : Calls a part program to edit in the editing job.
- **Erase part program** : Erases a part program.
- **Rename part program** : Alters a part program number.
- **Copy part program** : Copies a part program.

Details of the above functions are explained in the following:

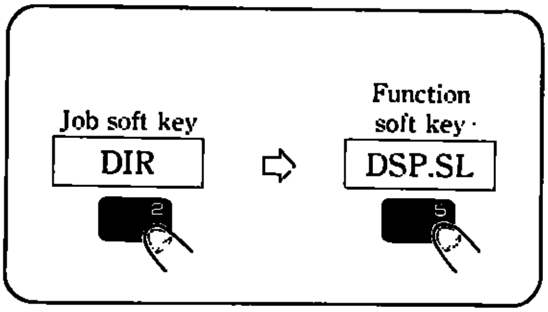
(1) Display part program entry list

This function displays a list of part program numbers that are stored in NC memory.

Comments and dates, or comments and the numbers of actually used characters can be added, or O-numbers only can be displayed. Use the DSP.SL function soft key to select the display items.

Up to 10 program numbers can be displayed on a full screen with comments. Up to 50 program numbers can be displayed on a full screen without comments.

KEYPOINT of operation

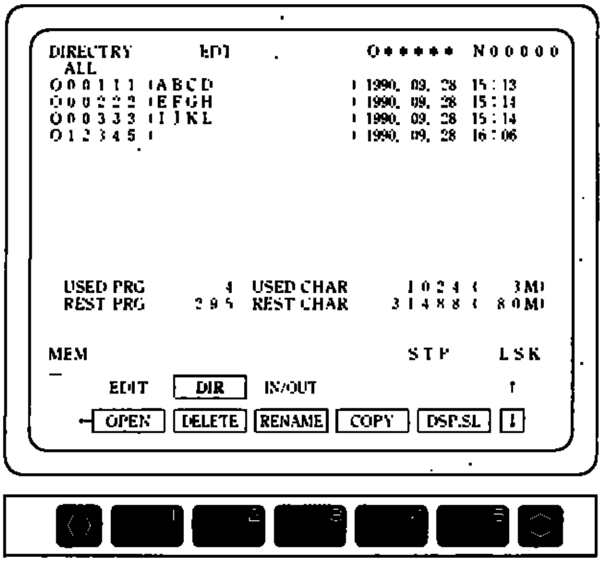


1. Depress the DIR job soft key.

   A list of part program numbers is displayed.

2. Depress the DSP.SL function soft key.

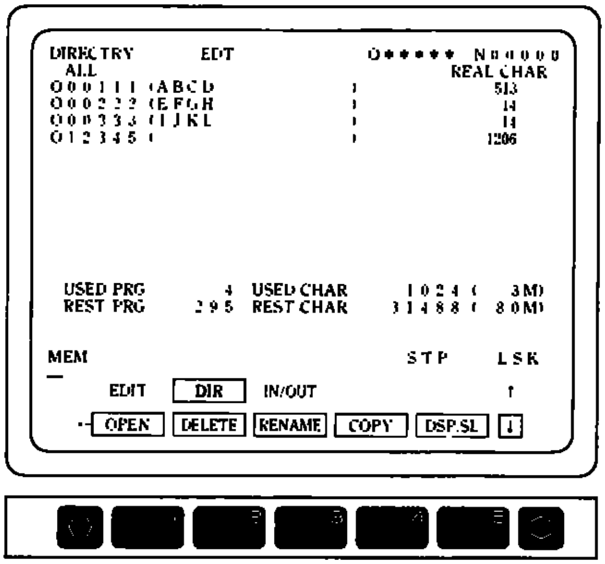
   Selections of display are switched cyclically from with comment and data to with comment and number of actually used characters to O-numbers only. If a list of subprograms is displayed, only O-numbers can be displayed.



```
DIRECTRY        EDT              O******  N00000
 ALL
O00111 (ABCD                ) 1990. 09. 28  15:13
O00222 (EFGH                ) 1990. 09. 28  15:14
O00333 (IJKL                ) 1990. 09. 28  15:14
O12345 (                    ) 1990. 09. 28  16:06




 USED PRG       4  USED CHAR     1024 (   3M)
 REST PRG     295  REST CHAR    31488 (  80M)

MEM                                STP    LSK
_
     EDIT  [ DIR ]  IN/OUT                 ↑
   ←[OPEN][DELETE][RENAME][COPY][DSP.SL][↓]
```

Fig. 3.4.25 Display of Part Program List with Comment and Data

Depressing the DSP.SL function soft key changes the screen to the following:

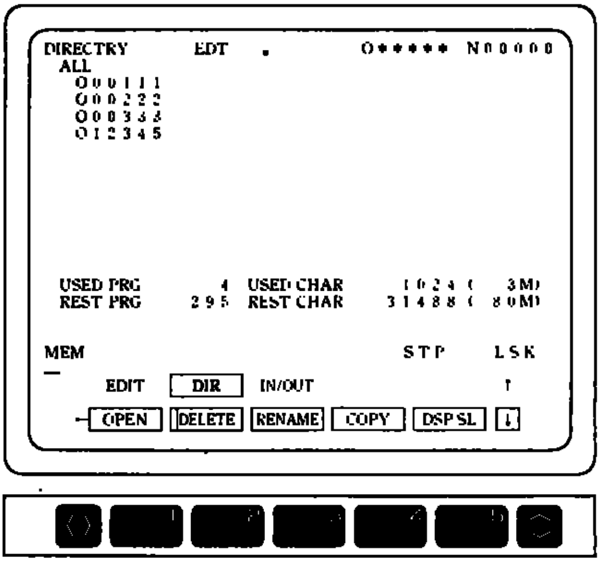


```
DIRECTRY        EDT              O******  N00000
 ALL                                   REAL CHAR
O00111 (ABCD                )              513
O00222 (EFGH                )               14
O00333 (IJKL                )               14
O12345 (                    )             1206




 USED PRG       4  USED CHAR     1024 (   3M)
 REST PRG     295  REST CHAR    31488 (  80M)

MEM                                STP    LSK
_
     EDIT  [ DIR ]  IN/OUT                 ↑
   ←[OPEN][DELETE][RENAME][COPY][DSP.SL][↓]
```

Fig. 3.4.26 Display of Part Program List with Comment and Number of Actually used Characters

NOTE The number of actually used characters is the number of characters actually used by the program. The number differs from the number of used characters or the number of remaining characters. The number of used characters and the number of remaining characters are multiples of 256 or 1024, depending on the hardware, and indicate memory size assigned to the characters.

Depressing the DSP.SL function soft key displays the following screen:



```
DIRECTRY      EDT    .         O******  N00000
 ALL
   O00111
   O00222
   O00333
   O12345




 USED PRG        4   USED CHAR      1024 (   3M)
 REST PRG      295   REST CHAR     31488 (  80M)

MEM                                 STP    LSK
—
      EDIT   [ DIR ]  IN/OUT                 ↑
   ←[ OPEN ] [DELETE] [RENAME] [ COPY ] [DSP SL] [↓]
```

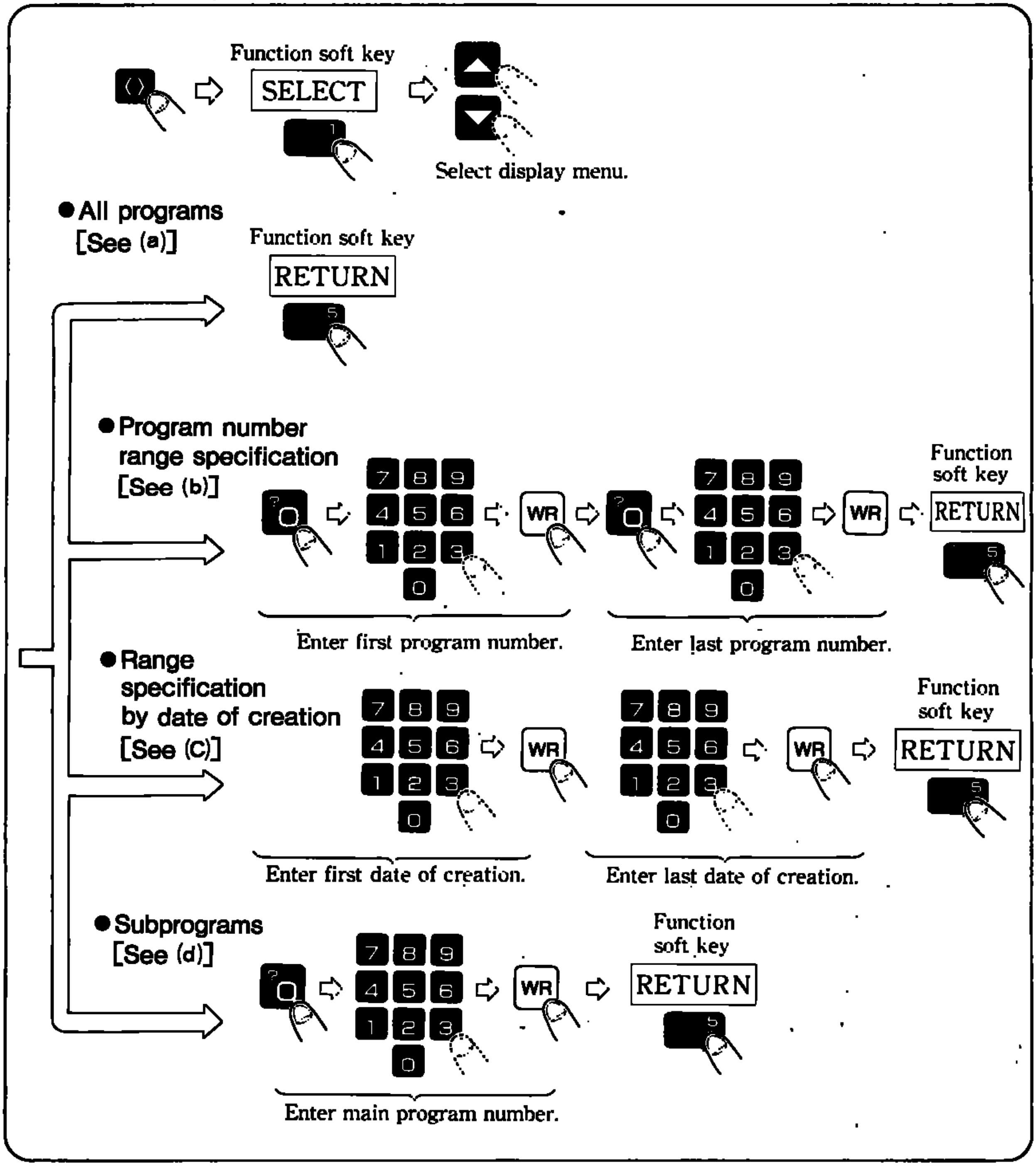
Fig. 3.4.27 Display of Part Program List by O-numbers only

There are the following four display method menus. Use the DSP.SL function soft key to select one of them:

- **All programs** : Display all part program numbers that are stored.
- **Program number range specification** : Display part program numbers in the specified range.
- **Range specification by date of creation** : Display part program numbers in the specified range of date of creation.
- **Subprograms** : Display all part programs that are called by the specified main program.

NOTE When subprograms are displayed, the DSP. SL key is invalid.

KEYPOINT of operation

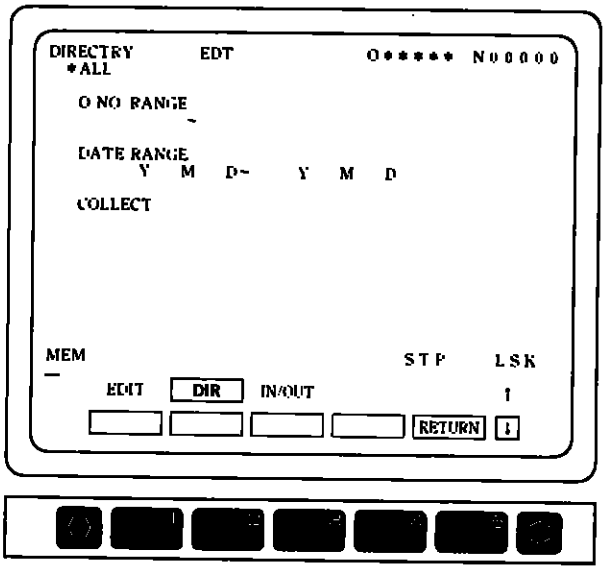


(a) **Selecting all programs**

1 Set display selection to "all programs" which is the initial state.

2 Depress the RETURN function soft key.

The screen returns to the list display and displays a list of all program numbers.



```
DIRECTRY          EDT                  O******  N00000
 *ALL
   O NO RANGE
              -
   DATE RANGE
        Y    M    D-      Y    M    D
   COLLECT




MEM                                       STP      LSK
-
     EDIT     [ DIR ]   IN/OUT                      ↑
    [      ] [      ] [      ] [      ] [RETURN] [↓]
```

Fig. 3.4.28 Selecting "all programs"

(b) Selecting program number range specification

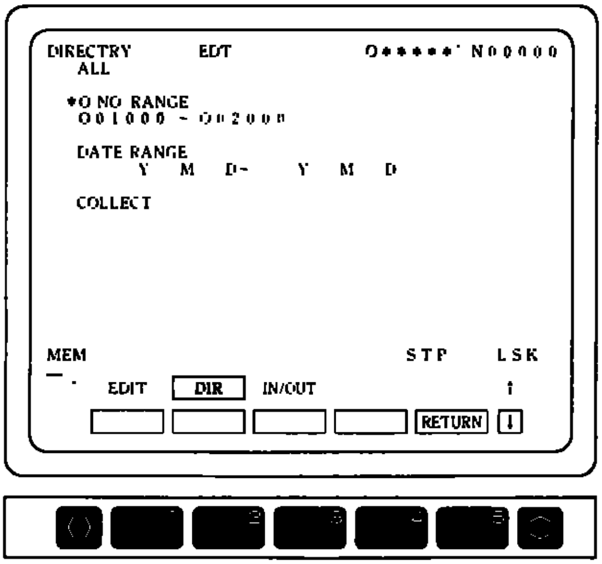
1. Set display selection to "program number range specification".
2. Enter the first program number.

   (Example : 0 1 0 0 0 WR)
3. Enter the last program number.

   (Example : 0 2 0 0 0 WR)
4. Depress the RETURN function soft key.

The screen returns to the list display and displays a list of program numbers between the first and last program numbers.

NOTE If the input range of program numbers is invalid, a warning is displayed.



```
DIRECTRY        EDT                 O******' N00000
  ALL

 *O NO RANGE
  O01000 - O02000

  DATE RANGE
       Y   M   D-     Y   M   D

  COLLECT




MEM                                  STP     LSK
      EDIT    DIR    IN/OUT                    ↑
                                     RETURN    ↓
```

Fig. 3.4.29 Specification of Range of Program Numbers

(c) Range specification by date of creation

1. Set display selection to "range specification by date of creation".
2. Enter the first date of creation.

   (Example : 1 9 9 0 . 1 . 1 WR)

   Entering the year only sets the first day of the year.
   Entering the year and the month only sets the first day of that month of the year.
3. Enter the last date of creation.

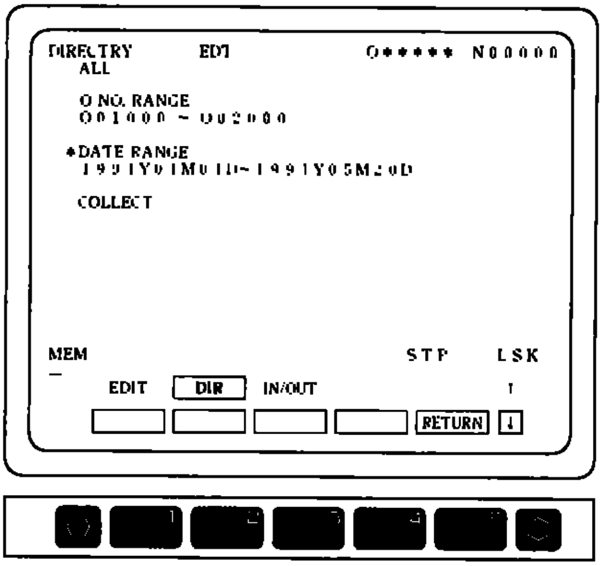
   (Example : 1 9 9 0 . 5 . 2 0 WR)

   Entering the year only sets the last day of the year.
   Entering the year and the month only sets the last day of that month of the year.
4. Depress the RETURN function soft key.

   5

   The screen returns to the list display and displays a list of program numbers created in the specified period.

NOTE If the input range of date of creation is invalid. a warning is displayed.



```
DIRECTRY        EDT                O******  N00000
   ALL

   O NO. RANGE
   O01000 ~ O02000

  *DATE RANGE
   1991Y01M01D~1991Y05M20D

   COLLECT




MEM                                  STP     LSK
—
       EDIT   [ DIR ]  IN/OUT                 ↑
      [     ][     ][     ][     ][RETURN][↓]
```

Fig. 3.4.30 Specification of Range of Data of Creation

(d) Subprograms

1. Set display selection to "subprograms".
2. Enter the main program number.

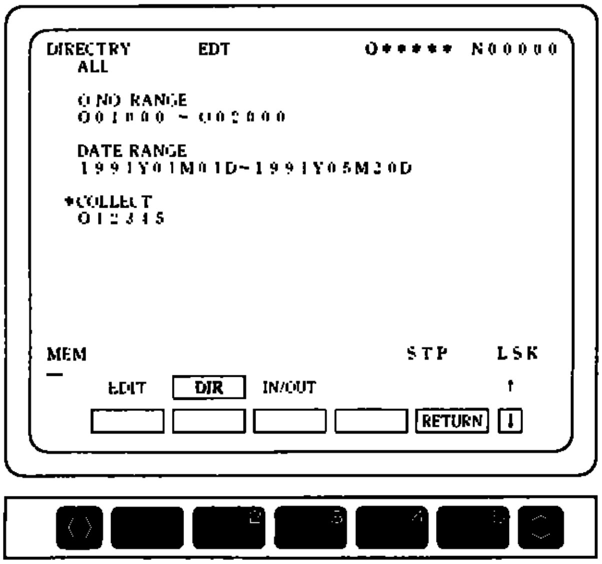
   (Example : O 1 2 3 4 5 WR)

3. Depress the RETURN function soft key.

The screen returns to the list display and displays a list of programs that are called from the main program.

NOTE 1. When subprograms are displayed, only O-numbers can be displayed. COMMENT function soft key becomes invalid.

2. Programs that are called as subprograms but not stored are indicated with their numbers blinking.



```
DIRECTRY          EDT                     O******  N00000
   ALL

   O NO RANGE
   O01000 ~ O02000

   DATE RANGE
   1991Y01M01D~1991Y05M20D

  *COLLECT
   O12345

MEM                                       STP    LSK
```

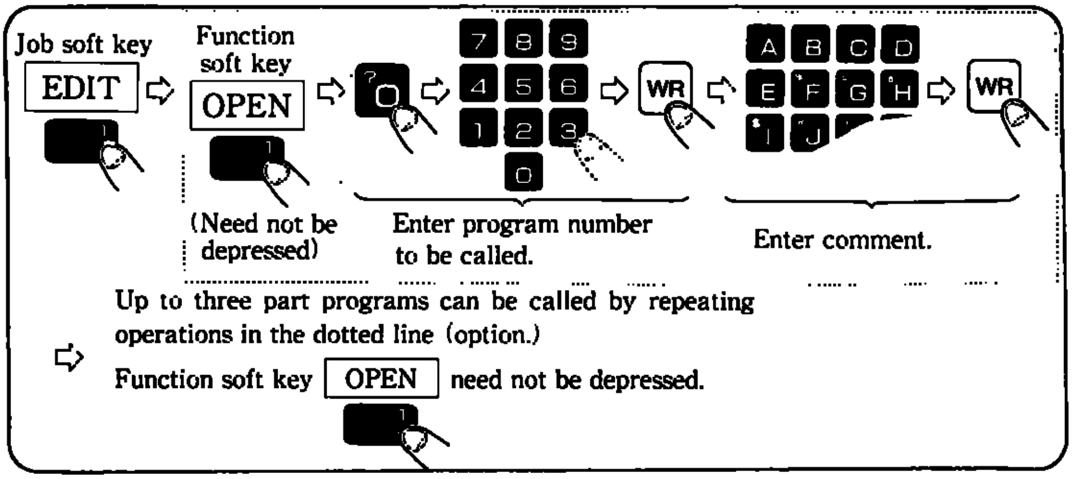
Fig. 3.4.31 Specification of Subprograms

(2) Calling part program

From the program list, part programs can be called onto the editing screen by simple operation. After a stored program number is entered, the screen changes to the editing screen and the program is ready for editing. After a program number that is not yet stored is entered, comment must be entered to create a new part program before the screen changes to the editing screen and the program becomes ready for editing.

Call part programs in the same way as calling a part program for editing. In the program list screen, programs can be called by depressing function soft keys or by entering the program number and depressing the WR key.

KEYPOINT of operation

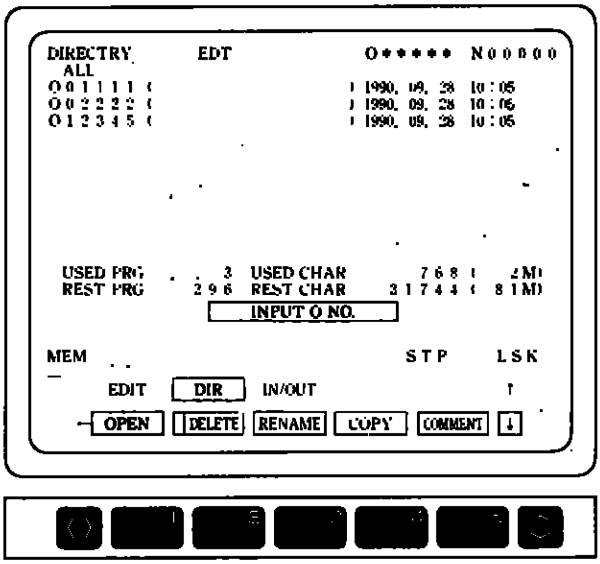


Up to three part programs can be called by repeating operations in the dotted line (option.)

Function soft key OPEN need not be depressed.

NOTE

1. New program numbers cannot be created in the editing prohibited state. Clear the editing prohibited state before calling programs.
2. Programs being executed can also be called, but cannot be edited.

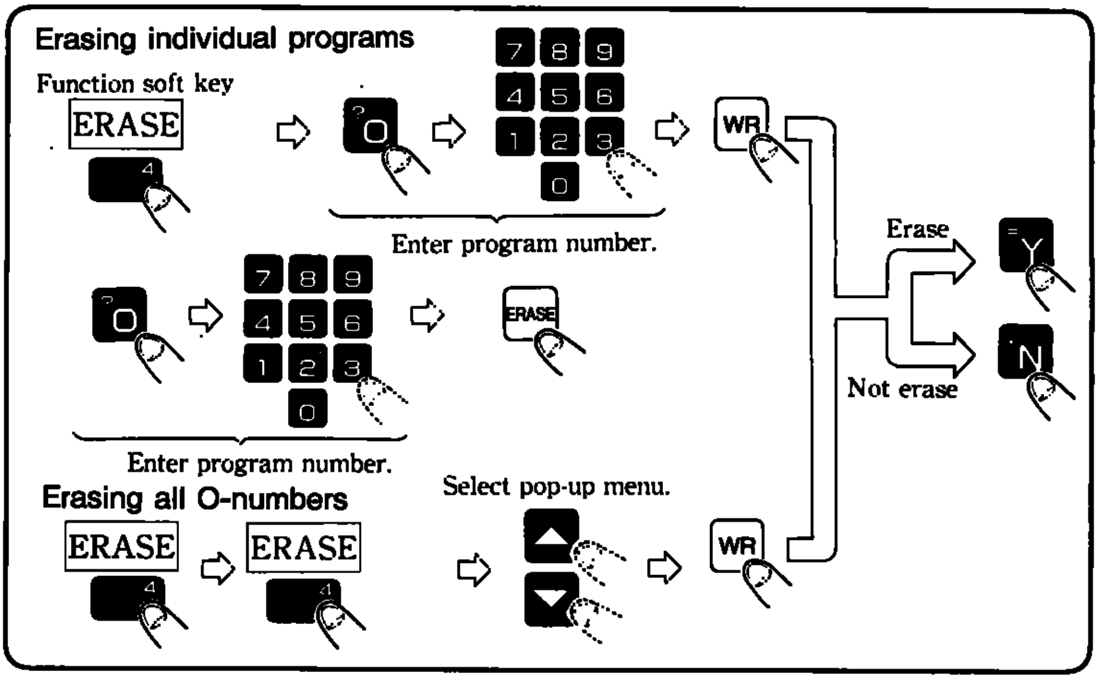


```
DIRECTRY        EDT                  O******   N00000
  ALL
O01111 (                    ) 1990. 09. 28  10:05
O02222 (                    ) 1990. 09. 28  10:06
O12345 (                    ) 1990. 09. 28  10:05



  USED PRG        3  USED CHAR       768 (    2M)
  REST PRG      296  REST CHAR     31744 (   81M)
                     INPUT O NO.

MEM                                 STP     LSK

      EDIT    DIR    IN/OUT                  ↑
    OPEN  DELETE  RENAME  COPY  COMMENT  ↓
```

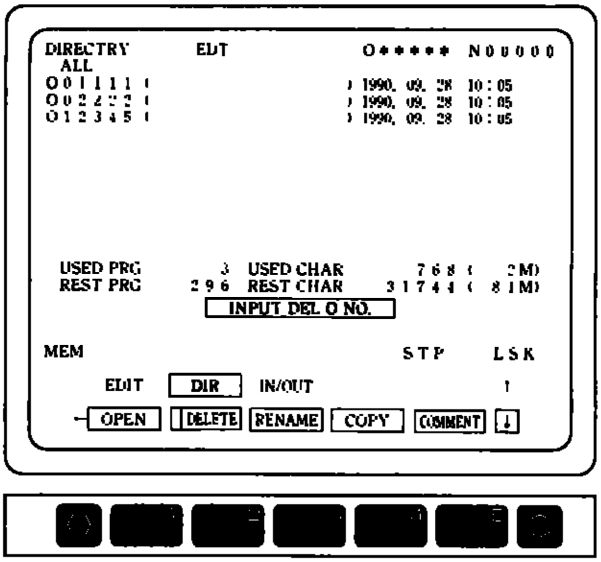
Fig. 3.4.32 Calling Part Program

(3) From the program list, part programs can be erased by simple operation. The operation is similar to erasing part programs in editing job. In the program list screen, programs can be erased by depressing the function soft key or by entering the program number and depressing the ERASE key.

KEYPOINT of operation



NOTE
1. Part programs cannot be erased in the editing prohibited state. Clear the editing prohibited state before erasing programs.
2. Programs being executed cannot be erased. Wait until the program terminates.



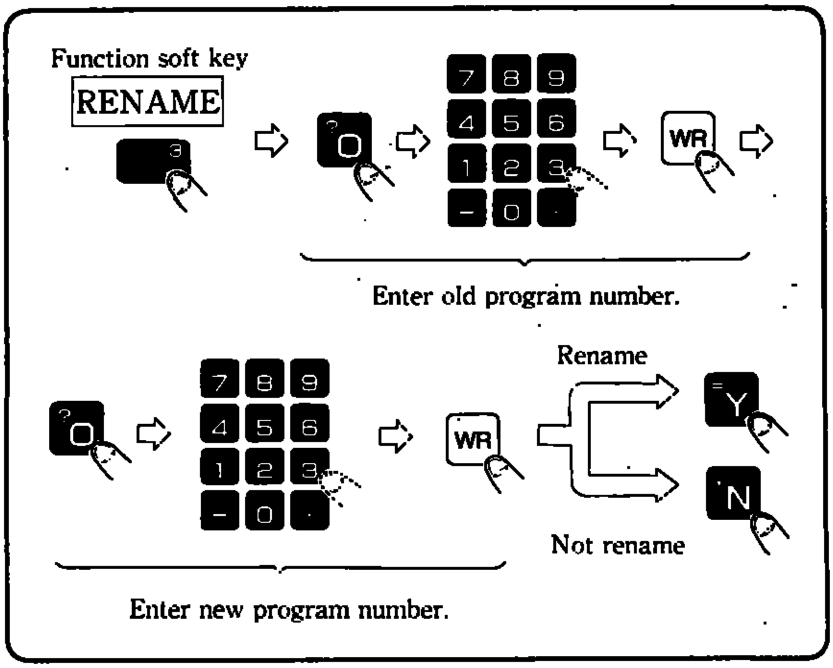
```
DIRECTRY        EDT                 O******  N00000
  ALL
O01111 (                      ) 1990. 09. 28  10:05
O02222 (                      ) 1990. 09. 28  10:05
O12345 (                      ) 1990. 09. 28  10:05





 USED PRG       3  USED CHAR       768 (   2M)
 REST PRG     296  REST CHAR     31744 (  81M)
               INPUT DEL O NO.

MEM                                 STP    LSK

     EDIT    DIR    IN/OUT                   ↑
  ← OPEN  DELETE  RENAME  COPY  COMMENT  ↓
```

Fig. 3.4.33 Erasing Part Program

(4) Renaming part programs

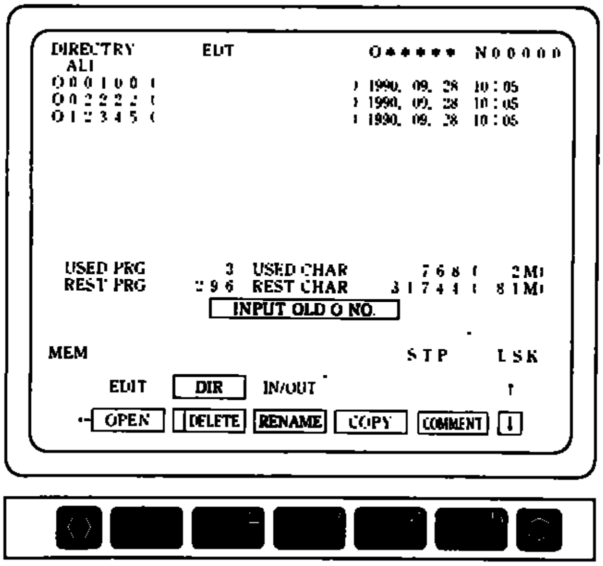
Part program numbers can be changed by the following operation:

KEYPOINT of operation



1 Depress the RENAME function soft key.

A message requests : "Enter old program number."



```
DIRECTRY        EDT                  O******  N00000
  ALL
O00100 (                           ) 1990. 09. 28  10:05
O02222 (                           ) 1990. 09. 28  10:05
O12345 (                           ) 1990. 09. 28  10:05



   USED PRG        3  USED CHAR      7680 (   2M)
   REST PRG      296  REST CHAR    317440 (  81M)
                   INPUT OLD O NO.

MEM                                      STP     LSK
```

Fig..3.4.34 Renaming Part Program

2 Enter the old program number.

(Example : O 1 0 0 WR)

A message requests : "Enter new program number."

3 Enter the new program number.

(Example : O 2 0 0 WR)

4 A message asks : "Are you sure to rename the program (Y/N)?"

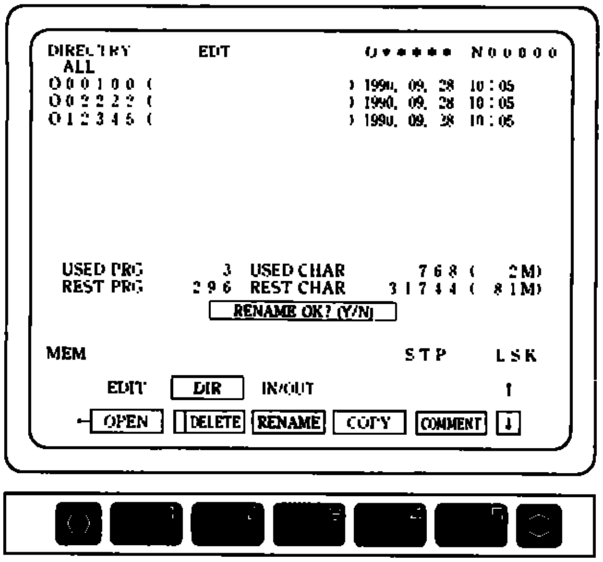
- To rename the program, depress Y .
- Not to rename the program, depress N .

For a program with the program number at the beginning, whether the program number is automatically edited when the program is renamed or copied [see (5)], can be determined by the following parameter :

pm3005 D4 = 0 : Does not edit the O-number automatically when the program is renamed, copied, or stored.

pm3005 D4 = 1 : Edits the O-number automatically when the program is renamed, copied, or stored.

NOTE The data of the program is not updated when the program number is renamed.



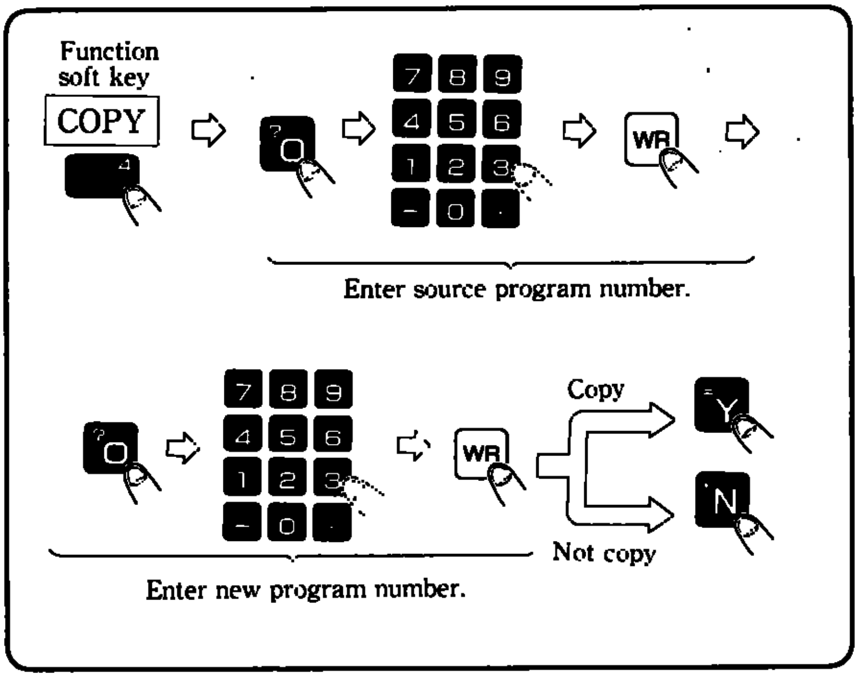
```
DIRECTRY         EDT              O******  N00000
  ALL
O00100 (                   ) 1990. 09. 28  10:05
O02222 (                   ) 1990. 09. 28  10:05
O12345 (                   ) 1990. 09. 28  10:05



 USED PRG       3  USED CHAR      768 (   2M)
 REST PRG     296  REST CHAR    31744 (  81M)
              RENAME OK? (Y/N)

MEM                               STP     LSK
```

Fig. 3.4.35 Part Program Rename Selection Screen

(5) Copying part programs

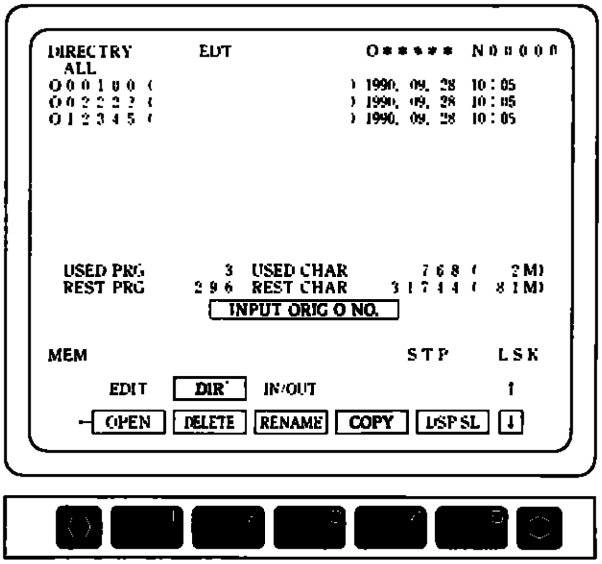
Part programs can be copied by the following operation:

KEYPOINT of operation



■ Depress the COPY function soft key.

A message requests : "Enter source program number."



```
DIRECTRY       EDT                O******  N00000
  ALL
O00100 (                    ) 1990. 09. 28  10:05
O02222 (                    ) 1990. 09. 28  10:05
O12345 (                    ) 1990. 09. 28  10:05

  USED PRG         3  USED CHAR       768 (   2M)
  REST PRG       296  REST CHAR     31744 (  81M)
                    INPUT ORIG O NO.

MEM                                 STP    LSK

      EDIT   [ DIR ]  IN/OUT                 1
  -[ OPEN ][ DELETE ][ RENAME ][ COPY ][ DSP SL ][ ↓ ]
```

Fig. 3.4.36 Copying Part Program

2 Enter the source program number.

(Example : 0 1 0 0 WR)

A message requests : "Enter copy program number."

3 Enter the new program number to be created.

(Example : 0 2 0 0 WR)

4 A message asks : "Were you sure to copy the program (Y/N)?"

- To copy the program, depress Y
- Not to copy the program, depress N

NOTE The new program created by copying has a new date of creation.

3.4.5.3 Part program I/O verification ($\overset{\text{job soft key}}{\boxed{\text{IN/OUT}}}$)

Depress the IN/OUT job soft key. The following functions are available for verifying input and output part programs:

(1) **Part program input function** : Stores part programs from external equipment to controller memory.

(2) **Part program output function** : Outputs part programs from controller memory to external equipment.

(3) **Part program verification function** : Collates part programs in external equipment with that in controller memory.

(4) **I/O equipment setup function** : Determines data I/O interface baud rate, stop bit length, and use of the control code.

(5) **Other reverse functions** : Enables operations such as temporary stop, interruption, restart, etc, by using ☐, ☐, ☐, ☐ or BGREST during I/O or verification.

Details of the above functions are described in the following.

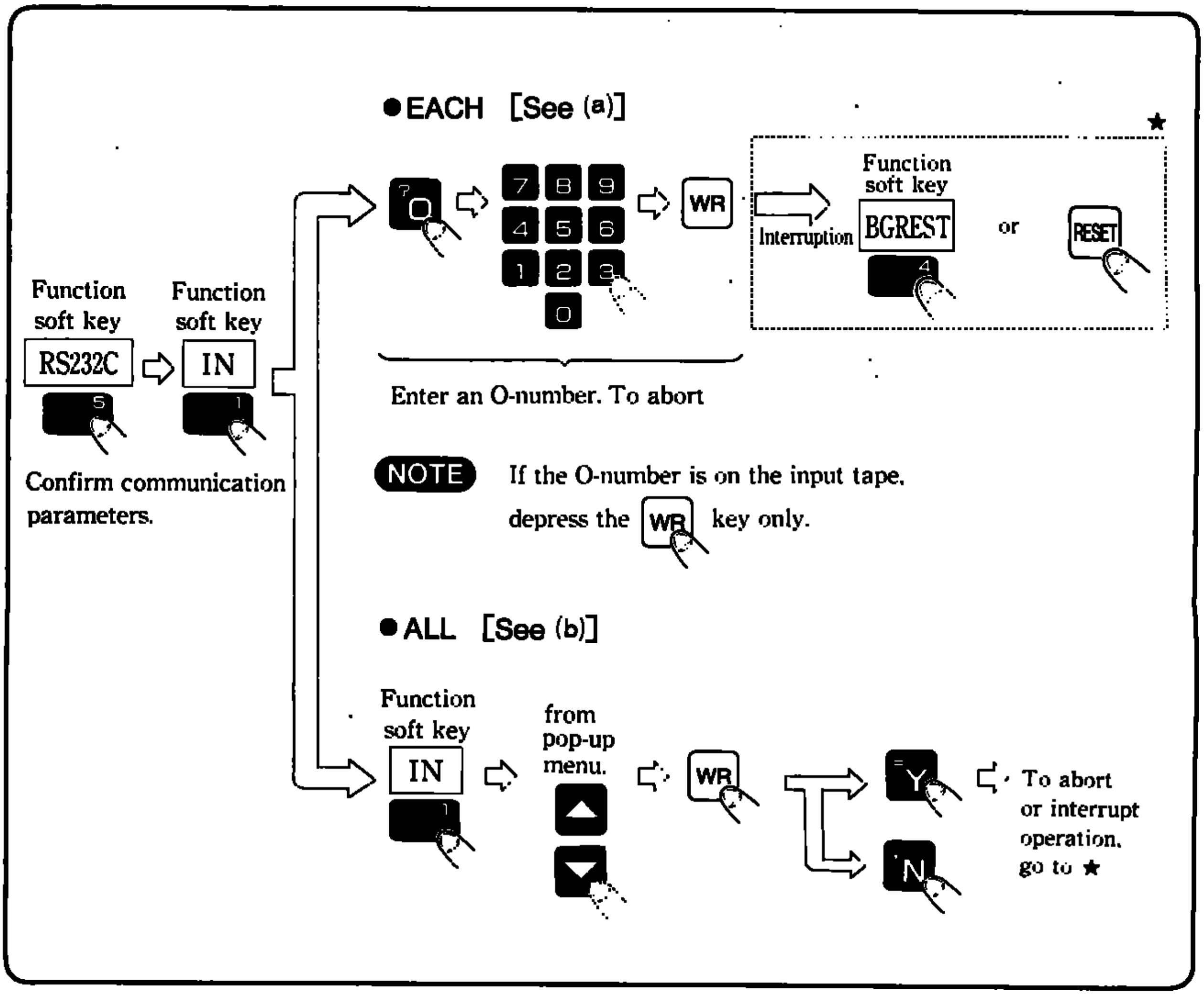
NOTE In the editing prohibited status, part programs cannot be input, output, or verified. Clear the editing prohibited status in advance.

(1) Part program input function

There are EACH and ALL input menus, which can be selected from the pop-up menu.

- EACH : Part programs are entered one by one.
- ALL : All O-numbers on a single tape are entered.

KEYPOINT



Depress the RS232C function soft key to confirm communication parameters.

(a) **EACH**

1 Enter the O-number of the part program to be input.

(For example, O , 1 , 1 , 1 , WR)

- If the O-number is on the input tape, the number need not be entered from the screen. Depress the WR key only.
- Even if the O-number of the program is on the input tape, the program can be entered with a new O-number. To do this, enter the new O-number. Whether or not to change the O-number at the beginning of the part program can be determined by the parameter:

  Pm3005 D4 = 0 Does not edit O-number automatically at program rename/copy/save.

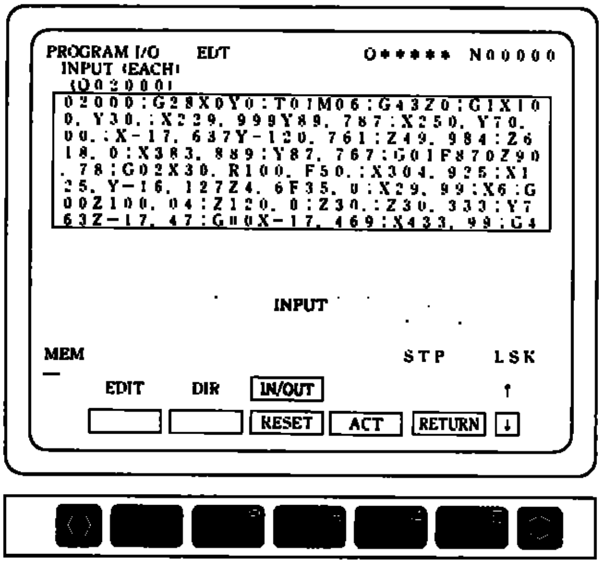
  Pm3005 D4 = 1 Edits O-number automatically at program rename/copy/save.

- If an input O-number has already been registered, a warning can be issued or the existing O-number can be deleted. Select either by parameter:

  Pm3005 D0 = 0 : If the O-number input from tape is already registered, issue a warning.

  Pm3005 D0 = 1 Deletes the existing program and saves the input program if the O-number input from tape is already registered.

2 Program input is started and the part program number input screen shown in Fig. 3.4.37 is displayed:



```
PROGRAM I/O     EDIT                 O******  N00000
  INPUT (EACH)
  (O02000)
02000:G28X0Y0:T01M06:G43Z0:G1X10
0. Y30.:X229. 999Y89. 787:X250. Y70.
00.:X-17. 637Y-120. 761:Z49. 984:Z6
18. 0:X383. 889:Y87. 767:G01F870Z90
. 78:G02X30. R100. F50.:X304. 925:X1
25. Y-16. 127Z4. 6F35. 0:X29. 99:X6:G
00Z100. 04:Z120. 0:Z30.:Z30. 333:Y7
63Z-17. 47:G00X-17. 469:X433. 99:G4

                    INPUT

MEM                                   STP    LSK
```

Fig. 3.4.37 Part Program Number Input Screen

By switching parameter, no part program display can be enabled.

Pm0007 D5= 0 : Does not perform I/O verification display.

Pm0007 D5= 1 : Performs I/O verification display.

3 To cancel the input, depress the BGREST function soft key or the RESET key.

Difference between BGREST and RESET :

- BGREST : Cancels only tape I/O verification. Does not affect operation.
- RESET : Resets all NC movements.

4 An input operation is completed when a completion M code (M02, M30, or M99) or code % is found on the input tape.
After completion, "INPUT COMPLETE" is displayed.

NOTE The RETURN function soft key is disabled until input operation is completed.

The process keys are effective throughout the operation.

Whether or not a completion M code is regarded as the end of the program can be specified by the following parameter:

Pm3005 D3 = 0 : Does not assume the completion M codes (M02, M30, and M99) as the end of the program.

Pm3005 D3 = 1 : Assumes the completion M codes (M02, M30, and M99) as the end of the program.

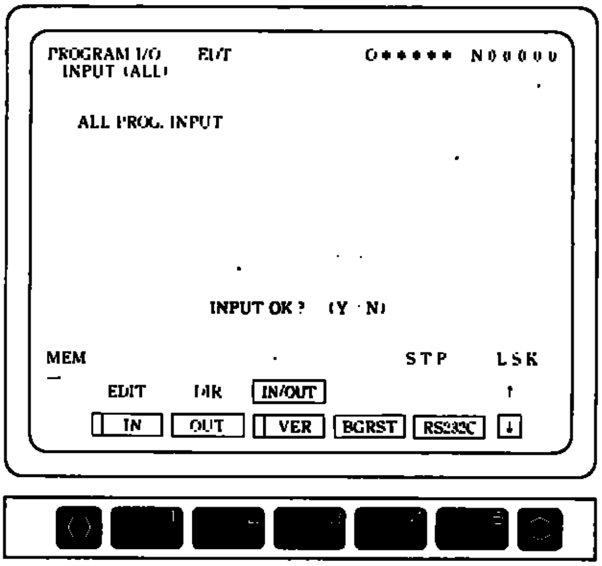
Whether or not space should be deleted at verification of program input, can be specified by the following parameter setting.

Pm0020 D0 = 0 : Space is deleted.

Pm0020 D0 = 1 : Space is not deleted.

●ALL

1 "ALL PROG. INPUT" and "INPUT OK? (Y/N)" are displayed as shown in Fig. 3.4.38:



```
PROGRAM I/O     EDIT              O******   N00000
 INPUT (ALL)

   ALL PROG. INPUT



              INPUT OK ?   (Y/N)

MEM                            STP     LSK
```

Fig. 3.4.38 ALL Input Screen

2 ●To input programs, depress Y . Program input is started, the screen changes, and the contents of the part programs being input are displayed as shown in Fig. 3.4.37.

●To cancel input operation, depress N .

3 To halt or abort input operation, do the same as in EACH.

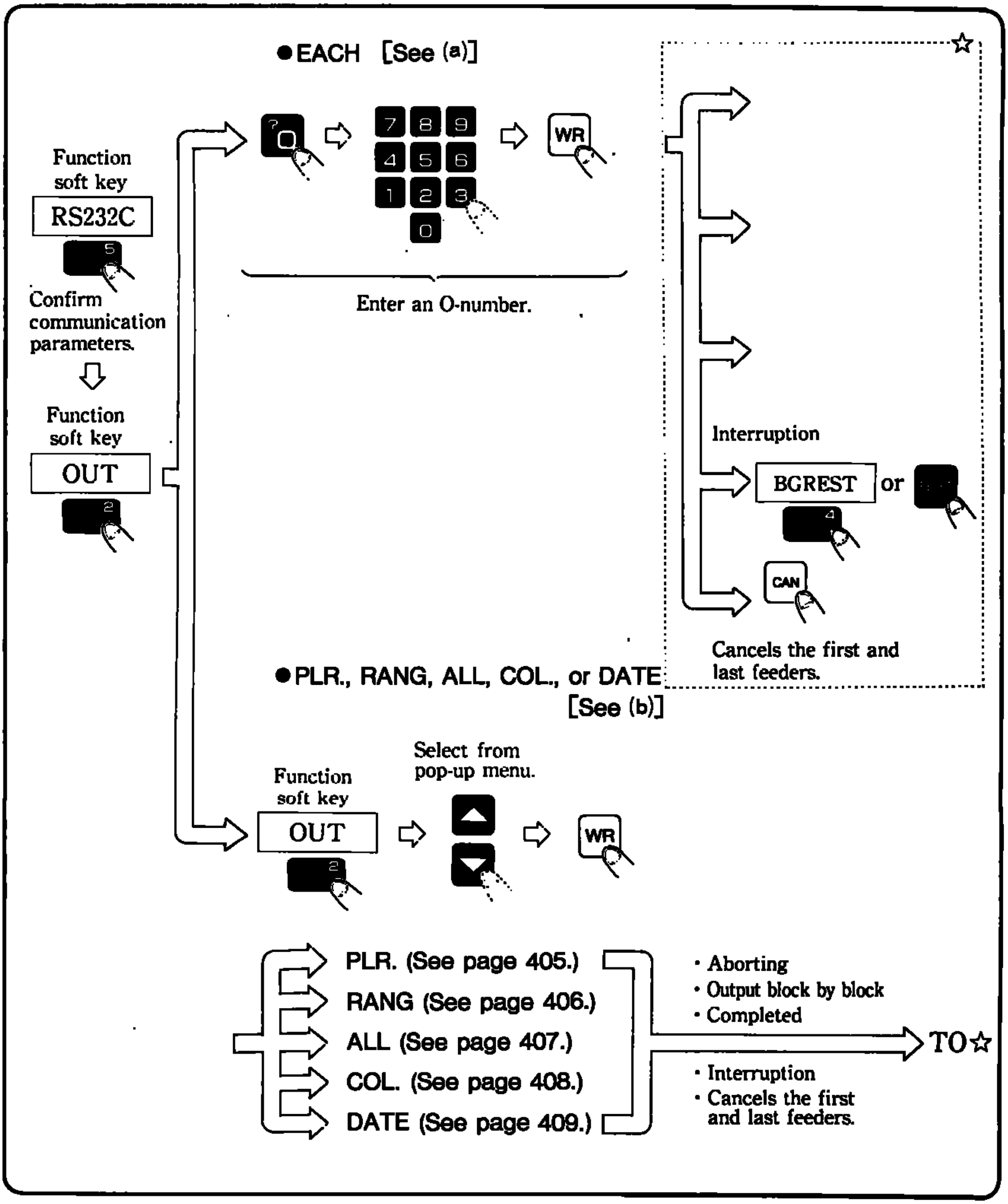
4 The ALL input operation is completed when code % is input. When the operation is completed, "INPUT COMPLETE" is displayed.

(2) **Part program output function**

The following six output menus can be selected from the pop-up menu:

- **EACH** : Part programs are output one by one.
- **PLR. (plural)** : Two or more specified part programs are output.
- **RANG (range)** : Part programs in the range between specified O-numbers are output.
- **ALL** : All part programs in controller memory are output.
- **COL. (correlated)** : Up to 500 subprograms that are called from a specified part program are output.
- **DATE** : Part programs created on a specified day are output.

KEYPOINT


●EACH [See (a)]
Function soft key
RS232C
5
Confirm communication parameters.
Function soft key
OUT
2
?
O
7 8 9
4 5 6
1 2 3
0
WR
Enter an O-number.
☆
Interruption
BGREST
4
or
CAN
Cancels the first and last feeders.
●PLR., RANG, ALL, COL., or DATE [See (b)]
Function soft key
OUT
2
Select from pop-up menu.
WR
PLR. (See page 405.)
RANG (See page 406.)
ALL (See page 407.)
COL. (See page 408.)
DATE (See page 409.)
· Aborting
· Output block by block
· Completed
TO☆
· Interruption
· Cancels the first and last feeders.

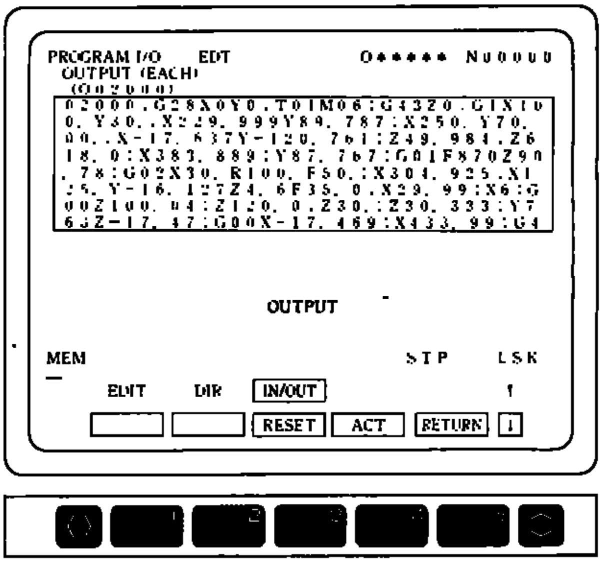
Depress the RS232C function soft key to confirm communication parameters.

(a) **EACH**

1 Enter the O-number of the program to be output.

(For example, O , 1 , 1 , 1 , WR)

2 Program output is started, and the screen changes and displays the contents of the part program being output, as shown in Fig. 3.4.39



```
PROGRAM I/O     EDT            O****** N00000
 OUTPUT (EACH)
  (O02000)
O2000.G28X0Y0.T01M06:G43Z0.G1X10
0.Y30..X229.999Y89.787:X250.Y70.
00..X-17.637Y-120.761:Z49.984.Z6
18.0:X383.889:Y87.767:G01F870Z90
.78:G02X30.R100.F50.:X304.925.X1
25.Y-16.127Z4.6F35.0.X29.99:X6:G
00Z100.04:Z120.0.Z30.:Z30.333:Y7
63Z-17.47:G00X-17.469:X433.99:G4
```

Fig. 3.4.39 O-number Output Screen

By switching parameter, no part program display can be enabled.

Pm0007 D5 = 0 : Does not perform I/O verification display.

Pm0007 D5 = 1 : Performs I/O verification display.

3 **<Aborting output>**

To abort output operation, depress the BGREST 4 function soft key or [illegible] key.

<Canceling the first and last feeder blocks of output data>

To cancel the feeder blocks at the top and the end of output data, depress the CAN key while the feeders are being output.

4 When output operation is completed, "OUTPUT COMPLETE" is displayed.

NOTE The RETURN 5 function soft key is disabled until output operation is completed.

The process keys are effective throughout the operation.

(b) **PLR., RANG, ALL, COL., and DATE**

1. Depress the OUT function soft key. "OUTPUT (EACH)" is displayed.

2. Depress the OUT function soft key again. The pop-up menu appears to provide choice of EACH, PLR., RANG, ALL, COL., or DATE. (See Fig. 3.4.40)

3. Depress cursor key ▲ or ▼ to select an output pop-up menu.

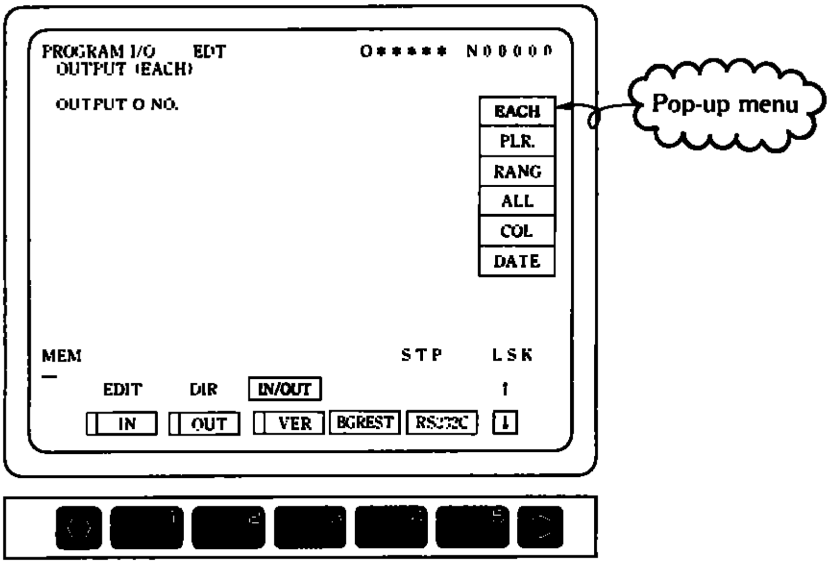
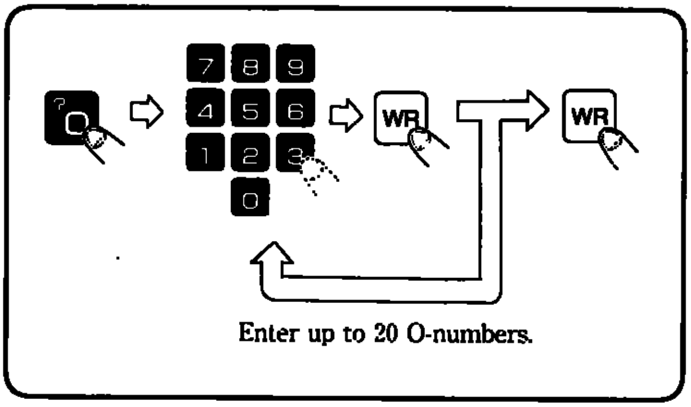


Fig. 3.4.40 Output Menu Select Screen

4. Depress the WR key. The output pop-up menu disappears and the specified output menu is displayed.

(i) **PLR.**

KEYPOINT



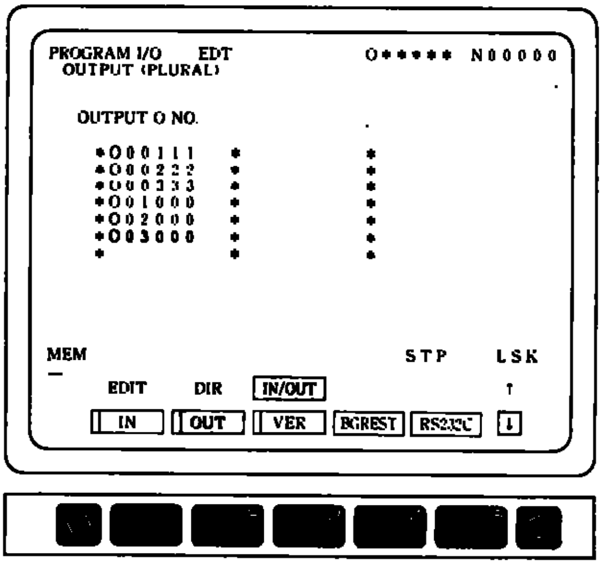
Enter up to 20 O-numbers.

1 Enter the O-number of the program to be output.

(For example, O 1 1 1 WR)

The cursor moves to the next line. Up to 20 O-numbers can be entered consecutively.

2 Depress the WR key.



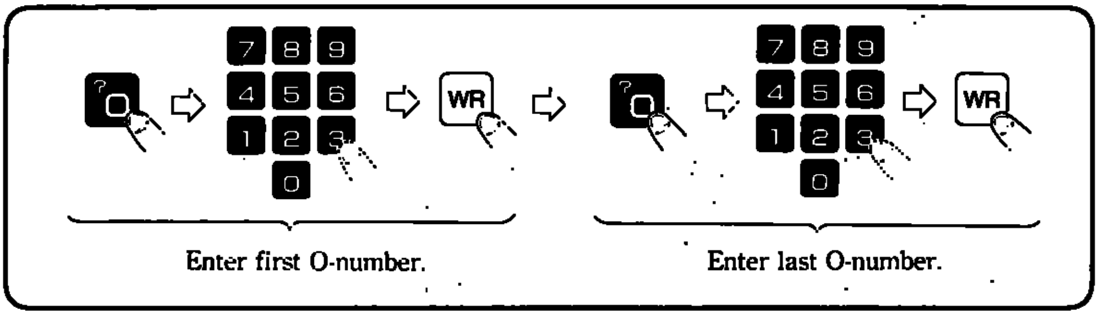
```
PROGRAM I/O     EDT                    O******  N00000
 OUTPUT (PLURAL)

  OUTPUT O NO.

    *O00111       *              *
    *O00222       *              *
    *O00333       *              *
    *O01000       *              *
    *O02000       *              *
    *O03000       *              *
    *             *              *



MEM                                    STP     LSK
—
     EDIT     DIR    [IN/OUT]                   ↑
    [ IN ]  [ OUT ]  [ VER ]  [BGREST] [RS232C]  [↓]
```

Fig. 3.4.41 O-number Input Screen

3 Program output is started and the contents of the part program being output are displayed. The specified programs are output on a single NC tape.

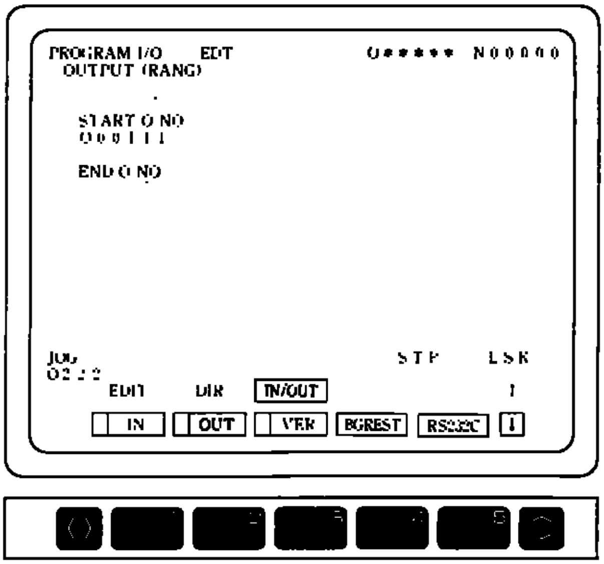
(ii) RANG

KEYPOINT



1 The operator is requested to enter the first and last O-numbers of the range to be output.

(For example, O 1 1 1 WR for the first O-number, and O 2 2 2 WR for the last O-number)



```
PROGRAM I/O     EDT                    O******  N00000
 OUTPUT (RANG)

  START O NO
  O00111

  END O NO




JOG                                      STP      LSK
O2:2
```

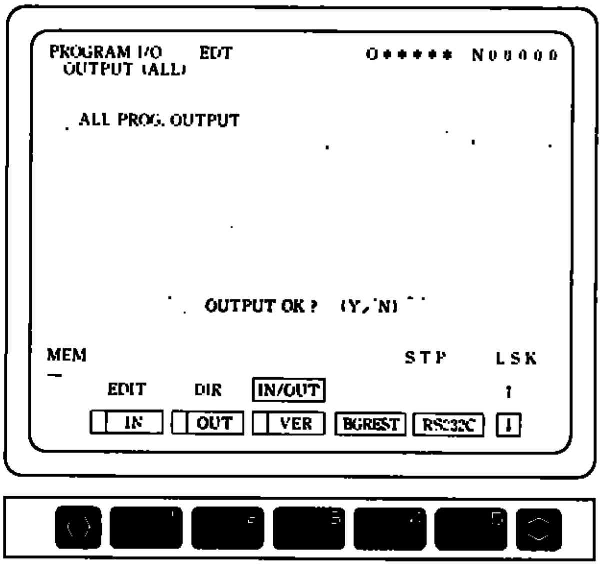
Fig. 3.4.42 Screen for Entering First and Last O-numbers

2 Program output is started and the contents of the part program being output are displayed. Part programs between the first and the last O-number are output onto a single NC tape. If no O-number is found in the range, a warning is issued.

(iii) **ALL**

1 "ALL PROG. OUTPUT" and "OUTPUT OK ?(Y/N)" are displayed.

To start output operation, depress Y



```
PROGRAM I/O     EDT              O****** N00000
 OUTPUT (ALL)

 ALL PROG. OUTPUT

            OUTPUT OK ?  (Y/N)

MEM                              STP     LSK
```

Fig. 3.4.43 ALL Output Screen

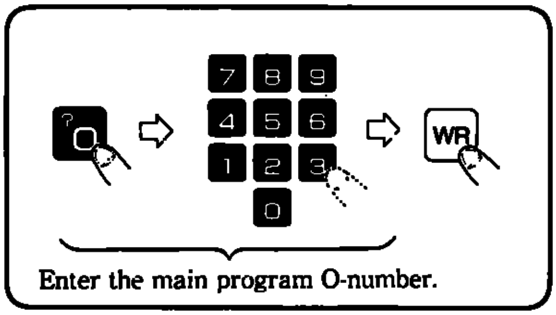
2 Program output is started and the contents of the part program being output are displayed. All the part programs that are registered are output onto a single NC tape.

3 To cancel output operation, depress N.

(iv) **COL.**

Specify the main program among part programs, and the subprograms, program copies, and macro programs that are called in the main program can be output.

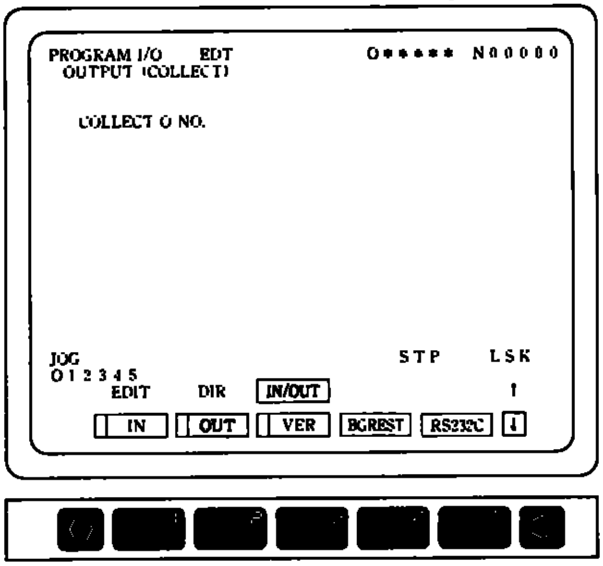
KEYPOINT



Enter the main program O-number.

1 The operator is requested to enter an O-number. Enter the O-number of the main program to output.

(For example, )



```
PROGRAM I/O     EDT                    O******  N00000
  OUTPUT (COLLECT)

    COLLECT O NO.
```

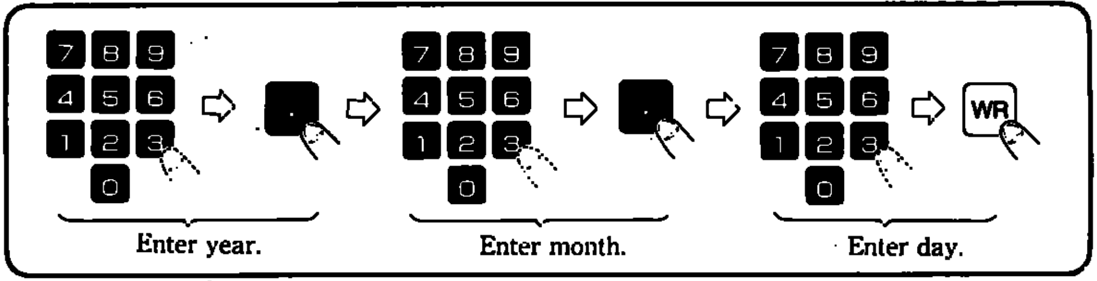
Fig. 3.4.44 O-number Input Screen

2 Program output is started and the contents of the part program being output are displayed. The main program and the function programs called by the main program output onto a single NC tape.

(v) **DATE**

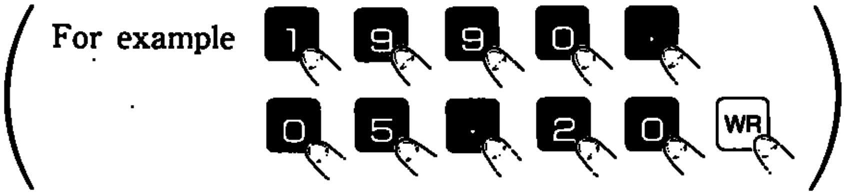
The date of the creation and registration of a part program is kept in controller internal memory. A part program can be output using this date.

KEYPOINT

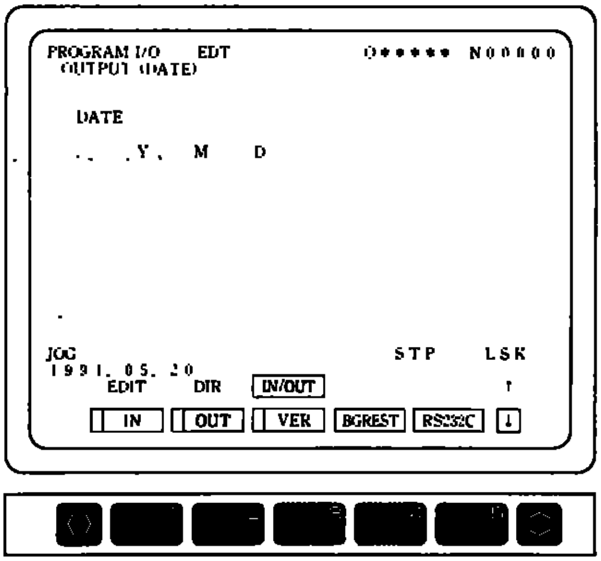


1 The operator is requested to enter the date. Enter the date of creation of the part program to output.

For example



If only the year is entered, all the programs created in that year are output. If only the year and the month are entered, all the programs created in that month of that year are output.



```
PROGRAM I/O    EDT                    O****** N00000
 OUTPUT (DATE)

  DATE

  . .   .Y.    M     D




JOG                                   STP    LSK
1991. 05. 20
```

Fig. 3.4.45 Date Input Screen

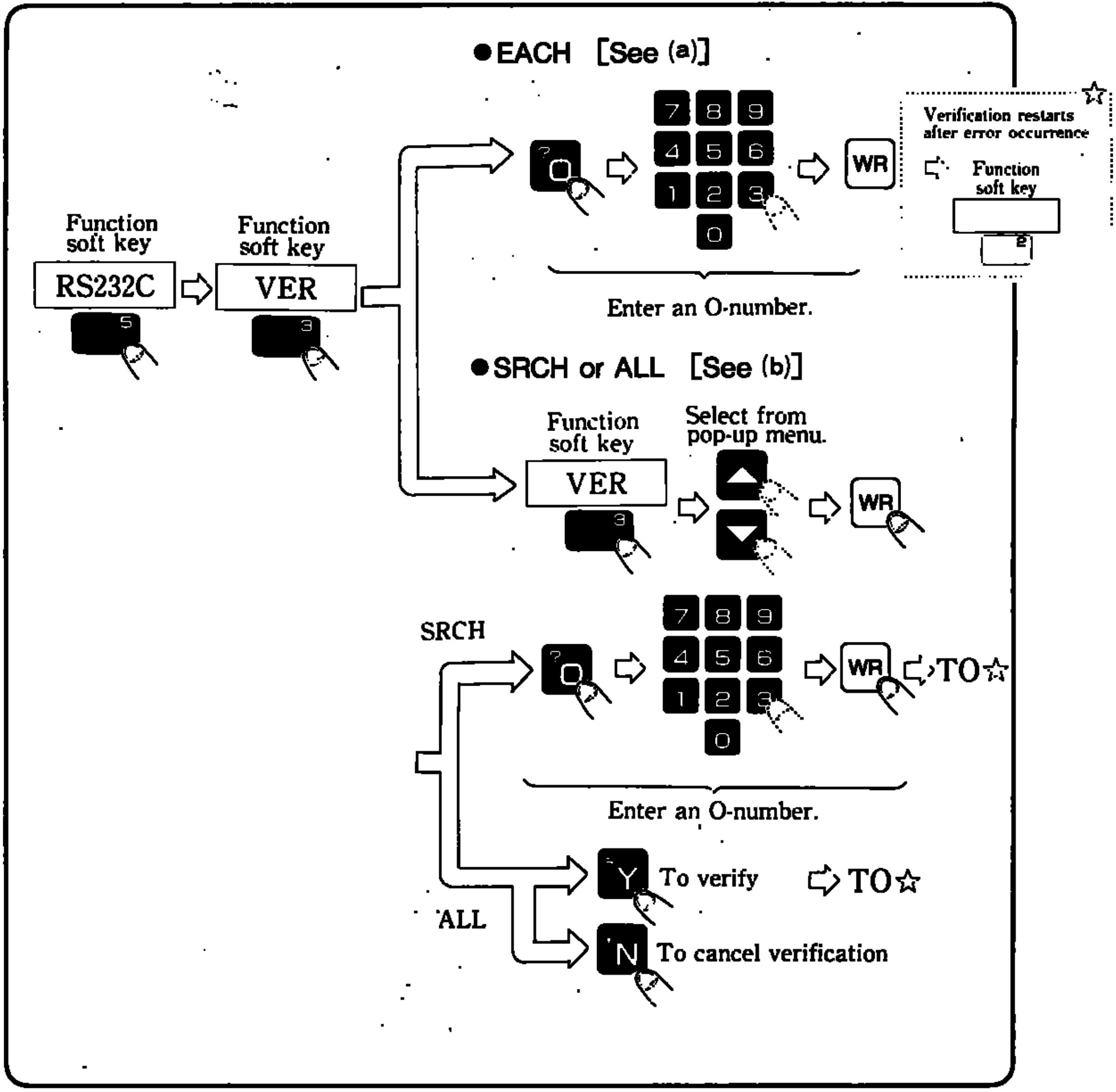
2 Program output is started and the contents of the part program being output are displayed. Part programs created on the specified date are output onto a single NC tape.

(3) Part program verification function

The following three verification menus can be selected from the pop-up menu:

- **EACH** : Part programs are verified one by one.
- **SRCH (search)** : If more than one O-number is on a single tape, a specified O-number is searched for automatically, and only that O-number is verified.
- **ALL** : All O-numbers on a single tape are verified.

KEYPOINT



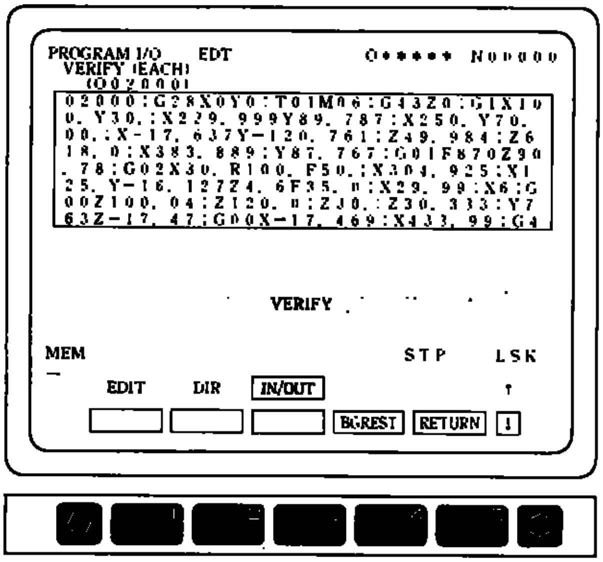
Depress the RS232C function soft key to confirm communication parameters.

(a) **EACH**

1. Enter the O-number of the part program to verify.

   (For example, O 1 1 1 WR)

2. Verification is started and the contents of the input tape part program to be collated with that in controller memory are displayed.



```
PROGRAM I/O     EDT                  O****** N00000
 VERIFY (EACH)
   (O02000)
O2000:G28X0Y0:T01M06:G43Z0:G1X10
0.Y30.:X229.999Y89.787:X250.Y70.
00.:X-17.637Y-120.761:Z49.984:Z6
18.0:X383.889:Y87.767:G01F870Z90
.78:G02X30.R100.F50.:X304.925:X1
25.Y-16.127Z4.6F35.0:X29.99:X6:G
00Z100.04:Z120.0:Z30.:Z30.333:Y7
63Z-17.47:G00X-17.469:X433.99:G4

                    VERIFY

MEM                                 STP    LSK
```

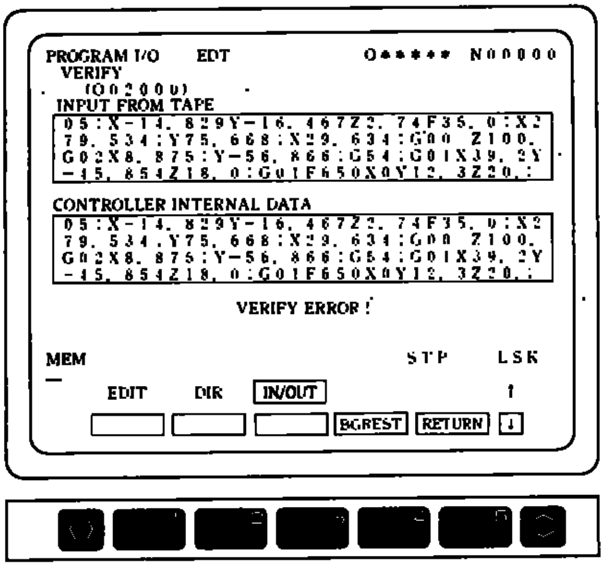
Fig. 3.4.46 O-number Verification Screen

By switching parameter, no part program display can be enabled.

Pm0007 D5= 0 : Does not perform I/O verification display.

Pm0007 D5= 1 : Performs I/O verification display.

3 If inconsistency between the memory contents and the input from external equipment is found, the screen is divided into two halves. The upper half displays the input from tape. The lower half displays controller internal data. The unmatched parts are shown highlighted.



```
PROGRAM I/O    EDT                 O******  N00000
 VERIFY
   (O02000)
 INPUT FROM TAPE
05:X-14.829Y-16.467Z2.74F35.0:X2
79.534:Y75.668:X29.634:G00 Z100.
G02X8.875:Y-56.866:G54:G01X39.2Y
-45.854Z18.0:G01F650X0Y12.3Z20.:

CONTROLLER INTERNAL DATA
05:X-14.829Y-16.467Z2.74F35.0:X2
79.534.Y75.668:X29.634:G00 Z100.
G02X8.875:Y-56.866:G54:G01X39.2Y
-45.854Z18.0:G01F650X0Y12.3Z20.:

              VERIFY ERROR !

MEM                              STP    LSK
—
     EDIT    DIR    IN/OUT                ↑
                           BGREST RETURN  ↓
```

Fig.3.4.47 Mismatch found

A verification error also occurs if the input from tape is longer than memory contents.

4 To restart verification, depress ☐ function key.

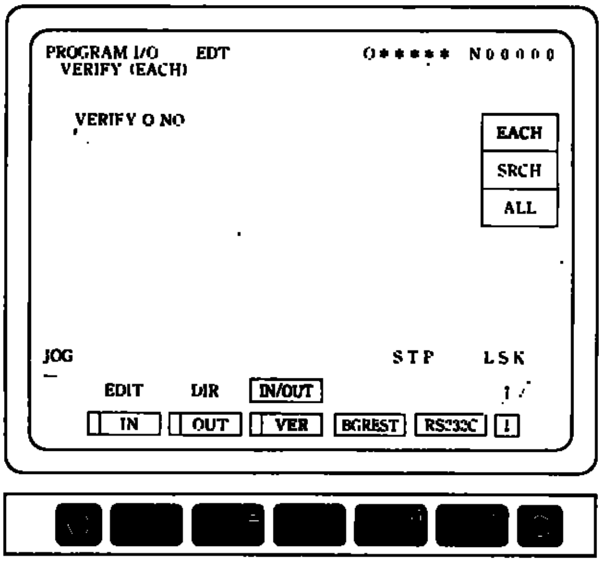
Verification restarts from the top of the next block where an error occured. However, if a verification error occurs in the last block, verification cannot be restarted.

(b) SRCH and ALL

1. Depress the VER

2. Depress the VER function soft key again. The pop-up menu appears and the operator is requested to select EACH, SRCH, or ALL. (See Fig. 3.4.48)

3. Depress the cursor key ▲ or ▼ to select from the pop-up menu.



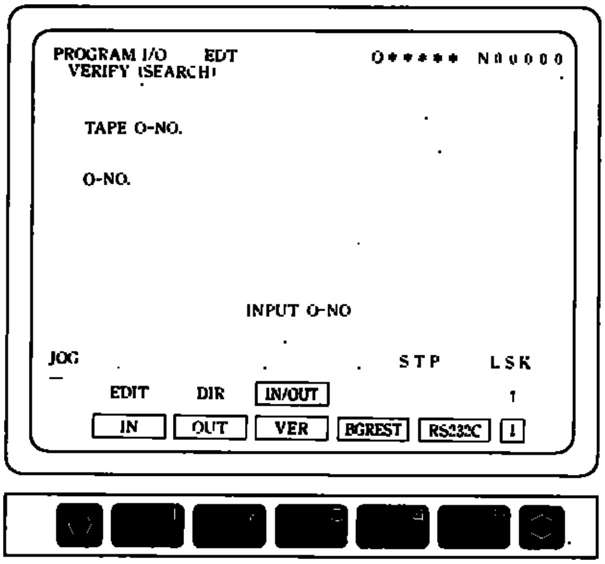
Pop-up menu (for verification)

Fig. 3.4.48 Veification Menu Select Screen

4. Depress the WR key. The pop-up menu disappears and the specified verification menu is displayed.

● **SRCH**

❶ The SRCH verification screen shown in Fig. 3.4.49 appears.



```
PROGRAM I/O    EDT                    O******  N00000
 VERIFY (SEARCH)

   TAPE O-NO.

   O-NO.

                      INPUT O-NO

JOG                                     STP    LSK
```

Fig. 3.4.49 SRCH Verification Screen

❷ Enter the O-number of the tape program to be verified, then depress the WR key. The O-number is displayed for both TAPE O-NO. and O-NO.

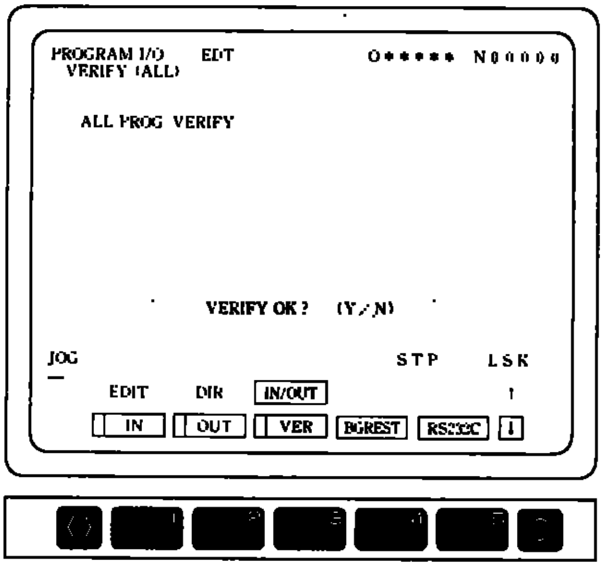
❸ If the programs to be collated have the same O-number, depress the WR key.

If they have different O-numbers, enter the O-number in controller memory, then depress the WR key. The O-number specified in ❷ (that is, TAPE O-NO. on the screen) is searched for from the tape, and the program is collated with another in controller memory with the O-number specified in ❸ (that is, O-NO. on the screen).

❹ If inconsistency between the memory contents and the input from external equipment is found, the screen shown in Fig. 3.4.47 is displayed, and the unmatched parts are shown highlighted.

●ALL

1 "ALL PROG. VERIFY" and "VERIFY OK ? (Y/N)" are displayed as shown in Fig. 3.4.50. To start verification, depress Y.



PROGRAM I/O EDIT O****** N00000
VERIFY (ALL)

ALL PROG VERIFY

VERIFY OK ? (Y/N)

Fig. 3.4.50 ALL Verification Screen

2 Program verification is started, and the contents of the part programs being verified are displayed.

3 To cancel verification operation, Depress N

4 If inconsistency between the memory contents and the input from external equipment is found, the screen shown in Fig.3.4.47 is displayed, and the unmatched parts are shown highlighted.

(4) **BG reset**

This is a reset switch for the background which is different from the operator panel key.

At BG reset, the following contents are affected :

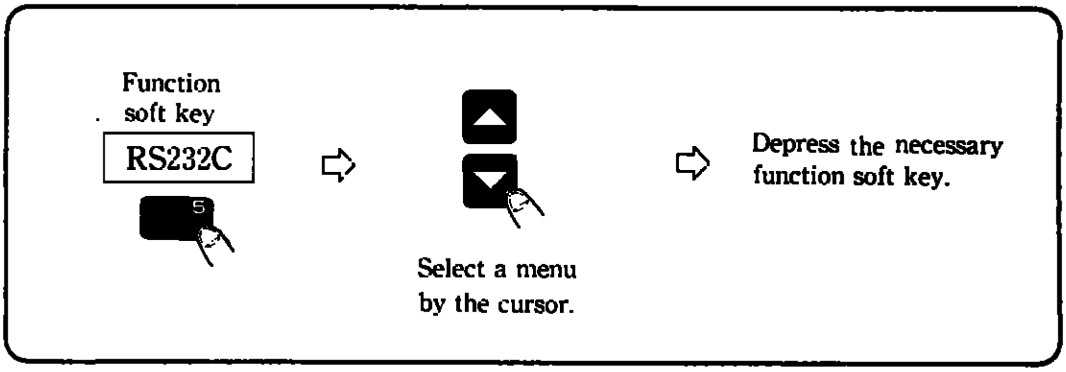
- Cancels tape I/O verification.
- Turns ON the label skip function.
- Releases background alarms (ALM 9000's)

Even by depressing the BGREST function soft key during operation, it does not affect the operation.

Additionally, even by depressing the BGREST function soft key, the "during resetting" status RST on the screen is not displayed.

(5) I/O equipment setup function

KEYPOINT



Function soft key

RS232C

⇨

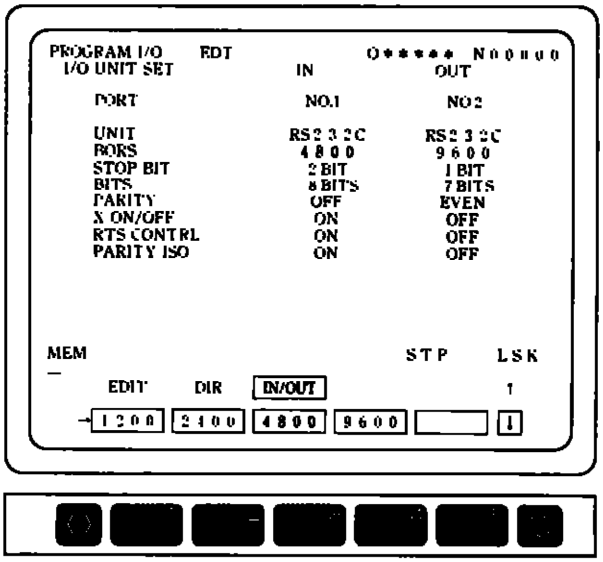
Select a menu by the cursor.

⇨

Depress the necessary function soft key.

■1 Depress the RS232C function soft key.

The screen in Fig. 3.4 51 appears. The screen is used to display and set up communication parameters for interfacing with external equipment according to RS-232C.
There are the first and second ports for input and output.



```
PROGRAM I/O      EDT                 O******  N00000
 I/O UNIT SET               IN              OUT

    PORT                   NO.1             NO2

    UNIT                 RS2 3 2C         RS2 3 2C
    BORS                   4 8 0 0          9 6 0 0
    STOP BIT               2 BIT            1 BIT
    BITS                   8 BITS           7 BITS
    PARITY                 OFF              EVEN
    X ON/OFF               ON               OFF
    RTS CONTRL             ON               OFF
    PARITY ISO             ON               OFF




MEM                                    STP      LSK
—
      EDIT      DIR     [IN/OUT]                 1
   →[1200]    [2400]    [4800]    [9600]  [     ]  [1]
```

Fig. 3.4.51 I/O Equipment Setup Screen

2 Place the cursor at the menu item to be set by cursor key ▲ or ▼.

Different function soft keys appear as the selected menu item changes.

3 Select and depress a soft key. Data are written to the cursor location.

(Example) In baud rate setup, depressing the 9600 soft key sets 9600 baud.

NOTE "* * * *" indicates that the item is automatically set up and cannot be changed.

Table 3.4.2 lists possible setting for serial interface:

Table 3.4.2 Serial Interface Setting

Menu item	Input	Output
Equipment	YE tape reader	
	General-purpose RS-232C	General-purpose RS-232C
Port	First	First
	Second	Second
Baud rate	100 or 75	100 or 75
	110 or 150	110 or 150
	300	300
	600	600
	1200	1200
	2400	2400
	4800	4800
	9600	9600
Stop bit	1 bit	1 bit
	2 bits	2 bits
Bit length	7 bits	7 bits
	8 bits	8 bits
Communication parity check	Even parity	Even parity
	Odd parity	Odd parity
	No	No
Control code	Yes	Yes
	No	No
RTS control	Yes	Yes
	No	No
ISO parity check	Yes	Yes
	No	No

NOTE

1. The two RS-232 ports cannot be used at the same time. The setting items for the first and second ports are independent from each other.
2. For the communication at the baud rate 9600 bps, the following parameter is recommended.

Baud rate	9600 bps
Stop bit	2 bits
Bit length	8 bits
Parity check	None/Even/Odd
NC data control	No

The menu items are explained in the following :

(1) **Equipment**

Select YE tape reader when a YASNAC specialized tape reader is to be connected. If YE tape reader is selected, the baud rate and the following items are set up automatically, and "* * * *" is displayed for these items.

Automatic setting : 4800 baud
stop bit : 2 bits
bit length : 8 bits
communication parity check : no
control code : yes

If general-purpose RS-232C is selected, various equipment can be connected by setting proper parameters.

(2) **Ports**

Select the first or second port.

(3) **Baud rate**

Set up the communication speed. In Table 3.4.2, there are two numbers each in the lowest two baud rate classes. Whether 100 and 110 or 75 and 150 are available is determined by parameters Pm11D7, Pm13D7, Pm16D7, and Pm18D7. If 1 is set for Pm11D7, 100 and 110 are displayed as soft keys for the first port input. If 0 is set for this parameter, 75 and 150 are displayed. The same rates apply to Pm13D7 with the first port output, Pm16D7 with the second port input, and Pm18D7 with the second port output.

(4) **Stop bit**

Set the stop bit length.

(5) **Bit length**

Set the data bit length.

(6) **Communication parity check**

Specify whether to use parity check by hardware.

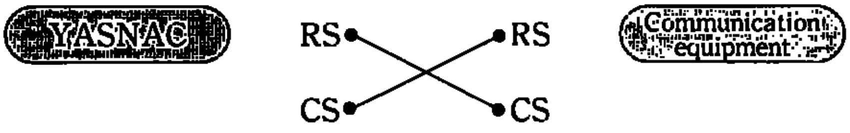
(7) **Control code**

Specify whether to use control code.

(8) **RTS control**

The RTS control governs YASNAC and connected communication equipment using RS and CS.

To perform RTS control, use RS and CS connection cables as shown in the figure :



(9) **ISO parity check**

Specify whether to perform parity check with data higher bit at ISO code I/O. This item can be specified by the operator even if YE tape reader or YE tape puncher has been selected.

Other communication control parameters

- **DR line check**

	Input	Output
First port	Pm0012 D2	Pm0014 D2
Second port	Pm0017 D2	Pm0019 D2

When the corresponding parameter is 1, DR check is performed for the specified input or output at specified port.

The DR line check tests whether the DR line of the remote equipment is on (that is, whether the equipment is ready.)

To perform the DR check, connect the remote equipment that supports the DR line control function to the NC controller and the remote equipment DR line with a cable.

(10) **NC data special control**

Sets whether NC special checks or character codes are to be used. There are some differences as follows when this control is provided or when not provided.

(a) **At input**

(i) **When control is provided**

pm0004 : D1, D2, D3, and pm3005 : D0, D2, D3, D4, D5, are to be parameter settings.

pm4100 to 4109 and pm4144 to 4146 are validated.

EIA/ISO automatic indentification provided.

(ii) **When control is not provided**

pm0004 : XXXX100X, pm3005 : XX0000X0

Any part other than Xs shown above become fixed.

pm4100 to 4109 and pm4144 to 4146 are invalidated.

ISO fixed.

(b) **At output**

(i) **When control is provided**

pm0004 : other than D3 and pm3005 : D5 are to be parameter settings.

pm4100 to 4109 and pm4144 to 4146 are validated:

(ii) **When control is not provided**

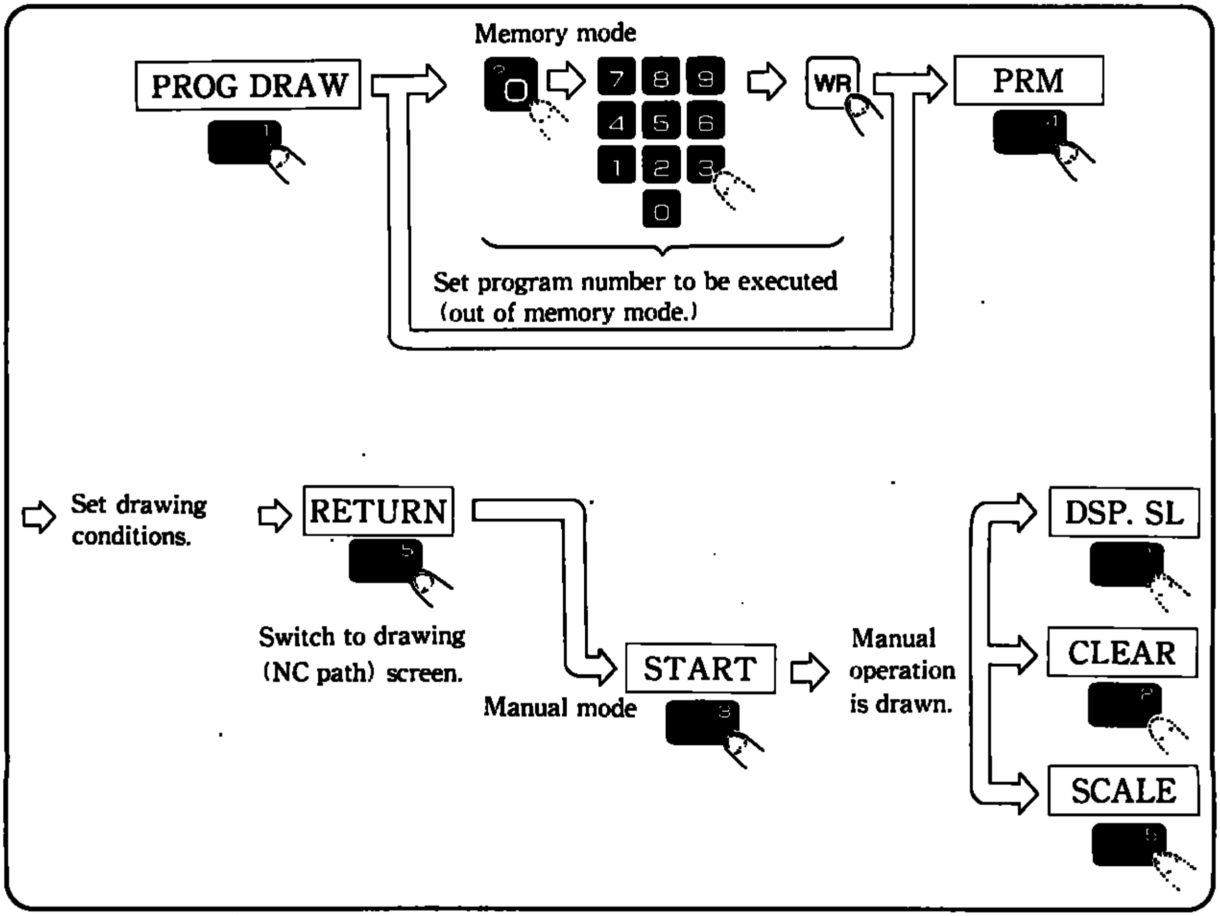
pm0004 : 0110XXX0, pm3005 : XX0XXXXX

Any part other than Xs shown above become fixed.

3.4.5.4 Program path drawing job ✻

On the screen, the path of the processing program tool is diagnosed and drawn on the screen. Conditions of drawing such as the range and the angle can be set on the drawing condition screen.

KEYPOINT operation



(1) **Features of i80M program path drawing function**

(a) The following describes the features of the drawing function realized by i80M.

- Line drawing : Drawing method to display tool path by lines is employed.
- Drawing plane change : A plane on which path is drawn can be selected (XY, YZ, ZX, XYZ, XY/YZ, XY/XZ).
- Visual angle free selection : On XYZ plane, visual angle can be selected freely in units of 5°.
- Drawing scale change : A scale to draw can be selected freely from all tool paths.

- Automatic scaling : Multiplication can be automatically set so that all paths can be within the screen.
- Arbitrary process draw : Cutting by one tool is regarded as one process, and only processes that are to be displayed can be selected for drawing.
- Auxiliary function indicator : During auxiliary function execution, the status is displayed by symbols.
- Continuous/single-change : Either single-block drawing or continuous drawing can be selected.

(b) Difference between program path drawing and NC path drawing

(i) Program path drawing

A path is drawn disregarding NC mode or execution status.
By using this function, the path can be drawn in the background under automatic operation.

(ii) NC path drawing

NC tool path is monitored and drawn on the screen.

(2) Basic Operation

The following describes the basic drawing procedures.

(a) Enter the edit process.

Depress the EDIT key.

(b) Select the program path drawing.

Depress the PROG DRAW key and select the program path draw job.

(c) Specify the program to be drawn. (Refer to (3) (a) (ii) of this paragraph.)

Input the data in the form of O + "O No. " + WR.

When no data are input, the program drawn in the former operation is entered.

(d) Set the drawing conditions. (Drawing plane, visual angle, etc.)
Depress the PRM key on the drawing screen.
→The drawing coniditjon setting screen is displayed.
Set the drawing plane and visual angle.
Set the other required items (block skip, compensation drawing mode, drawing scale, etc.)

* : Drawing process selection is disabled in this stage (before automatic scaling).

(e) Determine the drawing scale.

Depress the AUTO key on the drawing screen.

→Multiplication is automatically calculated and a message "AUTO SCALING" is displayed.

* : When automatic scaling is not performed, the drawing scale can be set by inputting the data to the drawing condition setting screen drawing scale (work coordinate system) item.

(f) Execute drawing.

Depress the START key.

→Drawing starts.

To stop drawing, depress the START key again.

To erase the drawn diagram, depress the CLEAR key.

With the procedures described above, drawing is completed.

Additionally, the following functions are enabled.

(g) A drawn diagram can be expanded.

Depress the SCALE key on the drawing screen.

→The drawing scale changing screen is displayed.

By using the cursor key and soft keys, the drawing frame position and size are changed and the next drawing scale is set.

Return to the drawing screen and depress the START key.

(h) Draw an arbitrary process.

Depress the PRM key on the drawing screen.

→The drawing condition screen is displayed.

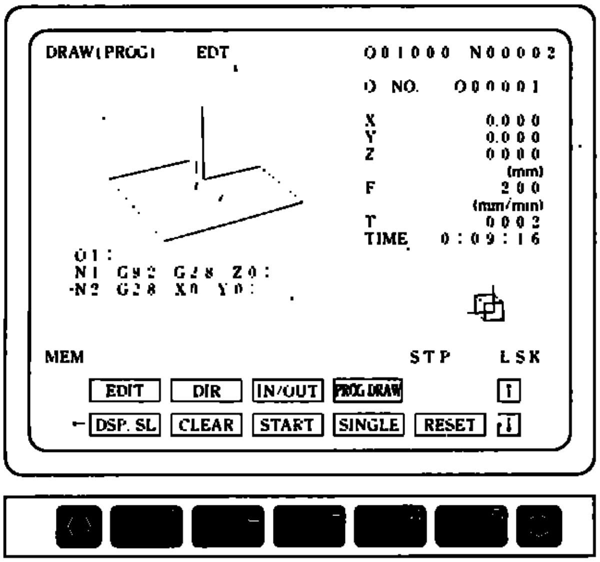
Move the cursor to the item of "drawing process" and select the tool No. of the process to be drawn by using the soft keys.

Return to the drawing screen and depress the START key after depressing the soft key of the selected drawing.

(3) Description of drawing

(a) Drawing screen

When the program path drawing job is selected on the edit process screen, the following drawing screen is displayed. The drawing screen is the main screen for drawing the specified program. From this screen, the drawing condition setting screen or drawing scale setting screen is called.



```
DRAW(PROG)     EDT               O01000  N00002
                                 O NO.    O00001
                                 X         0.000
                                 Y         0.000
                                 Z         0000
                                           (mm)
                                 F          200
                                       (mm/min)
                                 T         0002
                                 TIME   0:09:16
 O1;
 N1  G92  G28  Z0;
-N2  G28  X0  Y0;

MEM                              STP       LSK
```

PRM | SCALE | AUTO | PROCES | ☐

(i) Functions

The following are the functions contained in the drawing screen.

- Drawing O No. setting
- Drawing execution
- Automatic scaling
- Drawing clear
- Drawing condition setting screen or drawing scale setting screen call
- Drawing reset
- Drawing condition setting (single-block, selection screen)

(ii) Drawing program setting

<Drawing O No. setting>

It is necessary to specify the program to draw when drawing in the program path.

- Drawing O No. setting method

After inputting the O No. following "O" on the drawing screen, depress the WR key.

- Drawing O No. is held unless the power supply is turned OFF. Therefore, it is not necessary to set the drawing O No. again when the same program is drawn as before. The number is set to O when the power supply starts up.
- When the sub-program is drawn or scale is automatically calculated, the O No. is rewritten.

(iii) Execution of drawing

According to the drawing condition setting contents, a tool path is drawn. Whether drawing is performed by one block or continuously depends on the continuous/single soft key setting.

〈Drawing start〉

By depressing the START key on the drawing screen, drawing starts.

The status where the START key is displayed in reverse video is indicated in during drawing.

〈Drawing temporary stop〉

Drawing is stopped temporarily by depressing the START key during drawing.

However, in this case, it stopped in units of single-block.

By depressing the START key during temporary stop, the rest of the program is drawn.

〈Drawing completion〉

When the program drawing is completed, a message "DRAWING COMPLETE" is displayed. To complete the drawing forcedly, depress the RESET key.

Then depressing the START key starts drawing the program from the beginning.

〈Alarm process〉

If an error occurs in the drawing program, the alarm No. and alarm message are displayed. However, even if more than one alarm occurs, only one alarm with the largest alarm No. is displayed.

〈Drawn diagram clear〉

Depressing the CLEAR key erases the drawn diagram.

When this key is depressed during drawing, the diagram drawn at that time is cleared, and drawing is continued.

〈Types of lines〉

High-speed feeding and cutting feeding are identified by the types of lines as shown below.

High-speed feeding : Dotted line (tool center path)
Two-point chain line (compensated path)
Cutting feeding : Bold line (tool center path)
One-point chain line (compensated path)

NOTE

1. Drawing speed is constant disregarding program reference value or override.

2. The size of the drawing is usually determined by the maximum value and minimum value of the program with the item of drawing condition scale. However, when the drawing scale has been changed on the drawing scale setting screen, the drawing is expanded according to the set value.
3. In drawing, the tool offset amount and the work shift amount are referenced.

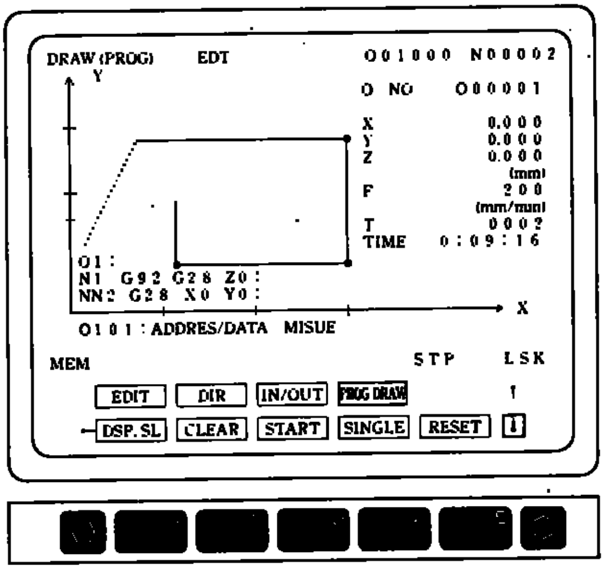
   Those data are validated when the drawing O-number is set or the RESET key is pressed.

   Therefore, if the tool offset amount and/or the work shift amount is changed in setup process, the drawing O-number should be re-set or the RESET key should be pressed.

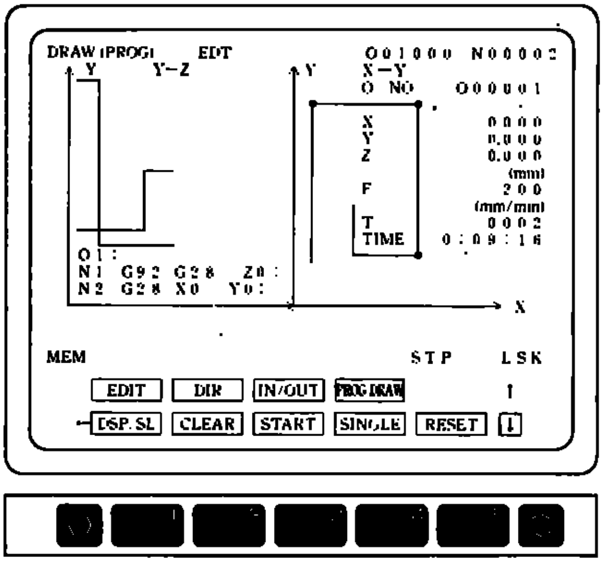
(iv) Display contents

<Drawn diagram>

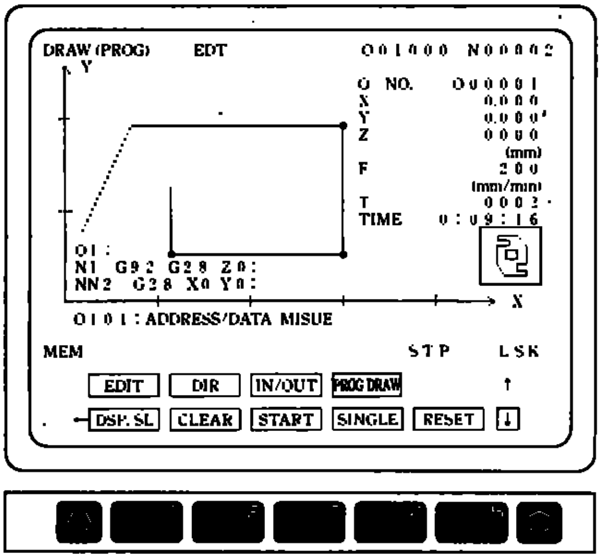
- The specified program tool path is displayed in lines.
- When a two-dimensional drawing is selected (XY, YZ, ZX, XY/YZ, XY/XZ planes) the coordinate axis is displayed on the screen as shown below. When a three-dimensional drawing is selected (XYZ plane) , the coordinate axis is not displayed.
- The display area is 304×544 dots at the upper left side of the screen.
- The drawing size of XY/YZ or XY/XZ plane is determined so that the drawing will be the maximum size.



```
DRAW (PROG)     EDT            O01000  N00002
                               O NO    O00001
                               X          0.000
                               Y          0.000
                               Z          0.000
                                           (mm)
                               F            200
                                       (mm/min)
                               T           0002
                               TIME   0:09:16
O1:
N1  G92 G28 Z0:
NN2 G28 X0 Y0:
O101 : ADDRES/DATA  MISUE
MEM                            STP      LSK
```



```
DRAW (PROG)      EDT                  O01000  N00002
  Y        Y-Z          Y    X-Y
                             O  NO       O00001
                             X              0000
                             Y             0.000
                             Z             0.000
                                            (mm)
                             F               200
                                        (mm/min)
                             T              0002
                             TIME        0:09:16
 O1 :
 N1  G92 G28  Z0 :
 N2  G28 X0   Y0 :
                                                X
MEM                                 STP      LSK
```



```
DRAW (PROG)      EDT                  O01000  N00002
  Y
                             O  NO.      O00001
                             X             0.000
                             Y             0.000'
                             Z              0000
                                            (mm)
                             F               200
                                        (mm/min)
                             T              0002
                             TIME        0:09:16
 O1 :
 N1  G92 G28 Z0 :
 NN2  G28 X0 Y0 :
                                                X
 O101 : ADDRESS/DATA MISUE
MEM                                 STP      LSK
```

<Drawing program No.>

The O No. of the program to be drawn in the program path is indicated. When the sub-program is drawn or scale is automatically calculated, the O No. is rewritten.

<Coordinate symbol>

When a three-dimensional drawing is selected, the coordinate symbol is displayed at the lower right side of the screen.

When a two-dimensional drawing is selected, the coordinate symbol is not displayed.

<Drawing program>

Three blocks are displayed during drawing of the programs.

<Position>

Positions of axes X, Y and Z are displayed. However, these values are not added with diameter compensated values.

<Cammand value>

F command value and T command value are displayed.

<Auxiliary function indicator>

Auxiliary functions which are effective during drawing are displayed as symbols.

There are three display positions according to the type of M code as shown in the drawing.

Left : M code symbol to indicate spindle status
Spindle forward run, spindle reverse run, spindle stop, spindle constant position stop

Center : M code symbol to indicate coolant ON/OFF status

Right : Non-modal M code symbol
Program stop, optional stop, end of program, tool replacement

The application of auxiliary functions and M code numbers are different depending on the machine manufacturers. They are determined by the setting status of NC function parameters (pm3442 to pm3448).

<Constant display>

Constant display section is displayed.

<Processing time>

Actual processing time until block being currently drawn is calculated and displayed.

(v) Soft keys

<Description of soft keys>

① CLEAR key
- Depressing this key clears the drawn diagram displayed at that time.
- Character information is not cleared.

② CONT / SINGLE key
- This key determines continuous drawing or single-block drawing.
- CONT / SINGLE key is switched by alternate operation.

③ START key
- Depressing this key starts the drawing.
- This key is displayed in reverse video during drawing.
- Depressing this key during drawing stops drawing.
- Depressing this key while drawing is stopped, restarts drawing from the stopped block.

④ DSP. SL key
- Depressing this key cyclically switches operation information (position, command, drawing program, etc.) being displayed or not displayed.

⑤ RESET key
- Depressing this key clears key buffer and drawn diagram, and returns the drawing execution cursor to the head of the program. Additionally, the offset data entered for drawing is renewed, and drawing and automatic multiplication calculation are stopped.

⑥ PRM key
- Depressing this key changes the screen to the drawing condition screen.

⑦ SCALE key
- Depressing this key changes the screen to the drawing scale setting screen. However, when XY/YZ plane or XY/XZ plane is displayed, this key is disabled.

⑧ AUTO key
- Depressing this key calculates multiplication automatically.
- During automatic multiplication calculation, a message "AUTO SCALING" is displayed at the lower part of the screen. Additionally, the character "AUTO" of the soft key is displayed in reverse video.

- When automatic scaling is completed, the character of "AUTO" is returned to normal status and the display at the lower part of the screen is changed to the front function display.
- Depressing this key during automatic scaling interrupts calculation, and the value calculated until that time is effective.
- Depressing the DSP. SL key displays the program, position, reference value, etc. of which scales are being calculated.

⑨ PROCES key

- This key determines whether all processes are drawm or only selected processes are drawn.
- By alternate operation, the character of "PROCES" is switched from normal display to reverse-video or vice versa. The reverse-video display indicates the selected drawing mode.
- When automatic scaling is not executed, this soft key is not displayed.

<Effective timing of soft keys>

- Keys effective at automatic scaling
  DSP. SL, CONT / SINGLE, AUTO, START, RESET keys.
- Keys effective during drawing
  CLEAR, CONT / SINGLE, START, RESET, PROCES keys. The soft key of PROCES is not displayed unless automatic scaling is completed.

(vi) Message

In this screen, the following messages are displayed.

"AUTO SCALING"
Indicates that automatic scaling is being performed.
(Highlighted, blinking)

"SCALING COMPLETE"
Indicates that automatic scaling is completed.
(Highlighted)

"DRAWING COMPLETE"
Indicates that drawing is completed.
(Highlighted)

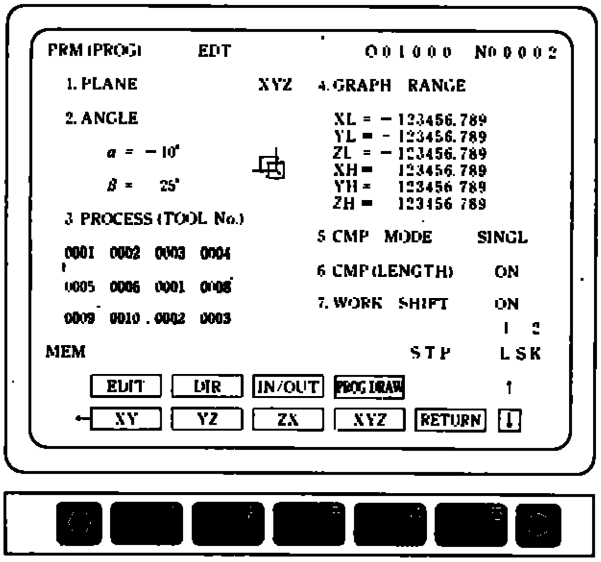
(vii) Others

- Depressing the RESET key (panel reset) does not affect the drawing or automatic scaling.
- When an additional axis is provided, the position is displayed but drawing is executed disregarding the additional axis.
- MST command excution time is considered at processing time calculation.

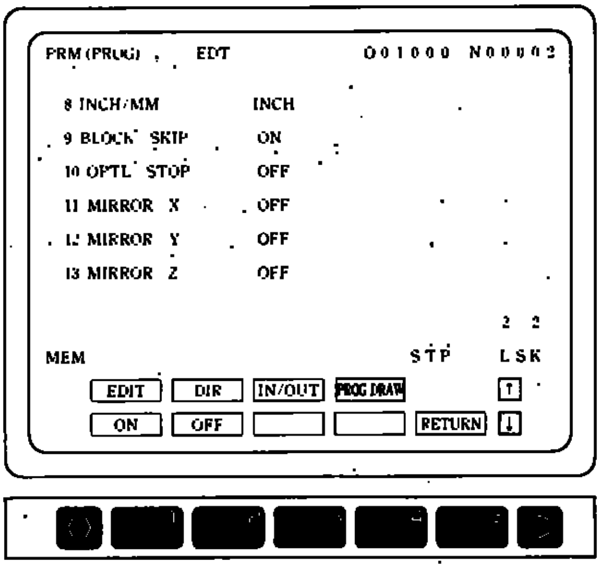
  Each set value of pm3449 to pm3451 is calculated as average time of M, S or T command.

(b) Drawing condition setting screen

When the PRM key is depresed on the drawing screen, the drawing condition setting screen as shown below is displayed. This screen is to display and set the conditions required for the program path drawing.



```
PRM (PROG)        EDT              O 0 1 0 0 0   N0 0 0 0 2
 1. PLANE               XYZ  4. GRAPH  RANGE
 2. ANGLE                     XL = − 123456.789
                              YL = − 123456.789
       α = − 10°              ZL = − 123456.789
                              XH =   123456.789
       β =   25°              YH =   123456 789
                              ZH =   123456 789
 3 PROCESS (TOOL No.)
                             5 CMP  MODE        SINGL
 0001  0002  0003  0004
                             6 CMP (LENGTH)      ON
 0005  0006  0001  0008
                             7. WORK  SHIFT      ON
 0009  0010 . 0002  0003
                                                 1  2
MEM                                   STP       L S K
```



```
PRM(PROG)      EDT                 O01000  N00002

 8 INCH/MM           INCH
 9 BLOCK  SKIP        ON
10 OPTL  STOP         OFF
11 MIRROR  X          OFF
12 MIRROR  Y          OFF
13 MIRROR  Z          OFF

                                           2  2
MEM                              STP       LSK
```

(i) Functions

The following drawing conditions can be set.

- Drawing plane
- Visual angle in three-dimensional drawing
- Selection status of process for selection drawing
- Drawing scale
- Tool compensation process
- Work coordinate system shiting value process
- Inch/millimeter change
- Optional stop process
- Mirror image process
- Block skip process

(ii) Parameter description and operation

Use the cursor keys $\boxed{\uparrow}$ or $\boxed{\downarrow}$ to move the cursor (highlighted) to each drawing condition item. Then each item contents are set by using the soft keys, cursor keys, $\boxed{\text{WR}}$ key and data key.

Depressing the $\boxed{\wedge}$ or $\boxed{\vee}$ key changes the page. The cursor move also changes the page.

The following describes the setting method of each item.

<Drawing plane>

Indicates that the drawing is either on two-dimensional plane (XY plane, YZ plane, ZX plane, XY/YZ plane or XY/XZ plane) or on three-dimensional plane (XYZ plane).

When the cursor is on this item, the soft key lower stage is changed as follows and the depressed soft key plane is selected. Additionally, a plane can be selected by using the $\boxed{\text{WR}}$ key.

XY	YZ	ZX	XYZ	RETURN
XY YZ	XY · XZ			RETURN

<Visual angle>

Indicates drawing angle with a three-dimensional drawing.

"α" is the angle with Z-axis as the rotation center and X-axis → Y-axis as the forward direction.

"β" is the angle made by XY plane and visual angle and XY plane → Z-axis as the forward direction.

When the cursor is on this item, the soft key display is changed as shown below.

RIGHT	LEFT	UP	DOWN	RETURN

①Setting by soft keys :

By depressing the RIGHT or LEFT key, value of α is changed by -5° or $+5^\circ$, respectively.

By depressing the UP or DOWN key, value of β is changed by -5° or $+5^\circ$, respectively.

②Setting by numerical value input

When the cursor is on this item, the second cursor is displayed at the position of α or β.

When the WR key is depressed after inputting a numerical value in this status, the visual angle is changed.

The second cursor is moved by the cursor → or ← key or WR key.

<Drawing process>

- Indicates the tool No. for selection drawing.
- Tool No. to be displayed is used in the program of which scale has been automatically calculated, and stored at automatic scaling.
- The highlighted number is the tool No. for selected drawing.

- When more than nine tool Nos. are provided, they are displayed by scrolling according to the cursor (displayed, blinking) movement.
- Tool No. display status and meaning

  Normal character : Not selected and not at the second cursor position
  Highlighted character : Selected and not at the second cursor position
  Blinking character : Not selected and at the second cursor position
  Highlighted, blinking character: Selected and at the second cursor position
- When the cursor is on this item, the second cursor is displayed (blinking) in the tool No. list and the soft key display is changed as shown below.

UP DOWN LEFT RIGHT RETURN

- Depressing the cusor key → or ←, or soft key RIGHT or LEFT moves the second cursor right or left.
  Depressing the soft key UP or DOWN, or the cursor key ↑ or ↓ moves the second cursor up and down.
  WR key alternate operation changes "selected/not selected" of the tool where the second cursor is located.
  The highlighting displayed No. is the selected tool No.
- The number of tools to be displayed at one time is 15 in T2 digits and 9 in T4 digit.
  If more than the number specified are provided, the screen is scrolled to display them with the second cursor movement.
- In the following cases, the cursor is not moved.
  When the ← cursor key is depressed where the cursor is at the first tool.
  When the → cursor key is depressed where the cursor is at the last tool.
- When the → cursor key is depressed where the cursor is at the right side, or when the ← cursor key is depressed where the cursor is at the left side, the cursor moves to the former or next tool respectively.

<Drawing scale (reference coordinate system) >

- Indicates which scale on the reference coordnate system is drawn on the screen.
- When automatic scaling is not performed, any desired scale can be drawn by setting this item to the scale to be drawn with a value on the reference coordinate system.
- Automatic scaling sets this item automatically to the maximum value or minimum value in the program.
- When the cursor is on this item, the soft key display is changed as shown below, and the second cursor (reversed cursor) is displayed.

UP	DOWN			RETURN

- Depressing the soft key UP or the cursor key ← moves the second cursor up while depressing the soft key DOWN or cursor key → moves it down.
- By "numerical value" + WR key input, the value of the coordinate where the second cursor is located is changed and the cursor is moved down.

<Compensation drawing mode>

- Indicates whether tool compensation is considered to be drawn or to be disregarded.
- There are three types of drawing methods.
  Selecting "OFF" draws a path disregarding tool compensation.
  Selecting "SINGLE" draws a path considering tool compensation.
  Selecting "DOUBLE" draws paths both considering and disregarding tool compensation.
- When the cursor is on this item, the soft key display is changed as shown below.

DOUBLE	SINGLE	OFF		RETURN

- Drawing status is selected by soft key DOUBLE, SINGLE or OFF or the WR key.

<Length compensation>

- Indicates whether tool compensation is considered or disregarded to be drawn.
- Drawing method depends on the setting of compensation drawing mode "OFF" , "SINGLE" or "DOUBLE".
  When "OFF" is selected, this item setting is disabled.
  When "SINGLE" or "DOUBLE" is selected, for the path considering tool compensation, length compensation is selected whether to be disregarded or considered.
- When the cursor is on this item, the soft key display is changed as shown below.

ON	OFF			RETURN

<Input unit>

- Selects whether a path is drawn in the units of inches or millimeters.
- When G20 or G21 is commanded in the program, the value of this item is changed.
- When the cursor is on this item, the soft key display is changed as shown below.

MM	INCH			RETURN

- Input unit inch/millimeter is selected by the soft key MM or INCH or the WR key.

<Work coordinate system>

- Selects whether a path is drawn considering or disregarding work coordinate system shifting value.
- Work coordinate system shifiting value effective/ineffective is selected by the soft key ON or OFF or the WR key.

<Block skip>

- Selects whether a block with "/" at the head is to be effective or ineffective (block skip function).
- Selects block delete effective/ineffective by the soft key ON or OFF or WR key.

<Optional stop>

- Selects whether temporary stop is performed in MO1 block (optional stop function).
- After temporary stop, drawing starts again by depressing the start key.
- Optional stop effective/ineffective is selected by the soft key ON or OFF or the WR key.

<Mirror image>

- Specifies mirror image effective axis.
- Mirror image other than axes X, Y and Z are ineffective.
- There are three items : mirror image X, mirror image Y and mirror image Z.
- Mirror image effective/ineffective is selected by the soft key ON or OFF or the WR key.

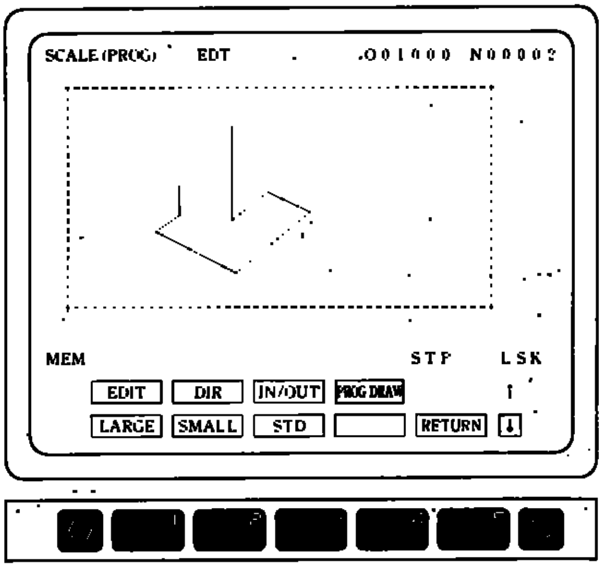
(iii) Others

The drawing conditions are not cleared even when the power supply is turned OFF.

However, the tool information of the drawing processes are cleared.

(C) Drawing scale changing screen

Depressing the SCALE key in the drawing screen displays the drawing scale changing screen as shown below. This screen is used for partial expansion of the program which has been drawn once.



(i) Function

By specifying the diagram drawn before by framing the section to be expanded, the size of the next drawing is determined.

Then by drawing in the drawing screen, the framed section is drawn fully on the screen.

(ii) Display contents

- Drawn diagram

  The same diagram (drawn in the former operation) as the drawing main screen is displayed.
- Drawing frame

  A frame to indicate the drawing scale. When this frame position and size are changed, the drawing window for the next operation is changed.

(iii) Description of operation

(1) Drawing frame position and size are set.

- Depressing the cursor ↑, ↓, → or ← key moves the drawing frame up, down, right or left by one character.
- Depressing the LARGE or SMALL soft key expands or reduces the drawing frame in units of characters.
- Depressing the STD key returns the drawing frame to the same size as the standard drawing scale.

  However, when the drawing frame is at the end of the screen, the frame is not moved or expanded.

(2) Depressing the RETURN key changes the screen to the drawing screen.

(iv) Others

- When XY/YZ plane or XY/XZ plane is displayed on the drawing screen, it is not possible to change to this screen.
- Even by changing the drawing scale, the drawing scale items of the drawing conditions are not affected.

(4) Description of other functions

(a) Scaling

It is necessary to set how large the program tool path is to be drawn on the screen before starting drawing. There are two methods : automatic scaling and key input of drawing scale value.

① Automatic scaling

Automatic scaling is a function to assure that all program tool paths are fully displayed on the screen. At the same time, program grammar error check and processing time calculation are also available.

- Automatic scaling start

  Depressing the AUTO key starts automatic scaling.

- Completion of automatic scaling

  When M30, M99, or M02 is commanded in the program, automatic scaling is completed. However, M99 command in the subprogram does not complete it. To interrupt automatic scaling, depressing the START key (in highlighted status) or the AUTO key (in highlighted status) completes automatic scaling at that time.

  In this case, scaling is calculated for the tool path until stopped.
- Setting items by automatic scaling.

  By executing automatic scaling, the following data items are set.

  Drawing scale (drawing condition) : Sets the maximum value and minimum value on the X, Y and Z reference coordinates in the program.

  Drawing process (drawing condition) : Sets the tool Nos. used in the program.
- CONT / SINGLE change

  Selecting "SINGLE" by the CONT / SINGLE key enables automatic scaling by one block.
- Display selection

  Display status (position, program display provided/not provided) during automatic multiplication calculation is switched.
- Program error check

  If there is an error in the program, automatic scaling is canceled and an alarm is displayed. As well as at drawing, the alarm No. and alarm message are displayed at the lower part of the screen.

NOTE

At scaling, the drawing conditions (work cooridnate system considered/disregarded, tool compensation provided/not provided) is referenced.

② Drawing scale input

Automatic scaling is omitted and scaling can be set by numerical value input.

- Input method.

  Four drawing scale values are set on the drawing condition setting screen.
- Input value

  The scale to draw among tool paths is input with the value on the reference coordinate system.

NOTE

Once automatic scaling is executed, the scale can not be changed unless automatic scale is performed for the other program or the drawing scale is input. Therefore, to draw the same program more than one time, automatic scaling is not needed for the second time and after.

(b) Arbitrary process drawing

<Arbitrary process drawing operation>

① Depress the PRM key on the drawing screen.
The drawing condition setting screen is displayed.

② Use the cursor ↑ or ↓ key to move the cursor to the item of "3. PROCESS".

③ Use the cursor → or ← key to move the cursor (displayed, blinking) to the tool No. of the process to be drawn, and use the WR key to select "draw/not draw".
The process of the highlighting displayed tool is drawn.

④ Depress the RETURN key on the drawing condition screen to display the drawing screen.

⑤ Depress the PROCES key to enter the arbitrary process drawing mode.
The PROCES key is reversely displayed.

⑥ Start drawing.
Only the selected process is drawn.

<Relation between tool and process>

One process in the arbitrary process drawing is from the time a certain tool is replaced and mounted on the spindle to when the next tool is replaced. Then by selecting the tool No., it is specified which process is to be drawn.

<Tool No. to set drawing process (drawing condition)>

Tool Nos. to set drawing condition process at automatic scaling are only tools to be replaced according to M06 command. That is, if the tool is not replaced even by T command, the T command is disregarded.

(Example)

```
        :
        :
T01 ;
T02 ;
G28  Z0 ;
G28  X0  Y0 ;
M06 ;          .........①
        :
        :
T03 ;
G28  Z0 ;
G28  X0  Y0 ;
M06 ;          .........②
        :
```

Since T01 tool is not replaced, it is disregarded.

T02 process is from block ① to block ②.

NOTE

- Even if drawing process (drawing condition) is changded, arbitrary process drawing is disabled unless the arbitrary process drawing mode is entered.
- Processes for arbitrary process drawings are from the program beginning to the 45th process.
- If the same tool No. is specified twice, it is regarded as one process.
- In the program commanded by M06 in the status without T command from the program beginning, the first process tool No. is unknown. In this case, this process is always drawn.

(5) **Screen change**

The drawing screen, drawing condition setting screen or drawing scale changing screen is displayed again even if other processes or other jobs are switched ; the display status is also returned.

(a) Switching to other processes

(i) Drawing screen

It is possible to switch to other processes even during automatic scaling or drawing.

Even if other process screens are displayed during automatic scaling or drawing, automatic scaling or drawing is executed in the background.

(ii) Both drawing condition setting screen and drawing scale changing screen can be switched to other processes and the screen status is also held.

(b) Switching in job

(i) Drawing screen → drawing condition setting screen, drawing screen → drawing scale changing screen

Cannot be changed during drawing or automatic scaling.

(ii) Drawing condition setting screen → drawing screen → drawing condition setting screen

Returns to the former status including the cursor position.

(iii) Drawing scale changing screen → drawing screen → drawing scale changing screen

Returns to the former status including the drawing frame.

(c) Switching between program path drawing and NC path drawing

(i) Drawing screen

When switching between program path and NC path, the drawn diagram in the former path is displayed.

When another path drawing screen is displayed in the drawing status in the background, drawing in the background (another path drawing) is displayed.

When drawing or automatic scaling is performed in the another path (background), the START and AUTO keys become ineffective.

(ii) Drawing condition setting screen

Does not affect another path.

(iii) Drawing scale changing screen

In addition, the drawing screen, when another path drawing screen is displayed in the drawing status in the background, drawing in the background (another path drawing) is displayed.

(d) Others

Changing mode does not affect any other process or function.

(6) DRAWING RELATED PARAMETERS

WORD parameter

pm3442 : Spindle forward run M code No.
pm3443 : Spindle reverse run M code No.
pm3444 : Spindle stop M code No.
pm3445 : Spindle constant position stop M code No.
pm3446 : Tool replacement M code No.
pm3447 : Coolant ON
pm3448 : Coolant OFF
pm3449 : Average time required for M code command (msec)
pm3450 : Average time required for S code command (msec)
pm3451 : Average time required for T code command (msec)

(7) Notes for drawing of machining programs

In drawing of machining programs, some parts of NC statement does not make the same movement as it should be normally.

(a) G codes

The following G codes make different movement than NC statement.

- G10

  When the work offset amount or the tool offset amount is changed using G10 command, the setting of parameter will not be changed.

  However, to change the inside area, the change takes effect in drawing.

- G22

  G22 and G23 (stored stroke limit) commands does not change the parameter settings.

- G27, G28, G29 and G30

  From the drawing starting point as the machine zero point, the machine returns to the reference point.

- G31

  As skip input can not be made at G31 command, it is drawn as G01 command up to the G31 command.

- G93

  Solid tap command makes movement of ordinary tap command.

- G53

  The drawing is made with the supposition of the machine coordinate system with the drawing starting point as the machine zero point.

- G106

  The movement by G106 (automatic override) command is drawn as the movement by ordinary G01 command.
- G125

  G125 (tool life count multiplication) is disregarded.

(b) Program copy

G25 (program copy) can not be executed.
The block of G25 is disregarded.

(c) High-speed cutting does not cause an alarm but is drawn as ordinary cutting command.

(d) User macro

- G code makes the same movement as NC

  Macro program simple call (G65)
  Macro program modal call (G66)
  Macro program call cancel (G67)
  GMSTB macro program call
  Operation command and control command
- G code makes different movement as NC

  The following system variables do not take the actual status of NC into account.

  Modal information #4000's

  The modal information commanded by the program path drawing execution, are read-in from the data area for drawing.

  Position information #5000's

  The axis position information diagnosed by the program path drawing execution, are read-in from the data area for drawing.

  The program path drawing is made with supposition of the machine coordinate system with the drawing starting point as the machine zero point.
- Variables do not execute drawing

  Drawing diagnosis is not executed by the following variables, which are disregarded.

  Interface input signal (#1000 to #1031, #1032)
  Interface output signal (#1000 to #1131, #1132)
  Alarm message display (#3000)
  Clock (#3001 and #3002)

Control for single-block stop, and miscellaneous function complete wait (#3003)

Feed hold stop (#3004)

Feedrate override (#3004)

Exact stop (#3004)

RS232C data output 2 (#3100, BPRNT, DPRNT)

(e) Tool life control

Tool life control command can not be used.

Accordingly, tool life control T command is handled as ordinary T command.

(f) Circle cutting

Circle cutting is drawn as the consecutive command of G02/G03 with radius offset.

The high-speed feed section is drawn as cutting feed (solid line or dotted chain line).

3.4.6 Setup Process SET

The setup process is preparation for cutting. In this process, the following can be displayed and set up : work coordinate system shift amount, tool offset, and tool life control data. The data can be displayed or set up any time regardless of the controller operation mode.

1. Depress the SET key.

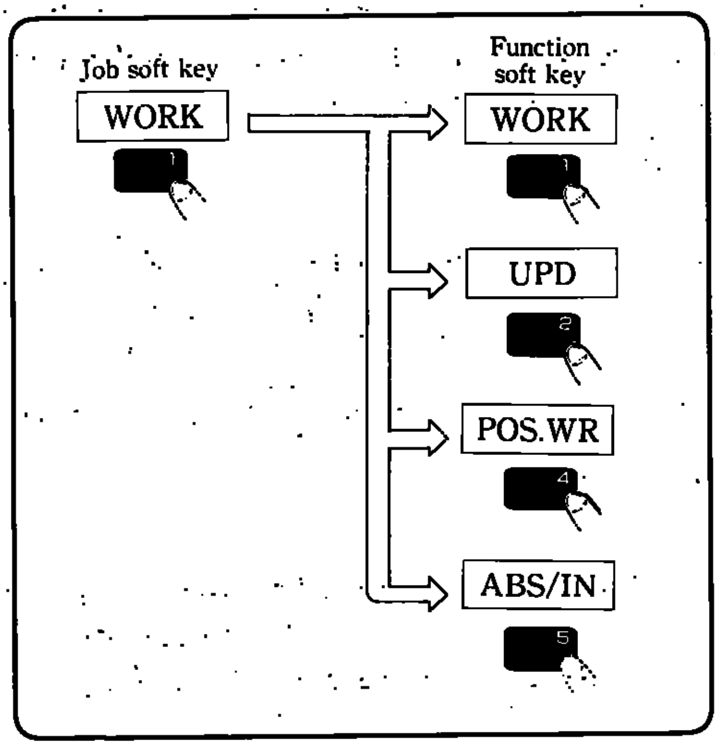
2. Select a job by depressing the WORK 1, TOOL 2, or T-LIFE 3 job soft key. Otherwise, depress the SET key a number of times until the target job appears.

- Work coordinate system job → See Par. 3.4.6.1 (page 448.)
- Tool job ——————→ See Par. 3.4.6.2 (page 454.)
- Tool life control job ———→ See Par. 3.4.6.3 (page 473.)

NOTE Job at power ON can be selectd using these parameters :
Pm3003 D2 = 0 : Selects work coordinate system job at power ON.
Pm3003 D2 = 1 : Selects tool job at power ON.

3.4.6.1 Work coordinate system job (job soft key WORK)

KEYPOINT



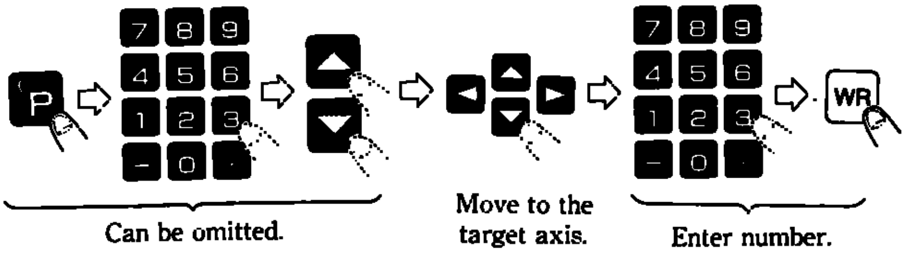
1 Depress the WORK job soft key.

2 The WORK, UPD, POS.WR, and ABS/IN function soft keys are displayed.

(1) **Work shift**

Work shift amount data can be set up and displayed.

KEYPOINT

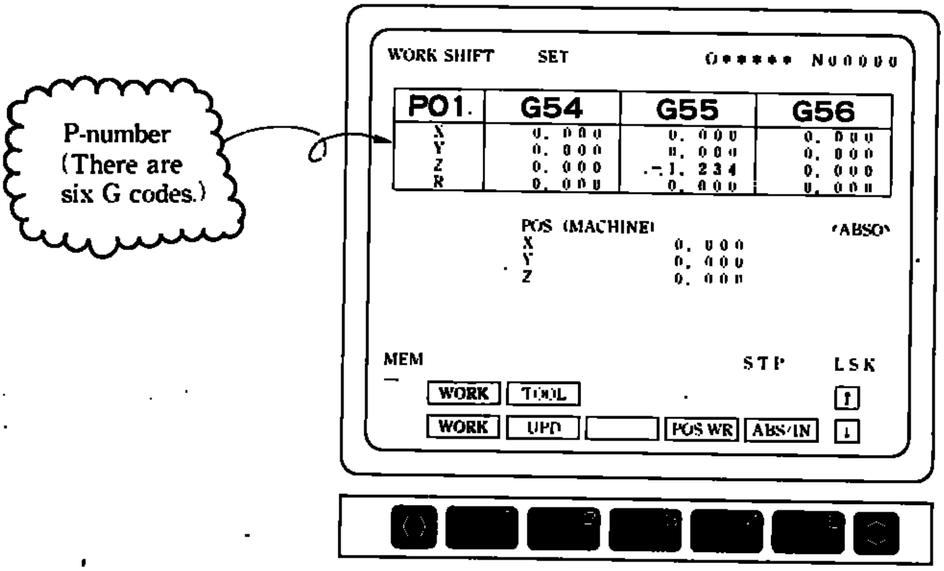


NOTE Do this operation for handling many coordinate systems.

1 Depress the WORK function soft key.

2 Depress the page keys ▲ PAGE ▼, or depress P ⇨ enter number, then depress cursor key ▲ or ▼ to display the work coordinate system shift amount data of a target P-number.

NOTE In addition to six standard work coordinate systems, 54 or 162 optional ones are available. The P-number indicates the number of the coordinate system : P1 for six, P2 for 12, ... and P27 for 162.



P-number (There are six G codes.)

```
WORK SHIFT     SET                  O******  N00000

 P01.  |   G54    |   G55    |   G56
  X    |   0.000  |   0.000  |   0.000
  Y    |   0.000  |   0.000  |   0.000
  Z    |   0.000  |  -1.234  |   0.000
  R    |   0.000  |   0.000  |   0.000

            POS (MACHINE)                      (ABSO)
            X                  0.000
            Y                  0.000
            Z                  0.000

MEM                                     STP    LSK
_
   [WORK] [TOOL]                               [↑]
   [WORK] [UPD ] [    ] [POS WR] [ABS/IN]      [↓]
```

Fig. 3.4.52 P-number Input Screen

NOTE Position data displayed on this screen can be converted to machine coordinates using the parameter :
Pm 3000 D3 = 0 : Machine coordinate system
1 : Work coordinate system

3 Move the cursor to the axis of the work coordinate system where data are to be written. G codes are classified into two groups : G54 to G56 and G57 to G59.

▲ ▼ : Selects an axis.
◀ ▶ : Moves among G-numbers.

4 Enter data.

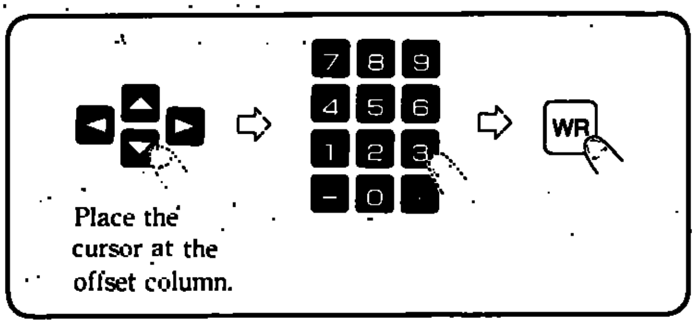
(For example, - 1 . 2 3 4 WR)

The input data are displayed as shown in Fig. 3.4.52.

(2) Offset

The offset for shifting all work coordinate systems can be set up and displayed.

KEYPOINT



1 Depress the UPD function soft key.

2

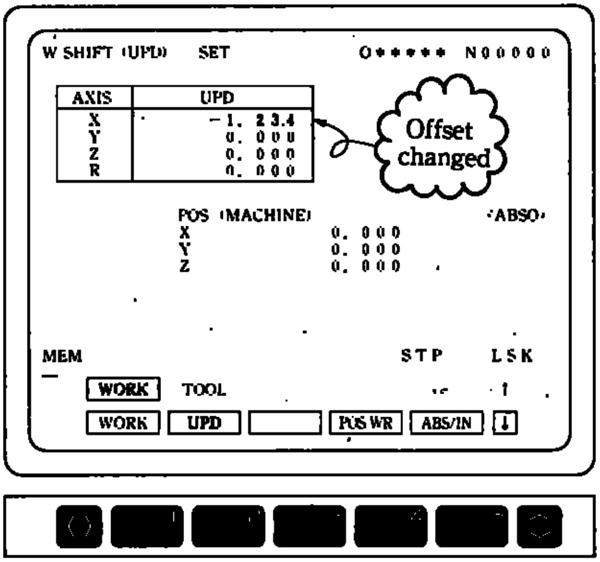
2 Place the cursor at the offset column of the work shift coordinate system where data are to be written.

3 Enter data.

(For example, - 1 . 2 3 4 WR)

The input data are displayed as shown in Fig. 3.4.53.

NOTE According to the setting parameter, the following status is entered.
Pm0007 D3 = 0 : Does not add the work shift offset amount.
Pm0007 D3 = 1 : Adds the work shift offset amount.
Work shift offset amount is written when Pm0007 D3=0, "CAN'T WRITE !" is displayed.



AXIS	UPD
X	−1. 23.4
Y	0. 000
Z	0. 000
R	0. 000

POS (MACHINE) (ABSO)
X 0. 000
Y 0. 000
Z 0. 000

Fig. 3.4.53 Offset Input Screen

NOTE Position data displayed on this screen can be converted to machine coordinates using the parameter :
Pm 3000 D3 = 0 : Machine coordinate system
Pm3000 D3 = 1 : Work coordinate system

(3) Work shift position write

Position data displayed on the work shift screen can be written as work shift coordinates.

1. Depress the POS. WR  function soft key. The key is highlighted and "ALL SET (Y/N)?" is displayed.
2. Depress  to set the entire position data for the work shift coordinates.
3. Depress  to select axes. "INPUT AXIS NAME" is displayed.
4. Enter axis names to write position data, then depress the WR key.

NOTE 1. The position write mode is canceled by depressing the POS.WR function soft key or the CAN or [illegible] key, or by switching to another screen.

2. Machine or work coordinates can be selected for the position data to be written.
Set the parameter as follows:
Pm3003 D3 = 0 : Machine coordinate system
Pm3003 D3 = 1 : Work coordinate system

● **Examples of writing position data**

❶ **Example of writing all data**

"ALL SET (Y/N)?"

P 01	G 54	
X	0.000	
Y	0.000	
Z	0.000	
R	0.000	

POS. (MACHINE)	
X	123.456
Y	654.321
Z	−1.234

↓ Press Y

P 01	G 54	
X	123.456	
Y	654.321	
Z	−1.234	
R	0.000	

POS. (MACHINE)	
X	123.456
Y	654.321
Z	−1.234

● **Example of writing on specific axes (writing offsets on Y and Z axes)**

"ALL SET (Y/N)?"

Press N

"INPUT AXIS NAME"

P 01	G 54
X	0.000
Y	0.000
Z	0.000
R	0.000

POS. (MACHINE)	
X	123.456
Y	654.321
Z	−1.234

Press Y SHIFT N Z WR

P 01	G 54
X	0.000
Y	654.321
Z	−1.234
R	0.000

POS. (MACHINE)	
X	123.456
Y	654.321
Z	−1.234

NOTE

1. Position data are written to the work coordinate system indicated by the cursor.
2. The data displayed on the screen are written.
3. If an invalid axis name is entered, "INPUT ERROR!" is displayed.
4. If two or more axis names are specified and any of them is invalid, no data will be written at all.
5. Position data are written as absolute values regardless of input selection.

(4) Input select function

Select incremental or absolute for data input. Each time the ABS/IN 5 function soft key is depressed, the incremental input mode and the absolute input mode changes alternately.

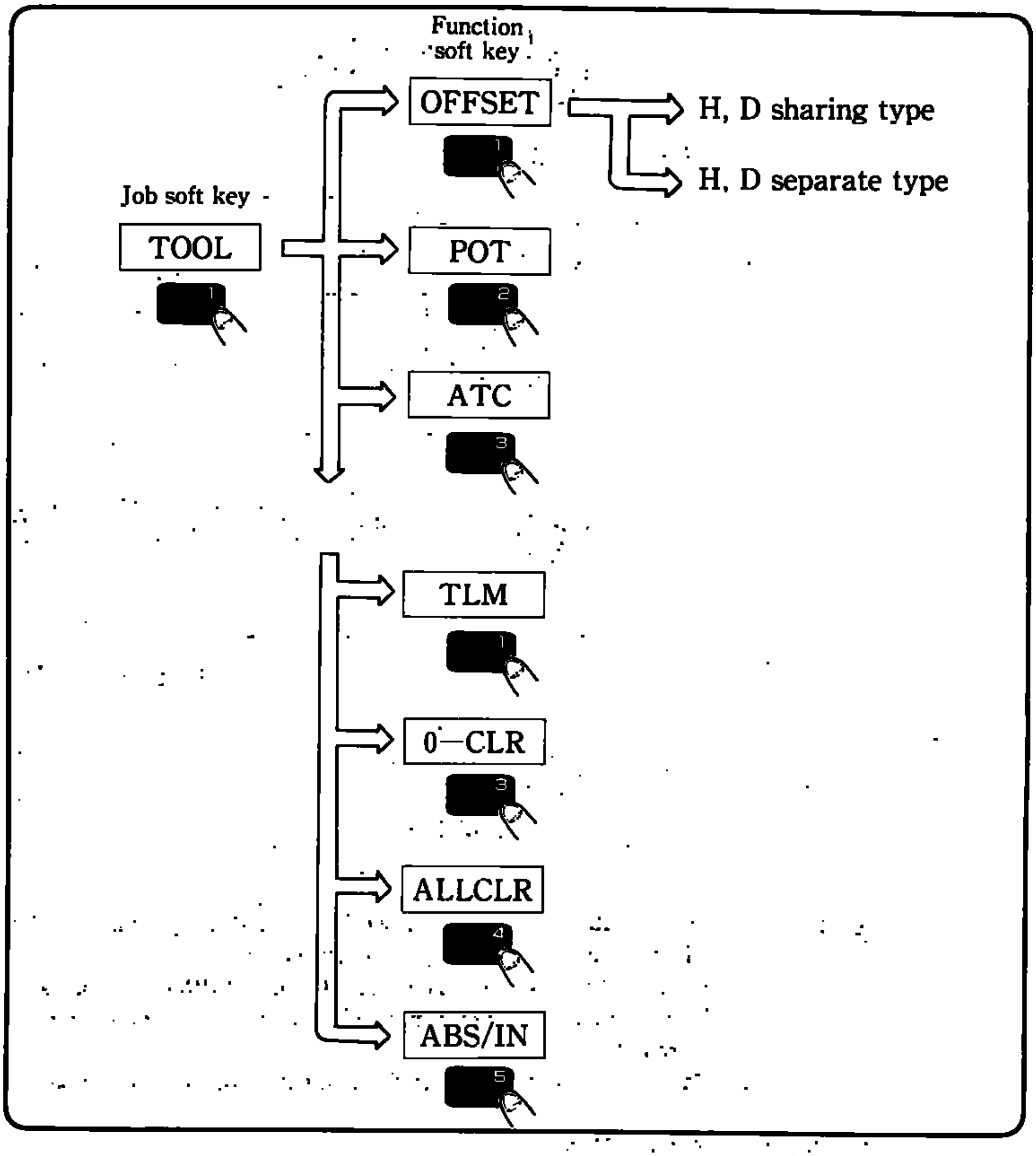
NOTE I/O switch can be selected using the parameter.

Pm3003 D3 = 0 : Enters the absolute input status at power ON.

Pm3003 D3 = 1 : Enters the incremental input status at power ON.

3.4.6.2 Tool job (job soft key TOOL)

KEYPOINT



1. Depress the TOOL job soft key.

2. Select OFFSET, POT, ATC, or TLM function select soft key.

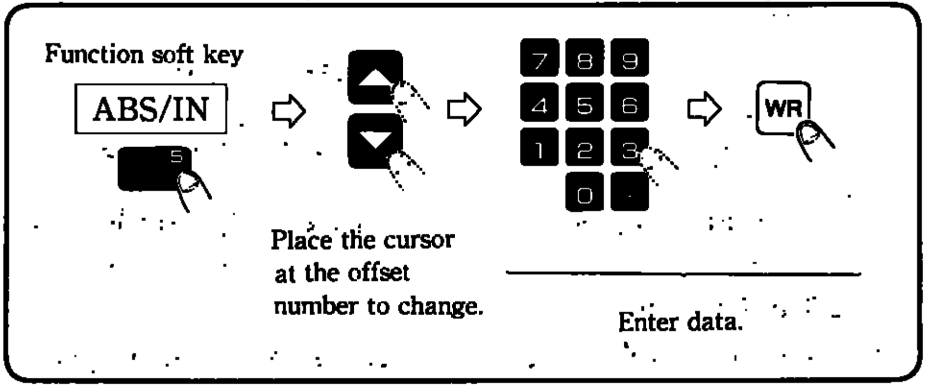
OFFSET operation → [See (1) (page 455)]
POT operation → [See (2) (page 460)]
ATC operation → [See (3) (page 463)]
TLM operation → [See (4) (page 464)]

(1) Offset function

Either of the following types is selected according to parameter setting:

- H, D sharing type → [See (a) (page 456)]
- H, D separate type → [See (b) (page 458)]

(a) H, D sharing type



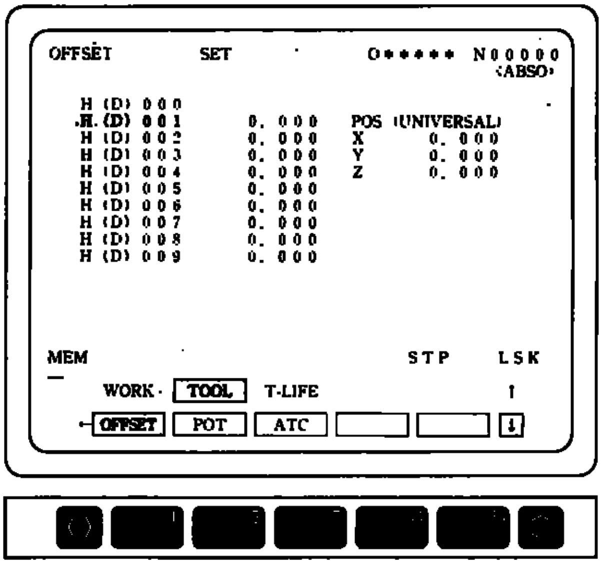
Place the cursor at the offset number to change.

Enter data.

1 Depress the ABS/IN function soft key to select absolute or incremental input.

NOTE 1. The ABS/IN function soft key is an alternate switch. <INCR> or <ABSO> is displayed on the upper right part of the screen indicating the current state.

2. I/O switch can be selected using the parameter.
Pm3003 D3 = 0 : Enters the absolute input status at power ON.
Pm3003 D3 = 1 : Enters the incremental input status at power ON.



```
OFFSET          SET              O****** N00000
                                         <ABSO>

   H (D) 000
  .H.(D) 001        0. 000     POS (UNIVERSAL)
   H (D) 002        0. 000     X        0. 000
   H (D) 003        0. 000     Y        0. 000
   H (D) 004        0. 000     Z        0. 000
   H (D) 005        0. 000
   H (D) 006        0. 000
   H (D) 007        0. 000
   H (D) 008        0. 000
   H (D) 009        0. 000

MEM                                  STP     LSK
—
      WORK · [TOOL]  T-LIFE                   ↑
   ←[OFFSET] [POT]  [ATC]  [    ]  [    ]    [↓]
```

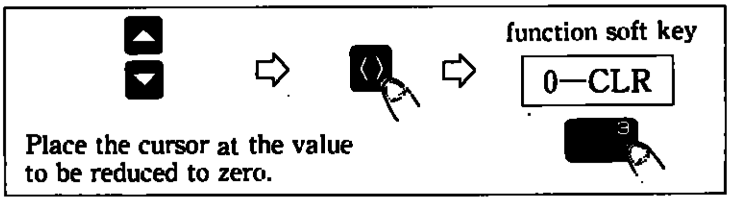
Fig. 3.4.54 Offset Screen (for H, D sharing type)

NOTE Position data displayed on this screen can be converted to machine coordinates using the parameter :
Pm3000 D4 = 0 : Work coordinate system
Pm3000 D4 = 1 : Machine coordinate system

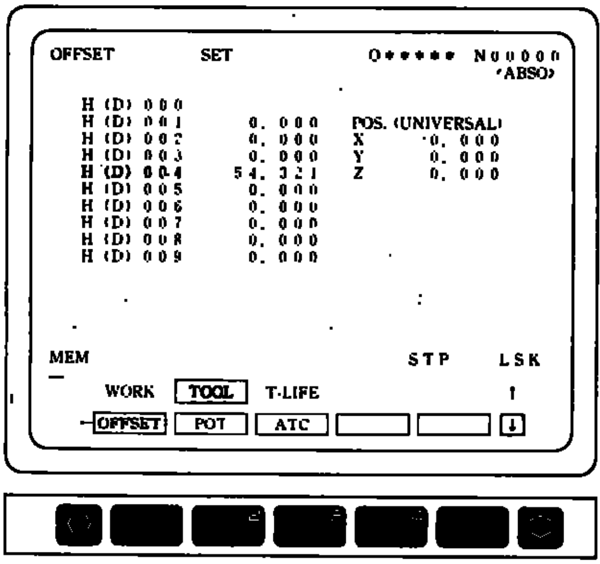
2 Place the cursor at the offset number to be changed, enter new data, then depress the WR key. The offset is changed.

(For example, 5, 4, ., 3, 2, 1, WR)

NOTE To reduce an offset to zero in the incremental input mode, do as follows:



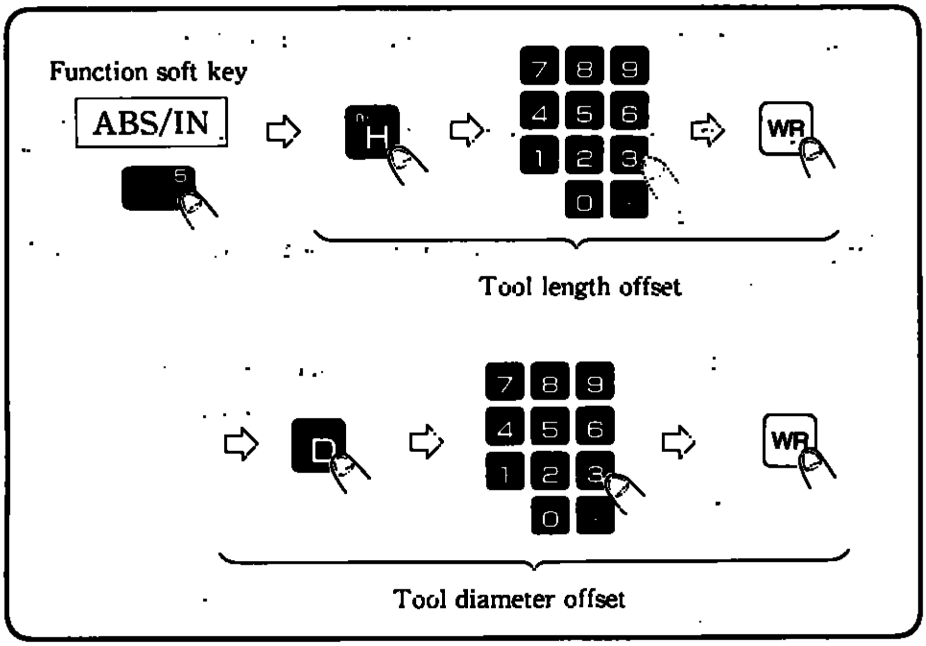
Place the cursor at the value to be reduced to zero.



```
OFFSET            SET                O****** N00000
                                            <ABSO>
   H (D) 000
   H (D) 001          0. 000   POS. (UNIVERSAL)
   H (D) 002          0. 000   X         0. 000
   H (D) 003          0. 000   Y         0. 000
   H (D) 004         54. 321   Z         0. 000
   H (D) 005          0. 000
   H (D) 006          0. 000
   H (D) 007          0. 000
   H (D) 008          0. 000
   H (D) 009          0. 000

MEM                                   STP    LSK
      WORK    TOOL     T-LIFE                 ↑
    OFFSET    POT      ATC                    ↓
```

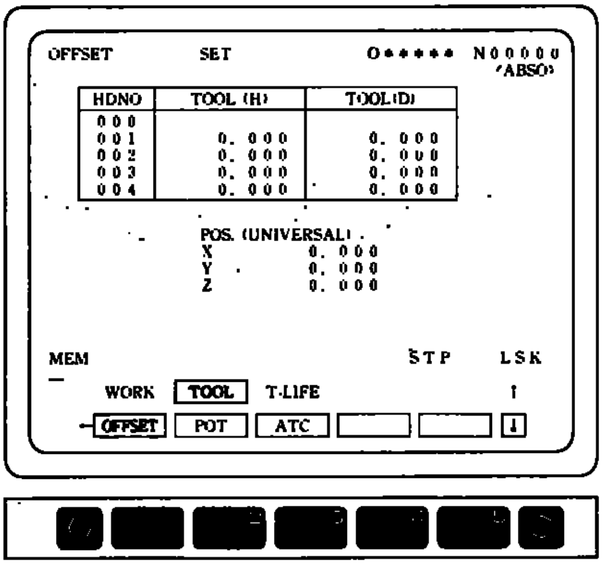
Fig. 3.4.55 Screen after Changing Offset Number

(b) H, D separate type



■ Depress the ABS/IN function soft key to select absolute or incremental input.

NOTE The ABS/IN function soft key is an alternate switch. ‹INCR› or ‹ABSO› is displayed on the upper right part of the screen indicating the current state.



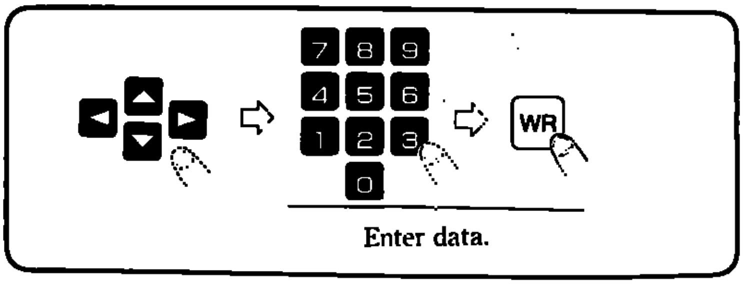
```
OFFSET          SET           O******  N00000
                                         ‹ABSO›
     HDNO    TOOL (H)       TOOL(D)
     000
     001        0.000          0.000
     002        0.000          0.000
     003        0.000          0.000
     004        0.000          0.000

             POS. (UNIVERSAL)
             X          0.000
             Y          0.000
             Z          0.000

MEM                                STP    LSK
—
    WORK   [TOOL]   T-LIFE                 ↑
  ←[OFFSET] [POT] [ATC] [    ] [    ] [↓]
```

Fig. 3.4.56 Offset Screen (for H, D separate type)

NOTE Position data displayed on this screen can be converted to machine coordinates using the parameter:

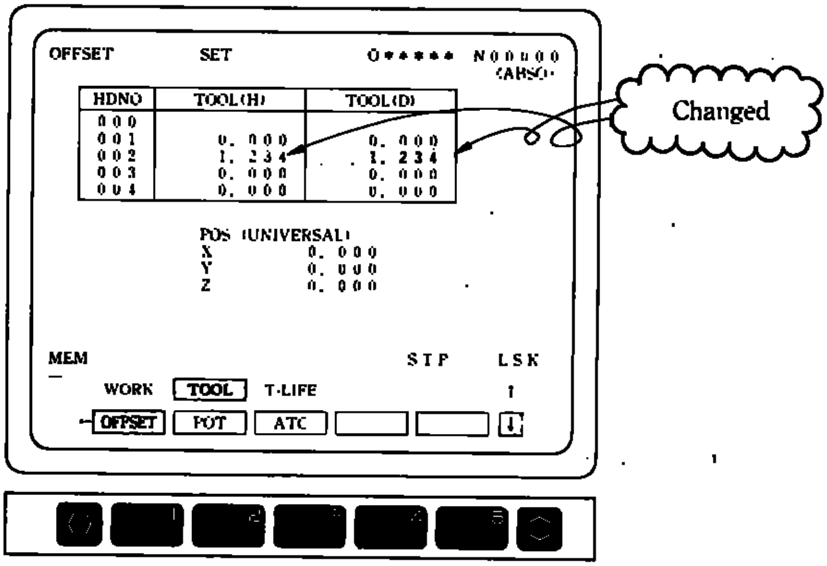
Pm3000 D4 = 0 : Work coordinate system
Pm3000 D4 = 1 : Machine coordinate system

2 Depress cursor keys ▲, ▼, ◀ or ▶ to move the cursor to the offset column to be written. Then enter offset data and depress the WR key.



Enter data.

3 The offset is changed as shown in Fig. 3.4.57:



```
OFFSET          SET                 O*****  N00000
                                             <ABSO>
   HDNO    TOOL(H)      TOOL(D)
   000
   001       0.000        0.000
   002       1.234        1.234
   003       0.000        0.000
   004       0.000        0.000

          POS (UNIVERSAL)
          X          0.000
          Y          0.000
          Z          0.000

MEM                                 STP     LSK
       WORK   [TOOL]   T-LIFE                ↑
   ←[OFFSET] [POT] [ATC] [    ] [    ]      [↓]
```

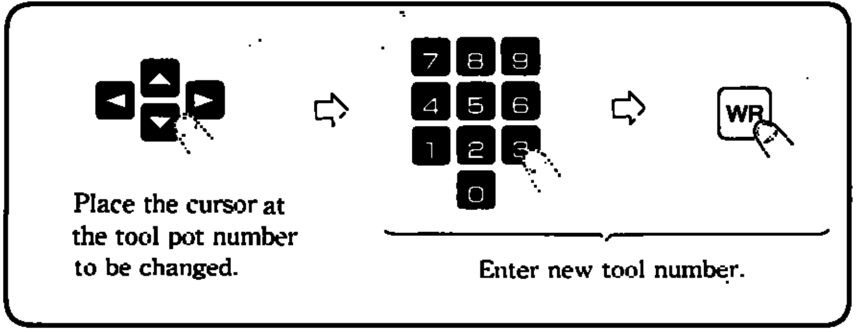
Fig. 3.4.57 Offset Change Screen

(2) Tool pot function

Either of the following types is selected according to parameter setting:

Pm3001 D6 = 0 : Byte indication type ——— See (a)
Pm3001 D6 = 1 : Word indication type ——— See (b)

(a) Byte indication type

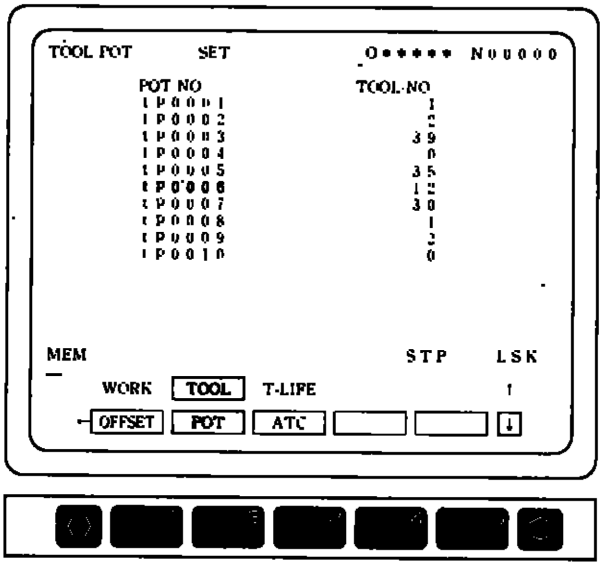


Place the cursor at the tool pot number to be changed.

Enter new tool number.

1. Place the cursor at the tool pot number to be changed. The cursor can be moved by cursor key ▲ or ▼. Otherwise, the target tool pot number can be searched for by entering the number by digit keys and depressing the cursor key ▲ or ▼.

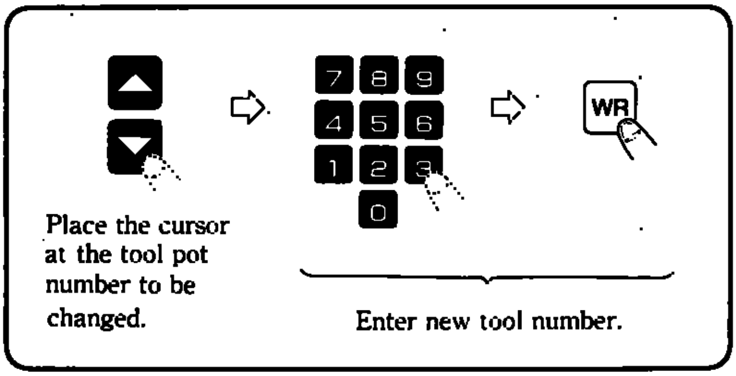
2. Enter a new tool number and depress the WR key.
   (For example, 1 , 2 , WR)



```
TOOL POT        SET             O******  N00000

        POT NO                  TOOL-NO
        1P0001                        1
        1P0002                        2
        1P0003                       39
        1P0004                        0
        1P0005                       35
        1P0006                       12
        1P0007                       30
        1P0008                        1
        1P0009                        2
        1P0010                        0

MEM                                 STP    LSK
```

Fig. 3.4.58 Byte Indication Screen

(b) Word indication type



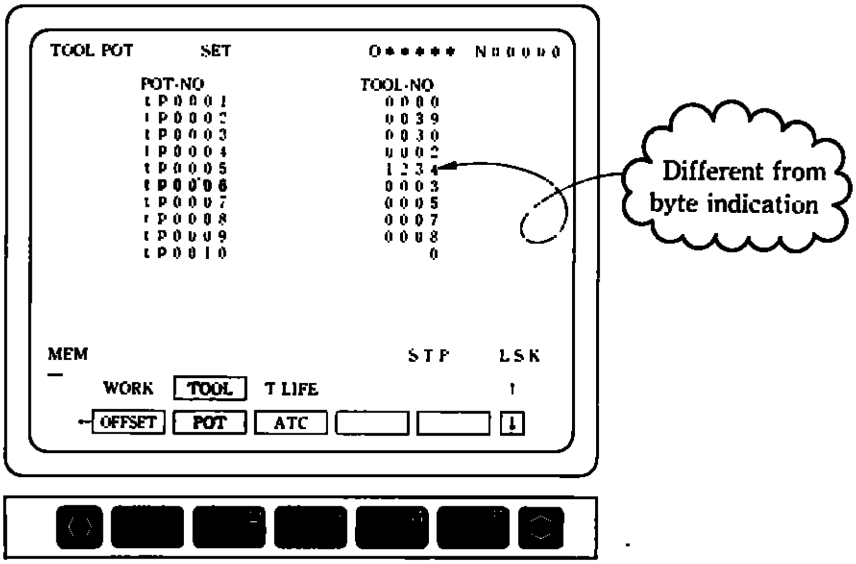
Place the cursor at the tool pot number to be changed.

Enter new tool number.

1. Place the cursor at the tool pot number to be changed. The cursor can be moved by cursor key ▲ or ▼. Otherwise, the target tool pot number can be searched for by entering the number by digit keys and depressing the cursor key ▲ or ▼.

2. Enter a new tool number and depress the WR key.

   (For example, 1, 2, 3, 4, WR)



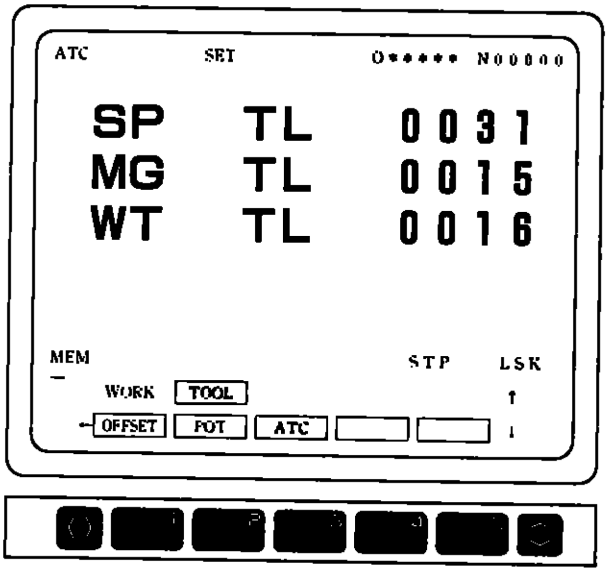
```
TOOL POT        SET               O******  N00000

        POT-NO                   TOOL-NO
        P0001                     0000
        P0002                     0039
        P0003                     0030
        P0004                     0002
        P0005                     1234
        P0006                     0003
        P0007                     0005
        P0008                     0007
        P0009                     0008
        P0010                        0

MEM                                  STP     LSK
    WORK   [TOOL]   T LIFE                    ↑
  ←[OFFSET] [POT]   [ATC]   [    ]  [    ]   [↓]
```

Different from byte indication

Fig. 3.4.59 Word Indication Screen

(3) ATC function

Depress the ATC function soft key. The ATC tool status is displayed.



```
ATC            SET                O****** N00000

  SP     TL     0031
  MG     TL     0015
  WT     TL     0016

MEM                               STP    LSK
```

Fig. 3.4.60 ATC Tool Status Screen

SP : Tool number of the tool mounted on the spindle.
MG : Tool number of the tool at the tool replacement position in the tool magazine.
WT : Tool number of the tool in the middle pot.

NOTE Whether or not the middle pot tool number is displayed depends on the hardware.

(4) Tool length measurement function

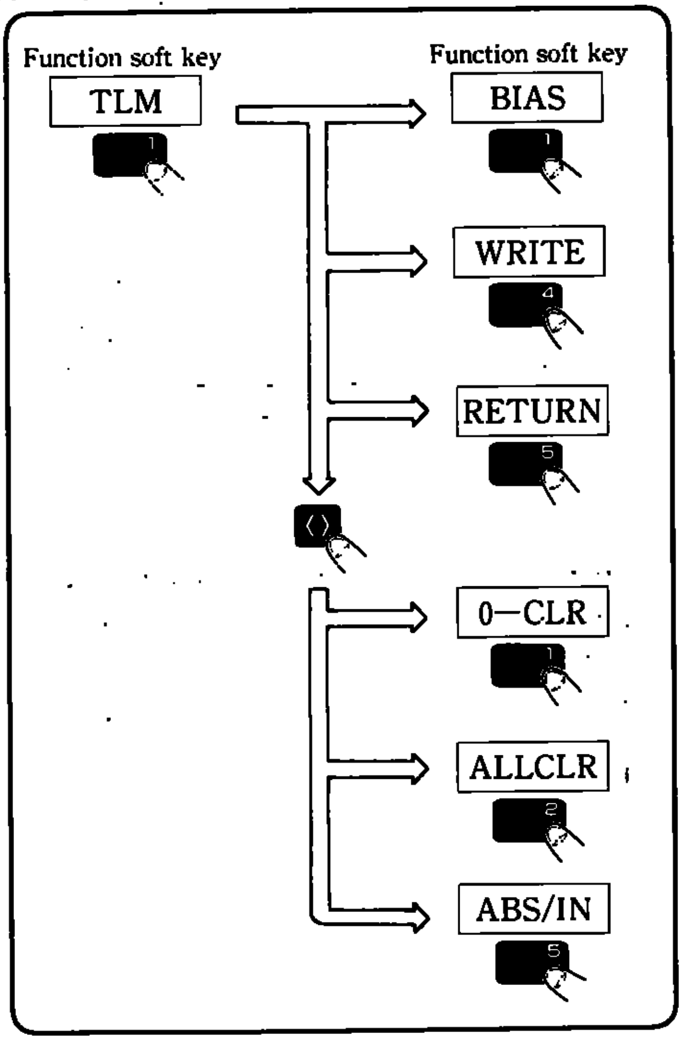
Measure tool offsets for the tool length offset commands. (G43, G44, and G49.)

Either of the following types is selected according to parameter setting:

Pm6008 D0 = 0 : Measurement of length of travel —— **See (a)**
Pm6008 D0 = 1 : Measurement of remainder length —— **See (b)**

(a) Measurement of length of travel

KEYPOINT



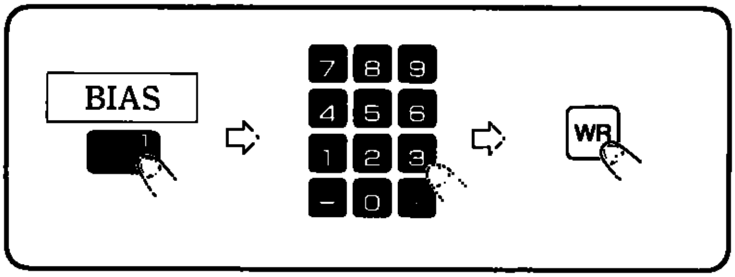
1. Depress the TLM function soft key.

2. The BIAS, WRITE, and RETURN function soft keys are displayed.

(i) Bias function

Bias for tool length measurement can be set up.

KEYPOINT

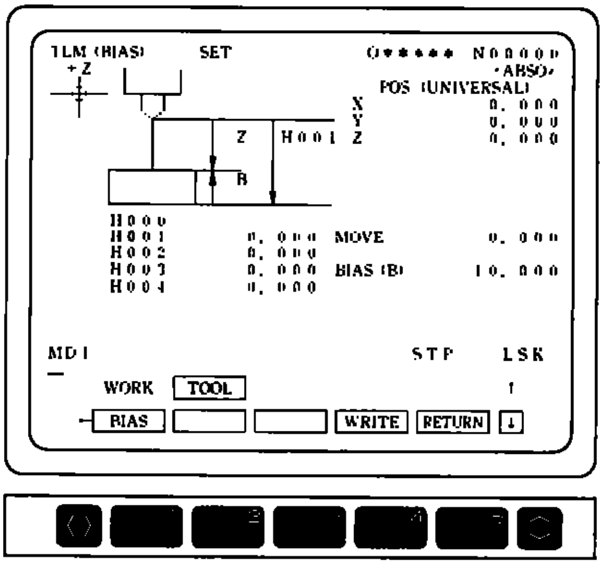


1. Depress the BIAS function soft key. The "BIAS (B)" is highlighted.

2. Enter new bias data, then depress the WR key.

   (For example, 1 , 0 , . , 0 , 0 , 0 , WR)

   The bias data are rewritten and the cursor highlight goes out.



```
TLM (BIAS)       SET               O****** N00000
 +Z                                       ·ABSO·
                                  POS (UNIVERSAL)
                  Z    H001  X             0.000
                             Y             0.000
                             Z             0.000
               B
H000
H001            0.000  MOVE                0.000
H002            0.000
H003            0.000  BIAS (B)           10.000
H004            0.000

MDI                               STP    LSK
     WORK  [TOOL]                          ↑
  -[BIAS] [    ] [    ] [WRITE] [RETURN] [↓]
```

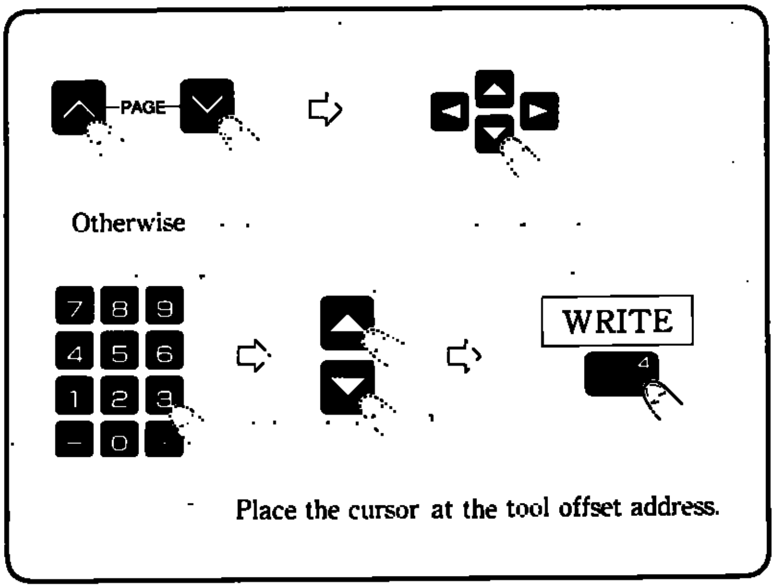
Fig. 3.4.61 Screen for Setting Bias for Tool Length Measurement (length of travel)

NOTE Align the tip of the tool to the contact level by manually operating the Z axis. The contact level is an arbitrary level determined in reference to the surface of the workpiece or the reference level. The difference between the contact level and the reference level must be preset as the tool length measurement bias for parameter Pm806. The input unit for the parameter is 1. The difference represents the thickness of a block gage, which must be positive.

(ii) **Write function soft key**

Distance from the reference position to the reference level can be stored at a specified address of offset memory.

KEYPOINT



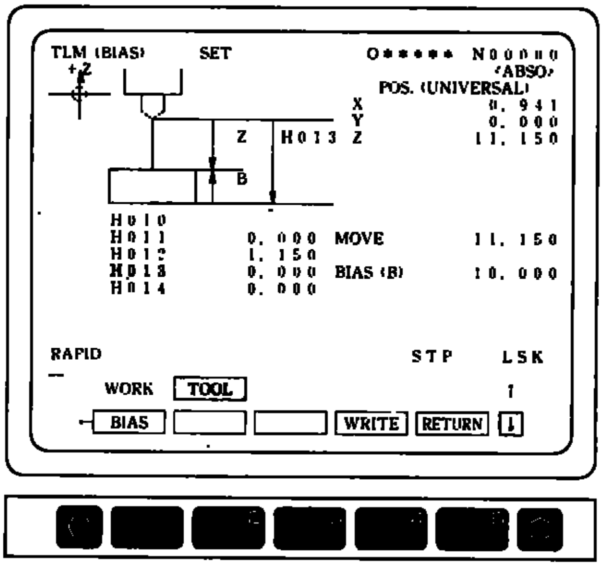
Place the cursor at the tool offset address.

■ Place the cursor at the tool offset address where data are to be written using the page key ▲ or ▼ and cursor key ▲, ▼, ◀, or ▶.

Otherwise, enter the target tool offset address and depress the cursor key ▲ or ▼. The tool offset page containing the input offset address is displayed and the cursor moves to the input offset address.

(For example, 1, 2, ▼)

2 Depress the WRITE function soft key.



```
TLM (BIAS)      SET                O****** N00000
 +Z                                        <ABSO>
                                  POS. (UNIVERSAL)
                             X              0. 941
                             Y              0. 000
               Z   H013      Z             11. 150
                  B
      H010
      H011         0. 000  MOVE            11. 150
      H012         1. 150
      H013         0. 000  BIAS (B)        10. 000
      H014         0. 000

RAPID                              STP      LSK
—
     WORK  [TOOL]                             1
  ←[BIAS] [    ] [    ] [WRITE] [RETURN] [↓]
```

Fig. 3.4.62 Display of Tool Offset Address Contents (measurement of length of travel)

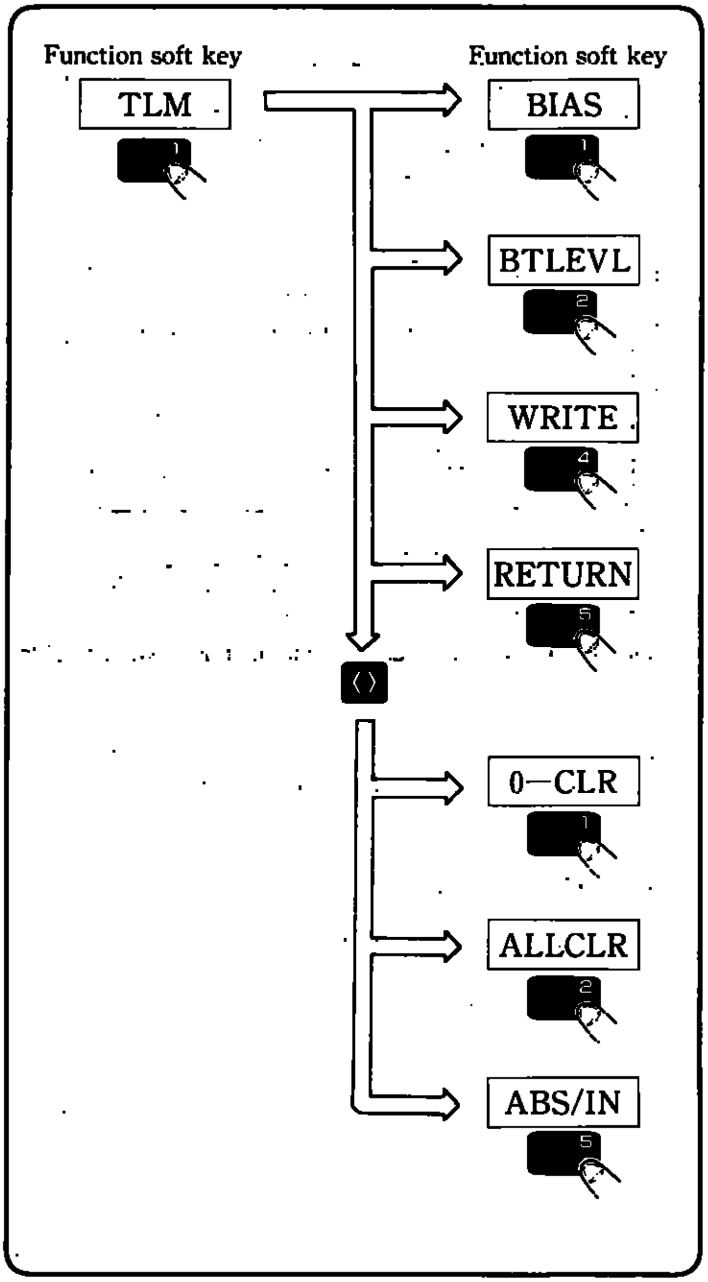
NOTE After the above operation, an offset address greater than the one specified by 1 is added on the screen to get ready for the next tool offset write. (After H99 has been specified, H01 is added.)

(iii) Return function soft key

Depress the RETURN function soft key to return to the offset screen.

(b) Measurement of remainder length

KEYPOINT



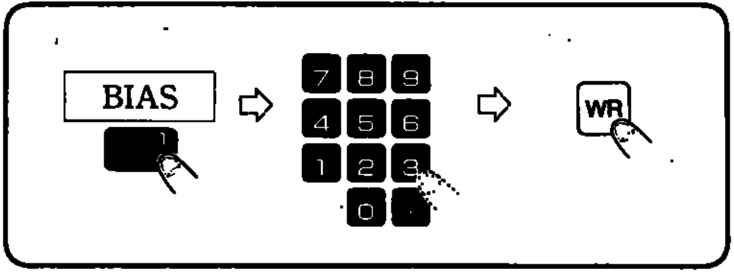
1. Depress the TLM function soft key.

2. The BIAS, BTLEVL, WRITE, and RETURN function soft keys are displayed.

(i) Bias function

Bias for tool length measurement can be set up.

KEYPOINT

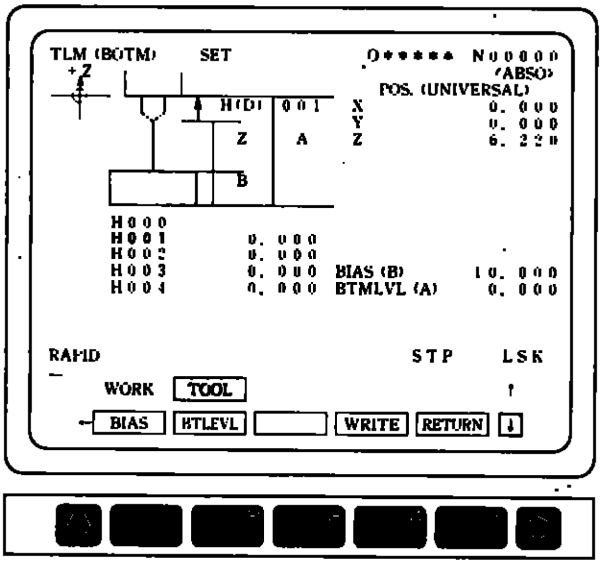


1. Depress the BIAS function soft key. "BIAS (B)" is highlighted.

2. Enter bias data, then depress the WR key.

   (For example, 1 , 0 , . , 0 , 0 , 0 , WR)

The bias data are written and the cursor highlight goes out.

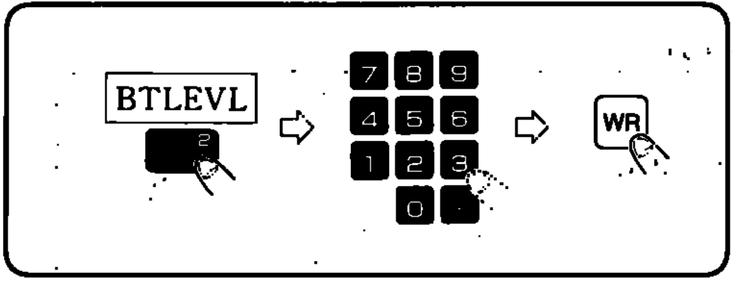


```
TLM (BOTM)      SET                 O******  N00000
                                              (ABSO)
                                    POS. (UNIVERSAL)
               H(D)  001      X              0. 000
                              Y              0. 000
                  Z    A      Z              6. 220

                 B

H000
H001        0. 000
H002        0. 000
H003        0. 000  BIAS (B)         10. 000
H004        0. 000  BTMLVL (A)        0. 000

RAPID                             STP     LSK
      WORK  TOOL
   BIAS  BTLEVL        WRITE  RETURN  ↓
```

Fig.3.4.63 **Screen for Setting Bias for Tool Length Measurement (remainder length)**

NOTE Align the tip of the tool to the contact level by manually operating the Z axis. The contact level is an arbitrary level determined in reference to the surface of the workpiece, or the reference level. The difference between the contact level and the reference level must be preset as the tool length measurement bias for parameter Pm806. The input unit for the parameter is 1. The difference represents the thickness of a block gage, which must be positive.

(ii) Bottom level function soft key

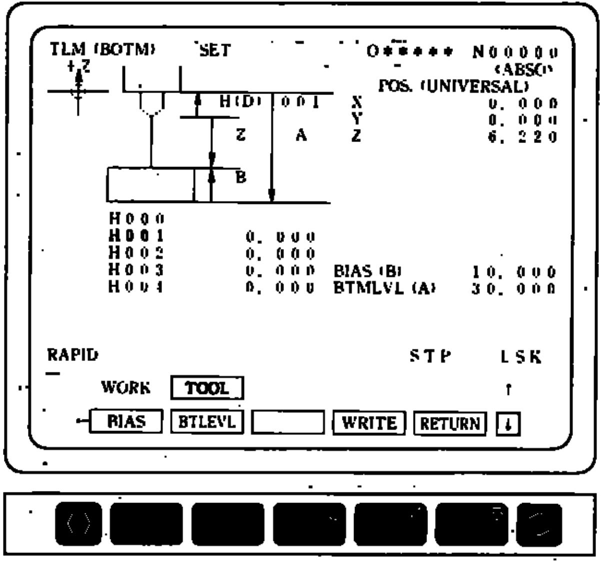


1 Depress the BTLEVL function soft key. BTMLVL (A) is highlighted.

2 Enter bottom level data, then depress the WR key.

(For example, 3, 0, ., 0, 0, 0, WR)

The bottom level data are written and the cursor highlight goes out.



```
TLM (BOTM)     SET              O*****  N00000
 +Z                                      (ABSO)
                                 POS. (UNIVERSAL)
            H(D)  001    X                 0.000
                         Y                 0.000
              Z     A    Z                 6.220
              B
      H000
      H001         0.000
      H002         0.000
      H003         0.000  BIAS (B)         10.000
      H004         0.000  BTMLVL (A)       30.000

RAPID                             STP      LSK
      WORK  TOOL                            ↑
   ←  BIAS  BTLEVL        WRITE  RETURN     ↓
```

Fig.3.4.64 Bottom Level Setting Screen

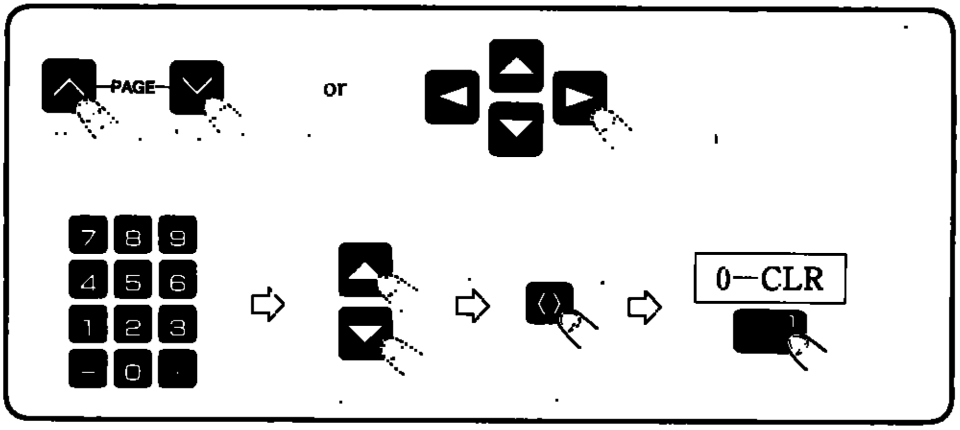
NOTE The bottom level, which is the reference level for measuring a remainder length, can also be set with a signed value as the distance from the reference position by parameter Pm807. The input unit for the parameter is 1.

The subsequent measurement can be made in the same way as in (2) and (3) of measurement of length of travel.

(iii) **Other function soft keys**

(1) **Memory clear**

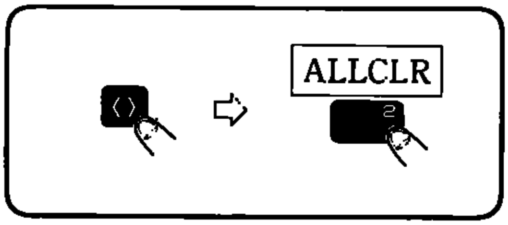
KEYPOINT



Depress the 0—CLR function soft key. The offset data at the address indicated by the highlighted cursor are reduced to zero.

(2) **Memory all clear**

KEYPOINT



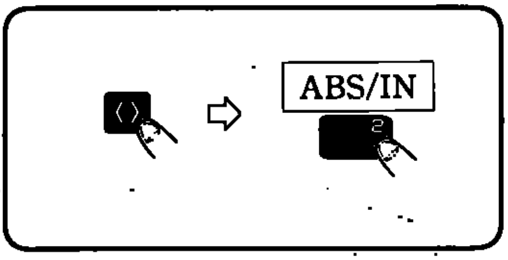
1 Depress the ALLCLR function soft key. "CLEAR ALL OFFSET (Y/N)" is displayed.

2 Depress Y to clear the entire offset memory contents.

Depress N to cancel memory all clear.

(iii) Write input mode switch function soft key

KEYPOINT



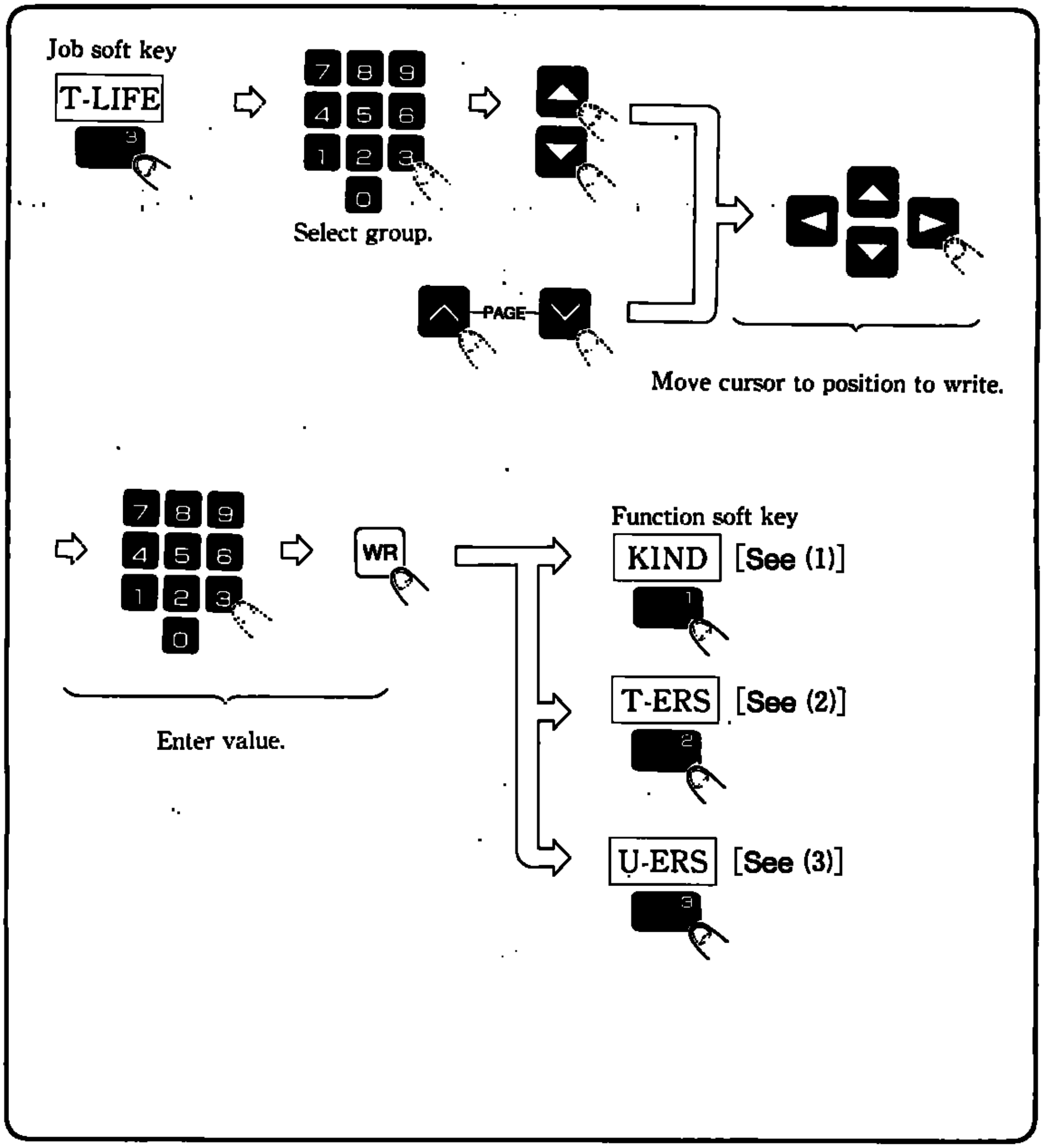
Depress the ABS/IN function soft key to select the incremental or absolute data input. Each time the ABS/IN function soft key is depressed, the incremental input mode and the absolute input mode change alternately.

NOTE Offset data indicated by the cursor can be rewritten by entering new data and depressing the WR key on the tool length measurement screen.

3.4.6.3 Tool life control job

Tools are classified into groups and life is specified for each tool.

KEYPOINT of operation

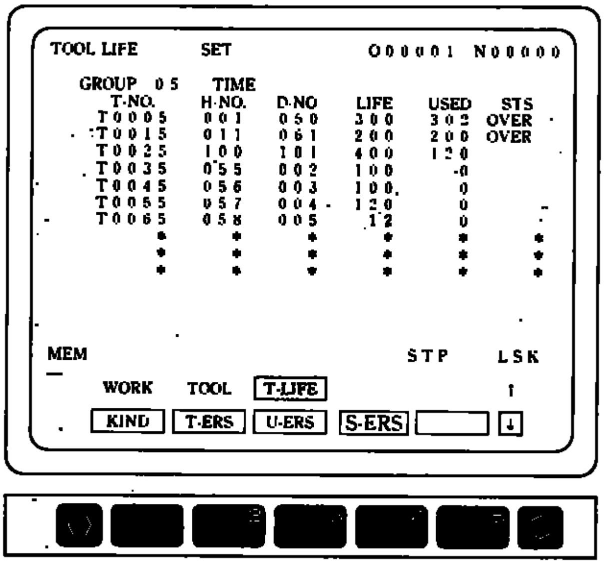


1. Depress the T-LIFE job soft key.

2. Depress page keys ⌃ or ⌄, or digit keys and cursor keys ▲ or ▼, to select the group to be displayed.

   Depress cursor keys ▲ or ▼ to move the cursor to the position of write. Depress digit keys and the WR key to enter data.

   (Example: 1 2 WR)



```
TOOL LIFE      SET                O00001  N00000

   GROUP  05   TIME
     T-NO.     H-NO.   D-NO    LIFE    USED   STS
     T0005     001     050     300     302    OVER
     T0015     011     061     200     200    OVER
     T0025     100     101     400     120
     T0035     055     002     100       0
     T0045     056     003     100       0
     T0055     057     004     120       0
     T0065     058     005      12       0
         *       *       *       *       *      *
         *       *       *       *       *      *
         *       *       *       *       *      *

MEM                                   STP    LSK
      WORK    TOOL    [T-LIFE]                 ↑
     [KIND] [T-ERS] [U-ERS] [S-ERS] [     ]  [↓]
```

Fig.3.4.65 Tool Life Control Data Input Screen

(1) Selection of tool life control type (KIND)

The type of tool life control can be selected.

By depressing the KIND function key, the following pop-up menu and the message "SELECT BY CURSOR & WR" are displayed.

To select the type of tool life control, move the cursor to the desired type in the pop-up menu by depressing the ▲ or ▼ key, and depress the WR key. When nothing should be changed, depress the KIND function key again so that the pop-up menu disappears.

Pop-up menu

TIME	Tool life is controlled by the length of time the tool has been used.
COUNT	Tool life is controlled by the number of times the tool has been used.
DIST	Tool life is controlled by the distance in which the tool has been used.
DRILL	Tool life is controlled by the number of holes machined by the tool.

NOTE The selected tool life control type is valid only for the tool group for which it is written in.

(2) Erasing tool number (T-ERS)

The data of the selected tool number are erased.

By depressing the T-ERS 2 function key, the following pop-up menu and the message "SELECT BY CURSOR & WR" are displayed. Move the cursor to the item to be erased by depressing the ▲ or ▼ key, and press the WR key. If nothing should be changed, depress the T-ERS 2 function key again so that the pop-menu is disappeared.

Pop-up menu

EACH	The data of the tool number indicated by the cursor are erased.
GROUP	The data of the tools in the displayed group are erased.
ALL	The data of all groups are erased.

NOTE When tool data are erased for all groups or the selected group, the tool life control type of such groups is automatically set at "TIME".

(3) Erasing tool use count data (U-ERS)

The data of tool use count is erased.

By depressing the U-ERS 3 function key, the following pop-up menu and the message "SELECT BY CURSOR & WR" are displayed. Move the cursor to the item to be erased by depressing the ▲ or ▼ key, and depress the WR key. If nothing should be changed, press the U-ERS 3 key again so that the pop-up menu disappears.

Pop-up menu

GROUP	The data of the tools in the displayed group are erased.
ALL	The data of all groups are erased.

(4) Erasing tool life status data (S-ERS)

The data of tool life status are erased.

By depressing the S-ERS function key, the following pop-up menu and the message "SELECT BY CURSOR & WR" are displayed. Move the cursor to the item to be erased by depressing the ▲ or ▼ key, and depress the WR key. If nothing should be changed, depress the S-ERS function key again so that the pop-up menu disappears.

Pop-up menu

EACH	The status data of tool number indicated by the cursor are erased.
GROUP	The status data of tools in the displayed group are erased.
ALL	The status data of all groups are erased (blank).

NOTE

1. The LIFE, USE, and STS are closely related to each other.
2. "OVER" is displayed when "LIFE" ≦ "USE".
   The data can be corrected when the edit lock is OFF and the NC is in the automatic operation suspended state (reset status). This also applies to the erase operation. For the change and erase of the LIFE, USE, and STS data, data change and erase operation are possible in the single-block stop status, with an exception of the tool data in the tool life control mode.
3. When the setting of the number of groups and the number of tools (parameter Pm0009 D6, D7) is changed, the display changes at the time the parameter setting is valid (at the time the reset key is pressed).

3.4.7 Run Process RUN

Created part programs are executed in the run process.
During the process, status of program operation and command values of the running program are displayed. Also, internal switches can be displayed and set up.
The above jobs are classified as the program job, command value job, and setting job.

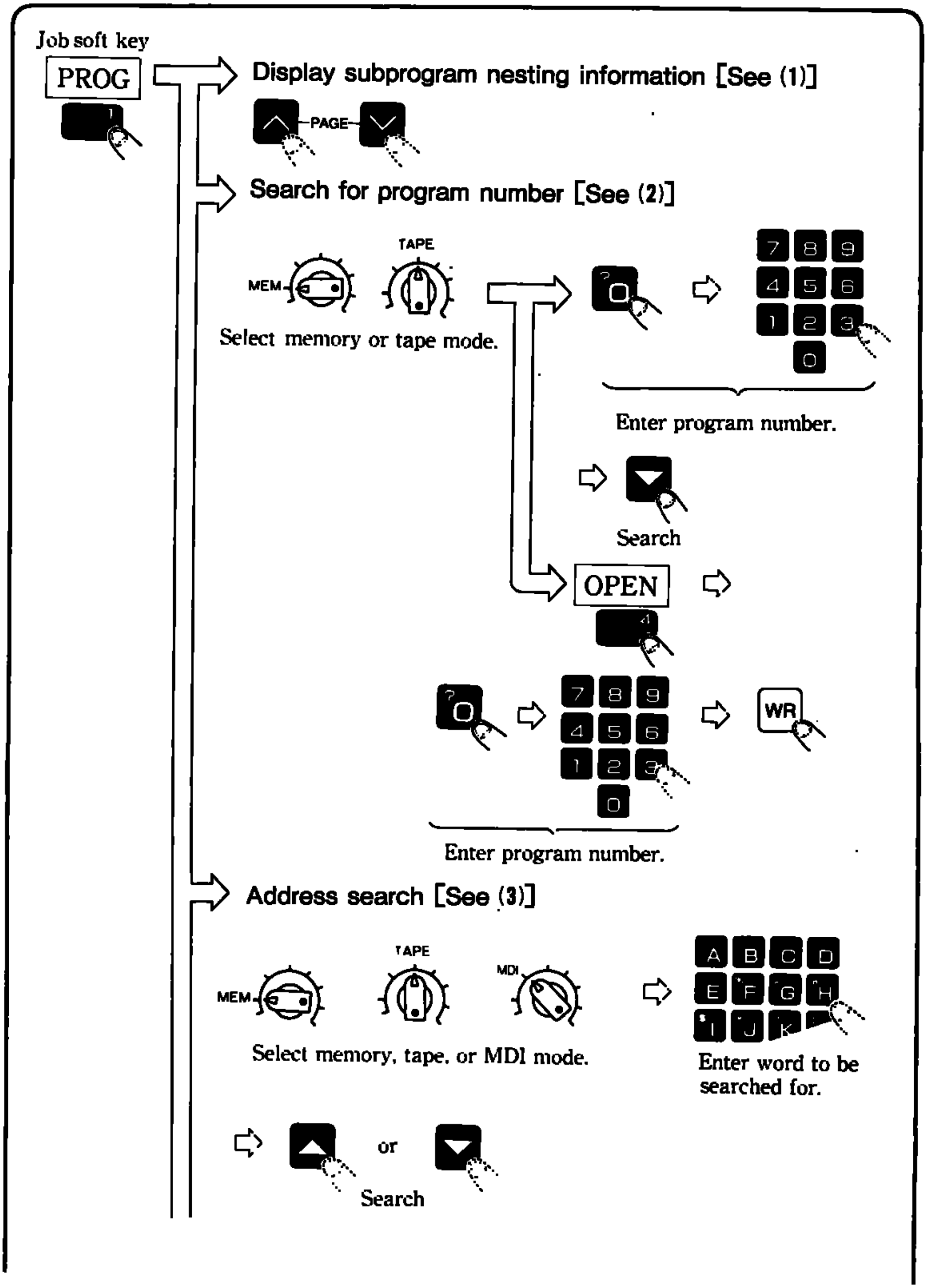
1 Depress the RUN key. Any of the following jobs is displayed:

- Program job ——— See Par. 3.4.7.1 (page 479.)
- Command value job ——— See Par. 3.4.7.2 (page 493.)
- Setup job ——— See Par. 3.4.7.3 (page 495.)
- NC path drawing job ——— See Par. 3.4.7.4 (page 505.)

2 To display a necessary job, depress the corresponding job soft key (PROG 1, COMMND 2, or SETING 3.) Otherwise, depress the RUN key to switch the displayed job.

3.4.7.1 Program job (job soft key PROG)

KEYPOINT operation



Display subprogram nesting information [See (1)]

Search for program number [See (2)]

Select memory or tape mode.

Enter program number.

Search

Enter program number.

Address search [See (3)]

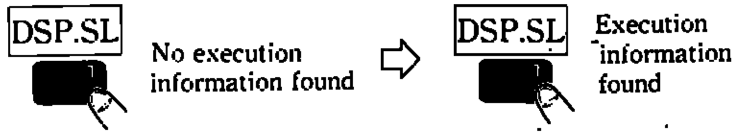
Select memory, tape, or MDI mode.

Enter word to be searched for.

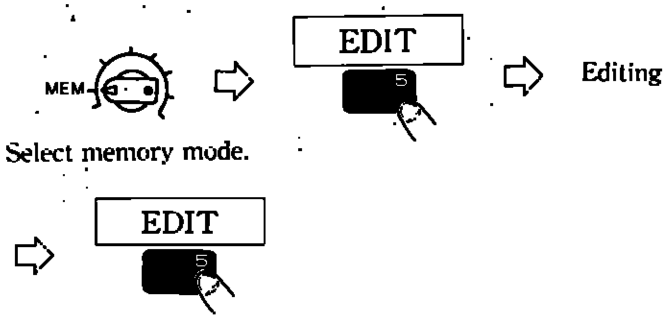
or

Search

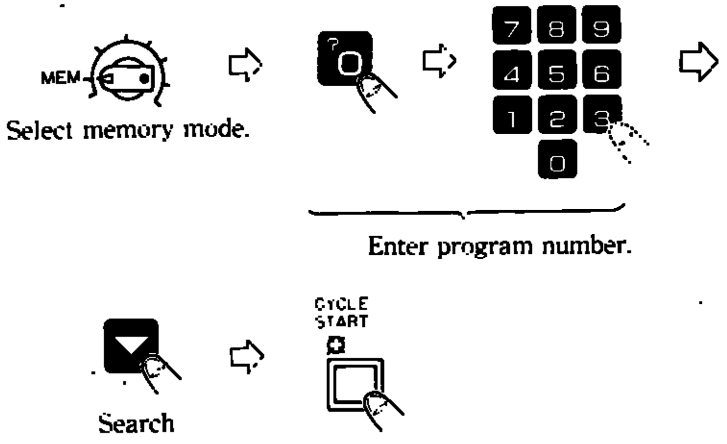
Select display [See (4)]



Edit part program memory [See (5)]

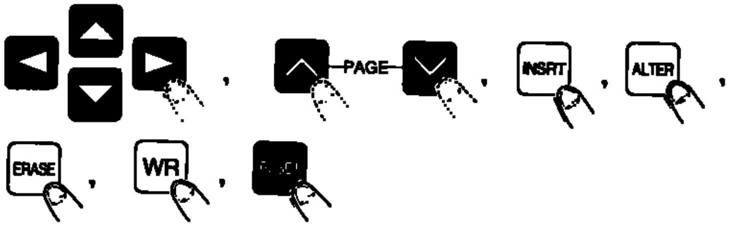


Run part program [See (6)]

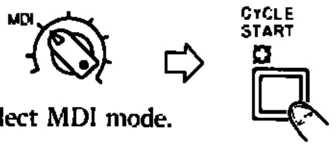


Enter MDI program [See (7)]

Edit MDI program [See (8)]



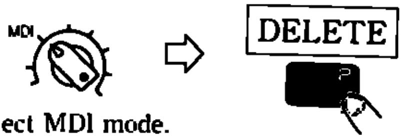
Run MDI program [See (9)]



Select MDI mode.

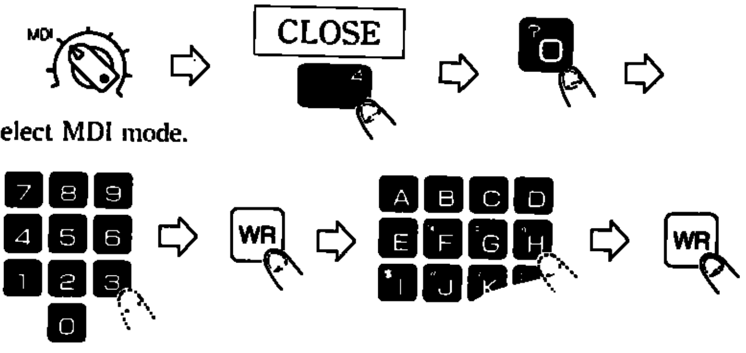
NOTE Place the cursor at the beginning of the MDI program.

Erase MDI program [See (10)]



Select MDI mode.

Store MDI program [See (11)]

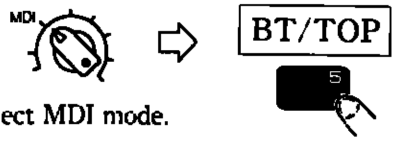


Select MDI mode.

Enter program number.

Enter comment.

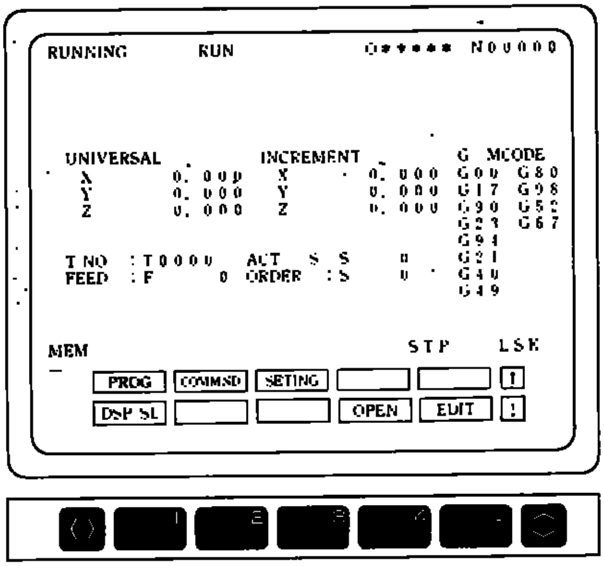
Move to last/first line of MDI program [See (12)]



Select MDI mode.

Depress the PROG job soft key to display the currently running program

and information of execution.



```
RUNNING        RUN                    O****** N00000

UNIVERSAL          INCREMENT            G   MCODE
 X        0.000    X         0.000      G00  G80
 Y        0.000    Y         0.000      G17  G98
 Z        0.000    Z         0.000      G90  G52
                                        G23  G67
                                        G94
T NO  :T0000    ACT    S  S     0       G21
FEED  :F       0 ORDER  : S     0       G40
                                        G49

MEM                                   STP    LSK
```

Fig. 3.4.66 Program Execution Display Screen

NOTE

1. The work coordinate system positions displayed on the screen can be converted to the mechanical coordinate system by changing the following parameters:
   Pm3000 D5=0 : Work coordinate system
   Pm3000 D5=1 : Mechanical coordinate system
2. When Pm3005 D5=0, Z-axis position can be displayed without adding the compensation amount (G43 Hxx).
   Pm3000 D2=0 : Adds the Z-axis compensation amount.
   Pm3000 D2=1 : Does not add the Z-axis compensation amount.

The program screen is divided into the following four blocks:

● **Execution program display and editing**

Different items are displayed by the execution program display depending on the mode of NC operation.

In the MDI mode, the MDI program is displayed. In the tape mode, the tape program is displayed when operation is started. In other modes, the part program being searched for is displayed.

Depressing the EDIT function soft key in the memory mode makes it possible to edit the part program being displayed. In the edit enable state, the program is displayed in a frame.

● **Commanded G code, M code display**

In the auto mode operation, the modal G codes, non-modal G codes, and M codes being executed or immediately before execution are displayed. The displayed modal G codes are the most important 14 groups among all 23 groups of modal G codes supported by the equipment. On the non-modal G code column, if not commanded, only G is indicated. On the M code column, the commanded M code is displayed with three digits.

- **Work coordinate system and remainding length display**

  This function is helpful for checking a program.

- **Tool number, feedrate, actual r/min., and commanded r/min.**

  The tool number, feedrate, actual r/min., and commanded r/min. being executed or immediately before execution are displayed.

  The following can be also displayed by switching the parameter.

  Pm3003 D5 = 0 : Tool number, feedrate, actual r/min and commanded r/min are displayed.

  Pm3003 D5 = 1 : Actual r/min, feedrate, spindle tool and secondary tool are displayed.

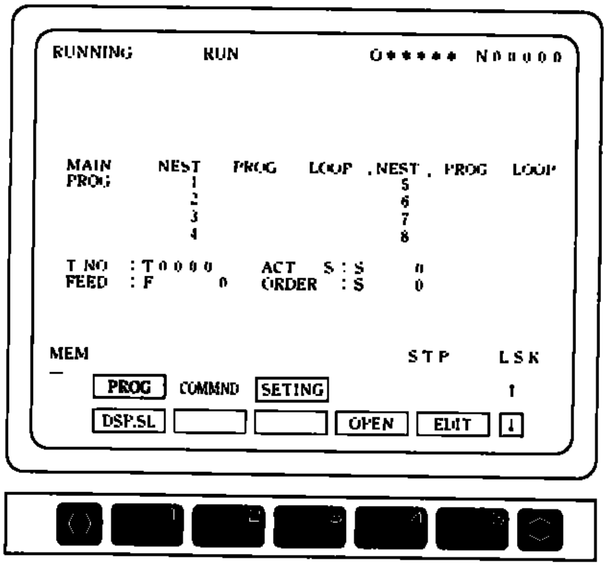
  Pm1000 D4 = 0 : "Specified speed×Override" is displayed for "actual r/min"

  Pm1000 D4 = 1 : "Speed of feed back value" is displayed for "actual r/mim".

NOTE The display of actual r/min is an optional function.

(1) Subprogram nesting information display

Depress page key ∧ or ∨ to display subprogram nesting information.



```
RUNNING          RUN                 O*****  N00000



MAIN      NEST    PROG    LOOP  .NEST .  PROG    LOOP
PROG       1                       5
           2                       6
           3                       7
           4                       8

T NO   : T0000      ACT    S : S        0
FEED   : F        0 ORDER    : S        0


MEM                                  STP      LSK
—
     [PROG]  COMMND  [SETING]                   ↑
     [DSP.SL] [    ] [    ] [OPEN] [EDIT]      [↓]
```

Fig. 3.4.67 Nesting Information Display Screen

In Fig. 3.4.67, the main program number, subprogram numbers, and the number of loops are displayed to a maximum nesting level of eight. The program number of the current running program is indicated with

an arrow. Subprogram call is commanded by M98.
On this screen, macro program call is also displayed.

NOTE If a program is being edited in the memory mode, depressing page key ∧ or ∨ will not display subprogram nesting information. Instead, the editing program page is changed.

(2) **Search for program number**

A program number to be executed can be called by the following two methods:

(a) **Using OPEN function soft key**

1. Select the memory or tape mode on the machine control station.
2. Depress the OPEN function soft key.

   A message requests : "Specify program number to call".
3. Depress O, then enter the program number to call.
4. Depress the WR key.

(b) Entering program number

1. Select the memory or tape mode on the machine control station.

2. Depress O ,then enter the program number to call.

3. Depress cursor keys ▲ or ▼ .

● In memory mode

The specified program number is searched for among stored programs, and called to the screen. During search for the program, the execution program number displayed on the upper right part of the screen changes. If the specified program number is not stored, a warning message indicates "No O-number".

Whether to call the previous target of program number search after power is turned on can be determined by setting the following parameter:

Pm3005 D1 = 0 : Clears the program number at power-on.
Pm3005 D1 = 1 : Does not clear the program number at power-on.

● In tape mode

Program data are read (from tape) via the RS-232C input port (first or second port) that is currently selected, and the specified program number is searched for. During search, message "searching" is displayed. After the number is found, the message stops. If the program number was not found in the read data (from tape), a warning message indicates "O-number not found".

Depressing the CYCLE START button on the machine control station when the search for the program number has terminated restarts reading of program data (from tape), runs the program, and displays the running program.

NOTE
1. If program number search is attempted in the manual mode, a message indicates: "Search is impossible in this mode".
2. Leading zeros in a program number need not be entered.

(3) Address search

Input data (character string) from the NC operator panel are collated with the input data (on tape) from the RS-232C port or memory data (character string). Display stops when these data match. The object of search in the tape mode is the data (on tape) input via the RS-232C port. In the memory mode, the part program ; in the MDI mode, the MDI program.

1 Enter the word to search for.

2 Depress cursor key ▼ or ▲.

- **In memory mode**

Depress cursor key ▼ to search from the cursor toward end of data.

Depress cursor key ▲ to search from the cursor toward beginning of data.

During search, message "searching" is displayed.

- **In tape mode**

Program data (on tape) are from the RS-232C port and the specified character string is searched for. During search, message "searching" is displayed, but the read data are not displayed. Data read is stopped after the specified character string is read.

If the specified character string was not found in the read data (from tape), a warning message indicates "Search word not found".

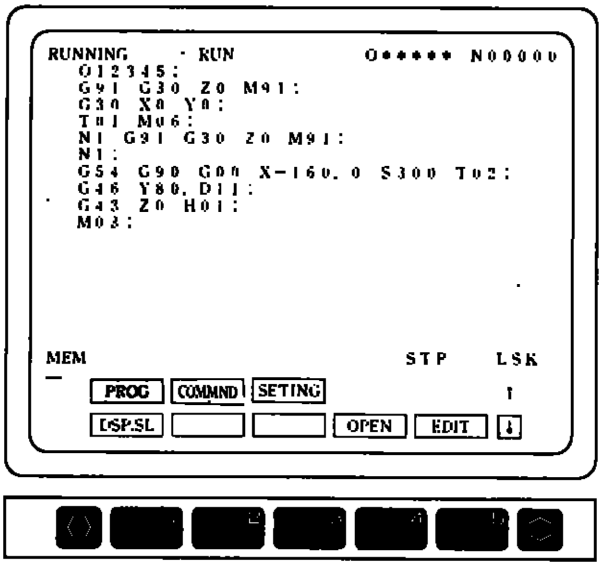
Depressing the CYCLE START ☐ button on the machine control station when address search has terminated restarts reading of program data (from tape), runs the program, and displays the running program.

3 When the search has completed, the message disappears and the cursor indicates the found word. If the specified word was not found a warning message appears.

4 Cycle start after address search starts the program from the block where the cursor is located.

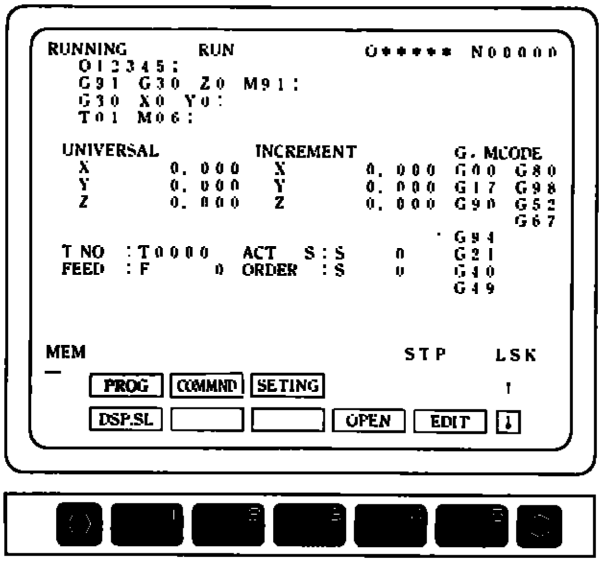
NOTE

1. Leading zeros in data need not be entered for address search. Decimal points are processed during search, according to the status of input in millimeters or inches. Therefore, addresses of commands having different representations of the same meaning (for instance X1. = X1000) can be searched for. (Binary search function)
2. The addresses can not be searched for alarm status.



```
RUNNING      · RUN              O****** N00000
   O12345:
   G91 G30 Z0 M91:
   G30 X0 Y0:
   T01 M06:
   N1 G91 G30 Z0 M91:
   N1:
   G54 G90 G00 X-160.0 S300 T02:
   G46 Y80. D11:
   G43 Z0 H01:
   M03:

MEM                                STP    LSK
```

Fig. 3.4.68 Execution Information Display Screen



```
RUNNING        RUN              O****** N00000
   O12345:
   G91 G30 Z0 M91:
   G30 X0 Y0:
   T01 M06:
 UNIVERSAL          INCREMENT           G. MCODE
   X       0.000      X        0.000   G00  G80
   Y       0.000      Y        0.000   G17  G98
   Z       0.000      Z        0.000   G90  G52
                                            G67
                                       G94
 T NO  :T0000    ACT   S:S      0      G21
 FEED  :F      0 ORDER  :S      0      G40
                                       G49

MEM                                STP    LSK
```

Fig. 3.4.69 Execution Program Display Screen

(4) Editing part program memory

1 · Depress the `EDIT` function soft key when the object part program is selected in the memory mode. The program currently being selected can be edited. When the editing status is entered, a frame appears on the program display screen.
The editing function provides expansion-edit functions, similar to that of the `EDIT` process editing.

2 Four lines of the part program are displayed. Depress the `DSP.SL` function soft key to display 13 lines of the part program.

3 After editing, depress the `EDIT` function soft key again.

The part program is stored and the editing mode is terminated. While the part program is being stored, a message indicates "Storing".

NOTE 1. While the program is running, it cannot be edited if the `EDIT` function soft key is depressed. If the program is in the single-stop state, it can be edited.
2. While the program is being edited, cycle start is impossible. Use cycle start after terminating editing. Note that cycle start starts the program where the cursor was at the end of editing.
3. The `EDIT` function soft key is not displayed in the MDI or tape mode.
4. Even while a program is being edited, the cursor can be moved to the beginning of the program by depressing the REW key.
5. Even while a program is being edited, another program can be called by operation of (2), program number search. If cursor key ▲ or ▼ is depressed after O, the program being edited is searched for character "0".

(5) Running part program

1. Select the memory mode on the machine control station.

2. Enter the program number from the NC operator panel to search for. The searched part program is displayed on the screen and the cursor is highlighted in reverse.

3. Depress the CYCLE START button on the machine control station. The part program is executed and the cursor moves from the beginning of a block to another as the program runs.

NOTE CYCLE START when the cursor is on the block executes the program according to the parameter setting as shown below:
Pm4008 D3 = 0 : Executes from the middle of the block.
Pm4008 D3 = 1 : Executes from the block head.

(6) Entering MDI program

An MDI program of up to 1024 characters can be entered by the following operation:

1. Select the MDI mode on the machine control station.

2. EOB is displayed in the execution program display area. From this state, an MDI program can be entered using the address and data keys.

3. Enter an appropriate address and depress the WR key.

4. The input data are stored in the MDI buffer. Repeat the above operation. UP to 1024 characters can be stored in the MDI buffer. If more than 1024 characters are entered, warning message "OVER MDI BUFFER!" appears.

(7) **Editing MDI program**

An MDI program that has been entered can be edited similar to a part program stored in memory.

1. Depress cursor keys ▲ or ▼ to move the cursor to the preceding or next block in the MDI program.
2. Depress cursor keys ◀ or ▶ to move the cursor to the preceding or next word.
3. Depress page keys ∧ . or ∨ to turn page of the MDI program.
4. Depress the INSRT key to insert the entered data after the current cursor position.
5. Depress the ALTER key to replace the world indicated by the cursor with the entered word.
6. Depress the ERASE key to erase the word indicated by the cursor.
7. Depress the WR key to insert the entered data at the end of the MDI program.
8. Whether to clear the MDI buffer by the [illegible] key can be determined by the following parameter:

   Pm3002 D1 = 0 : Clears MDI buffer by panel or external reset.
   Pm3002 D1 = 1 : Does not clear MDI buffer by panel or external reset.

If the MDI buffer is not cleared by the RESET key, the cursor is as shown below:

Pm4008 D0 = 0 : Cursor position does not change.
Pm4008 D0 = 1 : Cursor position changes.

NOTE
1. While an MDI program is running, it cannot be edited. If the program is in the single-stop state, it can be edited.
2. Use CYCLE START after editing. Note that CYCLE START starts the program where the cursor was at the end of editing.

(9) Running MDI program

1. Retain the MDI mode on the machine control station and depress the CYCLE START button. The entered MDI program is started. The program is started from the current cursor position.
2. As the MDI program runs, the cursor moves from the beginning of a block to another.
3. The MDI program need not be executed in the RUN process. Once an MDI program is entered, it can be executed on any screen.
4. Whether to erase an MDI program after its operation terminates can be determined by the following parameter:

   Pm3002 D0 = 0 : Clears the MDI buffer when MDI has a terminater.
   Pm3002 D0 = 1 : Does not clear the MDI buffer when MDI has terminated if there is the end M code (M02 or M30).

NOTE In the single-block stop state, cursor keys ▲, ▼, ◀, and ▶, and the BT/TOP 5 function soft key can be used to move the cursor or edit the program.

(10) Erasing MDI program

When the mode of the machine control station is changed to MDI, the DELETE 2 function soft key is displayed. Depress the DELETE 2 function soft key to erase the MDI program stored in the MDI buffer.

This key is convenient when the MDI buffer cannot be cleared by resetting from the control station because of the parameter setting, or when the ■ key cannot be used during MDI intervention during memory operation.

(11) **Storing MDI program**

The program in the MDI buffer can be stored into memory.

1 Depress the CLOSE function soft key in the MDI mode.

4

2 A message requests : "INPUT 0 NO". Enter the program number to be stored.

(Example) O 2 O WR

3 A message requests : "INPUT COMMENT". Enter comment. The MDI program is stored with the program number and comment.

(Example : D R I L L WR)

NOTE The CLOSE function soft key appears in the MDI mode only.

(12) **MDI program Bottom/Top**

1 Enter an MDI program in the MDI mode.

2 Depress the BT/TOP function soft key to move the cursor to the last

5

line of the MDI program.

3 Depress the BT/TOP function soft key again to move the cursor to

5

the first line o.f the MDI program.

3.4.7.2 Command value job (job soft key COMMND)

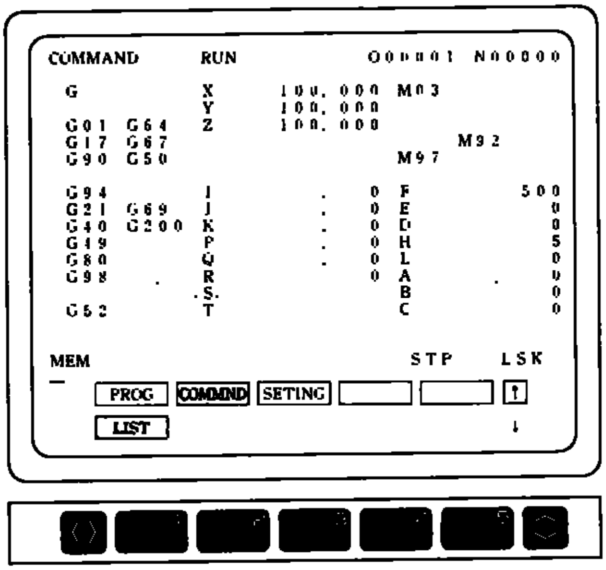
Command values in the currently running program can be displayed in a list.

Depress job soft key COMMND

The command value list screen shown in Fig. 3.4.70 is displayed. On this screen, data of the block being executed in the auto mode are displayed.

Displayed data

- During auto operation or feed hold, the contents of the execution register are displayed.
- During block stop, the contents of the preread buffer to be executed next are displayed. If the preread buffer is <blank>, the preceding block that has been executed is displayed.



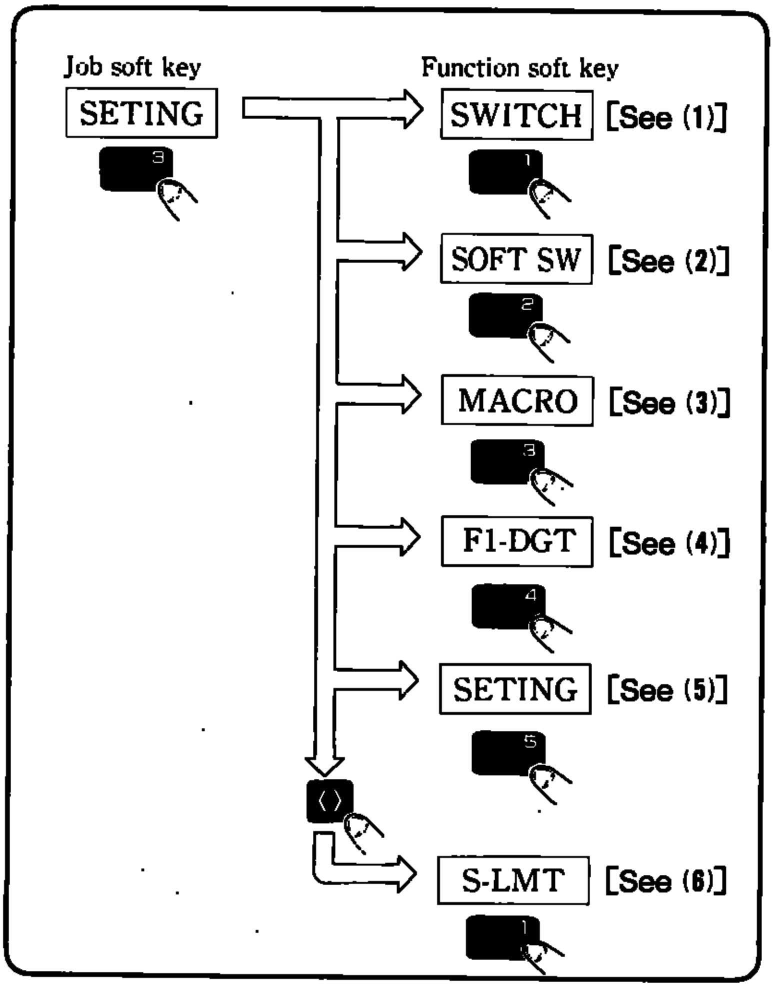
```
COMMAND          RUN              O00001  N00000

  G               X       100.000   M03
                  Y       100.000
  G01   G64       Z       100.000
  G17   G67                                 M92
  G90   G50                         M97

  G94             I           .   0  F            500
  G21   G69       J           .   0  E              0
  G40   G200      K           .   0  D              0
  G49             P           .   0  H              5
  G80             Q           .   0  L              0
  G98             R               0  A              0
                  S                  B              0
  G52             T                  C              0

MEM                                  STP      LSK
```

Fig. 3.4.70 Command Value Display Screen

3.4.7.3 Setting job (job soft key SETING)

Setting data necessary for operation can be set.

KEYPOINT of operation

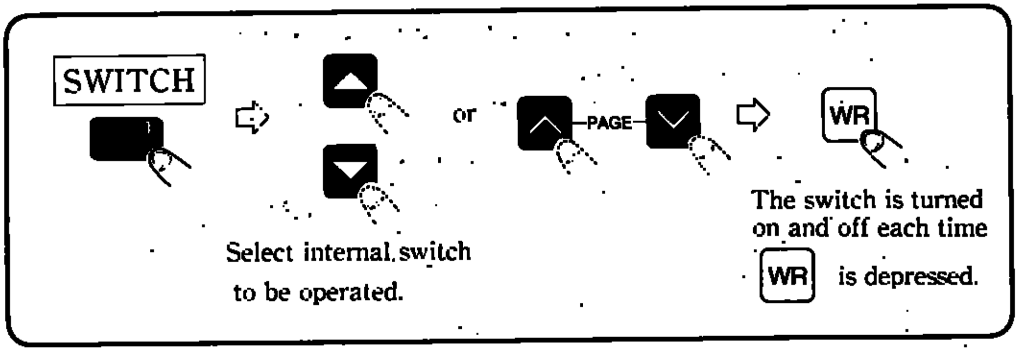


Depress job soft key SETING to display one of the following functions:

- Internal switch function
- Software switch function
- Macro program variable function
- F1-digit function
- Setting function
- Stored stroke limit function

(1) Internal switch function ※

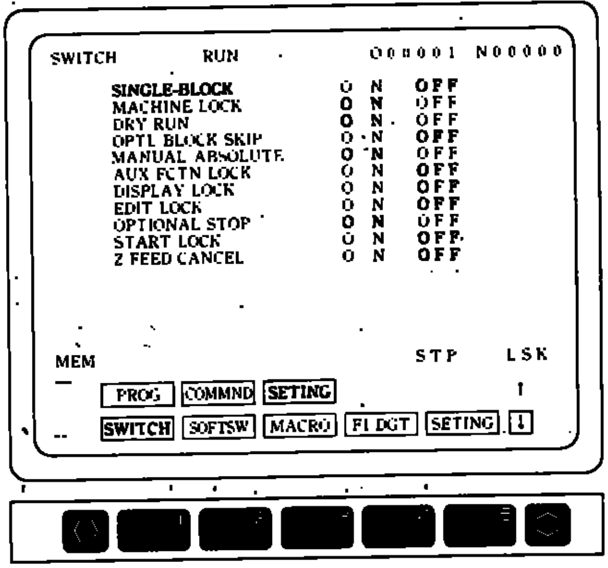
KEYPOINT of operation



Select internal switch to be operated.

The switch is turned on and off each time WR is depressed.

1 Depress function soft key SWITCH. The internal switch screen shown in Fig. 3.4.71 is displayed. The switches on the screen can be turned on or off readily by setting from the NC operator panel even when switch setting is omitted on the machine control station.
There are up to 15 switches on two pages.



```
SWITCH          RUN                O00001  N00000

        SINGLE-BLOCK              O  N   OFF
        MACHINE LOCK              O  N   OFF
        DRY RUN                   O  N   OFF
        OPTL BLOCK SKIP           O  N   OFF
        MANUAL ABSOLUTE           O  N   OFF
        AUX FCTN LOCK             O  N   OFF
        DISPLAY LOCK              O  N   OFF
        EDIT LOCK                 O  N   OFF
        OPTIONAL STOP             O  N   OFF
        START LOCK                O  N   OFF
        Z FEED CANCEL             O  N   OFF


MEM                                      STP    LSK
     PROG   COMMND  SETING                       ↑
     SWITCH SOFTSW  MACRO   FI DGT  SETING       ↓
```

Fig. 3.4.71 Internal Switch Screen

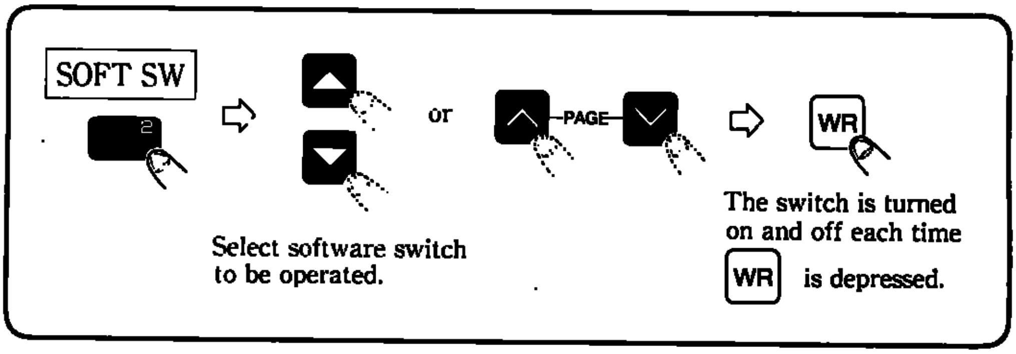
2 Move cursor to the internal switch to be turned on or off, then depress the WR key.

Each time the WR key is depressed, the switch is turned on or off.

The on or off state of the switch takes effect immediately. The number of internal switches depends on the option.

(2) Software switch function ※

KEYPOINT of operation



Select software switch to be operated.

The switch is turned on and off each time WR is depressed.

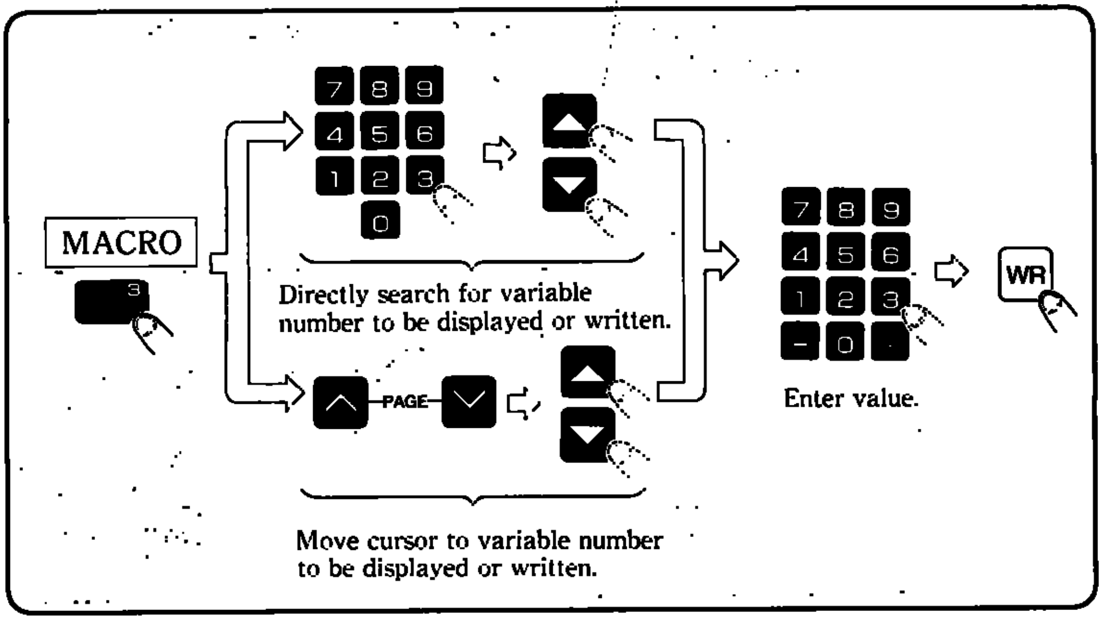
Switches on the machine control station can be substituted by this screen depending on the option.

Depress the SOFT SW function soft key to display the software switch selection screen. Up to 64 switches can be used, which are contained in eight pages. For details, refer to the operation manual by the machine tool builder.

(3) Macro program variable function ※

Local variables (#1 to #33) and common variables (#100 to #299, #500 to #999) can be displayed and written for using macro program function options.

KEYPOINT of operation



■1 Depress the MACRO function soft key. Local variables and the level of nesting are displayed.

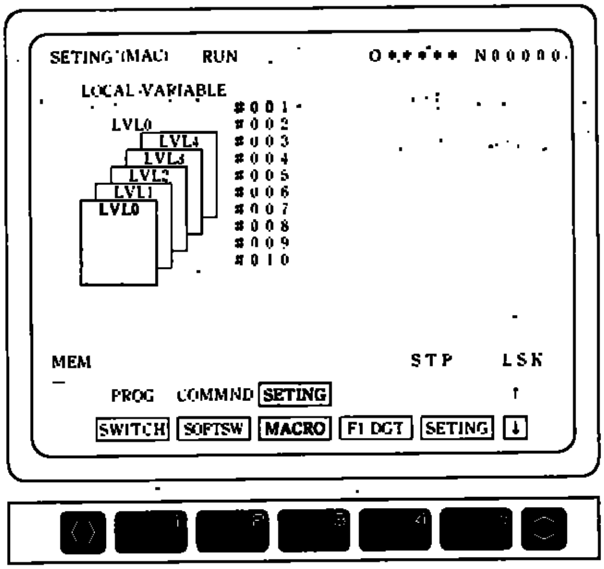


Fig. 3.4.72 Local Variable and Nesting Level Display Screen

NOTE

1. The level of nesting is displayed by the degree of multiplication of the currently running macro program from level 0 to level 4, with an illustration.
2. A local variable at the currently executed level is displayed with a signed number of the integer part with eight digits and fractional part with seven digits.
3. Values of the local variables at another nesting level cannot be displayed on the screen.
4. If data field is blank, the local variable is "empty". After execution, all local variables are cleared to blank.
5. No nesting level is displayed for common variables.

2 ● **Searching directly for variable number**

Enter a variable number and depress cursor key ▲ or ▼.

The variable is directly searched for.

● **Selecting variable number by cursor**

Depress page keys ∧ or ∨ to turn pages of variable display. Depress cursor keys ▲ or ▼ to move the cursor to the variable number to be selected.

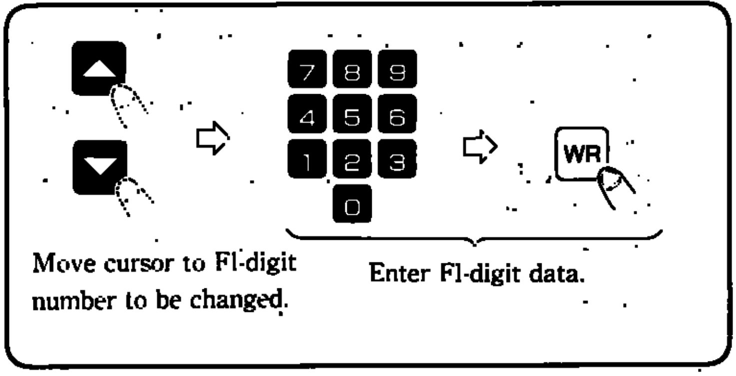
3 Enter a value to be written, then depress the WR key.

The input value is stored for the data of the variable number indicated by the cursor.

NOTE

1. Common variables can be written any time.
2. Local variables can be written only while macro is being executed. At other times, write operation is disregarded. However, it may be dangerous to change a local variable while the macro is runnung. For safety, terminate operation by single-block stop, confirm that the variable can be changed safely, then rewrite the variable.

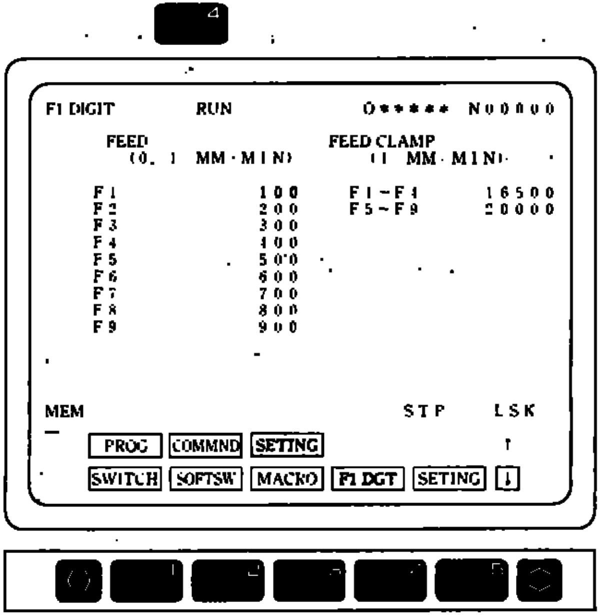
(4) F1-digit function ※



Move cursor to Fl-digit number to be changed.

Enter Fl-digit data.

■1 Depress the F1-DGT function soft key. The F1-digit data setting



```
F1 DIGIT          RUN                O*****  N00000

      FEED                   FEED CLAMP
        (0. 1  MM·MIN)          (1  MM·MIN)

     F1                 100    F1~F4        16500
     F2                 200    F5~F9        20000
     F3                 300
     F4                 400
     F5                 500
     F6                 600
     F7                 700
     F8                 800
     F9                 900



MEM                                    STP     LSK
—
       PROG    COMMND   SETING                  ↑
       SWITCH  SOFTSW   MACRO   F1 DGT  SETING  ↓
```

Fig. 3.4.73 F1-digit Screen

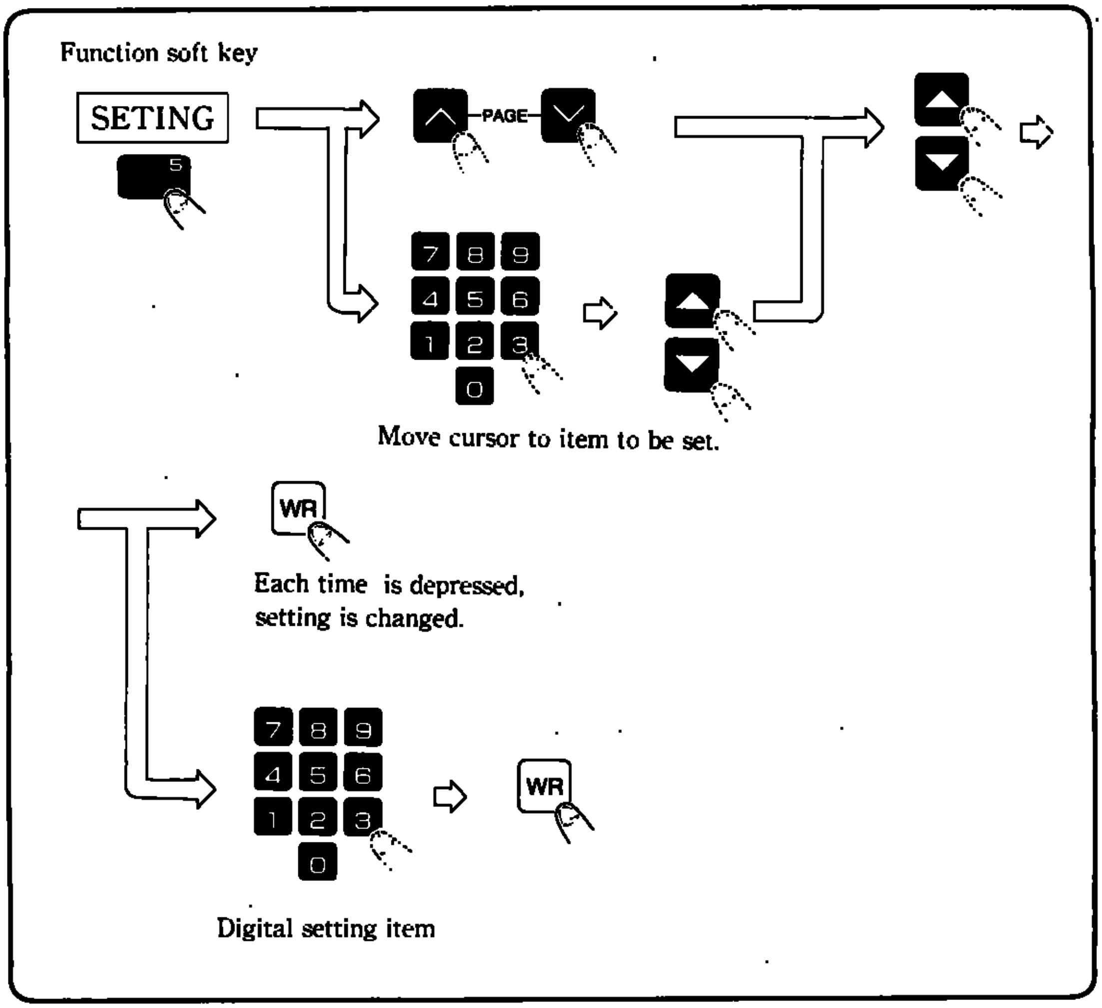
■2 Move the cursor to the Fl-digit number to be changed, enter new Fl-digit data, then depress the WR key. The Fl-digit data are rewritten.

(Example : )

NOTE For details of the Fl-digit function, see Par, 2.5.3.

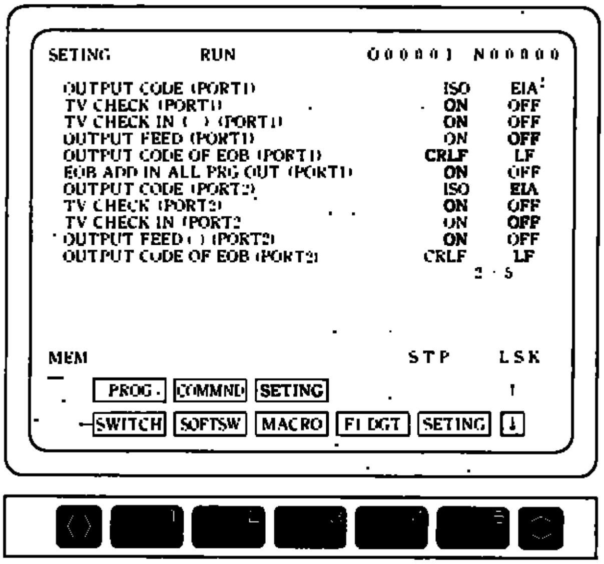
(5) Setting function

KEYPOINT of operation



■ Depress the SETING function soft key. The setting screen

shown in Fig. 3.4.74 appears.



```
SETING            RUN                      O00001  N00000

  OUTPUT CODE (PORT1)                          ISO     EIA
  TV CHECK (PORT1)                             ON      OFF
  TV CHECK IN ( ) (PORT1)                      ON      OFF
  OUTPUT FEED (PORT1)                          ON      OFF
  OUTPUT CODE OF EOB (PORT1)                   CRLF    LF
  EOB ADD IN ALL PRG OUT (PORT1)               ON      OFF
  OUTPUT CODE (PORT2)                          ISO     EIA
  TV CHECK (PORT2)                             ON      OFF
  TV CHECK IN (PORT2                           ON      OFF
  OUTPUT FEED ( ) (PORT2)                      ON      OFF
  OUTPUT CODE OF EOB (PORT2)                   CRLF    LF
                                                  2 - 5

MEM                                          STP     LSK
```

Fig. 3.4.74 Setting Screen

2 Depress page keys ∧ or ∨ and enter the page number, or that which contains the item to be set.

3 Depress the cursor keys ▲ or ▼ to move the cursor to the item to be set.

- **For non-digital setting items.**

  Depress the WR key to change the setting. The setting may be changed from use to not use, on to off, ISO to EIA, inch to millimeter, and so on.

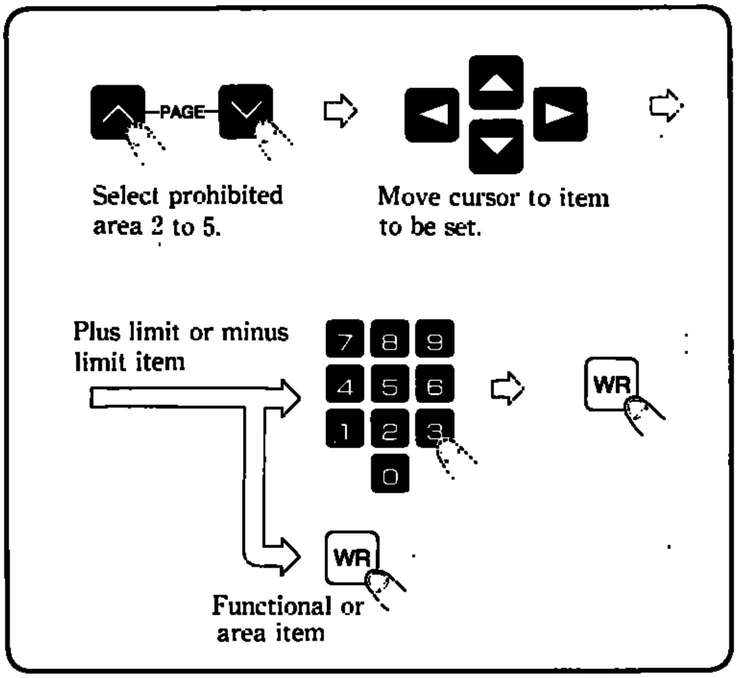
- **For digital setting items**

  Enter a value to be set, then depress the WR key.

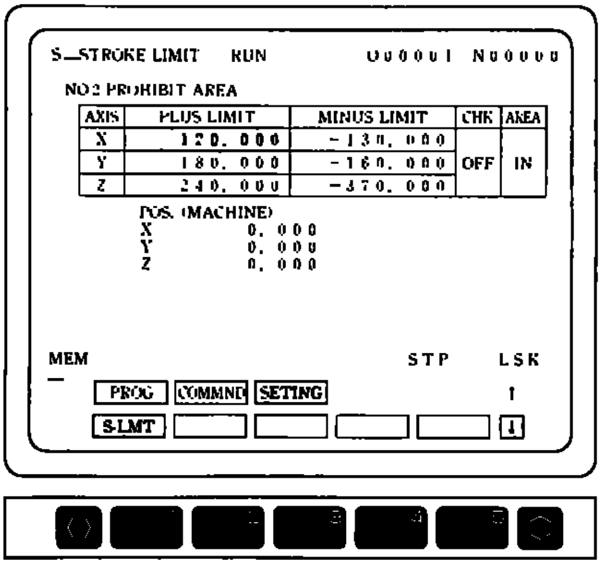
NOTE All items set on this screen correspond to the setting parameters that can be displayed by the setting function of the maintenance process parameter job.

(6) · Stored stroke limit function※

KEYPOINT of operation



■ Depress the S-LMT function soft key to display the stored stroke limit screen.



S_STROKE LIMIT RUN O00001 N00000

NO2 PROHIBIT AREA

AXIS	PLUS LIMIT	MINUS LIMIT	CHK	AREA
X	120. 000	-130. 000	OFF	IN
Y	180. 000	-160. 000		
Z	240. 000	-370. 000		

POS. (MACHINE)
X 0. 000
Y 0. 000
Z 0. 000

MEM STP LSK

Fig. 3.4.75 Stored Stroke Limit Screen

2 Depress page keys ∧ or ∨ to display prohibited areas 2 to 5 to be set.

NOTE The number of prohibited areas that can be set depends on the option.

3 Depress cursor keys ▲, ▼, ◀, or ▶ to move the cursor to the item to be set.

4 Depress WR key to set.

- **Setting plus limit or minus limit**

  Enter a value to be set, then depress the WR key.

- **Setting function item**

  Depress the WR key to validate or invalidate the stored stroke limit function.

- **Setting area item**

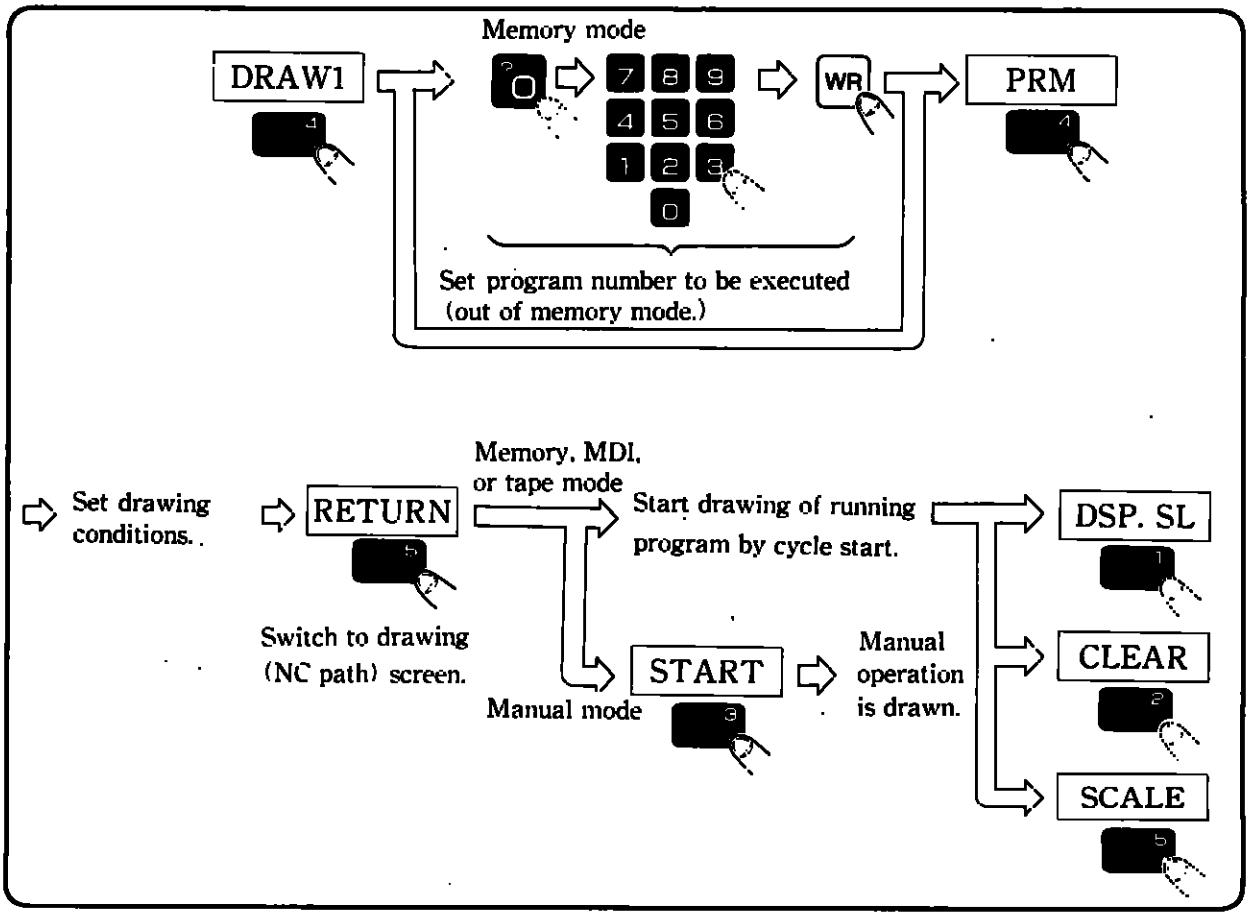
  Depress the WR key to switch whether the inside or the outside of the set area is to be prohibited.

NOTE For the stored stroke limit check function, see Par. 2.16.

3.4.7.4 NC path drawing job ※

On the NC path drawing screen, the trajectory of the NC tool is monitored and drawn on the screen. Conditions of drawing, such as the range and the angle, can be set up on the drawing condition screen.

KEYPOINT operation



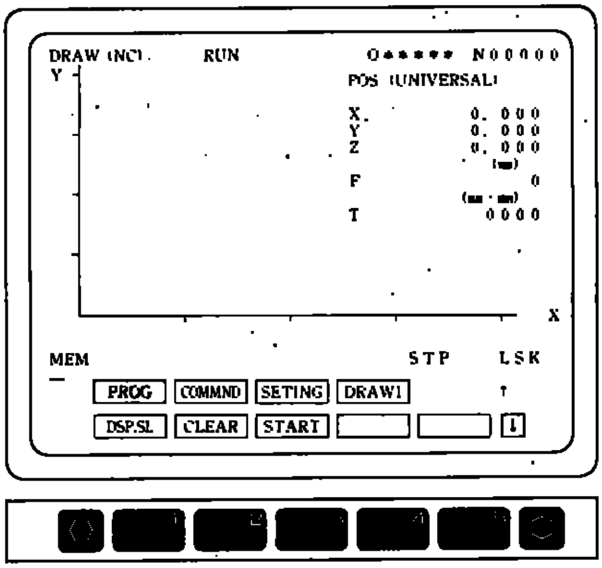


Fig. 3.4.76 NC Drawing Screen

(1) Setting program number

The program number to be executed (drawn) can be called. Depress

O, enter the program number; then depress the WR key.

(2) Setting drawing conditions

Set drawing conditions including drawing plane, view angle, drawing range, and drawing time coordinates. See explanation about the drawing condition function.

(3) Drawing

(a) When memory, MDI, or tape mode is selected

Depress the CYCLE START button. The START function soft key is highlighted in reverse, the program is executed, and the NC motion is drawn on the screen. Cutting feed is indicated by a continuous line. Rapid traverse is indicated by a dotted line.

(b) When manual mode is selected

Depress the START function soft key. Manual motion is drawn.

NOTE Motion path is drawn only in the range specified by drawing range of drawing conditions. Nothing is drawn on the screen while the NC tool is moving out of the specified range.

(4) Temporary stop of drawing

If the START function soft key is depressed during drawing (while the START function soft key is displayed in reverse video,) the monitor halts although NC keeps on moving. In this state, drawing is stopped. To restart drawing, depress the START function soft key again.

(5) Terminating drawing

When program operation is terminated by execution of M02 or M30 by depressing the key, a message is displayed "Drawing completed", and drawing is stopped.

(6) Display selection function

Depress the DSP.SL function soft key. Operation information such as positions and the program are displayed. To stop display of the information, depress the DSP.SL function soft key again.

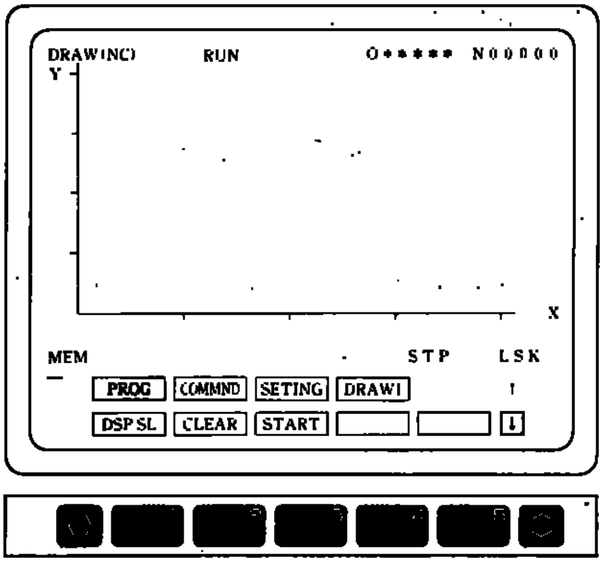


Fig. 3.4.77 NC Drawing Screen

(7) Clear function

Depress the CLEAR function soft key to clear the drawn diagram

that has been displayed.

(8) Start function

Depress the START function soft key to start or stop drawing.

When the START function soft key is displayed in reverse video, motion of the NC is being drawn.

When the START function soft key is displayed normally, motion of the NC is not being drawn.

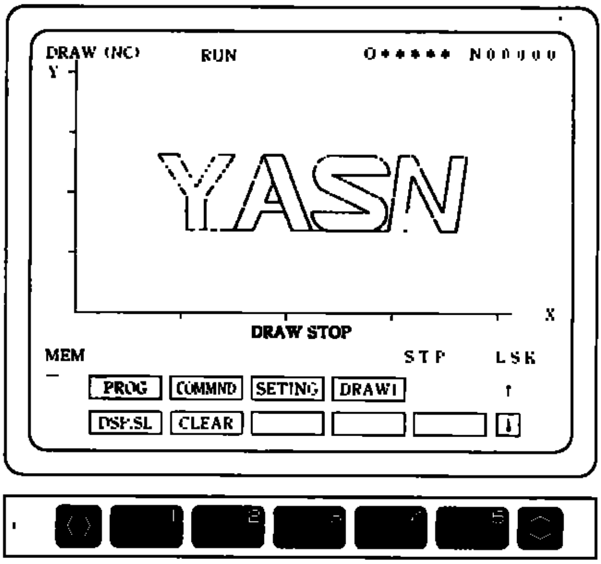
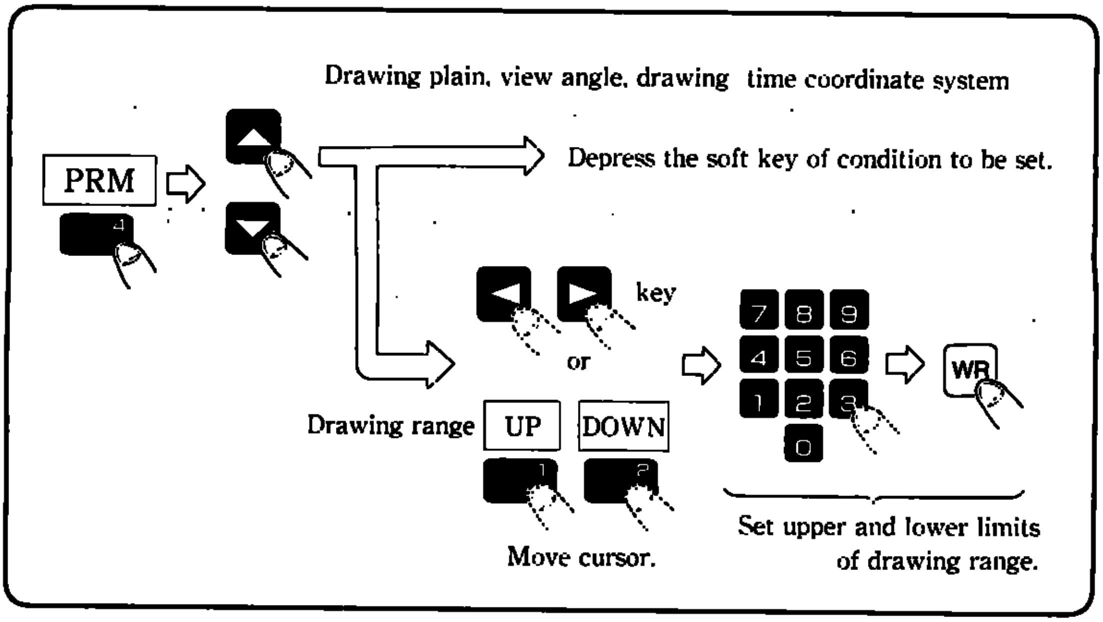


Fig. 3.4.78 NC Drawing Screen

(9) Drawing condition function

KEYPOINT of operation


Drawing plain, view angle, drawing time coordinate system

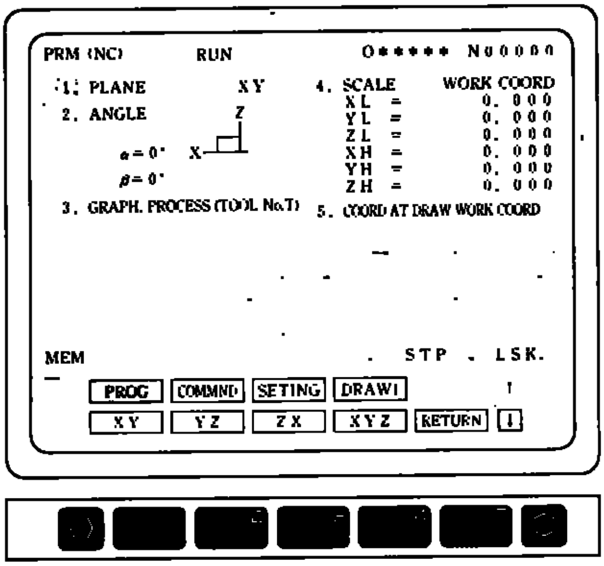
Depress the soft key of condition to be set.

Set upper and lower limits of drawing range.

Move cursor.

1. Depress the [PRM] function soft key. (Reverse function)

   The drawing condition setup screen appears.



```
PRM (NC)          RUN                  O****** N00000
 1. PLANE           XY      4. SCALE       WORK COORD
 2. ANGLE           Z          XL   =           0. 000
                               YL   =           0. 000
                               ZL   =           0. 000
        α= 0°   X              XH   =           0. 000
        β= 0°                  YH   =           0. 000
                               ZH   =           0. 000
 3. GRAPH. PROCESS (TOOL No.T)  5. COORD AT DRAW WORK COORD

MEM                                       STP   LSK.
    [PROG] [COMMND] [SETING] [DRAWI]
    [XY]   [YZ]     [ZX]     [XYZ]    [RETURN] [↓]
```

Fig. 3.4.79 Drawing Condition Display Screen

2. Use cursor key ▲ or ▼ to move the cursor to "1. PLANE."

   Select one of [XY], [YZ], [ZX], or [XYZ] function soft keys.

   (Example : Depress the [XY] function soft key.)

3. Use cursor key ▲ or ▼ to move the cursor to "2. ANGLE."

   Use function soft key [LEFT], [RIGHT], [UP], or [DOWN] to set up the view angle.
   α is an angle about the Z-axis in the direction from X-axis to Y-axis.
   β is the angle between the XY plain and the view point.

NOTE The view angle is valid only when the XYZ plain is selected.

4 Use cursor key ▲ or ▼ to move the cursor to "3. GRAPH. PROCESS. "Use the UP, DOWN, LEFT or RIGHT function soft key or cursor key ◀ or ▶ to move the cursor to prevent selection of unnecessary processes, SEL/DEL key is selected by WR key. Highlighted No. T is the selected number.

NOTE

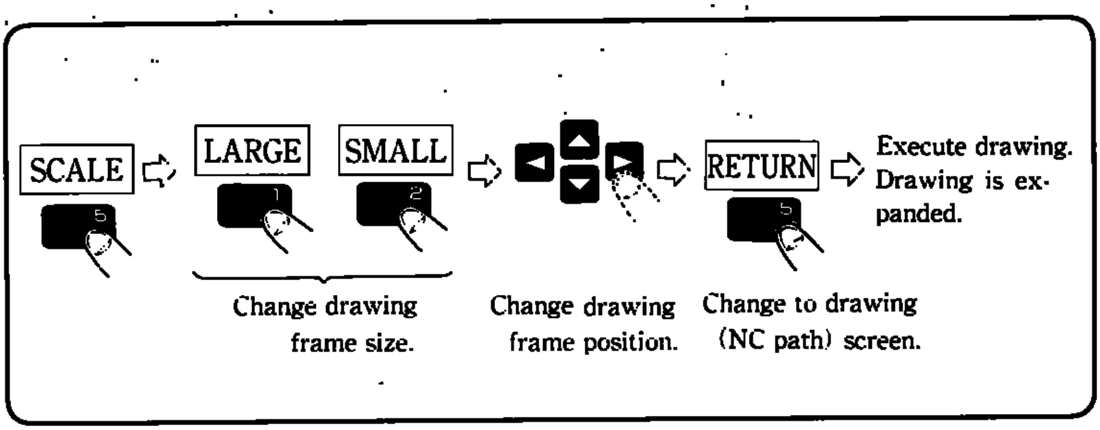
1. The following conditions must be satisfied : XL<XH; YL<YH; and ZL<ZH. If any of the conditions is not met, the drawing (NC path) screen cannot be returned.
2. If the drawing time coordinate system is the work coordinate system, set the range limits with the work coordinates. If the drawing time coordinate system is the mechanical coordinate system, set the range limits with the mechanical coordinates.

5 Use cursor key ▲ or ▼ to move the cursor to "5. COORD AT DRAW WORK COORD." Depress the UNIV or ABS function soft key to select the coordinate system.

6 Depress the RETURN function soft key to return to the drawing (NC path) screen.

(10) Drawing range function

KEYPOINT of operation



1 Depress the SCALE function soft key. (Reverse function)

The drawing range screen appears. Expanded drawing can be made for the next drawing by changing the range of the preceding drawing on this screen.
Once the drawing frame is changed on this screen, the setup frame is used for the next and following drawings.

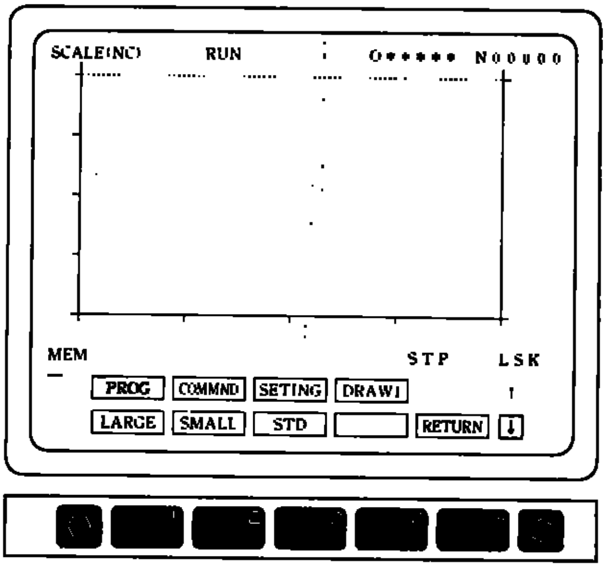


Fig. 3.4.80 NC Drawing Range Display Screen

2 Depress the LARGE or SMALL function soft key to change the drawing frame size for the next time.

3 Depress cursor key ▲, ▼, ◀, or ▶, to move the center of the drawing frame for the next drawing.

4 Depress the RETURN key.

The drawing (NC path) screen appears. The set drawing frame is used for the maximum on the screen for the next and following drawings.

5 To restore the original size after expanded drawing, depress the STD function soft key.

The drawing (NC path) screen appears.

After the STD key is depressed, the original drawing frame size is used.

NOTE The drawing range values remains unchanged after the drawing frame size is changed.

(11) Automatic scaling function (Reverse function)

1 Depress the AUTO function key.
When this key is depressed, the scale is automatically calculated so that all paths of the currently selected program will be displayed.
"AUTO SCALING" is displayed during automatic scating. When it is completed, a message "SCALING COMPLETE" is displayed.

3.4.8 Maintenance Process MAINT

In the maintenance process, necessary data for maintenance can be set up and displayed, or transferred to or from external equipment.

The parameter job sets up and displays parameter data, setting data, keep memory data, and pitch error offset data contained in the controller internal memory. These data are used to turn on and off specific functions and to determine operation conditions for functions.

The I/O verification job monitors the on/off state of I/O signals among the controller, PLC, and the milling machine.

The I/O verification job transfers parameter data from the parameter job to external equipment, and vice versa. The job also verifies data by comparing input data with internal data.

The internal information job can display a variety of NC unit internal information.

1 Depress the MAINT key. Any of the following jobs is displayed.

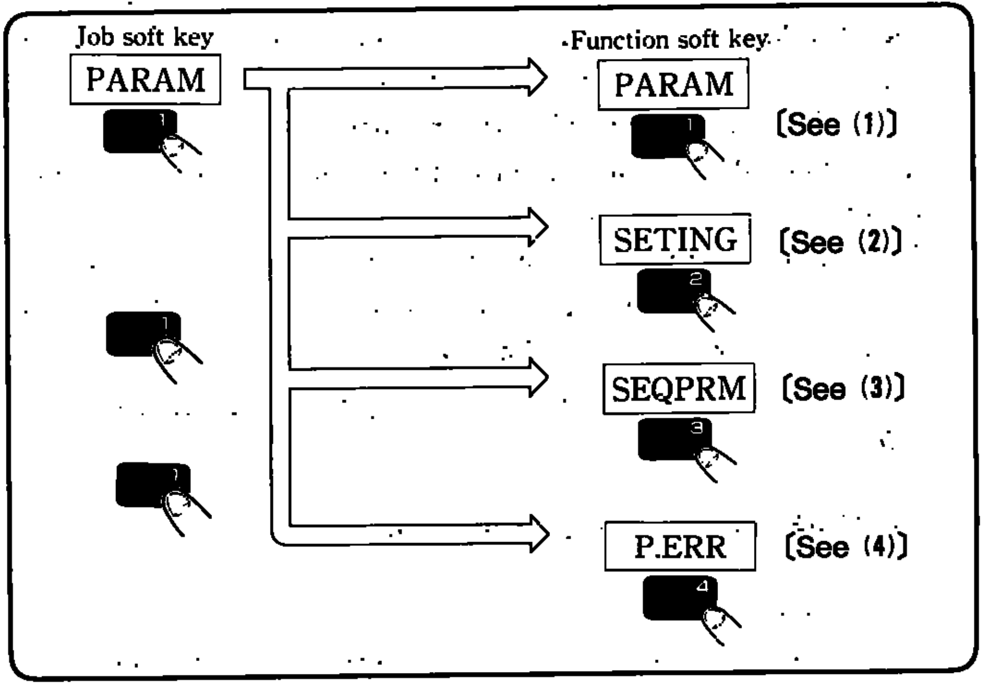
- Parameter job ———— See Par. 3.4.8.1 (page 516.)
- I/O monitor job ———— See Par. 3.4.8.2 (page 523.)
- I/O verification job ——— See Par. 3.4.8.3 (page 529.)
- Internal information job —— See Par. 3.4.8.4 (page 535.)

2 To display a necessary job, depress the corresponding job soft key (PARAM 1, DIAGN 2, IN/OUT 3 or DATA 4) Otherwise, depress the MAINT key to switch the displayed job.

Operations of the above jobs are explained below.

3.4.8.1 Parameter job (job soft key PARAM)

KEYPOINT



1. Depress the PARAM job soft key.

2. The PARAM, SETING, SEQPRM, and P.ERR function soft keys are displayed.

3. Turn on the "PARAMETER CHANG" internal switch by the SETING function soft key of the RUN process.
This makes it possible to change parameters, keep memory, and pitch error. After setting these items, turn off the internal switch again.

(1) Parameter function

Parameter data kept in controller internal memory can be displayed and set up.

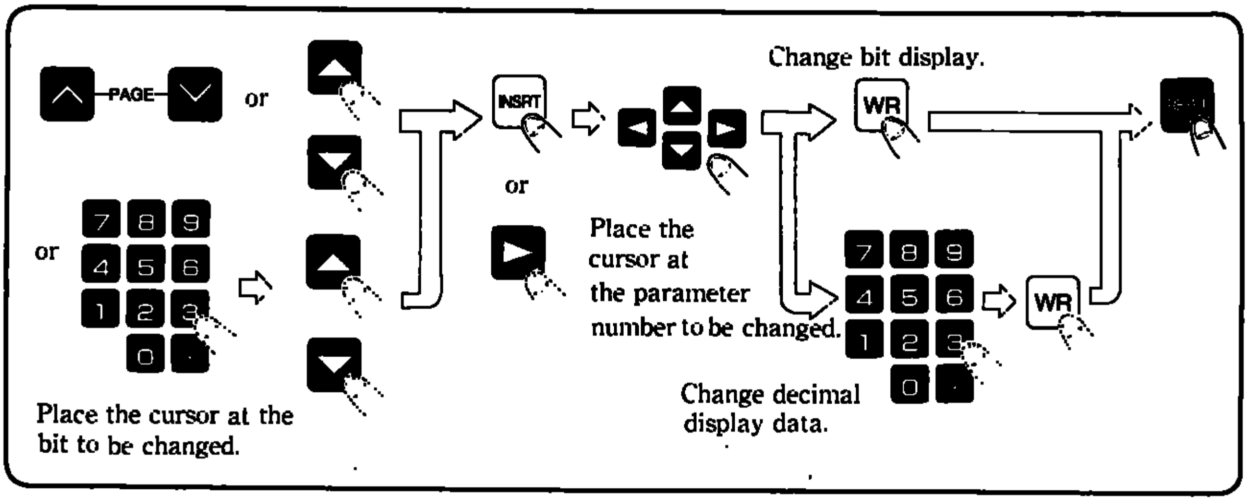
1 Depress the PARAM function soft key.

2 Parameters are classified into the bit type or the byte, word, and double word type by the parameter number.

NOTE For details about the parameter number, refer to YASNAC i80M Appendix Manual (TOE-C843-11.31.)

(a) Bit type

KEYPOINT



1 Place the cursor at the parameter number to be changed, by using the page keys ∧ PAGE ∨ or cursor key ▲ or ▼, or by entering the parameter number directly with digit keys and depressing the cursor key ▲ or ▼.

2 Depress the INSRT key or the cursor key ▶. If the INSRT key is depressed, the cursor moves to the decimal number display. If the cursor key ▶ is depressed, the cursor moves to the bits.

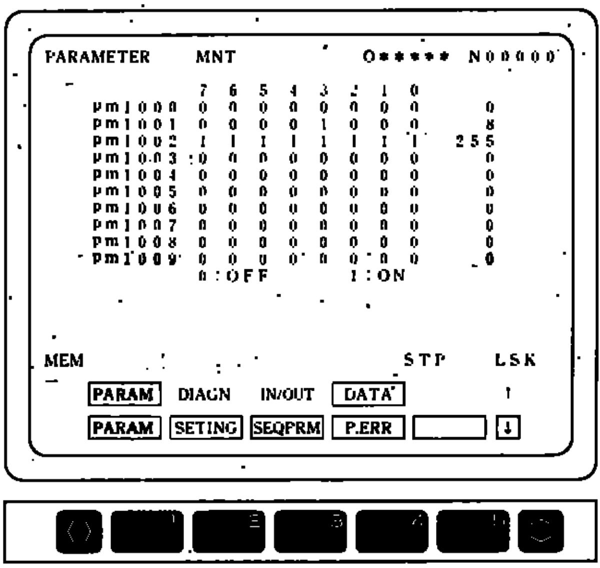
3 Place the cursor at the bit to be changed, then depress the WR key.

Each time the WR key is depressed, the bit is turned on and off alternately.

Data can be written by digit keys only when the cursor is placed at the decimal number display at the right end.

(Example) Decimal number write for bit display

Input data	Bit 7 6 5 4 3 2 1 0	Decimal display
0 WR	0 0 0 0 0 0 0 0	0
8 WR	0 0 0 0 1 0 0 0	8
2 5 5 WR	1 1 1 1 1 1 1 1	2 5 5

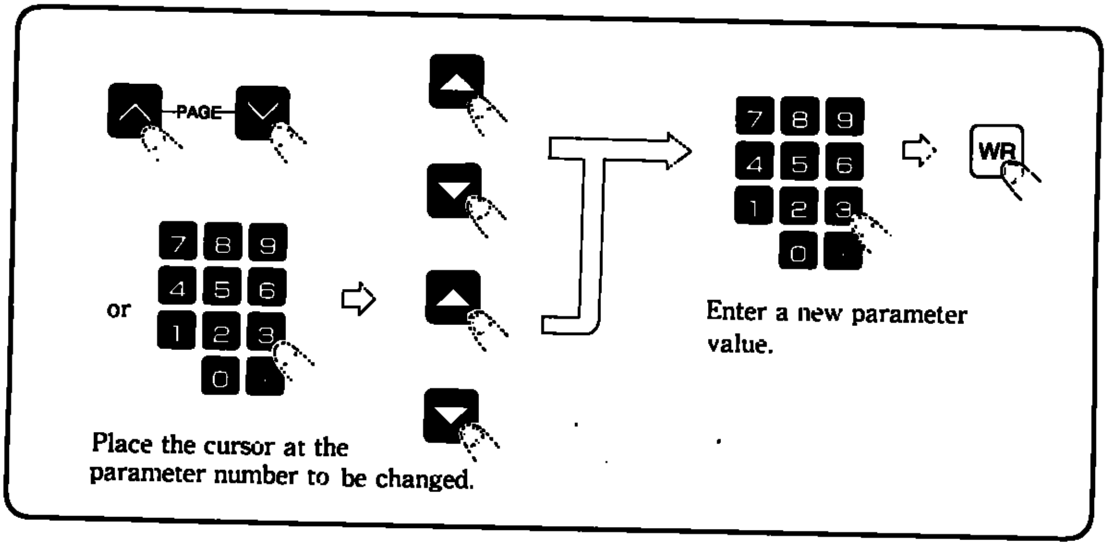


```
PARAMETER      MNT               O*****  N00000

                 7  6  5  4  3  2  1  0
      pm1000     0  0  0  0  0  0  0  0       0
      pm1001     0  0  0  0  1  0  0  0       8
      pm1002     1  1  1  1  1  1  1  1     255
      pm1003     0  0  0  0  0  0  0  0       0
      pm1004     0  0  0  0  0  0  0  0       0
      pm1005     0  0  0  0  0  0  0  0       0
      pm1006     0  0  0  0  0  0  0  0       0
      pm1007     0  0  0  0  0  0  0  0       0
      pm1008     0  0  0  0  0  0  0  0       0
      pm1009     0  0  0  0  0  0  0  0       0
                 0:OFF           1:ON

MEM                                  STP    LSK
```

Fig. 3.4.81 Parameter Input Screen (bit type)

(b) Byte, word, and double word type

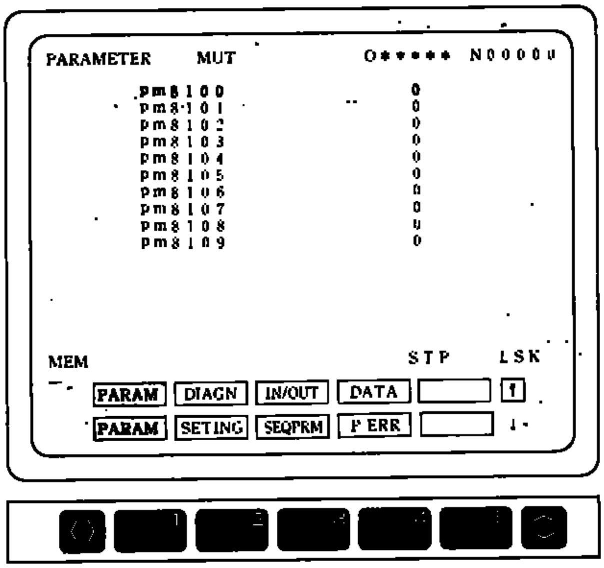
KEYPOINT



Place the cursor at the parameter number to be changed.

Enter a new parameter value.

1. Place the cursor at the parameter number to be changed, by using the page keys ∧ PAGE ∨ or cursor key ▲ or ▼, or by entering the parameter number directly with digit keys and depressing the cursor key ▲ or ▼. To search for a parameter number, the parameter value need not be specified.
2. Enter a new parameter value.
3. Depress the WR key. The parameter value is entered.



```
PARAMETER      MUT                    O******  N00000

              pm8100                     0
              pm8101                     0
              pm8102                     0
              pm8103                     0
              pm8104                     0
              pm8105                     0
              pm8106                     0
              pm8107                     0
              pm8108                     0
              pm8109                     0

MEM                                   STP     LSK
```

Fig. 3.4.82 Parameter Input Screen (byte, word, and double-word type)

4 After changing a parameter value, depress the key. The new parameter takes effect after the panel has been reset.

NOTE

1. If a parameter is written that takes effect only after power is turned off, alarm "ALM 0050 : POWER OFF PRM SET" is issued. To cancel the alarm, turn OFF power once, then turn ON again.
2. If a parameter is written that takes effect only after the part program is initialized, alarm "ALM 0051 : PROGRAM GENERATION PRM SET" is issued. If this alarm occurs, program generation is necessary. Contact your YASKAWA representative.

   Even after this alarm occurs, the system becomes available if you turn power OFF and ON, although the parameter change is canceled.

(2) Setting function

Setting data kept in controller internal memory can be displayed and set up.

1. Depress the SETING function soft key.

2. Place the cursor at the setting number to be changed.
3. Enter a new parameter value.

NOTE
1. The operation is similar to that of the parameter function, except that setting data can be changed without turning on the "PARAMETER CHANG" internal switch by the SETING function soft key of the RUN process.
2. The new setting data takes effect immediately after change.
3. Part of setting data can be set up on the SETING function screen in the RUN process.

(3) Keep memory function

Keep memory data kept in controller internal memory can be displayed and set up. The keep memory data are used by PLC.

1. Depress the SEQPRM function soft key.
2. Place the cursor at the keep memory number to be changed.
3. Enter a new parameter value.

NOTE
1. The operation is the same as the parameter function. See (1), "Parameter function".
2. On the keep memory bit type screen, the number on the right end is represented in hexadecimals.

(4) Pitch error function ※

Setting data kept in controller internal memory can be displayed and set up.

Pitch error offset data kept in controller internal memory can be displayed and set up.

1 Depress the P.ERR function soft key.

2 Place the cursor at the pitch error number to be changed.

3 Enter a new parameter value.

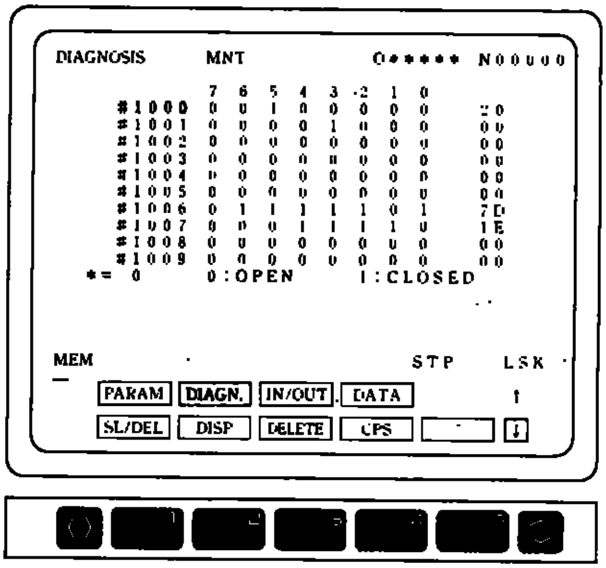
NOTE
1. The operation is the same as of the parameter function. See (1), "Parameter function".
2. The pitch error function soft key is displayed only when the pitch error offset option is available.

3.4.8.2 I/O monitor job (job soft key DIAGN.)

Depress the DIAGN. job soft key to display the on/off status of the NC

equipment, PLC, and machine I/O signals.

NOTE The status of I/O signals can be displayed any time, including the duration of auto operation.



```
DIAGNOSIS       MNT                      O******  N00000

                 7  6  5  4  3 ·2  1  0
       #1000     0  0  1  0  0  0  0  0      20
       #1001     0  0  0  0  1  0  0  0      00
       #1002     0  0  0  0  0  0  0  0      00
       #1003     0  0  0  0  0  0  0  0      00
       #1004     0  0  0  0  0  0  0  0      00
       #1005     0  0  0  0  0  0  0  0      00
       #1006     0  1  1  1  1  1  0  1      7D
       #1007     0  0  0  1  1  1  1  0      1E
       #1008     0  0  0  0  0  0  0  0      00
       #1009     0  0  0  0  0  0  0  0      00
   *=  0         0:OPEN           1:CLOSED


MEM                                      STP      LSK
```

Fig. 3.4.83 I/O Signal display screen

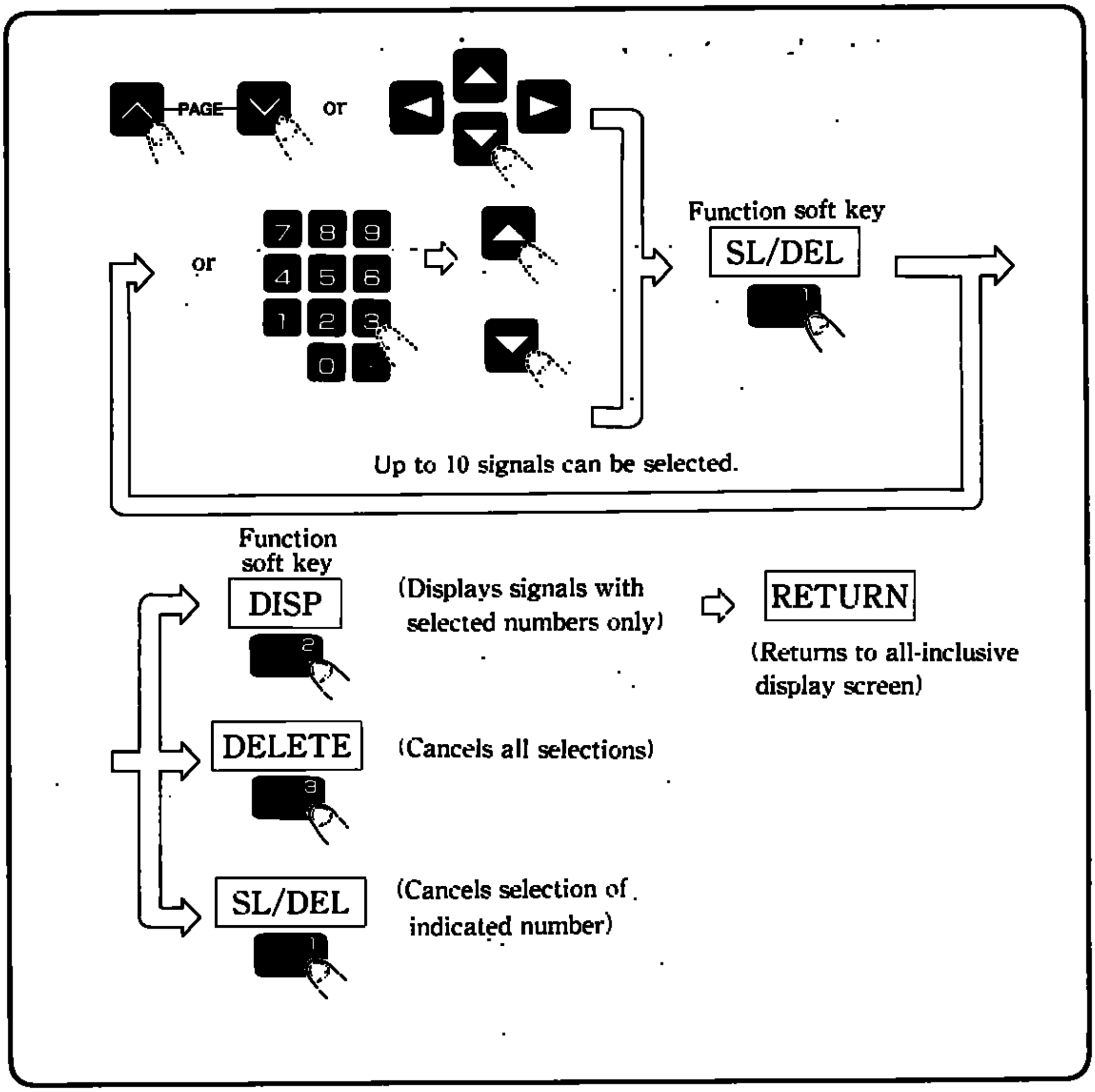
NOTE

1. For details of the display, see Section 4, "I/O SIGNAL DIAGNOSIS". Also refer to separate appendix (TOE-C843-11.31) "Standard I/O diagnosis Number Table" of YASNAC i80M Operation Manual.
2. The value on the far right in the I/O signal status display is represented in hexadecimals, for maintenance ease.

(1) I/O signal selection and display

I/O signals can be picked up and displayed.

KEYPOINT of operation



■ Use page keys ∧ or ∨, or the cursor keys ▲, ▼, ◀, or ▶, to move the cursor to the I/O signal number to be changed. Otherwise, enter I/O signal number using the digit keys, and depress the cursor key ▲ or ▼ to move the cursor to the I/O signal number to be changed.

2 Depress the SL/DEL function soft key. An asterisk appears to the left of the I/O signal number, indicating that the selected number is stored in memory. The number of I/O signal numbers is displayed on the lower part of the screen.

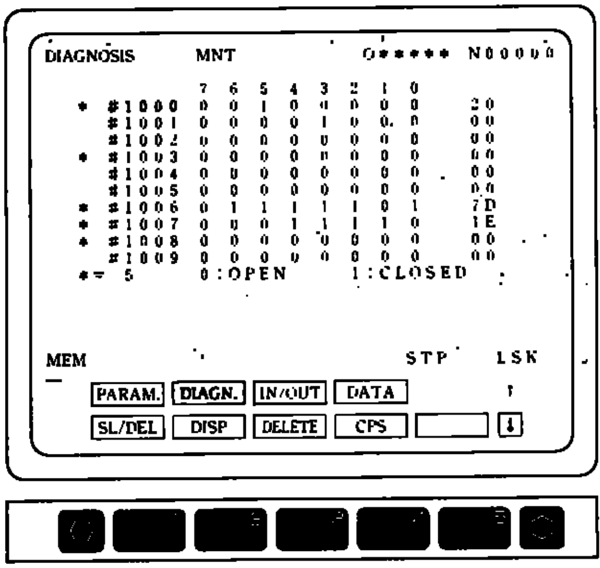
3 Repeat 1 and 2 to select up to 10 I/O signal numbers to be displayed.

4 ● Depress the DISP function soft key to display the selected numbers only.

● Depress the RETURN function soft key on the select display screen to return to the all-inclusive display screen.

● Depress the DELETE function soft key to cancel all selections that have been made.

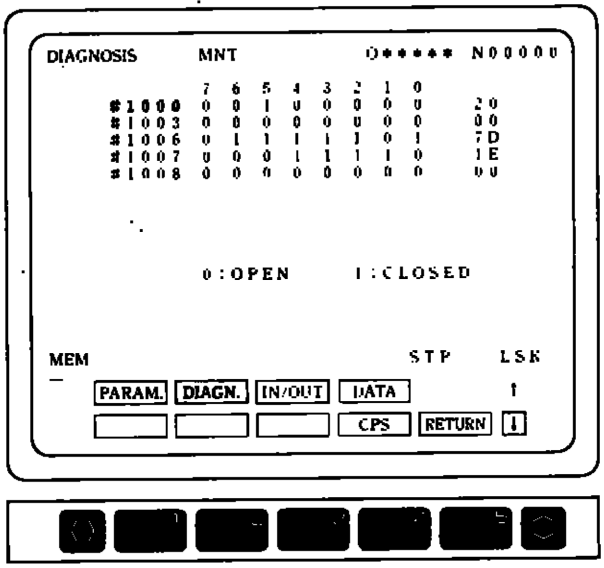
● Move the cursor to an already selected I/O signal number and depress the SL/DEL function soft key to cancel selection of that number.



```
DIAGNOSIS      MNT                 O*****  N00000

                 7  6  5  4  3  2  1  0
 *  #1000        0  0  1  0  0  0  0  0       20
    #1001        0  0  0  0  1  0  0  0       00
    #1002        0  0  0  0  0  0  0  0       00
 *  #1003        0  0  0  0  0  0  0  0       00
    #1004        0  0  0  0  0  0  0  0       00
    #1005        0  0  0  0  0  0  0  0       00
 *  #1006        0  1  1  1  1  1  0  1       7D
 *  #1007        0  0  0  1  1  1  1  0       1E
 *  #1008        0  0  0  0  0  0  0  0       00
    #1009        0  0  0  0  0  0  0  0       00
 *=  5           0:OPEN          1:CLOSED

MEM                                 STP    LSK
—   [PARAM.] [DIAGN.] [IN/OUT] [DATA]         ↑
    [SL/DEL] [DISP]   [DELETE] [CPS]  [      ] [↓]
```

Fig. 3.4.84 I/O Signal Selection Screen

Depress the DISP function soft key to change to the following screen:



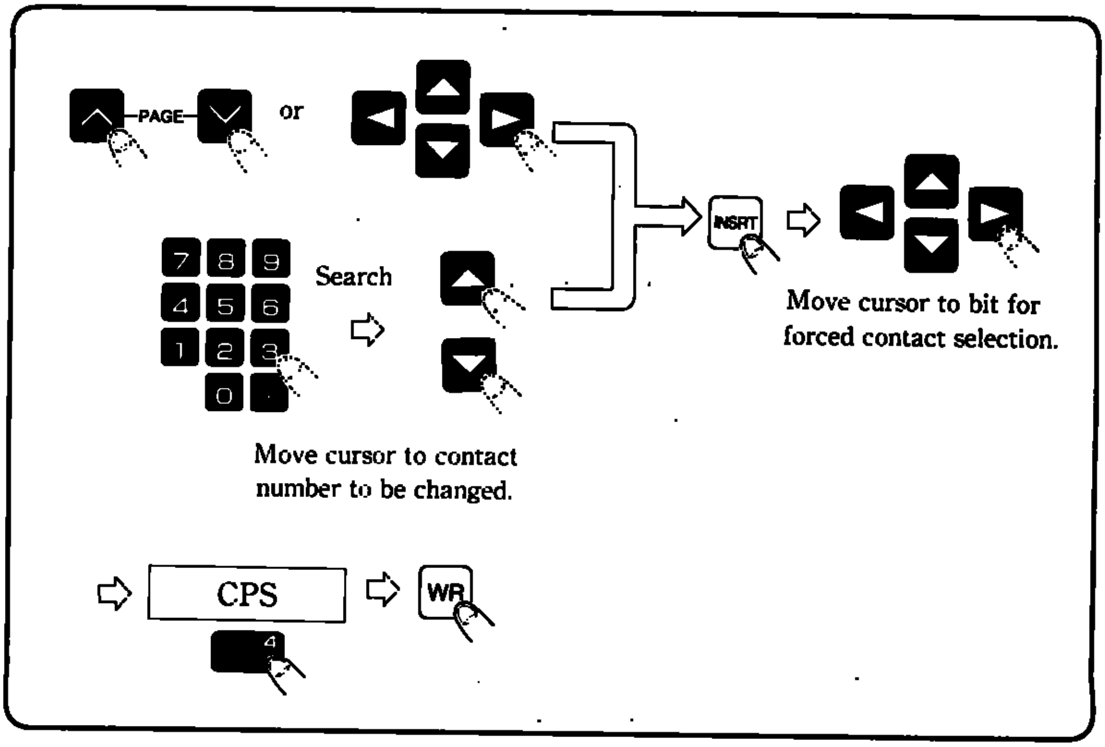
```
DIAGNOSIS      MNT                 O*****  N00000

                 7  6  5  4  3  2  1  0
    #1000        0  0  1  0  0  0  0  0       20
    #1003        0  0  0  0  0  0  0  0       00
    #1006        0  1  1  1  1  1  0  1       7D
    #1007        0  0  0  1  1  1  1  0       1E
    #1008        0  0  0  0  0  0  0  0       00


                 0:OPEN          1:CLOSED

MEM                                 STP    LSK
—   [PARAM.] [DIAGN.] [IN/OUT] [DATA]         ↑
    [      ] [      ] [      ] [CPS]  [RETURN] [↓]
```

Fig. 3.4.85 I/O Signal Selection Display Screen

(2) Operation of forced contact selection

The forced contact selection function is provided to check mechanical sequences. This function can forcibly change the state of contacts without changing the mechanical sequences. To enable this function, set "1" to pm109 of SETING function of PARAM job in the MAINT process.

KEYPOINT of operation



1. Use page key ∧ or ∨, or the cursor key ▲, ▼, ◀, or ▶, to move the cursor to the I/O signal number to be changed. Otherwise, enter I/O signal number using the digit keys, and depress the cursor key ▲ or ▼ to move the cursor to the I/O signal number to be changed.

2. Depress the INSRT key. The cursor moves to the selected bit.

3. Move the cursor to the bit for forced contact selection, then depress the CPS 4 function soft key. The bit is highlighted. If the function soft key is depressed again, the highlight disappears.

4. Depress the WR key. Each time the WR key is depressed, the bit is turned on or off.

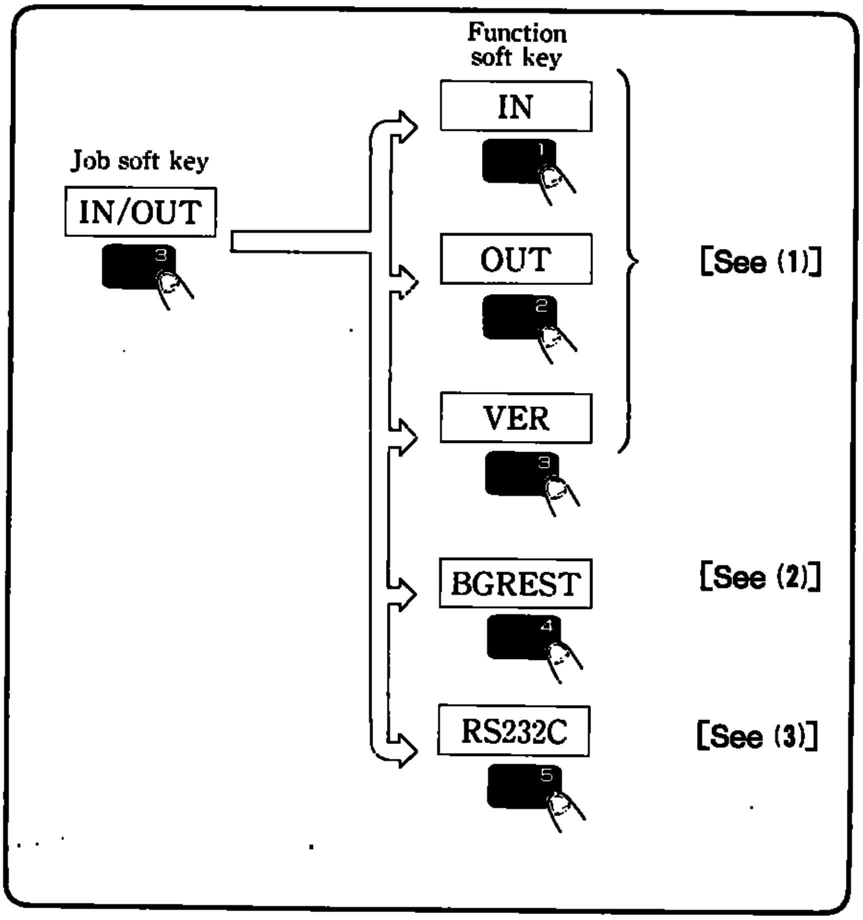
5. Move the cursor to an I/O signal number indicated by an asterisk and depress the SL/DEL 1 function soft key. The forced contact selection at the number is canceled.

6. Depress the DELETE 3 function soft key to cancel all forced contact selections.

3.4.8.3 I/O verification job (job soft key IN/OUT)

The I/O verification job is to input and output data between the controller and external equipment.

KEYPOINT



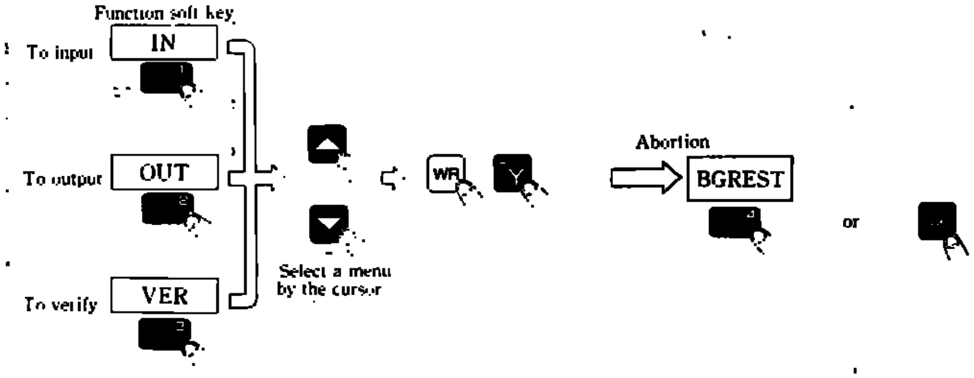
By changing the parameter, tape I/O verification data can be displayed.

Pm0007 D5 = 0 : Does not display data at tape I/O verification.
1 : Displays data at tape I/O verification.

When the data are displayed, the I/O verification time is extended.

(1) Input, output, and verification function

KEYPOINT



The following menus are available :

- All parameter I/O verification
  (Setting, parameter, sequencer parameter and keep memory, pitch error, and offset parameter)
- Setting I/O verification
- Parameter I/O verification (Sequencer parameters Pm7000 to Pm7999 are disabled)
- Sequencer parameters and keep memory I/O verification

NOTE Keep memory is displayed in hexadecimal but output is in decimal.

- Pitch error offset parameter I/O verification (option)
- Offset all I/O verification
  (Tool offset, work coordinate system shift)
- Tool life control data I/O verification (option)
- Macro common variable I/O verification (option)

I/O verification of the above data can be made regardless of the controller operation mode, except that automatic operation cannot be performed during I/O verification.

1 • To input, depress the IN function soft key.

• To output, depress the OUT function soft key.

• To verify, depress the VER function soft key.

The parameter input (or output/verification) screen shown in Fig. 3.4.86.

2 Depress cursor key ▲ or ▼ to move the cursor to the menu to input, output, or verify. Depress the WR key.

"INPUT (OUTPUT/VERIFY) OK?" is displayed.

Depress Y . I/O verification is started and the screen displays data being input, output, or verified.

Zero-suppress Function

Can specify whether 0-data are output at parameter output according to parameter pm0007 D6.

pm0007 D6=0 : Does not output parameter 0-data (zero-suppress).
1 : Outputs parameter 0-data.

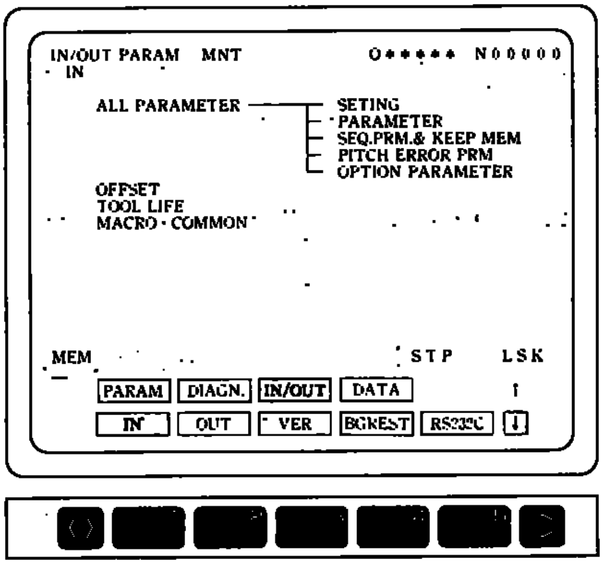
To input or verify the parameter which is output using this function, make sure of the following points.

- Input

  Does not execute writing-in 0-data since the data (other than 0) which are output using this function.

- Verification

  Does not verify any parts that are 0-data at output since only the data (other than 0) which are output use this function.



IN/OUT PARAM MNT O****** N00000

IN

ALL PARAMETER
- SETING
- PARAMETER
- SEQ.PRM.& KEEP MEM
- PITCH ERROR PRM
- OPTION PARAMETER

OFFSET
TOOL LIFE
MACRO·COMMON

MEM STP LSK

Fig. 3.4.86 Parameter Input (Output/verification) Screen

3 The following operation is needed during I/O verification.

⟨Aborting operation⟩

To abort I/O verification, depress the BGREST function soft key or 

the  key.

For the contents of BG reset, refer to (2) BG reset function.

NOTE Parameter, sequencer parameter & keep memory, and pitch error offset parameter can be selected only when 1 is set for #109 of SETING function soft key of the PARAM job soft key in the RUN process.

(2) BG reset function

The BGREST switch belongs to the background, different from the RESET switch on the NC operator's panel.

In BG reset operation, the following contents are processed:

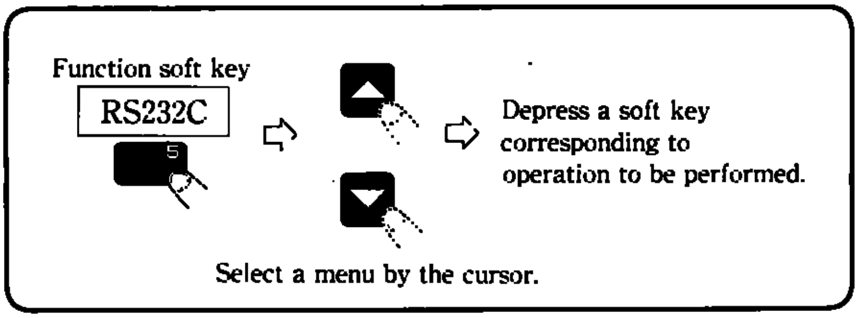
- I/O verification abortion
- Label skip function ON
- Background alarm (ALM No. 9000s) release

Depressing the BGREST function soft key during operation does not affect the operation.

Even by depressing the BGREST function soft key, the status "RST" during reset on the screen display is not displayed.

(3) I/O equipment setting function

KEYPOINT

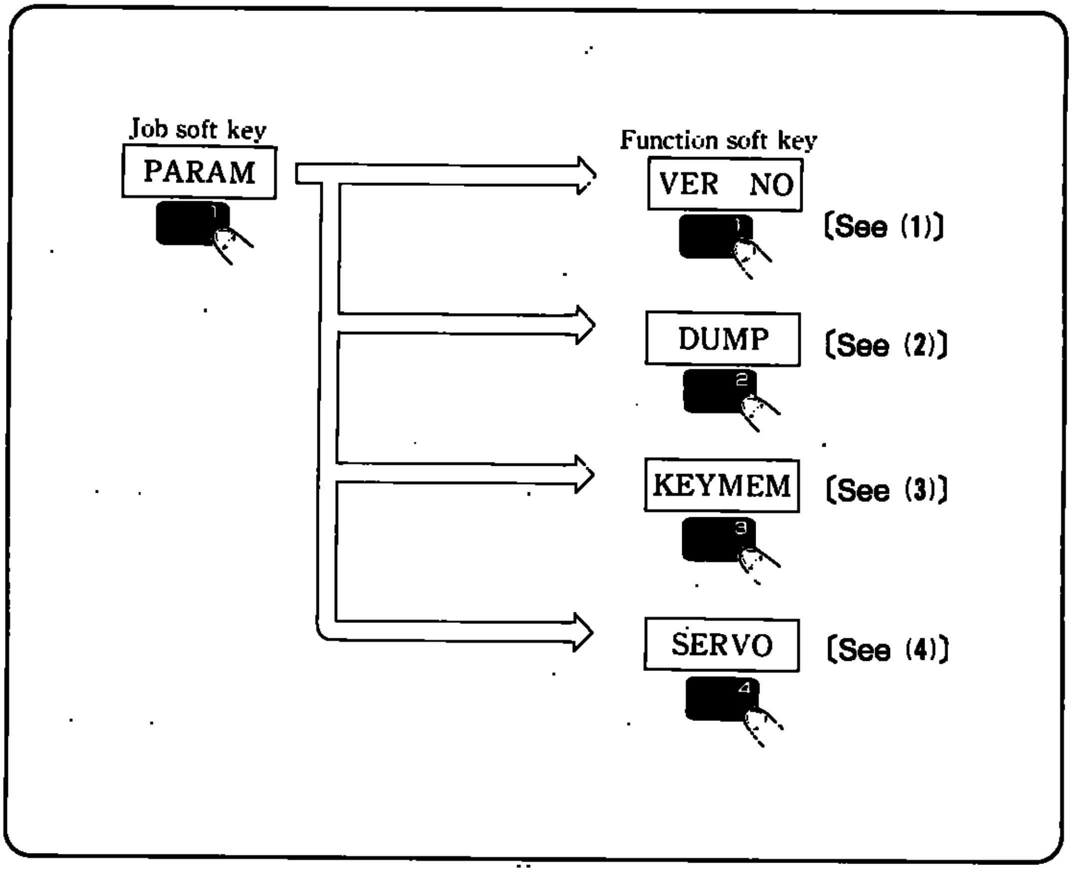


Depress the RS232C function soft key to set up I/O equipment. The setting operation is the same as with the RS232C function soft key of the IN/OUT job soft key in the PROG process. Data take the same effect regardless of which screen they were set up on.

job soft key

3.4.8.4 Internal information job (DATA)

KEYPOINT



1. Depress the PARAM job soft key.

2. The VER NO, DUMP, KEYMEM, SERVO function soft keys are displayed.

(1) Version No. list function

This screen is used exclusively for NC unit maintenance.

Depressing the VER NO function soft key displays the version No.

list screen. In this screen, the following contents are displayed.

(a) System No.

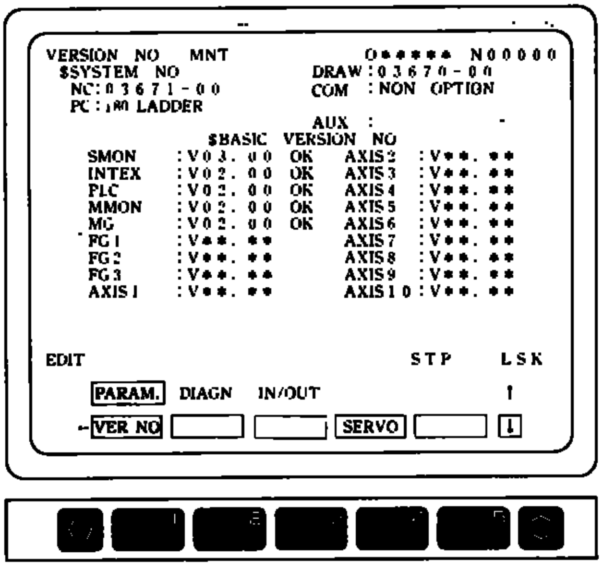
An item to be displayed on screen $ SYSTEM NO. There are two types of system numbers : those for total NC unit and for PC ladder.

(b) Option version No.

An item to be displayed on screen $ OPTION VERSION NO. The option board side software version when the option board is added is displayed.

(c) Basic version No.

An item to be displayed on screen $ BASIC VERSION NO. Each software version in the basic board is displayed. Additionally, whether each board software version consistency is correct is displayed.



```
VERSION  NO    MNT                    O*****  N00000
 $SYSTEM  NO                  DRAW :03670-00
   NC:03671-00                COM  :NON  OPTION
   PC :180 LADDER
                               AUX  :
              $BASIC  VERSION  NO
     SMON      :V03. 00  OK     AXIS2   :V**. **
     INTEX     :V02. 00  OK     AXIS3   :V**. **
     PLC       :V02. 00  OK     AXIS4   :V**. **
     MMON      :V02. 00  OK     AXIS5   :V**. **
     MG        :V02. 00  OK     AXIS6   :V**. **
     FG1       :V**. **         AXIS7   :V**. **
     FG2       :V**. **         AXIS8   :V**. **
     FG3       :V**. **         AXIS9   :V**. **
     AXIS1     :V**. **         AXIS10  :V**. **


EDIT                                      STP     LSK
```

Fig. 3.4.87 Version No. List Screen

(2) Memory dump function

This screen is used exclusively for NC unit maintenance.

Depressing the DUMP function soft key displays the memory dump

screen.

This screen must be used by the NC unit service engineers only.

(3) Key memory function

Depressing the KEYMEM function soft key displays the key memory

screen.

The record of key input from the NC operator's panel is displayed on the key memory screen.

The following shows the input keys and the corresponding key codes to be displayed.

- Key code list

Alphabets, numericals, SHIFT + A to Z

The character of the input key is displayed. However, only the space is displayed as "SP".

Special keys

Soft key

< > key	MN−L
◇ key	MN−R
1 key	F1
2 key	F2
3 key	F3
4 key	F4
5 key	F5

Cursor key

▶ key	CU−R
▲ key	CU−U
◀ key	CU−L
▼ key	CU−D

Page key

⍓ key	PG−U
⍌ key	PG−D

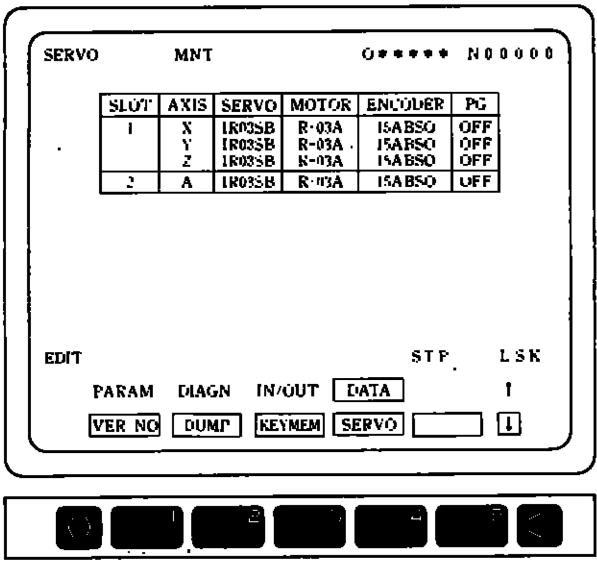
Process key, Other

PROG key	PRG
SET key	SET
RUN key	RUN
MAINT key	MNT
COMN key	CMN
AUX1 key	AUX1
AUX2 key	AUX2
PESET key	RES
CAN key	CAN
EOB key	EOB
INSRT key	INS
ALTER key	ALT
ERASE key	ERS
WR key	WR

(4) Servo configuration

Depressing the SERVO function soft key displays the servo configuration screen.

This screen displays the list of the SERVOPACK and SERVOMOTOR types mounted on the machine tools.



SERVO MNT O****** N00000

SLOT	AXIS	SERVO	MOTOR	ENCODER	PG
1	X	1R03SB	R-03A	15ABSO	OFF
	Y	1R03SB	R-03A	15ABSO	OFF
	Z	1R03SB	R-03A	15ABSO	OFF
2	A	1R03SB	R-03A	15ABSO	OFF

EDIT STP LSK

Fig. 3.4.88 Typical Servo Configuration Screen

3.4.9 Common Process (COMN)

The common process deals with screens that are used in every process, not particularly in one of the editing, setup, operation, or maintenance processes explained in the previous part of this manual.
The current value job displays and sets up positions.
The alarm job notifies alarms if one occurs.
The time job displays and sets up date and time with the calendar function.
Work times can be displayed and reset during operation.

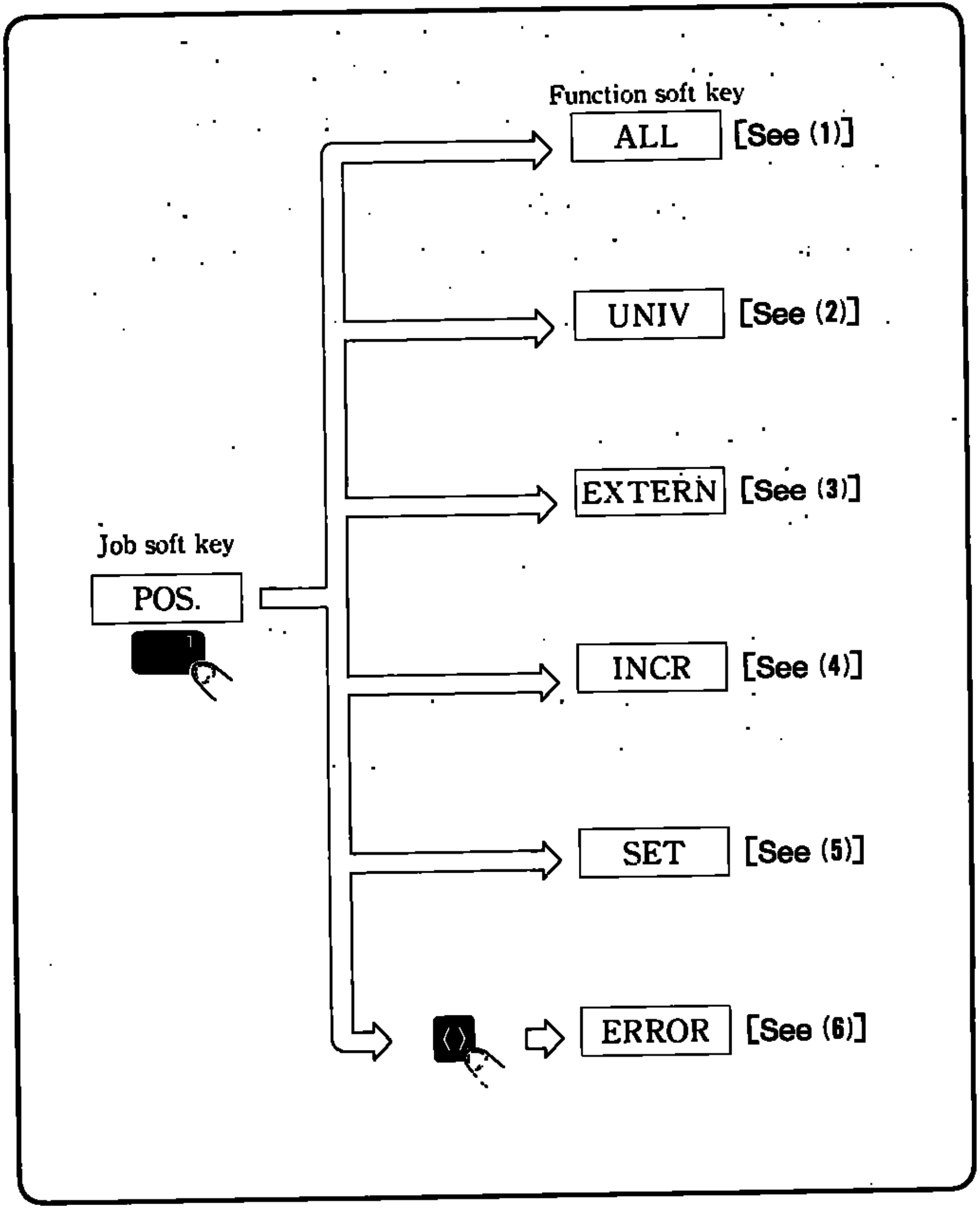
1. Depress the COMN key. Any of the following jobs is displayed.

- Current value job ———— See Par. 3.4.9.1 (page 540.)
- Alarm job ———— See Par. 3.4.9.2 (page 548.)
- Time job ———— See Par. 3.4.9.3 (page 551.)

2. To display a necessary job, depress the corresponding job soft key (POS. 1, ALARM 2, or TIME 3.)

3.4.9.1 Current value job (job soft key POS.)

KEYPOINT of operation

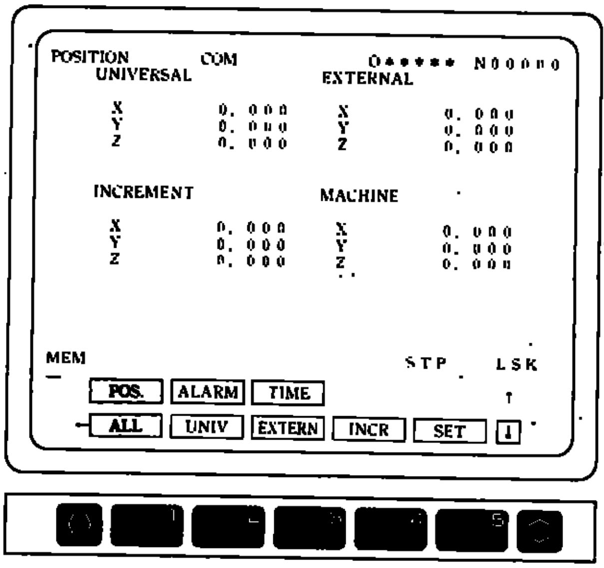


Depress the POS. job soft key to display any of the following functions:

- Current value batch function
- Work coordinate system function
- External coordinate value function
- Remainder length function
- Error pulse function

(1) Current value batch function

Depress the ALL function soft key to display the current value batch display screen.



```
POSITION          COM              O******  N00000
   UNIVERSAL                     EXTERNAL

   X        0.000                X        0.000
   Y        0.000                Y        0.000
   Z        0.000                Z        0.000

   INCREMENT                     MACHINE

   X        0.000                X        0.000
   Y        0.000                Y        0.000
   Z        0.000                Z        0.000


MEM                                     STP   LSK
```

Fig. 3.4.89 Current Value Batch Display Screen

Display of current values are explained in the following:

(a) **Work coordinate system**

The work coordinate system function displays the current positions of tools in the set coordinate system.

Whether to add the tool offset to work coordinate system representation can be determined by the following parameter:

Pm3000 D2 = 0 : Displays position in the work coordinate system adding the tool position offset.

Pm3000 D2 = 1 : Displays position in the work coordinate system without adding the tool position offset.

The work coordinate system is preset by the SET of the UNIV function.

(b) **External coordinate value**

The external coordinate value function displays the accumulated lengths of travel of the tool from the position preset by SET.

(c) **Remaindes length**

The remainder length function displays the following values:

- In the auto mode, the length from the current position of the tool to the end point of the block
- In the manual mode, the length from the current position of the tool to the starting point of manual operation. This display is reset to 0 when the auto mode is restored.

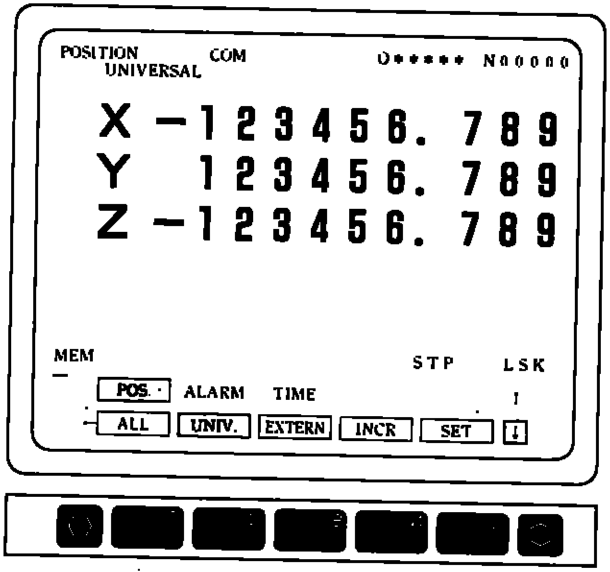
(d) **Error pulse**

The error pulse function displays the current position of the tool in the coordinate system of which origin is the reference point where the tool has returned by referece point return.

NOTE The axial name of the above current values are the display axial names determined by parameter Pm1100. If this parameter is not set, the NC machine default axial names are used.

(2) Work coordinate system function

Depress the UNIV function soft key to display the current values.

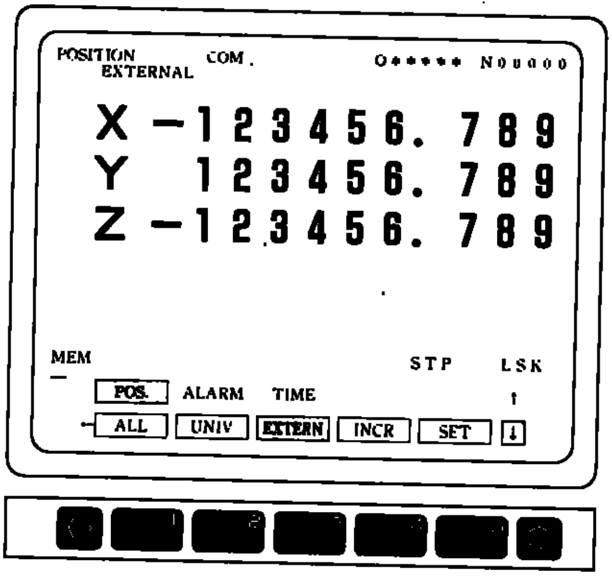


```
POSITION        COM          O****** N00000
  UNIVERSAL

   X -123456.789
   Y  123456.789
   Z -123456.789

MEM                           STP    LSK
     POS.  ALARM  TIME
   ALL  UNIV.  EXTERN  INCR  SET  ↓
```

Fig. 3.4.90 Work Coordinate System Display Screen

(3) Current coordinate value function

Depress the EXTERN function soft key to display the external coordinate values.



```
POSITION        COM.         O****** N00000
  EXTERNAL

   X -123456.789
   Y  123456.789
   Z -123456.789

MEM                           STP    LSK
     POS.  ALARM  TIME
   ALL  UNIV  EXTERN  INCR  SET  ↓
```

Fig. 3.4.91 External Coordinate System Display Screen

(4) Remainder length function

Depress the INCR function soft key to display the remainder lengths.

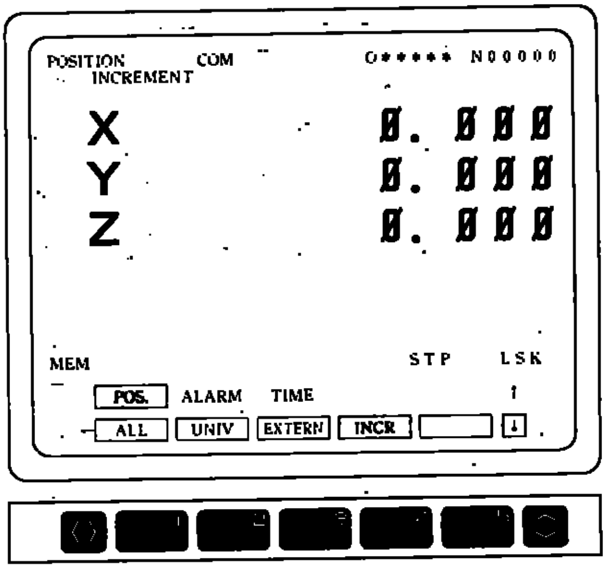


Fig. 3.4.92 Remainder Length Display Screen

(5) Coordinate system set-up function

The coordinate system can be set up when selecting function soft key ALL, UNIV, or EXTERN.

(a) When ALL function is selected

For the ALL function, the coordinate system of the external coordinate values can be set up. The set external coordinate system is always valid even during axis move by auto operation by the part program.

(b) When UNIV function is selected

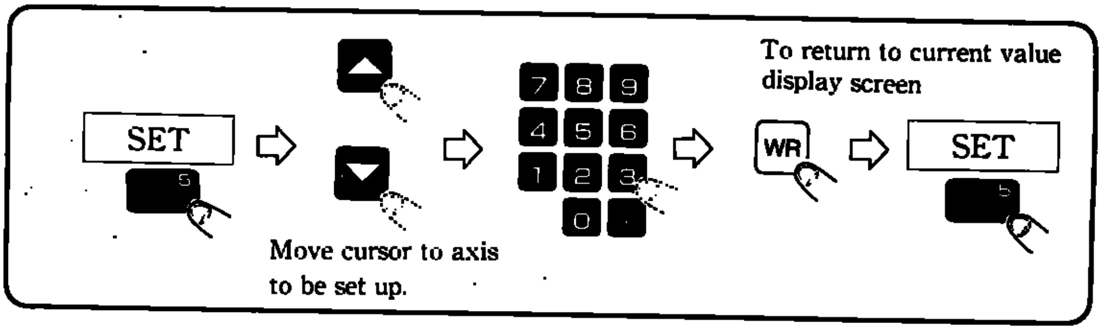
For the UNIV function, the work coordinate system can be set up.

This operation is valid in the manual operation (rapid traverse, manual feed, step, or handle) mode. If the coordinate system is set in other modes, a warning is issued.

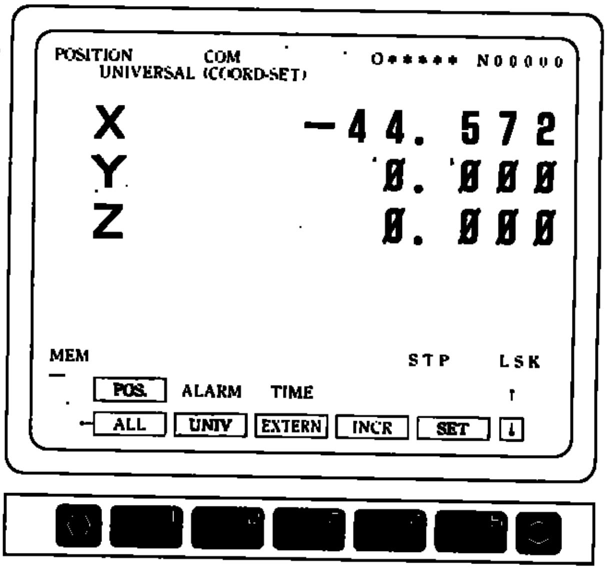
(c) When EXTERN function is selected

For the EXTERN function, the external coordinate system can be set up. The set external coordinate system is always valid even during axis move by auto operation by the part program.

KEYPOINT operation



1 Depress SET 5. The SET function soft key is highlighted in reverse and the name of the axis to be set up is displayed in reverse.



POSITION COM
UNIVERSAL (COORD-SET)
O****** N00000

X −44.572
Y 0.000
Z 0.000

MEM STP LSK

Fig. 3.4.93 Work Coordinate System Set-up Screen

2 Depress cursor key ▲ or ▼ to move the cursor to the axis to be set up.

3 Enter the coordinate system setup value.

Example : 2 1 0 . 5 WR …The input value is used for coordinate system set-up.

WR …0 is used for coordinate system set-up.

/ WR …1/2 of the position is used for coordinate system set-up.

After the coordinate system is set up, the cursor automatically moves to the next axis.

4 Depress the SET function soft key.

The cursor on the axial name disappears.

NOTE 1. To set 0 for the coordinate setup value, just depress the WR key.

2. If the above operation is performed in the mode where coordinate setup is impossible, a warning is issued.

(6) Error pulse display function

Depress the ERROR function soft key. The error pulse display screen appears.

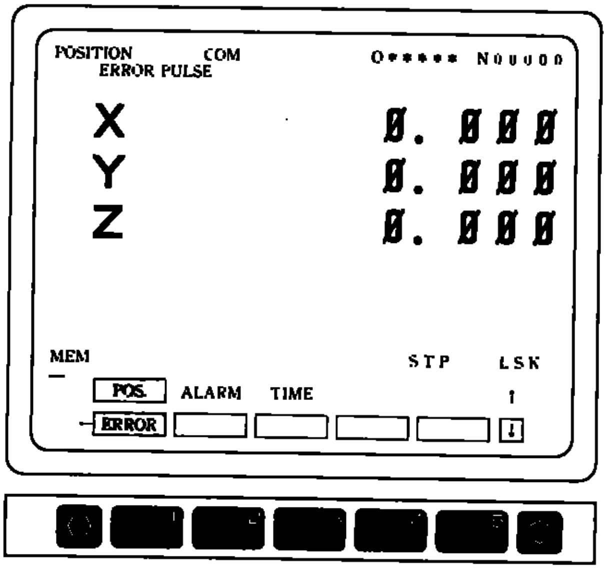
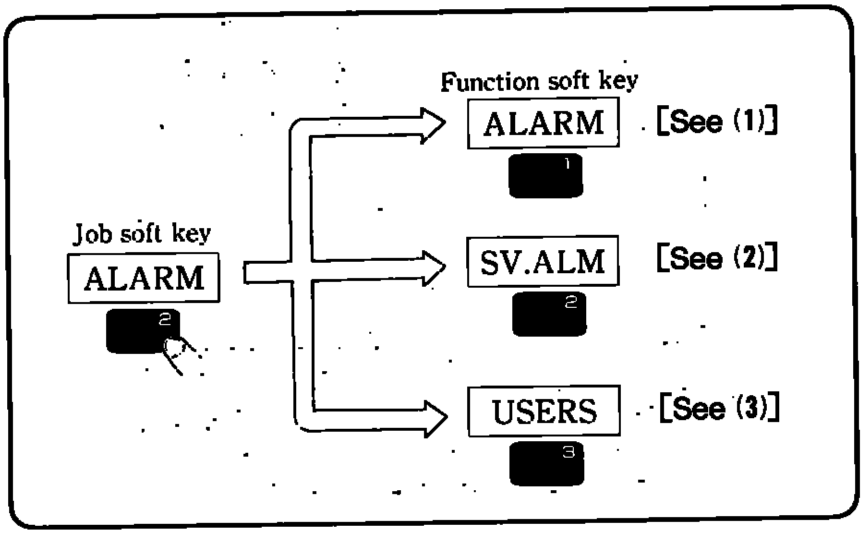


Fig. 3.4.94 Error Pulse Display Screen

Values displayed by the error pulse display function are the contents of the command pulse accumulation register in the NC equipment. The command pulse accumulation register is 0 when power is turned on, and accumulates all command pulses until power is turned OFF.

3.4.9.2 Alarm job (job soft key ALARM)

KEYPOINT



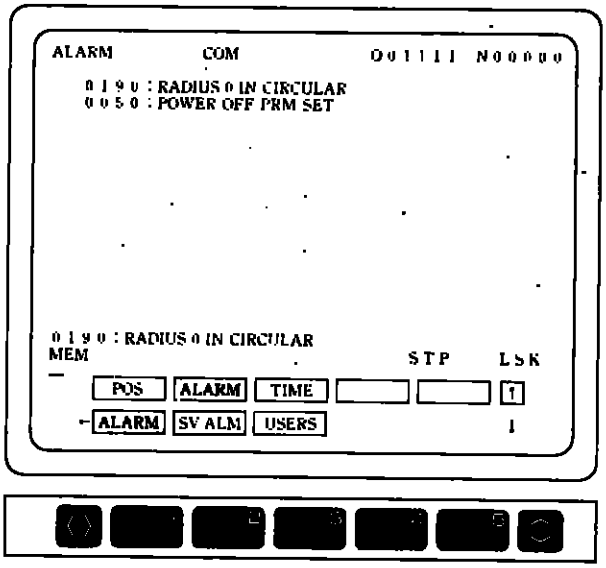
Depress the ALARM job soft key. The alarm function, servo alarm function, or users message function is displayed.

(1) **Alarm function**

If an alarm occurs in the controller, regardless of the mode or process, "ALM", "BGA" (background alarm,) or "A/B" or "B/B" (means that the battery alarm has also occurred) is displayed blinking in the alarm status field on the screen. Also, the highest priority alarm number, message, and the error section are displayed in the alarm message field.

Depress the ALARM function soft key to display the alarm screen for the entire system to monitor all current alarms.

Alarm numbers and the corresponding alarm messages are displayed in the order of the priority.



```
ALARM           COM           O01111  N00000
   0190 : RADIUS 0 IN CIRCULAR
   0050 : POWER OFF PRM SET

0190 : RADIUS 0 IN CIRCULAR
MEM                                STP    LSK
```

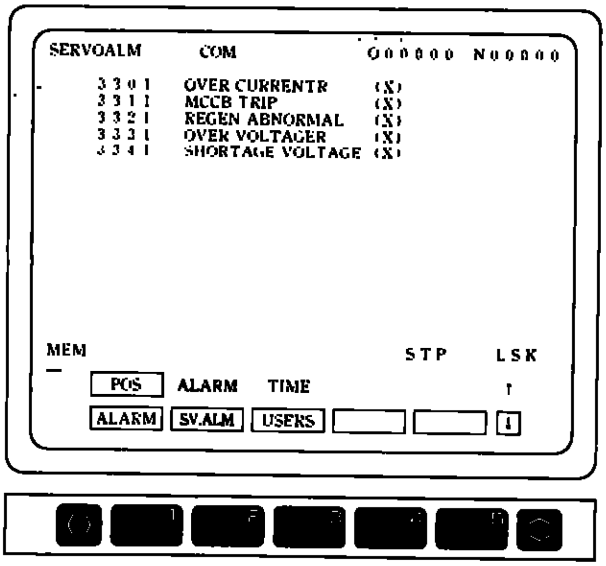
Fig. 3.4.95 Alarm Message Display Screen

(2) Servo alarm function

1 Depress the SV.ALM function soft key to display the servo alarm screen shown in Fig. 3.4.96.

The screen displays alarms related to the servo pack.

Call this screen to monitor the details of servo alarms if servo alarm messages are displayed on the alarm message display screen shown in Fig. 3.4.95.



```
SERVOALM        COM           O00000  N00000
     3301    OVER CURRENTR      (X)
     3311    MCCB TRIP          (X)
     3321    REGEN ABNORMAL     (X)
     3331    OVER VOLTAGER      (X)
     3341    SHORTAGE VOLTAGE   (X)

MEM                                STP    LSK
```

Fig. 3.4.96 Servo Alarm Screen

2 Depress page key ▲ or ▼ to turn to the next page. If all current alarms are displayed on the first page, the page keys are diregarded.

(3) Users message function

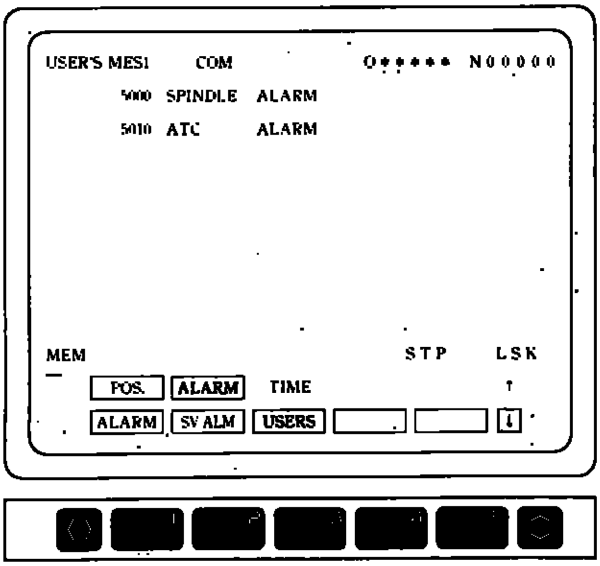
Messages can be displayed using the machine sequence controller (PLC system.)

Usually this function is used to display the alarm causes detected by the PLC. It can be used merely for message display. When a message display command (macro command) is executed in the PLC, "ALM" or "A/B" is displayed blinking in the alarm status field on the screen.

If the ALARM function soft key is depressed when "ALM" or "A/B" is displayed, the following appears:

```
ALM1080 SEQUENCE ERROR 0
ALM2180 SEQUENCE ERROR 1
ALM3240 SEQUENCE ERROR 2
```

Depress the USERS function soft key next to display the user's message screen for detailed error codes sent from the PLC:



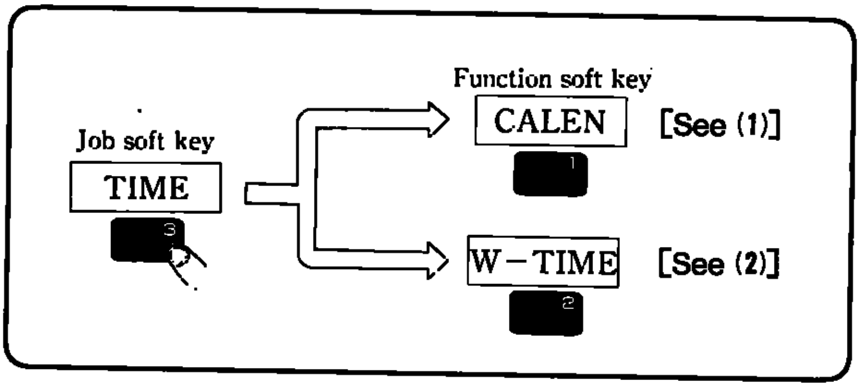
```
USER'S MES1      COM                O******  N00000
        5000  SPINDLE  ALARM
        5010  ATC      ALARM
```

Fig. 3.4.97 User's Message Screen

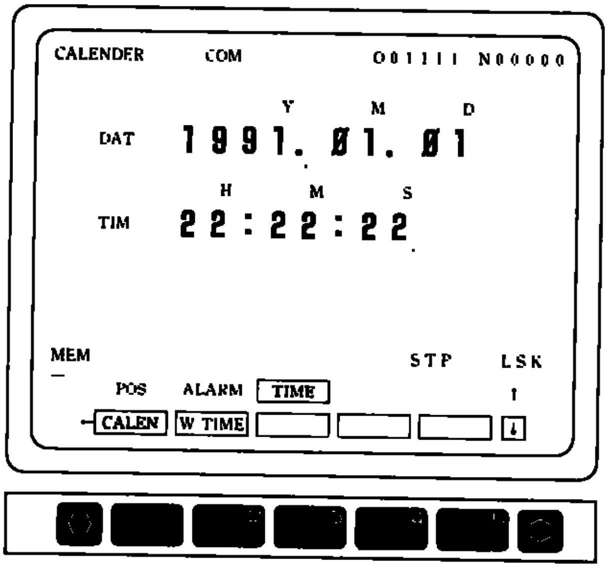
NOTE For more details, refer to the manual issued by the milling machine manufacturer.

3.4.9.3 Time job (job soft key TIME)

KEYPOINT



1. Depress the TIME job soft key. The calendar screen or the work time screen appears.



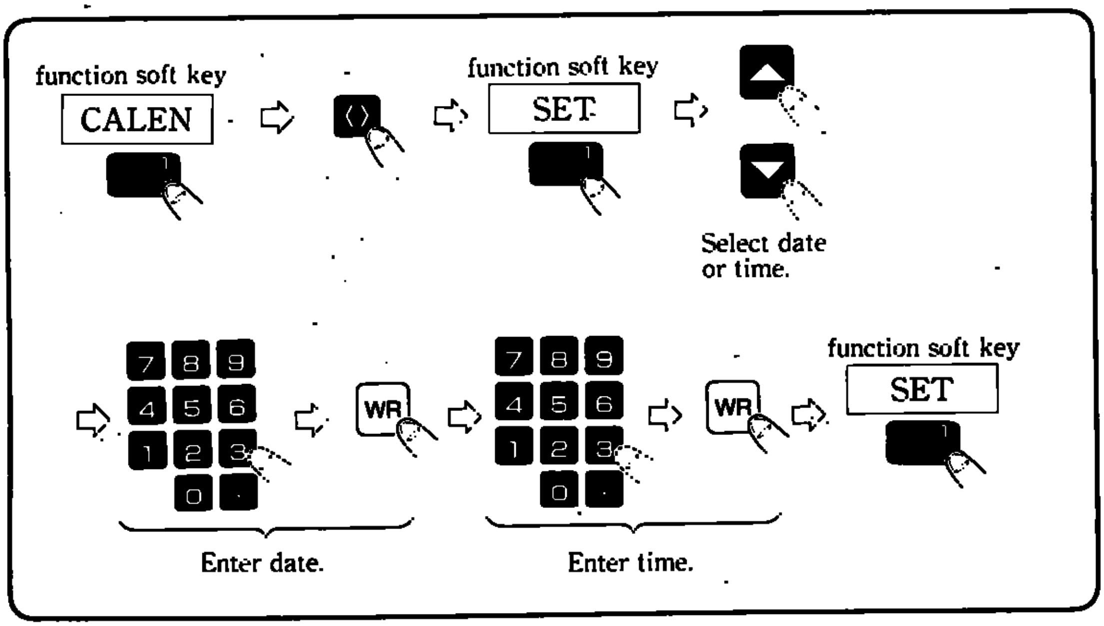
```
CALENDER     COM           001111  N00000

                   Y       M       D
DAT        1991. 01. 01
               H       M       S
TIM        22:22:22

MEM                            STP    LSK
   POS    ALARM  TIME                  ↑
 CALEN  W TIME                         ↓
```

Fig. 3.4.98 Calendar Screen

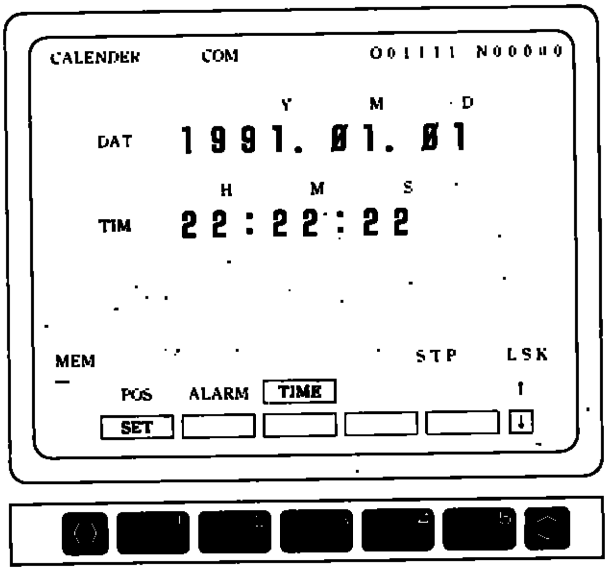
2. Select the CALEN or W-TIME function soft key.

(1) Calendar function

KEYPOINT



1 Depress the CALEN function soft key. The date and time held by the calendar function are displayed.



CALENDER COM O01111 N00000

DAT Y 1991. M 01. D 01

TIM H 22 : M 22 : S 22

Fig. 3.4.99 Calendar Setup Screen

2 To set date and time, depress 〈 〉 key . The SET function soft key is displayed.

3 Depress the SET function soft key. The key is highlighted and the cursor is displayed at the date field. The setup mode has started.

4 Select date or time using cursor key ▲ or ▼ .

5 Enter date.

(For example, 1 9 9 1 . 0 1 . 0 1 WR)

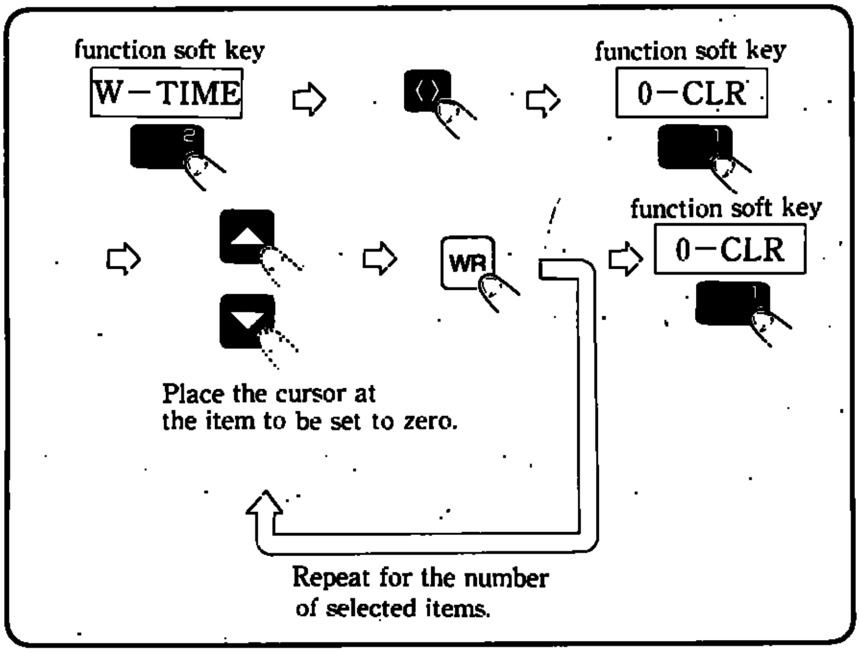
6 Enter time.

(For example, 1 2 . 0 0 . 0 0 WR)

7 Depress the SET function soft key. The cursor disappears and the entered date and time are displayed.

(2) Work time function

KEYPOINT



■ Depress the W−TIME function soft key.

The internally accumulated work times can be displayed. Use this function to check the processing time of a single workpiece or operation duration of a milling machine.
There are four types of work times, each counted in hours, minutes, and seconds :

(1)Power ON: Total time after power-ON

(2)Cycle start : Accumulated time of total duration of automatic operation.

(3)Cutting feed : Accumulated time of total duration of cutting.

(4)External timer : Accumulated time of total duration of external input signal being on.

2 To reset a work time, depress the operation change key ‹›. The 0−CLR function soft key is displayed.

3 Depress the 0−CLR function soft key.

The key is highlighted and the cursor is displayed at the power-on field. The zero clear mode has started.

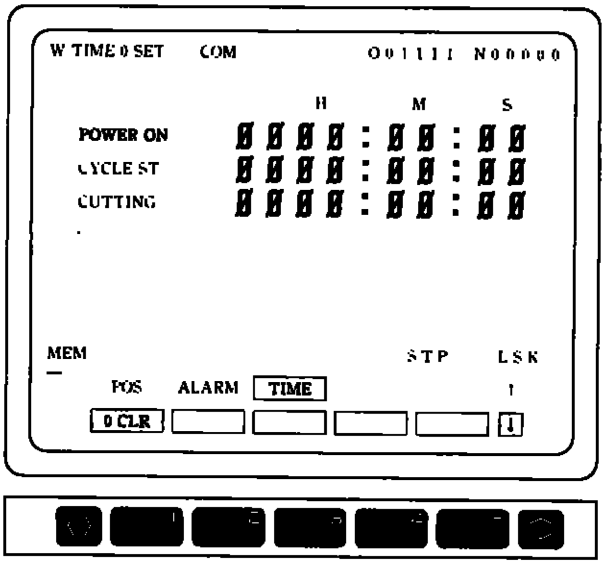


Fig. 3.4.100 Zero Clear Mode Screen

4 To reset a work time to zero, place the cursor at that work time using cursor key ▲ or ▼, then depress the WR key.

The work time at the cursor is cleared to zero. The cursor moves to the next item.

5 Depress the 0−CLR function soft key. The cursor disappears and the zero clear mode is canceled.

3.5 MACHINE CONTROL STATION.

3.5.1 Switching Units on Machine Control Station

(1) Fig. 3.5.1 shows an example of machine control station. The equipment layout and functions vary slightly with each machine. For details, refer to the operation manual issued by the machine tool builder.

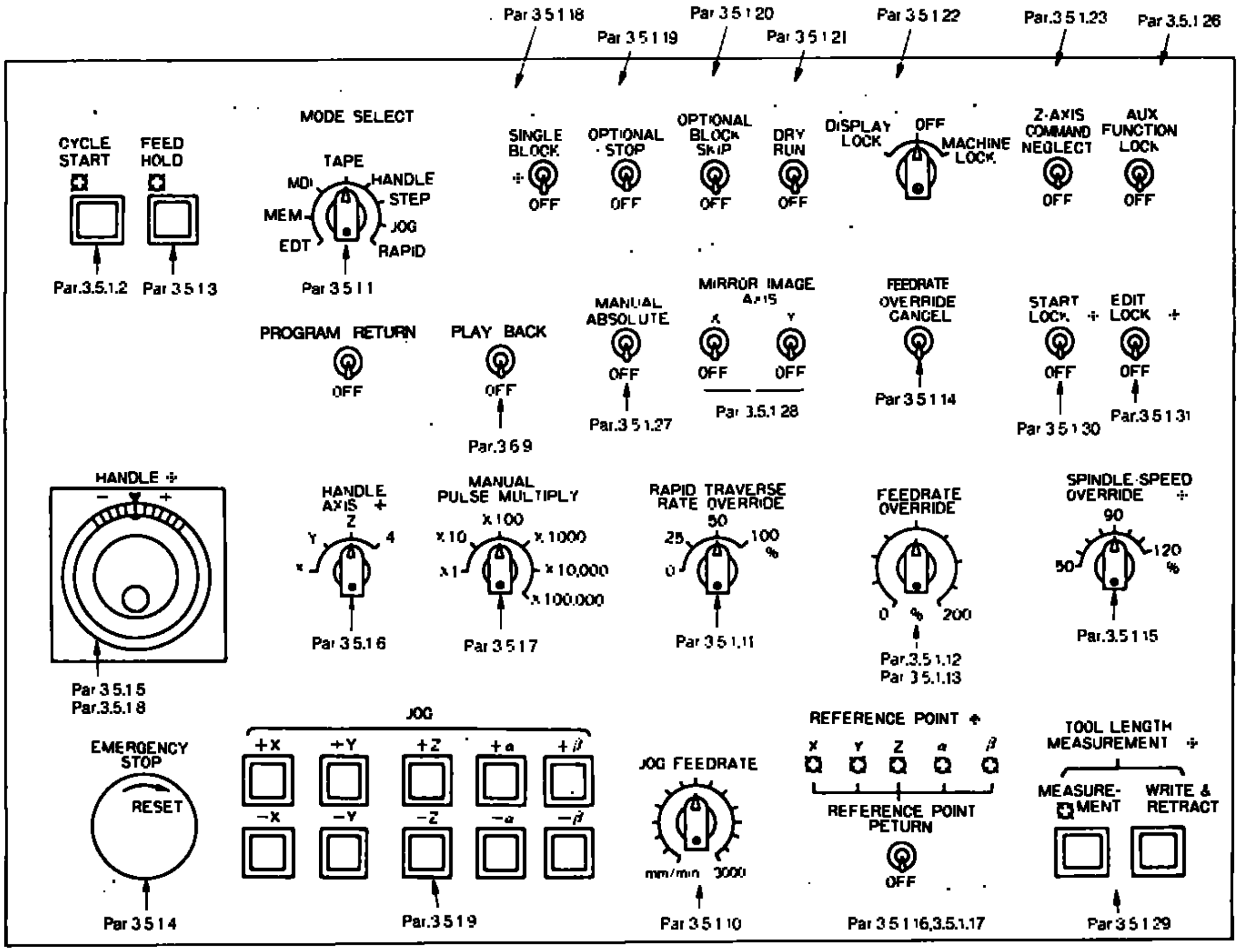
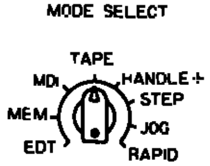


Fig. 3.5.1 Machine Control Station

3.5.1.1 Mode select switch (MODE SELECT)



(1) RAPID

Select this position to manually move the machine with rapid feedrate. Also use this position to manually return the machine to the origin.

(2) JOG

Select this position to manually move the machine with continuous feedrate. Specify the feedrate by using the JOG FEEDRATE switch.

(3) STEP

Select this position to move the machine with step feed. The machine moves one step when a JOG key is depressed.

(4) HANDLE ※

Select this position to use a manual pulse generator※.

(5) TAPE

Select this position to operate the machine automatically by using NC tape.

(6) MDI (manual data input)

Select this position to write command values and execute the contents of the command values by using manual data input (MDI).

(7) MEM (memory)

Select this position to operate the machine automatically by using the program stored in memory.

3.5.1.2 Cycle start key and lamp (CYCLE START)

This key instructs automatic operation to start in the automatic operation mode (TAPE, MDI, or MEM). When the machine starts operating, the CYCLE START lamp lights. Use this key to restart the machine after it is temporarily stopped by FEED HOLD or a change of mode.

3.5.1.3 Feed hold key and lamp (FEED HOLD)

This key temporarily stops automatic operation. When the machine is temporarily stopped, the CYCLE START lamp goes out and the FEED HOLD lamp lights.

(1) When the FEED HOLD key is depressed during axis movement, the machine feed is stopped after the speed is decelerated.

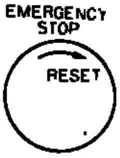
(2) FEED HOLD during tapping operation in G84: tapping cycle is disregarded. FEED HOLD during positioning is valid, not disregarded.

(3) If FEED HOLD is entered when the M, S, or T function is being executed without involving movement, each operation is executed through to completion although the FEED HOLD lamp lights immediately. (The same also applies for the B function.) After execution is completed, the indicator lamp lights and operation is stopped.

(4) To restart operation after FEED HOLD, depress the CYCLE START key.

NOTE When operation is stopped in the middle of a canned cycle due to single block ON, the FEED HOLD lamp lights automatically (to indicate that the process is in the middle of a canned cycle).

3.5.1.4 Emergency stop switch (EMERGENCY STOP)



Use this switch to stop the machine in an emergency or turn the power off. When this switch is depressed, the servo power supply of the NC equipment turns off and the machine feed is stopped by a dynamic brake. As a result of this operation, the READY lamp goes out and the NC ALARM lamp lights (alarm code 3002).

Follow the procedure shown below to restore the machine after emergency stop:

(1) Turn the EMERGENCY STOP switch clockwise to unlock.

NOTE EMERGENCY STOP switches of a lock type are generally used.

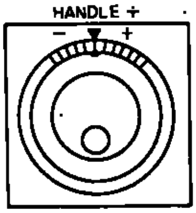
(2) Depress the  key on the CRT operator's panel.

— The alarm code changes from "3002" to "3000."

(3) Turn the servo power supply on by using the  switch.

— "ALM" indication on the CRT screen goes off, and "STP" is displayed.

3.5.1.5 Simultaneous one-axis control, manual pulse generator (HANDLE) ※



This pulse generator is used to move the machine manually. After selecting the HANDLE mode, turn the handle and the machine moves in the positive or negative direction depending on the direction the handle is turned. This is simultaneous one-axis operation by an axis select switch.

The following shows how to operate the handle:

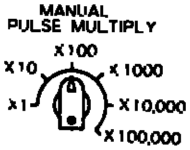
(1) Select the HANDLE mode.
(2) Select the axis with the HANDLE AXIS switch.
(3) Specify the quantity of movement per scale division of the handle with the MANUAL PULSE MULTIPLY switch.
(4) Turn the handle clockwise or counterclockwise, and the axis selected in (2) is moved accordingly.

Clockwise: Moves in the positive direction Counterclockwise: Moves in the negative direction

3.5.1.6 Handle axis select switch (HANDLE AXIS SELECT) ※

This switch selects the axis to be moved by handle operation. For details, see Par. 3.5.1.5.

3.5.1.7 Manual feed pulse multiply select switch (MANUAL PULSE MULTIPLY)



Select the quantity of movement per scale division of the handle by HANDLE (manual pulse generator) operation.

(1) This switch permits changing the movement quantity per scale division to × 1, × 10, or × 100 pulses.

NOTE The × 100 pulses can be changed to any desired rate of multiplication by parameter.
Pm2003 D7=0 : When selecting × 100, the multiplier is regarded as × 100.
Pm2003 D7=1 : When selecting × 100, the parameter setting (#2459) is used.

(2) Specify the quantity of movement (one step) to be accomplished each time the JOG (manual feed) key is depressed in the STEP mode.

NOTE Inertial run of the handle can be prevented by parameter Pm2865 (pulse buidup clamp quantity of handle). Note however that if the handle is turned quickly after setting this parameter, there will be a difference between the handle rotation quantity and movement quantity.

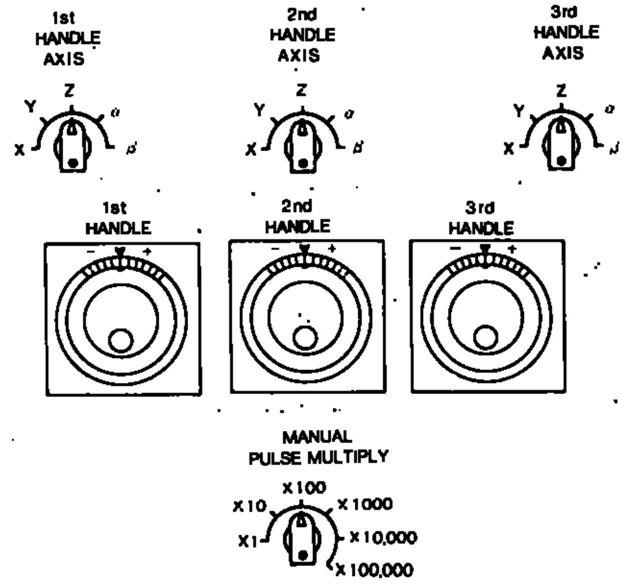
Table 3·5·1 HANDLE *

	mm	inch	* DEG
×1	0.001mm/scale	0.0001in/scale	0.001deg/scale
×10	0.01 mm/scale	0.001 in/scale	0.01 deg/scale
×100 ×1000 ×10000 ×100000	0.1 mm/scale	0.01 in/scale	0.1 deg/scale

Table 3·5·2 STEP *

	mm	inch	* DEG
×1	0.001mm/step	0.0001in/step	0.001deg/step
×10	0.01 mm/step	0.001 in/step	0.01 deg/step
×100	0.1 mm/step	0.01 in/step	0.1 deg/step
×1000	1.0 mm/step	0.1 in/step	1.0 deg/step
×10000	10.0 mm/step	1.0 in/step	10.0 deg/step
×100000	100.0mm/step	10.0 in/step	100.0deg/step

3.5.1.8 Simultaneous 3-axis control, manual pulse generator (HANDLES) ※

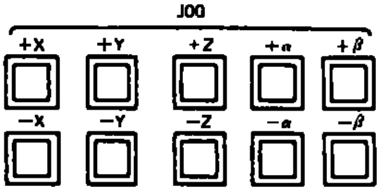


By fitting a manual pulse generator to each axis, it is possible to exercise handle control over a total of three axes among X, Y, Z, α, and β.

(1) The movement quantity per scale division of handle conforms to" manual feed pulse multiply switch" (Table 3.5.1) described above. One switch is installed for shared use by three axes.

(2) Select the HANDLE mode, then turn the handle of the desired axis to move it in the positive or negative direction.

3.5.1.9 Manual feed keys (JOG)



These keys are used to manually feed the machine.

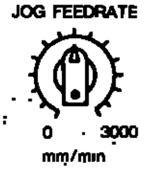
(1) While any key among + X, − X, + Y, − Y, + Z, or − Z (※ + α, − α, + β, or − β) is depressed in the RAPID mode, the machine moves with rapid feedrate in the corresponding direction.

(2) While these keys are depressed in the JOG mode, the machine moves at the feedrate selected by the JOG FEEDRATE select switch.

(3) Each time this button switch is depressed in the STEP mode, the machine moves one step as specified by the MANUAL PULSE MULTIPLY select switch. The maximum speed of one-step movement is determined by Pm2860 (linear axis) and Pm2861 (rotary axis).

NOTE The feed by the JOG keys described above can be applied for simultaneous all-axis operation.

3.5.1.10 Jog feedrate select switch (JOG FEEDRATE)



(1) This switch specifies a manual continuous feedrate in the JOG feed mode.

(2) Speed can be specified up to 32 steps.
This speed is preset by parameters (#2400 − #2431).

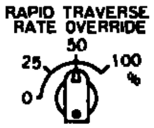
NOTE The number of steps used and the speed in each step vary depending on the machine, so refer to the instruction manual issued by your machine tool manufacturer (Table 3.5.3.).

Table 3.5.3 JOG Feedrate

STEP	JOG FEEDRATE		STEP	JOG FEEDRATE	
	Parameter No.	Typical Speed		Parameter No.	Typical Speed
0	Pm2400	0	16	Pm2416	100
1	Pm2401	1	17	Pm2417	120
2	Pm2402	2	18	Pm2418	150
3	Pm2403	4	19	Pm2419	200
4	Pm2404	6	20	Pm2420	250
5	Pm2405	8	21	Pm2421	300
6	Pm2406	10	22	Pm2422	400
7	Pm2407	12	23	Pm2423	500
8	Pm2408	15	24	Pm2424	600
9	Pm2409	20	25	Pm2425	800
10	Pm2410	25	26	Pm2426	1000
11	Pm2411	30	27	Pm2427	1200
12	Pm2412	40	28	Pm2428	1500
13	Pm2413	50	29	Pm2429	2000
14	Pm2414	60	30	Pm2430	2500
15	Pm2415	80	31	Pm2431	3000
		mm/min			mm/min

NOTE
1. Varies depending on the machine. Refer to the instruction manual issued by your machine tool manufacturer.
2. When the 4th and 5th rotation axes are installed, the machine moves at a rate directly converted from mm/min into deg/min.

3.5.1.11 Rapid feedrate override select switch (RAPID TRAVERSE RATE OVERRIDE)



(1) Some machines have a switch to override a rapid feedrate.

(2) There are four steps: 100%, 50%, 25%, F_0. The 100% speed is the same as the rapid feedrate set by parameters (Pm2801 to Pm2805).

(3) Rapid feedrate override is valid in both automatic operation (G00 command) and manual operation (RAPID).

(4) F_0 is the speed given by parameter (Pm2447).

(5) In addition to rapid feedrate override in four steps (100%, 50%, 25%, F_0), the rapid feedrate override in six steps (100%, 50%, 25%, F_0, F_1, F_2) is available by an option.
F_1 and F_2 are the speeds given by parameters (Pm2448, Pm2449).

3.5.1.12 Feedrate override select switch (FEEDRATE OVERRIDE)

In the automatic operation mode (TAPE, MEM, MDI), this function enables override to be applied in 32 steps from 0% to 540% for the feedrate instructed by the F code.

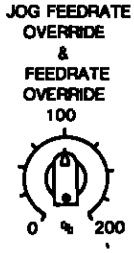
NOTE

1. Feedrate in G84: tapping cycle G74 is not affected by this switch; the machine is fed as instructed by the F code.
2. When the OVERRIDE CANCEL switch is on, the machine is fed as instructed by the F code without being affected by this switch.
3. The units of increment of overriding are 10% from 0 to 200%; 20% from 220% to 300%; and 40% from 340% to 540%.
4. STEP21 and above are optional.

Table 3·5·4 FEEDRATE OVERRIDE

STEP	%	STEP	%
0	0	16	160
1	10	17	170
2	20	18	180
3	30	19	190
4	40	20	200
5	50	21	220
6	60	22	240
7	70	23	260
8	80	24	280
9	90	25	300
10	100	26	340
11	110	27	380
12	120	28	420
13	130	29	460
14	140	30	500
15	150	31	540

3.5.1.13 JOG feedrate override select switch (JOG FEEDRATE OVERRIDE) ※



(1) When options are installed, override can be applied for JOG feedrate in 21 steps in 10% increments from 0% to 200%.

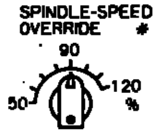
(2) This function is shared with the feedrate override select switch (FEEDRATE OVERRIDE). That is to say, when the JOG mode is selected, the feedrate override select switch turns to the JOG feedrate override select switch.

NOTE Use this function in the following way: Although the JOG feedrate select switch can be changed in up to 32 steps, limit the speeds to approximately three steps (low speed, middle speed, high speed) and supplement the intermediate speeds with the JOG feedrate override function.

3.5.1.14 Feedrate override cancel switch (FEEDRATE OVERRIDE CANCEL)

When this switch is turned on, the FEEDRATE OVERRIDE select switch is nullified.

3.5.1.15 Spindle speed override switch (SPINDLE-SPEED OVERRIDE) ※.



(1) By using the option, irrespective of the selected mode, this switch permits overriding the spindle rotation speed in increments of 10% from 50% to 120%.

(2) The spindle speed in tapping cycles G84 and G74 can be made unaffected by this switch (rotated as instructed by S-code). This can be accomplished by setting parameters (Pm2000 D2=0: override not 100% clamped or D2=1: override 100% clamped).

(3) In addition to 50% to 120% override of S-code, an extended function is available by an option that permits override in the range of 10% to 200%.

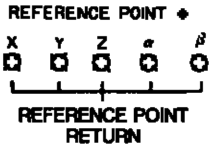
Pm1000 D6 = 0 : Spindle override 50 to 120%

Pm1000 D6 = 1 : Spindle override 10 to 200%

3.5.1.16 Manual reference point return switch (MANUAL REFERENCE POINT RETURN)※

Use this switch to manually return the machine to the reference point. For details on how to operate this switch, refer to Par. 3.5.3.1.1 "Manual reference point return."

3.5.1.17 Reference point position lamp (REFERENCE POINT) ※



This lamp indicates that the machine is at the reference point. It lights when the machine is positioned at the reference point by manual or automatic reference point return (G28) or reference point return check (G27). The lamp goes out when the machine moves off the reference point by machine or switch operation after that.

3.5.1.18 Single block switch (SINGLE-BLOCK)

Use this switch to execute the program stored in tape or memory block by block. When a cycle is started after turning this switch on, the program is executed one block and, when completed, the machine is stopped. When this switch is turned on in continuous operation, the machine is stopped after the block under execution at that time is completed.

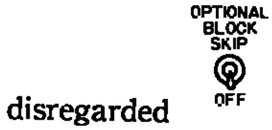
3.5.1.19 Optional stop (OPTIONAL STOP)

This switch specifies whether or not the M01 command is to be executed in the automatic operation modes (TAPE, MEM, MDI).

(1) When the switch is on, the machine stops after the M01-containing block is executed with its cycle start lamp remaining lit (goes out if the FIN signal is returned, however).

(2) When the switch is off, the M01 command is disregarded.
Operation of this switch is invalid for the block under execution. While in automatic operation, blocks become valid beginning with newly read blocks.

3.5.1.20 Optional block skip switch (OPTIONAL BLOCK SKIP)



This switch specifies whether or not data in blocks containing " " (slash) are to be disregarded in the automatic operation mode.

(1) When the switch is on, all commands from "/" (slash) to the end of that block are ignored. Block data preceding " " (slash) are valid.

(2) When the switch is off, blocks containing "/" (slash) are executed as are others.

Operation of this switch is invalid for the block under execution and blocks stored in the preread buffer. While in automatic operation, blocks become valid beginning with newly read blocks.

NOTE

1. " " and " 1" are equivalent commands.
2. When the optional block skip B-function * is installed, up to eight kinds of block skip operations can be independently performed by turning switches corresponding to "2" to "/9" on and off.
3. Operation of this switch is invalid for preread blocks.

3.5.1.21 Dry run switch (DRY RUN)

(1) When this switch is on in the automatic operation mode (TAPE, MEM, MDI), feed commands (F function)in the program are disregarded and the machine is operated with manual continuous feedrate. Because F-codes in the program are displayed as command values as are, this function permits efficient checking of the program flow.

(2) Rapid feed (G00) during dry run becomes as shown below depending on parameter specification (#2000 D2):

Table 3.5.5 Rapid Feedrate in Dry Run

Parameter Pm2000 D0	G00 in Dry Run
"0"	Rapid feedrate (See Note 2)
"1"	Manual continuous feedrate (JOG)

NOTE

1. When the dry run switch is changed over during automatic operation, dry run becomes valid immediately.
   In the mm/rev mode ※ and tapping, however, dry run does not become valid until the block is completed.
2. For rapid feed rate, the machine is fed with rapid feed override applied.
3. During tapping, the speed specified at the start of tapping is retained. Override cannot be changed prior to completion.

3.5.1.22 Display lock/machine lock switch (DISPLAY LOCK/MACHINE LOCK)

Use this switch to move the machine with the current value display fixed, or, conversely, to update the display with the machine fixed.

(1) OFF
Select this position for normal manual or automatic operation. Both machine and current value display move as instructed.

(2) DISPLAY LOCK
External current value display does not change even when the machine moves. Use this position when machine shift quantity is desired to be included in the display.

(3) MACHINE LOCK
Although the current value display changes according to manual or automatic operation as instructed, the machine does not move except when the M, S, and T functions are executed. Use this position to manually preset the display or check tape. Note however that when machine lock is on, return to the origin is not executed.

NOTE This switch must always be operated when the machine is stopped. The switch is not changed over in any machine state other than block stop and feed hold.

3.5.1.23 Z-axis command cancel switch (Z-AXIS COMMAND DISREGARD)

This switch is used for idling operation on the XY plane.
When this switch is on, the machine state is such that the Z-axis only is locked, so the Z-axis does not move.

NOTE However, the Z-axis position display is updated.
This switch must always be operated when the machine is stopped. The switch is not changed over in any machine state other than block stop and feed hold.

3.5.1.24 4th-axis disregard input

When the power is turned on while the 4th-axis disregard input is on (contact closed), the machine state is equivalent to one in which the 4th-axis is not controlled.

NOTE However, if the 4th-axis move command is issued in this state, the position display only is changed for the 4th-axis and the axis is not actually moved (machine locked state).

The position display of the 4th axis can be updated in the above state.

This switch must be operated when the machine is not operating. The switch is effective only in the block stop status or feed hold status.

3.5.1.25 5th-axis disregard input

When the power is turned on while the 5th-axis disregard input is on (contact closed), the machine state is equivalent to one in which the 5th-axis is not controlled.

NOTE However, if the 5th-axis move command is issued in this state, the position display only is changed for the 5th-axis and the axis is not actually moved (machine locked state).

The position display of the 5th axis can be updated in the above state.

This switch must be operated when the machine is not operating. The switch is effective only in the block stop status or feed hold status.

3.5.1.26 Auxiliary function lock switch (AUXILIARY FUNCTION LOCK)

When this switch is on, the M, S, T, (※ B) function commands are disregarded. This switch is generally used in combination with MACHINE LOCK to check the NC tape.

NOTE However, the following M-codes are processed normally:

1. M00, M01, M02, M30

   Both decode signal and BCD code are sent out.

   M90 to M99(internal processing M codes)

   These codes inherently do not send out BCD code.
2. When this switch is turned on during automatic operation, blocks become valid beginning with the next block under execution.
3. The auxiliary function lock is ineffective for the S5-digit command.

3.5.1.27 Manual absolute switch (MANUAL ABSOLUTE)

When the tool is moved in the middle of automatic operation by intervention of manual operation, this switch specifies how the manually moved quantity be handled when operation is restarted.

(1) When ON

When manually intervened, the program coordinate system does not change so that when automatic operation is restarted, the manually moved quantity is canceled in time and the tool is returned to the previous locus before intervention occurrs.

Movement cancellation is accomplished when position in the G00 or G01 mode is executed.

If the block after restart is a circular interpolation (e.g., G02 or G03), the machine operates while maintaining parallel movement of locus until the G00 or G01-mode command is issued.

For blocks in which execution was interrupted, the remaining steps are executed while parallel movement is left intact even in the G00 or G01 mode.

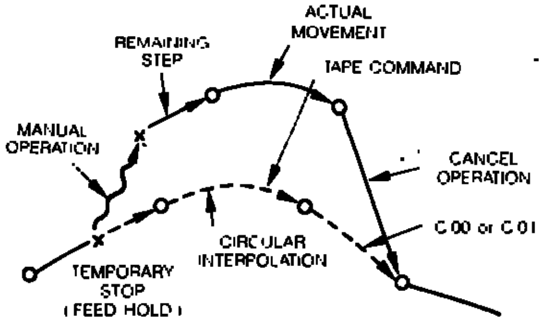


Fig. 3.5.2 Manual Absolute Switch ON

NOTE By using Pm4001 D7

It is possible to determine whether or not manual absolute be turned on in the G91 mode.

0: Valid 1: Invalid

Note however that even if G91 is nullified by setting "1" to this parameter, manual absolute becomes valid when G90 is encountered next.

(2) When OFF

When manually intervened, the program coordinate system shifts according to the manually moved quantity. Therefore, when automatic operation is restarted, subsequent operation is executed as with the parallel moved tool locus left intact.

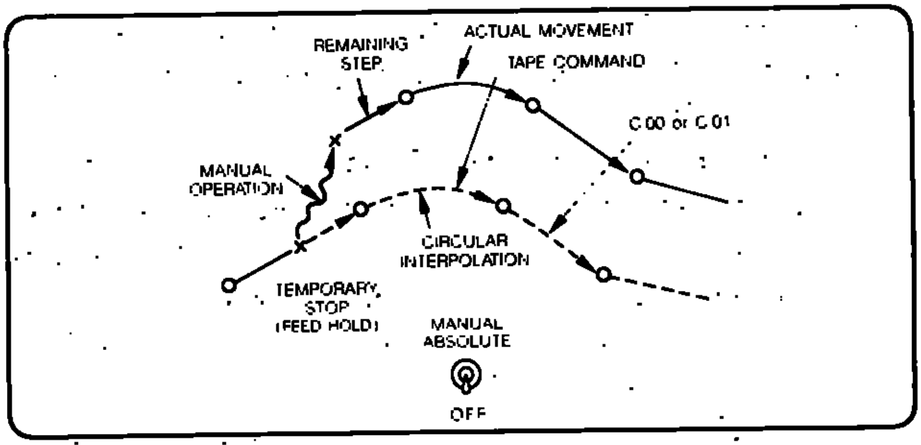


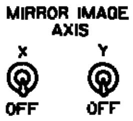
Fig. 3.5.3 Manual Absolute Switch OFF

The moved value by manual operation will be canceled if the following command or operation executes prior to cancellation.

- Manual or Automatic Zero return
- Reference Coordinate System Setting
- G31 Skip Execution
- [illegible] Operation

The command values are converted to agree with the current values.

3.5.1.28 Mirror image axis specification switch (MIRROR IMAGE AXIS)



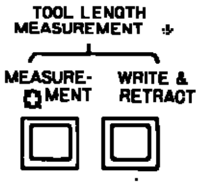
It is possible to specify the mirror image axis with a switch to perform symmetrical machining.

(1) For some machines, this switch may be activated simultaneously with mirror image axis specification by parameter setting. To specify on/off for the mirror image axis with this switch, parameter Pm5001 D1 must be set to "0."

(2) Turn on the switch for the axis that is desired to be a mirror image axis. When this is performed, mirror image is applied to move commands for that axis from when M95 is commanded in the program to when M94 is commanded. For details on operation, refer to Par. 2.13.4 "Mirror Image ON/OFF."

NOTE Do not change over the mirror image axis during the M95 (mirror image ON) mode.

3.5.1.29 Tool length measurement key and lamp ※



(TOOL LENGTH MEASUREMENT) ※

This key is used after actually installing a tool for Z-axis to have the manually moved quantity from the designated home position to the reference level automatically written directly into tool offset value memory.

NOTE The Z-axis returns from the reference level to the home position at 100% of rapid traverse rate (set by Pm2803.)

3.5.1.30 Start lock switch (START LOCK) ※

When input STLK is on, operation of the cycle start key is nullified. Use this function as start interlock when machine state is hazardous if put into automatic operation. (This switch may also be used as an operation switch.)

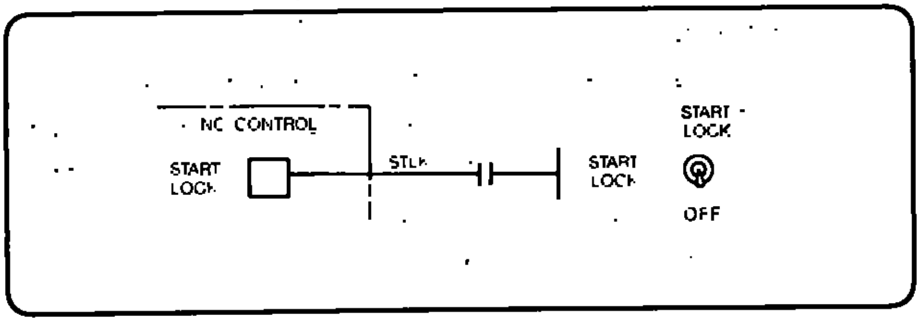


Fig. 3.5.4 Start Lock

NOTE Start lock switch during feed hold is invalid.

3.5.1.31 Edit lock switch (EDIT LOCK) ※

When this switch is on,

(1) ERASE, INSERT, ALTER keys are nullified.

(2) NC tape storing operation can be valid or invalid by the setting of parameter Pm3000.

D2=0 : Input operation valid
  1 : Input operation invalid
D3=0 : Output verification valid
  1 : Output verification invalid

NOTE When edit operation is performed while edit lock switch remains on, "EDIT LOCK" is displayed.

3.5.1.32 Interlock Input (INTERLOCK)

This interlock input is used to prohibit axis movement, and is provided for each axis.

(1) When an axis under movement is interlocked, it is stopped after deceleration. When the interlock is cleared, the remaining movement is continued, and control advances to the next block when that movement is completed.

(2) For simultaneous 2-axis and 3-axis interpolation commands, if one of those axes is interlocked, interpolation operation is prohibited.

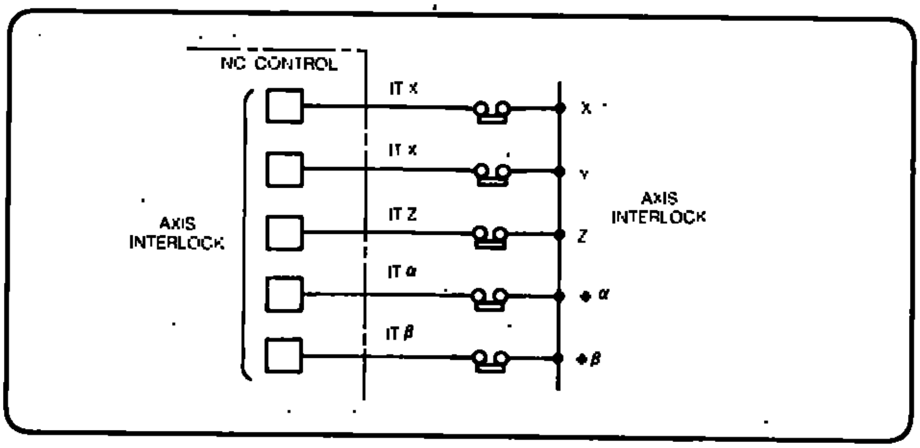


Fig. 3.5.5 Interlock Input

3.5.1.33 External deceleration input (EXTERNAL DECELERATION) ※

To prevent accidents due to erroneous move commands, an external deceleration limit switch is provided for some machines.

(1) For rapid feedrate command (G00) and manual operation
For movement in a direction in which the external deceleration limit switch operates, deceleration is started from that operating point and the machine is moved at a low speed set with parameter (Pm2444).

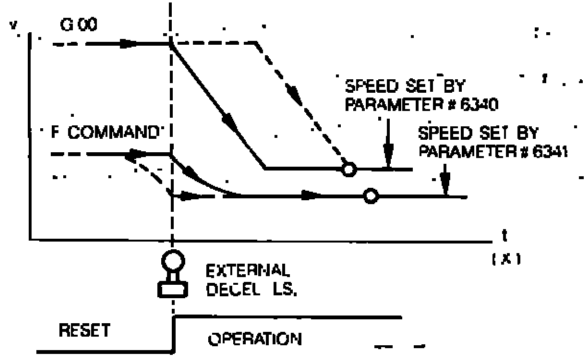


Fig. 3.5.6

(2) For cutting feed command (G94)
While the external deceleration limit switch is operating, the speed is reduced to a low speed set by parameter (Pm2445). If the speed instructed by F-command is lower than that set by this parameter, the machine is moved as instructed by F-command.

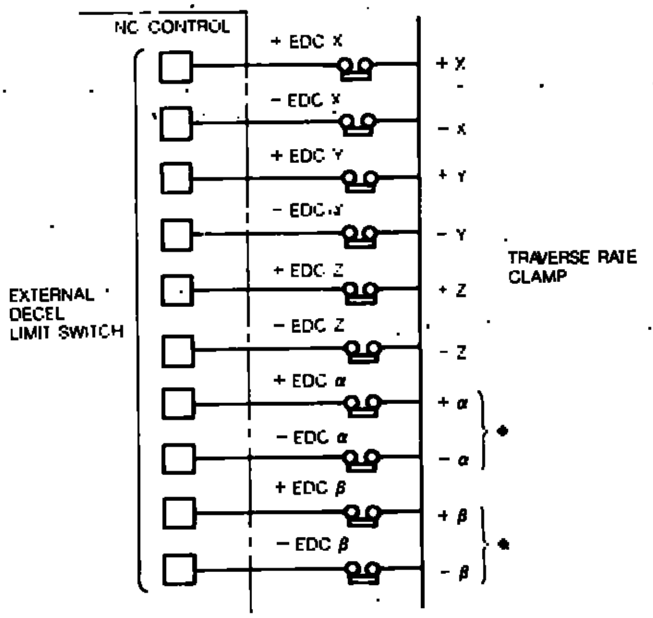


Fig. 3.5.7

NOTE
1. This deceleration function is invalid for specification of ※ feed per minute (mm/rev).
2. External deceleration is invalid for feed by HANDLE (manual pulse generator).

3.5.2 Operation Procedure

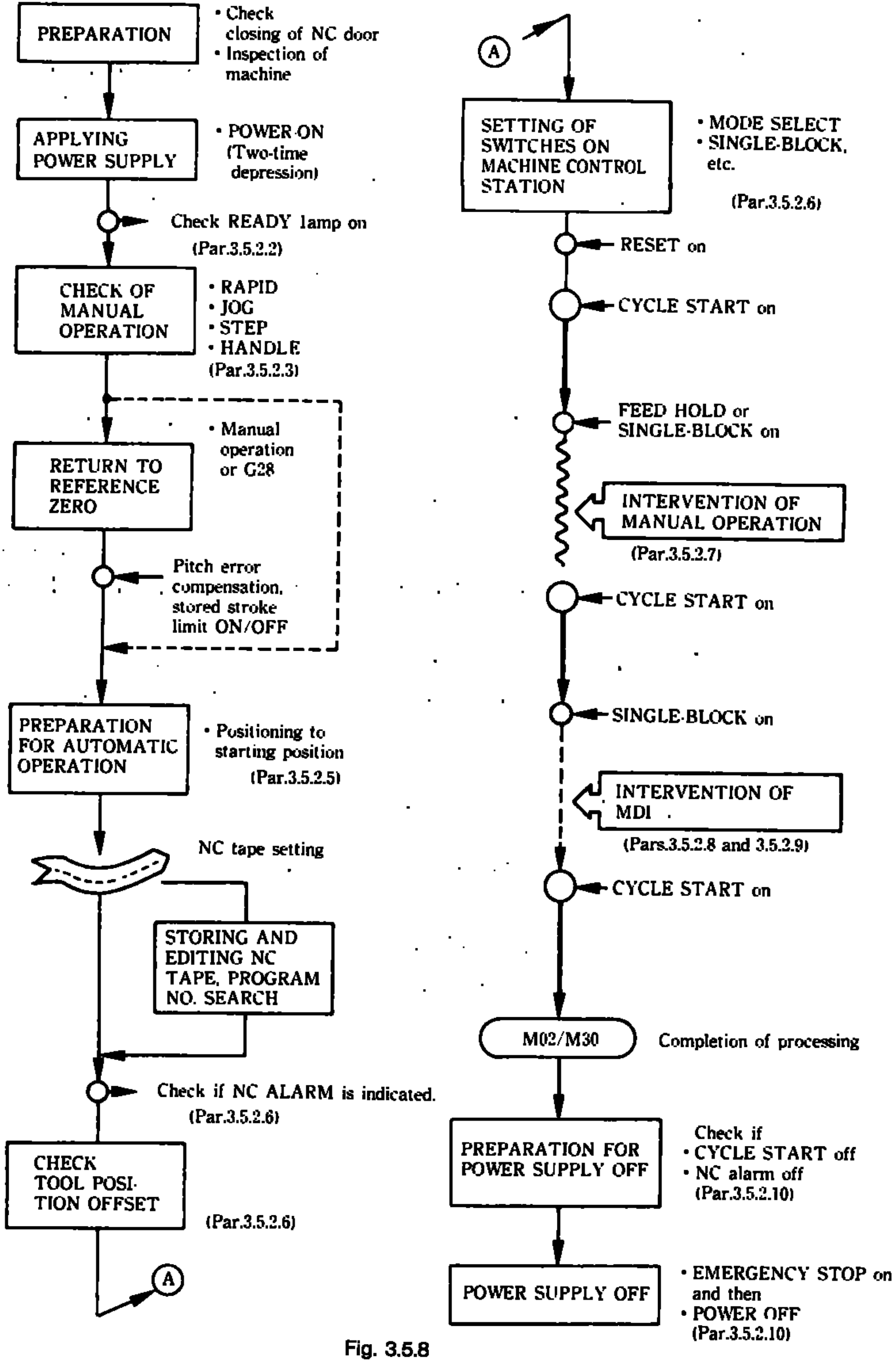


Fig. 3.5.8

3.5.2.1 Before turning power ON

(1) Confirm that the front and rear doors of the equipment are closed.

NOTE The equipment uses enclosed dustproof construction to prevent air from entering. If the door is open or there is a gap in the door, firmly close the door by using door open close metal fitting.

(2) Check the machine for the designated checkpoints.

NOTE See the instruction manual issued by your machine tool builder.

3.5.2.2 Power ON

(1) Confirm that external power supply is fed to the equipment.

(2) Depress the  switch ON the NC operation panel.

(i) When this is done, the control power supply turns ON, and the cooling fan starts operating. Confirm that air is flowing from the upper side location of the equipment.

(ii) The timer runs out approximately 20 seconds later, making the machine ready for the servo power supply to be turned on (alarm code 3000).

(3) Depress the  switch again.

(i) The servo power supply turns ON and, if the machine is readied, the NC is placed in a READY state.

(ii) When the power supply of the control unint is correctly turned ON, NRD (NC preparation complete) signal is sent out.

(iii) The power supply on the machine side is turned on by the NRD signal and when MRD (machine preparation complete) signal is returned, the READY lamp lights.

(4) If the READY state cannot be entered, examine the causes of malfunction by checking symptoms according to Par. 3.1.4.7.2 "Display of alarm codes" and take appropriate action. Also note that when the power is turned on, various checks must be made depending on the machine used. For details, refer to the instruction manual issued by your machine tool builder.

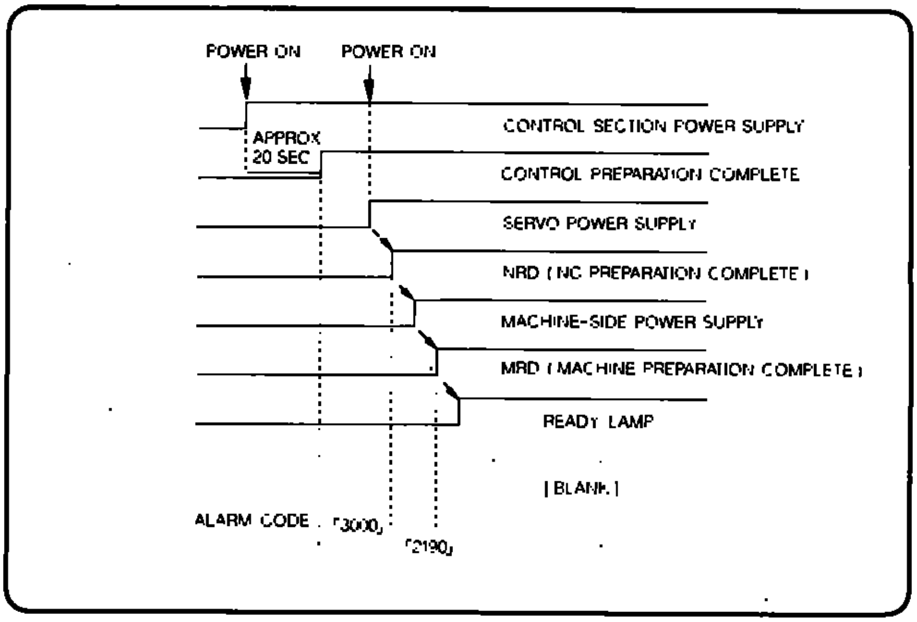
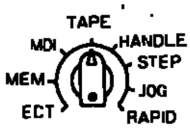


Fig. 3.5.9 Turning Power ON

3.5.2.3 Manual operation

When moving the machine manually, select the RAPID, JOG, STEP, or HANDLE mode depending on the purpose.



(1) Manual rapid feedrate (RAPID)

① Select the RAPID mode.

② Select the speed by using the RAPID TRAVERSE RATE OVERRIDE select switch.

100%, 50%, 25%, F_0

③ Depress the JOG key for the axis and the desired move direction. While the key is held down, the machine moves with rapid feedrate.

(2) Manual continuous feedrate (JOG)

① Select the JOG mode.

② Select the speed by using the JOG FEEDRATE select switch. Available in up to 32 steps.

③ Depress the JOG key for the axis and the desired move direction. While the key is held down, the machine moves at the specified speed.

(3) STEP feed

① Select the STEP mode.

② Select movement of one step with the MANUAL PULSE MULTIPLY selection switch.

mm System	0.001, 0.01, 0.1, 10.0, 100.0 mm/step
inch System	0.0001, 0.001, 0.01, 0.1, 1.0, 10.0 in/step

③ Depress the JOG key of the axis to be moved and the direction in which it is to be moved.

→ The machine moves an amount equal to the movement per step selected each time the key is depressed.

(4). HANDLE feed ※

The following operation is possible if equipped with a manual handle.

① Select the HANDLE mode.

② Select the axis to be moved with the HANDLE AXIS switch.

③ Select the amount of movement per handle graduation with the MANUAL PULSE MULTIPLY switch.

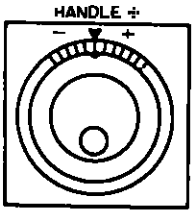
mm System	0.001 – 0.01 – 0.1 mm/scale
inch System	0.0001 – 0.001 – 0.01 in/scale

NOTE Even if ×1000 or ×10000 is selected, it becomes the same as ×100.

④ Turn the manual handle

→ The machine moves forward or backward corresponding to the direction of rotation of the handle.

Clockwise (CW)	Foward direction
Counterclockwise (CCW)	Backward direction



3.5.2.4 Compensation of stored type pitch error and preparation of stored stroke limit.

(1) Reference point return

In equipment provided with pitch error compensation or stored stroke limit function, reference point return operation (a) or (b) below will be required before automatic operation after switching power ON.

(a) Conduct manual reference point return operation. (See Par. 3.6.1(1))

(b) Execute G91 G28 XO YO ZO; in the MDI mode.

— This operation is to teach the equipment the reference point since the operating data are decided with the reference point as the standard, together with pitch error compensation and stored stroke check.

(2) Confirmation of parameter Pm4018 D0 to D4

(For setting parameter switch at 1)

Alarm "0411 to 0415" (reference point return incomplete) will occur if the CYCLE START key is depressed without executing reference point return immediately after switching the power ON.

NOTE

1. Be sure to perform reference point return in (1) above.
2. If provided with pitch error compensation or stored stroke limit, set this parameter to "1".

3.5.2.5 Preparation for automatic operation

To start automatic operation after turning on the power, it will be necessary to position the starting point in accordance with the program. It will then be necessary to set the coordinate system correctly to operate or process according to the program. A few examples will be shown, but refer to the instruction manual of the machine tool builder, for details.

(1) When there is no coordinate setting command (G92) in the program

① Return the machine to reference point with the "Manual reference point return" operation in Par.3.6.1(1).

② Set parameter with the MDI operation in accordance with the program and execute the following.

```
G92 X······Y······Z······ ;
```

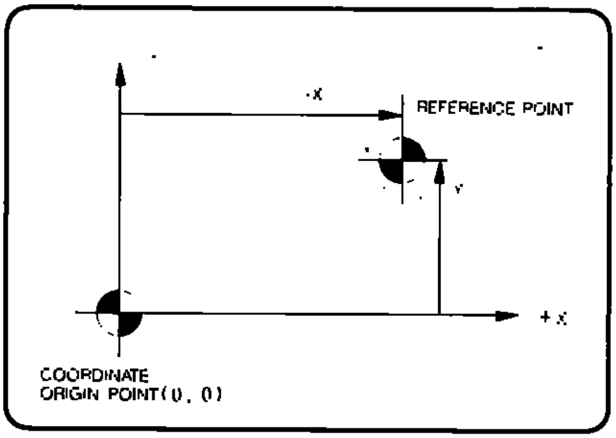


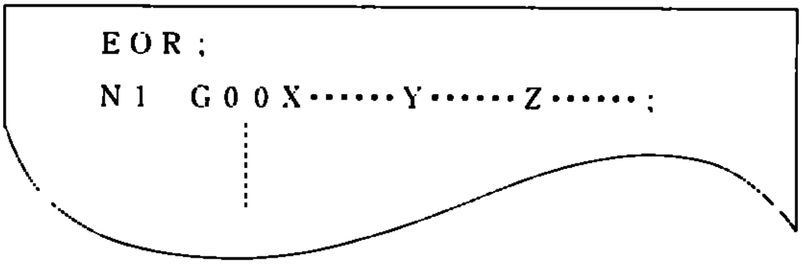
Fig. 3.5.10 Machining Coordinate System

③ The machining coordinates are decided with the reference point as the standard.

When the coordinate system setting is

```
G92 X0 Y0 Z0 ;
```

corresponding to the program, 0 can be easily set for the coordinate value by depressing the ZERO SET key following① above. See Par.3.4.4.7.



```
EOR ;
N1  G00X······Y······Z······ ;
     :
```

Fig. 3.5.11 Example of Programming

(2) When G92 is specified in the program
When G92 is the program that must be executed at the reference point, return the machine to the reference point by operating "Manual reference point return" in Par.3.6.1(1).

```
EOR;
N1   G92X······Y······Z······;
       :
```

Fig. 3.5.12 Example of Programming

(3) When G28 and G92 are specified in the program
If G28 is specified at the head of the program, bring the machine within the range possible for reference point return by moving it a certain distance away from the reference point.

```
EOR;
N1   G28X······Y······Z······;
N2   G92X······Y······Z······;
       :
```

Fig. 3.5.13 Example of Programming

3.5.2.6 Memory operation

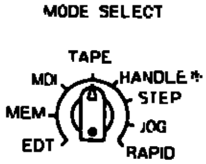
(1) Confirm that the alarm indicating lamp on the operator panel is not lit.

→If the lamp is lit, check cause according to Par.3.4.9.2 "Alarm code display" and cancel the alarm.

(2) Confirm and correct the amount of tool offset, and position the machine at its starting point.

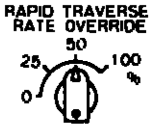
(3) Set the switches on the machine operating panel as follows.

(a) MODE SELECT



(b) SINGLE-BLOCK

(c) RAPID TRAVERSE RATE OVERRIDE



(d) MANUAL ABSOLUTE

(e) OPTIONAL BLOCK SKIP

(f) OPTIONAL STOP

(g)

DRY RUN

(h)

FEEDRATE OVERRIDE & JOG FEEDRATE

(4) Depress the RESET key. LABEL SKIP will be displayed and memory rewind will be conducted.

(5) Start automatic operation by depressing the CYCLE START key.

(6) Depress the FEED HOLD key to pause during operation.

NOTE

1. If an unexpected state occurs, stop the machine by depressing the EMERGENCY STOP switch.
2. Operation can be started from the middle of a part program.
   Set the cursor in the middle of the part program using address search, then start the cycle. The modal G code must be set in advance.
   Enter an address and numeric data for address search in the memory mode.

3.5.2.7 Intervening with manual operation during automatic operation

(1) Interrupt operation by depressing the FEED HOLD key or SINGLE-BLOCK switch.

(2) Record the stopping position of the machine according to the value displayed.

(3) Select MANUAL mode and conduct manual operation.

(4) Return the machine to the point of intervention.
Move the machine manually to the position recorded in (2) above.

(5) Select the auto mode (MEM/MDI) that was set prior to intervention.

(6) Automatic operation will restart if the CYCLE START key is depressed.

NOTE

1. The following will occur if mode is changed over during automatic operation without conducting the stopping operation.
   (a) Will decelerate and stop immediately if changed to MANUAL mode from AUTO mode.
   (b) Will stop at end of block if changed from one AUTO mode to another AUTO mode.
2. If automatic operation is restarted with the machine still in the manually shifted position, the tool locus after restarting will differ according to ON/OFF of the MANUAL ABSOLUTE switch. See Par.3.2.1.27 "MANUAL ABSOLUTE Switch".
3. Operation will become effective immediately and output signals such as the BCD code will be sent in the MANUAL mode when the CYCLE START key is depressed after first writing the F, M, S, T or B code by the same procedure as when writing MDI. This is called manual MDI.
   This is effective as a means of intervention to the actual buffer currently being executed instead of the advance read buffer. However, writing M00, M01, M02, M30, M90 to M99, M190 to M199 is not possible.

3.5.2.8 Automatic operation with MDI

(1) Select the MDI mode.

(2) Write the command value of the block by MDI operation.

(3) Start execution by depressing the CYCLE START key.

3.5.2.9 Intervening with MDI operation during automatic operation

Operate as follows when changing command values and intervening with MDI operation during TAPE or MEM mode operation.

(1) Interrupt automatic operation by depressing the SINGLE BLOCK switch to ON.

— Operation stops after completing the block being executed. There may be cases when advance read is conducted on several blocks at this time.

(2) Select the MDI mode and execute writing of the necessary command values.

(3) Select the mode(TAPE/MEM) prior to interruption.

(4) Return the SINGLE-BLOCK switch to OFF.

(5) Automatic operation restarts when the CYCLE START key is depressed.

NOTE

1. Generally, intervention with MDI operation is not possible in the TOOL RADIUS OFFSET mode (G41, G42) since advance reading of block is being conducted. However, output is possible in the MANUAL mode as shown in par 3.2.2.7 (3) relative to the F, M, S, T and B codes.
2. Intervention with MDI operation must not be conducted during the canned cycle modes (G73, G74, G76, G77, G81 to G89). There will be instances where correct operation cannot be guaranteed.
3. Intervention with MDI operation will be possible in cases other than the foregoing.

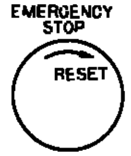
3.5.2.10 Preparation prior to power OFF and during power OFF operation

(1) Confirm that the machine has stopped and that the CYCLE START lamp is out.

(2) Confirm that the alarm display on the CRT is OFF.

NOTE If still lit, check cause according to Par.3.4.9.2 "Alarm code display" and cancel the alarm.

(3) Conduct other necessary checks on the machine side.

(4) Depress the  switch and turn OFF the servo power supply.

(5) Depress the POWER OFF switch ON the operator panel and turn off all power to the control panel.

(6) Turn OFF the power to the unit.

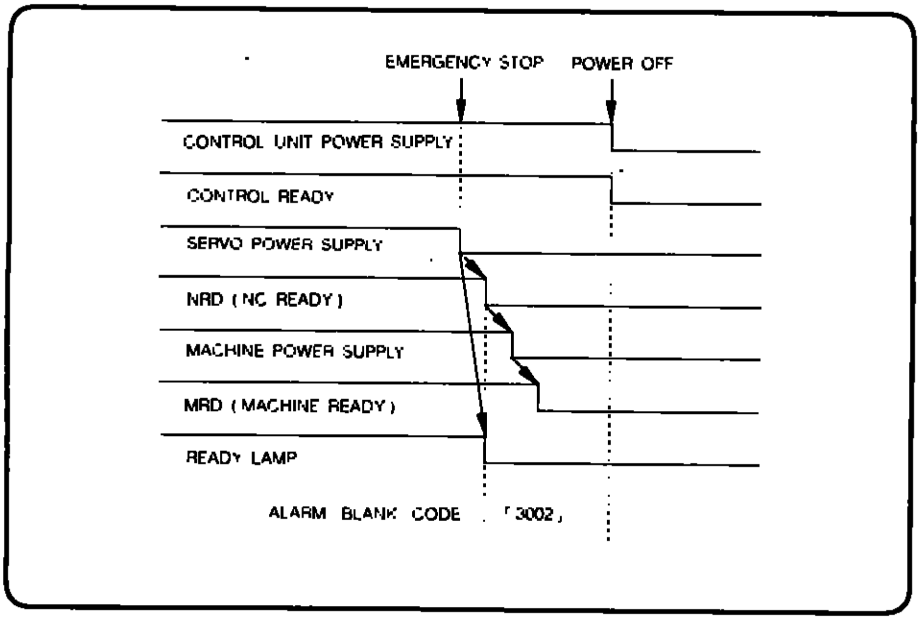


Fig. 3.5.14 Preparation Prior to Power OFF and During Power OFF Operation

3.6 VARIOUS OPERATIONS AND FUNCTIONS

3.6.1 Reference Point Return Operation

3.6.1.1 Manual reference point return

This is the function to return manually to the reference point and the operation is as follows.

1. Select the RAPID (or JOG) mode.
2. Move the moving part of the machine manually to a position away from the reference point.

NOTE Reference point return can be executed properly if its starting position is within area A of Fig.3.6.1.

3. Set the REFERENCE POINT RETURN switch to ON.
4. Hold down the JOG key in the direction of reference point return.
   — Machine movement starts similar to normal manual feed. Traverse becomes low speed when the deceleration point is passed and the machine stops automatically at the reference point.
5. When reference point return ends, the REFERENCE POINT lamp lights.

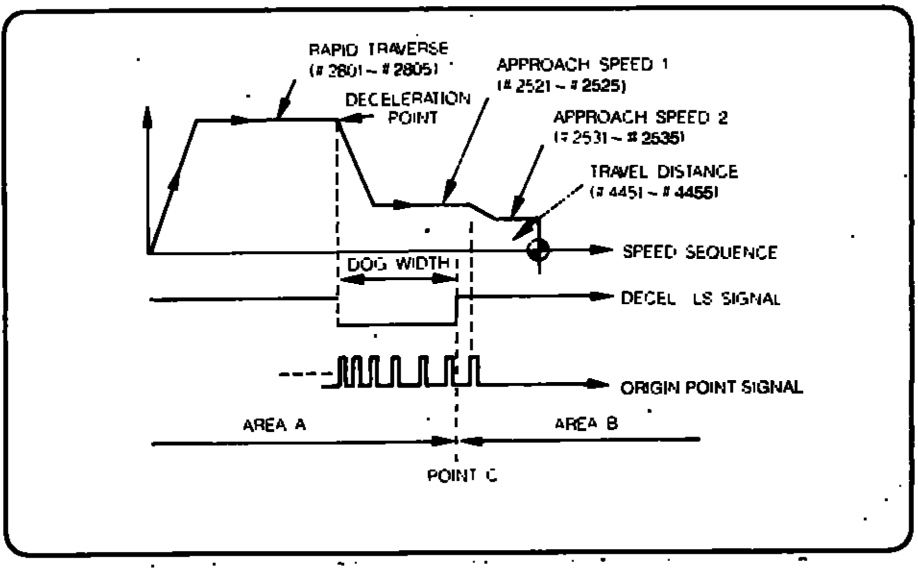


Fig. 3.6.1 Return of Manual Reference Point

NOTE

1. Although manual or auto reference point return operation starts after power is turned on regardless of the position of the moving part of the machine, correct return is not possible if the moving part is in area B. Always conduct reference point return after the moving part is in area A.
2. Error will occur if reference point return is conducted while in area B since point C will be stored in the equipment once reference point return is conducted. Therefore, return to area A and conduct reference point return again.
3. The direction of reference point return of the axis on which reference point return is completed cannot be changed by manual operation as long as the REFERENCE POINT RETURN switch is OFF.
4. An alarm occurs if the mode is changed during reference point return self-running.
   Alarm 2141 to 2145 : Reference point return interruption error.
5. Reference point return is not executed if the machine lock switch is ON.
6. Do not conduct manual reference point return during auto operation with an advance read block in the buffer since the movement data in the advance read buffer will be erased.
7. Rotary axis manual return to reference point.
   The rotary axis can also be returned to the reference point by the same operation as the linear axis. After moving ±360.000°, the rotary axis returns to the reference position by turning a minimum amount in the home positioning direction.

   For example, from positions A and B, the axis returns to positions A' and B', respectively. (The direction of approaching the home position is determined by Pm4002 (D3, D4).

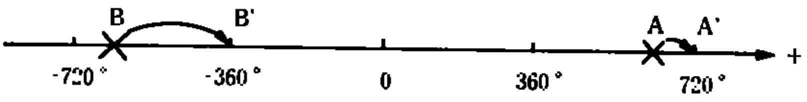


Fig. 3.6.2 Return of Rotation Axis to Reference Point
(Approach to home position from negative to positive)

8. High-speed return to reference point

After once returning to the reference point, the axis is positioned to the determined reference point in the second and subsequent returns.

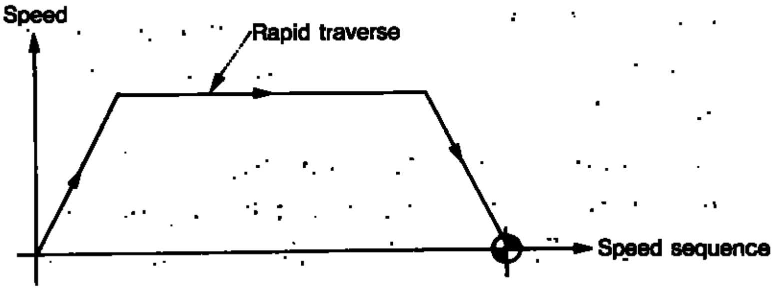


Fig. 3.6.3

To perform second and subsequent operations at low speed, set the parameter for low speed manual return to reference point (Pm4003 D6=1).

3.6.1.2 Easy return to reference point ※

(1) Outline

Since the place where the actual phase-C pulse appeared cannot be measured for the zero-point LS signal position at the conventional machine zero-point setting, the optimum position was measured after several trial times by adjusting the zero-point LS signal.

On the other hand, by rewriting the internal data, the optimum reference point position can be set quickly and easily.

This function employs a new concept of imaginary phase-C pulse. The phase-C pulse position output by one pulse per motor revolution is fixed mechanically and adjustment is not needed. Therefore, besides this, an imaginary phase-C pulse which can occur at any desired position during the motor rotation is assumed. This pulse is called imaginary pulse-C and indicated as the shift distance from the phase-C pulse.

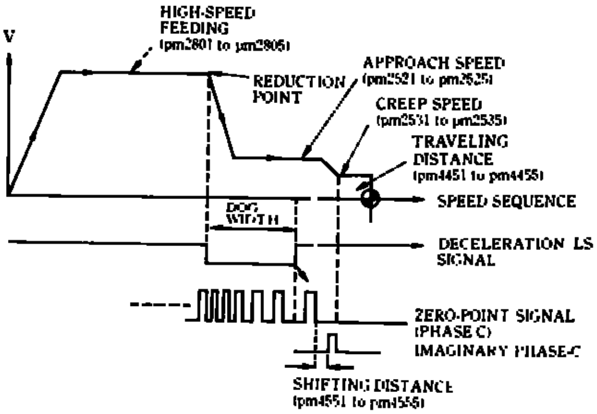


Fig. 3.6.4

(2) Function description

The same operation method of reference point return is employed as the conventional method.

By using the concept of imaginary phase-C, this function can change the relation between the zero-point LS signal and phase-C pulse positions or the relation between the zero-point LS signal and reference point positions only by rewriting the parameters instead of adjustment of zero-point LS signal position. Additionally, by adjusting the final traveling distance, the distance from the zero-point LS signal to the reference point can be kept constant. That is; it is possible to set the reference point position at an arbitrary constant distance from the zero-point LS signal, disregarding the phase-C pulse position.

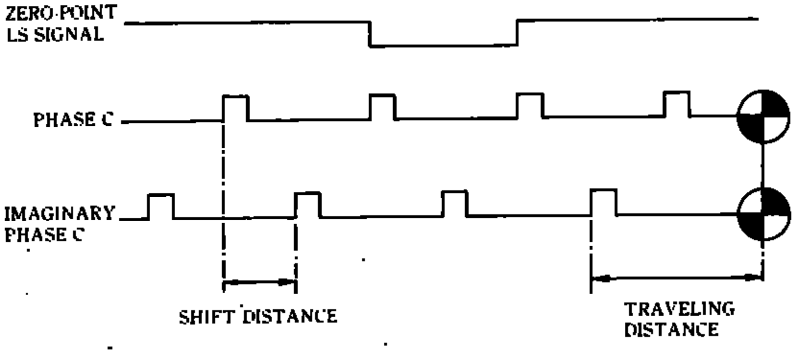


Fig. 3.6.5 Imaginary Phase-C Shift Distance

(3) Related parameters

pm1821 to 1825 : Moving amount per motor revolution (Axes X to 5th)
Setting unit : 1 = Least detection unit
Setting range : 0 to 999999999

pm2521 to 2525 : Reference point return approach speed (Axes X to 5th)
Setting unit : 1 = 1 mm/min = 1 deg/min
Setting range : 0 to 32767

pm2531 to 2535 : Reference point return creep speed (Axes X to 5th)
Setting unit : 1 = 1 mm/min = 1 deg/min
Setting range : 0 to 32767

pm4002 D0 to D4 : Reference point return direction (Axes X to 5th)
0 : +direction, 1 : −direction

pm4003 D6 : Manual reference point return
0 : Manual reference point return high-speed type from the second time and after
1 : Manual reference point return low-speed type from the second time and after

pm4003 D7 : Automatic reference point return
0 : Automatic reference point return high-speed type from the second time and after
1 : Automatic reference point return low-speed type from the second time and after

pm4551 to 4555 : Imaginary phase-C shift distance (Axes X to 5th)
Setting unit : 1 = 0.001mm/0.001deg
Setting range : −32767 to +32767

pm4451 to 4455 : Reference point return traveling distance (Axes X to 5th)
Setting unit : 1 = 0.001mm/0.001deg
Setting range : −32767 to +32767

NOTE The above-mentioned pm4551 to 4555 are newly added parameters following the addition of functions.
Since the above parameters need resetting, do not fail to perform reset operation when the setting is changed.

(4) How to set imaginary phase-C shiting values

(i) When the machine zero-point and dceleration limit switch are at arbitrary positions.

① Set the parameter. As temporary setting, set the reference point return traveling distance (pm4451 to 4455) to 0 and the imaginary phase-C shift distance (pm4551 to 4555) to 1000.

② Execute reference point return.

③ When the tool stops before the machine zero-point, the distance from the stopped position to the machine zero-point is added to the imaginary phase-C shift distance.

When it stops after the machine zero-point, the distance from the stopped position to the machine zero-point is subtracted from the imaginary phase-C shift distance.

④ Inching adjustment to the actual machine zero-point is possible at the imaginary phase-C shit distance or reference point return traveling distance.

(ii) When the machine zero-point and deceleration limit switch are at a distance of 1/2 of the ball screw pitch

- Perform adjustment assuming that the machine zero-point described above (c) is at a distance of 1/2 of the ball screw pitch. Inching adjustment to the actual machine zero-point is possible at the reference point return traveling distance.

NOTE

1. This function is optional. Therefore, it is not effective if the option is not set.
2. When the reference point return is executed by selecting this function, do not fail to retract the deceleration limit switch sufficiently. If the distance beyond the deceleration limit switch is too short, an alarm (2061 to 2065: reference point return area error) occurs.
3. If the imaginary phase-C shift distance (pm4551 to 4555) or reference point return traveling distance (pm4451 to 4455) set value is not proper, retraction is generated and an alarm (2091 to 2095: reference point return parameter error) occurs.

   In this case, adjust the parameter by increasing the reference point return traveling distance set value, etc.

   (Example 1) When the distance from the deceleration limit switch to the imaginary phase-C pulse becomes larger by increasing the imaginary phase-C shift distance.

   (Example 2) When the final traveling distance set value is minus and exceeds the distance from the deceleration limit switch to the imaginary phase-C pulse.
4. When the deceleration limit switch is turned OFF and then ON at reference point return before completion of phase-C latch, alams 2151 to 2155: reference point return phase-C creation error(axes X to 5th) occur.

3.6.1.3 2nd manual reference point return ✱

This is a function in the MANUAL mode to automatically position the machine in its 2nd reference point. Positioning is possible regardless of which side of the 2nd reference point the current position is located.

(1) Description of the function

(a) If ZRN2 (2nd reference point return request) and + X (or + Y, + Z) are set to ON in the JOG or RAPID mode, the X-axis (or Y- or Z-axis) will be positioned at the 2nd reference point. The moving speed of positioning will be of JOG or RAPID speed.

(b) If ZRN2 is set to OFF (= 0) when moving to the 2nd reference point, movement will stop. To restart 2nd reference point return, set ZRN2 to ON and depress the + or − direction input of manual feed.

(c) Although movement will also stop if + X (or + Y, + Z) is set to OFF when moving to the 2nd reference point, movement will also start if + X (or + Y, + Z) is set to ON (= 1) again.

NOTE

1. Input of 2nd reference point return mode will be invalid if still in 1st reference point unreturned state.
2. In the 2nd reference point return mode, input of − X (or − Y, − Z) will also be invalid. The mode switching is regarded as ZRN2 input OFF.
3. The 1st reference point return mode and 2nd reference point return mode cannot be used simultaneously. When both inputs are ON at the same time, both modes will become invalid and, from a safety standpoint, JOG feed and rapid feed will not operate.
4. The − X (or = Y, − Z) input is the same as the + X (or + Y, + Z) input and is valid as the 2nd reference point return input.
5. In states such as machine lock (including machine lock by axis) and Z − axis command cancel, it becomes normal JOG feed operation
6. instead of 2nd reference point return.
Movement by manual operation is not possible in relation to an axis on which manual 2nd reference point return was completed unless the "2nd
7. Reference Point Return" switch is set to OFF.
Reverse setup is used on the axis on which manual 2nd reference point return is complete.
8. If the + and − direction manual feed inputs are set to ON at the same time, it will be considered as OFF and the machine will stop.

3.6.2 Coordinate System Setting Operation

3.6.2.1 Automatic coordinate system setting ※

Automatic coordinate setting will be possible at the point where manual reference point return operation is completed. The coordinate values to be set are initially set by parameters. The coordinate system created by this operation is equal to the coordinate system created with G92.

(1) Parameter setting values when an mm input system

Table 3.6.1 Parameter Setting Value (mm Input System)

Parameter	Meaning
Pm4801	X-axis coordinate
Pm4802	Y-axis coordinate
Pm4803	Z-axis coordinate
Pm4804	4th axis coordinate
Pm4805	5th axis coordinate

(2) Parameter setting values when an inch input system

Table 3.6.2 Parameter Setting Value (inch Input System)

Parameter	Meaning
Pm4811	X-axis coordinate
Pm4812	Y-axis coordinate
Pm4813	Z-axis coordinate
Pm4814	4th axis coordinate
Pm4815	5th axis coordinate

(3) Selection of valid axes with parameters (Pm4006 D0 to D4) is possible for both (1) and (2) above.

3.6.3 Handle Operation

3.6.3.1 Simultaneous 2nd and 3rd handle axis feed ※

Simultaneous handle control of up to a total of three of any of the axes (X, Y, Z, α or β) is possible by connecting a manual pulse generator to each axis.

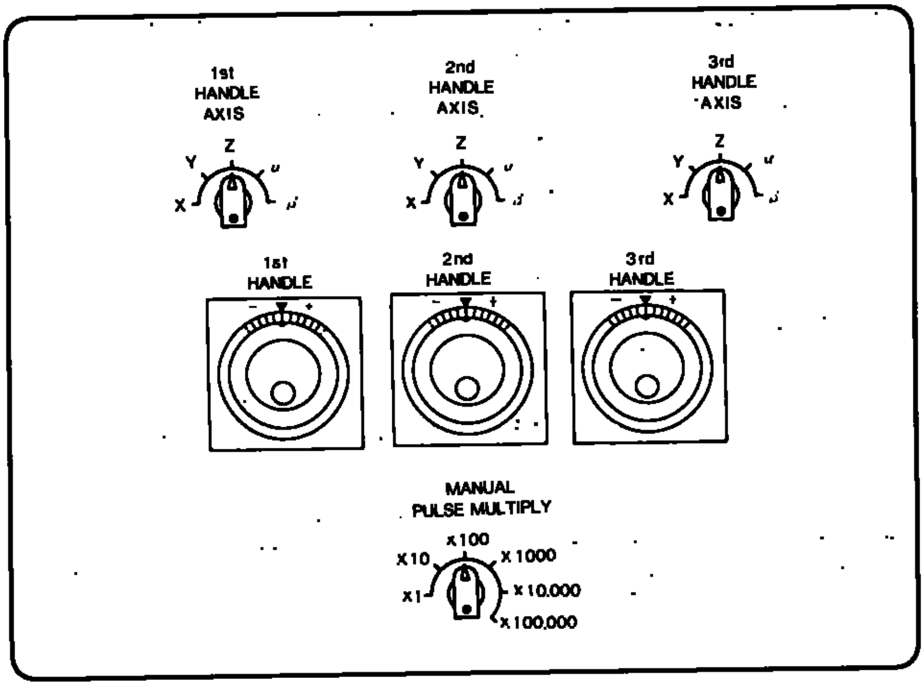


Fig. 3.6.6

(1) Movement per handle graduation complies with the previously described "Manual Feed Pulse Multiple Switch" (Table 3.5.1). Only one of these switches is provided in common with three axes.

(2) Select the HANDLE mode and turning the handle moves the axis forward or in reverse.

3.6.3.2 AUTO mode handle offset ✻

This is a function that evaluates and weighs movement by manual pulse generator to movement during automatic operation (TAPE,MDI,MEM) This function enables offset of shift caused by mounting the workpiece. The operating procedure is as follows

(1) Set the AUTO MODE HANDLE OFFSET switch to ON.

(2) Select the axis to be moved with the HANDLE AXIS switch.

NOTE Coaxial 3-axis movement is possible when a "coaxiai 3-axis control manual pulse generator" is added.

(3) Select movement per handle graduation by means of the MANUAL PULSE MULTIPLY switch. The amount of movement per graduation can be switched to 1, 10, or100 pulses.

(4) If the HANDLE switch is turned during AUTO operation, the amount of movement with the handle will be that with the movement of AUTO operation added in relation to the axis selected in (2) above.

Clockwise (CW)	Positive direction
Counterclockwise (CCW)	Negative direction

(5) Set the AUTO MODE HANDLE OFFSET switch to OFF.

(6) Subsequent movement will be in shifts equal to the handle offset.

NOTE Setup with command value only will be conducted without the handle offset being added later in relation to coordinate setup commands (G92, etc.).

NOTE

1. Movement by AUTO MODE HANDLE OFFSET is not possible in an alarm state.
2. Movement by AUTO MODE HANDLE OFFSET is not possible with the axis interlock input (IT) in ON state.
3. Various AUTO MODE HANDLE OFFSET movements can be invalidated by parameter setting.

Table 3.6.3

Axis	Parameter		Valid/Invalid
	No.	Setting	
X-axis	Pm2002 D0 (HOFSX)	1 0	Valid Invalid
Y-axis	Pm2002 D1 (HOFSY)	1 0	Valid Invalid
Z-axis	Pm2002 D2 (HOFSZ)	1 0	Valid Invalid
4th axis	Pm2002 D3 (HOFS4)	1 0	Valid Invalid
5th axis	Pm2002 D4 (HOFS5)	1 0	Valid Invalid

4. The condition for movement by the AUTO MODE HANDLE OFFSET can be set by the parameter Pm2003 D1.

Pm2003D1 = 0	Movement is possible during rapid feedrate and interpolation.
Pm2003D1 = 1	Movement is possible only during interpolation.

5. The MANUAL ABSOLUTE function is invalid in this function.

3.6.4 Spindle Indexing Function ※

Spindle indexing is a function that stops the spindle at optional positions (position having optional rotating angle).

(1) Conditions for adding spindle indexing function

In addition to understanding this function, the following conditions must be satisfied to use the spindle indexing function.

① Must be equipped with S5 digit analog output options.

② The YASKAWA VS − 626MT III or VS − 626VM3 must be mounted on the spindle drive.

③ A PG must be mounted on the spindle or spindle motor and the gear ratio used must be fixed at a ratio of 1 : 1 (Ideal performance can be displayed if the spindle and motor are directly connected).

④ There must be 4096 feed back pulses per rotation of the spindle.

⑤ Solid tap function option must be valid.

(2) Spindle indexing procedure

In addition to arranging the hardware resoures, control of the I/O signal with a sequencer is important in using the spindle indexing function. Refer to the instruction manual issued by the machine tool builder for detailed operating methods and precautions since only general operating methods will be introduced here.

① M code commands specified by the machine tool builder are issued at the point where spindle indexing is desired in the part program.

② Execute this part program. When the spindle indexing operation M code is executed, spindle indexing will end.

③ Execution of the part program will continue thereafter.

NOTE

Spindle indexing operation cannot be executed in the SOLID TAP mode. Execute spindle indexing after cancelling the SOLID TAP mode.

3.6.5 Return to Suspended Operation Point ※

This function is used to return the system to the position where auto operation was suspended after manually moving the tool for measuring the workpiece or removing swarf.

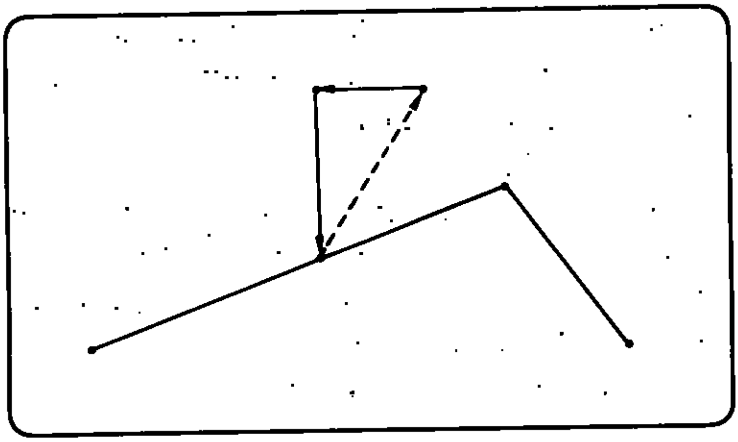


Fig. 3.6.7 Return to Suspended Operation Point

The operation procedure for returning to the processing-suspended operation point is explained in the following :

(1) To stop auto operation, turn on the single-block switch or depress the temporary stop (feed hold) push-button.
(2) Enter the manual (JOG, RAPID, HANDLE, or STEP) mode.
(3) Move the axis in the manual mode to retract the tool from the workpiece.
(4) Perform necessary operation such as workpiece measurement or removal of swarf.
(5) Turn on the return-to-suspended operation point switch.
(6) Set the manual feed selection push-button in the direction toward the suspended operation point (where the auto mode is changed to the manual mode). The tool moves at a selected speed toward the suspended operation point and automatically stops at the point.
(7) Turn off the return-to-suspended operation point switch.
(8) Return to the auto mode and depress the start pushbutton switch to start auto operation.

NOTE

1. Manual jog feed for returning-to-suspended operation point Manual jog feed cannot be used to return to the processing-suspended operation point unless the return-to-suspended operation point switch is turned OFF.
2. Position of suspended operation point after changing from auto mode to manual mode and resetting.

   If the system is reset after changing from auto mode to manual mode, the tool returns to the suspended operation point. While the tool is on the way toward the suspended operation point, manual jog feed cannot be used unless the return-to-suspended operation point, switch is turned off, as explained in NOTE 1.
3. Suspended Operation point

   After a series of moves in successive changes from the auto to manual mode and vice versa, the last point where the auto mode is changed to the manual mode is used for the suspended operation point.
4. Return-to-suspended operation point

   If the home positioning switch or the second home positioning switch is ON, return-to-suspended operation point is not performed. Instead, the tool returns to the home position or the second home position.

3.6.6 Tool Length Measurement ✲

This function measures tool length offset values for the length offset commands (G43, G44, and G49.)

(1) Outline

Mount the tool on the spindle. Contact the tip of the tool to the Z-axis contact level by manual operation. Depress the WRITE & RETRACT return button switch. The system performs as follows :

(a) Stores the length from the set Z-axis home position to the reference level to the currently specified offset number memory. Difference between the contact level and the reference level can be set by a parameter.

(b) Add "I" to the offset number to get ready for the next Write-in.

(c) Increments the tool to the Z-axis home position.

(2) Operations

(a) Measurement of length of travel (parameter pm6008 D0=0)

[1] Mount the tool on the spindle and position to the point to be the home position. The home position can be set at any position. For easy replacement of tools, it is convenient to set the tool replacement position for the home position.

[2] Set the MODE select switch to manual (RAPID, JOG, HANDLE, or STEP.)

[3] Depress PREPARATION key to enter the preparation process.

[4] Select TOOL job.
This enters the tool offset value display mode, where offset numbers that are already specified and data near the number are displayed.

[5] Depress the MEASUREMENT push-button switch.
The MEASUREMENT lamp lights and the current position of the Z-axis is stored as the home position.
(This push-button switch is effective only in the manual mode. Confirm that the system stops before depressing the switch.)
The preparation process is automatically selected and the preparation process screen appears.

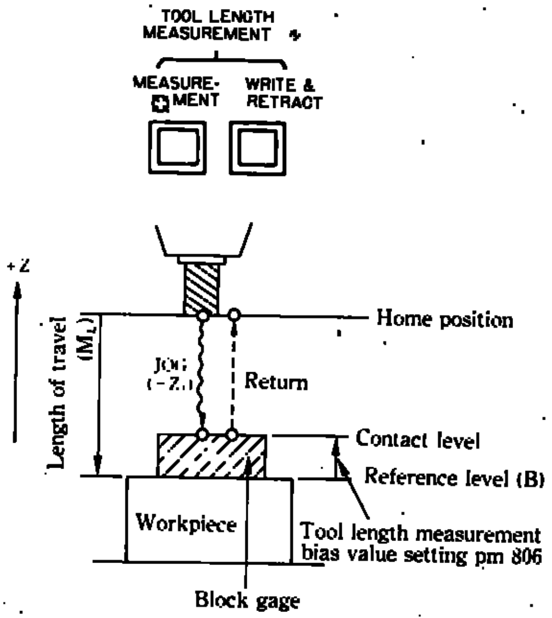


Fig. 3.6.8 Measurement of Tool Length Offset Value

Measurement is started at home position. The position is assumed to be the zero point.

NOTE
1. To assume this position as the zero point of the workpiece coordinate system, perform workpiece coordinate system zero setting. Measurement is normally made without the setting.
2. For details of the tool length measurement screen, refer to Par 3.4.6.2, "Tool job", (4) tool length measurement function.

[6] Use the page keys and cursor keys to move the cursor to the tool offset number where data are to be written. Otherwise, enter the tool offset number and depress the ▼ key.

The page of the input offset value is displayed and the cursor is at the input offset number.

[7] Align the tool tip to the contact level by Z-axis manual operation. The contact level can be set at an arbitrary distance from the reference level (workpiece processing surface). The difference between the levels is the tool length measurement bias, which must be preset in Pm806 (input unit is 1.) The value represents the thickness of the block gage, and must be a positive value.

The bias can also be set by the following procedure on the tool length measurement screen :

① Depress the BIAS function key. "BIAS (B)" is highlighted in reverse display.

② Enter the bias value, then depress the WRITE key. The bias value is written.

[8] Depress the WRITE & RETRACT push-button switch. The system performs as follows :

① The distance from the home position to the reference level, that is, the measured length of travel (M_L) is stored at the specified offset number. $(M_L) = Z-B$.

② At the same time, the Z-axis automatically returns to the home position in rapid traverse.

③ After the above operations, a new offset number that is greater than the specified one by 1 is prepared for the next write of a tool offset value. (After H99, H01 is used.)

[9] Replace the tool by manual or MDI operation. The MEASUREMENT lamp remains on even after change to the MDI mode. Return to the manual mode after tool replacement.

[10] Repeat steps [7] to [9] to save all offset values for necessary tools.

[11] Depress the RETURN key. The offset screen appears.

NOTE This step can be omitted.

[12] Depress the MEASUREMENT push-button switch.

The MEASUREMENT lamp goes off and the auto write function is canceled. If the tool length measurement screen has been on, it changes to the offset screen.

(b) Measurement of remainder (parameter Pm6008 D0 = 1)

If parameter Pm6008 D0 = 1 is set, remainder (R_L) is stored in the tool offset memory instead of length of travel (M_L).

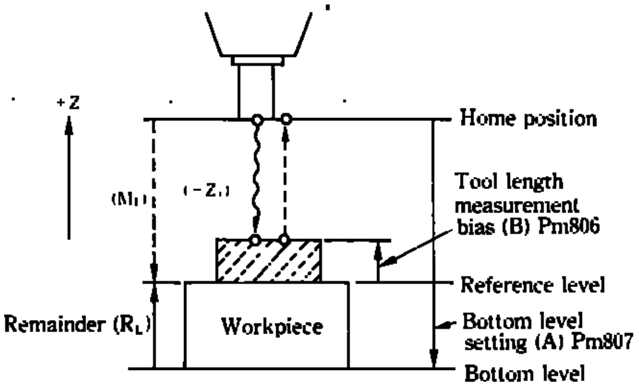


Fig. 3.6.9 Measurement of Remainder

The bottom level to be used as the reference for measurement of remainder is a signed value of a distance from the home position and can be set in Pm807 (bottom level setting; input unit is 1.)

The bottom level can also be set by the following procedure on the tool length measurement screen :

① Depress the BTLEVL key. "BOTTOM LEVEL(A)" display is highlighted in reverse display.

② Enter the bottom level value, then depress the WR key. The bias bottom level value is written.

After this setting, the remainder value can be written by the same procedure as (a).

$R_L = -(A - M_L)$

(3) Length measurement on NC operating panel

(a) Measurement of lengths can be made on the NC operating panel without using specialized push-button switches such as MEASUREMENT.

(b) To accomplish this, do as follows instead of depressing the MEASUREMENT push-button switch.

[1] Depress COMN , POS , and UNIV .

[2] Clear the coordinate system set value to zero by depressing the SET and WR . This means that the home position is set at 0.

[3] Depress SET , OFFSET , 〈〉 , and TLM keys.

(C) Substitute operation for the WRITE & RETRACT ☐ push-button switch. Depress WRITE. (This soft key is not displayed in the TLM mode.)

This operation performs auto write of tool measurement values similar to normal tool length measurement, except that the Z-axis is not returned to the home position.

(d) Setting of parameter Pm6008 D0 (selection of measurement of length of travel/remainder)is valid for this measurement operation. Which value is to be stored is determined by the parameter.

NOTE

1. In this auto write mode, the measured value is stored as an absolute value into memory.
2. If the home position is not the tool replacement position, the tool can be moved from the tool replacement position to the contact level directly after replacement. Once the home position is determined by depressing the MEASUREMENT push-button switch, positioning to the home position is not unnecessary.
3. If H00 is specified, depressing the WRITE & RETRACT push-button switch makes no change of offset values. Move the cursor to H01 or after to perform TLM operations.
4. To rewrite an offset value on the tool length measurement screen, set the cursor to the value, enter a new numeric value, then depress WRITE.

3.6.7 Manual Skip ※

Make contact with the touch sensor by manual operation to read the coordinates and the direction of contact of the touch sensor. This function is manual skip.

The manual skip function implements measurement with human interface (such as ACGC) software.

There are two types of manual skip which can be selected by parameter setting.

Pm5010 D7 = 0 : Manual skip A

Pm5010 D7 = 1 : Manual skip B

Manual skip A uses I/O input for touch sensor signal, while manual skip B uses direct-in signal.

(1) Operation

[1] Enter the manual mode.

[2] Return to the home position manually.

NOTE This step can be omitted if the axes once returned to their home positions after power was turned ON.

[3] Turn on the manual skip entry signal (# 3009D6) by PLC. At the rising of this signal, NC enters the manual skip mode.

[4] After NC enters the manual skip mode, the manual skip mode on signal is output to PLC.

[5] Move the axis in the JOG or HANDLE mode to contact the touch sensor to an arbitrary point.

NOTE When the tool touches the sensor, feed is immediately stopped. At this time, NC automatically detects the machine position values and the direction of contact, and writes them into memory.

[6] After measurement, the PLC turns OFF the manual skip entry signal (# 3009D6). At the falling of this signal, NC cancels the manual skip mode.

[7] Output of the manual skip mode ON signal to PLC turned OFF.

(2) Related parameters and I/O signals

(a) Parameters

- Position data parameters

Parameter value 1 represents 0.001 mm.

Pm920 : X-axis first point position data
Pm921 : Y-axis first point position data
Pm922 : Z-axis first point position data
Pm923 : X-axis second point position data
Pm924 : Y-axis second point position data
Pm925 to pm935 : Z-axis second point position data to X-axis sixth point position data
Pm936 : Y-axis sixth point position data
Pm937 : Z-axis sixth point position data

- Contact direction parameters

Pm420 : First point contact direction
Pm421 : Second point contact direction
Pm422 : Third point contact direction
Pm423 : Fourth point contact direction
Pm424 : Fifth point contact direction
Pm425 : Sixth point contact direction

NOTE : The direction of contact is indicated by bit data as follows :

d0 ON : X-axis positive direction
d1 ON : Y-axis positive direction
d2 ON : Z-axis positive direction
d8 ON : X-axis negative direction
d9 ON : Y-axis negative direction
d10 ON : Z-axis negative direction

- Measurement point monitor parameter

Pm107 : Indicates the number of points of which data are currently held in memory. (The initial value is 0. The value is incremented by 1 each time measurement is performed.)

- Measurable retraction length

Pm6847 : Determines the length of retraction between two successive measurement operations.

Parameter value 1 represents 0.001 mm.

(b) Input signals

- Manual skip entry signal (#3009D6 : MSKP)
- Touch sensor signal (#3009D7 : SPST)···When manual skip A is used.

(C) Output signal

- Manual skip mode (#3507D7 : MSKP0)

NOTE Notes on manual skip function

1. To move axes to the contact point, move a single axis at one time (simultaneous 1-axis operation.)
2. The skip contact position data are canceled only by an external instruction such as the manual skip mode entry signal or a request to reset. The data are retained unless change is commanded externally.
3. The manual skip function is ineffective for the 4th and 5th axes.
4. If the manual skip entry signal is input out of the manual mode, the signal is disregarded and the manual skip mode ON signal will not be output.
5. If simultaneous return of all axes to home position has not been performed after power was turned on before the rising of the touch sensor signal is detected in the manual skip mode, the touch sensor signal is disregarded and no measurement will be made.
6. Axes do not move in the direction of the contact with the touch sensor.
7. After contacting the touch sensor, the tool cannot be moved in the direction of contact with the touch sensor without a certain length (determined by Pm6847) of retraction.
   Retraction is not needed when Pm6847=0.

3.6.8 Manual Centering ※

This function determines the coordinates of the center of given two points by manual operation. The function can be used to locate the center of a circle.

(1) Operation

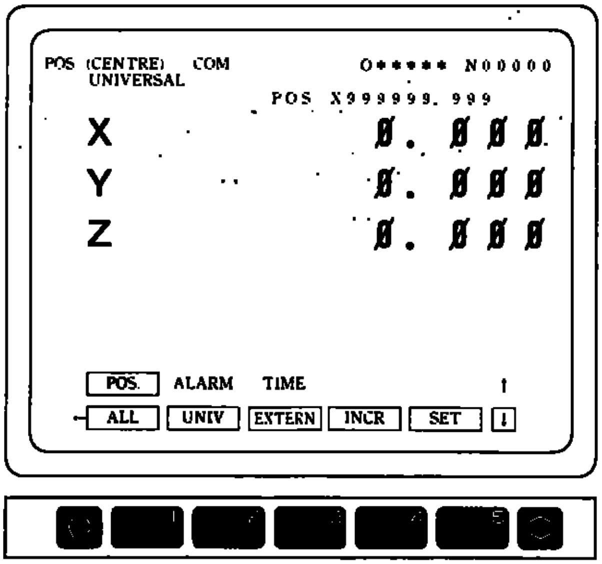
[1] Enter the manual mode.

[2] POS → UNIV / EXTERN Depress appropriate key.

[3] Turn on the manual centering mode signal (#3009D5). At the same time, the manual skip mode signal (#3009D6) also goes on.

[4] Specify the centering axis.

A coordinate can also be specified as shown in the figure. The data are stored temporarily and displayed on the screen.



```
POS (CENTRE)   COM                O****** N00000
    UNIVERSAL
                         POS X999999.999
    X                              0.000
    Y                              0.000
    Z                              0.000
```

Fig. 3.6.10 Centering Axis Selection Screen

[5] Move the X-axis so that the sensor touches the circumference. When the sensor outputs a signal, the axis is stopped and the data stored in [4] are set up for position data of the contact point (similar to coordinate system setting.) The axis cannot be moved further in the contacting direction unless the axis is retracted for the specified length (by Pm6847.)

[6] Retract the axis from the contacting surface, then place the sensor on the other side of the circumference. When the sensor outputs a signal, the axis is stopped and the current value is set up for position data of the center of the two points where the sensor has touched.

[7] Repeat steps [4] to [6] for the Y-axis.

[8] Turn off the manual centering mode signal (#3009D5). At the same time, the manual skip mode signal (#3009D6) also goes OFF.

NOTE This function is available only when manual skip takes effect.

3.6.9 Playback ⁕

Current positions of the axes can be used for axes commands, and arbitrary values can be used for F, S, and T commands by editing the part program. That is, an axis can be moved to a necessary position and the position can be used in an axis command. For an F, S, or T command, a value that is currently used or a new input value can be used.

The values can be written by entry from the keyboard or by external signals.

(1) Operation

(a) Entering playback mode

Depress the RUN, PROG, and EDIT function soft keys to enter the edit mode. Then turn on the PLAY BACK ◉ OFF switch.

(b) Writing by key entry

[1] Writing axis data

① Move the axis to the position to be used in the part program by an appropriate method of moving.

② Depress the axis key of the position which is to be written (by the address keys.) then depress the WR key. The current value (workpiece coordinate) being displayed is set up for the axis command. Entering a value from the keyboard before depressing the WR key adds the value to the current position data and stores the sum for the command value.

NOTE Each time the WR key is depressed, the current value is added.

(Examples)

1. If the current value of the X-axis (in the workpiece coordinate system) is 1.000, depressing the X and WR keys stores X1. for the command value.
2. Entering X 2 . 0 0 0 WR adds 2.000 to the current value.
3. Entering X 2 . 0 0 0 WR WR adds a current value of 1.000 twice to the input value of 2.000, resulting in X4. for the command value.

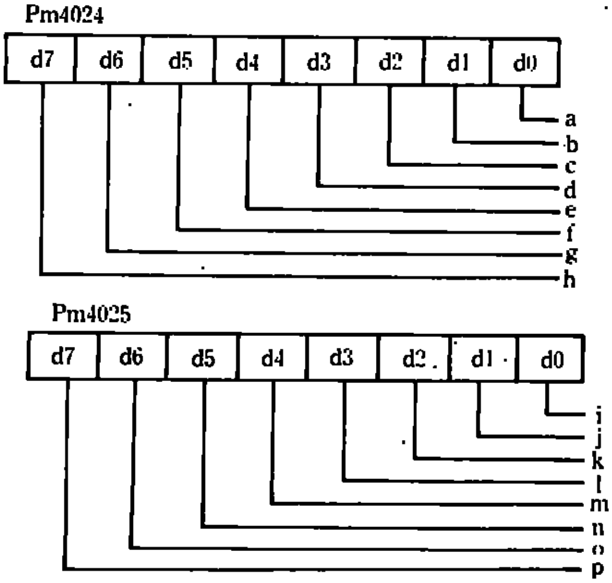
[2] Writing F, S, and T data

F, S, and T data can be written similar to axis data, except for the following:

- A value entered before the WRITE key is depressed is used for the command value.
- The currently used F and S values are written for the F and S data.

(c) Writing by external write signal

(i) When the external write signal is turned on, write of the axis and cutting feed specified by parameters Pm4024 and Pm4025 is enabled.



a : 1st axis write enabled
b : 2nd axis write enabled
c : 3nd axis write enabled
d : 4nd axis write enabled
e : 5nd axis write enabled
f : 6nd axis write enabled
g : 7nd axis write enabled
h : 8nd axis write enabled
i : 9nd axis write enabled
j : Not used
k : Not used
l : Not used
m : Not used
n : F write enabled
o : S write enabled
p : T write enabled

(ii) New data are compared to the preceding data, and only the changed data are written.

(d) Storing edited program to memory

After editing, depress the EDIT function soft key to store the edited part program.

(e) Supplementary

(i) Data other than axis data and F, S, and T data can be edited by normal editing operation.

(ii) Current values written by the WR key are the workpiece coordinates. Therefore, the origin of processing must be set up by G92 or other comniand before setting playback data.

NOTE : Editing can be performed only when the EDIT function key is highlighted in reverse display.

(2) I/O signals

Playback function ON/OFF

#30087

–Playback can be turned ON or OFF without this signal by setting Pm0005 D5 ON or OFF.

–External write signal

#30086

NOTE

1. Out of the edit status (when the EDIT function key is not in reverse display.) the playback mode is off.
2. If the WR key is depressed before the data address (axis address or F, S, or T) is specified, a warning message of "No address specified" appears. Specify the address before depressing the WR key.
3. If the WR key is depressed or the external write signal is turned on while the axis is moving, a warning message of "Axis moving" appears. Writing can be performed after the axis stops.
4. For the F, S, or T command, the following values are used in playback operation :

F : Current command F
S : Current command S
T : Current command tool

3.6.10 Program Restart ✱

Processing can be restarted in the middle of a program by specifying the sequence number of the block. Use this function after terminating auto operation because of breakage of a tool or to continue processing from the end of the previous day's operation.

(1) Operation

[1] Turn on the PROGRAM RETURN OFF switch (#30063) on the machine operating panel.

[2] Find the beginning of the program to be restarted in the MEM mode (operation process program job.)

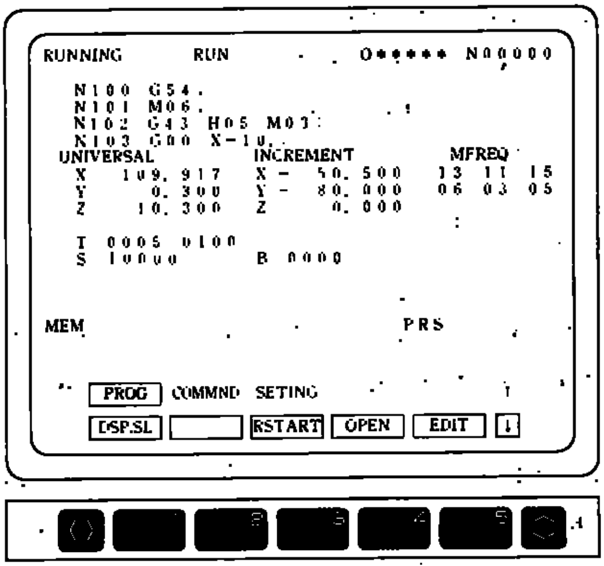
[3] Depress the RSTART function soft key on the operation process program job screen and executes the program restart request.
When the key is depressed, the key entry request message appears for the restart position.

(1) A message of "Enter sequence number" appears. Enter the sequence number to be searched for, then depress the WR key.

(2) A message of "Enter the occurrence number" appears. If the same sequence number occurs more than once, enter the position of the object once, then depress the WR key. If specification of the position is not necessary, depress the WR key only.

[4] At the second depressing of the WR key, the specified sequence number is searched for from the current cursor position. During the search, "PREPARING FOR RESTART" is displayed on the screen. Also, the program restarting PRS blinks.
PRS disappears when the processing restart position is found.

[5] After the search, the screen displays program restart data :



```
RUNNING        RUN              O****** N00000

  N100 G54.
  N101 M06.
  N102 G43 H05 M03;
  N103 G00 X-10.
 UNIVERSAL           INCREMENT           MFREQ
  X  109.917         X -  50.500         13 11 15
  Y    0.300         Y -  80.000         06 03 05
  Z   10.300         Z     0.000

  T 0005 0100
  S 10000            B 0000

MEM                                  PRS
```

Fig. 3.6.11 Example of Program Restart Data Screen

Workpiece coordinate system : Indicates the processing restart position.

Remaining length of travel : Indicates the distance from the current tool position to the processing restart position.

M code	: Displays all M codes that occur from the sequence number search start block to the restart block. If there are more than 24 command M codes, the last 24 are displayed.
MCOUNT	: Indicates the number of command M codes.
T code	: Displays up to two latest command T codes in the restart block.
S code	: Displays the last command S code before the restart block.
B code	: Displays the last command B code before the restart block (only when the second auxiliary function option is effective.)

The M and T codes are displayed in the order of their appearance.

[6] If the found sequence number is for the object block, turn on the PROGRAM RETURN OFF switch. If it is not, request to restart the program again.

[7] Specify necessary M, S, T, and B codes as explained in the following to restart processing on the program restart data screen :

(1) Select the manual mode.

(2) Enter necessary M, S, T, or B code, then depress the CYCLE START push-button switch to start. (Execute M, S, T, or B by the 1-line MDI function.)

(3) Return to the memory mode.

[8] Depress the CYCLE START push-button switch to restart processing.

With cycle start, the 5th, 4th, X, Y, and Z axes move to the processing restart position one by one in this order. Then automatic operation is restarted from the next block after the specified sequence number.

(2) Notes on operation

(i) While "Preparing to restart" is displayed, all keys except the reset key are ineffective. In the meantime, the G/M code display of the operation process program job is not changed.

(ii) After "Preparing to restart" disappears, display on the lower half of the screen can be changed from G/M code data to sub-program nesting data to program restart data, using the page keys. The pages can also be changed in reverse order.

(iii) While PRS is displayed, memory programs cannot be called nor edited.

(iv) The RSTART soft key is displayed only when the PROGRAM RETURN switch is on and the label is skipped.

(v) Program restart request can be executed without the PROGRAM RETURN switch.

Pm4015 D0 = 0 : Use PROGRAM RETURN switch.
Pm4015 D0 = 1 : Do not use PROGRAM RETURN switch.

When the parameter is set for "Do not use PROGRAM RETURN switch", the program restart mode is turned on or off by the RSTART soft key. Under this condition, steps [1] and [6] in(1) above are not necessary.

The RSTART soft key is displayed only when the label is skipped.

(3) Remarks on program restart and operations of related functions

In order to restore the processing conditions of suspended operation to restart, options are prepared for the search for a specified sequence number.

Nevertheless, some of the program commands require extra care. The details are explained in the following :

(a) The program restart position is generated in NC. Therefore, axes are regarded as moving according to the NC program during search for the restart position.

Automatic reference point return (G28), reference point return check (G27), return from reference point return (G29), second to fourth reference point return (G30), machine coordinate system (G53).

(b). There are commands that perform different operations depending on input signals. Some such input signals retain the status they had when the program restart search was started. Others are completely disregarded.

(i) Signals that retain status at search start Mirror image (M95, M94,) block delete, machine lock, manual absolute, F1-digit selection

(ii) Signals that are disregarded

Feed hold, single-block, auxiliary function lock, skip, program interrupt (M91, M90,) optional stop

The skip signal is assumed being input.

(c) Some commands cannot be executed as programmed because the special operation mode is impossible.

(i) Solid tap (M93)

When restarted, the G74 and G84 commands in the solid tap mode (M93) are replaced with normal tapping.

(ii) High-speed cutting commands (HON, HOF)

When restarted, HON and HOF commands in the HON mode are replaced with normal text operations.

(iii) F1-digit command

Processing can be restarted at the speed according to the F1-digit command, except that acceleration and deceleration of handle operation requires new F1-digit commands.

(iv) Tool life control :

Before restarting operation in the tool life control mode, the life control group must be specified.

The group is indicated by command T code in the program restart data.

(d) Macro programs require extra care :

(i) System variables .

- Interface I/O signals (#1000 to #1032, #1100 to #1132)

All of the above signals are assumed to be 0, consequently the result will be marred.

- Clock (#3001, #3002)

Clock signals are read and written normally, but the time count cannot be guaranteed in restarted processing.

- Control of single-block stop, wait for auxiliary function completion (#3003)

The signal is disregarded and specification of control becomes ineffective.

- Control of feed hold, feed rate override, exact stop (#3004)

The signal is disregarded and specification of control becomes ineffective.

- Position data (#5000 and above)

Position data are read and written normally, but the actual data used are generated in the NC system.

For the servo position deviation(#5101 to #5105),0 is read.

(ii) RS-232C data output 2 (BPRNT, DPRNT)

If processing is restarted after the RS-232C open command, an alarm occurs at the RS-232C output command.

(4) Program restart related alarms

(a) Alarms 411 to 415 "Reference point return not completed"

Restart was requested without return to the reference point after power was turned ON.

(b) Alarm 290 "Program restart sequence number not found"

The specified sequence number in the request to start could not be found.

(C) Alarm 291 "Coordinate system shifted after program restart"

After searching, a command that changes the coordinate system for program restart was entered.

(d) Alarm 292 "Impossible to start the command when restarting program"

Before restarting memory operation, the 

push-button switch was depressed in the MDI or TAPE mode.

NOTE

1. This function is an optional function. If the option is unavailable, the function cannot be used.
2. After depressing the CYCLE START switch, if the workpiece or other object obstructs the axes moving to the processing restart position, remove the obstacle by hand.
3. If the CYCLE START push-button switch is depressed while the PROGRAM RETURN OFF switch (#30063) is ON, a warning message of "Program being restarted" is issued.
4. After program restart search, intervening operations are disabled until processing is restarted. If the CYCLE START push-button switch is depressed in the MDI or TAPE mode, alarm 292 "Impossible to start the command when restarting program" is issued.
5. Axes cannot be moved in the manual mode when the PROGRAM RETURN switch is on. The axis switches and handle pulse input signals are disregarded.
6. When requesting program restart, the conditions of suspended processing, such as input signals and the manual mode, must be restored. Otherwise, processing restart position and conditions will be changed.

   For manual operation during restarting a program, turn on the MANUAL ABSOLUTE switch.
7. The FEED HOLD push-button switch and the SINGLE BLOCK switch are disregarded while searching.

   If the system is reset, search is stopped immediately and the program restart request is canceled. Request to restart if necessary.
8. The coordinate system must be set properly before using the program restart function. If restart is requested without reference point return after power is turned on, alarms 411 to 415, "Reference point return not completed" are issued.

   Proper restart position cannot be determined for a program that contains incremental commands without prior coordinate system setting.
9. If the sequence number specified in the restart request was not found, alarm 290, "Program restart sequence number not found" is issued.
10. When reference point return is performed after switching into manual operation mode, or when manual coordinate system setting is performed, alarm 291 "Coordinate system shifted after program restart" is issued.
11. Program cannot be restarted at DNC operation, tape operation or high-speed cutting operation.

3.6.11 FS Automatic Edit Function※

The actual value of feedrate (F) and spindle rotation speed (S) are automatically taken and edited into the program of NC memory, to provide the optimum F and S values to the program.

The machining program is prepared with the estimated values of F and S. Then, an actual cutting is performed with this program to make the machining more accurate using feed override and spindle override.

The optimum data and its corresponding block on the program are stored in the prgram by closing the FS memory change signal.

(1) I/O signals

The following I/O signals are used in FS automatic edit function.

(a) Input signals

- FSED M : FS edit mode

  I/O #30114

  By closing this signal, the preparation for storing and editing of FS actual values is performed.

- FSED MEN : FS memory

  I/O #30115

  By closing this signal during memory operation, F and S values and their corresponding block positions are entered in NC.

- FSED CH : FS memory change

  I/O #30116

  By closing this signal during interruption of memory operation, the entered F and S values are reflected in the program.

- FSED CLR : FS data clear

  I/O #30117

  By closing this signal, the F and S values stored in NC and cleared.

(b) Output Signals

- FSED S : In FS edit mode

  I/O #35040

  This signal is closed when the FSED S is closed and FS memory and FS memory edit are in enable status.

- FSED CE : FS memory change completed

I/O #35041

This signal is closed when the FSED CH is closed and the F and S values are entered in NC program, and opened when the FSED CH is opened again.

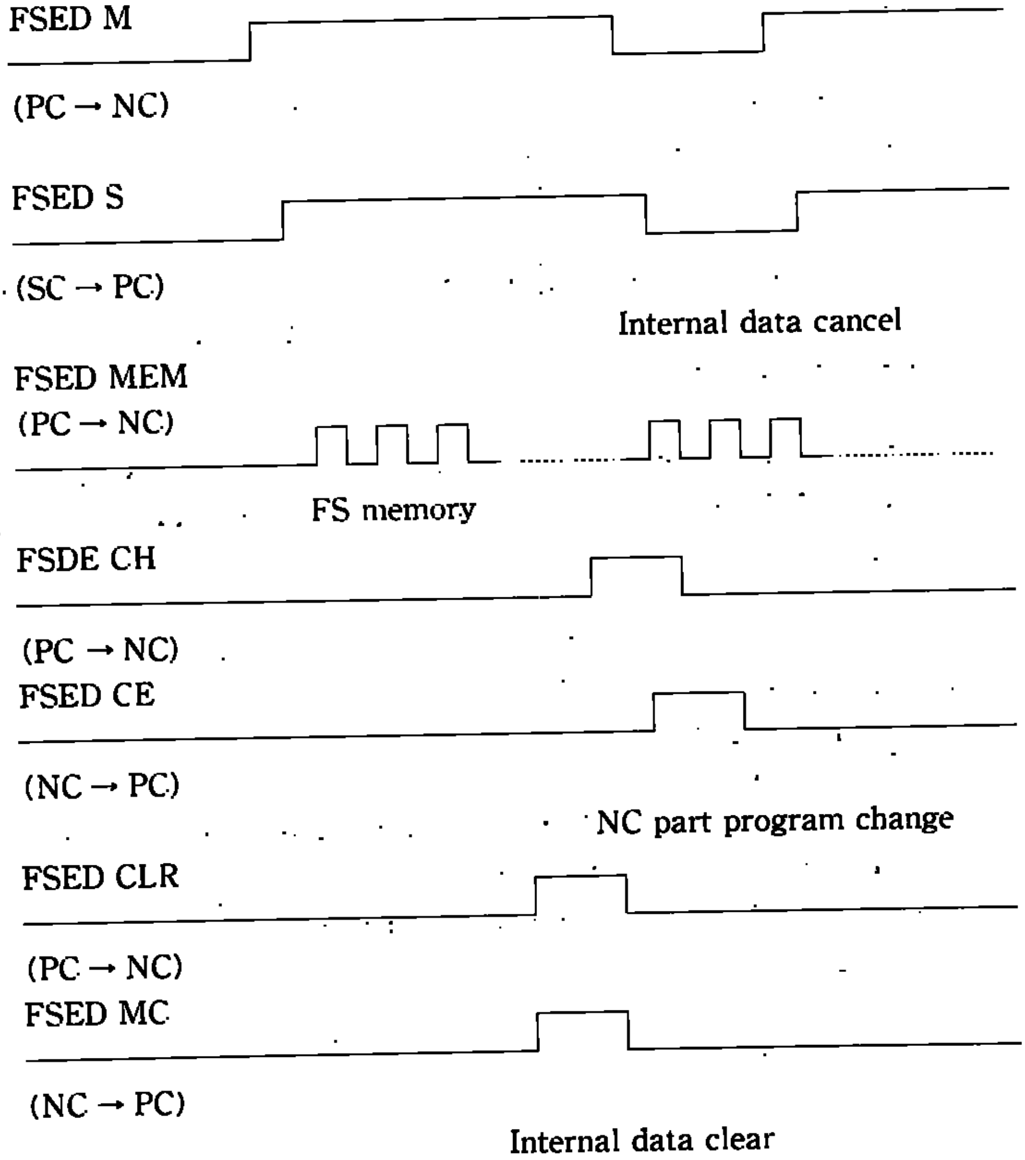
- FSED MC : FS data clear completed

I/O #35042

This signal is closed when the FSED CLR is closed and the F and S values stored in NC are cleared, and opened when the FSED CLR is opened again.

An example of the timing chart of FS automatic edit function signals is shown below.

(Example)



(2) Operations

Use this function in the following procedures.

① Search the program in MEM mode to prepare for memory operation.

② Close the FSED M to enter FS edit mode.

③ After confirming FSED S (in FS edit mode output) in closed status, perform the cycle start to start the memory operation.

④ The F and S and changed to its optimum values.

⑤ When the F and S are changed to its optimum values, close the FSED MEM (FS memory) to store the actual values of F and S in NC.
The F and S values can be stored only during FSED S output and also memory operation.
The F and S values can be stored up to 128 times (number of times FSED MEM is closed).

⑥ When memory of the optimum values for one processing is completed, set the memory operation in hold state (STP).

⑦ When FSED CH (FS memory change) is closed, the message "FSED? (Y/N)" is displayed on the operation screen.
When "Y" is entered, the stored F and S values are reflected in NC program.
When "N" is entered, the message disappears. By closing the signal again, the message will be redisplayed.
The F and S values can be reflected in NC program only during FSED S signal output and also memory operation in hold or interrupted status.

⑧ Open the FSEC CH (FS memory change).

⑨ Open the FSED M (FS edit mode). Then the stored F and S values are canceled.

An example of ameliorated NC program by the above operations is shown below.

<Program before FS edit>

```
0100;
G28 X10. Y10. Z10.;
G92 X100. Y100. Z100.;
G00 G90 F100.;
M03 S3000;
G01 X200.;
Y200.;←---- FSED MEM is closed
  •          with override 80%,
  •          and S overried 110%
  •
X500.;←———Override 60%
  •          S override 100%
M30;
```

<Program after FS edit>

```
0100;
G28 X10. Y10. Z10.;
G92 X100. Y100. Z100.;
G00 G90 F100.;
M03 S3000;
G01 X200.;
Y200. F80. S3300;
  •
  •
  •
X500. F60. S3000;
  •
M30;
```

NOTE

1. When FSED MEM (FS memory) is closed more than 2 times in one block, the F and S values obtained when FSED MEM closed last time are stored. The count of FSED MEM is not increased in this case.
2. When FSED MEM (FS memory) closes during execution of sub program. The F and S values are reflected in the sub program. Therefore, pay attention when the sub program is called from other program.
3. When FSED CH (FS memory change) is closed, CYCLE START is invalid.
4. When FSED MEM (FS memory) is closed, CYCLE.START is invalid.
5. During TAP CYCLE operation. FSED MEM (FS memory) is invalid.
6. When FSED MEM (FS memory) closes during RAPID TRAVERSE operation, only S value is stored.
7. Closing FSED MEM (FS memory) more than 128 times causes a warning alarm and no more data are stored.
8. When FSED M (FS edit mode) closes during memory operation. FSED.S (in FS edit mode) is not closed.
9. When FSED M closes during automatic operation or manual operation, FS edit mode is not entered. A message "FSED MODE INVLD" displayed.
10. When FSED MEM closes during interruption of memory operation (SINGLE BLOCK stop, FEED HOLD), a message "CAN'T STORE FS!" is displayed and the F and S values are not stored.
11. When FSED CH closes during memory operation, a message "CAN'T CHANGE FS!" is displayed and the memory is not validated.
12. When FSED CLR closes during FS memory validation (FS memory change), a message "CAN'T CLEAR FS!" is displayed and the data are not cleared.
13. The following edit operations can not be performed in FS edit mode. EDIT LOCK takes offects. Insert, Change, Delete, Copy, Move, Replace, O-number Copy, O-number Rename, and O-numbr Delete.
14. When FS edit mode is entered with FSED M closed during editing on operation screen, the edit job is released.
15. Closing FSED CH while the tool lenght measurement screen and work preset screen are displayed, the screen is not switched to the opration screen. A message "FSED ERR!" is displayed.

3.6.12 Machine and Servo Systems Correction Function

3.6.12.1 Backlash correction

Corrects lost motion of the machine system

(a) Sub-correction amount 0 to ±32767 pulses (least detection unit)

Parameter	pm1701	X-axis
	pm1702	Y-axis
	pm1703	Z-axis
	pm1704	4th axis
	pm1705	5th axis

(b) Correction amount 0 to ±32767 pulses (least output unit)

Parameter	pm1551	X-axis
	pm1552	Y-axis
	pm1553	Z-axis
	pm1554	4th axis
	pm1555	5th axis

NOTE The backlash correction amount (least output unit) is calculated as follows. (The example is for the X-axis.) Backlash correction amount (least output unit)

= Pm1551 + △

−When separate PG is not used.

△ = L/PG • pm1701

Where,

L : Length of travel per rotation of motor (Pm1821)

PG : Number of pulses generated per rotation of PG (Pm1071)

−When rotary encoder is used for separate PG,

△ = L/α • pm1701

Where,

L : Length of travel per rotation of motor (Pm1821)

α : Number of pulses generated by separate PG per rotation of motor (Pm1841)

−When scale feedback is used for separate PG,

△ = α/1000 • Pm1701

Where,

α : Minimum detection unit of scale feedback (pm1841)

3.6.12.2 Pitch error compensation ※

This is a function that automatically corrects ball screw errors in accordance with correction data for each fixed distance from a reference point on each axis.

This function becomes valid after completion of reference point return.

(1) The specifications are as follows.

Table 3.6.4

Corrected axes		X, Y, Z, 4th (including rotary axis), and 5th axes (including rotating axis)
Number of offset points		Up to 1152 for all axes
Offset reference point		Reference point
Offset interval		Over 10000 pulses
Data setting method		Absolute/incremental (Parameter Pm6001 D0 changeover)
Offset amount	Minimum offset unit	1 pulse (=Least output unit)
	Offset amount each time	Max. 127 pulses

NOTE

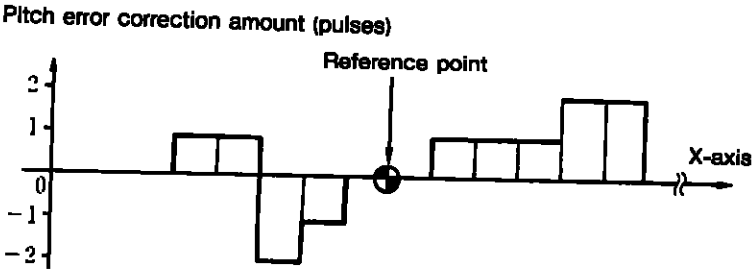
1. The maximum setting value is 127 pulses in the case of absolute settings. This is then multiplied by the offset multiple.
2. Allocation of the number of points per axis is optional if the total number of correction points is within 1152.
3. If the 4th and 5th axes are rotating axes, operation is possible within a maximum of ± 200 rotations.
4. Pitch error valid ′invalid is available for each axis of a parameter. Pm 6000 D0 to D4(X, Y, Z, 4th, and 5th axes).
5. Set correction amount at each axis base point to"0".

(2) The related parameters will become as shown in Table 3.6.5.

Table 3.6.5 Pitch Error Compensation and Parameter

No.	Item	Axis	Parameter	Contents and Remarks
1	Pitch error compensation effective/ineffective	X to β	Pm 6000 D0 to Pm 6000 D4	"0" = Ineffective "1" = Effective
2	Compensation interval	X to β	Pm 6801 to Pm 6805	Over 10000 "1" = 1 Pulse
3	ABS/INC set switching		Pm 6001 D0	"0" = Incremental setting "1" = Absolute setting
4	Origin point No. of compensation	X to β	Pm 6421 to Pm 6425	
5	Max.point of compensation	X to β	Pm 6411 to Pm 6415	
6	Min.point of compensation	X to β	Pm 6401 to Pm 6405	
7	Amount of compensation at each point	X to β	Pm 0000 to Pm 1151	0 to ± 127 "1" = 1 Pulse
8	Compensation scale	X to β	Pm 6101 to Pm 6105	0 to 3 "1" = one time

(3) A write example in the X-axis is shown below.



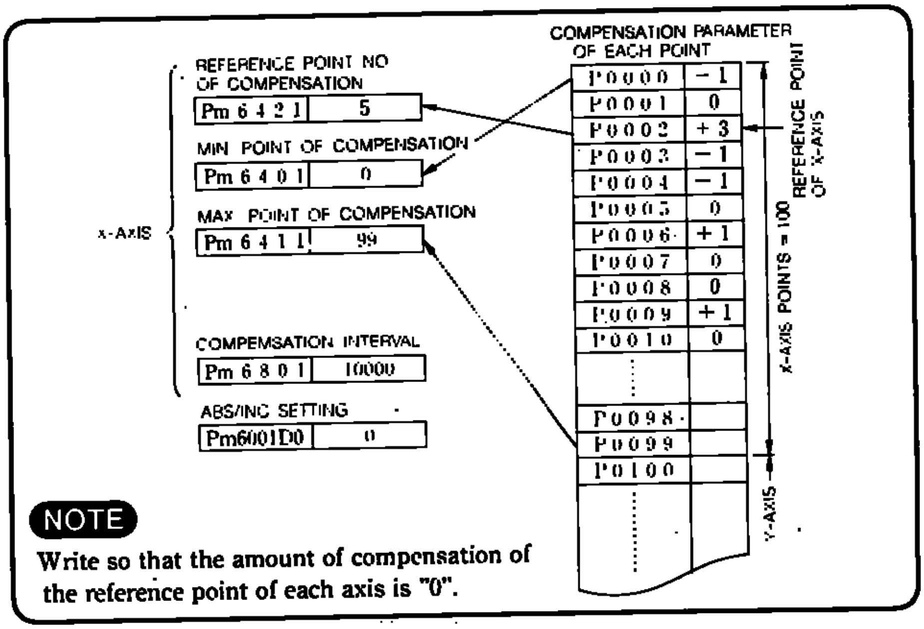
Absolute setting	0	+1	+1	−2	−1	0	+1	+1	+1	+2	+2	
Incremental setting	−1	0	+3	−1	−1	0	+1	0	0	+1	0	
Correction amount (P)	0000	0001	0002	0003	0004	0005	0006	0007	0008	0009	0010	

Fig. 3.6.12 Write Example in the X-axis

Set as shown in Fig. 3.6.12

Compensation interval :10000 pulses
ABS/INC :Incremental
Compensation points of X-axis :100 points

Each parameter will then be set as follows.



NOTE

Write so that the amount of compensation of the reference point of each axis is "0".

Fig. 3.6.13 Parameter Setting

(4) When using the 4th and 5th axes as rotating axes, observe the following rules in addition to the setting direction of the X, Y and Z axes.

① Compensation interval

Set the correction interval at over 10000 pulses and to a value in which the quotient will be a positive integer when 360,000 is divided by the compensation interval. Pitch error compensation will not be performed if the setting is in error.

② Reference point compensation

Set the compensation amount at the reference point as follows.

ABS setting	0
INC setting	0

When setting the compensation amount at the Max. point in INC setting, set so the sum of the compensations at each point becomes "0".

③ Set each parameter as shownn in Fig. 3.6.15, if the compensation interval is 45000 pulses (8 divisions per rotation) as shown in Fig.3.6.14.

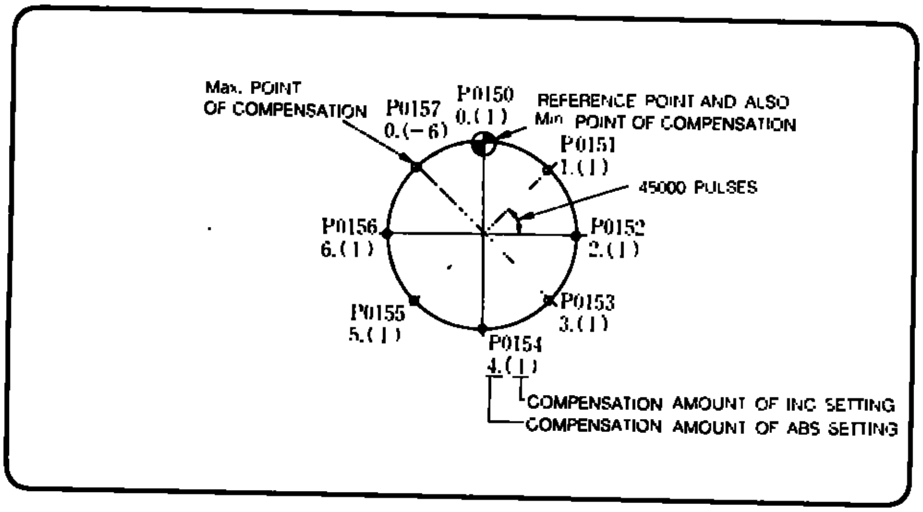
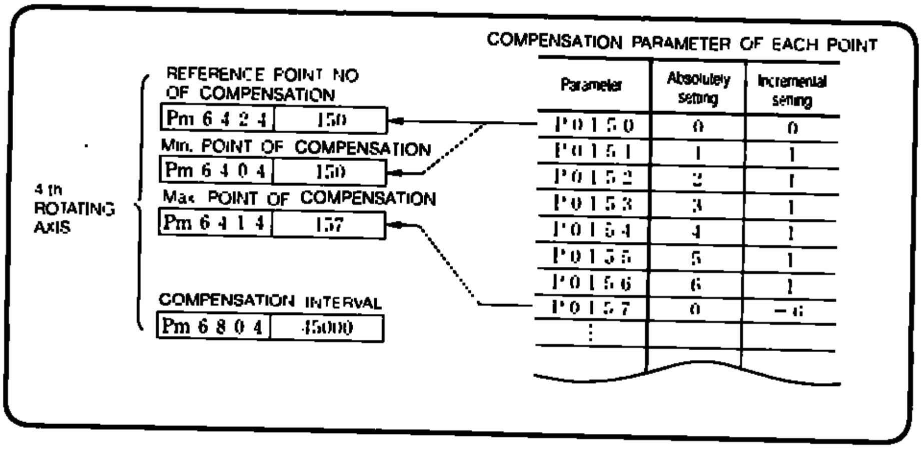


Fig. 3.6.14 Compensation Interval



COMPENSATION PARAMETER OF EACH POINT

Parameter	Absolutely setting	Incremental setting
P 0 1 5 0	0	0
P 0 1 5 1	1	1
P 0 1 5 2	2	1
P 0 1 5 3	3	1
P 0 1 5 4	4	1
P 0 1 5 5	5	1
P 0 1 5 6	6	1
P 0 1 5 7	0	−6
⋮		

Fig. 3.6.15 Parameter Setting

3.6.13 Absolute Value Detecting Function*

Outline

By detecting and holding the machine movement even when the power supply is turned OFF, the absolute value detecting function enables automatic setting of the machine coordinate system and work coordinate system for immediate restart of operation without zero-point return after the power supply is turned ON.

Description of operation

Manual low-speed type zero-point return method
[At pm8002 (D0, D1 = 0)]
Floating machine zero-point manual setting method
[At pm8002 (D0 =1, D1 =0)]

Key Point Operation

Function soft key

SETOFF 1 ⇨ WR

The cursor is displayed in the zero-point setting status.

Only when Zero-point setting status is "FINISH" it is changed to "UNFINISH"

Function soft key

GAPOFF 2 ⇨ WR

The cursor is displayed at the position error.

Only when the position error is "EXIST", it is changed to "NOT EXIST".

●Only in floating machine zero-point manual setting

Function soft key

ON/OFF 3 ⇨ WR

The cursor is displayed at the axis name.

Zero-point setting is performed where the cursor is displayed.

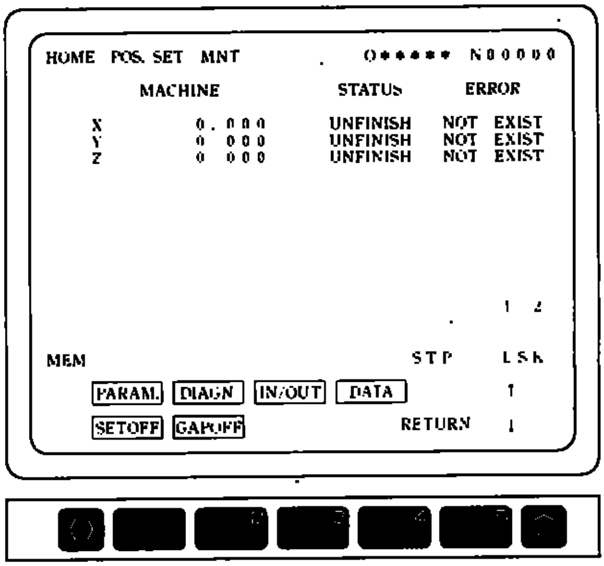
■ Manual low-speed type zero-point return method

<Zero-point setting release function soft key>

Displays whether zero-point return has been performed.

1 Depress the SETOFF function soft key.

The cursor is displyed at the position in zero-point setting status. (Fig.3.6.16)



```
HOME  POS. SET  MNT              O******  N00000
          MACHINE              STATUS      ERROR
    X         0.000           UNFINISH   NOT  EXIST
    Y         0 000           UNFINISH   NOT  EXIST
    Z         0 000           UNFINISH   NOT  EXIST




                                                1  2

MEM                                    STP     LSK
      PARAM.   DIAGN   IN/OUT   DATA            1
      SETOFF   GAPOFF              RETURN       1
```

Fig. 3.6.16 Zero-point Setting Release

2 Use the cursor ▲ or ▼ key to select the axis and depress the WR key.

When zero-point setting is "FINISH", it is changed to"UNFINISH".

3 Depress the SETOFF function soft key again.

The cursor display disappears.

NOTE
1. When the axis with zero-point setting cnmpleted is set to "UNFINISH", perform zero-point setting (zero-point return) again after turning OFF the power supply.
2. Depressing the "zero-point setting release" function soft key when zero-point setting is "UNFINISH" does not change any data.

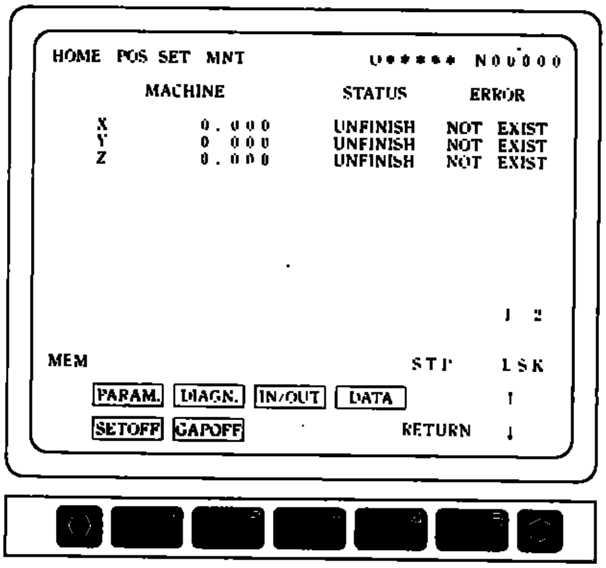
<Position shifting release function soft key>

The machine coordinate value at power OFF and the current machine coordinate value are compared when the power supply is turned ON ; if the positions are shifted more that the value set in the parameter (pm8411 to pm8415), position error "EXIST" is displayed. (However, if the parameter setting is 0, it is not checked.)

When position error "EXIST" is displayed and ALM2131 to 2135 occurs, the alarm can be released by the following operation.

1 Depress the GAPOFF function soft key.

The cursor is displayed at the position error. (Fig.3.6.17)



```
HOME  POS SET  MNT                  U******  N00000

          MACHINE                 STATUS      ERROR

     X          0.000           UNFINISH    NOT  EXIST
     Y          0 000           UNFINISH    NOT  EXIST
     Z          0.000           UNFINISH    NOT  EXIST




                                                  1   2

MEM                                     STP       LSK
     PARAM.  DIAGN.  IN/OUT  DATA                  ↑
     SETOFF  GAPOFF                   RETURN       ↓
```

Fig. 3.6.17 Position Shifting Release

2 Use the cursor ▲ or ▼ key to select the axis and depress the WR key.

When zero-point setting is "EXIST", it is changed to "NOT EXIST".

3 Depress the RESET key.

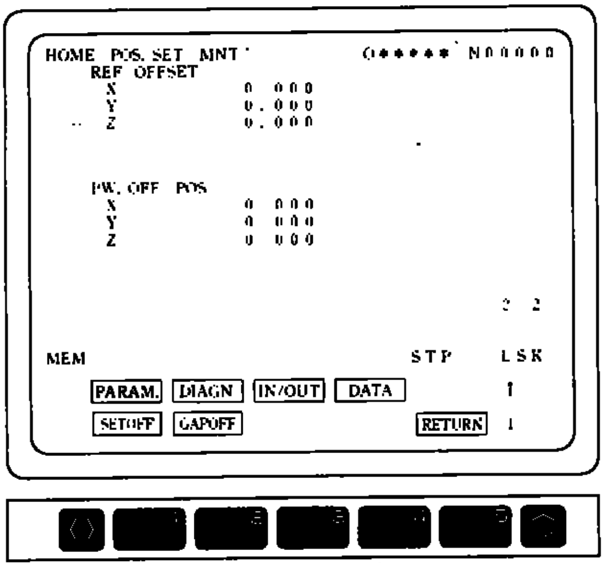
Alarms2131 to 2135 "absolute position detection error" disappear.

4 Depress the GAPOFF function soft key again.

The cursor displayed at the position error disappears.

Depress ∨ – page – ∧.

On the next page, the zero-point setting offset value and machine position at power OFF are displayed. (Fig. 3.6.18)



```
HOME  POS. SET  MNT                  O******  N00000
    REF  OFFSET
     X                 0  000
     Y                 0.000
     Z                 0.000


    PW. OFF   POS
     X                 0  000
     Y                 0  000
     Z                 0  000

                                                  2   2

MEM                                       STP     LSK
    PARAM.  DIAGN  IN/OUT  DATA                    ↑
    SETOFF  GAPOFF                     RETURN      ↓
```

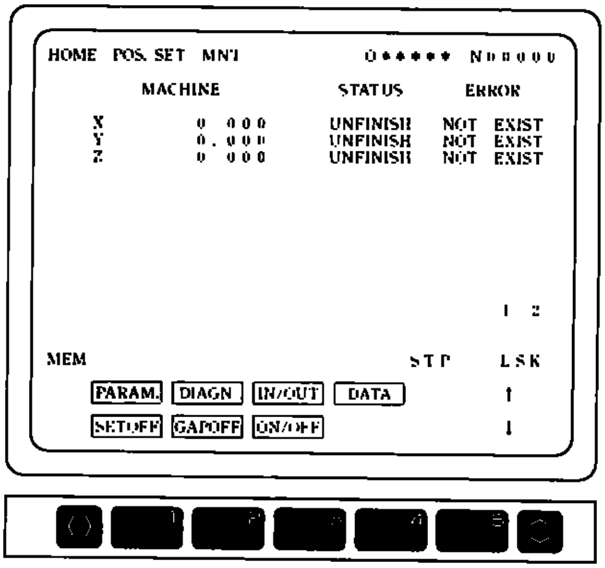
Fig. 3.6.18 Manual Low-speed Type Zero-point Return Method

② Floating machine zero-point manual setting method

- The operation of zero-point setting release and position shifting release functions is the same as that of the manual low-speed type zero-point return method.
- Setting/release function soft key
  By setting the zero-point status to "UNFINISH", zero-point setting can be performed again.

1 Depress the ON/OFF function soft key again.

The screen shown below is displayed and the cursor is displayed at the axis name. (Fig. 3.6.19)



```
HOME  POS. SET  MNT                    O******  N00000

          MACHINE                  STATUS      ERROR

   X          0  000           UNFINISH    NOT  EXIST
   Y          0. 000           UNFINISH    NOT  EXIST
   Z          0  000           UNFINISH    NOT  EXIST




                                                   1  2

MEM                                       STP      LSK
     PARAM.  DIAGN   IN/OUT   DATA                  ↑
     SETOFF  GAPOFF  ON/OFF                         ↓
```

Fig. 3.6.19 Zero-setting Mode Screen

2 Use the cursor ▲ or ▼ key to move the cursor to the axis where zero-point setting is performed.

3 Move the axis to the position to be the zero-point by handle or jog feeding and depress the WR key.

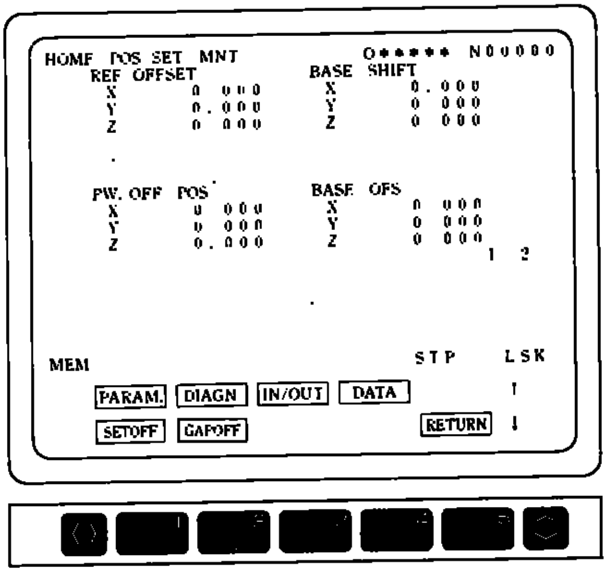
→ The zero-point setting status is "FINISH" and zero-point setting is completed.

4 Depress the ON/OFF function soft key again.

The zero-point setting mode is released.

Depress ∨ − page − ∧

On the next page, the zero-point setting offset value, machine position at power OFF, reference shifted value and reference shift fine-adjusted value are displayed. (Fig. 3.6.20)



```
HOMF  POS SET  MNT                 O******  N00000
    REF  OFFSET                 BASE  SHIFT
     X         0  000            X        0.000
     Y         0.000             Y        0  000
     Z         0  000            Z        0  000

    PW. OFF  POS                BASE  OFS
     X         0  000            X        0  000
     Y         0  000            Y        0  000
     Z         0.000             Z        0  000
                                                   1   2

MEM                                     STP     LSK
```

Fig. 3.6.20 Floating Machine Zero-point Manual Setting Method

NOTE

1. If the ON/OFF function soft key is depressed in the automatic mode, a warning "RUN MODE ERROR !" is displayed.
2. If zero-point setting is performed for any axis with the zero-point setting status "FINISH", a warning "AXIS REF. POS.!" is displayed.
3. If zero-point setting is performed by the WR key during axis movment, a warning "AXIS MOVING" is displayed.

By depressing the RETURN 5 function soft key, the setup menu display screen is returned.

The following describes each display item.

① Machine coordinate value
Displays the distance from the servo axis machine zero-point.
(Same as position machine coordinate value)

② Zero-point setting offset value
Displays the absolute position obtained by the encoder at zero-point setting. (This value is stored in parameters pm8811 to pm8815.)

③ Machine position at power OFF
Displays the machine coordinate value obtained at the previous power OFF.
(This value is stored in parameters pm8801 to pm8805.)

④ Reference point shifted value
Displays the shifted value from the set point at zero-point setting in the floating machine zero-point setting method. (pm8821 to pm8825)

⑤ Reference point shift fine-adjusted value
Displays the reference point shift fine-adjusted value.
(pm8401 to pm8405)

NOTE
1. The absolute position detecting function is valid when a motor provided with an absolute position encoder is used.
2. Since the absolute position encoder has battery backup, when the battery is dead, it does not function. To prevent this, battery failure is detected in advance and a battery alarm warning is output.
3. To release the warning, it is necessary to replace the battery.

SECTION 4

MAINTENANCE

To enable YASNAC i80M to provide long and reliable service, this section shows routine maintenance, battery replacement, alarms, the corrective action for alarms, etc.

CONTENTS PAGE

4.1 ROUTINE MAINTENANCE

Always perform maintenance according to the following simple routine check to enable YASNAC to provide long and reliable service.
Always turn OFF the power source to the NC unit when carrying out routine maintenance except when checking is possible with the NC unit power on, such as external visual check for dirt, grease, etc. or check for noise or vibration.

NOTE

It is not sufficient to simply "switch off power" by depressing the "POWER OFF" switch on the NC operator panel. There will be the danger of electrical shock since power is supplied to specific parts in the NC unit even when the power is OFF.
Therefore, turn OFF the power to the NC unit by setting the MCCB molded case (circuit breaker) located on (or inside) the power panel on the machine side.

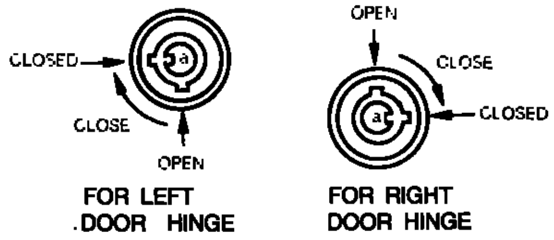
Cut off of all power to the NC unit can be confirmed by the fans in the unit since all will stop when power is turned OFF.

4.1.1 Control Panel Maintenance

(1) Check to ensure the door is fully closed (daily).

① The unit is of fully airtight design to prevent entrance of air, oil mist, dust, etc. from the outside. It is mandatory that the control panel doors be closed at all times not only during operation.

② When it is necessary to open the door for maintenance, use the special key supplied (Type: YE001) and lock (2 locks per door) the door positively when finished. The key can be inserted with the door either open or closed.



Notch in "a" horizontal	"Closed"
Notch in "a" vertical	"Open"

Fig. 4.1.1 Closing Door

NOTE In units equipped with a door interlock switch※,if the door is opened all operations become impossible, in addition to the power being turned off.

(2) Checks for gaps or damage in the airtight structure (once a month)

① Open each door and check for damage of the airtight packing on the edges of the door.

② Check for abnormal contamination inside the unit. If stains or other contamination are present, check the cause and clean immediately.

③ Check to ensure that there is no gap with the door completely closed and locked.

YASNAC Controllers can be used for a longer period of time safely by executing the above checking.

4.1.2 Maintenance of the SERVOMOTOR and Spindle Motor

(1) Vibration and noise check (Daily or as appropriate)

Check for vibration and sound different from normal by feel for vibration and aurally for noise, and contact maintenance personnel if any abnormality is detected.

(2) External visual check of contamination or damage. (Daily or as appropriate)

Check visually for excessive dirt, oil, etc. or major damage. When abnormally dirty, remove the motor cover on the machine side and check in accordance with application (Refer to the instruction manual issued by the machine tool builder.)

4.1.3 Battery Check

Check whether "BAT" is displayed on the lower right corner of the NC operating panel screen. If displayed, contact maintenance personnel since the battery must be replaced within one month.

NOTE Replacement of battery after the power turns OFF should be performed as fast as possible since leaving the unit without battery for a long period may erase the stored memory.

Also, turn off the power after a few seconds of the power ON.

4.2 BATTERY REPLACEMENT

The memory in which the parameters, tool corrections and processing programs are stored is provided with a backup battery so data are not lost if a power failure occurs. The alarm "BAT" will be displayed on the NC operator panel screen when the battery becomes weak.
When this alarm is displayed, it will be necessary to replace the battery within one month.

NOTE 1. If the existing battery is removed with the power OFF, the stored data will be lost.
2. Since commercial batteries cannot be used as is, always consult your YASKAWA representative when replacing the battery.

(REPLACEMENT PROCEDURE)

(1) Depress the POWER OFF switch.

(2) When equipped with a door interlock switch
Open the front door and pull out the moving part of the door interlock switch (There is a one-stage snap action.)
— Power can then be turned on with the front door open.

(3) Open the front door sufficiently so the NC unit board mounted on the inside of the door is visible.

NOTE Depending on the model, the NC unit may be mounted on the main body instead of on the door.

(4) Switch on the power.

(5) Confirm that the LED (red) of the FC200 module lights. If this LED is lit, it indicates that the battery "ER6V3" must be replaced. The layout of the LED and the battery is shown in Fig. 4.2.1.
— The procedure up to now is to confirm that the battery ER6V3 (with lead connector) is defective.

(6) Set POWER switch to OFF.

(7) Remove the old battery from the connector and take it out of the holder.

(8) Place the new battery in the holder and insert the connector.

NOTE The connector can be inserted in either right or left direction.

(9) If there is a door interlocking switch, depress the moving part of the switch.

→The switch will reset and power cannot be turned ON while the door is "open".

(10) Close the front door completely.

(11) Switch ON the power.

(12) Confirm that the battery alarm display on the NC operator panel screen is OFF.

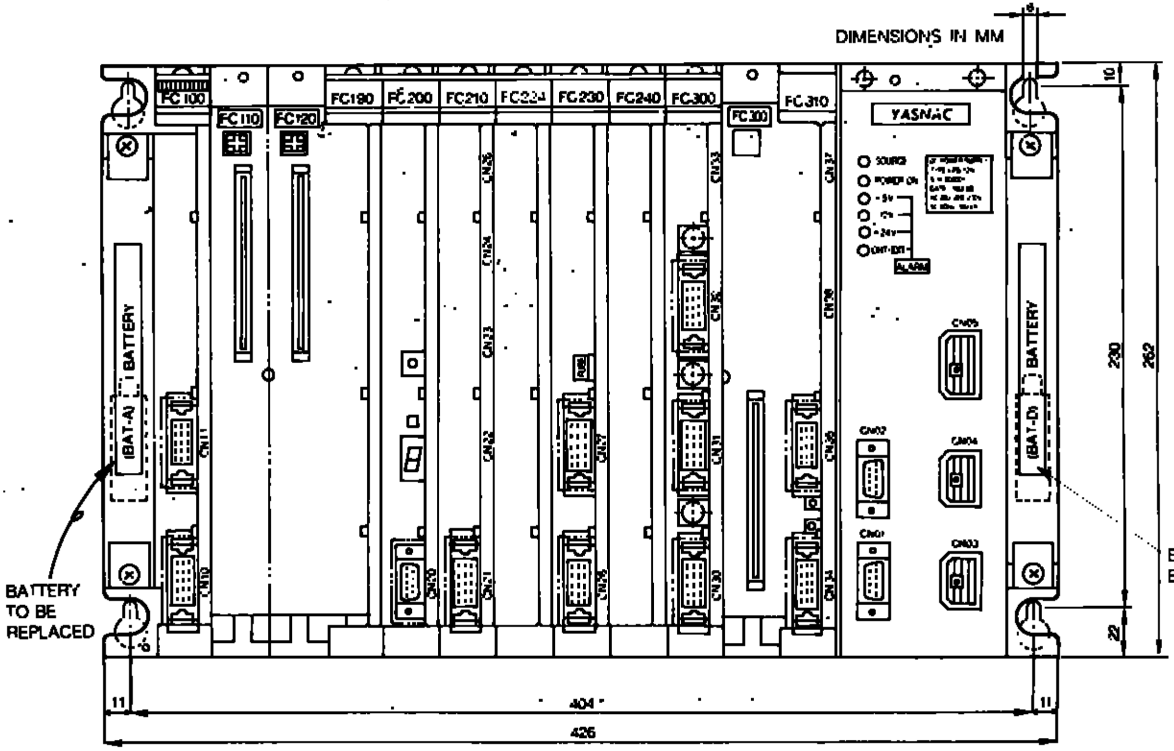


Fig. 4.2.1 Arrangement of LED and Battery

NOTE
1. Since work is conducted with the front door open, it is important that the work be performed within the shortest period of time (to prevent entrance of contaminants)
2. Care is required to prevent water drops, oil and dust from adhering to the various parts (printed circuit board, connector, cable) in the unit.

4.3 POWER SUPPLY

Although composite power supplies (CPS-12N) for control are provided with various protective functions, it will be necessary to contact maintenance personnel. immediately after checking the following items if an abnormal situation arises such as inability to switch ON NC power.

(1) The "SOURCE" (green) LED goes out

① Check whether the main circuit breaker of the machine or the factory has tripped or if there is an open phase.

② Trouble may be considered in the composite power supply itself for control.

(2) " + 5, ± 12V" LED (red) lights

① This lights when overcurrent due to a short circuit in the + 5V or ± 12V, or when a + 5V overvoltage is detected.

② If overcurrent is the cause of the failure, the power can be turned ON again by switching power on  ⇨  after removing the cause.

③ If + 5V overvoltage is the cause, switch power ON again after first confirming that the "SOURCE" LED is out by setting the main breaker to OFF once.

(3) " + 24V" LED (red) lights

① This LED lights if overcurrent due to a short circuit in the + 24V output is detected.

② Switch power ON  ⇨  after removing the cause of the failure.

(4) "0HT" LED (red) lights

① This LED lights when abnormal temperature rise of internal parts is detected.

② Switch power ON  ⇨  after removing the cause of the failure.

4.4 SERVO CONTROL UNIT ALARM AND CORRECTIVE ACTION

Servo-related alarms consist of those having detecting functions on the NC unit side and those having detecting functions on the servopack side. They can both be confirmed with the alarm screen, servo alarm screen and CRT display.

(1) Alarms detected on the NC unit side

Table 4.4.1 Alarms Detected on the NC Unit Side

Alarm	No.	Contents
Overdeviation Detection	# 3041 – # 3045	Detected when the value determined by machine followup deviation exceeds the value determined by the machine maker.
Overload Detection	# 3061 – # 3065	Detected when excessively deep cuts are made, when variable speed frequency is excessive, and when friction of the mechanical system increases abnormally.
PG Open Circuit Detection	# 3081 – # 3095	Detected when there is an open circuit in the signal line from PG.
Overspeed Detection	# 3121 – # 3125	Detected when there is an overspeed command and maximum motor speed is exceeded.
Overtravel Prevention Detection	# 3141 – # 3145	Overtravel operation due to erroneous connection of the motor or PG signal is detected in advance.
Absolute Error Detection	# 3161 – # 3165	Detects malfunction of absolute encoder when an absolute encoder is used.
Position Error Detection	# 3181 – # 3185	Detects pulse count servopack of the internal PG when using an absolute encoder.
Servopack Communication Error Detection	# 3201 – # 3205	Detects malfunction in serial communications between the NC and the servopack.

NOTE For details, refer to the Maintenance Manual attached. (TOE-C 843-11.2)

(2). Alarm detected on the servopack side (servo alarm)

If trouble occurs in the servopack side, servo alarm is displayed on the alarm screen in the form of #3101 to #3105 for each axis. The contents of the servo alarm can be confirmed on the servo alarm screen (refer to par.3.4.9.2(2) "Servo alarm function").

Table 4.4.2 Alarm Detected on the Ser

No.	Alarm	Contents
D 0	–	–
D 1	Overcurrent detection	Detects overcurrent flow in the main circuit.
D 2	MCCB trip detection	Detects tripping of MCCB.
D 3	Regeneration error detection	Detected when the regeneration processing circuit in the servopack is inoperative.
D 4	Overvoltage detection	Overvoltage is detected when the DC voltage in the main circuit is abnormally high.
D 5	Undervoltage detection	Undervoltage is detected when the DC voltage in the main circuit is under 150 V after power is turned on.
D 6 – D 9	–	–
DA	Heat sink Overheat detection	Overheat is detected when the heat sink in the servopack has abnormally high temperature (over about 85℃)
DB, DC	–	–
DD	Fuse blown detection	Detects blown fuse in a servopack with 2 or 3 axes.
DE	Open circuit detection of the current reference cable	Detects open circuit in the current reference cable.
DF	Open phase detection	Detects an open phase in 3-phase power supplies.

NOTE For further details, refer to the accompanying Service Manual. Instruction Manual. If any of the above alarms occur, contact your YASKAWA representative immediately after confirming details of the alarm display.

4.5 MCCB (WIRING CIRCUIT BREAKER)

MCCB units that can be set ON/OFF externally are sometimes provided in models in which all power sequence control units are consolidated in special case form on the NC unit side.

Power to the NC unit can generally be cut off since the MCCB is in OFF position. For details, refer to the instruction manual issued by the machine tool manufacturer.

4.6 CAUSE OF FAULTS AND CORRECTIVE ACTION

4.6.1 On-line Self-diagnosis

(1) Most causes of faults in the equipment are those that can be detected by simple checks and those that are reparable. This unit is provided with the following functions for easy and rapid detection of causes.

① Four-digit alarm code and message display
② State display (with M, S, T, F, P, D classification)
③ I/O signal display

(2) The above displays are possible regardless of whether during operation or when stopped.

4.6.2 Cause due to Alarm Code, and Corrective Actions

(1) If operation stops with "ALM" blinking in the lower right corner of the screen, depress COMN key and ALARM job function soft key.

— Both the alarm code and message will then be displayed on the screen.

NOTE Although one alarm code and message is normally displayed, important alarms will be displayed if multiple alarms occur simultaneously.

(2) List of alarm codes

A list of alarm codes of high order is shown in YASNAC i 80M APPENDIX (TOE-C843-11.31).

4.6.3 I/O Signal Diagnosis

(1) When the cause of trouble is assumed to be in the I/O part of the unit, it will be possible to check by displaying the present state of the I/O signal.

(2) Standard I/O signal and custom I/O signal
The standard I/O signal can be observed by specifying the diagnosis No. with the keys on the NC operator panel.
If a power sequence control option is provided as optional, the system will have its own characteristic "custom I/O signal" and this state can be observed by setting an exclusive diagnosis No. in the unit.
For details, refer to your machine tool builder's manual.

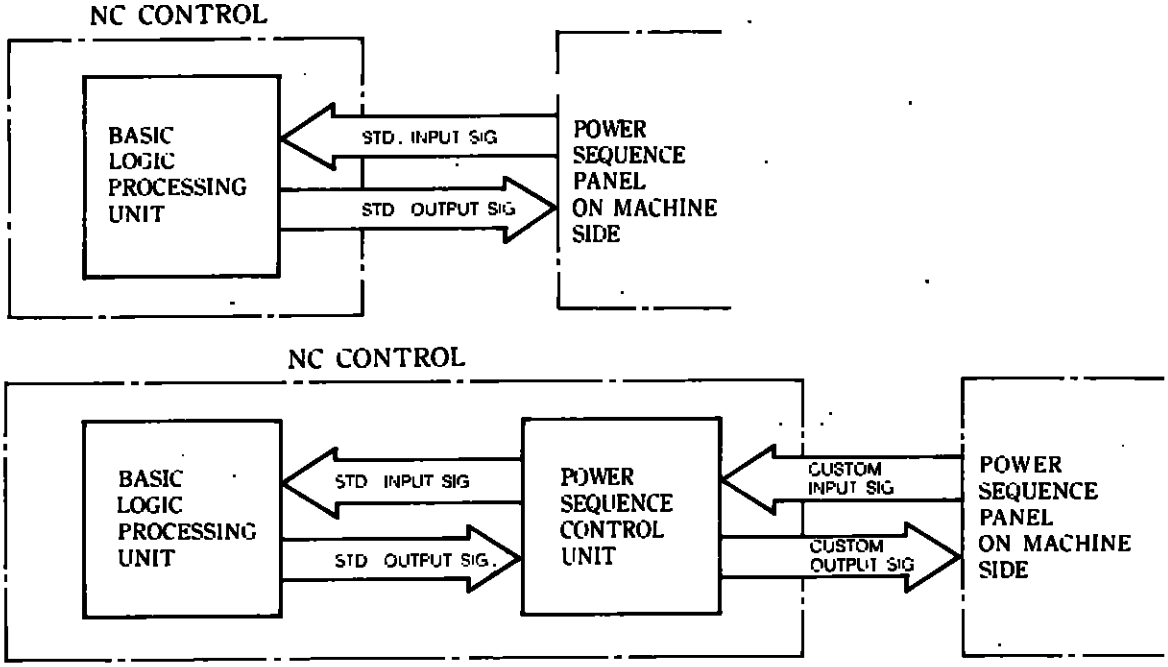


Fig 4.6.1 YASNAC i 80 M System

YASNAC i80M
INSTRUCTIONS

TOKYO OFFICE
New Pier Takeshiba South Tower, 1-16-1, Kaigan, Minatoku, Tokyo 105 Japan
Phone 81-3-5402-4511 Fax 81-3-5402-4580

YASKAWA ELECTRIC AMERICA, INC.
Chicago-Corporate Headquarters
2942 MacArthur Blvd. Northbrook, IL 60062-2028, U.S.A.
Phone 1-847-291-2340 Fax 1-847-498-2430
Chicago-Technical Center
3160 MacArthur Blvd. Northbrook, IL 60062-1917, U.S.A.
Phone 1-847-291-0411 Fax 1-847-291-1018

MOTOMAN INC. HEADQUARTERS
805 Liberty Lane West Carrollton, OH 45449, U.S.A.
Phone 1-937-847-6200 Fax 1-937-847-6277

YASKAWA ELÉTRICO DO BRASIL COMÉRCIO LTDA.
Avenida Brigadeiro Faria Lima 1664-5° CJ 504/511, São Paulo, Brazil
Phone 55-11-815-7723 Fax 55-11-870-3849

YASKAWA ELECTRIC EUROPE GmbH
Am Kronberger Hang 2, 65824 Schwalbach, Germany
Phone 49-6196-569-300 Fax 49-6196-888-301

Motoman Robotics AB
Box 504 S38525 Torsås, Sweden
Phone 46-486-48800 Fax 46-486-41410

Motoman Robotec GmbH
Kammerfeldstraße 1, 85391 Allershausen, Germany
Phone 49-8166-900 Fax 49-8166-9039

YASKAWA ELECTRIC UK LTD.
3 Drum Mains Park, Orchardton Woods, Cumbernauld, Scotland, G68 9LD, United Kingdom
Phone 44-1236-735000 Fax 44-1236-458182

YASKAWA ELECTRIC KOREA CORPORATION
Paik Nam Bldg 901 188-3, 1-Ga Euljiro, Joong-Gu Seoul, Korea
Phone 82-2-776-7844 Fax 82-2-753-2639

YASKAWA ELECTRIC (SINGAPORE) PTE. LTD.
151 Lorong Chuan, #04-01, New Tech Park Singapore 556741, Singapore
Phone 65-282-3003 Fax 65-289-3003

YATEC ENGINEERING CORPORATION
Shen Hsiang Tang Sung Chiang Building 10F 146 Sung Chiang Road, Taipei, Taiwan
Phone 886-2-563-0010 Fax 886-2-567-4677

BEIJING OFFICE
Room No. 301 Office Building of Beijing International Club, 21
Jianguomenwai Avenue, Beijing 100020, China
Phone 86-10-6532-1850 Fax 86-10-6532-1851

SHANGHAI OFFICE
27 Hui He Road Shanghai 200437 China
Phone 86-21-6553-6600 Fax 86-21-6531-4242

YASKAWA JASON (HK) COMPANY LIMITED
Rm. 2909-10, Hong Kong Plaza, 186-191 Connaught Road West, Hong Kong
Phone 852-2803-2385 Fax 852-2547-5773

TAIPEI OFFICE
Shen Hsiang Tang Sung Chiang Building 10F 146 Sung Chiang Road, Taipei, Taiwan
Phone 886-2-563-0010 Fax 886-2-567-4677

SHANGHAI YASKAWA-TONGJI M & E CO., LTD.
27 Hui He Road Shanghai China 200437
Phone 86-21-6531-4242 Fax 86-21-6553-6060

BEIJING YASKAWA BEIKE AUTOMATION ENGINEERING CO., LTD.
30 Xue Yuan Road, Haidian, Beijing P.R. China Post Code: 100083
Phone 86-10-6233-2782 Fax 86-10-6232-1536

YASKAWA ELECTRIC CORPORATION

YASKAWA

MANUAL NO. TOE-C843-11.30F
Printed in Japan December 1997 91-3 0.5Z · 8 ·