YASKAWA ELECTRIC

TOE-C843-7.30C
INSTRUCTIONS

CNC SYSTEM FOR MACHINING CENTERS

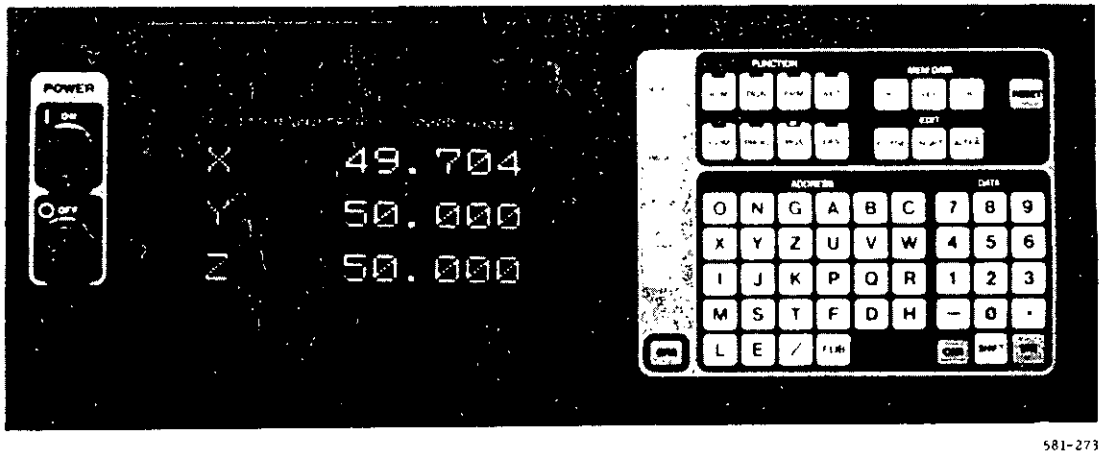
YASNAC® MX1

OPERATOR'S MANUAL

Before initial operation read these instructions thoroughly, and retain for future reference.

This manual is primarily intended to give operators instructions for YASNAC MX1 programming, operation and maintenance.

This manual applies to the basic and optional features of YASNAC MX1. The optional features are marked with a dagger. For the specifications of your YASNAC MX1, refer to the machine tool builder's manual.



581-273

YASNAC MX1 OPERATOR'S STATION

PREFACE

Read this manual keeping in mind that the information contained herein does not cover every possible contingency to be met with the operation. The operations not described in this manual should not be attended with the control.

The functions and performance as NC machine are determined by a combination of machine and the NC control. For operation of your NC machine, the machine tool builder's manual shall take priority over this manual.

Unless otherwise specified, the following rules apply to the description of programming examples shown in this manual.

- Least Input Increment: 0.01 mm
- Feed Function Selection: G94 (mm/min.)
- Absolute Zero Point: 
- Reference Zero Point
  (Return to reference zero by manual and automatic return): 
- Dimensions: in MM

TABLE OF CONTENTS

INDEX

INDEX

INDEX

INDEX

INDEX

INDEX

1. INTRODUCTION

The YASNAC MX1 is a high-performance CNC for simultaneously controlling 3 or 4 axes of a machining center, with emphasis placed on high-speed machining, unattended automatic operation, or feedback gauging control.

With the NC logic incorporating 16-bit microprocessors and various LSIs, the YASNAC MX1 incorporates a compact design with a wide range of capabilities. The memory comprises permanent, semi-permanent and programmable software storage used in combination to utilize each one to maximum advantage.

The data input-output interface has been expanded in concept, and, in addition to conventional interfaces such as FACIT and RS 232C, RS 422 is now available to accommodate requirements for new modes of operations such as high-speed, long-distance data transmission.

The YASNAC can incorporate a programmable machine interface, and the logic diagram can be edited easily from the NC operator's station.

2. PROGRMMING

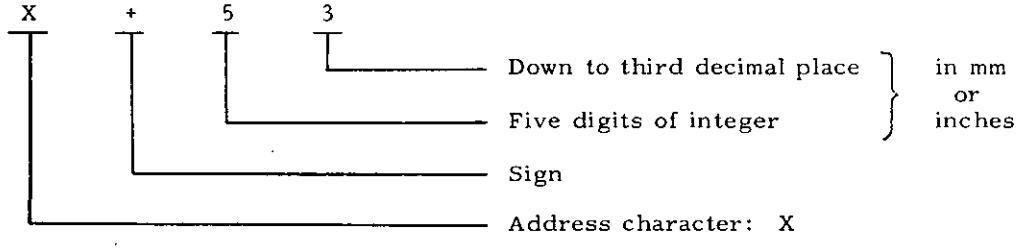
2.1 INPUT FORMAT

2.1.1 INPUT FORMAT

A variable block format conforming to JIS#B 6313 is used for YASNAC MX1.

Table 2.1 shows the input format. Numerals following the address characters in Table 2.1 indicate the programmable number of digits.

EXAMPLE



Note:

A decimal point should be omitted in actual programming when you make a program including decimal points, refer to 2.1.3 DECIMAL POINT PROGRAMMING.

The leading zeros can be suppressed for all address codes. Plus signs need not be programmed, but all minus signs must be programmed.

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 (;).

- Metric input format

  04 N4 G3 a+43 F5 S2 T2 M3 D(H)2 B3;

- Inch input format

  04 N4 G3 a+34 F31 S2 T2 M3 D(H)2 B3;

Notes:

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

Japanese Industrial Standard

Table 2.1 Input Format

No.	Address		Metric output		Inch input		B: Basic O: Optional
			Metric input	Inch input	Metric input	Inch input	
1	Program No.		O4		O4		B
2	Sequence No.		N4		N4		B
3	G function		G3		G3		B
4	Cordinate Word	Linear axis	a + 43	a + 34	a + 53	a + 34	B
		Rotary axis	b + 43	b + 43	b + 43	b + 43	O
5	Feed/min		F40	F31	F50	F31	B
6	Feed/min 1/10		F41	F32	F51	F32	B
7	Feed/rev		F22	F13	F22	F13	O
8	Feed/rev 1/10		F23	F14	F23	F14	O
9	S-function		S2		S2		B
			S5		S5		O
10	T-function		T2		T2		B
			T4		T4		O
11	M-function		M3		M3		B
12	Tool Offset No.		H2 or D2		H2 or D2		B
13	B-function		B3		B3		O
14	Dwell		P53		P53		B
15	Program No. designation		P4		P4		B
16	Sequence No. designation		P4		P4		B
17	No. of repititions		L8		L8		B

2.1.2 ADDRESS AND FUNCTION CHARACTERS

Address characters and their meanings are shown in Table 2.2.

Table 2.2 Address Characters

Address Characters	Meanings	B: Basic O: Optional
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	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	O, B
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	O
X	X-coordinate	B
Y	Y-coordinate	B
Z	Z-coordinate	B

2.1.2 ADDRESS AND FUNCTION CHARACTERS (CONT'D)

Table 2.3 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, User macro operator	
/	/	Optional block skip	
Del	DEL	Disregarded (Including All Mark)	
.	.	Decimal point	
Parameter setting	#	Sharp (Variable)	EIA: Special code
*	*	Astrisk (Multiplication operator)	
=	=	Equal mark	
[	[	Left bracket	
]	]	Right bracket	
:	:	User macro comment	
$	$	User macro comment	
@	@	User macro comment	
?	?	User macro comment	

Notes:

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

2.1.3 DECIMAL POINT PROGRAMMING

Numerals containing a decimal point may be used as the dimensional data of addresses related to coordinates (distance), time and speed.

Decimal points can be used in the following address words.

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

Time word: P

Feed rate word: F

EXAMPLE

	[mm]	[inch]
X15. ———	X15.000 mm	or X15.0000 inches
Y20.5 ———	Y20.500 mm	or Y20.5000 inches
(G94)F25.6 —	F25.0 mm/min (for F4.0)	or F25.6 inches/min (for F3.1)
G95F.2+ ——	F0.20 mm/rev (for F2.2)	or F0.200 inches/rev (for F1.3)
G04P1. ———	Dwell 1.000 sec	

Normally, when data without a decimal point is inputted, the control regards "1" as 0.001 mm (or 0.0001 inches, or 0.001 deg.), but with a parameter setting, the control may be made to regard "1" as 1 mm (or 1 inch or 1 deg.). Refer to parameter #6019$_{D6}$).

2.1.4 LABEL SKIP FUNCTION

In the cases named below, the label skip function becomes effective, and LSK is displayed on the CRT.

- When the power supply is turned on.
- When the RESET key is pushed.

While the label skip function is effective, all data on the punched tape up to the first EOB code are neglected. When LSK is displayed on the CRT in the MEM (memory) or EDIT (editing) mode, it indicates the presence of a pointer at the leading end of the part program.

2.1.5 BUFFER REGISTER

During normal operation, one block of data is read in advance and compensation computing is made for the follow-on operation.

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. One block can contain up to 128 characters including EOB.

2.1.6 MULTI-ACTIVE REGISTERS+

For the portion of part programs sandwiched in between M93 and M92, up to 5 blocks of data are read in advance.

M code	Meaning
M92 ◥	Multi-active register off
M93	Multi-active register on

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◥.

Inter-block stoppage can be eliminated when the program is so made that the automatic operation time of advance reading of 5 blocks is longer than processing time of advance reading of next 5 blocks of data.

NOTE:

Advance reading is not made for every 5 blocks but is always ready to be made up to 5 blocks in M93 mode.

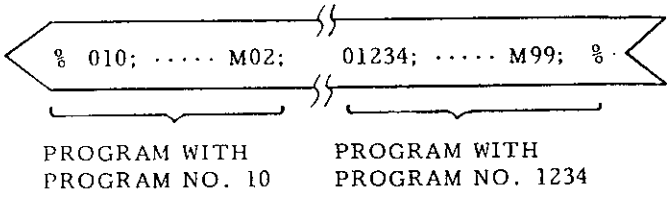
2.2 PROGRAM NUMBER AND SEQUENCE NUMBER

2.2.1 PROGRAM NUMBER

Program numbers may be prefixed to programs for the purpose of program identification.

Up to 4 digits may be written after an address character "O" as program numbers. Up to 99 program numbers can be registered in the control, and up to 199 or 999 can be registered employing an option.

One program begins with a program number, and ends with M02, M30 or M99. M02 and M30 are placed at the ends of main programs, and M99 is placed at the ends of subprograms.



ER (or % at ISO code) is punched on the tape at the top and end of the program.

NOTES:

- The blocks for optional block skip such as /M02;, /M30;, /M99; are not regarded as ends of programs.
- To make the reading of M02, M30, and M99 ineffective as a program end, and to make the succeeding ER (EIA) or % (ISO) as a sign of program ends is possible with a parameter change. (#6021$_{D0}$)

2.2.2 SEQUENCE NUMBER

Integers consisting of up to 4 digits may be written following an address character N as sequence numbers.

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

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

NOTES:

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

2.2.3 OPTIONAL BLOCK SKIP (/1 - /9)†

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

EXAMPLE

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

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

```
N 1234 G01 X100;.
```

With "1", "1" may be omitted.

NOTES:

- The optional block skipping process is executed while the blocks are read into the buffer resister. If the blocks have been read, subsequent switching on is ineffective to skip the blocks.
- While reading or punching out programs, this function is ineffective.
- The optional block skip /2 - /9 is an option function.

2.3 COORDINATE WORD

Generally, commands for movements in axis directions and commands for setting coordinate systems are called coordinate words, and coordinate words consist of address characters for desired axes and numerals representing dimensions of directions.

2.3.1 COORDINATE WORD

Table 2.4

Address		Description
Main axes	X, Y, Z	Position or distance in X, Y or Z coordinate direction.
4Th Axis†	A, B, C or U, V, W	These coordinate words are treated as commands in the directions of the 4th axes. A, B and C are used for rotary motion, and U, V and W are used for parallel motion.
Circular interpolation aux. data	Q	Circular arc increment in circle cutting (G12, G13)
	R	Generally, radius values of circles.
	I, J, K	Generally, distances from start point to arc center (in X, Y and Z components).

2.3.2 SIMULTANEOUS CONTROLLABLE AXES OF THREE-AXIS CONTROL

Table 2.3.2 shows simultaneously controllable axes.

Table 2.5

	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.)
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 2.9.5 HELICAL INTERPOLATION.
Manual control	Simultaneous control of X, Y and Z

Note:

Circular arc plane is determined according to the currently effective G codes for plane designation. (G17 to G19)
For details, refer to 2.9.4 CIRCULAR INTERPOLATION (G02, G03).

2.3.3 SIMULTANEOUSLY CONTROLLABLE AXES OF FOUR-AXIS CONTROL†

Table 2.3.3 shows simultaneously controllable axes.

Table 2.6

	Simultaneously controllable axes
Positioning G00	X, Y, Z, and $\alpha^{(1)}$ axes
Linear interpolation G01	X, Y, Z, and $\alpha^{(1)}$ axes
Circular interpolation G02, G03	Two axes,[2][3] XY; YZ, ZX, $X\alpha^{(1)}$, $Y\alpha^{(1)}$, or $Z\alpha^{(1)}$
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 2.9.5 HELICAL INTERPOLATION.
Manual control	One axis, X, Y, Z, or $\alpha^{(1)}$

(1) The α axis represents any one of axes A, B, C, U, V or W, selected as the 4th axis.

(2) Circular arc plane is determined according to the currently effective G codes for plane designation (G17 to G19). For details, refer to 2.9.4 CIRCULAR INTERPOLATION (G02, G03).

(3) For circular interpolation axis α, any one of linear axes U, V, and W should be designated.

2.3.4 4TH 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 and W.

2.3.4.1 ROTARY AXIS (A, B OR C AXIS)

The rotary axis is defined as follows.

Table 2.7

Rotary axis	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, any one of the three axes, A, B and C, is referred to as b-axis.

The unit of output increment and input increment for b-axis is "deg." instead of "mm" used with linear axes. For the other respects, the treatments are the same as those in mm. (Metric system)

Even when inch system is selected by parameter, the values for the b-axis remains "deg." unit. The control does not convert b-axis coordinate commands. However, feedrate command F is converted. (Refer to 2.9.3 LINEAR INTERPOLATION)

2.3.4.2 LINEAR AXIS (U, V OR W AXIS)

The linear axes are defined as follows.

Table 2.8

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 c-axis.

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

When inch system is selected by parameter, input values must be in inches for c-axis.

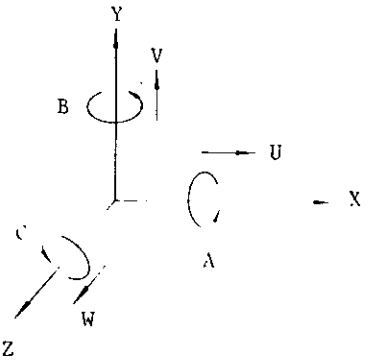


Fig. 2.1 4Th Axis in Right-hand Coordinate System

Table 2.9

	Rotary axis (b)	Linear axis (c)
Least output increment	0.001 deg./ pulse	0.001 mm/ pulse
Least input increment	0.01/0.001 deg.	0.01/0.001 mm
Maximum programmable value	±8388.607 deg.	±8388.607 mm
Rapid traverse rate	deg./min	deg./min
Cutting feedrate	deg./min	deg./min
Manual feedrate	deg./min	deg./min

2.3.5 LEAST INPUT INCREMENT AND LEAST OUTPUT INCREMENT

2.3.5.1 LEAST INPUT INCREMENT

The minimum input units that can be commanded by punched tape or MDI are shown in Table 2.10

Table 2.10 Least Input Increment (#6006D5 = "0.")

	Linear Axis	Rotary Axis†
Metric input	0.001 mm	0.001 deg
Inch input	0.0001 in	0.001 deg

Least input increment times ten can be set by setting parameter #6006D5 at "1."

Input Increment × 10 (#6006D5 = "1.")

	Linear Axis	Rotary Axis†
Metric input	0.001 mm	0.001 deg
Inch input	0.001 in.	0.001 deg

Metric input and inch input can be selected by setting #6001D0.

Notes:

- Selection of metric system or inch system is made by setting (#6001D0).
- Selection of x 1 or x 10 is made by parameter setting (#6006D5).

Tool offset value must always be written in 0.001 mm (or 0.0001 inch, or 0.001 deg†.), and offset is possible in these units.

In 0.01 mm increment system, the following operation must be made in the unit of 0.01 mm.

- Programming for operation in TAPE mode.
- Write operation in MDI mode.
- Programming for operation in MEMORY mode†.
- Program editing operation in EDT mode†.

NOTES:

- If NC tape programmed by 0.001 mm is fed into or stored in an equipment set by 0.01 mm increment, the machine will move ten times the intended dimensions.
- If the increment system is switched when the contents of NC tape are stored in memory, the machine will move by ten times or one tenth of the commanded dimensions.
- When the stored program is punched out on the tape†, the stored figures are punched out "as stored" regardless of switching of the increment system.

2.3.5.2 LEAST OUTPUT INCREMENT

Least output increment is the minimum unit of tool motion. Selection of metric or inch output is made by parameter (#6007D3) setting.

Table 2.11 Least output Increment

	Linear axis	Rotary axis†
Metric output	0.001 mm	0.001 deg
Inch output	0.0001 in.	0.001 deg

2.3.6 MAXIMUM PROGRAMMABLE DIMENSIONS

Maximum programmable dimensions of move command are shown below.

Table 2.12 Maximum Programmable Dimensions

		Linear axis	Rotary axis†
Metric output	Metric input	±8388.607 mm	±8388.607 deg
	Inch input	±330.2601 in.	±8388.607 deg
Inch output	Metric input	±21307.062 mm	±8388.607 deg
	Inch input	±838.8607 in.	±8388.607 deg

In incremental programming, input values must not exceed the maximum programmable value.

In absolute programming, move amount of each axis must not exceed the maximum programmable value.

Note: The machine may not function properly if a move command over the maximum programmable value is given. The above maximum programmable values also apply to distance command addresses I, J, K, R, Q in addition to move command addresses X, Y, Z, α.

The accumulative value must not exceed the maximum accumulative values shown below.

Table 2.13 Maximum Accumlative Values

	Linear axis	Rotary axis†
Metric input	± 99999.999 mm	± 99999.999 deg
Inch input	± 9999.9999 in.	± 99999.999 deg

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

2.4 TRAVERSE AND FEED FUNCTIONS

2.4.1 RAPID TRAVERSE RATE

2.4.1.1 RAPID TRAVERSE RATE

The rapid traverse motion is used for the motion for the Positioning (G00) and for the motion for the Manual Rapid Traverse (RAPID). The traverse rates differ among the axes since they are dependent on the machine specification and are determined by the machine tool builders. The rapid traverse rates determined by the machine are set by parameters in advance for individual axes. When the tool is moved in rapid traverse in two or three axial directions simultaneously, motions in these axial directions are independent each other, and the end points are reached at different times among these motions. Therefore, motion pathes are normally not straight.

For override rapid traverse rates, Fo, 25%, 50% and 100% of the basic rapid traverse rates, are available. Fo is a constant feed rate set by a parameter (#6231).

2.4.1.2 SETTING RANGE OF RAPID TRAVERSE RATE

For each axis, rapid traverse rates can be set at some suitable multiple of 7.5 mm/min (or deg/min).

The maximum programmable rapid traverse rate is 24,000 mm/min. However, respective machine tools have their own optimum rapid traverse rates. Refer to the manual provided by the machine tool builder.

2.4.2 FEEDRATE (F FUNCTION)

With five digits following an address character F, tool feedrates per minute (mm/min) are programmed.

The programmable range of feedrates is as follows.

Table 2.14

		Format	Feedrate (Feed/min) range
Metric output	Metric input	F40	F1.- F8100. mm/min
	Inch input	F31	F0.1 - F313.0 in./min
Inch output	Metric input	F50	F1.- F20574. mm/min
	Inch input	F31	F0.1 - 810.0 in./min

The maximum feedrate is subject to the performance of the servo system and the machine system. When the maximum feedrate set by the servo or machine system is below the maximum programmable feedrate given above, the former is set by a parameter (#6228), and whenever feedrates above the set maximum limit are commanded, the feedrate is clamped at the set maximum value.

F commands for linear and circular interpolations involving motions in simultaneously controlled two axial directions specify feedrates in the direction tangential to the motion path.

EXAMPLE G91 (incremental)

G01 X4000 Y3000 F500 ;

With this command,

$F = 500 = \sqrt{300^2 + 400^2}$ (mm/min)

X component

Y component

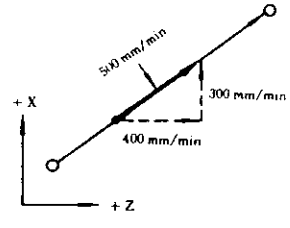


Fig. 2.2

G03 X···· Y···· I···· F200 ;

With this command,

$F = 200 = \sqrt{fX^2 + FY^2}$ (mm/min)

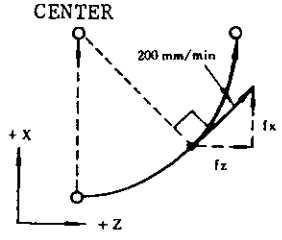


Fig. 2.3

F commands for linear interpolations involving motions in simultaneously controlled three axial directions specify feedrates also in the direction tangential to the motion path.

EXAMPLE

With G01 X··· Y··· Z··· F400 ;

$F = 400 = \sqrt{fX^2 + fY^2 + fZ^2}$ (mm/min)

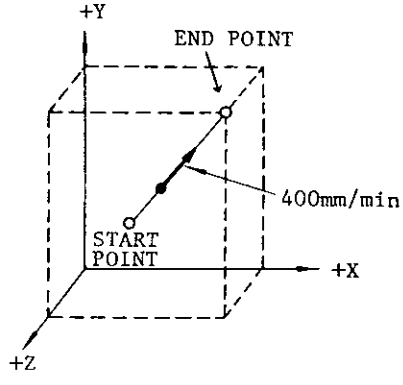


Fig. 2.4

F commands for linear interpolations involving motions in simultaneously controlled four axial directions specify feedrates also in the direction tangential to the motion path.

$F\ (mm/min) = \sqrt{fX^2 + fY^2 + fZ^2 + f\alpha^2}$

NOTES:

- If F0 is programmed, it is regarded as a data error. (alarm code "030)
- Do not program F commands with minus numerals, otherwise correct operation is not guaranteed.

EXAMPLE

F-250 ; ..... wrong

2.4.3 FEEDRATE 1/10

The feedrate programmed by F commands can be converted to 1/10-th value with a parameter setting as follows.

- When parameter #6020 D0 or D1 is set to "1," the feedrates range becomes as shown below.

Table 2.15

		Format	Feedrate (Feed/min) range
Metric output	Metric input	F41	F1.0 - F8100.0 mm/min
	Inch input	F32	F0.01 - F313.00 in./min
Inch output	Metric input	F51	F1.0 - F20574.0 mm/min
	Inch input	F32	F0.01 - F810.00 in./min

- When parameter #6020 D0 or D1 is set to "0," the feedrate range returns to normal.

2.4.4 F 1-DIGIT PROGRAMMING†

(1) Specification of a value 1 to 9 that follows F selects the corresponding preset feedrate.

(2) Set the feedrate of each of F1 to F9 to the setting number shown in Table 2.16(a).

(3) 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 increment or decrement value per pulse (F1-digit multiply) to the parameters listed in Table 2.16(b).

As a result of this operation, the contents of the setting number of the F1-digit feedrate are changed.

(4) Upper Limit of Feedrate

Set the maximum feedrate of F1-digit designation to the following parameter. If a value greater than the usual maximum cutting rate (the contents of #6228) is set, it is governed by the contents of #6228.

Table 2.16(a)

F command	Setting No. for F1-digit speed
F1	#6561
F2	#6562
F3	#6563
F4	#6564
F5	#6565
F6	#6566
F7	#6567
F8	#6568
F9	#6569

Setting "1" = 0.1 mm/min

Table 2.16(b)

F command	Parameter No. for F1-digit multiply
F1	#6141
F2	#6142
F3	#6143
F4	#6144
F5	#6145
F6	#6146
F7	#6147
F8	#6148
F9	#6149

Setting "1" = 0.1 mm/min/pulse

Table 2.16(c)

Parameter No.	Meaning
#6226	Max speed of F1 to F4
#6227	Max speed of F5 to F9

NOTES:

a. When this feature is installed, the specifying 1 to 9 mm/min by the usual F function is not allowed. Specifying 10 mm/min or more is allowed usually.

b. If F0 is specified, error "030" will be caused.

c. When DRY RUN switch is on, the rate of dry run is assumed.

d. For F1-digit specification, the feedrate override feature is invalid.

e. The feedrate stored in memory is retained after the power is turned off.

2.4.5 FEED PER REVOLUTION+

When a spindle pulse generator is installed+, the feedrate per revolution function can be used, and a new G code will be used for this function. Before F function for feedrate per revolution is given, G code of F group shown below must be designated. When the power supply is switched on, G94 is in effect.

G code of 05 group	Function
G94	Feed per minute (mm/min) designation
G95	Feed per revolution (mm/rev) designation

Since F code is modal, the code is effective until the next F code is given. However, when G94/G95 are switched over, F code must be designated again.

After the designation of G95, the feedrate-of-tool per spindle-revolution can be given by 4 digits following F. The command range of the F code is as follows.

Table 2.17

		Format	Feedrate (Feed/min) range
Metric output	Metric input	F22	F0.01 - F99.99 mm/rev
	Inch input	F13	F0.001 - F3.936 in./rev
Inch output	Metric input	F22	F0.01 - F99.99 mm/rev
	Inch input	F13	F0.001 - F3.936 in./rev

1/10 of minimum unit of F code programmable range can be programmed by setting the contents of parameter #6020D2,D3 to 1. See Table 2.18.

Table 2.18

		Format	Feedrate (Feed/min) range
Metric output	Metric input	F23	F0.001 - F99.999 mm/rev
	Inch input	F14	F0.0001 - F3.9366 in./rev
Inch output	Metric input	F23	F0.001 - F99.999 mm/rev
	Inch input	F14	F0.0001 - F3.9366 in./rev

However, the programming of feedrate is restricted by the spindle speed (S) as shown below.

$$F \times S \leq \text{Feedrate upper limit value or clamp value}$$

(mm/rev or in./rev) (rpm) (mm/min)

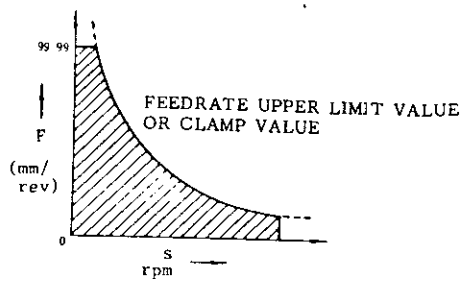


Fig. 2.4 Restriction of Feedrate (F) and Spindle Speed (S)

2.4.6 AUTOMATIC ACCELERATION AND DECELERATION

Acceleration and deceleration for rapid traverse and cutting feed are automatically performed.

2.4.6.1 ACCELERATION AND DECELERATION OF RAPID TRAVERSE AND MANUAL FEED

In the following operation, the pattern of automatic acceleration and deceleration is linear.

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

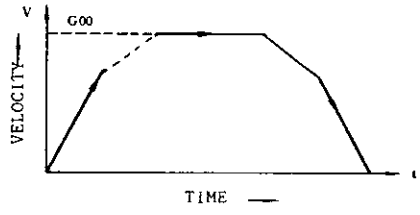


Fig. 2.5

Rapid traverse rate and acceleration/deceleration constant of rapid traverse rate can be set by parameter. (#6280 to #6301)

2.4.6.2 ACCELERATION/DECELERATION OF FEEDRATE

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

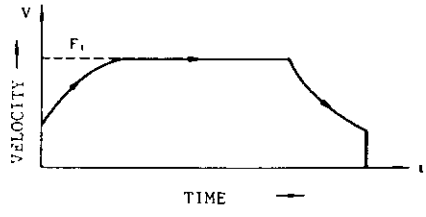


Fig. 2.6 Exponential acceleration deceleration

- Feedrate time constants are set at 4 msec intervals and feedrate bias is set at 2kpps intervals by parameters. (#6092, #6093)

Note:

The automatic acceleration/deceleration parameters are set to the optimum values for the respective machines. Do not change the setting unless this is required for special purposes.

2.5 SPINDLE-SPEED FUNCTION (S-FUNCTION)

2.5.1 S 2-DIGIT PROGRAMMING

The spindle speed is specified by two digits following the address S (S00 to S99).

For each S code and its corresponding spindle speed (rpm), refer to the machine tool builder's manual.

When a move command and an S code are issued in a block, whether the S command is executed together with the move command or after the completion of tool move depends on the machine tool builder. Refer to the machine tool builder's manual.

EXAMPLE

```
G00 S11 M03 ;
          ... S command
              Spindle CW      } S11: Effective
    X... Y... Z... ;
G01 Z... F... ;
    .
    .
G00 X... Y... Z... M05 ; ... Spindle stop
                ... M03 ;   } S11: Effective
    X... Y... Z... ;
G01 Z... F... ;
    S22 ;
    X... Y... F... ;        } S22: Effective
    .
    .
```

NOTE: The two-digit BCD output is sent to the machine when S and two-digit command is issued.

2.5.2 S 5-DIGIT PROGRAMMING†

2.5.2.1 S 5-digit Programming

With five digits written after an address character S (S □□□□□), spindle speeds in rpm are directly commanded.

The programmed speeds become effective upon the inputting of an S-command-completion-input-signal (SFIN).

When an S command is programmed in the same block with M03 (spindle forward run) or M04 (spindle reverse run), the execution of the next block starts only after the spindle speed reaches to the level specified by the S command, in most cases. However, for exact behavior of the machine tool under consideration, refer to the machine tool builder's manual.

EXAMPLE

S 1000 M03 ;

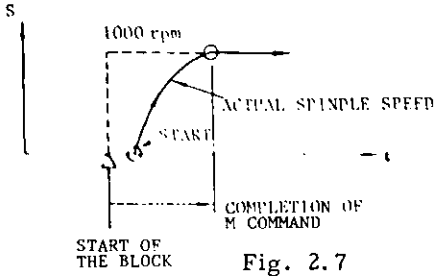


Fig. 2.7

The S commands are modal, and when it is programmed once, it remains effective until another command is programmed. Even when the spindle is stopped by a M05, the S command remains effective. Therefore, when the spindle starts again with an M03 (or M04), the spindle runs at the speed specified by the S command.

When the spindle speed is to be changed by a new S command after it is started with an M03 or M04, attention must be paid to the selected spindle speed range.

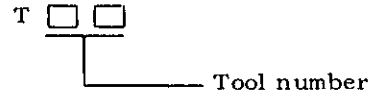
NOTES:

- The lower limit of programmable S commands (S0 and other S commands for near 0 values) is determined by the spindle motor of the machine tool. Refer to the machine tool builder's manual. Do not program minus values as S commands.
- When the control is equipped with the S 5-digit command function, spindle speed overriding is possible. That is, override speeds between 50 and 120% of the commanded spindle speed can be obtained at intervals of 10%.

2.6 TOOL FUNCTION (T-FUNCTION)

2.6.1 T 2-DIGIT PROGRAMMING

Two digits, following the address T, specify the tool number. Leading zeros may be omitted.



The figures used for the designation of tool number are determined by the machine. Refer to the machine tool builder's manual.

When a move command and a T code are issued simultaneously,

- the two commands are executed simultaneously, or
- the T command is executed upon completion of the execution of the move command,

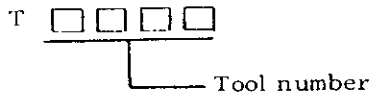
depending on the design of the machine.

For this, refer to the machine builder's manual.

- T codes are modal, and therefore, once they are given, they remain effective until another T command is given.
- T code commands are generally for making automatic tool changers (ATC) to select the tool number to be used next. Therefore, they can be given without regard to the G, H or D codes which are for offsetting for the length or radius of the tool currently in use.

2.6.2 T 4-DIGIT† PROGRAMMING

Four digits following the address T specifies the tool number.



- Leading zeros may be omitted.
- This tool code is the same as the T 2-digit codes, except for the increased number of digits.

2.7 TOOL COMPENSATION

2.7.1 OUTLINE OF TOOL COMPENSATION

The tool compensation function is in the following three modes.

- Tool length compensation

  This function is for compensating for differences in tool length, and is effective in the Z axis direction. Specified length compensation becomes effective from the block in which G43 or G44 is programmed together with an H code. It is cancelled with H00 or G49.

- Tool position offset (for simple compensation for tool radius)

  This function is for compensating for errors in machined dimensions to be introduced by the radius of tools. It is effective in the X, Y, and Z (4th†) axis directions. It is effective only for the block in which G45 - G48 is programmed together.

- Tool radius compensation C† (for compensating for tool radius effects with complicated machining contours)

  This function is for compensating for the tool radius effect with any given machining contours. It is effective in X-Y, Y-Z, and Z-X planes. It becomes effective from the moment G41, or G42 is commanded together with a D code, and is cancelled by G40.

NOTE: For details of these compensations functions, refer to 2.9 PREPARATORY FUNCTION (G-FUNCTION).

2.7.2 TOOL OFFSET MEMORY

For the three groups of offsets, all the necessary offset values must be stored in memory beforehand.

Up to 99 offset values can be stored in the tool offset memory.

Offset value storage: 99 max.

The setting range of offset values is as follows.

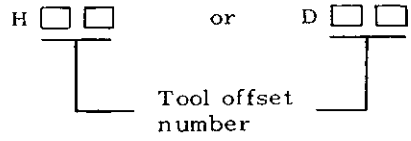
	Linear axis	Rotary axis†
Metric input	0 - ± 999.999 mm	0 - ± 999.999 deg
Inch input	0 - ± 99.9999 inch	0 - ± 999.999 deg

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

For the procedures of storing values into memory, refer to 4.3.5 WRITING OF TOOL OFFSET VALUE.

2.7.3 H- AND D-FUNCTION (H, D CODES)

Two digits, following the address H or D, specify tool offset numbers.



The tool offset numbers 01 through 99 directly correspond to the 99 offset-value memory numbers. That is, when certain numbers are designated, the corresponding offset values stored in the offset memories will be used to offset the tools.

Tool offset numbers 00 (H00 or D00) have differend meanings depending on the respective offset functions. For details, refer to the descriptions on the respective G functions.

H- and D-codes must be used properly according to their functions.

Code	Function
H code	Tool length offset
D code	Tool position offset, Tool radius compensation

The tool offset numbers 01 through 99 can be used freely in combination with the both H and D codes.

However, for programming case, it is recommended to divide the numbers into H code part and D code part.

H codes: H01 to H30

D codes: D31 to D99

Table 2.19

Offset method	G code	H or D code	Offset value memory
Tool length offset	G43 G44 G49	H [0][1]	No. / Offset value 01 02 03 04
Tool position offset	G45 G46 G47 G48	D [9][9]	. . . . 96 97 98 99
Tool dia. compensation C (Intersection computing system)	G40 G41 G42		

2.8 MISCELLANEOUS FUNCTIONS (M-FUNCTION)

The miscellaneous function is specified with the address M and three digits. The function of each M code (M00 to M89) is determined by the machine, except for several M codes. Refer to the machine tool builder's manual for the function of M codes except for the following M codes concerned with the control.

2.8.1 M CODES FOR STOP (M00, M01, M02, M30)

· M00 (Program Stop)

This code, when given in automatic operation mode, stops the automatic operation after the commands in the block containing M00 have been completed and M00 R signal is fed. The program may be continued by pressing the CYCLE START button.

· M01 (Optional Stop)

M01 performs the same function as program stop M00 whenever the OPTIONAL STOP switch is on. When the OPTIONAL STOP switch is off, the M01 code is disregarded.

· M02 (End-of-Program)

M02 is used at the end of program. When given in automatic operation mode, this code stops the automatic operation after the commands in the block containing M02 have been completed. Although the control is reset in most cases, the details are determined by the machine. Refer to the machine tool builder's manual.

· M30 (End-of-Tape)

M30 is given at the end of tape. When given in automatic operation mode, this code stops the automatic operation after the commands in the block containing M30 have been completed. In addition, in most cases, the control is reset and rewinds the tape (or memory). Since the details are determined by the machine, refer to the machine tool builder's manual.

NOTES:

· When M00, M01, M02 or M30 is given, it prevents the control from reading ahead the next block of information. The single decoded signal is fed in addition to the 2-digit BCD output for M codes.

· Whether M00, M01, M02 or M30 executes spindle stop, coolant off or some other executions, refer to the machine tool builder's manual.

· Whether the control is automatically reset or rewinds the tape (or memory), is determined by the following state.

(a) Input signal of the control "EOP" (internal reset input) is wired for "ON" or not.

(b) Input signal of the control "RWD" (rewind input) is wired for "ON" or not.

Refer to the machine tool builder's manual.

2.8.2 M CODES FOR INTERNAL PROCESSING (M90 TO M199)

M90 through M199 are used only for internal processing. Even when they are programmed, no external output signal (BCD and decoded output) is sent.

M90†: Program interrupt off

M91†: Program interrupt on

M92†: Multi-active register off

M93†: Multi-active register on

M94: Mirror image off

M95: Mirror image on

M96†: Tool radius compensation C: circular path mode

M97†: Tool radius compensation C: intersection computing mode

M98: Subroutine program call

M99: Subroutine program end

M100 to 199: Used for enhansed codes

2.8.3 PROGRAM INTERRUPTION ON/OFF (M90, M91)†

The following M codes are used for the program interruption function.

M code	Meaning
M90 ◥	Program interrupt function OFF
M91	Program interrupt function ON

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◥.

· M91 P ····· ;

During the time from this command to an M90 command, whenever a program interruption signal is received, the program under execution is interrupted (if the machine is in motion, it is stopped after deceleration), and the a jump is made to the program the number of which is written after the P.

· M90

With this command, the program interrupt function is cancelled.

2.8.4 MULTI-ACTIVE REGISTERS ON/OFF (M93, M92)†

M code	Meaning
M92 ◥	Multi-active register off
M93	Multi-active register on

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◥.

· M93:

During the time from this command to M92, the control assumes the 5 blocks-advance-reading mode. Namely, up to 5 blocks of data are read in advance for the following operation.

Inter-block stoppage can be eliminated when the program is so made that the operation time of advance reading of 5 blocks is longer than processing time of advance reading of next 5 blocks of data.

· M92:

This command cancels 5 blocks-advance-reading mode.

NOTE: In tool radius compensation C mode, the blocks without move command can be contained(up tc two blocks). Under this condition, 7 blocks, including the two blocks, may be read in advance.

2.8.5 MIRROR IMAGE·ON/OFF (M95, M94)

M code	Meaning
M94 ◥	Mirror image OFF
M95	Mirror image ON

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◥.

· With these codes, mirror image operation can be started and stopped at any desired point in the program. These commands must always be made on a single block.

· M94 and M95 are modal. When the power supply is turned on, M94 (OFF) is in effect.

· The axis on which mirror image is to be effected is specified by setting #6000D0 to D3 (or mirror image axis designation switch.) For this procedure, refer to 6.1.25 MIRROR IMAGE AXIS SELECT SWITCH.

· When M95 is given, the subsequent blocks will control the machine in mirror-image fashion, that is, movements in the specified coordinate direction will be reversed.

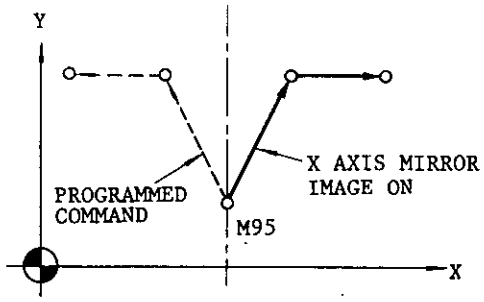


Fig. 2.8

With both the absolute and increment move commands, the same mirror image effect will be obtained. The block including M95 command constitutes the mirror point.

· When M94 is given, mirror image effect will be cancelled on the subsequent blocks. Mirror image operation must be started and cancelled at the same position.

NOTES:

· When G28 or G29 is used to change tools or for ending machining processes, make sure to cancel the mirror image effect by means of M94. If mirror image effect is not cancelled when G28 or G29 is given, an error "058" is shown.

· The mirror image effect is not effective on the offset movement resulting from the tool length offset function.

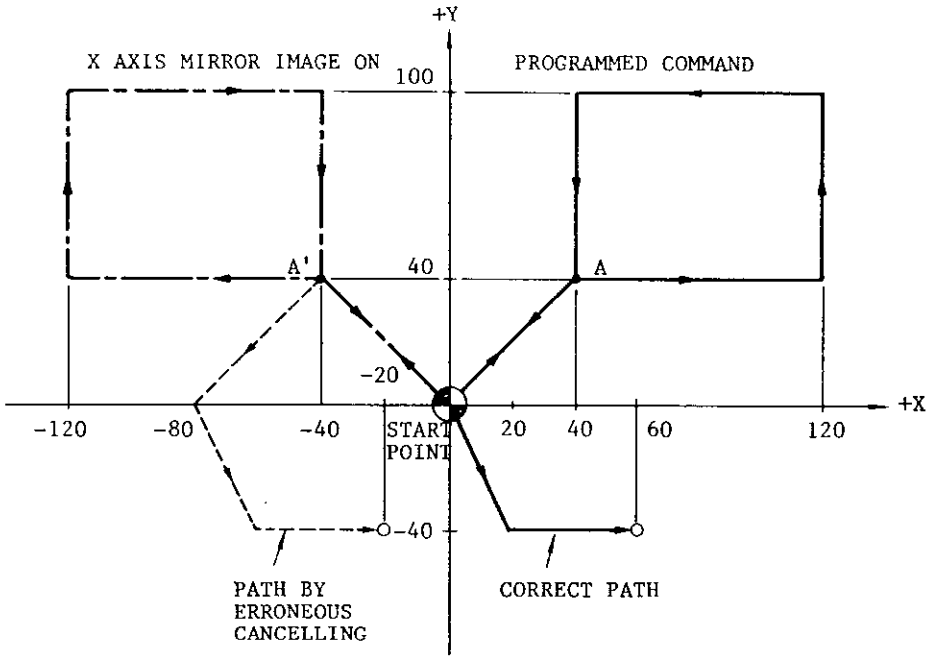
· Do not switch the designation of mirror image axis during operations under M95 (ON) mode.

· Displayed current position by POS key in mirror image fashion indicates the actual motion of tool. Displayed data by COM key show programmed commands.

· Program must be made so that mirror image operation starts and stops at the same position. If the start position and the stop position are not the same, movements of the machine after cancelling mirror image will be shifted by the difference between both positions.

· When the operation is reset, it will be in M94 mode.

EXAMPLE




```
N01  G92  X0  Y0  ;
N02       M95  ;          .....    Mirror image on.
N03  G90  G01  X4000  Y4000  F300 ;
N04       X12000  ;
N05       Y10000  ;
N06       X4000  ;
N07       Y4000  ;
N08       X0  Y0  ;
N09       M94  ;          .....    Correct mirror image off.
N10       X2000  Y-4000  ;
N11       X6000  ;
```

If "M94 ;" is programmed here, the tool moves on the dotted line.

Fig. 2.9

2.8.6 CIRCULAR PATH MODE ON/OFF ON TOOL RADIUS COMPENSATION C (M97, M96)†

M code	Meaning
M96	Tool radius compensation circular path on.
M97	Tool radius compensation circular path off. (Execution of intersection point)

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◥.

- In the G41 or G42 cutter radius compensation mode, when M96 is given, the tool moves along a circular path around a corner with an angle of 180° or larger. In the M97 mode, the tool does not move along a circular path at the corner, but moves along two intersecting straight lines intersecting at a calculated intersecting point shifted from the programmed contour by the tool radius.

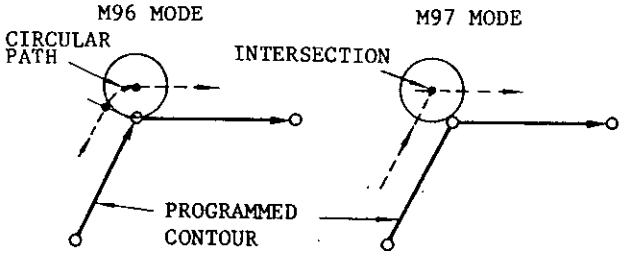


Fig. 2.10

- M96 and M97 are modal. When the power is turned on, M96 takes effect.

- M96 and M97 are effective on the following move command blocks.

```
G01  X··· Y··· F ;        }
                          }  Effective from
(G01) X··· Y··· M96 ;     }  the corner of
                          }  these 2 blocks

G01 X··· Y··· F··· ;      }
M96 (or M97) ;            }  Effective from
(G01) X··· Y··· ;         }  the corner of
                          }  these 2 blocks
```

2.8.7 SUBROUTINE PROGRAM (M98, M99)

With this function, call of subroutine programs which have been numbered and stored in advance is made and executed as many times as desired.

- The following M codes are used for this function.

Table 2.26

M code	Meaning
M98	Call of subroutine program
M99	Subroutine program end

- Call of subroutine program (M98)

M98 P··· L··· ;

With this command, call of the subroutine program with the number specified after P is made and is executed number of times specified after L. When no L code is programmed, the subroutine is executed once.
Subroutine programs can be nested up to 4 times.

- Format of subroutine program (M99)

Subroutine programs are written in the following format, and are stored in the part program memory in advance.

```
O ··· ;                    ← Program No.
····· ;
····· ;
····· ;
M99 ;                      ← Subroutine program end.
```

- Automatic return command from subroutine program

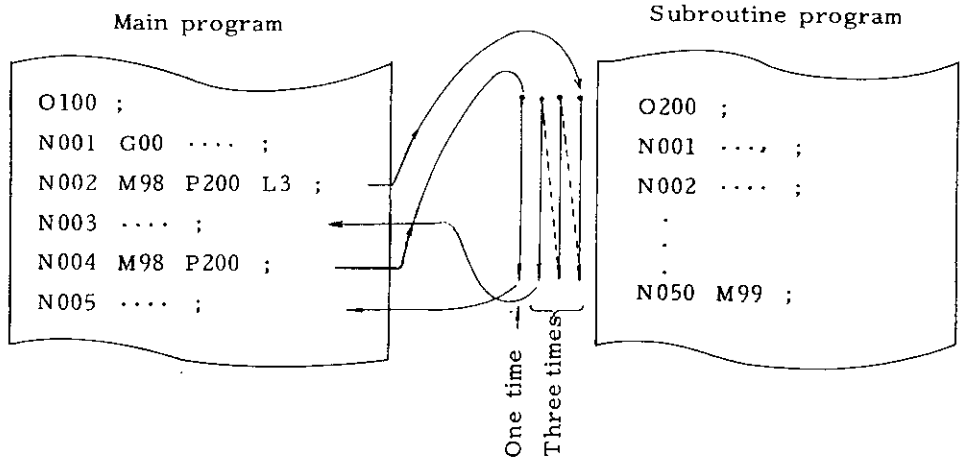
M99 ;

At the end of subroutine programs, M99 is written in a block of its own. When M99 is commanded in the subroutine program which has been called by M98, the execution of the main program is automatically restarted at the block immediately following the M98 block.

2.8.7 SUBROUTINE PROGRAM (M98, M99) (CONT'D)

EXAMPLE

Call of subroutine program and execution of it are made in the sequence shown below.



· Special use of M99

M99 P··· ;

With this command, the main program does not return to the block following the M98 block after executing the subroutine program, but returns to the block with a sequence No. specified by the P code.

NOTES:

· If the program number specified by the P code is not found, this is regarded as an error "041."

· While a subroutine program is repeated L times, the number of remaining repetitions may be displayed. For details, refer to 4.3 DISPLAY AND WRITING OPERATION.

· This function is usable when subroutine programs are stored in the part program memory. The main program can either be commanded from NC tape or the part program memory.

· When the nesting of subroutine programs is attempted more than 4 times, an error state is caused.

· Commanding M99; in main program will return the execution of the program to the head of the main program and control endless operation.

2.8.8 OTHER M CODES

For using M codes, other than those mentioned above, refer to the machine tool builder's manual.

Table 2.20 Typical Examples of M codes for Machine

M code	Meanings	Remarks
M03	Spindle forward running	M03 and M04 are not switchable. M05 (stop) must be intermediated.
M04	Spindle reverse running	
M05	Spindle stop	
M08	Coolant on	
M09	Coolant off	

When these M codes are given in a block together with move command, whether the M commands are executed simultaneously or after completion of move command, are determined by the type of machine. Refer to the machine tool builder's manual.

2.8.9 2ND MISCELLANEOUS FUNCTION (B-FUNCTION)†

B-function and T 4-digit† commands cannot be used simultaneously.

Three digits following the address B give index table positions.

The actual index positions corresponding to the respective B codes depend on the machine tool builder. For this, refer to the specifications of the machine tool builder. When a B-function is given together with a move command in one block,

- the B command is executed simultaneously with the move command, or
- B command is executed after the execution of the move command,

depending on the design of the machine tool. For this refer to the specifications of the machine tool builder.

B codes are modal. When one B code is given, it remains effective until another B-command is given.

NOTES:

- B function standard interface is in 3-digit BCD output.
- With MDI operation on NC panel, "B" is used to specify address for B codes. Therefore, when the control has B-function, the 4th axis control cannot be added.

2.9 PREPARATORY FUNCTION (G-FUNCTION)

2.9.1 LIST OF G CODES AND GROUPS

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

Ordinary G codes are either non-modal G codes marked with * or modal G codes belonging to groups 01 through 15. Those G codes belonging to the division B are included in the basic specifications.

- The G codes belonging to groups 01 through 15 are modal, remaining effective when once commanded until other G codes in the same group will be commanded.
  The G codes in the * group are non-modal, and are effective only for the block in which they are commanded.
- The G codes belonging to groups 01 through 15 may be programmed twice or more in the same block. However, when different G codes in the same group are programmed, the last appearing G code only is effective.
- The G codes belonging to the * group can not be programmed twice or more in a block. They must be programmed only once in a block of its own.
- 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 cancelled and these codes in the group 09 becomes G80.
- When the RESET key is depressed during the execution of a tool compensation C (G41, or G42) or a canned cycle, they become respectively G40 or G80 which cancells the programmed commands.
- 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.

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

- The transition state of the following G codes can be changed by setting of parameters.
- The G codes belonging to the following groups can be specified as to the state immediately after the application of supply power. (#6005D0 - D4)

Group	Initial state	Parameter
01	G00 or G01	#6005D2
03	G90 or G91	#6005D0
05	G94 or G95	#6005D1
08	G43, G44 or G49	#6005D3, D4

- G codes in the 01 group may selectively be changed to G00 or kept unchanged after resetting. (#6005D6)
- During the execution of G92, Display may selectively be made. (#6005D5).

2.9.1 LIST OF G CODES AND GROUPS (CONT'D)

Table 2.21 List of G codes

G code	Group	Function	B: Basic O: Optional
G00	01	Positioning	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	O
G21		Metric input designation	O
G22	04	Stored stroke limit ON	O
G23		Stored stroke limit OFF	O
G27	*	Reference point check	O
G28		Automatic return to reference point	O
G29		Return from reference point	O
G30		Return to 2nd, 3rd, 4th reference point	O
G31		Skip function	O
G33	01	Thread cutting	O
G36	*	Automatic centering	O
G37		Automatic centering	O
G38		Z-axis reference surface offset	O
G40	07	Tool radius compensation cancel	O
G41		Tool radius compensation, left	O
G42		Tool radius compensation, right	O
G43	08	Tool length compensation, plus direction	B
G44		Tool length compensation, minus direction	B
G49		Tool length compensation, 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	O
G53	*	Temporary shift to machine coordinate system	O
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
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	*	Programming of absolute zero point	B
G94	05	Feed per minute (mm/min) designation	O
G95		Feed per revolution (mm/rev.) designation	O
G98	10	Return to initial point for canned cycles	O
G99		Return to point R for canned cycles	O
G100	16	High-speed cutting cancel	O
G101		High-speed cutting in sequential processing mode ON	O
G102		High-speed cutting in processing mode ON	O

Notes:

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 twice or more in a block. They must be programmed only once in a block of its own.

2. The codes marked with ◥ is automatically selected at power on or reset.

2.9.2 POSITIONING (G00, G06)

G00 X··· Y··· Z··· (α±···) ;
(where α = A, B, C, U, V, or W)

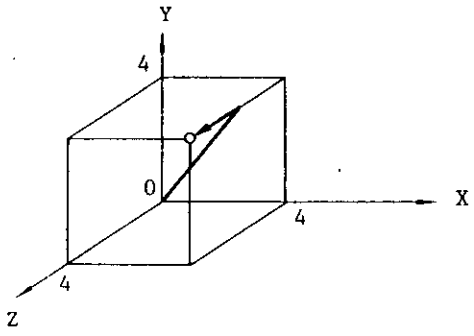
With this command, the tool is sent to the specified position in rapid traverse motions along the 3 axes (4 axes+) simultaneously. If any of the coordinate positions is not specified, the machine does not move along that coordinate axis.

The rapid traverse rate for the respective axes are inherent to the machine tool. Refer to the machine tool builder's manual.

Motions in the respective axis directions are independent each other, and therefore, the resultant tool path is not necessarily straight. When programming tool positioning commands, take care to avoid the possibility of tool and workpiece interference.

EXAMPLE

G01 X4000 Y4000 Z4000 ;



Rapid traverse rate

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

Fig. 2.11

G00 is a modal G code belonging to the 01 group.

Error detect OFF positioning (G06)

G06 X··· Y··· Z··· (α±···) ;

With this command, the same positioning motions are initiated as with a G00 command, with the following exceptions.

- After the completion of the positioning motion with G06 block, the program advances to the next block in the ERROR DETECT OFF mode (Note). Therefore, the tool path at the corner is rounded.
  G06 is a non modal G code belonging to the * group, and therefore, it is effective only in which it is programmed.

NOTE

- G00 commands position the tool in the ERROR DETECT ON mode, which means that the program advances to the next block only after the servo lag pulses are decreased below the permissible level, and this is detected by the control. With this command, therefore the corner of the workpiece is machined sharp.
- With the ERROR DETECT OFF mode commanded by G06, the program advances to the next block immediately after the completion of pulse distribution.

2.9.3 LINEAR INTERPOLATION (G01)

G01 X··· Y··· Z··· (α±···) F··· ;

where α = A, B, C, U, V, or W

With this command, the tool is moved simultaneously in the three (four+) axial directions resulting in a linear motion. When a certain axis is missing in the command, the tool does not move in the axial direction of that axis.
Feedrate is specified by an F code the feedrate in the component axial directions are so controlled that the resultant feedrate becomes the specified feedrate.

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

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

The end point can be programmed either in absolute coordinates or in incremental values with G90 or G91 respectively. (Refer to 2.9.31 Absolute/Incremental Programming (G90, G91)).

If no F code is given in the block containing the G01 or in preceding blocks, the block constitutes an error "030."

EXAMPLE

G01 X4000 Y4000 Z4000 F100 ;

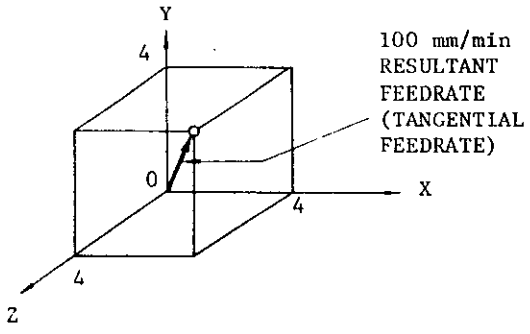


Fig. 2.12

Where the optional 4th axis is a 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.

2.9.3 LINEAR INTERPOLATION (G01) (CONT'D)

Table 2.22

F-function			In minimum F command unit	
			Feedrate in basic three axes	Feedrate of rotary axes
Metric output	Metric input	F40	1 mm/min	1 deg/min
	Inch input	F31	0.1 in./min	2.54 deg/min
Inch output	Metric input	F50	1 mm/min	0.3937 deg/min
	Inch input	F31	0.1 in./min	1 deg/min

Note: Feedrate of linear 4th as the same as that of basic three axes.

2.9.4 CIRCULAR INTERPOLATION (G02, G03)

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

XY plane G17 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ X··· Y··· $\left\{\begin{matrix}R\cdots\\I\cdots\ J\cdots\end{matrix}\right\}$ F··· ;

ZX plane G18 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Z··· X··· $\left\{\begin{matrix}R\cdots\\K\cdots\ I\cdots\end{matrix}\right\}$ F··· ;

YZ plane G19 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Y··· Z··· $\left\{\begin{matrix}R\cdots\\J\cdots\ K\cdots\end{matrix}\right\}$ F··· ;

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

G02: Clockwise

G03: Counter-clockwise

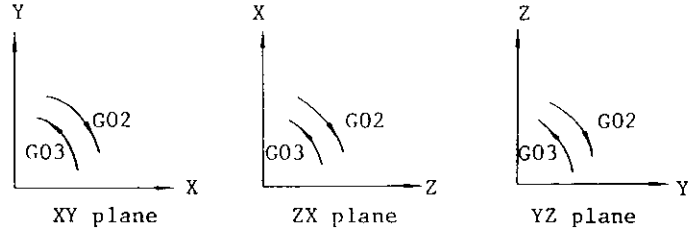


Fig. 2.13

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

G17: XY plane or X4 plane†

G18: ZX plane or Z4 plane†

G19: YZ plane or Y4 plane†

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 the switching of the power supply.

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

G17 G90 G03 X1500 Y4000 I-3000 J-1000 F150 ;

(a) Absolute command with (G90)

G17 G91 G03 X-4000 Y2000 I-3000 J-1000 F150 ;

(b) Incremental command

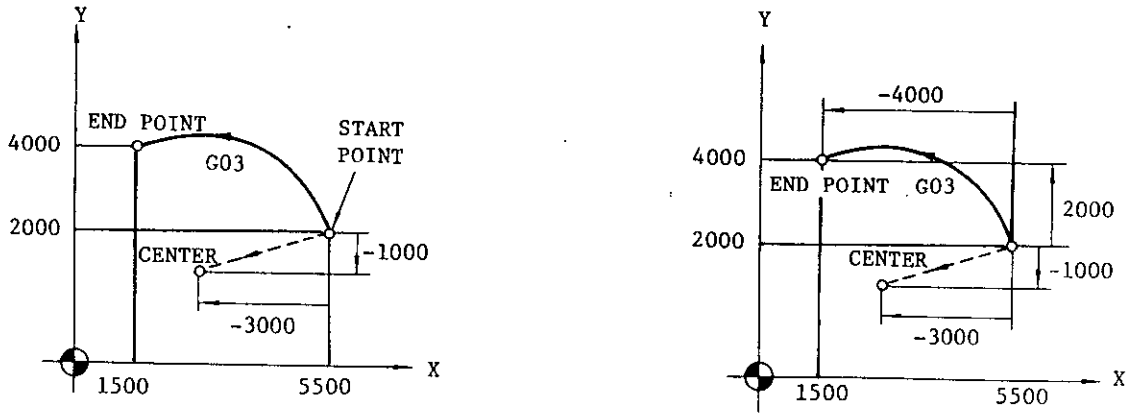


Fig. 2.14

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

G17 G02 X··· Y··· R±··· F··· ;

In this case,

when R > 0, a circular arc with the center angle less than 180°, and
when R < 0, a circular arc with the center angle larger than 180° are specified.

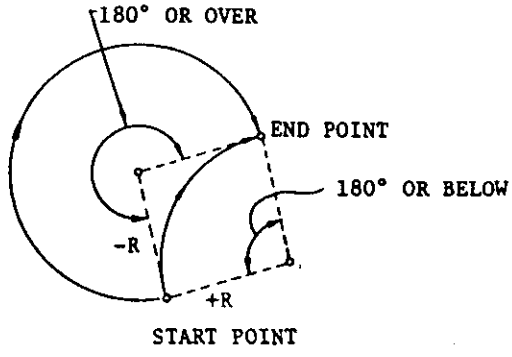


Fig. 2.15

2.9.4 CIRCULAR INTERPOLATION (G02, G03) (CONT'D)

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.

When a linear 4th axis option is used, circular interpolation is possible in the Xα , Zα , and Yα planes in addition to the XY, YX, and ZY planes (where α = U, V, or W)

Xα plane G17 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ X··· α··· $\left\{\begin{matrix}R···\\I··· \quad J···\end{matrix}\right\}$ F··· ;

Zα plane G18 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Z··· α··· $\left\{\begin{matrix}R···\\K··· \quad I···\end{matrix}\right\}$ F··· ;

Yα plane G19 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Y··· α··· $\left\{\begin{matrix}R···\\J··· \quad K···\end{matrix}\right\}$ F··· ;

NOTE:

· G17 G02 X··· $\left\{\begin{matrix}R···\\I··· \quad J···\end{matrix}\right\}$ F··· ;

Where address characters for the 4th axis is missing as in the above command, the XY plane is automatically selected. Circular interpolation cannot be performed on the axes including rotary 4th axis.

Circular pathes covering two or more quadrants can be programmed in a single block. A complete closed circle can also be programmed.

EXAMPLE

G00 X0 Y0 ;

G02 X0 Y0 I1000 J0 F100 ;
... complete circle

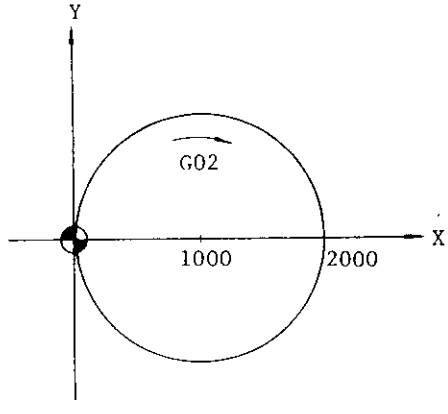
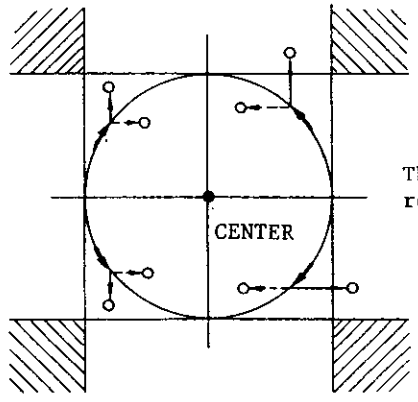


Fig. 2.16

When the coordinate values of the end point of a circular path is not exactly on the correct circular path due to calculation errors, etc., correction is made as shown below.
Points 0 are commanded as end point. (See the figure below.)



The end point is represented by "∘."

Fig. 2.17

When the end point is programmed in the hatched areas shown above, no alarm state is created, but the tool will keep on rotating. Especially when tool compensation is applied, coordinate values of the point and the center must be programmed accurately.

2.9.5 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 G17 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ X··· Y··· $\left\{\begin{matrix}R···\\I··· \ J···\end{matrix}\right\}$ Z(α)··· F··· ;

(b) For ZX plane G18 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Z··· X··· $\left\{\begin{matrix}R···\\K··· \ I···\end{matrix}\right\}$ Y(α)··· F··· ;

(c) For YZ plane G19 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Y··· Z··· $\left\{\begin{matrix}R···\\J··· \ K···\end{matrix}\right\}$ X(α)··· F··· ;

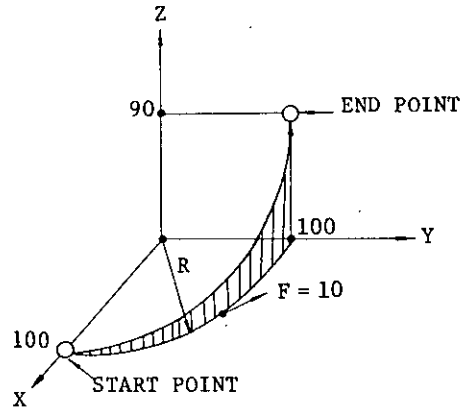
(d) For Xα plane G17 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ X··· α··· $\left\{\begin{matrix}R···\\I··· \ J···\end{matrix}\right\}$ Z··· F··· ;

(e) For Zα plane G18 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Z··· α··· $\left\{\begin{matrix}R···\\K··· \ I···\end{matrix}\right\}$ Y··· F··· ;

(f) For Yα plane G19 $\left\{\begin{matrix}G02\\G03\end{matrix}\right\}$ Y··· α··· $\left\{\begin{matrix}R···\\J··· \ K···\end{matrix}\right\}$ X··· F··· ;

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

EXAMPLE



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

Fig. 2.18

NOTES:

- The circular arc should be within 360°.
- As long as above note (a) is satisfied, the start and end points can be taken at any time.
- The feedrate F means the tangential speed on the plane of circular interpolation. Therefore, the speed (F') in the direction of linear interpolation is as follows.

$$F' = F \times \frac{\text{(Length covered by linear interpolation)}}{\text{(Length of circular path)}}$$

- Tool radius compensation C† can be applied only to the circular path on the plane of circular interpolation.

2.9.6 DWELL (G04)

G04 P··· ;

This command interrupts feed for the length of time designated by the address P.

Dwell is programmed as an independent block.

The maximum length of time which can be designated with address P is as follows.

Format	Dwell time (P programmable range)
P53	0 - 99999.999 sec

The value does not depend on metric/inch input or metric/inch output.

EXAMPLE

G04 P2500 ;

Dwell time: 2.5 sec.

Two types of dwell can be selected by parameter:

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.

2.9.7 EXACT STOP (G09, G61, G64)

- Exact stop (G09)

When a block containing G09 is executed, the program advances to the next block after completing a block in the Error Detect On mode (Note a). This function is used when sharp corners are desired. G09 is non-modal, and is effective only in the block in which it is contained.

2.9.7 EXACT STOP (G09, G61, G64) (CONT'D)

· Exact stop mode (G61)

When once G61 is commanded, all the following blocks will be completed in the Error Detect On mode before proceeding to the next block.

· Exact stop mode cancel (G64)

This G command is for cancelling the effect of G61.

NOTES:

· In the Error Detect On mode, the program proceeds to the next block only after the number of servo delay pulses is found to have decreased below a permissible limit following the complete distribution of circular interpolation command pulses.

· In the G09 and G61 off modes, the program proceeds to the next block immediately after the complete distribution of the pulses of ordinary linear and circular interpolations, and therefore, because of the servo delay, tool pathes are rounded at the corner. This mode is called "Error Detect Off" mode.

· For rapid traverse, the Error Detect On and Off modes are controlled only by G00 and G06, and not by the above G codes.

2.9.8 TOOL OFFSET VALUE DESIGNATION (G10)

With a G10 command, correction of tool offset values and work coordinate system can be made as follows.

· Designation of tool offset value (G10)

Normally, tool offset values are written in by MDI. On the other hand, with a program G10 P··· R··· ;, (where P = tool offset number and R = tool offset value), any programmed offset values can be replaced by a designated value.
When G10 is commanded in the G90 mode, R is regarded as an absolute value.
When G10 is commanded in the G91 mode, R is regarded as an incremental value.

· Changing work coordinate system†

Corresponding to G54 through G59, separate work coordinate systems are set up as setting data in advance.

G10 Q2 Pn X··· Y··· Z··· α··· ;

(where Q2 is used to discriminate from tool offset value designation and a means to set up a work coordinate system. Pn (n = 1 to 6) corresponds to the work coordinate system n to be set up.)

For P1··· G54

For P2··· G55

.
.
.

For P6··· G59 corresponds.

With the above command, data of any desired work coordinate system can be changed.

2.9.9 CIRCLE CUTTING (G12, G13)†

This is a canned cycle includes a complete series of movements for machining a circle in a single block. It includes the following functions.

· Format

G12(G13) I··· D··· F··· ;

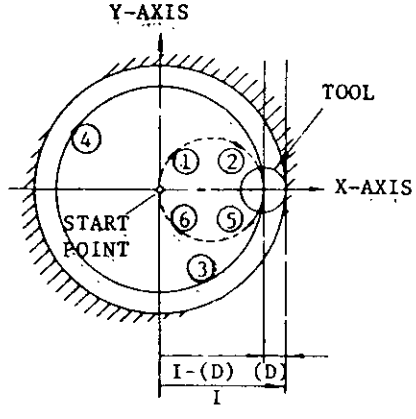
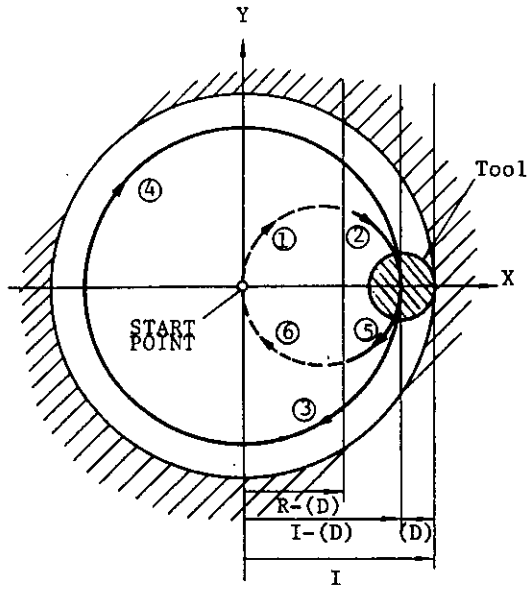


Fig. 2.19

· Designation of rapid traverse section R

G12 (G13) I··· R··· D··· F··· ;

With this command, a circular bore is machined as shown below. Numerals following an address character R specifies rapid traverse sections.



Tool path

G12: ① → ② → ③ → ④ → ⑤ → ⑥

G13: ⑥ → ⑤ → ④ → ③ → ② → ①

(D) represents a set value of tool radius compensation.

G12: Clockwise (CW)

G13: Counterclockwise (CCW)

I: Radius of finished circle (incremental value with sign)

R: Rapid traverse section (incremental value with sign)

D: Tool radius compensation No.

F: Cutting feed rate

Fig. 2.20

· Automatic calculation of rapid traverse section

G12(G13) I··· J··· D··· F··· ;

With this command, when depth of cut (incremental) is designated by numerals following an address character J in place of R, the tool rapid traverse section within which the tool can move at rapid traverse rate without making contact with the stock is calculated automatically. J is programmed without a plus or minus sign.

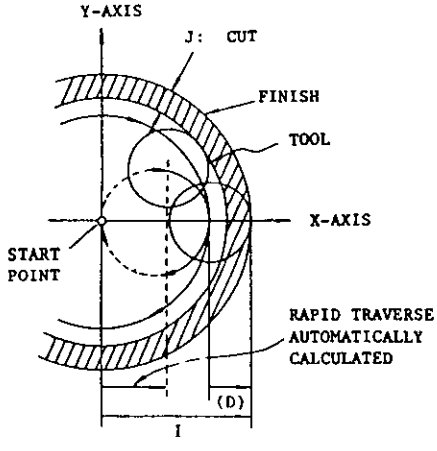


Fig. 2.21

NOTES:

Tool rapid traverse section can be set at $\frac{n}{10} \cdot i$ (n = 0, 1, 2 ··· 10) pitch for a returning semicircle of circle bore.

· Commanding repeated circle designation

G12(G13) I··· D··· L··· F··· ;

With this command, the circular bore surface can be executed L times.

· Commanding spiral circle Q, K

G12(G13) I··· D··· K··· Q··· F··· ;

With this command, the tool is moved along a spiral before finally finishing a circular hole, as shown below. For the sake of simplification, the diagram shows as if the tool has a zero radius (D = 0). Q (radius increment) must be programmed without sign.

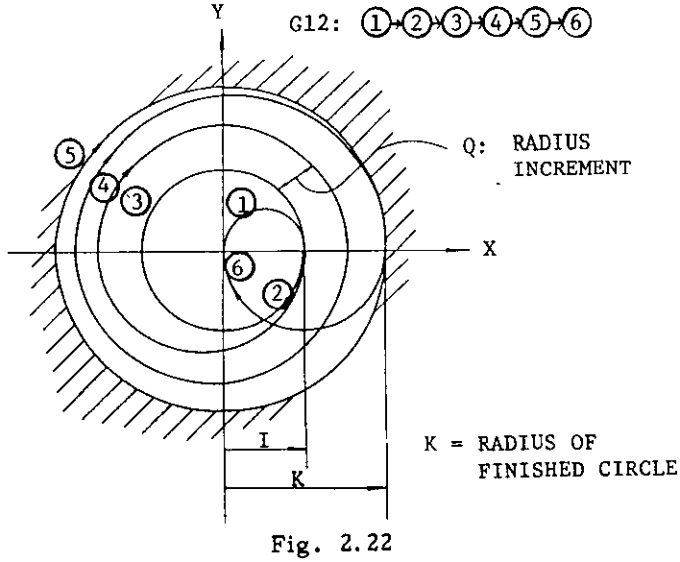
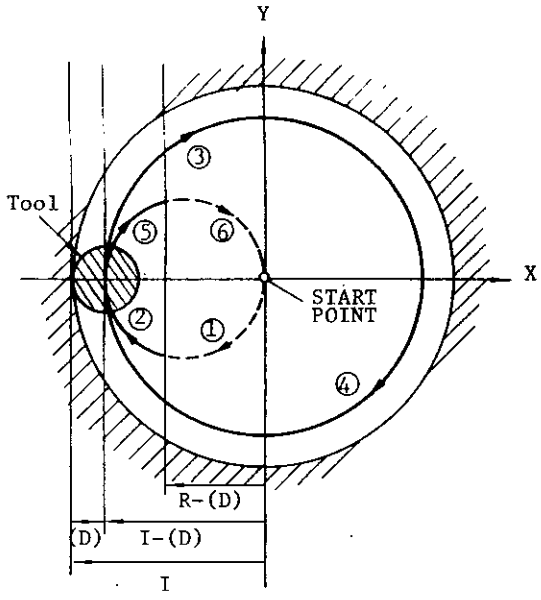


Fig. 2.22

2.9.9 CIRCLE CUTTING (G12, G13)† (CONT'D)



Tool path

G12: ① → ② → ③ → ④ → ⑤ → ⑥

G13: ⑥ → ⑤ → ④ → ③ → ② → ①

(D) represents a set value of tool radius compensation.

G12: Clockwise(CW)

G13: Counterclockwise (CCW)

I: Radius of finished circle (incremental value with sign)

R: Rapid traverse section (incremental value with sign)

D: Tool radius compensation value

Fig. 2.23

· Combined designation of rapid traverse section, repeated circle and spiral circle.

Rapid traverse section, repeated circle designation and spiral circle can be commanded in combination as shown below.

G12(G13) I··· R··· (or J···) K··· Q···
L··· F··· ;

NOTES:

· Circle cutting is possible only on the XY plane.

· The tool speed in the rapid traverse section is set by parameter #6225.

· With a circle cutting command (G12, G13), the tool is offset for its radius compensation without the use of G41 or G42 (tool radius compensation). When using G12 or G13, cancel tool radius compensation with G40.

· In the explanation above, only motions in the + direction of X-axis is considered. With proper use of signs for I, J, K and D codes, motions in the - direction of X-axis (symmetrical with respect to Y-axis) can be commanded. In Fig. 2.23, signs of I, R and (D) are minus. However, cutting in the Y-axis direction is impossible.

· I, J, K, R, Q and L codes in circle cutting command are effective only in the block containing them.

· The radius I of finished circle and the rapid traverse section R are subject to the following restriction. When values not in conformity with the restriction are programmed, this is regarded as an error.

$$|R - d| < |I - d|$$

If R-d and I-d have different signs, this is also regarded as an error.

· When programming G12(G13), always specify a tool radius compensation number D. If this is not specified, the tool moves without radius compensation.

EXAMPLE

· G00 Z-4000 ;
G12 I5000 R4000 D10 F300 ; D10 = 10.0mm
G00 Z4000 ;

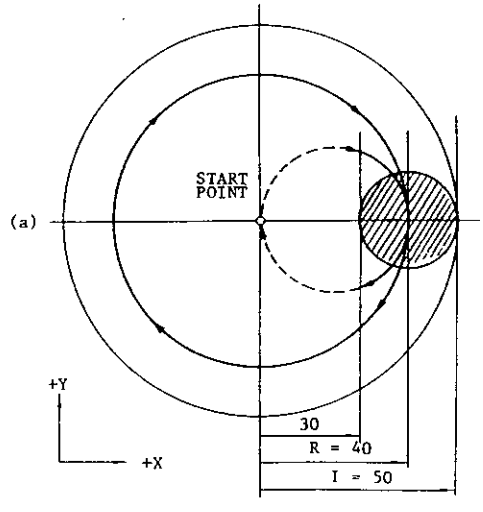


Fig. 2.24

. G00 Z-4000;

G13 I-5000 J700 D15 F300; D15 = -8.0 mm

G00 Z4000;

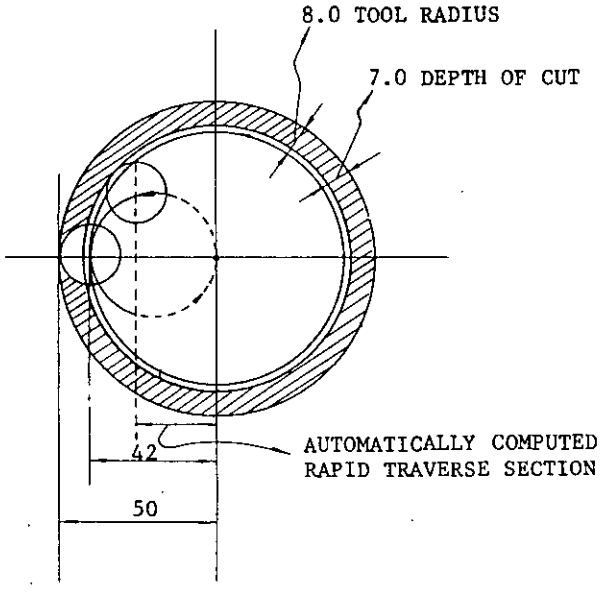


Fig. 2.25

2.9.10 PLANE DESIGNATION (G17, G18, G19)

The plane for making circular interpolation and tool radius compensation is designated by G codes G17/G18/G19.

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

When the 4th 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

α means U, V or W axis.

The move command in each axis can be programmed regardless of the plane designation by G17/G18/G19.

For example, if

G17 Z··· ;

is designated, motion is on Z axis.

The plane for making tool radius compensation by command G41 or G42 is univocally determined by G17, G18 or G19. It is not possible to designate compensation plane including the fourth axis.

The XY plane (G17) is selected when the power is turned on.

2.9.11 INCH/METRIC DESIGNATION BY G CODE (G20, G21)†

Unit of input data are selectively specified by the following G codes between metric and inch.

G code	Input unit
G20	Inch
G21	Metric

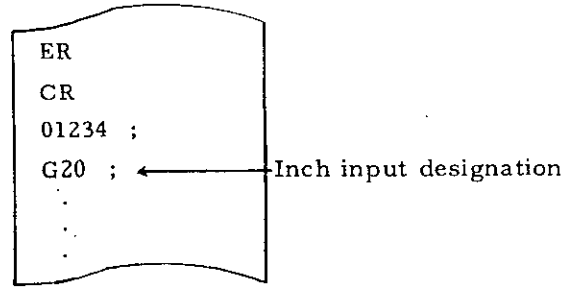
These G codes are programmed at the leading end of a block of its own. If one these G codes are commanded, the units of all the following motions are changed afterwards.

- subsequent programs
- tool offset values
- part of setting parameters
- part of manual movements
- displays

NOTES:

- When G20 or G21 is commanded, the setting of inch/metric selection is changed. Therefore, the state of G20/G21 at the time of power application depends on the setting by parameter #6001 D0.

EXAMPLE



- When G20/G21 selection is commanded in the program, take the following procedure beforehand.

A. When work coordinate system (G54 to G59) is used, return it to base coordinate system.

B. Cancel all tool compensation command. (G41 to G48)

2.9.11 INCH/METRIC DESIGNATION BY G CODE (G20, G21)† (CONT'D)

- Take the following procedure after the command of G20/G21 selection.
  - A. Program absolute zero point for all axes before move command.
  - B. In principle, make the display reset operation when current position display (external) is used.
- The tool offset values are processed differently in the G20 mode and the G21 mode. G20/G21 must be commanded after modifying the tool offset values.

Stored offset values	Processing in G20 (Inch)	Processing in G21 (Metric)
15000	→ 1.5000	15.000 mm

2.9.12 STORED STROKE LIMIT (G22, G23)†

This function is for checking the current tool position during manual or automatic operation for entry into the prohibited area specified by parameters or by G22. If the tool enters a prohibited area, machine operation is stopped and an error sign is displayed.

- 1st prohibited area

The area outside the area specified by a parameter is a prohibited area. Generally, this can be used as a substitute of overtravel checking function. Upper limit point A and lower limit point B are specified by parameters.

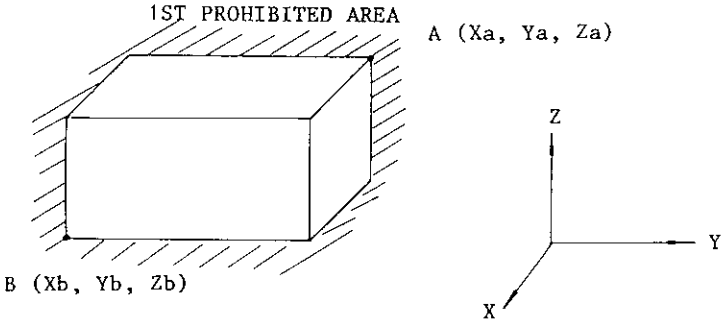


Fig. 2.26

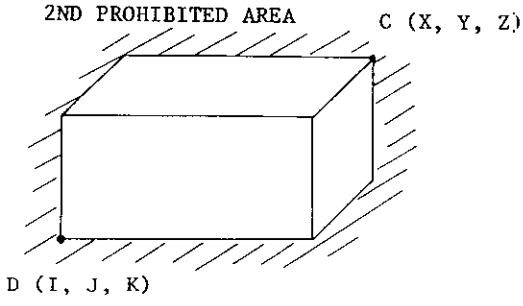
- 2nd prohibited area

The boundary of the 2nd prohibited area is specified by a parameter setting or by G22. The inside or the outside of the boundary can selectively be made a prohibited area by means of parameter setting.

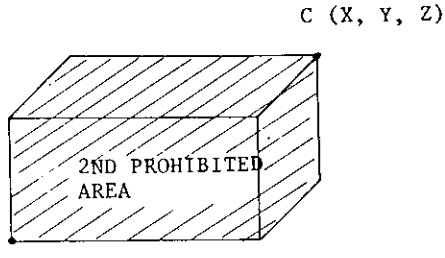
G22 X··· Y··· Z··· I··· J··· K··· ;

C point Upper limit — D point Lower limit

With this command, the checking of the 2nd prohibited area is started, and with G23 ;, the check function is cleared.



Where 2nd prohibited area is outside.



Where 2nd prohibited area is inside.

Fig. 2.27

· The parameter numbers and the setting numbers of these prohibited areas are as follows.

Table 2.23

		X	Y	Z	Division
1st prohibited area	A point	#6600	#6601	#6602	Parameter
	B point	#6606	#6607	#6608	
2nd prohibited area	C point	#6510	#6511	#6512	Setting
	D point	#6513	#6514	#6515	

· The parameters for specifying the inside and the outside of the 2nd prohibited area are as follows.

#6007$_{D0}$	Meaning
"0"	Inside prohibition
"1"	Outside prohibition

· The 2nd prohibited area checking function can also be turned on and off with the following setting number.

#6001$_{D1}$	Meaning
"0"	2nd prohibited area check; off
"1"	2nd prohibited area check; on

NOTES:

· The 1st and the 2nd prohibited area can be specified overlapping each other.

· Boundary lines are included in the prohibited areas.

· All the prohibited areas become effective after a manual return to reference point or a return to reference point by G28 after turning on of the power supply.

· If the tool is in the prohibited area at the time when the prohibited area becomes effective, this is immediately regarded as an error. In this case, turn off the 2nd prohibited area by the setting of the setting number, and either rewrite the data or move the tool out of the prohibited area manually.

· If an alarm state is created by the entry of the tool into the prohibited area, the tool can move only in the returning direction.

· No stored stroke limit can be set to the 4th axis.

· The stored stroke limit checking function may selectively be used or disused during a machine lock operation by the setting of the setting #6001D5.

2.9.13 REFERENCE POINT CHECK (G27)†

This function is for checking the correct return of the tool to the reference point after performing a cycle of operation in accordance with a program which starts at the reference point and ends at the reference point.

G27 X··· Y··· Z··· (α^{+}···) ;

With this command, the tool moves towards the specified position along the three axes (4 axes†) simultaneously but independently, and after the arrival at the specified point, the point is checked for the conformity to the reference point. If any of the axes is omitted in the command, the tool does not move in that axis and no check is made in that axis.

If the point is in conformity with the reference point, the reference point return lamp lights. If the tool is correctly in the reference point in all the axes, automatic operation is performed further, but if the tool is not in the reference point even in one axis, this is regarded as an error (alarm 241 - 244 display), and the automatic operation is interrupted. (Cycle start lamp goes off.)

If G27 is commanded in the tool offset mode, the tool return point is also offset. Cancel the tool offset mode when commanding G27.

2.9.13 REFERENCE POINT CHECK (G27)† (CONT'D)

Reference point as meant here is a fixed point relative to the machine to which the tool returns by the manual reference point return motion or by G28 automatic reference point return motion. Refer to 6.1.15 MANUAL REFERENCE POINT RETURN SWITCH. The mirror image function can be applied to the G27 command. To avoid non-conformity errors, clear the mirror image mode with M94 (Mirror image off) before commanding G27.

2.9.14 AUTOMATIC RETURN TO REFERENCE POINT (G28)†

G28 X... Y... Z... (α†...) ;

With this command, the tool is sent back to the reference point. The tool moves towards the specified points in rapid traverse, and automatically stops at the reference point.

The tool moves simultaneously in up to 3 axes (4 axes†). However, the tool will not move in the direction of the axis for which a coordinate instruction is omitted.

EXAMPLE

G28 X... Y... Z... ;

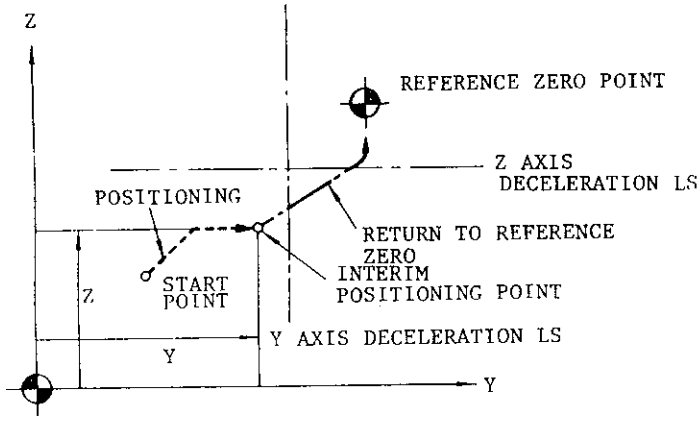


Fig. 2.28

"Return to reference point" involves the same series of motions as the manual return to reference point.

NOTES:

- If G28 is commanded in the tool radius compensation mode (G41, G42) or in a canned cycle, this is regarded as an input error "024."
- If G28 is commanded in the Mirror Image mode (M95), this constitutes an input error "058."
- The tool position offset command is not cancelled by G28. Make it a point to cancel it before commanding G28. If G28 is given in the tool position offset mode, the tool motion by the succeeding program becomes as described below. Care should be taken.

  A. When the succeeding program is made in the incremental mode:
  Tool moves by the amount of incremental value from the reference point. The tool offset is not effective.

  B. When the succeeding program is made in the absolute mode:
  Tool moves to the position which is specified by absolute value and tool offset value.

  C. When G29 is given immediately after the G28:
  By G29 command, the tool moves to the offset interim positioning point and the succeeding motion is made according to the item A and B.

- When returning the tool to the reference point for the first time after turning on the power supply, pay attention to the tool position. Refer to 6.2.1 MANUAL RETURN TO REFERENCE POINT†.

Return to reference point in rapid traverse

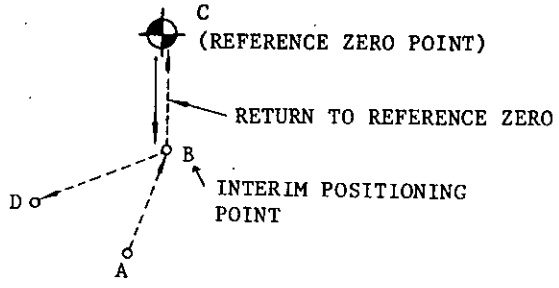
In addition to the above " Automatic Return to Reference Point," "Rapid Traverse Return to Reference Point" function may be incorporated in the control. With this function, the motion sequence is as follows.

- After positioning at an interim positioning point B, the tool directly moves to the reference point in rapid traverse. The returning time is shorter than that with the ordinary return to reference in which deceleration LSs are used in all the axes.
- With the "Rapid Traverse Return to Reference Point," point B may not necessarily be within the reference point return possible area.
- The rapid traverse return to reference point becomes possible only after the tool has been returned once to the reference point in all the axes by manual operation or by G28, following the turning on of the power supply.
- Rapid traverse return to reference point is effective only with G28. Manual return motions are not changed by it.
- Where a 4th axis is used, when no command is given for the 4th axis in a G28 command, and when the tool has been returned to the reference point in the X, Y, and Z axes, the tool moves to the reference point in the rapid traverse return mode. If a command for the 4th axis is included in the command, the tool returns to the reference point in the ordinary return mode, unless the return motions in all the 4 axes have been completed.

2.9.15 RETURN FROM REFERENCE ZERO (G29)†

This code is used to return the tool to its original position after return to reference zero by automatic return to reference zero, along the same path.

```
G28 Y··· Z··· ;        Point A → B → C
  .    Point B         (Reference zero point)
  .
  .
G29 Y··· Z··· ;        Point C → B → D
    Point D
```



Figl 2.29

When G29 is programmed, it is not necessary to consider the distance between point B and C in the program. Particularly when an incremental instruction is used, this is effective for returning tool to the original position, after returning to reference zero.

Movement of C → B and of B → D is made at rapid traverse rate simultaneously along three axes (simultaneously four axes†) by G29. However, in an axis for which a coordinate instruction was omitted, the tool will not move.

If G28 is programmed a number of times, the final coordinates of point B which the last G28 creates is effective for the move of G29.

EXAMPLE 1 (In the case of absolute input)

```
                          Interim point coordinates
                                X     Y     Z
N21 G90 ;
N22 G28 Z1000 Y2000 ; ⟶ (0, 2000, 1000)
N23 G28 X3000 ;       ⟶ (3000, 2000, 1000)
```

```
N24 G29 X-4000 Y-5000 ;
```

Equal to the two blocks =

```
G00 X3000 Y2000 ;
G00 X-4000 Y-5000 ;
```

EXAMPLE 2

```
N31 G91 ;
N32 G28 Z··· ;
N33 G28 X2000 Y4000 ;
N34 M06 ;
N35 G29 X4000 Y-4000 ;
 .
 .
 .
```

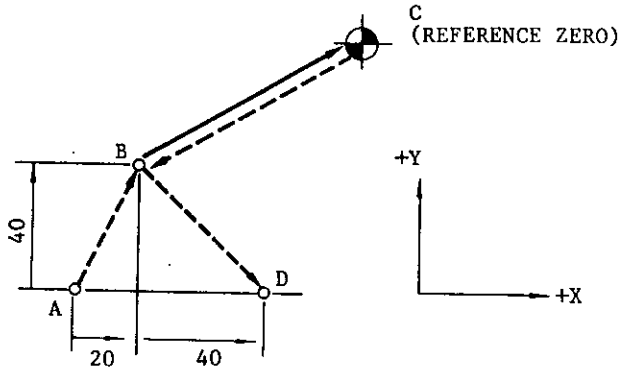


Fig. 2.30

NOTES:

- An input error "024" occurs if G29 is programmed in tool radius compensation mode (G41, G42) or during canned cycle mode (G73, G74, G76, G77, G81 to G89).
- An input error "059" occurs if G29 is given without execution of G28 after the control is turned on.
- In principle, cancel tool offset before programming G28 or G29. If they are programmed when offset is also effective, interim positioning point B will also be offset, and the tool passes point B'.

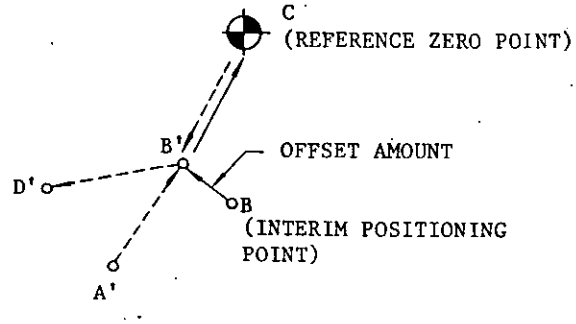


Fig. 2.31

2.9.15 RETURN FROM REFERENCE ZERO (G29)† (CONT'D)

- An input error "058" occurs if G29 is given during mirror image (M95).
- The following command or operation must not be taken because interim positioning point B of G28 does not meet with that of G29.

  (1) The following operations are made between G28 and G29 commands.
  - Setup of absolute zero (G92, ORG key)
  - Machine lock
  - Manual operation at Manual Absolute Off

  (2) G28 and G29 are commanded in the blocks following the block containing M94 which cancels mirror image at the different point from the starting point of mirror image.

  (3) G28 and G29 are commanded after manual operation at Manual Absolute Off.

2.9.16 2ND, 3RD AND 4TH REFERENCE POINT RETURN (G30)†

G30 Pn X··· Y··· Z··· (α†···) ;

(where Pn = P2, P3, P4)

With this command, the tool first moves to an interim positioning point, and then, moves to the 2nd, 3rd or 4th reference point.

P2:	2nd reference point	When P is omitted, the tool moves to the 2nd reference point.
P3:	3rd reference point	
P4:	4th reference point	

If any axis of the coordinate instruction is omitted in the command, the tool remains motionless in the direction of that axis.

Each reference point is specified by the parameters (#6612 to #6629) before hand.

EXAMPLE

```
G30 P3 X30 Y50 ; ···  The tool returns to the
                      3rd reference point
                      moving in the X and Y
                      directions.
```

NOTES:

- Three items except the last one in NOTES of 2.9.14 AUTOMATIC RETURN TO REFERENCE POINT (G28)† apply to G30 in the same manner.
- When G29 is commanded after G30, the tool moves to the designated point by G29 by way of interim positioning point designated by G30. However the interim positioning point is renewed on the axis designated by G30.

2.9.17 SKIP FUNCTION (G31)†

G31 X··· Y··· Z··· F··· ;

With this command, a special linear interpolation is commanded. During the interpolation movement under the command of this program, whenever a skip signal is inputted, the interpolation is interrupted immediately, and the program advances to the next block. From the moment that a skip signal is inputted to the time the control start to process the signal, delay time is less than 0.5 m sec. G31 is non-modal.

EXAMPLE

```
N100 G90 G31 X100.0 Y50.0 ;
N200 G01 X80.0 Y15.0 ;
      .
      .
      .
```

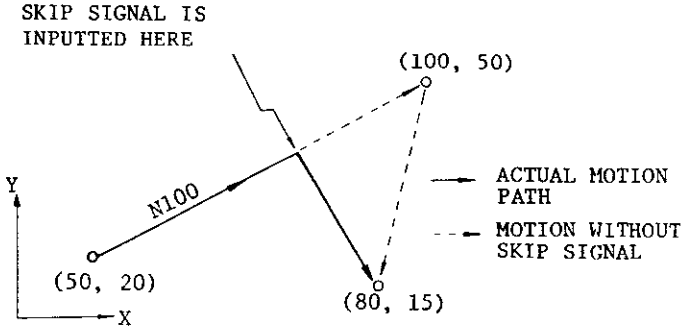


Fig. 2.32

When G31 block is executed without a skip signal being inputted, the machine stops at the end of the block, and the alarm code "087" is displayed.

Feedrate of the tool is set for G31 blocks selectively by one of the following two methods as specified by parameter #6019D4.

- To be specified by F similar to ordinary programs.
- To be set in advance by parameter #6232.

When a skip signal is inputted, the coordinate values at the moment are automatically stored as parameter data.

#6552 ··· storing X coordinate value
#6553 ··· storing Y coordinate value
#6554 ··· storing Z coordinate value
#6555 ··· storing 4th coordinate value

These data can be treated as coordinate data in user macros.

When a skip signal is not given in spite the execution of G31 by setting (#6004D0), the program moves on to the next block automatically.

2.9.18 THREADCUTTING (G33)†

Provided that the machine is equipped with a spindle pulse generator generating reference pulses, and a synchronous feed function (G95), screw threads can be cut with this function.

G33 Z··· F··· ;

With this command, threadcutting is made in the Z direction at a lead of F.
Lead range is shown below.

	Range of lead
Metric	0.01 - 99.99 mm
Inch	0.001 - 3.936 inches

However, lead is subject to the following restriction from the spindle speed.

F(mm/rev) x S(rpm) ≦ 24000(mm/min) or a clamp value

NOTES:

- G33 may be programmed with two or more axes simultaneously as for example in G33YyZzFf ,. In this case, the tool moves along the axis for which the longest distance is specified at the lead specified by F.
- G33 is a G code in the 01 group.
- During a threadcutting operation, feedhold and mode change are impossible. Feedrate override is also ineffective, and the feedrate is locked at 100%.
- G33 can only be commanded in the G95 (mm/rev) mode. If it is commanded in the G94 (mm/min) mode, it causes an alarm.
- G33 can not be programmed in the tool radius compensation C mode. If it is commanded, it causes an alarm.
- Threadcutting command can not include 4 axes. If 4 axes are included in a threadcutting command, an alarm is caused.

2.9.19 AUTOMATIC CENTERING FUNCTION (G36, G37)†

With this function, the spindle is aligned with the center line of the machined bore, with a touch sensor.

- G36 I···(or J···) F··· ;

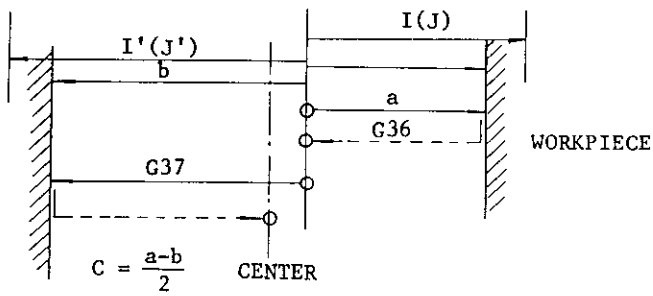
  With this command, the spindle moves in the X (or Y) direction at speed F until the touch sensor makes contact with the workpiece and gives a contact signal. The distance "a" from the start point to the contact point is stored with the function of G37. Then, the spindle returns to the start point in rapid traverse.

- G37 I'···(orJ'···) F··· ;

  With this command, the spindle moves in the X (or Y) direction at speed F until the touch sensor makes contact with the workpiece and gives a contact signal. Then, the spindle returns to the start point in rapid traverse.

$$D = \frac{a - b}{2}$$

where b = distance between G37 start point to contact point.



(I, J, I', J' = Incremental value)

Fig. 2.33

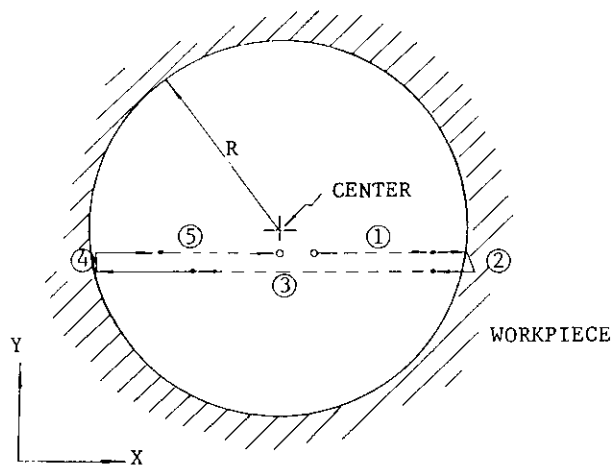
NOTES:

- When no contact signal is obtained during the movement through the specified incremental distances I (or J) and I' (or J'), this constitutes an error.
- When I and J are programmed together in a block of G36 or G37, this constitutes and error.
- When G36 or G37 is commanded in the tool radius compensation C mode or in a canned cycle, this command is treated as an error "034."

2.9.19 AUTOMATIC CENTERING FUNCTION (G36, G37)† (CONT'D)

EXAMPLE A: Automatic bore centering

M06 T10 ; ———	Selection of touch sensor
G00 X··· Y··· ; —	Positioning in X and Y axes to virtual center
Z··· ; ———	Motion in Z direction to measurable position
① G91 Xr ; ———	Motion through virtual radius r in X direction
② G36 Ii F··· ; —	Automatic centering (1) in X direction
③ X-2r ; ———	Motion through virtual diameter-2r in X direction
④ G37 I-i ; ———	Automatic centering (2) in X direction
⑤ Xr ; ———	Completion of centering in X direction
Yr ; G36 Jj ; Y-2r ; G37 J-j; Yr ;	Similar automatic centering in Y direction
G92 X0 Z0 ; ·····	Setting the automatically obtained center point as the coordinate (0,0) point

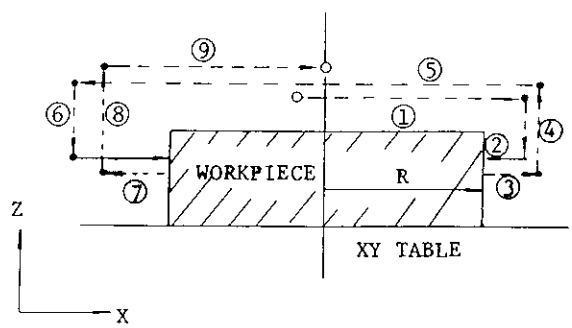


Note: r < R is assumed in the program.

Fig. 2.34

EXAMPLE B: Automatic outer diameter

M06 T10 ; ·····	Selection of touch sensor
G00 X··· Y··· ;···	Positioning in X and Y axes to virtual center
① G01 Xr ; ·····	Motion through virtual radius r in X direction
② Z-z ; ·····	Motion in Z direction to measurable position
③ G36 I-i F··· ; ····	Automatic centering (1) in X direction
④ Zz ; ·····	Retraction in Z direction
⑤ X-2r ; ·····	Motion through virtual diameter-2r in X direction
⑥ Z-z ; ·····	Motion in Z direction to measurable position
⑦ G37 Ii ; ·····	Automatic centering (2) in X direction
⑧ Zz ; ·····	Retraction in Z direction
⑨ Xr ; ·····	Completion of centering in X direction
Yr ; · · · Yr ;	Similar automatic centering in Y direction
G92 X0 Y0 ; ·····	Setting the automatically obtained center point as the coordinate (0,0) point



Note: r > R is assumed in the program.

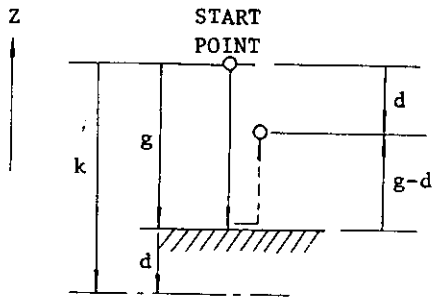
Fig. 2.35

2.9.20 Z-AXIS REFERENCE SURFACE OFFSET (G38)†

With this function, the tool is offset in the Z direction automatically for obtaining accurate dimension relative to a reference surface, with a touch sensor.

· G38 K··· F··· ;

With this command, the spindle moves in the Z direction at a speed F, and stops when the touch sensor gives out a contact signal. Then, d = k - g is calculated, and the spindle returns through g - d in rapid traverse, where k is a value specified by K and g is the distance from the start point to the contact point.



k: INCREMENTAL DISTANCE

Fig. 2.36

NOTES:

· If no contact signal is received during the travel through the specified incremental distance k, an error "087" is caused.

· When G38 is commanded in the tool compensation C mode, or in a canned cycle, this is regarded as an error "024."

EXAMPLE

M06 T11 ; ——— Selection of touch sensor

① G91 G00 Z-z ; —— Approach to the measuring point in Z direction

② G38 K-k F··· ; — Z reference surface compensation

③ Zz ; ——————— Return to the offset position in the Z direction

G92 Z0 ; ——— Setting of Z coordinate origin at this point

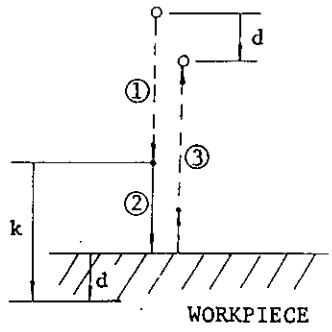


Fig. 2.37

2.9.21 TOOL RADIUS COMPENSATION C (G40, G41, G42)†

It is possible to specify the radius of the tool and to cause automatic tool path offset by this value. Store the offset value (tool radius value) in the offset value memory in advance by MDI, and program the tool offset number correspond to the tool radius value by a D code in the program.

1. Designation of compensation direction and of D code

   Tool radius compensation C is programmed with G41, G42 and is cancelled by G40. G41 and G42 indicate the directions of tool offset with respect to the direction of movement.

G code of tool radius compensation C

G code	Group	Meaning
G40	07	Cancellation of tool radius compensation C
G41	07	Tool radius compensation C, left
G42	07	Tool radius compensation C, right

Note: When the power is turned on, G40 is effective.

Note that the directions of compensation (right, left) indicated above are reversed when the sign of the tool radius value in the offset memory designated by a D code is negative. Make sure to designate a D code in the block containing G41, G42 or in a preceding block. If D00 is commanded, tool radius will be regarded as "0."

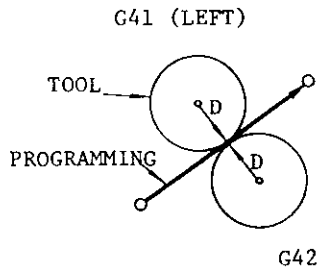


Fig. 2.38

Switching between G41 and G42 can be made in the compensation mode. Details will be given in item 5 below.

2. Designation of compensating plane

The plane in which tool radius compensation is made is designated by G17, G18, G19. They are G codes of 02 group. The XY plane (G17) is in effect at the time power is turned on.

G codes for designation of planes

G code	Group	Meaning
G17	02	XY plane
G18	02	ZX plane
G19	02	YZ plane

Note: When the power is turned on, G17 is effective.

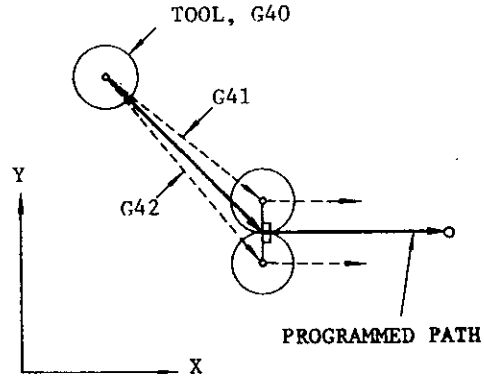
Make sure to designate a G code for plane designation in the same block as that of G41, G42 or in a preceding block. Plane designation cannot be made in a compensation mode. It is not possible to apply tool radius compensation in a plane including the fourth axis†.

3. Method of entry into compensation mode

When G41(G42) is programmed, the tool moves to an offset position with the distance equal to the radius. The offset position is on the normal line at the start point of the block immediately after G41(G42). If no coordinate instruction is programmed in the block of G41(G42), movement is made by the offset value only. Because G41(G42) accompanies a movement, it is necessary to program G00 or G01 for a G code in group A. An input error (alarm code "026) occurs if a G code other than G00, G01 is programmed.

EXAMPLE A

(a) G17 G01 F··· ;
G41(G42) D··· X··· Y··· ;
X··· ;



(b) G17 G01 F··· ;
G41(G42) D··· X··· Y··· ;
G02 X··· Y··· J··· ;

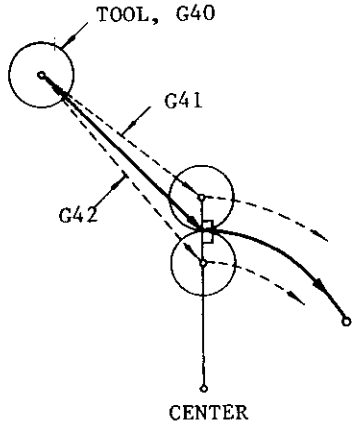


Fig. 2.39

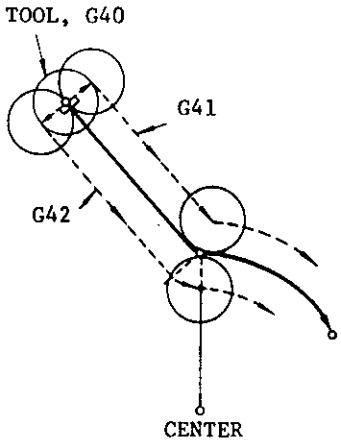
EXAMPLE B

(c) G17 G01 F··· ;

G41(G42) D··· ;

X··· Y··· ;

G02 X··· Y··· J··· ;



(d) G17 G01 F··· ;

G41(G42) D··· ;

G02 X··· Y··· J··· ;

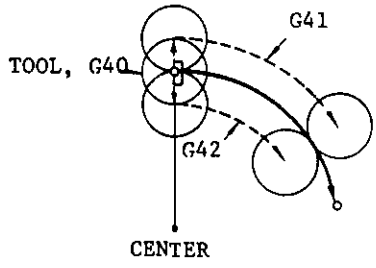


Fig. 2.40

Pay attention to the fact that offset is made on the normal line to the program line determined by the block after G41(G42) at the start point in all of the examples (a) to (d) above. When the movement on the compensation plane is not programmed in the block after G41(G42), the next one block is read ahead and the compensation start with the block. The blocks without move command can be programmed continuously up to two. Input error occurs if move commands on the compensation plane are not programmed in more than three blocks.

When compensation entry is programmed in the G00 mode, positioning movement is made independently by each axis to the offset point. Take care not to make the tool interfere with the workpiece.

4. Movement in compensation mode

When after the tool radius compensation is programmed by G41, G42, the tool moves along the offset path until the instruction G40 is given.

As calculation of the path is automatically made by the control, designate only the shape of the workpiece in the program. The tool path is controlled as follows depending on the angle between blocks.

A. Inside corner (180° or less): Intersection computing type

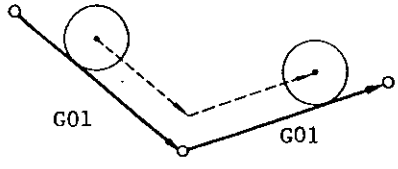


Fig. 2.41

2.9.21 TOOL RADIUS COMPENSATION C (G40, G41, G42)† (CONT'D)

B. Outside corner (over 180°):
Circular path type (in the case of M96)

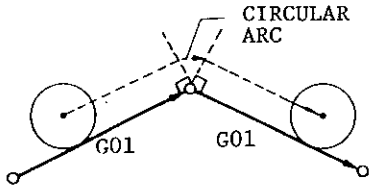


Fig. 2.42

In this case, movement of circular path is included in the former block.

Code M97 can be used to machine the outside corner by the intersection computation, depending on the work. Refer to 2.8.6 CIRCULAR PATH MODE ON/OFF ON TOOL RADIUS COMPENSATION C (M97, M96) for details.

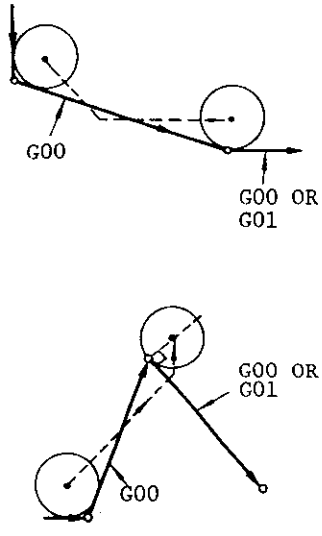
M96 ··· Tool radius compensation circular path ON

M97 ··· Tool radius compensation circular path OFF (execution of intersection computation)

Normally, M96 is used for this operation, however, when there is a possibility of an "overcut" in cutting special shapes with the M96, M97 should be used.

C. Movement in G00 mode

The instruction G00 positions tools independently along each axis toward the final offset position. Care should be taken on the cutter path.



(In M96 mode)

Fig. 2.43

D. Shape requiring care

Do not program a wedge shape having an accute angle.

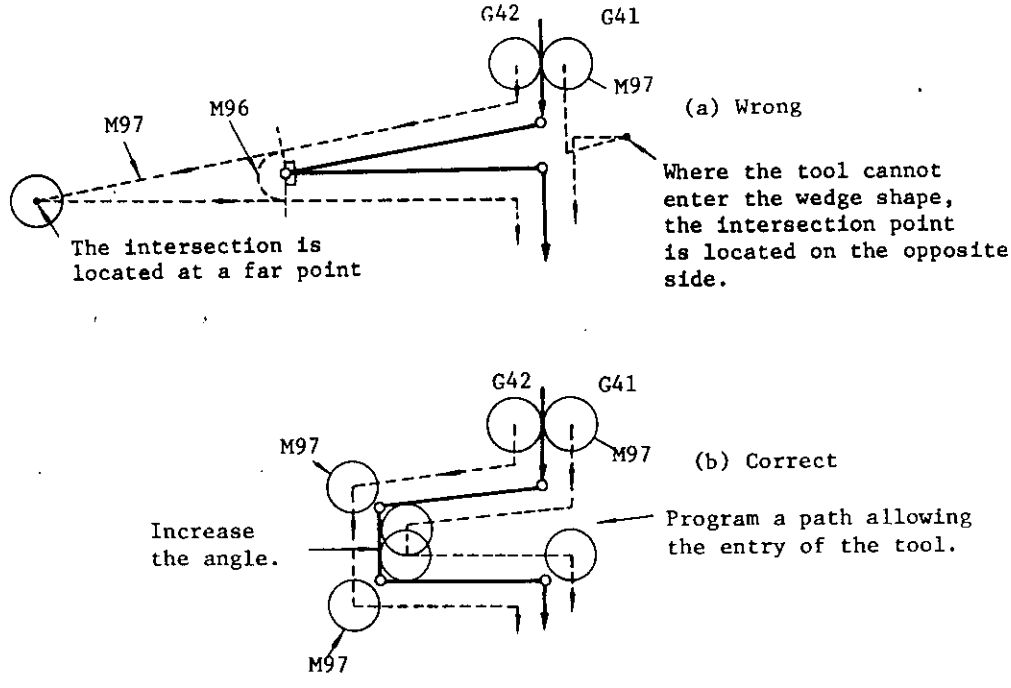


Fig. 2.44

5. Command involving no movement in compensation mode

The control normally reads in advance two blocks during tool radius compensation mode and computes the tool path. If either of these blocks give no coordinate instructions such as G04 (dwell), the control reads the block further ahead and makes computing.

The blocks with no coordinate instructions can be processed continuously up to two blocks. When coordinate instructions are missing in three or more blocks, tool radius compensation becomes impossible and accurate tool path cannot be obtained. Therefore, in a program where G41 or G42 is used, ensure that, after them, three or more blocks without movement command plane will not follow.

2.9.21 TOOL RADIUS COMPENSATION C (G40, G41, G42)† (CONT'D)

EXAMPLE

```
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··· ;
```

Blocks without movement in compensation plane. (When these blocks are within two, machining is made smoothly.)

· If no movement instruction is programmed in three continuous blocks, offset in the block immediate before them is made on the normal line at the end point. Where movement in the compensation plane cannot be programmed in three or more continuous blocks for retracting in the third axis or the like, and offsetting on the normal line is not satisfactory, a dummy block can be inserted by I, J or K.

EXAMPLE

```
N001 G17 G01 G41 X··· Y··· D··· F··· ;
N002 X··· Y··· ;
        .
        .
        .
N010 X··· Y··· ;
N011 I··· J··· ;
N012 Z··· ;
        .
        .
        .
N019 Z··· ;
N020 X··· Y··· ;
        .
        .
        .
N029 X··· Y··· ;
N030 G40 X··· Y··· ;
```

XY plane

Dummy block

Z axis
3 blocks
or above

XY plane

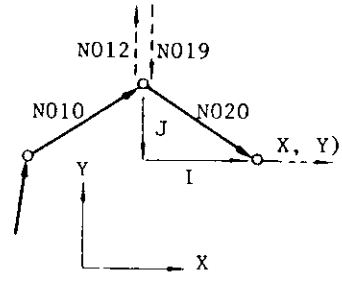


Fig. 2.45

The dummy block is not programmed for actual movement but it only provides data required for tool radius compensation computation. In the example indicated above, an instruction that is the same as the first block (N020) of restarted movement of the XY plane after movement of Z axis is programmed as a dummy by I and J. I, J and K are used as the addresses of this dummy instruction, and they correspond to X, Y, Z axes respectively. Suitably use them in accordance with the plane designation.

I: Dummy for X axis command

J: Dummy for Y axis command

K: Dummy for Z axis command

Programmed in incremental values

If X··· Y··· of N020 is in absolute values in the above example, give an instruction by converting into incremental values.

Note: Make a dummy block as follows if the object of the dummy block is circular interpolation.

EXAMPLE

```
N050 G01 X··· Y··· ;
N050 G01 I(b) J(-a) ;        ——— Dummy block
N052 Z··· ;
N053 Z··· ;   } Z axis
N059 Z··· ;
N060 G03 X··· Y··· I(a) J(b) ;  ——— Circular interpolation
N061 G01 X··· Y··· ;
```

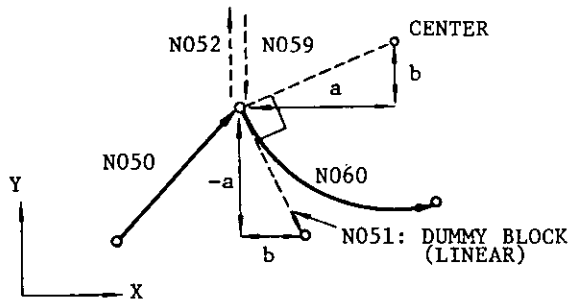


Fig. 2.46

This is, insert a linear dummy block that gives the tangential direction at the start point of the circular interpolation program block as shown above. Exercise care with the sign of the dummy block data depending on the shape of the circle. The tool stops at point A by the dummy block in preparation for the next circular command.

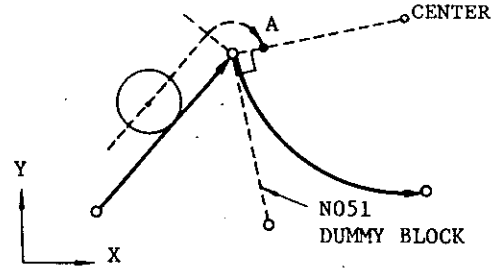


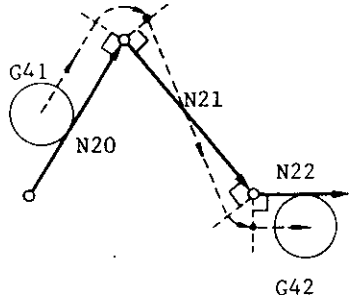
Fig. 2.47

6. Switching between G41 and G42 in compensation mode

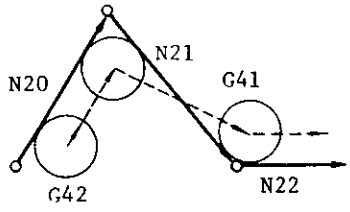
In this compensation mode, direct switching between G41 and G42 is possible without making cancellation with G40.

EXAMPLE

```
N10 G17 G01 F··· ;
N11 G41(G42) D··· ;
      .
      .
      .
N20 G01 X··· Y··· F··· ;
N21 G42(G41) X··· Y··· ;   ——— Block of switching
N22       X··· ;
```



(a) G41 → G42
(M96 mode)



(b) G42 → G41

Fig. 2.48

Note: The movement identical to what is shown above is also obtained when block N21 is split into two blocks as follows.

```
G42 (G41) ;
      X··· Y··· ;
```

7. Change of tool radius value in compensation mode

   New D code commanded in the compensation mode is effective in the block next to the commanded block.

8. Method of cancellation of compensation

   G40 is the command for cancelling tool radius compensation C and for positioning or feeding the tool to just programmed end point. In this case, the tool moves to a point on the normal line at the end point of the block immediately before the block containing G40.

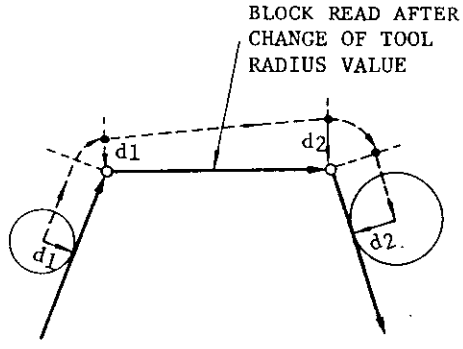


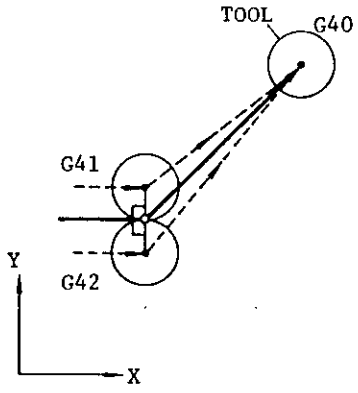
Fig. 2.49

Therefore, no portion will be left unmachined even when a cancellation with sharp angle is programmed. Because G40 accompanies cancelling movement, program it in the G00 or G01 mode like G41, G42. An input error "027" occurs if group A other than G00, G02 is used.

EXAMPLE A

(a) G41 (G42)

```
    .
    .
    .
G01 X···  F···  ;
G40 X···  Y···  ;
```



(b) G41 (G42)

```
    .
    .
    .
G02 X···  Y···  I···  J···  ;
G01 G40 X···  Y···  ;
```

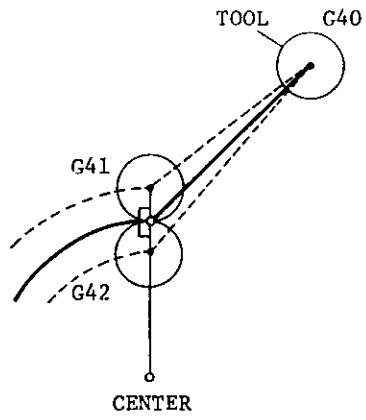
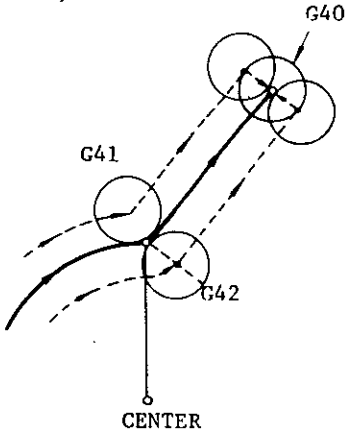


Fig. 2.50

EXAMPLE B

(c) G41 (G42)

```
    .
    .
    .
G02 X···  Y···  I···  J···  ;
G01 X···  Y···  ;
G40 ;
```



(d) G41 (G42)

```
    .
    .
    .
G02 X···  Y···  I···  J···  ;
G01 G40 ;
```

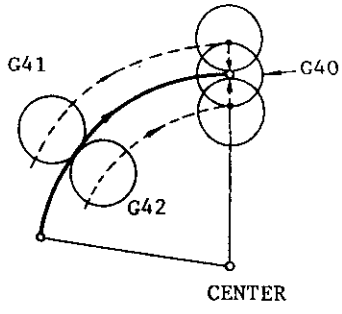


Fig. 2.51

In all cases (a) through (d) described above, the tool reaches the programmed end point via the offset position on the normal line at the end point of the block immediately before G40.

2.9.21 TOOL RADIUS COMPENSATION C (G40, G41, G42)† (CONT'D)

9. Cautions and remarks in tool radius compensation C

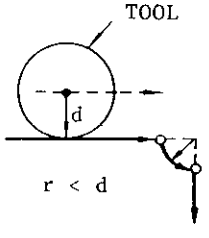
A. Maximum programmable value (Refer to Table 2.3.6.1) is not changed even in tool radius compensation C.

B. Programmed shapes that produce input errors

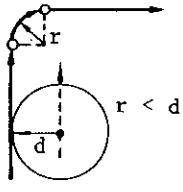
Input error "045" occurs with the following programmed shapes.

(1) When programming an inside arc with tool compensation, if

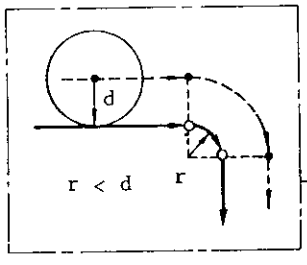
Programmed arc radius r + 5 ≤ tool radius d



(a) Inside compensation error



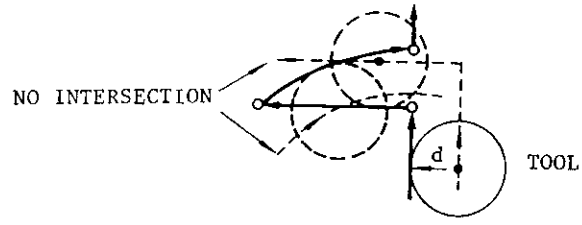
(b) Inside compensation error



Outside compensation is correctly made even when r < d

Fig. 2.52

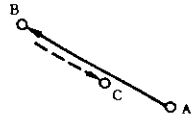
(2) When no intersection point exists on the locus of the offset tool center.

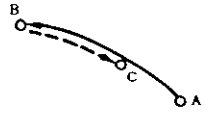


No-intersection error occurs when tool radius is too large relative to the programmed shape.

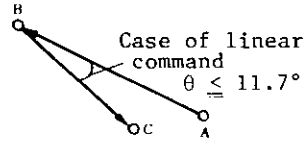
Fig. 2.53

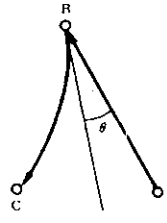
(3) When reversing command or an angle close to reversing command is programmed in M97 (Outside Corner Circular Arc Point Off) mode.





(a) Reversing command





(b) Command close to reversing

Note: With the circular arc command, tangent angle θ alone is insufficient.

Fig. 2.54

In M96 mode, all of the above shapes are correctly compensated.

C. Input errors occur when the following G codes are programmed in the compensation mode.

	Prohibited G codes
G codes producing input errors	G12, G13 (G17 to G19) G28, G29 G73, G74, G76, G77 G81 to G89 G92

* If a "reset operation" is performed in the compensation mode, compensation is cancelled and G40 remains.

D. Tool radius compensation C is applied to the movement path offset by tool length offset and tool position offset. However, in principle, avoid applying compensation C to the path using tool position offset for compensation of tool radius.

E. When programming G41, G42 and G40, G00 or G01 and an F code should be programmed in the same block or in a preceding block.

F. An input error occurs if a G code, G17 to G19 of plane designation for changing the compensation plane is programmed during compensation.

G. Program circle cutting (G12, G13), and canned cycles (G73, G74, G76, G77, G80 to G89) in the tool radius compensation cancel mode. Circle cutting and helical cutting incorporate tool radius compensating functions in themselves. Input error "024" occurs when they are programmed in the compensation mode.

H. Tool radius compensation C is also possible on circular interpolation by radius R designation.

I. Subprogram (M98, M99) can be programmed in the compensation mode.

J. Compensation is applied to the projection to the compensation plane designated by G17, G18 or G19 when simultaneous movement along three axes (four axes†) is programmed in compensation mode.

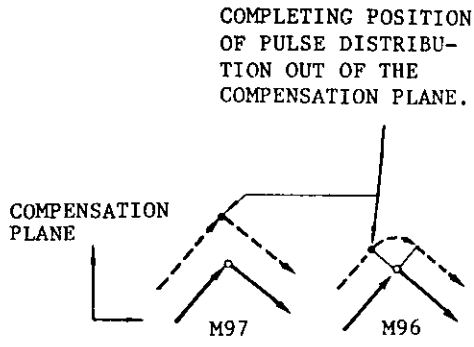


Fig. 2.55

K. Input error "046" occurs when circular interpolation is programmed out of the plane designated by G17, G18 or G19.

L. Offset position may be temporarily modified by programming a dummy block using addresses I, J, K.

```
(G42)
 .
 .
 .
N100  G01  X···  Y···  ;
N101       I···  J···  ;
N102       X···  ;
```

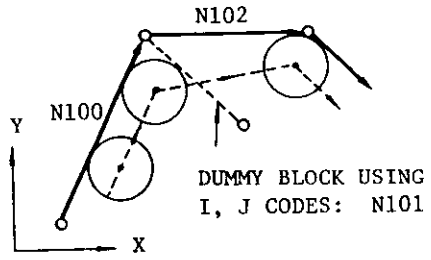


Fig. 2.56

2.9.21 TOOL RADIUS COMPENSATION C (G40, G41, G42) (CONT'D)

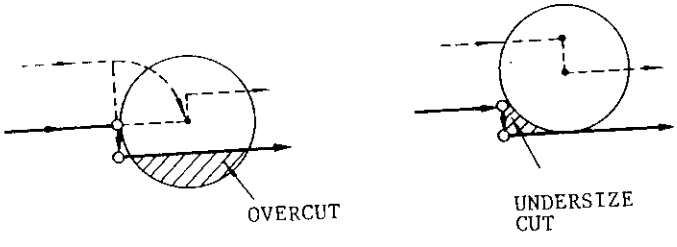
M. Advance reading of blocks is prohibited when M00, M01 (M02, M30) commands are given, and compensation is usually interrupted. Continuation of correct compensation is secured by programming I, J, K in a dummy block immediately before M00, M01 to avoid interruption.

```
(G41)
 .
 .
 .
N200 G01 X...  Y...  ;
N201         I...  J...  ;
N202         M00   ;
N203         X...  Y...  ;
```

Command movement data N203 using I, J

N. Up to 99 radius values can be stored in the offset memory in total for the tool radius compensation, together with the values for other compensation. Make designation by a D code. The maximum programmable value of tool radius compensation is ±999.999 mm (or ±99.9999 inch).

O. Overcut occurs if compensation is programmed on a step less than the tool radius in M96 mode. Keep this in mind. Although undersize cut occurs with the G97 mode, it is better than overcut with the M96 mode.



(a) M96 mode (b) M97 mode

Fig. 2.57

P. Even in M96 mode, the tool moves directly toward point B without making circular path, if both ΔX and ΔY are smaller than a fixed value as shown below. The fixed value in this case is the value set by parameter (#6230).

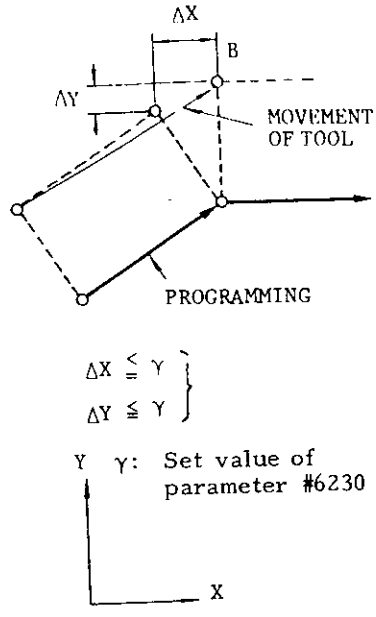


Fig. 2.58

10. Intervention of MDI operation in compensation mode

MDI operation can not be intervened in compensation mode.

11. Intervention to active buffer in compensation mode

The data given below can be programmed in the compensation mode of G41 or G42 with procedures identical to those of MDI operation, after turning on the SINGLE BLOCK switch to suspend the block, and then, selecting the RAPID or JOG mode.

Programmable data:
F, M, S, T and B† codes

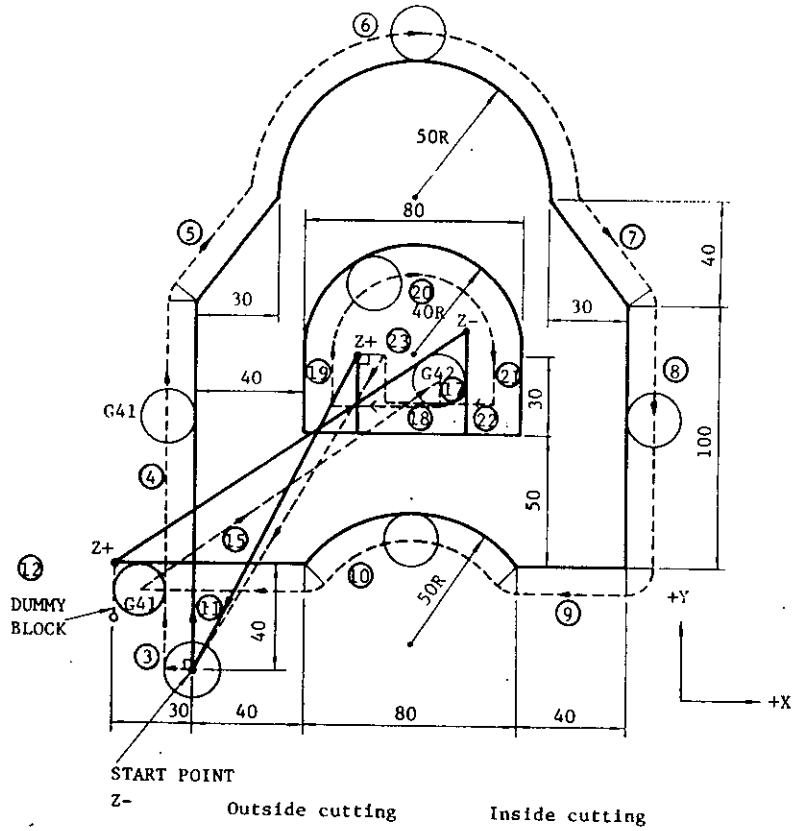
Programmable block:
In addition to the block of instructions of the active buffer just executed

When the CYCLE START button is pushed in the RAPID or JOG mode after programming, the instructions are immediately executed and signals such as BCD output are sent out. Automatic operation can be resumed when CYCLE START is made after returning to the original automatic operation mode.

Note: In the operation described in these items, the following M codes cannot be written.

M00, M01, M02, M30, M90 to M199

EXAMPLE A



Outside cutting G41 ⟶ Inside cutting G42

① G91 G01 A-2500 F150 ; — Incremental; Z axis lowering

② G17 (G01) F300 ; — XY plane designation, feed command

③ G41 D21 ; — Tool radius compensation start command with tool offset No. 21

④ Y14000 ; — Offset to a point on the normal line of start point of this block.

⑤ X3000 Y4000 ;

⑥ G02 X10000 I5000 ;

⑦ G01 X3000 Y-4000 ; — Outer Cutting

⑧ Y-10000 ;

⑨ X-4000 ;

⑩ G03 X-8000 R5000 ; — R designation circular arc

⑪ G01 X-7000 ;

⑫ J-2000 ; — Dummy block (for modifying offset position)

⑬ Z2500 ; — Z axis elevation

⑭ M01 ; — Optional stop

Two blocks without movement on the designated plane

⑮ G42 (G01) X13000 Y9000 F2000 ; — Switching of direction of compensation (left → right)

⑯ Z-2500 F150. ; — Z axis lowering

⑰ Y-4000 F300 ;

⑱ X-6000 ;

⑲ Y3000 ;

⑳ G02 X8000 I4000 ; — Inner Cutting

㉑ G01 Y-3000 ;

㉒ X-6000 ;

㉓ Y3000 ; — Offset to a point on the normal line of end point of this block

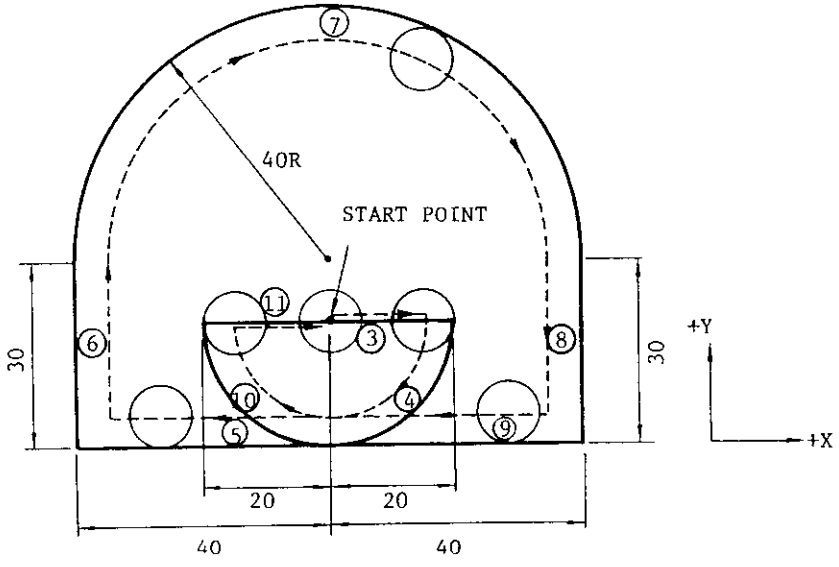
㉔ Z2500 ; — Z axis elevation

㉕ G40 (G01) X-6000 Y-12000 F2000 ; — Compensation cancel command

The same effect is obtained even when the commands in parentheses are not made. They are entered for ease of understanding.

Fig. 2.59

EXAMPLE B



(G40)

① G91 G01 Z-2500 F150 ;
② G17 F300 ;
③ G42 D20 X2000 ;
④ G02 X-2000 Y-2000 I-2000 ;
⑤ G01 X-4000 ;
⑥ Y3000 ;
⑦ G02 X8000 I4000 ;
⑧ G01 Y-3000 ;
⑨ X-4000 ;
⑩ G02 X-2000 Y2000 J2000 ;
⑪ G40 G01 X2000 ;
⑫ Z2500 ;

With the inner cutting in EXAMPLE A, the double cutting allowance at the cutting start and cutting end varies with the tool radius. An inner cutting case with zero double cutting allowance regardless of the cutter radius is shown in EXAMPLE B.

Fig. 2.60

2.9.22 TOOL LENGTH COMPENSATION (G43, G44, G49)

The tool length compensation function is for adding or subtracting the stored tool offset values to the Z-axis coordinate instruction values for the purpose of compensating for the deviations in tool length.

· G codes for tool length compensation

G code	Group	Meaning
G43	08	(+) direction
G44	08	(-) direction
G49	08	cancel

· G43 and G44 are modal functions, remaining effective when once commanded until cancellation by G49.

· G49 cancels tool length compensation effects.

· H00 also cancels tool length compensation effects.

· The tool length compensation function is programmed in the following format.

A. (G01)

G43(G44) Z··· H··· ;

With this command, the tool moves towards the Z coordinate position which is the sum of (or difference between) the H value and the Z value. As the result, the tool point is displaced from the specified Z-coordinate position by the distance specified by the H code.

B. (G01) Z··· ;

G43(G44) H··· ;

With this command, the tool is shifted by the distance specified by the G code.

C. G43(G44) Z··· H··· ;

H··· ; ····· (2)

With the command (2), the tool is shifted by the difference between the previous tool offset value and the new tool offset value.

· When G43, G44 and G49 are to be commanded, the accompanying 01 group G codes must be G00, G01 or G60. When G02 or G03 is used, this is regarded as an error.

· Direction of shift

The direction of tool shift is determined by the sign of tool offset value as programmed in the H code and by the G code used.

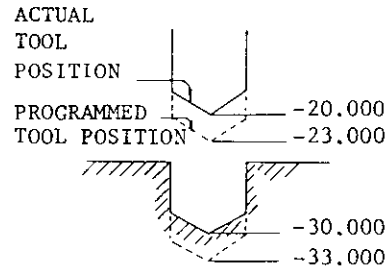
	Sign of tool offset value	
	+	-
G43	Plus direction	Minus direction
G44	Minus direction	Plus direction

EXAMPLE

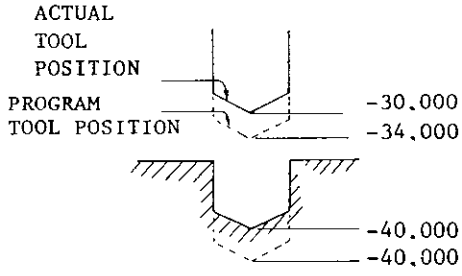
H10 ··· Offset value: -3.0

H11 ··· Offset value: 4.0

	CRT display including offset value (Z direction 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



NOTES:

- When the tool offset value is changed by the MDI function while programs in the offset mode is in execution, the change is effective from the block containing D code.
- The tool position offset function or the tool radius compensation function is effective on the tool which is already offset by the tool length compensation function.
- G43, G44 and G49 can be programmed in canned cycles. If they are programmed, this is regarded as an input error.
- When a G92 command involving the Z axis is given during the execution of a program in the tool length compensation mode, the tool length compensation is canceled. In principle, when G92 is to be programmed, the existing tool length compensation mode should first be canceled.
- During the automatic execution of a program in the tool length compensation mode, the number of the effective tool compensation memory (H code number) can be displayed. For this, refer to 4.3.2 DISPLAY OF COMMAND DATA.

2.9.23 TOOL POSITION OFFSET (G45 TO G48)

Tool position offset A is for extending or reducing the movement value designated in the program by the values in the tool offset memory, and is mainly used for tool radius compensation for square patterns. Therefore, this function is not required with controls equipped with G40, G41, G42 (tool radius compensation C).

1. G codes of tool position offset

G code	Group	Meaning
G45	*	Extension
G46	*	Reduction
G47	*	Expansion by double
G48	*	Reduction by double

2. G45 to G48 extend and reduce the movement value programmed in the block, in the direction of movement by the tool offset value. Extension or reduction is made only in the block in which G45 to G48 are programmed and movements in other blocks are unaffected. Therefore, to restore extended or reduced values to the original program values, an extension or reduction in the opposite direction must be programmed eventually.

3. Make program command by incremental designation (G91) for the sake of making the above operation clear. When the command is given by absolute designation (G90), extension and reduction are made along the direction of movement to the movement value from the end point of the preceding block, to the command target point. That is, extension and reduction are made to the incremental movement amount. The programming may become complicated.

4. When programming G45 to G48, designate the tool offset number by a D code simultaneously with axis designation. Because D codes are modal, they may be omitted if the same D code is used. Store the tool radius value in the tool offset value memory.

EXAMPLE

```
     G91
①    G00  G46  X···  Y···  D01  ;    · · ·   Reduction
②    G01  G47  Y···  (D01)  F···  ;   · · ·   Extension by double
③         G47  X···  (D01)  ;         · · ·   Extension by double
④         G47  Y···  (D01)  ;         · · ·   Extension by double
⑤         G47  X···  (D01)  ;         · · ·   Extension by double
⑥    G00  G46  X···  Y···  (D01)  ;   · · ·   Reduction
```

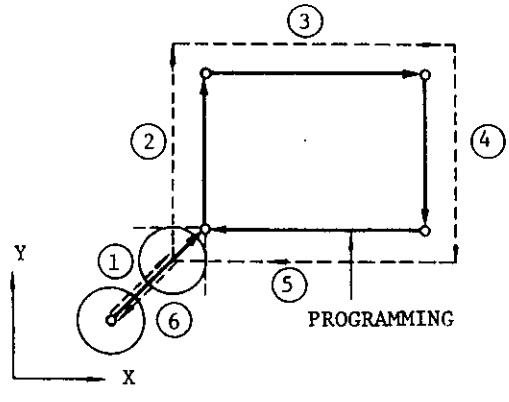


Fig. 2.61

2.9.23 TOOL POSITION OFFSET (G45 TO G48) (CONT'D)

5. Extension and reduction

Extension or reduction is determined by the sign of the tool offset value designated by a D code in addition to the G code.

	Sign of tool offset value	
	Positive	Negative
G45	Extension	Reduction
G46	Reduction	Extension
G47	Extension by double	Reduction by double
G48	Reduction by double	Extension by double

Note: In general, tool offset value should be "positive."

6. Values of extension and reduction

A. Programmed incremental move values are extended or reduced by the designated tool offset values or by twice their values.

G91 G00 G47 X6000 D10 ; D10 = 2000

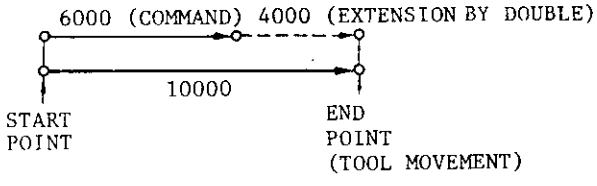


Fig. 2.62

B. Where extension or reduction is applied to an axis in the preceding block and the start point has already been offset, the total movement value is identical to that described above, but the distance is measured from the offset start point.

With an instruction same as that described above:

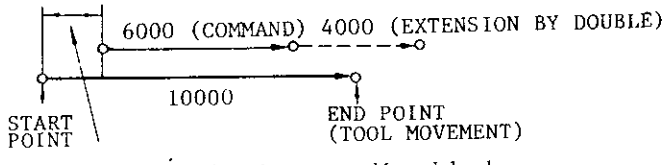


Fig. 2.63

Note: Where the tool offset value is larger than the programmed movement value, the direction of movement may be reversed when extension or reduction is applied.

G46 X1000 D10 ; D10 = 2000

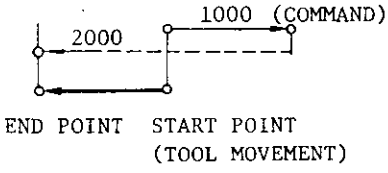


Fig. 2.64

7. The above applies to X and Y axes, but G45 to G48 may also be programmed to Z axis in the same manner.

8. Application to circular interpolation

If I, J, K are programmed in the block with G45 to G48, extension or reduction is made respectively in the same directions as X, Y, Z. Therefore, tool radius compensation is possible with 1/4 circle, 3/4 or full circle.

G91
G45 G02 X5000 Y5000 I5000 D10 ;
D10 = 2000

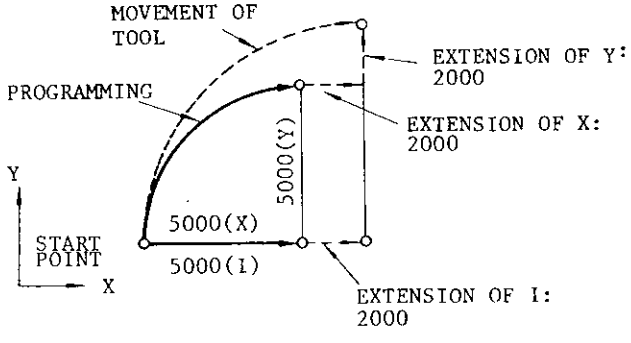


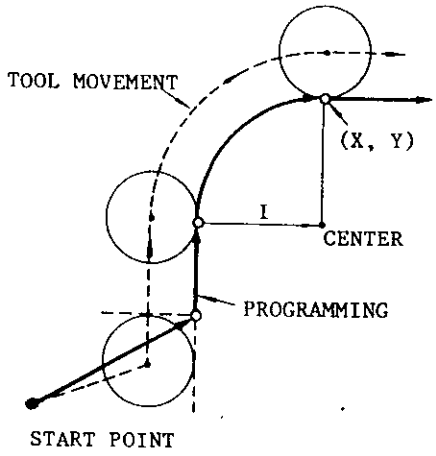
Fig. 2.65

In practice, correct radius compensation of circular arc is made if an offset is applied in the preceding block.

G91 G01 F··· ;
G46 X··· Y··· D10 ;
G45 Y··· ;

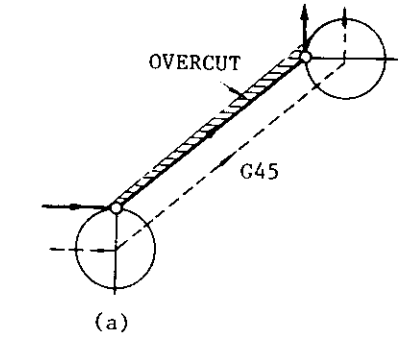
G45 G02 X··· Y··· I··· ;

G01 X··· ;



Note: When it is necessary to program 1/2 circle, assemble them using 1/4 circle.

Fig. 2.66



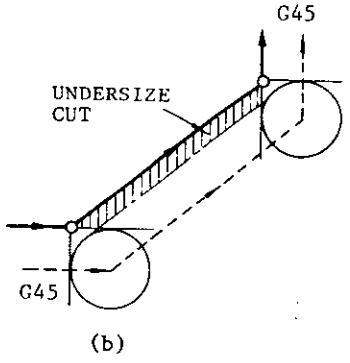


Fig. 2.67

9. When programming G45 to G48, the G code of 01 group can be given together in the same block. An input error occurs if instruction is given with other G codes.

10. When only movement by offset in the incremental designation (G91) is required, program "0" as the axis movement instruction.

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

... Movement is made in the positive direction along both X and Y axis by the offset value with D10.

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

... Movement is made in the negative direction along X axis by the offset value with D11.

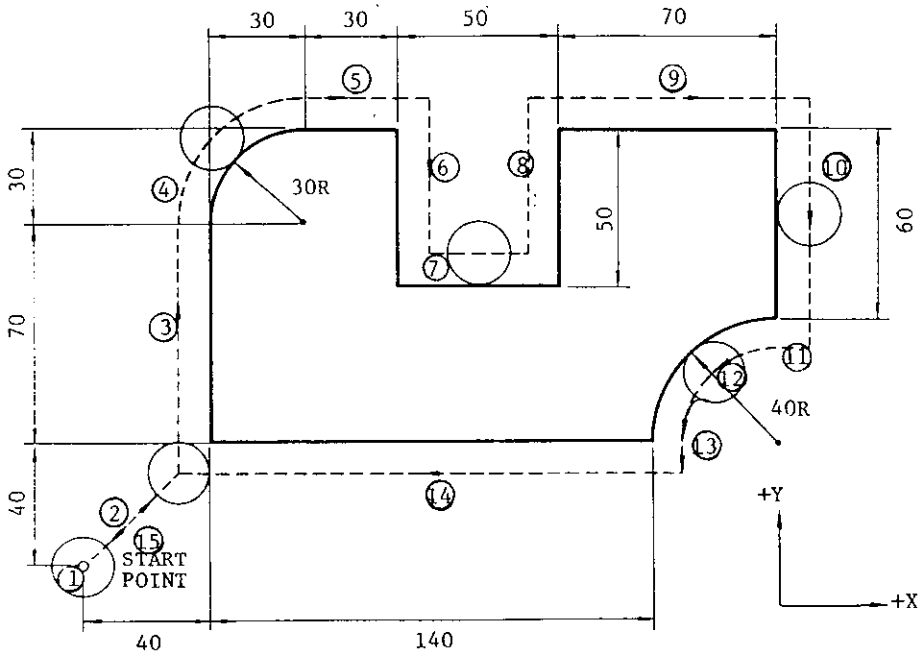
It is meaningless to give a sign to "0."

NOTES:

- When G45 to G48 are programmed as the simultaneous movement instruction along two axes, extension or reduction is made in the two axes. Overcut or undersize cut will occur if this is applied to cutting. Keep this in mind. (Fig. 2.9.23.7)
- Even when the offset value is changed by MDI, the offset instruction previously programmed will not be affected. It becomes operable when G45 to G48 are programmed thereafter.
- This tool position offset can be applied in addition to the tool length offset.
- Mirror image can be applied to tool position offset. That is, it is possible to perform symmetrical cutting with this offset applied.
- Tool position offset is independent of G codes (G17/G18/G19) of plane designation.
- G45 to G48 can not be programmed in the canned cycles mode. An input error will occur if this is programmed.
- If G92 is programmed in the offset mode, programming of absolute zero point is made after the offset value is canceled from the designated axes. In principle, program G92 after returning the offset value to the original value by programming extension or reduction in the opposite direction.
- During automatic operation, the offset distance in each axis from the programmed end point by tool position offset can be displayed Refer to 4.3.2.3 DISPLAY OF TOOL OFFSET STATE: COMMAND (OFFSET).

2.9.23 TOOL POSITION OFFSET (G45 TO G48) (CONT'D)

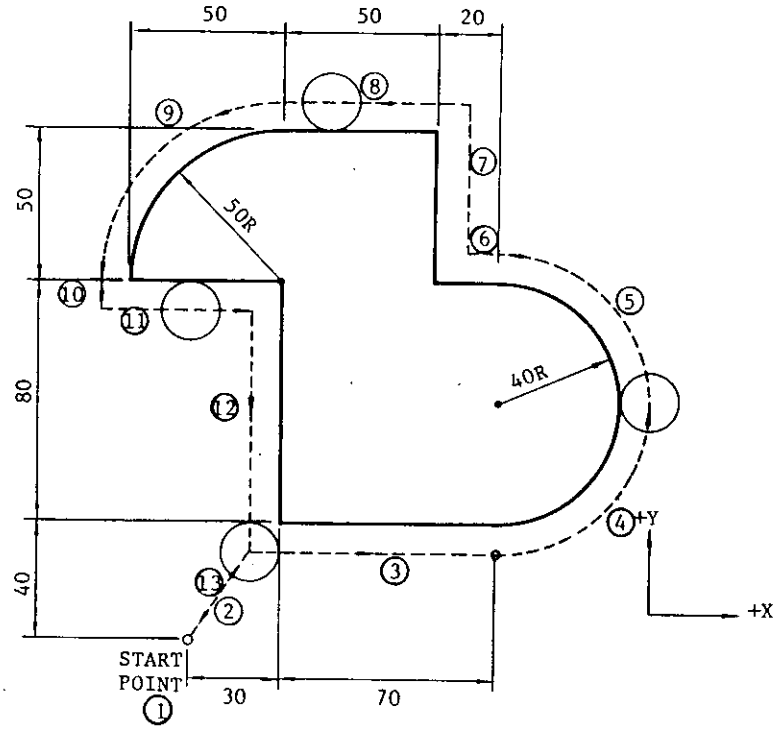
EXAMPLE A




```
①   G91  G01  Z-2500  F150  ;
②        G46  X4000  Y4000  D10  F300  ;
③        G45  Y7000  ;
④   G45  G02  X3000  Y3000  I3000  ;
⑤   G45  G01  X3000  ;
⑥             Y-5000  ;
⑦        G48  X5000  ;
⑧             Y5000  ;
⑨        G47  X7000  ;
⑩        G47  Y-6000  ;
⑪        G46  X0  ;
⑫   G46  G03  X-4000  Y-4000  J-4000  ;
⑬   G46  G01  Y0  ;
⑭        G47  X-14000  ;
⑮        G46  X-4000  Y-4000  ;
⑯             Z2500  ;
```

Fig. 2.68

EXAMPLE B




```
①   G91  G01  F300  ;
②        G46  X3000  Y4000  D15  ;
③        G45  X7000
④   G45  G03  X4000  Y4000  J4000  ;
⑤   G45  G03  X-4000  Y4000  I-4000  ;
⑥   G46  G01  X-2000  ;
⑦             Y5000  ;
⑧        G45  X-5000  ;
⑨   G45  G03  X-5000  Y-5000  J-5000  ;
⑩   G46  G01  Y0  ;
⑪             X5000  ;
⑫             Y-8000  ;
⑬        G46  X-3000  Y-4000  ;
```

Fig. 2.69

2.9.24 SCALING FUNCTION (G50, G51)+

With this function, workpiece contours programmed by part programs can be enlarged or reduced at any desired scale.

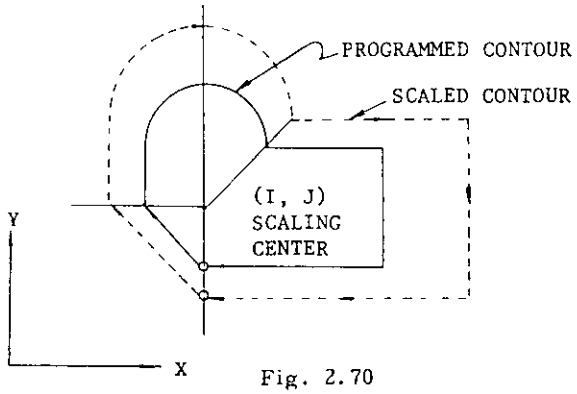


Fig. 2.70

The following G codes are used for this function.

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

Note: When power is applied or the control is reset, the control is in the state of G code marked with ◥.

- G51 I··· J··· K··· P··· ;

  With this command, the program is executed on an enalrged or reduced scale with the scale ratio specified by P, and the center of scaling specified by I, J, and K.

- G50; command cancels the scaling mode.
- The enlarging and reducing scales can be selected within the following range.

Enlarging and reducing range	0.000001-99.999999

When no scale ratio is specified in the program, the ratio set by parameter #6500 becomes effective as the scale ratio.

Command unit for P is: 1 = 0.000001.
When P command includes decimal number, numbers after decimal point are regarded as six-digit numbers.

Example

P0.999999	0.999999 time
P2.0	2 times
P2	0.000002 time

When P (designating multiplication) is omitted, multiplication is determined by setting #6500 and #6501.

$$\text{Multiplication} = \frac{\#6500}{\#6501}$$

Example

Where setting #6500 = 3, #6501 = 100

$$\text{Multiplication} = \frac{3}{100} = 0.03 \text{ times}$$

Multiplication should not exceed the enlarging and reducing range.

When I, J, or K is programmed in the G51 command, scaling functions on the axis designated: I··· X-axis, J··· Y-axis, K··· Z-axis. Scaling will work only on the axis selected by I, J, or K.

Example

G51 I100 J0 P08

With this command, scaling will work on X- and Y-axis and not on Z-axis.

Where the work coordinate system is specified, I, J, and K in the G51 block designates the distance between coordinate system zero point and scaling center.

Example

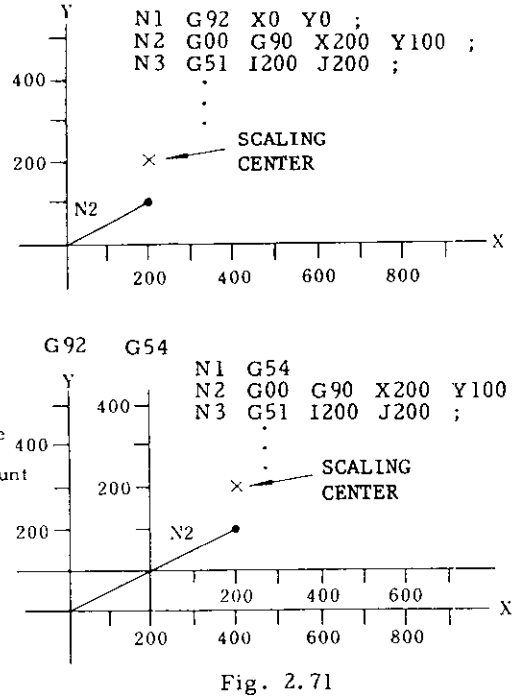


Fig. 2.71

NOTES:

- Scaling is turned on when approaching for usual machining and off after retraction on completion of approaching. Turning off and on scaling during machining will not form the correct contour.
- Scaling is executed on the two axes on machining plane. If scaling is executed on a single axis, an alarm occurs at circular command because scaling cannot work according to circular command.

- Block commands G51 I... J... K... P... ; and G50 ; should be programmed independently. If X, Y and Z commands coincide in the same block, an alarm will occur.
- When the scale ratio of one or more is programmed, the resultant command value should not exceed the maximum.
- Scale ratio 0 cannot be commanded. If commanded, an alarm will occur.
- Scaling is not effective on compensation value.
- Canned cycles cannot be executed with scaling commanded on Z-axis. If scaling is commanded on Z-axis during canned cycle execution, an alarm will occur.
- When operation is reset (reset pushbutton, M02, M30 command), scaling is turned off.
- Display of command and position will show the values of command and position after scaling is finished.
- The following G codes cannot be commanded during scaling. If commanded, an alarm will occur.

  G28, G29, G30, G31, G36, G37, G38, G53, G92
- Scaling (G51) command cannot be given during tool radius compensation C.
- Alarm codes for scaling are listed below.

EXAMPLE

Reduction by half in respect with the point (40, 30) specified as the center.

```
(1)  N100 G92 X0 Y0 Z0 ;
(2)  N101 G51 I40. J30. P 0.5   . . . Center of 1/2 scaled contour (40, 30)
(3)  N102 G90 G00 X10. Y20. ;
(4)  N103 G01 Y50. F1000 ;
(5)  N104 G02 X50. I20. ;
(6)  N105 G01 X90. ;                  Scaling range
(7)  N106 Y10. ;
(8)  N107 X50. ;
(9)  N108 X0 Y35. ;
(10) N109 G50 ;              . . . . . Scaling cancel
(11) N110 G00 X0 Y0 ;
```

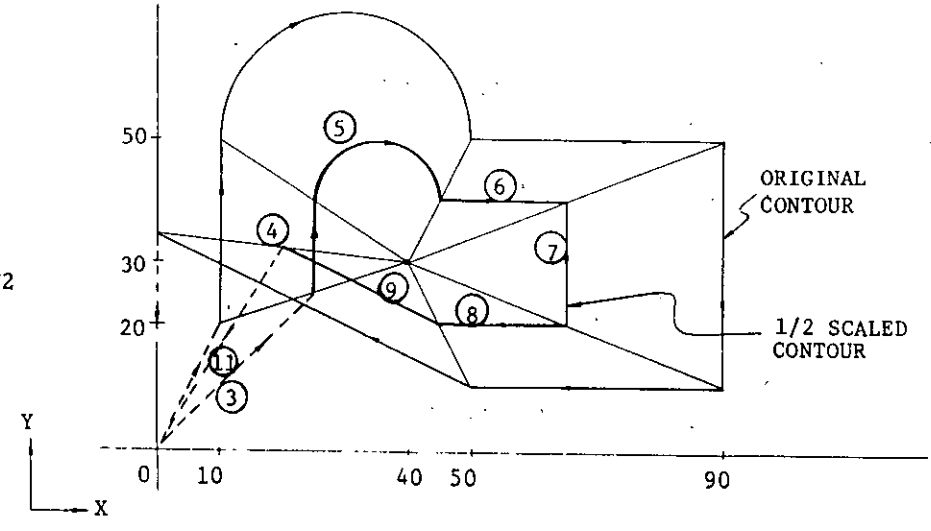


Fig. 2.72

2.9.25 WORK COORDINATE SYSTEM SETTING (G52 to G59) †

Six types of work coordinate systems corresponding to six G codes, G54 through G59, are available for selective use.

· There are three types of coordinate systems as follows.

A. Basic coordinate system

This is the basic coordinate system to be set up by G92, by the ORIGIN key, or by the automatic coordinate system setting function. When the power supply is turned on, until any of these actions will be made, the tool position at the time of turning on is treated as the temporary coordinate origin point.

B. Work coordinate system

When any of the G codes G54 through G59 is commanded, a coordinate system with the origin shifted by the amount set by the setting numbers corresponding to that G code is set up. The coordinate systems set up by these G codes are referred to as work coordinate systems, and when once a work coordinate system is set up, the tool will be controlled to it. Since there are six G codes for work coordinate systems, up to six work coordinate systems can be used.

C. Machine coordinate system

This is a coordinate system which is fixed to the machine, and is set up when the tool is returned to the reference point. This coordinate system has its (0, 0, 0) point at the reference point.

· The setting number for setting the shift amounts for G codes from G54 to G59 are as follows.

G code	Coordinate system	X	Y	Z	α
G54	Work coordinate system 1	#6516	#6517	#6518	#6519
G55	Work coordinate system 2	#6522	#6523	#6524	#6525
G56	Work coordinate system 3	#6528	#6529	#6530	#6531
G57	Work coordinate system 4	#6534	#6535	#6536	#6537
G58	Work coordinate system 5	#6540	#6541	#6542	#6543
G59	Work coordinate system 6	#6546	#6547	#6548	#6549

· Setting up work coordinate systems (G54 to G59)

G54 (G55, G56, G57, G58 or G59);

When this command is once given, from that time on, the tool will be controlled on the work coordinate system specified by it.

· Returning to basic coordinate system (G52)

G52 ;

With this command, the currently effective work coordinate system is cancelled, and the basic coordinate system becomes effective again. (Fig. 2.9.25.1)

· Temporary shift to positions on machine coordinate system

(G90) G53 G00 X··· Y··· Z··· ;

With this command, the tool is shifted to the position (X, Y, Z) on the machine coordinate system only in this block. G53 is a non-modal G code.

EXAMPLE

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

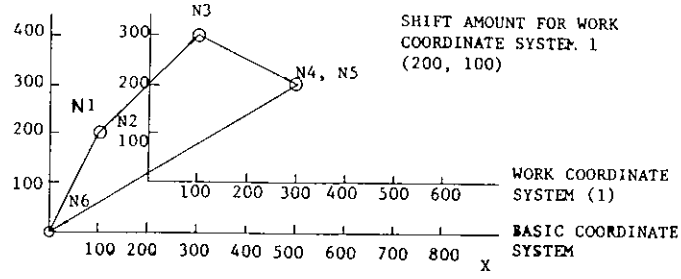


Fig. 2.73

EXAMPLE (Reference program)

```
N1  G92  X200  Y100  ;
N3  G54  G90  X100  Y200  ;
N4  G53  X300  Y100  ;
N5  X300  Y0  ;
N6  G52  ;
N7  X0  Y0  ;
```

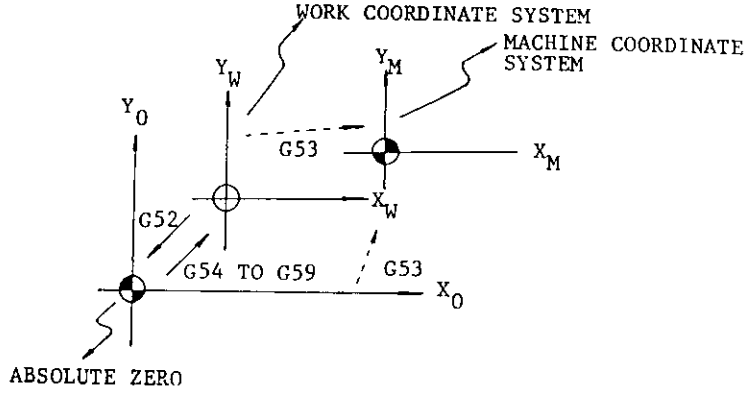


Fig. 2.74

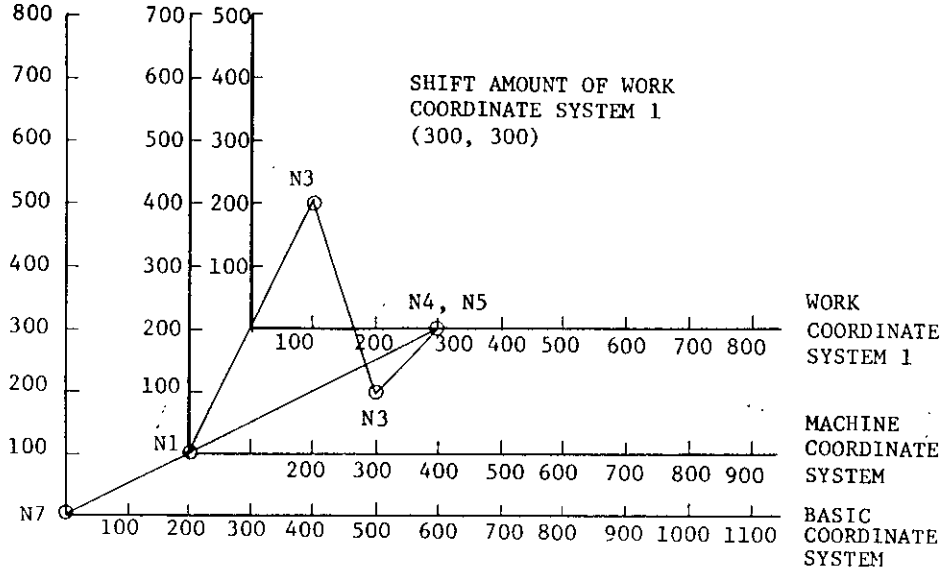


Fig. 2.75

NOTES:

- The shift amounts for work coordinate systems can be specified by programs with G10 commands, in addition to the MDI writing. For details, refer to 2.9.8 TOOL OFFSET VALUE DESIGNATION (G10).
- Work coordinate systems set up by G54 through G59 are canceled by the G52 command, and the basic coordinate system becomes effective again.
- When once a work coordinate system has been set up by any of the commands G54 through G59, the selected shift amounts can not be changed even when they are rewritten. The rewritten shift amounts will become effective when a new work coordinate system command is executed.
- G53 commands should only be given under the following conditions. If these conditions are not satisfied, the commands are regarded as errors.
  (1) The mirror image function is not used.
  (2) No canned cycle is in use and no tool compensation C is in use.
  (3) If a 01 group G code is used, it is G00, G01 or G60 and nothing else.

2.9.25 WORK COORDINATE SYSTEM SETTING (G52 to G59)† (CONT'D)

- If a G53 command is executed with the machine lock function on, the current value displayed changes sequentially until the command value corresponding to the machine lock function off state will be displayed. If the machine lock function is switched on and off during the execution of G53 blocks, correct positioning can not be achieved.
  However, when a complete G53 block is executed with the machine lock function off, correct positioning is achieved as programmed, even when the machine lock function is switched on and off before that block.
- G53 commands should be given in the G90 mode. If they are given in the G91 mode, the command values are regarded as G90 mode values.
- When work coordinate systems are to be changed with any of the G54 through G59 commands, the program should be so written that a new coordinate system will be set up in the G90 mode and the basic coordinate system will be reset in the G90 mode.
- If a G53 command is given while the tool length compensation or tool position offset function is on, the tool offset value is deleted temporarily. Generally, when giving a G53 command, the tool length compensation and tool position offset commands should be canceled in advance.
- When any of the commands G54 through G59 is given while the tool length compensation or tool position offset command is on, the compensation remains effective. Generally, when any of the commands G54 through G59 is to be given, the tool length compensation or tool position offset command should be canceled in advance.

```
G43  Z0  H01  ;      COM Z100
G54  ;
G90  Z1000  ;        COM Z1100
```

Z 1300 in incremental shift

G54 shift: Z = 300

Offset: H01 = 100

- If G92 is given during execution on the work coordinate system set up by G54 through G59, G54 through G59 or the basic coordinate system is shifted so that the current position is to be a shifted position by G92, G92 should not be used in G54 to G59 modes in general.

2.9.26 UNIDIRECTIONAL APPROACH (G60)†

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

- G60 X··· Y··· Z··· ;

  With this command, the tool moves and stops at the specified position. If the tool approaches the stop position in the direction specified by the parameter (#6014), it overtravels the stop position by the amount specified by parameters (#6062 - #6065) once, and then returns to the specified position to stop.

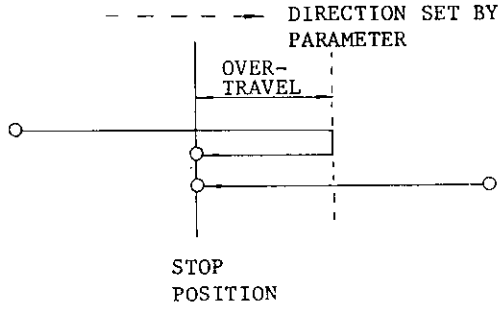


Fig. 2.76

2.9.27 HOLE PATTERN CYCLESS (G70, G71, G72)†

With this function, when a radius and a center angle are specified, the corresponding rectangular coordinate positions are computed automatically and the tool is brought to the required positions. This function is used in conjunction with one of the canned cycles G81 through G89, G73, G74, G76 and G77. With this function, the bolt hole cycle, the arc cycle, and the line at angle cycle are programmed. The tool moves to the position specified by a radius and an angle in rapid traverse (G00).

· Bolt hole cycle (G70)

G70 X··· Y··· I··· J··· L··· ;

With this command, the tool is positioned successively at equally spaced L points on a circle with the center at X, Y and the radius of I, starting at a point located on a line forming J degree with the X axis. In the command,

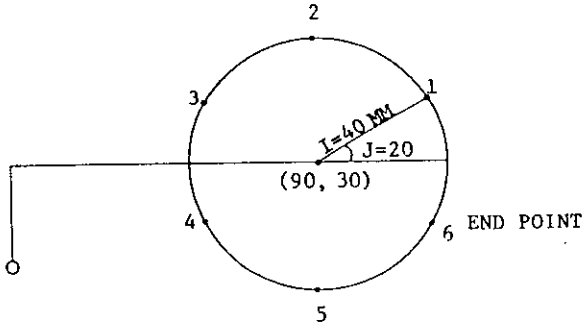
X,Y: Coordinates of the bolt hole cycle, defined either in G90 or G91 mode.

I is the radius of the bolt hole circle, programmed in a positive number and programmed with an accuracy of the least input increment.

J is the angular position of the first hole, programmed in degrees with an accuracy of 0.001 degree. CCW direction is regarded positive.

L is the number of division of the circumference.
For the counter-clockwise sequence, positive numbers are programmed, and vice versa.

EXAMPLE



G81 G98 G90 Z-50. R-20. F20 L0 ;
G70 X90 Y30 I40. J20. L6 ;
G80 G00 X0 Y0 ;

Fig. 2.77

· Arc cycle (G71)

With this function, when the following command is given, the tool is successively positioned to L points located on a circular arc with the center located at X, Y and with the radius of I, at a center angle of K degrees, starting from the point lying on a line intersecting the X axis at J degrees.

G71 X··· Y··· I··· J··· K··· L··· ;

X, Y: Coordinates of the arc center, defind either in G90 or G91 mode

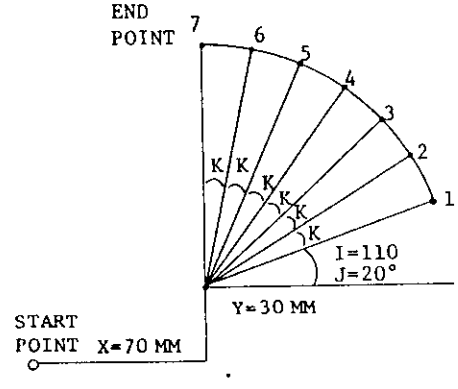
I: Radius of the arc programmed with an accuracy of the least programmable increment, and in positive numbers

J: Angular position of the first hole, programmed in 0.001 degrees. Positive values are used to command counter-clockwise direction.

K: Angular spacing in degrees with an accuracy of 0.001 degrees. Positive values are used to command counter-clockwise direction.

L: Number of holes, to be set in positive numbers.

EXAMPLE



G81 G98 G90 Z-50. R-20. F20 L0 ;
G71 X70. Y30. I110. J20. K15.2 L7 ;
G80 G00 X0 Y0 ;

Fig. 2.78

2.9.27 HOLE PATTERN CYCLESS (G70, G71, G72) (CONT'D)

· Line at angle cycle (G72)

When the following command is given, the tool is positioned successively at L points lying on a line forming J degrees with the X axis, with a uniform interval of I, starting at X, Y.

G72 X··· Y··· I··· J··· L··· ;

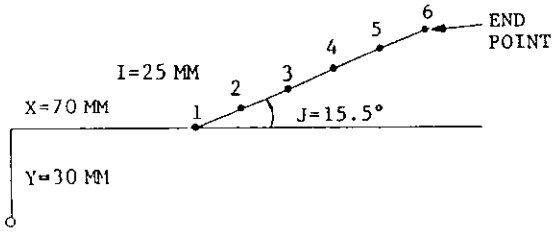
X, Y: Coordinates of the starting point, either in G90 or G91 mode

I: Interval is programmed in degrees, at an accuracy of the least input increment. When I is negative, the holes will be located on the line in the negative direction.

J: Angles programmed with an accuracy of 0.001 degrees.
Positive values are used for CCW direction.

L: Number of holes programmed in positive numbers.

EXAMPLES




```
G81 G98 G90 Z-50. R-20. F20. L0 ;
G72 X70. Y 30. I25. J15.5 L6 ;
G80 G00 X0 Y0 ;
```

Fig. 2.79

NOTES:

· When the hole pattern cycles are to be programmed with G70, G71 or G72, in principle, a canned cycle G73, G74, G76, G77, G81 to G89 should be programmed with L = 0 in the preceding block. Since L = 0, the canned cycle is not executed, but its mode becomes effective.

· G70, G71 or G72 may be programmed in the same block with a canned cycle G code.
However G73 and G83 which involve I, J, and K can not be programmed with G70, G71 or G72 in the same block. When G73 or G83 is to be used, either Q is to be used or I, J, and K should be programmed in the preceding block.

· The machining control of G70, G71 or G72 ends with the drilling of the last hole, and to move the tool to the next position, the G90 (absolute) mode is more convenient than the G91 (incremental) mode, because the latter involves complicated calculations.

· Immediately after the completion of the machining process as commanded by G70, G71 or G72, the canned cycle is still effective, and care must be taken in programming the subsequent block. Make it a point to cancel the canned cycle by G80.

· If G70, G71 or G72 command is given in the tool radius compensaion mode (G41 or G42), the ALARM code will be displayed.

· G70, G71 and G72 are non-modal G codes.

· Give G70, G71 or G72 command in a canned cycle mode only. If they are given without programming any canned cycle, the ALARM code will be displayed.

2.9.28 OUTPUT FOR EXTERNAL MOTION (G80, G81)†

G81 X··· Y··· L··· ;

With this command, the control outputs external motion signals to the machine after each positioning.

The L number specifies the number of repeated positioning motions. G81 is a modal G code, and remains effective until cancellation with G80. G81 may be used either for this purpose or as a canned cycle command code, depending on the setting of the parameter #6018$_{D0}$.

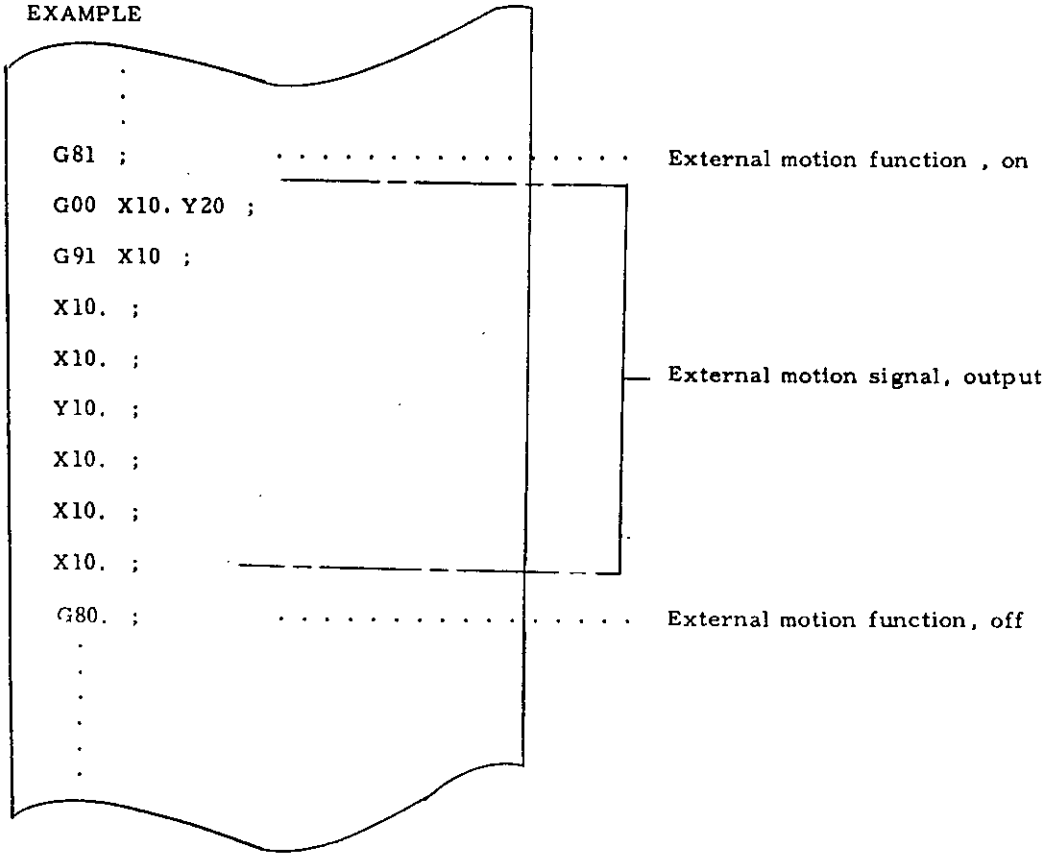


Fig. 2.80

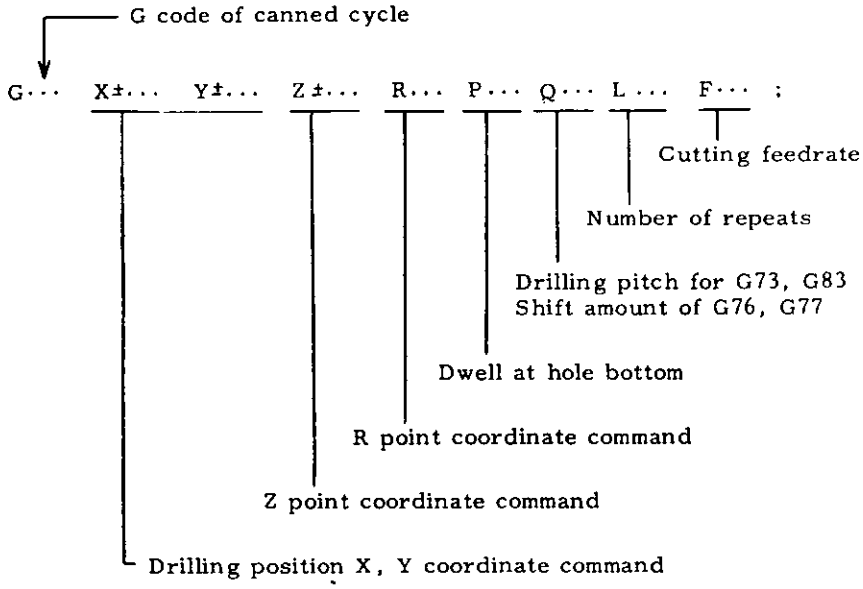
2.9.29 CANNED CYCLES (73, G74, G76, G77, G80 TO G89, G98, G99)†

Canned cycles (G73, G74, G76, G77, G80 to G89, G98, G99) are simplified programs that contain specific movements over a number of blocks in one block. 14 types of cycles are available, and G80 code is commanded for cancelling them.

Table 2.24

G code	Plunging	At hole bottom	Retraction	Application
G73	Wood pecker feed	-	Rapid traverse	High speed deep hole drilling
G74	Feed	Spindle forward running after dwell	Spindle reversing after feed	Reverse tapping
G76	Feed	Spindle indexing → shift	Rapid traverse → shift, spindle start	Boring
G77	Spindle indexing → shift → rapid traverse → shift → spindle start → feed	Dwell	Rapid traverse → spindle indexing → shift → rapid traverse → shift, spindle start	Back boring
G80	-	-	-	Cancel
G81	Feed	-	Rapid traverse	Drilling
G82	Feed	Dwell	Rapid traverse	Spot, facing
G83	Wood pecker feed	-	Rapid traverse	Deep hole drilling
G84	Feed	Spindle reversing after dwell	Spindle forward running after feed	Tapping
G85	Feed	-	Feed	Boring
G86	Feed	Spindle stop	Rapid traverse → Spindle start	Boring
G87	Feed	Spindle stop	Manual retraction → Spindle start	Boring
G88	Feed	Spindle stop after dwell	Manual retraction → Spindle start	Boring
G89	Feed	Dwell	Feed	Boring

· Command format:



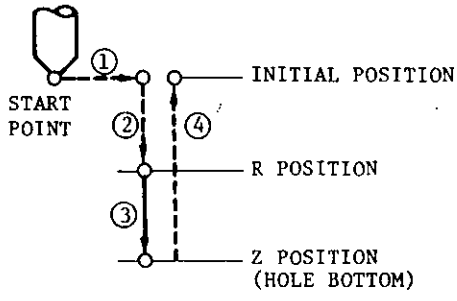


Fig. 2.81

Operations ① through ④ are executed in one cycle with the commands shown above.

① Positioning the drilling position (X,Y)
② Rapid traverse to R point
③ Drilling to Z point
④ Return to R point or to initial point

Number of repeats is specified by the address L. Where L is not given, number of repeats is regarded as "1."
If 0 is given for L, only positioning to (X, Y) is made. Shift direction of shift of G76, G77 can be made at the intended angle specified by the parameter. (#6019D0)
Z axis returning position at the end of canned cycle can be designated by the following G code.

G code	Meaning
G98	Initial level return
G99	R position level return

Note: When power is applied or the control is reset, the control is in the state of G code marked with ◥.

Where parameter #6019D0 is set to 0, the shift direction is made in the specified direction. The direction is set as listed below.

Bit / Direction	D2	D1
X(+)	0	0
X(-)	0	1
Y(+)	1	0
Y(-)	1	1

Where parameter #6019D_0 is set to 1, the shift direction is made at the intended angle. The shifting angle is set by setting #6506 (1 = 0.001 deg).

Table 2.25 Canned Cycle

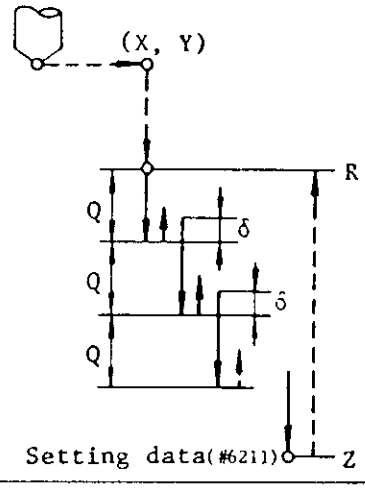
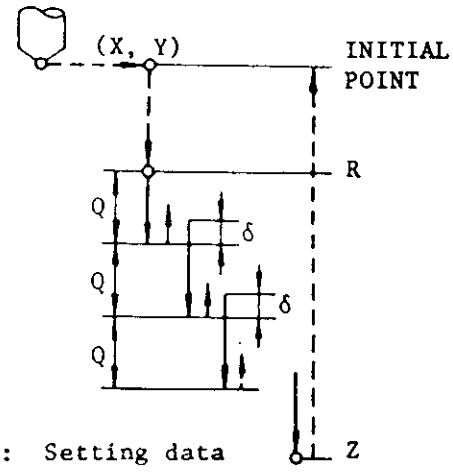
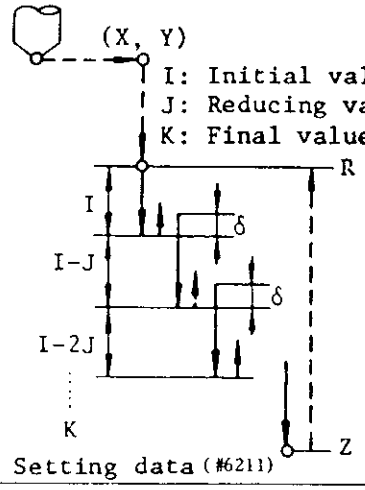
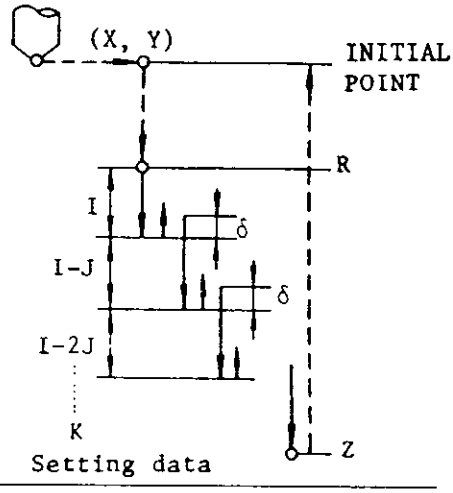
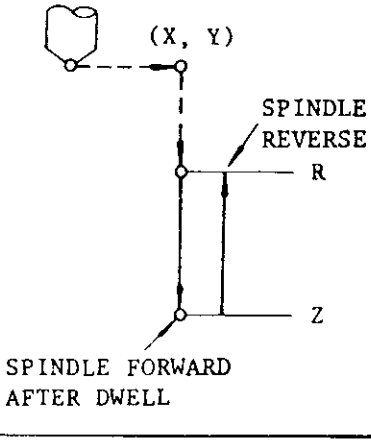
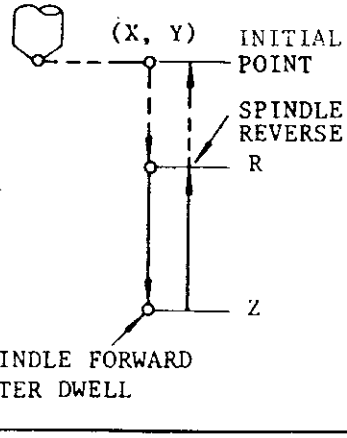
	With G99 (return to R)	With G98 (return to initial point)
G73 (Fixed pitch) High speed deep hole drilling	G73 X... Y... Z... R... Q... L... F... ; 	
G73 (Variable) High speed deep hole drilling	G73 X... Y... Z... R... I... J... K... L... F... ; 	
G74 Reverse tapping	G74 X... Y... Z... R... P... L... F... ; 	

Table 2.25 Canned Cycle (continued)

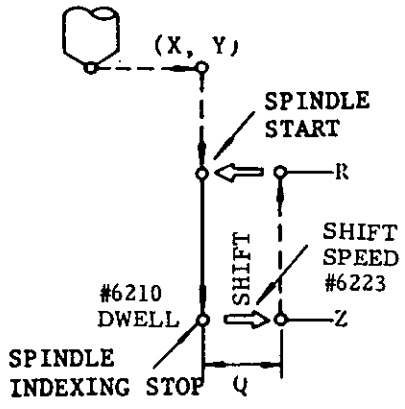
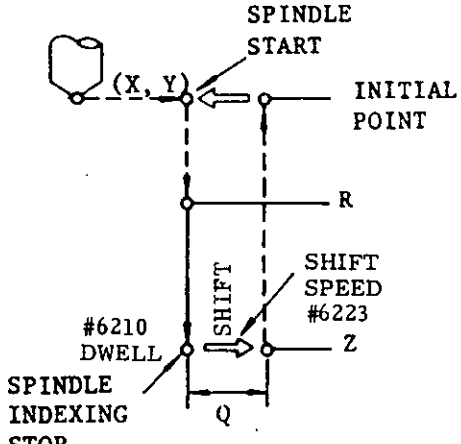
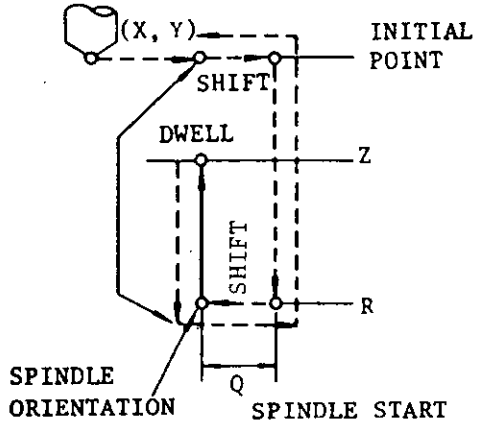
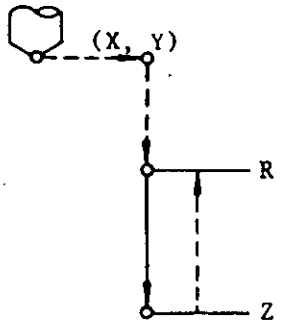
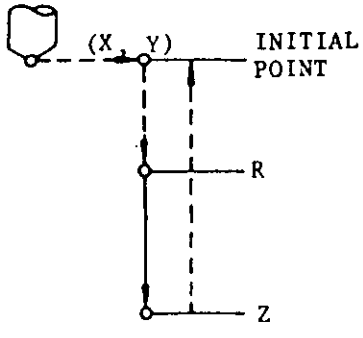
	With G99 (return to R)	With G98 (return to initial point)
G76	G76 X... Y... Z... R... Q... L... F... ;	
Boring		
G77	G77 X... Y... Z... R... Q... L... F... ;	
Back boring	NOT USED	
G80 Cancel	G80 ;	
G81	G81 X... Y... Z.. R... L... F... ;	
Drilling		

Table 2.25 Canned Cycle (continued)

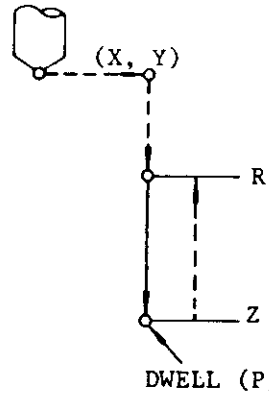
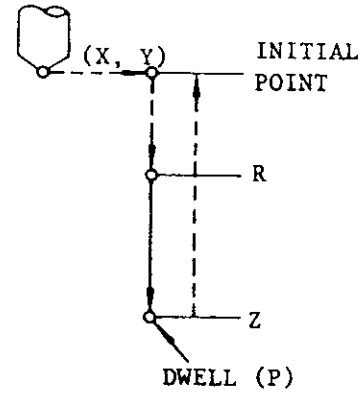
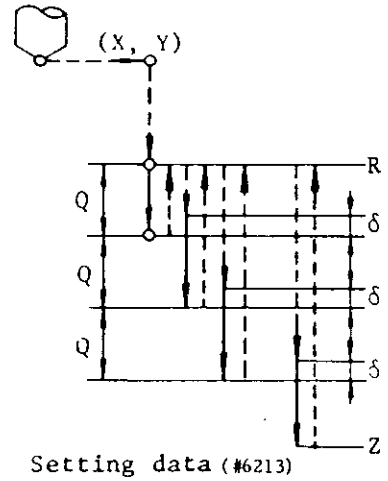
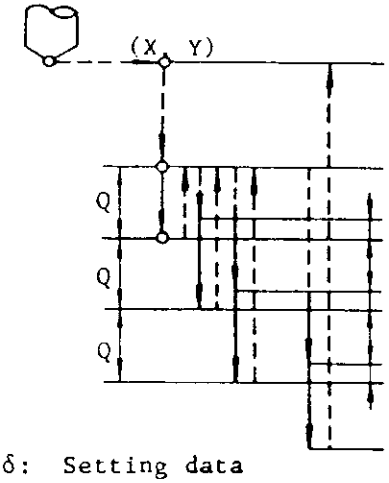
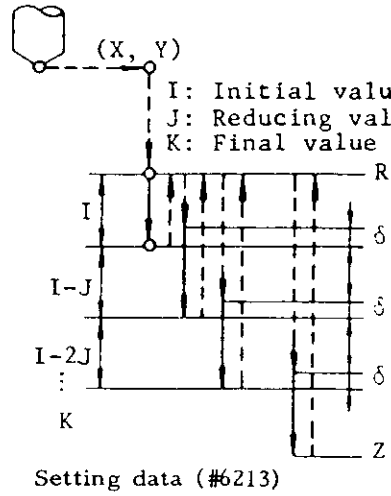
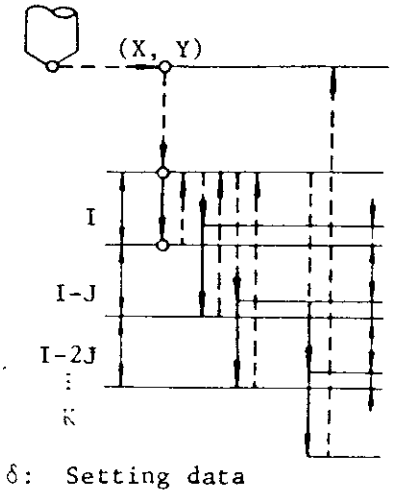
	With G99 (return to R)	With G98 (return to initial point)
G82 Spot facing	G82 X... Y... Z... R... P... L... F... ; 	
G83 (Fixed pitch) Deep hole drilling	G83 X... Y... Z... R... Q... L... F... ;  δ: Setting data (#6213)	 δ: Setting data
G83 (Variable pitch) Deep hole drilling	G83 X... Y... Z... R... I... J... K... L... F... ;  δ: Setting data (#6213)	 δ: Setting data

Table 2.25 Canned Cycle (continued)

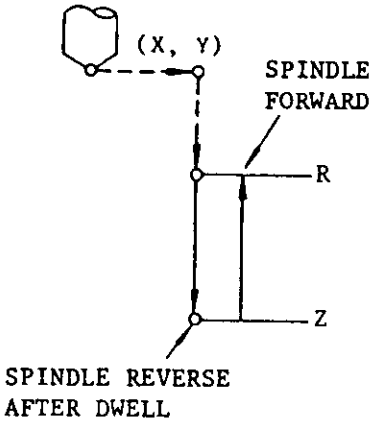
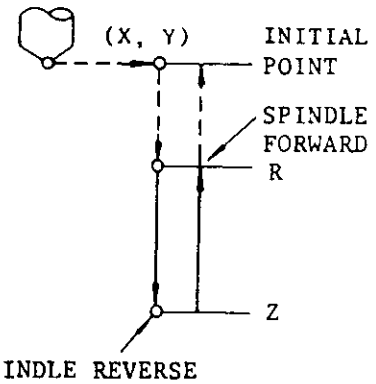
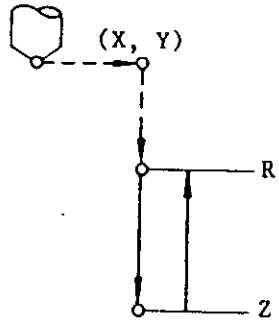
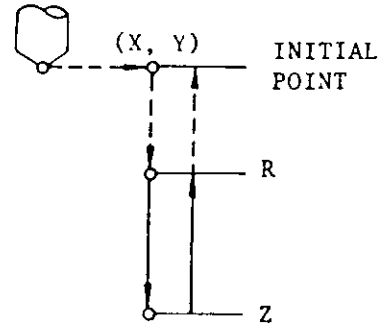
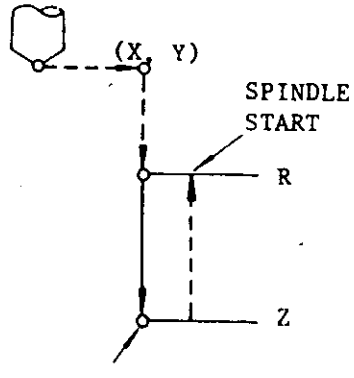
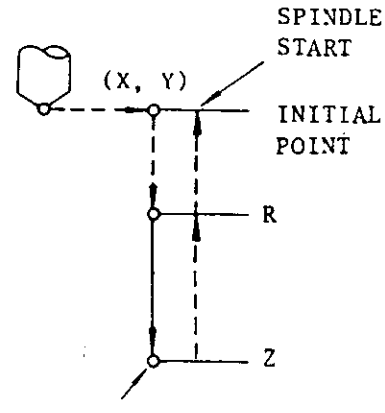
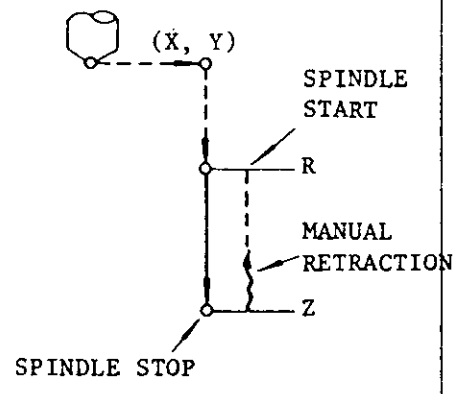
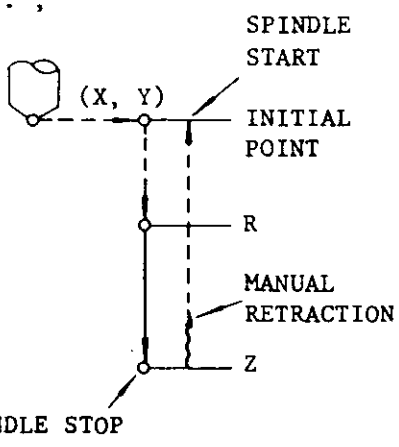
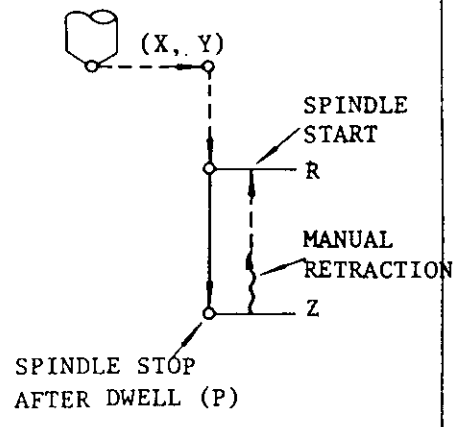
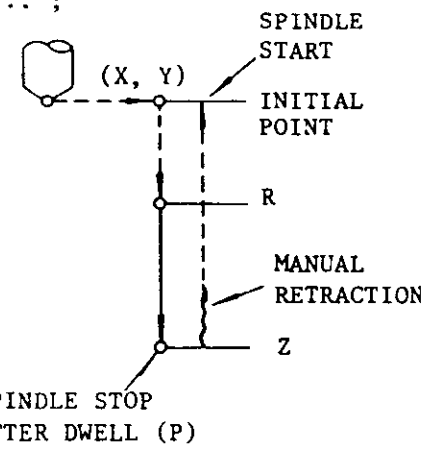
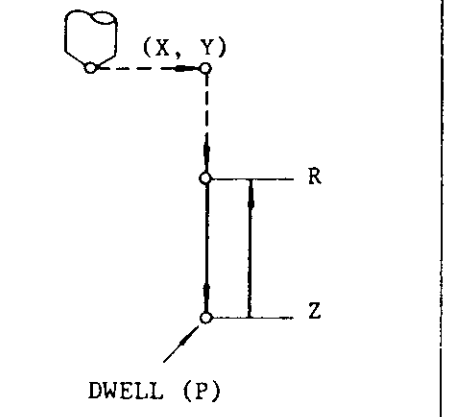
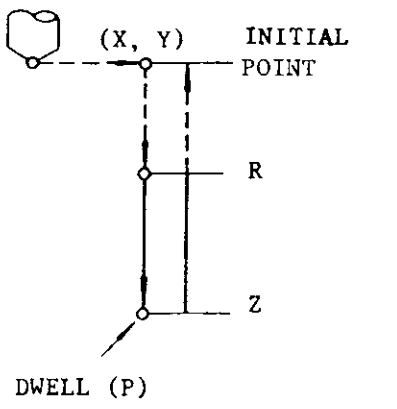
	With G99 (return to R)	With G98 (return to initial point)
G84	G84 X... Y... Z... R... P... L... F... ;	
Tapping		
G85	G85 X... Y... Z... R... L... F... ;	
Boring		
G86	G86 X... Y... Z... R... L... F... ;	
Boring		

Table 2.25 Canned Cycle (continued)

	With G99 (return to R)	With G98 (return to initial point)
G87	G87 X... Y... Z... R... Q... L... F... ;	
Boring		
G88	G88 X... Y... Z... R... P... L... F... ;	
Boring		
G89	G89 X... Y... Z... R... P... L... F... ;	
Boring		

EXAMPLE

A. G98 G90 G81 X··· Y··· Z-7000 R-4000 F··· ;

··· Return to initial point, absolute

B. G99 G91 G81 X··· Y··· Z-7000 R-4000 F··· ;

··· Return to R point, incremental

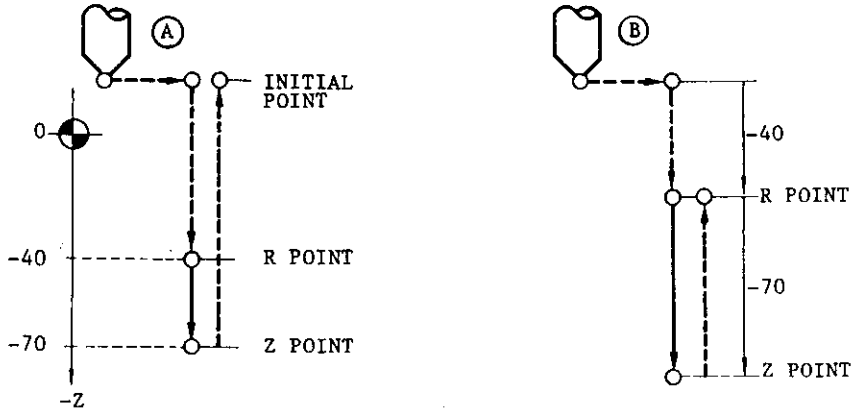


Fig. 2.82

C. N1 G92 X0 Y0 Z0 ;

N2 G98 G90 G81 X1000 Y1000 Z-5000 R-2000 F100 ;

N3 G91 X2000 R-3000 ;

N4 Z-5000 ;

N5 G99 Z-4000 R-5000 ;

N6 G80 G00 Z5000 ;

Absolute position		Remarks
Z =	R =	
-5000	-2000	Canned cycle start
-5000	-3000	Change in X, R
-8000	-3000	Change in Z
-9000	-5000	Change in R, Z
Tool moves to Z = 0		Canned cycle cancel

Newly programmed addresses only are changed including the case where switching is made from G90 to G91 such as N2 → N3 indicated in the above case. As for the non-programmed addresses, the positions programmed in the earlier blocks are maintained.

Note: Since address P, Q, I, J and K are modal in canned cycle mode, if once commanded, they are effective until the canned cycle is cancelled.

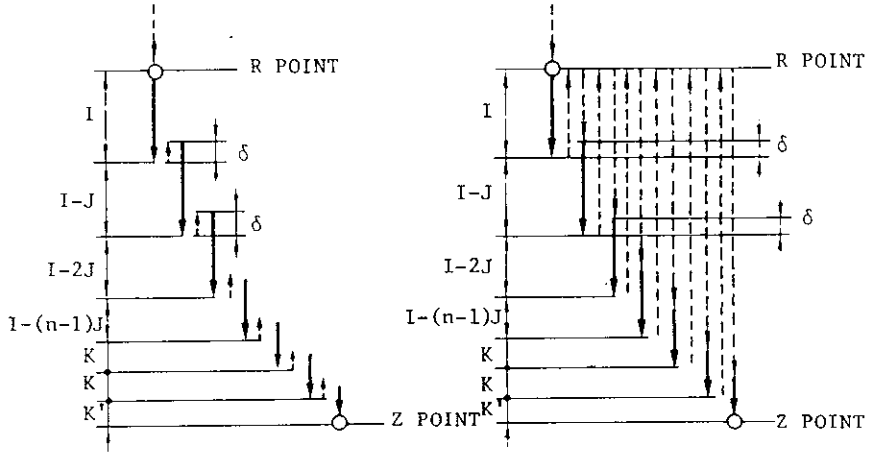
2.9.29 CANNED CYCLES (73, G74, G76, G77 G80 TO G89, G98, G99)†

· Variable pitch command (G73, G83)

In the deep hole drilling cycles of G73 and G83, variable drilling pitch can be programmed with addresses I, J, K instead of address Q for programming a constant drilling pitch.

I: Initial value
J: Reducing value in 2nd and subsequent plunges
K: Final value

} Command is given without signs



K': Final plunge to Z point

$K' \leq K$

(a) With G73 (b) With G83

Fig. 2.83

The value of δ is given by setting (#6211 for G73, #6213 for G83).

NOTES:

· Q, I, J, K are modal during canned cycle modes and are effective until the canned cycle is cancelled. Specify them without signs.

· Variable pitch can also be programmed by address Q instead of I. Furthermore, when instructions Q, I, J, K are given simultaneously, drilling cycle is executed with variable drilling pitch with Q as the initial value.
Q0 must be commanded in the block including modal G code before programming variable pitch with I, J, and K.

EXAMPLE

G91 G73 X··· Y··· T-3000 Z-5500 I1000···
J100 K400 F··· ;

Drilling pitch

1st plunge	10 mm	⇦	I1000
2nd plunge	9 mm		
3rd plunge	8 mm		
4th plunge	7 mm		
5th plunge	6 mm		
6th plunge	5 mm		
7th plunge	4 mm	⇦	K400
8th plunge	4 mm		
9th plunge	2 mm	⇦	K'
Total	55.00 mm	⇦	Z-5500

NOTES:

- When the canned cycles are executed by turning on the SINGLE BLOCK switch, a temporary stop is made in an intermediate position, and the FEED HOLD lamp lights up.

  (1) After positioning to (X, Y) point
  (2) After positioning to R point
  (3) After termination of each cycle, if L command has been given.

  The single block stop after the completion of canned cycles is as usual, and the FEED HOLD lamp does not light up.

- Be sure to designate R point and Z point by programming R and Z before entering the canned cycle mode. R point and Z point are cleared when canned cycles are cancelled.

- When executing canned cycles with the address data changed, the block requires any of the following address commands. The canned cycles will not be executed otherwise.

X, Y, Z, 4, R

- When M, S, T or B† code is given in the canned cycle, M, S, T signals are sent at the first positioning in the block. In general, M, S, T should be commanded in their own block.

- An input error occurs when any of the following G codes are programmed in the canned cycle mode.

G codes of * group except for G04 G70, G71, and G72
G codes of 07 group (G41, G42)

  When programming G92, G27, G28 etc., make sure to cancel the canned cycles in advance. Cancellation is made when a G code of 01 group is programmed.

- An independent block of dwell (G04) can be programmed in the middle of the canned cycle mode. Dwell is executed properly.

- An input error occurs when canned cycles are programmed in the tool radius compensation C mode (G41, G42).

- Start of spindle forward or reverse (M03 or M04) should be executed by automatic operation commands before entering canned cycles. Do not enter into canned cycles after manually switching the spindle between forward and reverse.

- Execution of subprogram (M98) in canned cycle mode. In a canned cycle mode, M98 P··· L··· ; can be programmed to call up subprogram and the canned cycle is continued in the subprogram. The address P (program No. of the first block of subprogram) with M98 command destroys temporary the contents of address P for designation of dwell time, but after the jumping to subprogram, it resumes the contents.

Note:

- Programming consideration of M98 in the canned cycle mode is the same as those of other than canned cycle modes. (e.g. Restriction of execution to no more than four levels, M98 command from punched tape and the like.)

- Address L for designation of repetition number of subprograms is nonmodal. But described below is a special case that the address L is retained temporarily.

EXAMPLE

G91 G81 X1000 R-2000 Z-3000 F100 ;

L3 ; ··· The canned cycle is not executed because X, Y, Z, 4 or R is not designated in this block. The L3 is retained.

X2000 ; ··· The canned cycle G81 is executed 3 times using the retained L3. After the execution, the L3 is erased.

As mentioned above, address L in canned cycle is retained until actually executed.

- Changing of R point and Z point

  When R is commanded instead of Z during the execution of canned cycle in G91 mode, Z becomes incremental value from the new R point. Care should be taken.

  G92 X0 Y0 Z0

  G91 X··· Y··· R-5.0

	R point	Z point
Z-10.0F ;	-5.0	-10.0
X··· R-7.0 ;	-7.0	-12.0
X··· Z-3.0 ;	-7.0	-10.0
R-4.0Z-11.0 ;	-4.0	-15.0

EXAMPLE

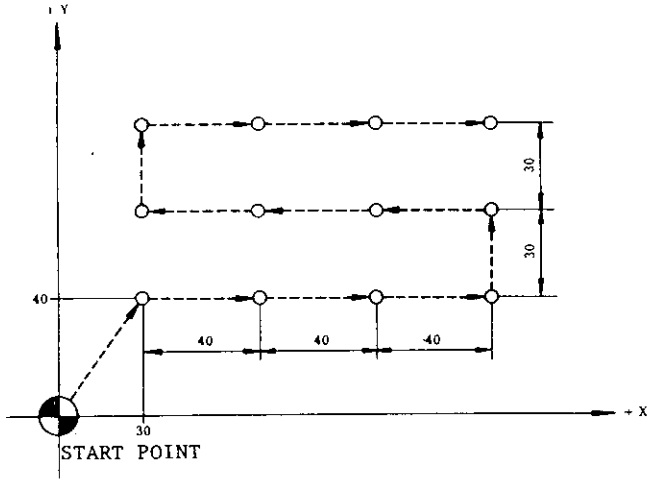


Fig. 2.84

```
N10  G92  X0  Y0  Z0  ;
N11  G90  G98  ;                          . . .   Return to initial point, Absolute
N12  G81  X3000  Y4000  R-2000  Z-3000  F200  ;  . . .  Drilling cycle
N13       M98  P100  ;                    . . .   Jump to subprogram
N14  G00  X0  Y0  ;
N15       T05  ;                          . . .   Tapper selection
N16       M06  ;                          . . .   Tool change
N17  G84  X3000  Y4000  R-2000  Z-3000  F2000  ;  . . .  Tapping cycle
N18       M98  P400  ;                    . . .   Jump to subprogram (Note)
N19  G00  X0  Y0  ;
  .
  .
  .
O400
N100  G91 X4000  L3  ;
N101      Y3000  ;
N102      X-4000  L3  ;
N103      Y3000  ;
N104      X4000  L3
N105  G90 G80  ;
N106      M99  ;
```

Subprogram for drilling position pattern.

2.9.30 ABSOLUTE/INCREMENTAL PROGRAMMING (G90, G91)

These G codes are for designating whether the movement data following the axis address are in absolute value or incremental value.

· G90 . . . Absolute designation

In the block including G90 and in the subsequent blocks, the movement data which follow addresses X, Y, Z, $\alpha^{\dagger}$ are regarded as absolute values.

G90 G00 X··· Y··· Z··· ;
··· Absolute designation

· G91 . . . Incremental designation

In the block including G91 and in the subsequent blocks, said data area is regarded as incremental values.

G91 G01 X··· Y··· Z··· ;
··· Incremental designation

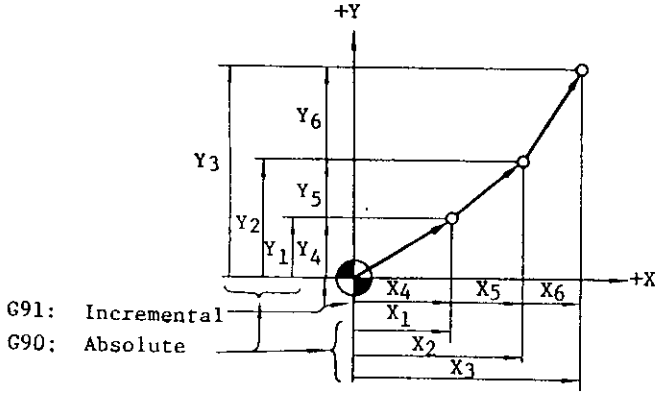


Fig. 2.85

· G90, G91 are modal G codes of 03 group.

· If both G90 and G91 are programmed in the same block, the G code which was programmed last is valid.

NOTE:

· The initial state of these G codes when the power is turned on can be designated by parameter $\#6005_{D0}$.

Parameter($\#6005_{D0}$)	Initial state
"0"	G90
"1"	G91

2.9.31 PROGRAMMING OF ABSOLUTE ZERO POINT (G92)

It is necessary to program the absolute zero point before programming movement command. When an absolute zero point is programmed, one absolute coordinate system is determined, and all absolute movement commands programmed thereafter will move the tool on the programmed coordinate.

· G92 X··· Y··· Z··· ($\alpha^{\dagger}$···) ;

With this command, the current position of the tool is programmed in the control as absolute coordinate point (X, Y, Z, $\alpha^{\dagger}$). That is, program the distance (with sign) from the desired absolute coordinate zero position (0, 0, 0, $0^{\dagger}$) to the current position. In other words, G92 command is for designating the position of the "absolute zero point."

EXAMPLE

G92 X50000 Y30000 Z40000 ;

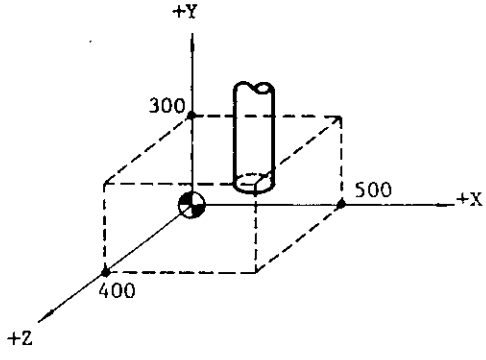


Fig. 2.86

· G92 is a G code of non-modal group which is valid only in the programmed block. It is not possible to program other G codes, F, M, S, T, $B^{\dagger}$ codes in the same block.

NOTES:

· In principle, program G92 in the state where all tool offset modes are cancelled.

· When the power is turned on, the current position of the tool is set as absolute zero point (0, 0, 0, $0^{\dagger}$). Make sure to reprogram absolute coordinate by G92 before executing the automatic operation.

2.9.31 PROGRAMMING OF ABSOLUTE ZERO POINT (G92) (CONT'D)

- The programmed absolute zero point is not affected by reset operation. Perform any of the following operations for resetting the absolute zero point.
  1. Use ORG key (see 4.1.9).
  2. Write G92 X0 Y0 Z0 α† 0 ; in MDI mode, and then execute.
  3. Turn the power off and on again.

2.9.32 FEED FUNCTION DESIGNATION (G94, G95)†

These G codes are for selecting whether to designate the feed in mm/min. or in mm/rev. prior to programming the F code for feed, in the case where the control is equipped with feed per revolution* function.

- When G94 ; is programmed, the F code programmed thereafter is executed in mm/min. (or inch/min.†, deg/min.†).
- When G95 ; is programmed, the F code programmed thereafter is executed in mm/rev. or inch/rev. (deg/rev.†).
- G94, G95 are modal G codes of F group, and G94 is selected when the power is turned on.
- When switching between G94 and G95 is made, the previously programmed F code is cancelled. Therefore, a new F code must be programmed.

2.9.33 HIGH-SPEED CUTTING FEATURE (G100 THROUGH G102)†

In this feature, a division in a part program is designated by a G code and the data processing of the designated division is performed before entering the automatic operation mode. This feature, therefore, eliminates the interblock stoppage time due to compensation calculation, providing an uninterrupted machining operation if short-distance blocks are consecutively specified. This feature is of two types: "sequential processing mode" and "memory processing mode," which may be selected by specifying one of the G codes shown below:

High Speed Cutting G codes

G code	Meaning
G100	High-speed cutting cancel
G101	High-speed cutting in sequential processing mode ON
G102	High-speed cutting in memory processing mode ON

2.9.33.1 High Speed Cutting in Sequential Processing Mode (G101)

```
G101 ;
  :
  :
G100 ;
```

Every time the above is specified, the data processing and data registration of this division are performed. The automatic operation is followed by the registered data. The above command may be used repeatedly in one part program. However, the number of blocks in a program between G101; and G100 ; is limited as shown in the table below. The part program memory capacity is also limited to 1/2 as shown in the same table.

No.	Part Program Capacity in Usual Operation	High-Speed Cutting Function ON	
		Part Program Memory Capacity	No. of Allowable Blocks for High-Speed Cutting
1	40 m	20 m	480 blocks
2	80 m	40 m	672 blocks
3	150 m	90 m	1504 blocks
4	320 m	150 m	4064 blocks

2.9.33.1 High-Speed Cutting in Memory Processing Mode (G102)

```
G102 P··· ;
  .      └── REGISTRATION NO. OF
  .          HIGH SPEED CUTTING
  .          PROGRAM
G100 ;
```

By the above command, data processing and data registration are first performed then the automatic operation is performed using the registered data. Thereafter, when the same command given, the automatic operation is immediately performed using the registered data. In address P, designate the registration number of the program to be registered in memory processing mode. The registration number is of a maximum of 4 digits (P0 through P9999). When P designation is omitted, P0000 is assumed. This P designation allows the registration of a maximum of 20 high-speed cutting programs.

Note: The "registration number" by P designation has no relationship with the "program number" by O designation. Hence, the registration number is not subjected to the address search operation. The contents of the program which was data-processed and registered cannot be displayed on the CRT screen. The number of blocks in one part program between G102 ; and G100 and the limits to the part program memory capacity are as shown in Table 2.9.32.2.
The total number of blocks in the maximum of 20 registered high-speed cutting programs should not exceed the value allowed in Table 2.9.33.2.

2.9.33.2 Restrictions on High-Speed Cutting Feature

(1) When the high-speed cutting feature is made valid by parameter setting, the part program memory capacity is decreased to about 1/2 (see Table 7.1.1 and

(2) The program between G101 (or G102); and G100 ; should follow the restrictions shown below:

a. The addresses that can be specified are as follows:

O, /(slash), N, G, X, Y, Z, I, J, K, R, D, H, M, L, F

The 4th-axis address cannot be specified.

b. The G codes that can be specified are as follows:

G00, G01, G02, G03, G09, G40 through G42, G43, G44, G45 through G48, G49, G90, G91

If any G code other than above is specified, program error "142" will be caused.

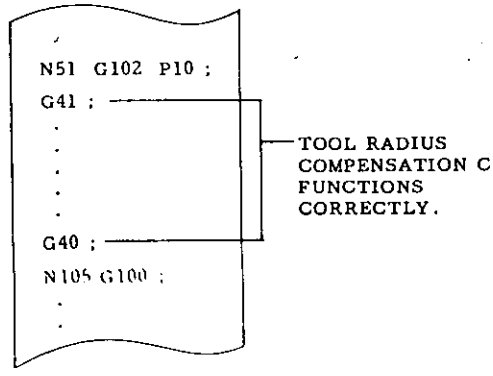
c. The M code that can be specified are as follows:

M90 through M99

If any M code other than above is specified, program error "142" will be caused.

Considerations and Remarks

(1) Do not enter the high-speed cutting mode (G101 or G102) with the tool radius compensation C mode being on. Cancel (by G40) the compensation C mode. Otherwise, program error "024" will be caused. With the high-speed cutting mode being on, the compensation C mode can be entered. However, before canceling the high-speed cutting by G100 ; command, cancel the compensation C by G40 command. Otherwise, program error "024" will be caused.



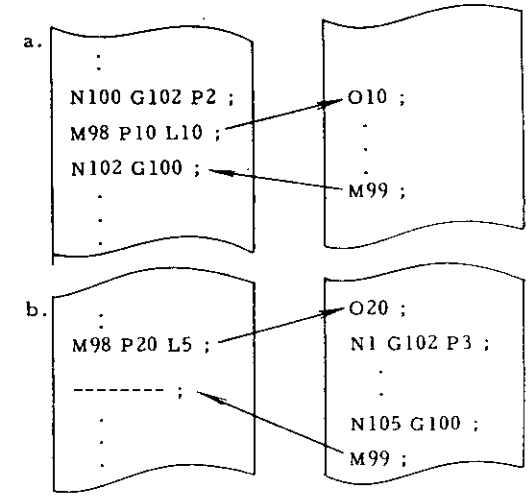
(2) The tool length compensation and tool position offset features are not affected by G100 through G102 commands and therefore always valid.

(3) The execution of G102 P··· ; command, thereafter, needs the use of the data processed and registered inthe first execution, so that the following points should be taken into account:

a. The tool offset amount to be used should be the same as that used in the first execution. To change the tool offset amount, reset the registered data and execute the command again for the first time (see 7.4, (7)).

b. If there is a single coordinate command of absolute value in a program of memory processing mode, the absolute coordinate value immediately before G102 P··· ; command must be in the same position as that used in the first execution. Otherwise, the correct position cannot be obtained as specified by the absolute command in the program.

(3) Specifying a subprogram (M98) is as follows: as follows:



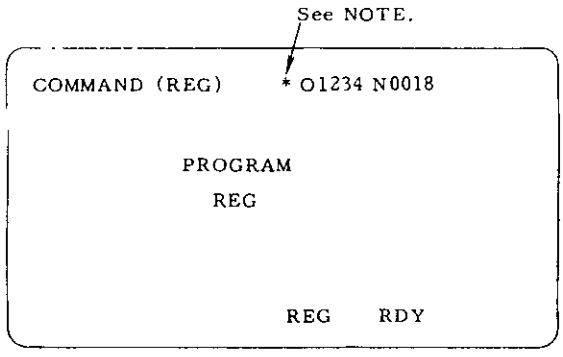
(4) G101 ;
G101 P··· ;
G100 ;

These commands should be specified independently.

(5) The blocks between G101 (or G102 P) ; and G100 ; are handled as a single block with regard to the single block switch. That is, the single block stop feature is disabled between these commands.

(6) During the data processing and registration of the high-speed cutting program, the display on the CRT screen is as shown below. The function keys are automatically put in the COM display mode, displaying the screen shown below. At this time, the NC operator's station keys except for RESET are disabled.

2.9.33.2 Restrictions on High-Speed Cutting Feature (Cont'd)



Note: During data processing and registration, * is displayed to the left of the program number. With G101 ; command, * is erased after the completion of registration, upon which the automatic operation mode is entered. With G102 P ; command, * remains displayed after the completion of registration until the registered program is erased.

When a reset operation is performed, the data being registered is erased and the reset state is provided. If FEED HOLD button is depressed or a mode switching operation is performed, the data being registered is erased and the label skip state is provided.

(7) The processed and registered program data is erased by any of the following operations:

a. NEXT key on NC operator's station was pressed.

b. Address search was performed in MEM or EDIT mode.

c. Some edit operation was performed either on the stored part program or on the high-speed cutting program.

Note: To change the tool offset amount after data processing and registration, depress NEXT key to erase the registered program data. In the case of the high-speed cutting (G101) in sequential processing mode, however, the data processing and registration are performed each time, eliminating the need for depressing NEXT key.

(8) The following parameter setting may invalidate the high-speed cutting feature temporarily. When this feature is invalidated, the part program memory may be used to its full capacity.

Parameter Number

#6008

D7 = 1 ··· Disables high-speed cutting function

D7 = 0 ··· Enables high-speed cutting function

However, to change the part program memory capacity, key in 0 - 9 9 9 9 ERASE (by EDIT, PROG) for both changes of D7 from 0 to 1 and from 1 to 0. Hence, in either change, all the part programs are erased. If G101 or G102 is specified when #6008 D7 = "1," error "147" will be caused.

(9) The following setting may provide the mode in which only data registration is performed:

Setting Number

#6004

D5 = 1 ··· The high-speed cutting is executed normally.

D5 = 0 ··· The high-speed cutting is not executed for G101 ; command but the usual automatic operation is performed. For G102 P··· ; command, only data registration is performed during the usual automatic operation.

This setting may be used for the checking of programs as follows:

a. Set #6004D5 to 0.

b. Select low-speed feed by FEEDRATE OVERRIDE or DRY RUN switch.

c. Cycle-start the program to be tested including G102 P ; through G100 ; commands. The usual automatic operation is performed, so that visually check the tool path. In some cases, the program may be single-block-stopped for checkups. Upon termination of the operation, the processed data up to G100 ; command have been registered in the machine.

d. Set #6004 D5 to 1.

e. Set the feedrate to the original value.

f. Cycle-start the same program. G102 P ; through G100 ; are immediately executed using the registered data.

2.10 USER MESSAGE DISPLAY

2.10.1 ALARM MESSAGE DISPLAY BY #8000 COMMAND

The user can display any alarm message on the CRT screen. By specifying the following command in the part program, 3-digit alarm number n and the alarm message (less than 32 characters) may be designated:

#8000 = n (<alarm message>);

The alarm number should be 3 digits and the one that is not used by the control. For the alarm message, alphanumeric characters may be used. When this #8000 command is executed, the alarm message is displayed and "ALM" flashes at the bottom of the CRT screen.

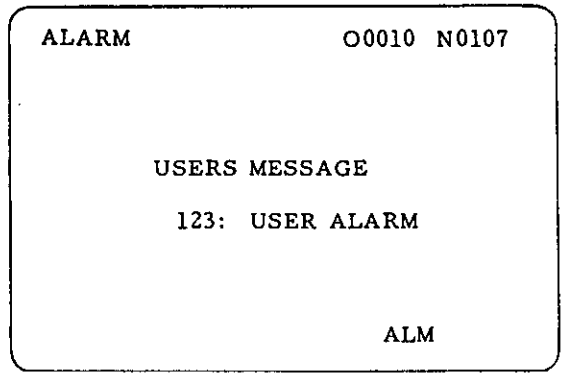


Fig. 2.87 Sample Display of Alarm Message by #8000 Command

Note: If a command with a wrong alarm message is specified, a format error will be caused.

2.10.2 MESSAGE DISPLAY BY CONTROL-OUT AND CONTROL-IN

(1) Programming and Display by Control-Out and Control-In

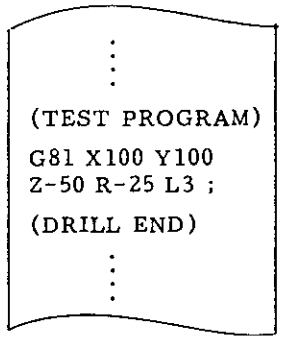
The user can program any message between the information enclosed with control-in and control-out in the part program in order to display the message on the CRT screen. The enclosed information is assumed to be insignificant.

(2) How to Edit Control-Out And Control-In

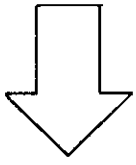
The control-in and control-out part may be edited by the usual edit operation.

a. Depress [(2] key and [SHIFT] key, and character "(" may be entered.

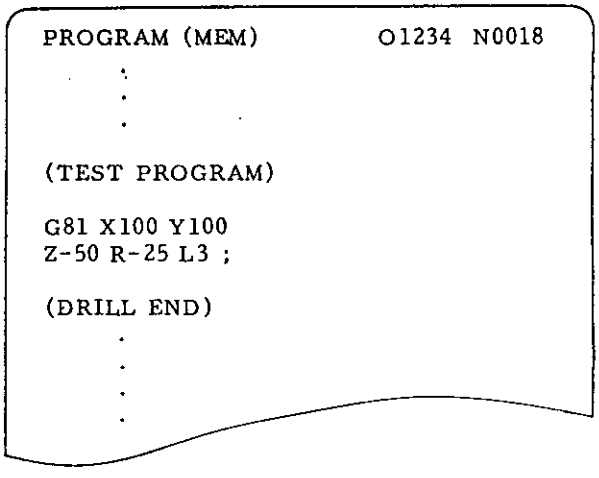
b. Depress [(3] key and [SHIFT] key, and character ")" may be entered.



Program Example



Message Display by Control-out and Control-in

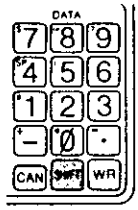


Notes:

1. The characters which are indicated in the thick-lettered keys shown below may be enclosed in control-out and control-in.

2. The number of characters that may be enclosed in control-out and control-in less than 32.

3. Nesting of the control-out and control-in is not allowed.



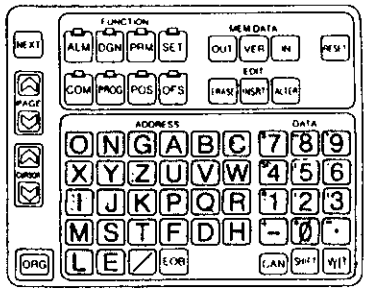


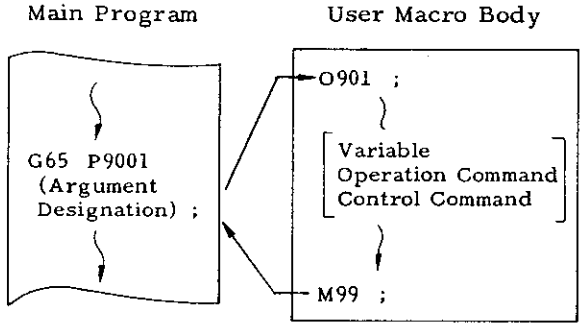
Fig. 2.88

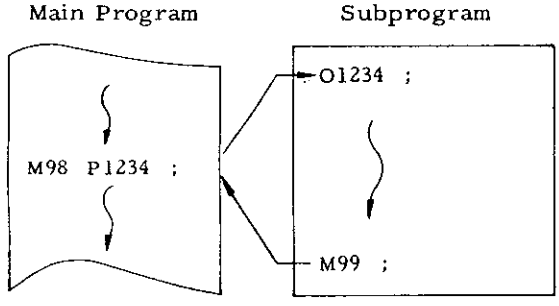
Characters Enclosed in Control-Out and Control-In (Thick-Lettered Keys)

2.11 USER MACRO (G65 AND G66)

Special programs written by the machine builder or user by the use of a group of instructions are registered in the part program memory. These programs can be called by the use of G65 or G66 command to execute them.

These special programs are referred to as the user macro body, which can be written and stored in the same format as a subprogram.





However, unlike a subprogram, a user macro allows:

(1) Use of variables.

(2) Computation between variable or between constants.

(3) Use of control commands such as conditional branch.

These features enable the user macro body to provide a generalized program that requires complicated computations and decisions.

The "argument designation" in calling a user macro body from the main program makes it possible to assign the real numbers to the variables in the body. This enables this user macro to run as a series of specific program that provide tool movements.
In this manual, the user macro body is sometimes referred to as "macro program" or, simply, "macro."

2.11.1 USER MACRO CALL COMMANDS

A user macro body may be called in the following five manners:

No.	Type of Call	Code	Remarks
1	Simple call	G65	
2	Modal call	G66	G67: For cancel.
3	Call by arbitrary G code	Gxx	
4	Call by M code	Mxx	
5	Call by T code	Txxxx	4 digits max.

(1) Simple Call (G65)

G65 P··· L··· (argument designation) ;

The macro program whose program number was specified by P is called and is executed L times. The default value of L is 1. When the designation of an argument to the user macro is desired, specify it in (argument designation). "Argument designation" is the assignment of real numbers to the "local variables" used in the user macro. For details, see 1.3 ARGUMENT DESIGNATION.

(2) Modal Call (G66 and G67)

G66 P··· L··· (argument designation) ;

This command provides the mode to call the macro program of the program number specified by P. Each time a move command is executed, the specified macro is run L times.

G67 ;

This command cancels the modal call mode.

(3) Macro Call by Arbitary G Code

Gxx (argument designation) ;

This provides the command which is equivalent to G65 P··· (argument designation) ;. For Gxx, ten sets of G codes of G01 through G199 except for those designated by NC maker may be set for the parameter. The macro program numbers which correspond to these G codes are as follows:

#6120 ··· Sets G code which calls the macro of program number O9010.

#6121 ··· Sets G code which calls the macro of program number O9011.

)

#6129 ··· Sets G code which calls the macro of program number O9019.

NOTE: Macro call by arbitrary G code permits only single nesting. Namely, the macro which was called by the macro call using arbitrary G code or the macro call with M code or T code does not permit another macro call by arbitrary G code.

(4) Macro Call by M Code

G··· X··· Y··· Mxx ;

This command may call macros. In this case, the macro is executed after the execution of the move command in the same block. MF and M codes are not transmitted. For Mxx, four sets of M codes of except for M00, M01, M02, M30, M90 through M99 may be set for the parameter.

#6130 ··· Sets M code which calls the macro of program number O9001.

#6131 ··· Sets M code which calls the macro of program number O9002.

#6132 ··· Sets M code which calls the macro of program number O9003.

#6133 ··· Sets M code which calls the macro of program number O9004.

NOTE THAT THE MACRO CALL BY M CODE DOES NOT PERMIT ARGUMENT DESIGNATION.

When these M codes are specified in the macros called by the macro call using arbitrary G code or by the macro call using M code or T code, the macro call is not executed but is handled as a usual M code.

(5) Macro Call by T Code

All the T code commands provide a macro call command.

G··· X··· Y··· Txxxx ;

With this command, the macro of program number O9000 is executed after the move command in the same block is executed:

Transmission of T code and TF signal is not performed. Whether a T code is to be used as a macro call command may be specified by the following parameter:

Parameter No.

#6134

0 ··· T code designation is handled as a T code.

1 ··· T code designation is handled as a macro call command to call the macro of program number O9000.

When a T code is specified as a macro call command, the value designated by T "xxxx" (up to decimal 4 digits) becomes the argument of common variable #149. NOTE THAT THE ARGUMENT DESIGNATION OTHER THAN THIS IS NOT PERMITTED. When a T code is designated in the macro called by the macro call by arbitrary G code or by the macro call by M code or T code, the macro call is not executed but is handled as a usual T code.

(6) Multiple Call

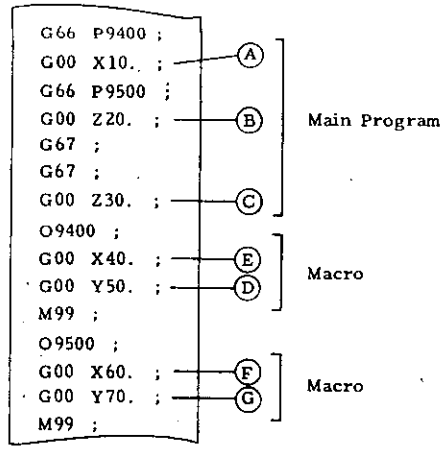
A. G65 Simple Call And G66 Modal Call

As a subprogram is called from another subprogram, a user macro may be called from another user macro. Quadruple nesting is permitted for simple call and modal call combined. Multiple call is disabled for the macro call by arbitrary G code, or M code or T code.

B. Multiple Call by G66 Modal Call

In modal call, each time a move command is executed, the designated macro is run. This is also valid for the move command in the macro called by multiple call. The macros are sequentially executed from the one designated latest.

Sample Program



The above sample program is executed in the following order:

Ⓐ → Ⓔ → Ⓓ → Ⓑ → Ⓕ → Ⓔ → Ⓓ → Ⓖ → Ⓔ → Ⓓ → Ⓒ

2.11.2 ARGUMENT DESIGNATION

Argument is the real value to be assigned to a variable used in the user macro body. Argument designation, therefore, is the act of assigning real values to variables. Argument designation is of type I and type II, which can be selected as required.

(1) Argument Designation I

Argument may be designated in any address except for G, L, N, O, and P. The relationships between the argument designation addresses and the variables are as shown below.

Address of Argument Designation I	Variable in User Macro Body
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

For the address in which no argument need be designated, the command may be omitted.

Sample Program

```
                #1   #2    #24     #26
                 ↑    ↑     ↑       ↑
G65 P9100 L3 A30  B60  X34.8  Z50.2  ;
```

Argument Designation Part

(2) Argument Designation II

A, B, and C arguments and 10 sets of I, J, and K arguments may be designated. I, J, and K must be designated in this order. The relationships between the argument designation addresses and the variables are as shown below.

Address of Argument Designation II	Variables in User Macro Body
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	#13
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

The suffixes 1 through 10 to I, J, and K are determined by the order of the designated I, J, and K combinations.

For the address in which no argument need be designated, the command may be omitted.

Sample Program

```
                                #4   #5   #6   #7   #9
                                 ↑    ↑    ↑    ↑    ↑
G65 P9005 A... B... C... I... J... K... I... K... ;
```

Argument Designation Part

(3) Position of Decimal Point Argument

An argument may generally be designated with a sign and decimal point. For the designation without decimal point, the position of decimal point is as shown on the next page.

Address in Argument Designation	Metric Input	Inch Input
A, C	3 (2)	3 (2)
B (Without B 3-digit option)	3 (2)	3 (2)
B (With B 3-digit option)	0	0
D, H	0	0
E, F (In G94 mode)	0 (1)	1 (2)
E, F (In G95 mode)	2 (3)	3 (4)
I, J, K	3 (2)	4 (3)
M, S, T	0	0
Q, R	3 (2)	4 (3)
U, V, W	3 (2)	4 (3)
X, Y, Z	3 (2)	4 (3)

The value shows the position of decimal point as counted from the least significant digit. The value in parentheses indicates the number of digits that follows decimal point at the time of parameter #6020D_0 = 1, D_2 = 1 for addresses E and F, and parameter #6006D_5 = 1 for the other addresses.

(4) Considerations in Argument Designation

A. Argument designation types I and II may be used concurrently. If the same variable has been designated duplicately, the last one is validated.

B. For both types I and II, addresses I, J, and K should be designated in this order. The other addresses may be designated in any order.

C. In the argument designation part, negative sign and decimal point may be used regardless of the address.

D. In G65 and G66 blocks, G65 and G66 should always be specified before each argument designation. This holds true with the macro call by G code.

2.11.3 OVERVIEW OF USER MACRO BODY

A user macro body is programmed using the combination of the following commands.

(1) Variables

A. Local variable (#1 through #33)

B. Common variable (#100 through #509)

C. System variable (#1000 through #5104)

(2) Operation Commands

A. Arithmetical operations (+, -, *, /, ···)

B. Functional operations (SIN, COS, ROUND,···)

(3) Control Commands

A. Branch command (IF ⟨qualification⟩ GO TO n)

B. Repeat command (WHILE ⟨qualification⟩ DO m)

Using these commands, a program which requires complicated operations and conditional judgements may be written in the general format. Hence, the feature of user macro is to enable the programming of the wide range of NC functions from a simple machining cycle which is rather a subprogram to a special, complicated canned cycle, and the storing of these cycles in the machine. Described below are details of the commands mentioned above.

2.11.4 VARIABLES

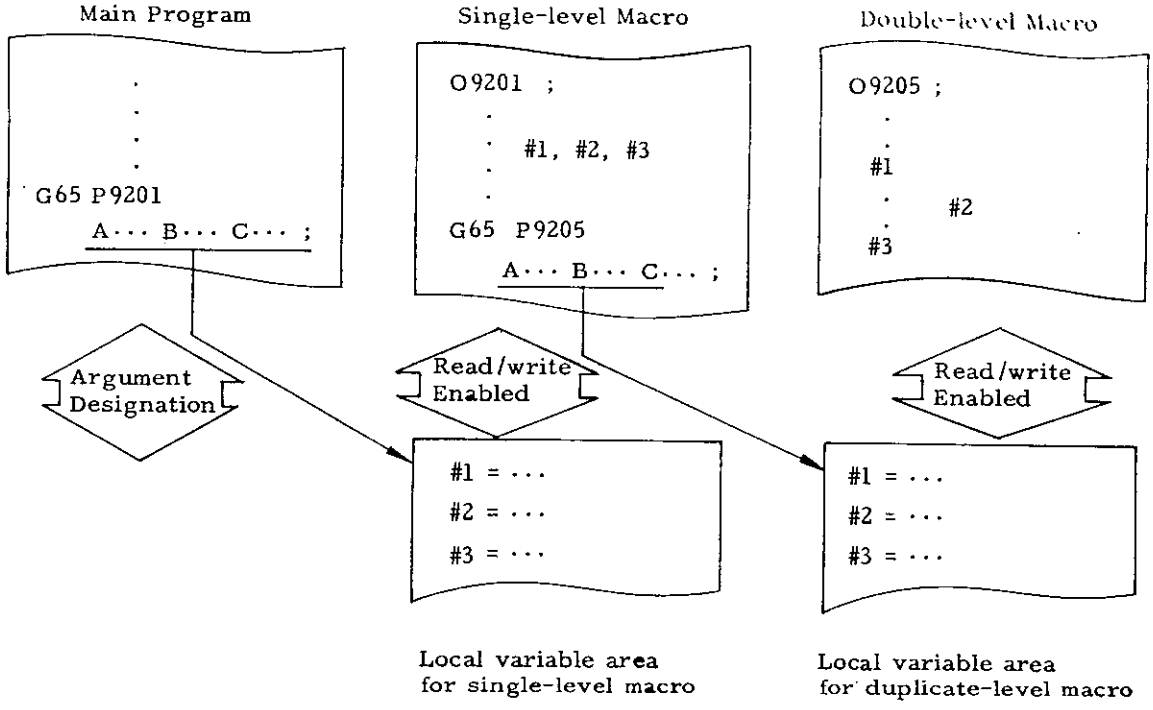
Instead of directly assigning a value to an address in a user macro body, the address may be designated by a variable. When this variable is called during execution, the corresponding value is fetched from the variable area to provide the address value.

There are three types of variables: local variable, common variable, and system variable. Each is identifiable by a variable number.

To the local variables, real numbers can be assigned using the argument designation part of macro call command by G65 or G66.

(1) Local Variables (#1 through #33)

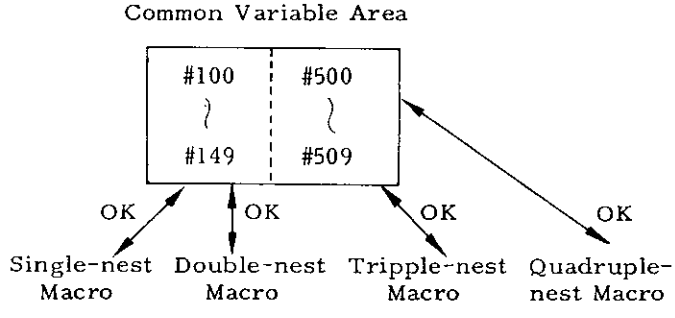
A local variable is the one that is used for each macro locally, That is, when the local variable is used, the variable area (#1 through #33) is independently allocated for each macro call, certain values are stored by argument designation, and the results of operations in macro are retained.



Hence, the variables #1, #2, #3, ··· of the same macro assume different values each time it is called. Each local variable is reset for each macro call and is registered by argument designation. The variable not designated becomes "blank." Each local variable is set to "blank" at the time of power-on and reset operations.

(2) Common Variables (#100 through #149, #500 through #509)

A common variable may be shared by all macros and through all macros of all nesting levels. That is, the common variable enables a macro to refer to the results obtained by another macro.

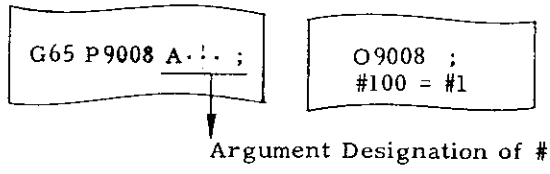


Common variables are divided into the following two types depending on clear conditions:

A. #100 through #149: These common variables are cleared at the time of power-on and reset operations and are set to "blank." They are not cleared by reset operation in some controls if parameter #6008 D1 is set to "1."

B. #500 through #509: These common variables are not cleared at the time of power-on and reset operations.

The common variables are available to the user without restrictions. They cannot be designated by arguments. Indirectly, however, they can be designated as follows:



(3) System Variables

A system variable is the one whose use is unique to the system. There are following types of system variables:

A. Interface input signals ... #1000 through #1015, #1032†

B. Interface output signals ... #1100 through #1115, #1132†

C. Tool offset amount and work coordinate-system shift amount ... #2001 through #2099, #2500 through #2806

D. Alarm message display ... #3000

E. Clock ... #3001, #3002

F. Single-block stop and auxiliary-function completion wait control ... #3003

G. Feed-hold, feedrate-override, and exact-stop control ... #3004

H. RS232C data output ... #3100 (print out feature).

I. Modal information ... #4001 through #4120

J. Position information ... #5001 through #5014

Note: The interface input and output signals of (A) and (B) may not be installed. Follow the specifications of the machine tool builder.

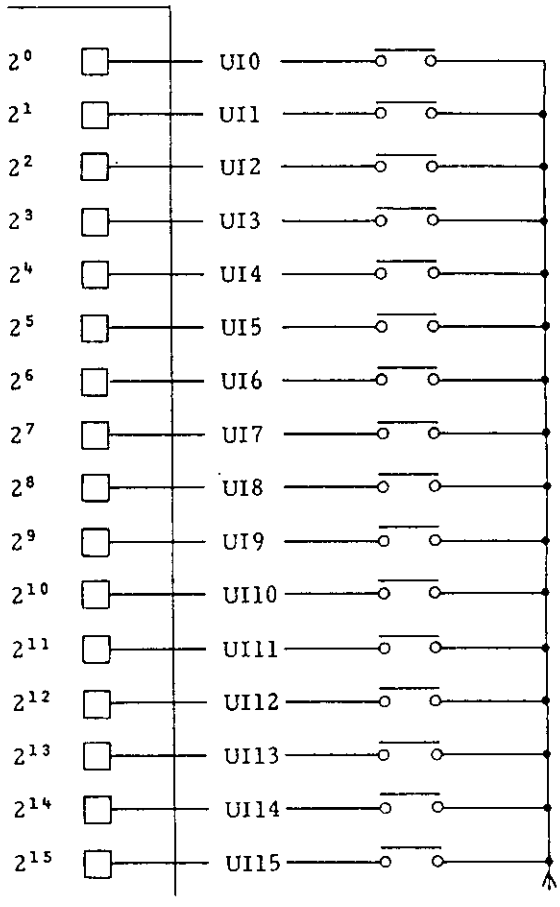
The following paragraphs describe the details of the variables mentioned above.

A. Interface Input Signals (#1000 Through #1015, #1032)†

a. When one of system variables #1000 through #1015 is specified to the right-hand of an operational expression, the on/off state of each of user-macro-dedicated 16-point input signals is read. The relationships between the input signals and the system variables are as shown below.

#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000
UI7 2^7	UI6 2^6	UI5 2^5	UI4 2^4	UI3 2^3	UI2 2^2	UI1 2^1	UI0 2^0
#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008
UI15 2^{15}	UI14 2^{14}	UI13 2^{13}	UI12 2^{12}	UI11 2^{11}	UI10 2^{10}	UI19 2^9	UI18 2^8

Variable Value	Input Signal
1	Contact Closed
0	Contact Open



Each read variable is 1. 0 or 0.0 when the associated contact is "closed" or "open" respectively, regardless of the unit system of the machine.

b. When system variable #1032 is designated, the input signals (UI0 through UI15) that consist of 16 points (16 bits) are collectively read as a decimal positive value.

$$\#1032 = \sum_{i=0}^{15} \#\left[1000 + i\right] * 2^i$$

Sample Program

IF [#1015 EQ 0] GO TO 100 ;

Bit 2^{15} (UI15) is read and, if it is "0," a branch is made to sequence number N100.

#130 = #1032 AND 255

Bits 2^0 through 2^7 (UI0 through UI7) are collectively read to be stored in common variable #130 as a decimal positive value.

Note: System variables #1000 through #1032 cannot be placed to the left-hand of operational expressions.

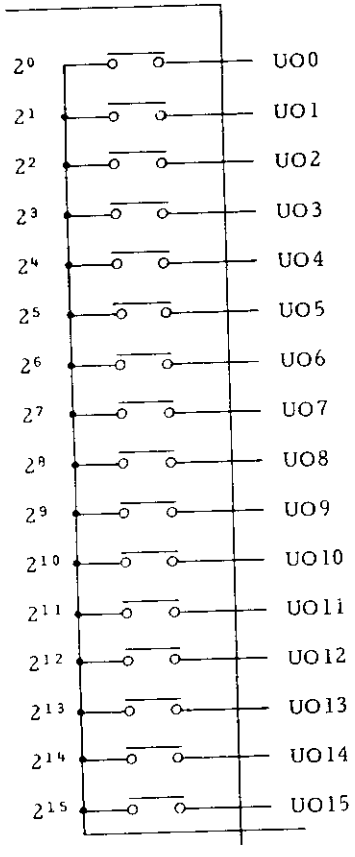
B. Interface Output Signals (#1100 Through #1115, #1132)†

a. When one of system variables #1100 through #1115 is specified to the left-hand of an operational expression, an on or off signal can be sent to each of user-macro-dedicated 16-point output signals. The relationships between the output signals and the system variables are as shown below:

#1107	#1106	#1105	#1104	#1103	#1102	#1101	#1100
UO7 2^7	UO6 2^6	UO5 2^5	UO4 2^4	UO3 2^3	UO2 2^2	UO1 2^1	UO0 2^0
#1115	#1114	#1113	#1112	#1111	#1110	#1109	#1108
U15 2^{15}	U14 2^{14}	U13 2^{13}	U12 2^{12}	U11 2^{11}	U10 2^{10}	U09 2^9	U08 2^8

Variable Value	Output Signal
1	Contact Closed
0	Contact Open

YASNAC



When 1.0 or 0.0 are substituted in any of #1100 through #1115, the associated output contact is output in the "closed" or "open"state.

b. When system variable #1132 is specified, the output signals (UO0 through UO15) that consist of 16 points (16 bits) are collectively output. At this time, the decimal positive value substituted in #1132 is output in the form of binary 16-bit value.

$$\#1132 = \sum_{i=0}^{15} \#\left[1100 + i\right] * 2^i$$

c. With system variables #1100 through #1132, the value sent last is retained. Hence, when one of them is written to the right-hand of an operational expression, its value is read.

d. Considerations

When any values other than 1.0 or 0.0 are substituted into one of #1100 through #1115, the values are handled as follows:

"Blank" is assumed to be "0."
Values other than "blank" and 0 are assumed to be "1."

Sample Program

#1107 = #10 ; (#10 = 1.5)

The output signal of bit 2^7 (UO7) is output in the contact (closed) state.

#1132 = (#1132 AND 240) OR (#8 AND 15;)

The output signal of bits 2^4 through 2^7 (UO4 through UO7) are output without change and the contents of local variable #8 are output to the output signals of bits 2^0 through 2^3 (UO0 through UO3).
(Decimal 240) = 11110000,
(Decimal 15) = 00001111)

C. Tool Offset Amount And Work Coordinate System Shift Amount (#2001 Through #2099, #2500 Through #2086)

a. When one of system variables #2001 through #2099 is specified to the right-hand of an operational expression, the tool offset amount can be read.

b. When one of system variables #2500 through #2806 is specified to the right-hand of an operational expression, the work coordinate system shift amount (and the external work coordinate system correction amount) can be read.

c. The relationships between the tool offset numbers and the system variables are as shown below:

System Variable	Tool Offset No.
#2001	01
#2002	02
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
#2098	98
#2099	99

System Variable	Work Coordinate System Shift Amount	Spindle
#2500	External work coordinate system correction amount	X
#2501	G54	
.	.	
.	.	
.	.	
#2506	G59	
#2600	External work coordinate system correction amount	Y
#2601	G54	
.	.	
.	.	
.	.	
#2606	G59	
#2700	External work coordinate system correction amount	Z
#2701	G54	
.	.	
.	.	
.	.	
#2706	G59	
#2800	External work coordinate system correction amount	α
#2801	G54	
.	.	
.	.	
.	.	
#2806	G59	

(4) When one of the above system variables is specified to the left-hand of an operational expression, its value can be changed.

Sample Programs

a. #116 = #2016 ;

The contents of tool offset number 16 are substituted for common variable #116.

b. #2506 = #4 ;

The work coordinate system shift amount of G59 X-axis is erased and the contents of local variable #4 are set.

D. Alarm Message Display (#3000)

When a condition to be alarmed occurs in a user macro program, system variable #3000 may be specified to put the machine in the alarm state.

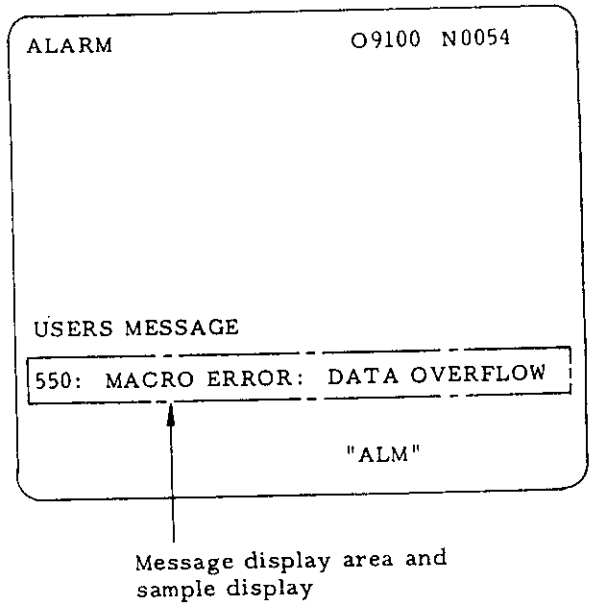
#3000 = n (<alarm meassage>);

Using this command, specify the alarm message (less than 32 characters) enclosed by 3-digit alarm number n and control-in and control-out. The alarm number should be three digits and be the one that is not used by the machine.

When this #3000 command is executed, "ALM" or "A/B" is displayed on the bottom of CRT screen regardless of the mode and function. Its message can be seen by the following operation:

a. Press ALM function key.

The alarm number and message are displayed on the bottom of CRT screen.



Message display area and sample display

b. When RESET key is pressed after removal of the cause of alarm, the message display and the alarm state can be cleared.

Sample Program

```
#3000 = 550 (MACRO ERROR: DATA OVER-
            FLOW)
```

E. Clock (#3001, #3002)

When system variable #3001 or #3002 for clock is specified, the clock can be read.

System Variable	Type	Unit	At Power-On	Count Condition
#3001	Clock 1	1 ms	Reset to "0"	Always
#3002	Clock 2	1 s	Same as power-off time	When STL signal is on

To preset the clock, cubstitute the value with this system variable put at the left-hand of the expression.

Sample Program

```
#3001 = 0 ; ... The clock is preset to value "0."
```

Restrictions

a. The accuracy of clock 1 is 8 ms. When 4294968000 msec has been reached, an overflow occurs, setting the clock to "0."

b. The accuracy of clock 2 is 8 ms. When 429496800 sec has been reached, an overflow occurs, setting the clock to "0."

Sample Program

Main Program

```
G65 P9351 A5 ;
```

Argument designation of variable #1

... Macro to wait for 5 seconds

Macro Program

```
O9351 ;
#3002 = 0 ;
WHILE [#3002 TE #1] DO1 ;
END 1 ;
M99 ;
```

F. Single Block Stop and Auxiliary Function Completion Wait Control (#3003)

When the value listed in the following table is substituted in system variable #3003, the single block switch can be disabled or the next block may be entered without waiting for the checking of the finish signal (FIN) of the auxiliary function (MST).

When the finish signal is not waited for, the distribution end signal (DEN) is not transmitted. In this case, the FIN is waited for in the block with the check skip cleared. Hence, when the FIN is not waited for, be careful not to specify the next auxiliary function.

#3003	Single Block Switch	FIN Signal
0	Valid	Waited
1	Invalid	Waited
2	Valid	Not waited
3	Invalid	Not waited

G. Feed-Hold, Feedrate-Override, And Exact -Stop Control (#3004)

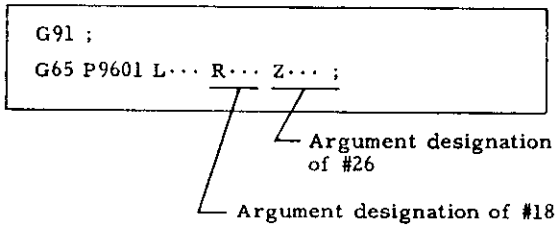
When the value listed in the following table is substituted in system variable #3004, feed hold, feedrate override, and exact stop can be made valid or invalid.

#3004	Feed Hold	Feedrate Override	Exact Stop
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

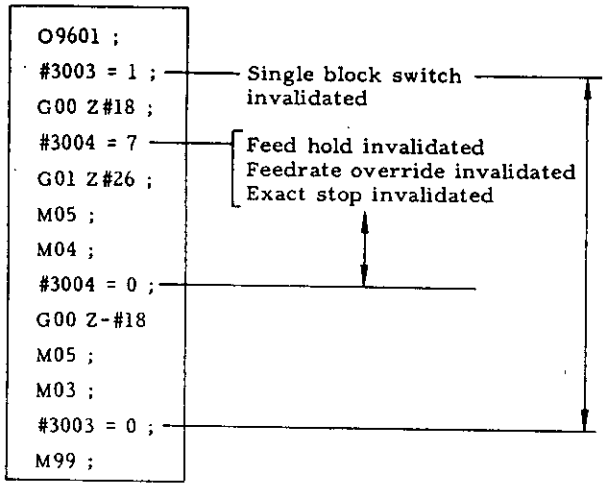
Sample Program

Tapping Cycle (for Incremental Command)

Main Program



Macro Program



H. RS232C Data Output (#3100)

When system variable #3100 is specified, messages and NC internal data can be outputted to external equipment via RS232C data input/output interface. If the external equipment is a printer, the above information is printed.

a. Output of Messages

#3100 = (<Message>)

When this command is specified, the message enclosed by control-in and control-out is outputted via RS232C interface.

Each output message is followed by CR/LF (Carriage Return/Line Feed). Hence, when #3100 = ( ) is specified, only CR/LF is outputted, which is useful in tabulating the punched data.

b. Output of Data

#3100 = < variable >

When this command is specified, the value of the local variable, common variable, or system variable at the right-hand is outputted vis RS232C interface as the signed decimal 8-digit data.

c. The above output is performed when system variable #3100 is executed in the macro program. It is required, therefore, to previously attach the external equipment such as a printer via RS232C interface and preset the parameters that use the interface.

Sample Program

#3100 = ( ); ··· Carriage return/line feed

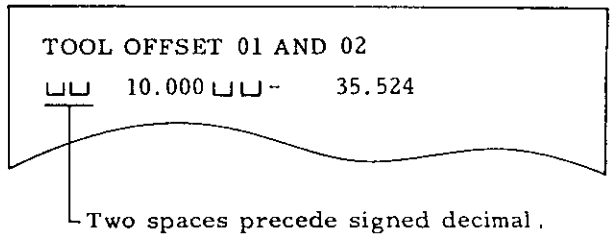
#3100 = (TOOL OFFSET 01 AND 02):

#3100 = #2501 ; ··· = 10.000 mm

#3100 = #2502 ; ··· = -35.524 mm

#3100 = ( );

Printout Data



I. Modal Information (#4001 Through #4120)

When one of system variables #4001 through #4120 is specified, the modal commands that are specified up to the immediately preceding block can be known. These modal commands are sometimes called the current values of modal information commands.

System Variable	Modal Information
#4001	G code (group 01)
⎩	⎩
#4021	G code (group 21)
#4102	B code
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Sequence number
#4115	Program number
#4119	S code
#4120	T code

#4001 through #4120 connot be placed to the left-hand of the operation expression.

Sample Program

Main Program

```
G65 P9602 <Argument Designation> ;
```

Macro Program

```
O9602 ;
#1 = #4001 ;          ← G codes (G00 through G03) of 01 group are retained.
G00 X··· Y··· ;
G01 Z··· F··· ;
G03 X··· Y··· R··· ;
G00 Z··· ;
G#1 ;                 ← G codes of 01 group are restored.
M99 ;
```

J. Positional Information (#5001 Through #5014)

When system variables #5001 through #5104 are specified, various positional information can be obtained.
The unit of the information is millimeters or inches.

	Unit
Metric input	0.001 millimeter
Inch input	0.0001 inch
Deg input†	0.001 degree

In the user macro body, the "input unit x 10" feature is invalid.

System Variable	Positional Information	Read During Move
#5001	X-axis block end position (ABSIO)	Enabled
#5002	Y-axis block end position (ABSIO)	
#5003	Z-axis block end position (ABSIO)	
#5004	4th block end position (ABSIO)	
#5021	X-axis current position (ABSMT)	Enabled(Note)
#5022	Y-axis current position (ABSMT)	
#5023	Z-axis current position (ABSMT)	
#5024	4th current position (ABSMT)	
#5041	X-axis current position (ABSOT)	Enabled(Note)
#5042	Y-axis current position (ABSOT)	
#5043	Z-axis current position (ABSOT)	
#5044	4th current position (ABSOT)	
#5061	X-axis skip signal position (ABSKP)	Enabled(Note)
#5062	Y-axis skip signal position (ABSKP)	
#5063	Z-axis skip signal position (ABSKP)	
#5064	4th skip signal position (ABSKP)	
#5083	Tool length correction amount	Enabled
#5101	X-axis servo position deflection amount	Enabled(Note)
#5102	Y-axis servo position deflection amount	
#5103	Z-axis serve position deflection amount	
#5104	4th servo position deflection amount	

Note: Reading of #5021 to #5024, #5041 to #5044, and #5101 to #5104, when commanded during movement, will be performed after completion of the movement.

Mnemonic	ABSIO	ABSMT	ABSOT	ABSKP
Meaning	End position of block immediately before.	Command current position (same as POS. MACHINE display).	Command current position (same as POS. UNIVERSAL display).	Position at which skip signal did not go on in G31 block.
Coordinate system	Work coordinate system	Machine coordinate system	Work coordinate system	Work coordinate system
Tool position, Tool length, Tool radius compensation Amount	Not included		Included	Included

Notes:

1. When the skip signal is not turned on in G31 block, the skip signal position is at the end of G31 block.

2. The "input unit x 10" feature is valid up to the macro call block (the argument designation part by G65 or G66) but is invalid in the user macro body.

3. System variables #5001 through #5104 may not be placed to the left-hand of operational expression.

Sample Program

The tool is positioned to the specified location (X, Y, Z) on machine coordinate system, performs the specified M feature, and returns to the start point.

Main Program

```
G65 P9603 X··· Y··· Z··· M··· ;
```

#13
#26
#25
#24

Macro Program

```
O9603 ;
#1 = #5001 ;
#2 = #5002 ;
#3 = #5003 ;
G91 ;
G00 X [#24-#5021] Y [#25-#5022];
G00 Z [#26-#5023] ;
M#13 ;
G00 Z#3 ;
G00 X#1 Y#2 ;
M99 ;
```

List of Variables

Variable No.	Meaning
#1 to #33	Local variables.
#100 to #149	Common variables (reset to"blank"at power-off).
#500 to #509	Common variables (retained at power-off).
#1000 to #1015	Interface input signals (each signal for each bit).
#1032	Interface input signal $(\sum_{i=0}^{15} \#[1000 + i] * 2i)$.
#1100 to #1115	Interface output signals (each signal for each bit).
#1132	Interface output signal $(\sum_{i=0}^{15} \#[1100 + i] * 2i)$.
#2001 to #2099	Tool offset amount.
#2500 to #2806	Work coordinate system shift amount, external work coordinate system correction amount.
#3000	Alarm message display.
#3001	Clock 1 (in units of 1 ms).
#3002	Clock 2 (in units of 1 s).
#3003	Single block stop, auxiliary function complete wait control.
#3004	Feed-hold, feedrate-override, and exact-stop control.
#3100	RS232C data output (print out feature).
#4001 to #4120	Current value of modal information command.

Variable No.	Meaning
#5001 to #5004	End position of immediately preceding block (for each axis).
#5021 to #5024	Current position of machine coordinate system (for each axis).
#5041 to #5044	Current position of POS. UNIVERSAL (for each axis).
#5061 to #5064	Position at which G31 skip signal is turned on (for each axis).
#5083	Valid tool length compensation amount.
#5101 to #5104	Servo position deflection amount (for each axis)

K. Variable Reprentation

Each variable is represented in a variable number that follows #.

a. How to designate a number directly:

#i (i = 1, 2, 3, 4, ···)

Sample
#10
#130
#2000

b. How to designate an expression as a variable number:

#[<expression>]

Sample
#[#100]
#[#500 + 1]
#[#20/2]

In the following description, variable #i may be replaced with variable #[< expression >].

L. Variable Reference

a. The value that follows an address may be replaced by a variable.

When < address> #i or <address> -#i is specified, the value of the variable or its negative value (complement, more exactly) is made the specified value of the address.

Sample
```
#30 = 1. 0 ;
#101 = 100. ;
#102 = 200. ;
#103 = 300. ;
#140 = 250. ;
G#30 X#101 Y#102 Z-#103 F#140 ;
```

The above specification is equivalent to the specification below:

```
G01 X100 X200 Z-300 F-250 ;
```

Notes:

(1) Address /, O, and N may not refer to variables.

Sample /#8, N#100 ··· Error.

(2) A variable number may not be replaced with a variable.

Sample
##20 ··· Error.
[#20] ··· Correct.

(3) When a variable is used as address data, the values below the least significant digit are rounded.

Sample

(i) When #1 = 45.2346
X#1 ··· = X45.235 mm (for mm input)

(ii) When #2 = 350.85
F#2 ··· F351 (mm/min)

(iii) When #3 = 5.37672
G04 P#3 ··· = G04 P5.377 (sec)

(iv) When #4 = 2.7236
M#4 ··· M03
G#4 ··· G03

(4) Value for each address should not exceed the maximum programmable value.

(5) The value that follows an address may be replaced with <expression>.

(6) The constant without decimal point enclosed in brackets[ ]is assumed to have a decimal point at its end.

M. Undefined Variable

The value of an undefined variable is assumed to be "blank." An undefined variable occurs in the following situations:

a. The local variable for which argument designation was not performed in macro call command.

b. Common variables #100 through #149 at the time of power-on and reset operations.

c. The local variables and common variables for which the values were not written from MDI panel.

Designation and function of <blank> is classified in the following two versions A and B. The control is set for either version. Versions A and B cannot be changed.

		Version A	Version B
1	Concept of "#0"	. No conception of #0. . Commanding #0 causes alarm.	. #0 defined as variables of <blank> . Commanding #0 at the left-hand side of the equation.
2	Variable <blank> is commanded in the replacement equation.	. Where #2 is <blank> , command #3=#2; means #3=0.	. Where #2 is <blank> command #3=#2; means #3= <blank>.
3	Variable <blank> is commanded in the part program.	. Where #2 is <blank>, command 600 x #2; is equivalent to command 600 x 0;	. Where #2 is <blank> command 600 x #2; is equivalent to 600; (Adress is Ignored.)
4	Variable <blank> is commanded in the condition of EQ and NE.	. Where #2 is <blank>, #3 is 0 ① Condition "IF #3 EQ #2" is established ② Condition "1F #3 NE #2" is not established.	. Where #2 is <blank> #3 is 0 ① Condition "1F #3 EQ #2" is not established. ② Condition "IF #3 EQ #2 is established.
5	Others	#3=#[#0+#0] #3=#2 * #0; #3=#0 + #0; #3=#0/#0; #3=5*#0; In these commands, #3=0, #3=2-#0; means #3=2 #3=5/#0; causes alarm. Blank in the replacement described above is treated as "0." . Condition IF#3 GE#2 is established when #2 and #3 are <blank> , or #2 is 0 and #3 is <blank> . . Condition IF #3LT #2 is not established when #2 and #3 are <blank> , or #2 is <blank>, and #3=0.	

2.11.5 OPERATION COMMANDS

Various operations can be performed between variables and between variables and constants. The operation expression is represented in the form of #i = ⟨expression⟩, in which ⟨expression⟩ is a general arithmetic operational expression produced by combining variables and constants with operators and functions. The available opearations and functions are as follows. Instead of #j and #k, constants may be used.

(1) Variable Definition and Replacement

#i = #j ··· definition, replacement.
#i = #[#j + #k] ...

(2) Add-Type Operations

#i = #j + #k ··· Sum.
#i = #j - #k ··· Difference.
#i = #j OR #k ··· Logical sum (for each of 32 bits).
#i = #j XOR #k ··· Exclusive logical sum (for each of 32 bits).

(3) Multiply-Type Operations

#i = #j * #k ··· Product.
#i = #j / #k ··· Quotient.
#i = #j AND #k ··· Logical product (for each of 32 bits).

Note: In OR, XOR, or AND operation, the variable value (or constant) is converted into the binary 32-bit equivalent and the operation is performed on each bit.

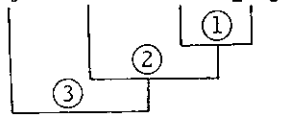
(4) Functions

#i = SIN [#j] ··· Sine (in degrees).
#i = COS [#j] ··· Cosine (in degrees).
#i = TAN [#j] ··· Tangent (in degrees).
#i = ATAN [#j/#k] ··· Arctangent (in degrees).
#i = SQRT [#j] ··· Square root.
#i = ABS [#j] ··· Absolute value.
#i = BIN [#j] ··· Convert from BCD.
#i = BCD [#j] ··· Convert into BCD.
#i = ROUND [#j] ··· Produce integer by rounding.
#i = FIX [#j] ··· Truncate the fractions.
#i = FUP [#j] ··· Raise the fractions to a unit.

(5) Combinations of Operations

The above operations and functions may be used in combinations. A functional operation is performed first. Then, a multiply-type operation is performed. An add-type operation is performed last.

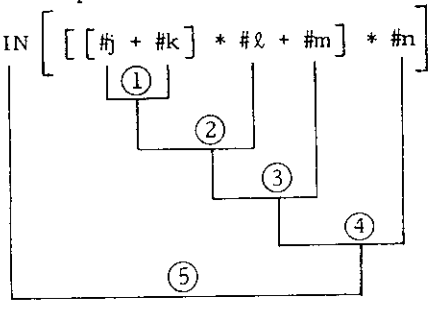
Sample #i = #j + #k * SIN [#ℓ]



(6) Change of Operational Order by [ ]

Priority may be given to an operation by enclosing it in brackets [ ]. Up to quintuple (five-hold) nesting of brackets is permitted including those of functional operations.

Sample #i = SIN [[[#j + #k] * #ℓ + #m] * #n]



(7) Considerations for Operational Commands

A. The constant without decimal point used in ⟨expression⟩ is assumed to have a decimal point at its end.

B. When used in conditional expression IF or WHILE, function ROUND truncates the fractions.

C. When used in address data, function ROUND rounds the part below the least significant digit.

Sample (a)

#10 = 12.3758

When the least significant digit of address X is 0.001 mm, the following command

G00 X [ROUND [#10]];
means

G00 X12.376 ;
because 8 of 12.3758 is rounded.

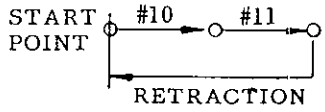
This command is also equivalent to

G00 X#10 ;

Usually, ROUND is not used as mentioned above; it is used as shown below:

Sample (b)

When ROUND is used as follows, the program returns to the start point correctly:



① #10 = 12.3758 ;
② #11 = 13.1236 ;
③ G91 G00 X#10 ;
④ G01 X#11 F··· ;
⑤ G00 X- [ROUND [#10] + ROUND [#11]] ;

This is because the data of #10 and #11 in ③ and ④ blocks are substantially rounded before being executed.

If ⑤ block is

⑤' G00 X- [#10 + #11] ;

then, the movement is made by the following amount:

X- [#10 + #11] = X - [12.3758 + 13.1236]
= X - [25.4994]
= X - [25.499]

On the other hand, block movement of ③+④is

X#10 + X#11 = X12.376 + X13.124
= X25.500

Hence, the program of ⑤' is not correct.

(8) Operational Errors

The data format and the operational errors in the user macros are as follows:

A. Data Format

The numeric data handled in user macros are of the floating point format.

M * 2E

where, M is sign + data 52-bit binary,
E is sign + data 10-bit binary.

B. Operational Errors

Each time an operation is performed, the following error is caused and is accumulated. The number of significant digits is 15 to 16, which compensates the error sufficiently.

2.11.6 CONTROL COMMANDS

The commands which control the flow of micro-program are of the following two types:

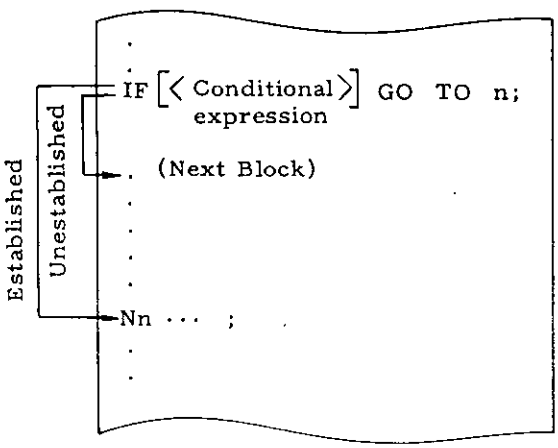
A. Branch Command ... IF [<conditional expression>] GO TO n ;

B. Repeat Command ... WHITE [<conditional expression>] DO m ;

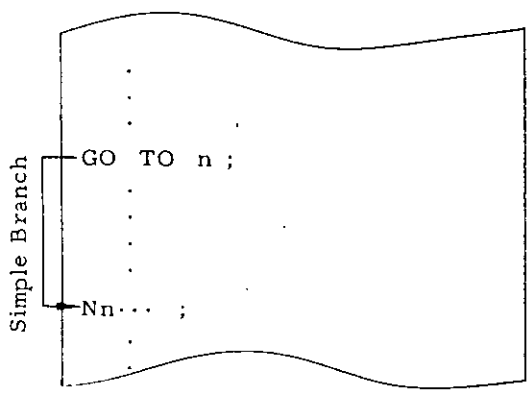
(1) Branch Command

A. IF [< conditional expression>] GO TO n ;

If < conditional expression> of this command is established, a branch is made to the block of sequence number n within the same program. When a variable or an expression is used for n, the branch destination may be changed. If the condition is not satisfied, the program proceeds to the next block.



IF [<conditional expression>] may be omitted to provide a simple branch command as shown below:



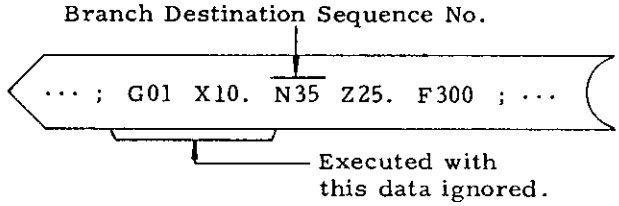
Conditional expressions are EQ, NE, GT, LT, GE, and LE. They are represented as follows:

Conditional Expression	Meaning
#i EQ #j	(#i = #j)
#i NE #j	(#i ≠ #j)
#i GT #j	(#i > #j)
#i LT #j	(#i < #j)
#i GE #j	(#i ≥ #j)
#i LE #j	(#i ≤ #j)

A constant and <expression> may be used to #i and #j. A variable and <expression> may be used for n.

Notes:

1. The sequence number of the destination of the branch by a branch command must be located at the head of that block. Otherwise, the data prior to the sequence number is ignored as shown below:

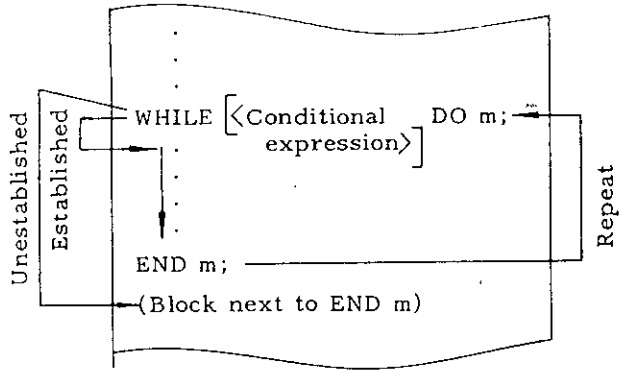


2. The reverse branch on the program takes longer execution time than the forward branch.

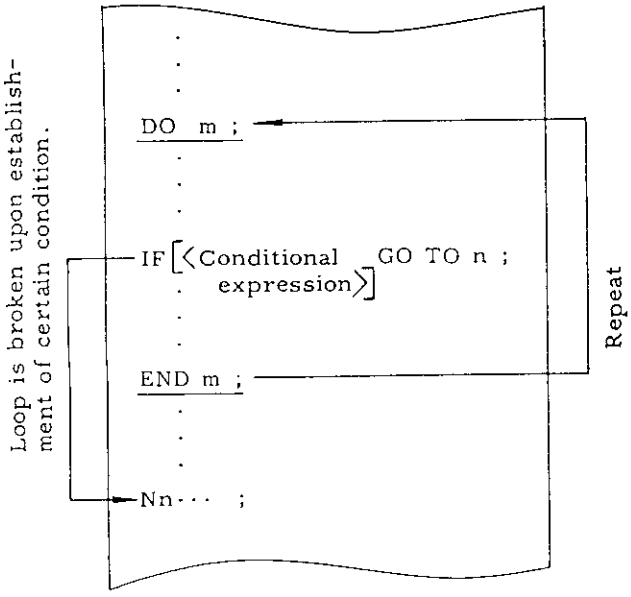
(2) Repeat Command

A. WHILE [<conditional expression>] DO m;
(m = 1, 2 and 3)
.
.
.
END m ;

While <conditional expression> is satisfied, the blocks between DO m and END m are repeated. When it is unsatisfied, the processing branches to the block next to END m.



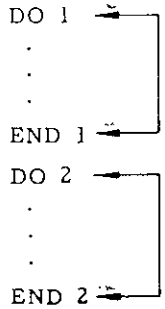
When the specification is made omitting WHILE [<conditional expression>], the blocks between DO m and END m are repeated infinitely. Generally, this is used in the format shown below.



Notes:

1. DO m should be specified before END m.

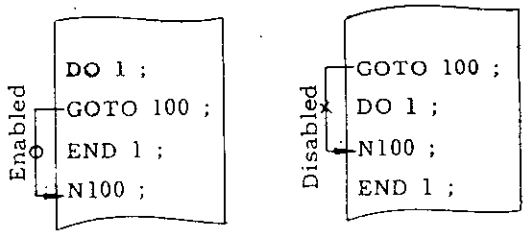
2. m of DO m and END m should have the same value. However, only 1, 2, or 3 may be specified in m.



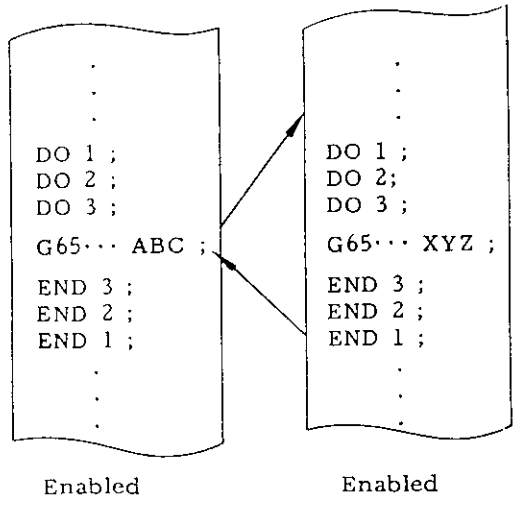
The value of m, 1, 2, or 3 is called the identification number, which is used by DO and END to call each other.

3. The same identification number may be used repeatedly except where repeat ranges overlap.

4. GO TO n enables to get out of DO loop but it does not enable to get into DO loop as shown below:



5. Triple DO-loop nesting is permitted for each micro program.



6. The codings shown below cause an error:

(i)
```
DO 1 ;  ?
  .
  .
DO 1;   ?
  .
  .
END 1 ;
```
The return destination of END 1 is unknown.

(ii)
```
DO 1 ;
  .
  .
END 1 ;
  .
  .
END 1 ;  ?
```
One END 1 is missing.

(iii)
```
DO 1 ;  ?
  .
  .
  .
DO 2 ;  ?
  .
  .
  .
END 1 ;
  .
  .
  .
END 2 ;
```
Overlap of DO ranges is not permitted. Rewrite as shown right.
```
DO 1 ;
DO 2 ;
END 2 ;
END 1 ;
```

(iv)
```
DO 1 ;
  .
  .
DO 2 ;
  .
  .
DO 3 ;
  .
  .
DO 1 ;
  .
  .
  .
END 1 ;
  .
  .
END 3 ;
  .
  .
END 2 ;
  .
  .
END 1 ;
```
Quadruple nesting. Max. nexting permitted is triple.

(v)
```
DO 1 ;
  .
  .
  .
N7000··· ;
  .
  .
  .
END 1 ;
  .
  .
  .
IF···GO TO 7000 ;
```
DO loop may not be entered from outside.

2.11.7 REGISTRATION OF USER MACROS

(1) How To Make Registration of User Macros

The registration and edit of user macro bodies (macro programs) are performed in the same manner as usual part programs and subprograms. Hence, there is no program size restriction that applies to the user macro body only. Part programs, subprograms, and macro programs may be stored together in the part program memory to its full capacity.

(2) Classification of Program Numbers

The program numbers are classified into the following:

Program No.	Classification
O1 to O7999	These programs may be registered, erased, or edited without restrictions.
O8000 to O8999	When D4 of #6004 is set to 1, the registration, erase, and edit of programs are disabled.
O9000 to O9999	When D7 of #6021 is set to 1, the registration, erase, and edit of programs are disabled.

2.11.8 DISPLAY AND WRITE OF LOCAL VARIABLES AND COMMON VARIABLE

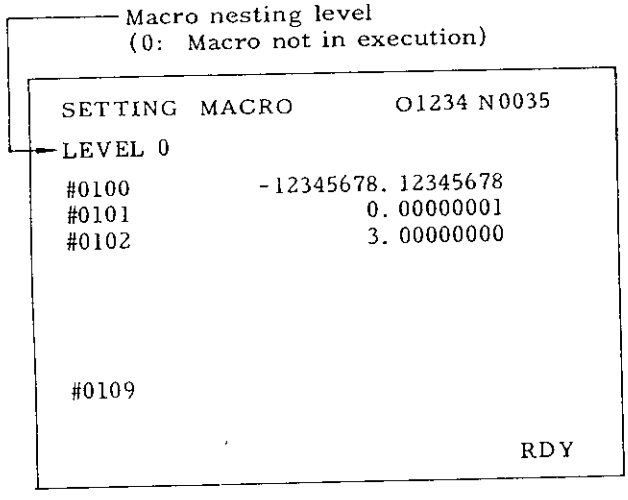
Local variables (#1 through #33) and common variables (#100 through #149, #500 through #509) can be displayed and written by the following operations:

(1) Display Operations

A. Display of Variables

a. Press SET function key. Mode select position may be provided anywhere.

b. Key-in the variable number and press [CUSOR ↓] key or [↑ CURSOR] key. However, # need not be keyed-in. Ten sets of variable numbers including the specified variable number and their data are displayed. The data are displayed in the signed 8-digit integer part and the 8-digit fraction part.



Sample Display of Common Variables

c. Press [PAGE ↓] key or [↑ PAGE] key, and the display may be scrolled up or down.

B. Remarks

a. Common variables may always be displayed for review.

b. For local variables, those of the macro currently executed are displayed. Consequently, when a macro of a nesting level is in execution, the local variables of the macros of the other nesting levels cannot be seen. The local variables after completion of execution are all rest to "blank."

(2) Write Operations

A. Write of Values to Variables

a. Press SET function key. Mode select position may be provided anywhere.

b. Key-in the variable number to the written. Press [CURSOR ↓] key or [↑ CURSOR] key. However, # need not be keyed-in. The keyed-in variable number is specified and the cursor is positioned to it.

c. Key-in the value to be written. Press WR key. The keyed-in value is stored as the data of the variable number with the cursor positioned.

d. Press [CURSOR ↓] key or [↑ CURSOR] key or [↑ PAGE] key or [PAGE ↓] key to move the cursor.

e. Repeat operations in c. and d. to write the values to the desired variables.

B. Remarks

a. Common variables can always be changed.

b. Local variables may not be written at any time other than when a macro is in execution. Any attempt to do so is invalidated. However, rewriting of local variables during macro execution may cause an unexpected failure. Before attempting the rewriting, stop the machine operation by single stop function and check to see if it is safe to rewrite variables.

c. The written local variables and common variables #100 through #149 are reset to "blank" by the reset operation or the power-on operation.

2.11.9 CONSIDERATIONS AND REMARKS FOR USER MACROS

(1) Summary of Restrictions

A. Available Variables

#1 through #33 ··· Local variables.
#100 through #149 ┐
#500 through #509 ┘ Common variables.
System variables

B. Available Variable Values

Maximum value ... $\pm 10^{+308}$

Minimum value ... $\pm 10^{-308}$

C. Constant Values Usable in ⟨Expression⟩

±(8 digits above decimal point). (7 digits below decimal points).

Sample Maximum value ±99999999.9999999

Minimum value ±0.0000001

D. Operational Accuracy

Decimal 15 digits significant.

E. Macro Call Maximum Nesting Level

Quadruple (four-hold).

F. Maximum Nesting Level of Repeat Command

Triple (thread-hold) for each macro.

G. Repeat Command (DO) Identifier m

m = 1, 2, and 3.

H. Maximum Nesting Level of Brackets

Quintuple (five-hold).

(2) Differences Between User Macro and Subprogram

A. User macros G65 and G66 allow argument designation but the subprogram (M98) does not.

B. The user macro directly branches to the user macro body without executing any command that was specified in G65 or G66 block and has no relationship with the macro. With the subprogram, however, a branch is performed after the execution of the command (if any) other than P and L in M98 block.

C. The maximum nesting level of user macro is quadruple including G65 and G66 calls. That of subprograms is also quadruple but separately.

D. If user macros are specified via the MDI during automatic operation, the maximum nesting level is restricted to quadruple. With subprograms, up to four levels of nesting are permitted in tape mode or memory mode, or separately in MDI mode.

(3) Relationship with MDI Operation

A. MDI writing permits the macro call and the execution of the called macro.

B. MDI writing does not permit or execute macro body commands such as operational commands and control commands.

C. When a macro program being executed is stopped by the single block stop function, any MDI writing command not related to the macro may be specified and executed.

(4) Relationship with Address Search

The address search function is not permitted to search for the sequence numbers in the user macro body.

(5) Relationship with Single Block Switch

A. The operational command and control command blocks do not single-block stop if the single block switch is turned on. This switch is enabled for the other macro program blocks.

B. However, when setting number #6004D1 = 1, the single block switch is enabled for the operational command and control command.

C. System variable #3003 (for the control of single block stop, see 1.5.3.6) and setting #6004 D1 mentioned above operate as shown below:

Setting #6004	System Variable #3003	When Single Bock Switch is on
D1 = 0	= 1 or 3	None of operational command, control command, and general command stop.
D1 = 0	= 0 or 2	Operational command and control command do not stop. General command stops.
D1 = 1	= 1 or 3	None of operational command, control command, and general command stop.
D1 = 1	= 0 or 2	All of operational command, control command, and general command stop.

(6) Relationship with Operational Block Skip

The slash "/" character used in the right-hand of an operational expression or in brackets is assumed to be the operator for quotient. It does not mean the optional skip.

(7) Setting and Parameter of Program Number Classification

A. Disabling of Program Registration, Erase, And Edit

The following setting is permitted to protect the registered user macros and subprograms from inadvertent destruction:

Setting Number

#6004

D4 = 1 ··· The programs of program numbers #8000 through #8999 are disabled for registration, erase, and edit.

D4 = 0 ··· Registration, erase, and edit are enabled.

Parameter Number

#6021

D7 = 1 ··· The programs of program numbers #9000 through #9999 are disabled for registration, erase, and edit.

D7 = 0 ··· Registration, erase, and edit are enabled.

(8) Effects of Reset Operation

A. A reset operation resets all local variables (#1 through #33) and part of common variables (#100 through #149) to "blank."

B. A reset operation resets the user-macro multiple call state and the multiple DO loop state, making the program pointer return to the program head.

(9) Special Codes Usable in User Macro Body

A. The special codes listed below may be used in the user macro body:

	Code	Use	EIA Code 8	7	6	5	4	∘	3	2	1	ISO Code 8	7	6	5	4	∘	3	2	1
	SP	For comment				○		∘				○		○			∘			
*	(	For alarm message comment				○	○	∘		○				○		○	∘			
*	)			○			○	∘		○		○		○		○	∘			○
	+	Add		○	○	○		∘						○		○	∘		○	○
	-	Subtract		○				∘						○		○	∘	○		○
	:	For comment		○				∘	○	○				○	○	○	∘		○	
	/	Divide			○	○		∘			○			○		○	•	○	○	○
	#	Variable	Parameter designation											○			∘		○	○
*	*	Multiply	○			○	○	∘				○		○		○	∘		○	
*	=	Equal	○				○	∘	○			○		○	○	○	∘	○		○
*	[	Bracket(open)	○		○	○		∘				○	○		○	○	∘		○	○
*	]	Bracket (close)	○		○			∘		○		○	○		○	○	•	○		○
	$	For comment	○			○		∘	○					○			•	○		
	@		○				○	∘	○	○	○	○	○				∘			
	?		○			○	○	∘	○	○				○	○	○	∘	○	○	○
	.	Decimal point		○	○		○	∘		○	○			○		○	•	○	○	

Notes:

1. For the hole pattern of EIA code of the character attached with an asterisk, the pattern shown above is standard. However, other patterns may be specified by using the following parameters:

#6110 ··· [
#6111 ···]
#6112 ··· *
#6113 ··· =
#6114 ··· (
#6115 ···)

Read the desired hole pattern in the binary value, convert it into the decimal equivalent, and set it to the parameter. For example, the hole pattern shown below is set as "152":

8	7	6	5	4	0	3	2	1
○			○	○	∘			

When the value of the parameter is "0," the hole pattern listed in the above table is provided.

2. When the codes shown below are output from the NC unit for punch-out or other purposes, the upper code (UC) or lower code (LC) is output immediately before.

a. Codes preceded by UC ··· #, +, $, ?.

b. Code preceded by LC ··· @.

c. Codes preceded by UC only at parameter designation ... (,), *, =.

2.11.10 ALARM NUMBER OF USER MACROS

Shown below are the user-macro-associated alarms and their causes.

105 MACRO ERROR (CONSTANT)

The number of constants is in excess of the specified range.

106 MACRO ERROR

There are too many G67 cancel codes.

107 MACRO ERROR (FORMAT)

A format other than expression has an error.

108 MACRO ERROR (UNDEFIN #NO)

The value not defined as a variable number is designated.

109 MACRO ERROR (#NO NOT LEFT)

The variable of assignment statement is the one that is disabled for assignment.

110 MACRO ERROR ([ ] 5 LIMIT)

The bracket nesting level is in excess of the upper limit (5).

111 MACRO ERROR (MOVE G66 - M99)

A move command is specified in the macro end command M99 called by G66.

112 MACRO ERROR (5)

The macro call nesting level is in excess of the upper limit (4).

113 ——

114 MACRO ERROR (DO FORMAT)

DO and END are not paired.

115 MACRO ERROR ([ ] UNMATCH)

The format of 〈expression〉 has an error.

116 MACRO ERROR (DO - END NO.)

DO m is not in the range of $1 \leq m \leq 3$.

117 ——

118 MACRO ERROR (GO TO N)

GO TO n is not in the range of $0 \leq n \leq 9999$.

2.11.11 EXERCISES OF USER MACRO

(1) Canned-Cycle G82

T (Teacher): We have discussed many complicated rules you have to understand to write user macros. Now, let's create some user macros as exercises. Let's take G82 Spot Facing Cycle of canned cycles for example, because it is a simple operation.

S (Student): Where shall we start?

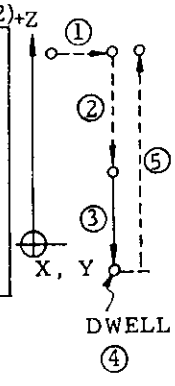
T: An example of usual G82 command takes the following format:

(P1)

```
G91 ; (··· Incremental Designation)
G82 X100. Y50. R-80. Z-40. P3.0 F250 ;
```

This command is divided into the following and executed within the NC unit:

```
    G91 ;
① G00 X100. Y50 ;
② G00 X-80. ;
③ G01 Z-40 F250 ;
④G04 P3. ; (Dwell)
⑤G00 Z-((-80) + (-40)) ;
```



2.11.11 EXERCISES OF USER MACRO (CONT'D)

First, these moving distances may all be converted into variables.

S: They are local variables #1 through #33, aren't they? But which type of local variable?

T: Type 1 for small number of variables. This type allows the use of X, Y, and Z and therefore makes the argument designation easier to understand.

S: OK. When type I is used, we have the following variables:

```
X100.  Y50.  R-80.  Z-40.  P3.0  F250
  ↓     ↓     ↓      ↓      ↓     ↓
 #24   #25   #18    #26     ?    #9
```

(P3)

Address P of dwell time cannot be used for argument designation, can it?

T: No. Use some other address. Then, write the address U instead of P.

```
P3.0  →  U3.0
           ↓
          #21
```

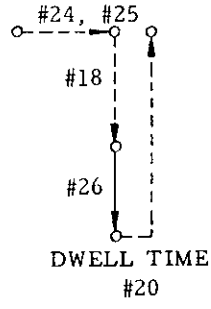
(P3')

Using these variables, rewrite the former program (P2).

S: OK.

(P4)

```
   G91 ;
①  G00 X#24 Y#25 ;
②  G00 Z#18 ;
③  G01 Z#26 F#9 ;
④  G04 P#21 ;
⑤  G00 Z-[#18 + #26] ;
```



Is this all right?

T: Sorry, it isn't right. You have forgotten to specify something in ⑤, havn't you?

S: Oh, I should have specified "ROUND."

(P5)

```
⑤  GOO Z-[ROUND [#18] + ROUND [#26]];
```

Is this good?

T: Yes. Write as follows after the above, and we have a complete user macro body.

```
⑥  M99 ;
```

S: That's easy.

T: Then, using G65, create this macro call and the user macro body in the complete formats.

S: Let me try it.

The macro call command is as follows:

(P6)

```
G91 ;

G65 P9082 X100. Y50. R-80. Z-40.
          U3.0 F250. ;
```

The user macro body is as follows:

(P7)

```
O9082 ;
G00 X#24 Y#25 ;
G00 Z#18 ;
G01 Z#26 F#9 ;
G04 P#21 ;
G00 Z-[ROUND [#18] + ROUND [#26]] ;
M99 ;
```

Program number "O9082" of the user macro is arbitrary.

T: That looks OK.

****

S: I think something is wrong. With this program, I have to specify points R and Z every time!

T: That's true. With a usual canned cycle, when points R and Z have been specified once, their values are retained.

S: Do you have any trick to overcome this in covenience?

T: I do. In such a case, common variables help. Using common variables, write the macro to designate the position of points R and Z. U and F may also be used for the same purpose.

S: I've got it! Now, I divide the macro body into two parts as follows:

(P8)

```
O9000 ;
#100 = #18 ;
#101 = #26 ;
#102 = #21 ;
#103 = #9 ;
M99 ;
```

(P9)

```
O9082 ;
G00 X#24 Y#25 ;
G00 Z#100 ;
G01 Z#101 F103 ;
G04 P#102
G01 Z-[ROUND [#100] + ROUND [#101]] ;
M99 ;
```

and I write macro call as follows:

(P10)

```
G91 ;
G65 P9000 R-80. Z-40. U3.0 F250. ;
G65 P9082 X100. Y50. ;
G65 P9082 X··· Y··· ;
  .
  .
  .
```

T: Very good.

****

S: Wait a minute! This canned cycle always returns to the initial point.

T: I also overlooked it. A program would be useless if it is disabled for the designation of the initial point return (G98) and point-R return (G99). To solve this problem, use the system variable called "current value of modal information command" to know which state, G98 or G99, is provided, and change the specification of the tool return destination.

S: G98 and G99 belong to group "10." So, I have to use system variable #4010. Is it right?

T: Yes. Then, using "IF ··· GO TO ···" command, change the specification of the tool return destination.
There is one more point to be improved. Write the program so that the group "01" G codes before execution may be retained after the execution of this macro. Group "01" G codes include G00 through G03.

S: It looks difficult ···
What would you say to the following program?

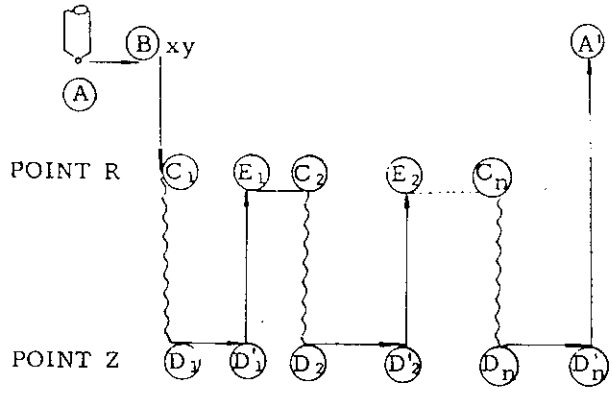
T: Well done!

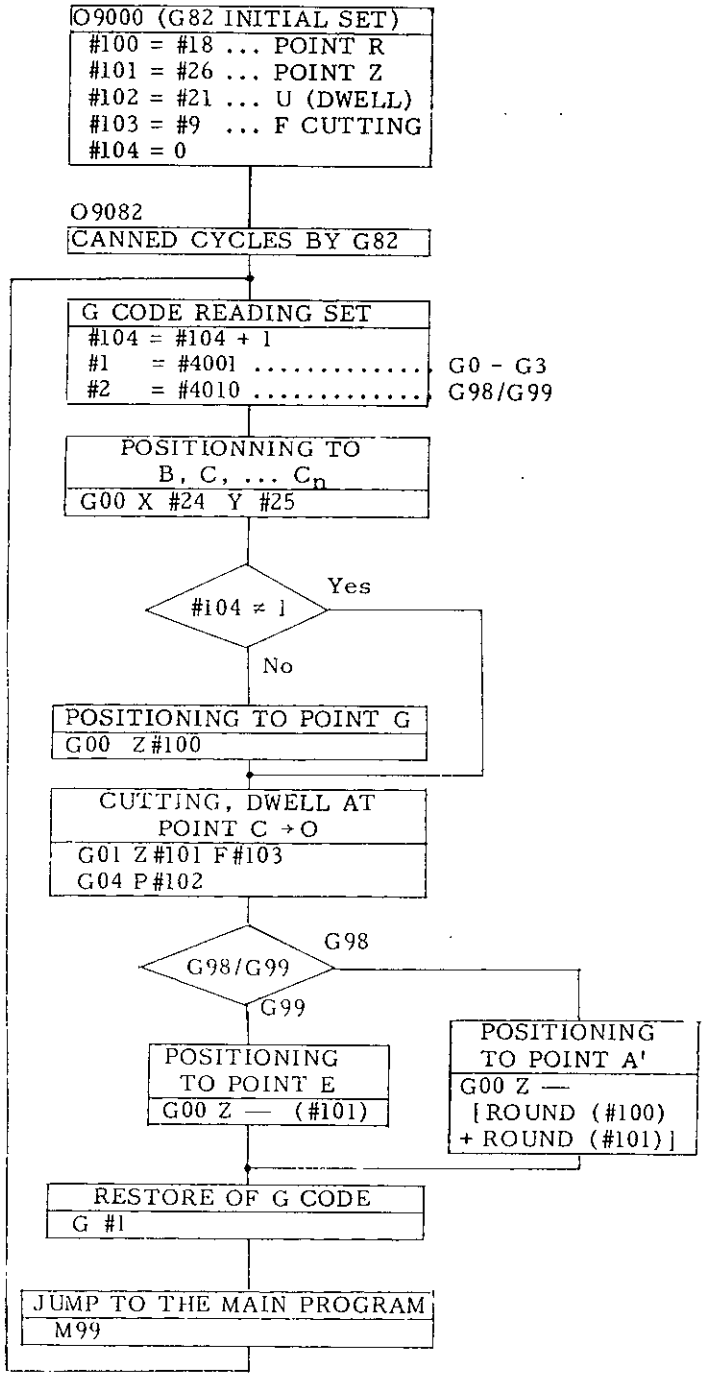
(P11)

```
Macro Call Program
G91 G99;
G65 P9000 R-80. Z-40. U3.0 F250;
G65 P9082 X100. Y50.;
G65 P9082 X...Y...;
  .
  .
  .
G98;
G65 P9082 X...Y...;
```

User Macro Body (P12)

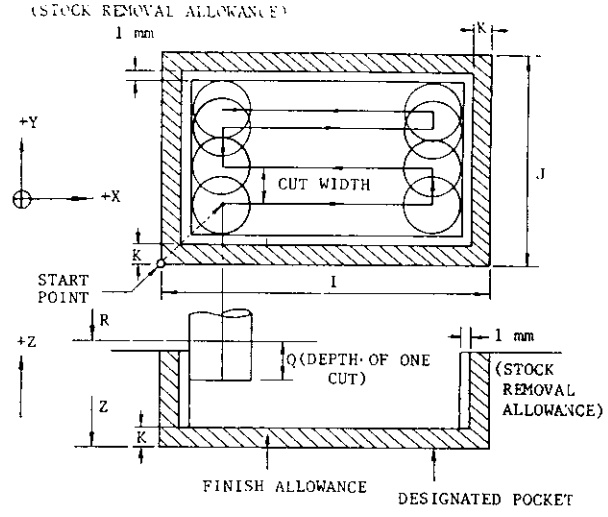
```
O9000;
#100 = #18;
#101 = #26;
#102 = #21;
#103 = #9;
#104 = 0
-----------------------------------------
O9082;
#104 = #104 + 1
#1 = #4001; ... G0 to G3
#2 = #4010; ... G98/G99
G00 X#24 Y#25;
IF[ #104 NE1] GO TO 1;
IF[ #2 EQ99] GO TO 2;
G00 Z#100;
N1 G01 Z#101 F#103;
G04 P#102;
IF[ #2 EQ 98] GO TO 2;
G00 7-[ #101]
GO TO 3;
N2 G00 Z-[ROUND [ #100] + ROUND [ #101]];
N3 G#1; ... Restore G Code
M99;
```





(2) Pocket Mill

A. The cycle for the pocket machining shown below is created by user macros as follow:



B. Macro Call Command

G65 P9061 X··· Y··· Z··· R··· I··· J··· K···
T··· Q··· D··· F··· E··· ;

where,

X, Y: The absolute coordinate values of the start point (the lower left position of the pocket).

Z: The absolute position of the bottom of the pocket.

R: The absolute position of rapid traverse tool return.

I, J: X-axis and Y-axis lengths of the pocket (unsigned).

K: Finish allowance (left-over allowance, unsigned). Default value is 0.

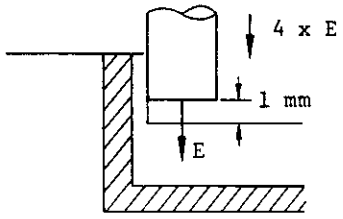
T: Cut width rate (designated in %).
Cut width = tool radius x T/100

Q: Z-axis cut depth for each time (unsigned).

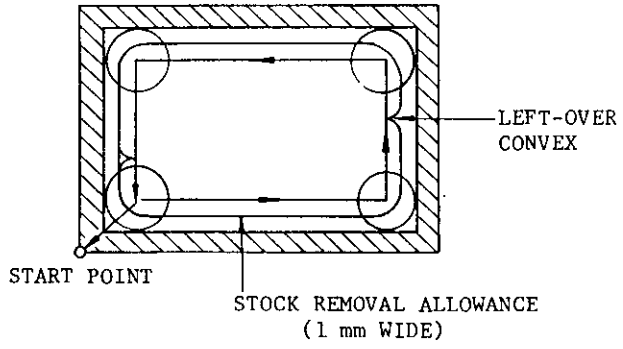
D: Tool offset number.

F: Feedrate on XY plane.

E: Feedrate at Z-axis cut.
(Tool is fed 4 times as fast as E up to the point 1 mm to the preceding cut bottom.)



The stock removal allowance (1 mm) inside the finish allowance is all cut by a single operation in the final process as shown below. Then, the tool returns to the start point, completing the cycle.



C. User Macro Body

```
O9061 ;
#10 = # [2000 + #7]; ··· Tool radius
#11 = #6 + 1.0 + #10 ;
#12 = #5 - 2 * #11 ;
#13 = 2 * #10 * #20/100 ; ··· Cut width
#14 = FUP [#12/#13] ; ··· X-axis cut count: -1
#27 = #24 + #11 ; ···   } X, Y coordinates of
#28 = #25 + #11 ; ···   } machining start point
#29 = #26 + #6 ; ··· Z-axis coordinates of
                     cut bottom
#30 = #24 + #4 - #11 ;
#15 = #4003 ; ··· Read of G90/G91
G90 ; ··· Absolute command
G00 X#27 Y#28 ;
G00 Z#18 ;
#32 = #18 ; ··· #32: Cut bottom in execution
DO 1 ;
#32 = #32 - #17 ;
IF [#32 GT#29] GO TO 1 ;
#32 = #29 ;
N1 G01 Z#32 F#8 ;
G01 X#30 F#9 ;
#33 = 1 ;
WHILE [#33 LE#14] DO 2 ;
IF [#33 EQ#14] GO TO 2 ;
G01 Y [#28 + #33 * #13] F#9 ;
GO TO 3 ;
N2 G01 Y [#25 + #5 - #11] ;
N3 IF [#33 AND 1 EQ 0] GOTO 4 ;
G01 X#27 ;
GO TO 5 ;
N4 G01 X#30 ;
N5 #33 = #33 + 1 ;
END 2 ;
G00 Z#18 ;
IF [#32 LE#29] GO TO 6 ;
G00 X#27 Y#28 ;
G01 Z [#32 + 1.0] F [4 * #8] ;
END 1 ;
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 ;
G01 Z#32 F#8 ;
G01 X#30 F#9 ;
    Y#31 ;
    X#27 ;
    Y#28 ;
G00 Z#18 ;
G00 X#24 Y#25 ; ··· Return to start point
G#15 ; ··· Restore of G90/G91
M99 ;
```

α cut loop

Stock removal cycle

3. PART PROGRAM TAPE CODING

3.1 TAPE CODE

3.1.1 TAPE CODE

With this control, both the EIA and the ISO codes can be used.

EIA code: EIA RS-244-A

ISO code: ISO 84

Table 3.1.1 shows the EIA and ISO punched tape formats.

Before starting programming any machining operations, decision must be made as to the code to be used.

3.1.2 EIA/ISO AUTO-SELECT

Before starting to use part program tapes, the control must be switched to the same code as the tapes, inaccordance with the procedure for writing-setting under 4.3.6 DISPLAY AND WRITING OF SETTING DATA.

When "1" is set with setting #6001$_{D6}$, the control is automatically adapted to the code used for the part program tape. The control recognizes the code used when it reads the first EOB code in the label skip mode, and all the subsequent data will be read automatically in that code.

When "0" is set with #6001$_{D6}$, the control will not discriminate the code used automatically, and will read all tapes in the code specified by the setting with #6000$_{D7}$. That is:

When "0" is set with #6000$_{D7}$ ··· EIA code

When "1" is set with #6000$_{D7}$ ··· ISO code

If the code setting of the control and the code of the tape are different, alarm code 015 will be displayed.

NOTE: For punching tapes, the code must be selected by the setting of #6000$_{D7}$.

3.2 PROGRAMMING

3.2.1 PROCESS SHEET

Programs are first drafted on process sheets.

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

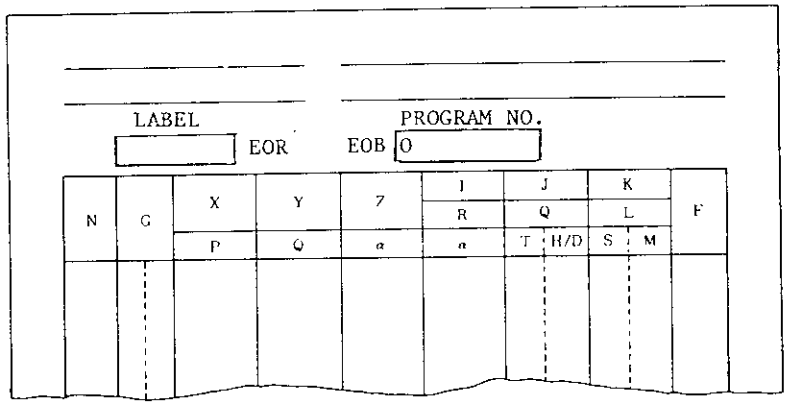


Fig. 3.1 Example of Process Sheet

Table 3.1 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	—									
									—	(			○		○	○			
									—	)	○		○		○	○			○
	○	○	○		○				+				○		○	○		○	○
	○				○				−				○		○	○	○		○
	○				○	○	○		o	:			○	○	○	○		○	
		○	○		○			○	/		○		○		○	○	○	○	○
	○	○	○	○	○	○	○	○	Del	DEL	○	○	○	○	○	○	○	○	○
○	○	○	○	○	○	○	○	○	All Mark		○	○	○	○	○	○	○	○	○
Sec Note 2.									#		○		○			○		○	○
○			○	○	○				*		○		○		○	○		○	
○				○	○				=		○		○	○	○	○	○		○
○		○	○		○				[		○	○		○	○	○		○	○
○	○				○		○		]		○	○		○	○	○	○		○
○			○		○				$				○			○	○		
○				○	○				@		○	○				○			
○			○	○	○				?				○	○	○	○	○	○	○
	○	○		○	○		○	○	.				○		○	○	○	○	

Notes:

1. For characters from # to ?, EIA codes have not been agreed upon. In the present system, for the time being, the above provisional codes are used.

2. EIA code of character # can be designated by the parameter #6017.

3.2.2 GENERAL PART PROGRAM FORM

Part programs are generally written on tapes in the following formats.

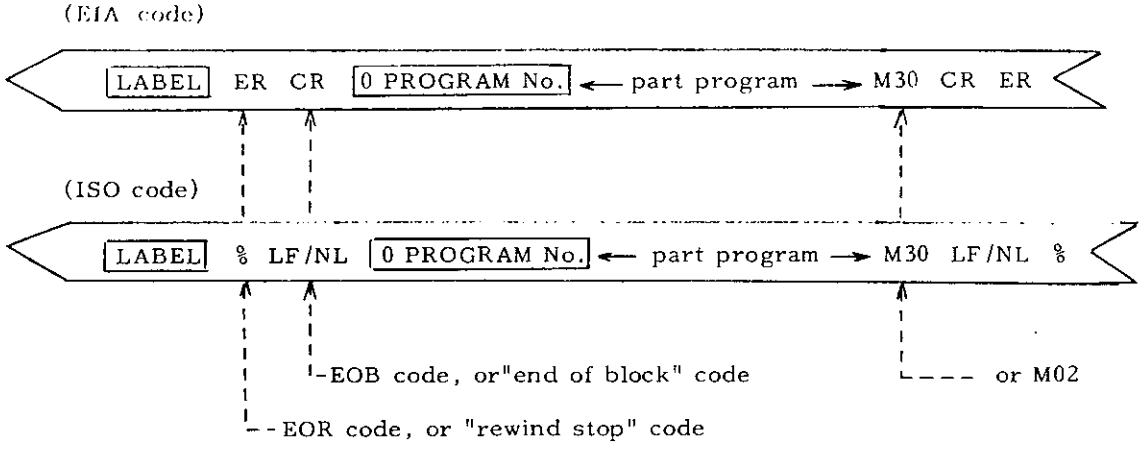


Fig. 3.2

To facilitate classification and handling of tapes, any identifying labels may be written at the leading end of all part program tapes. Since all data appearing before the first EOB code are skipped by the NC reading these tapes using the label skip function, even addresses and function codes not specified may be written here, and also out-of-parity codes may be written.

The EOR (rewind stop) code following the label is the stopping point of rewinding motion which is initiated by a rewind command.

When programs on a part program tape is transferred into the memory, the data between the first EOB code, following label skipping, and the next earliest EOR code is transferred. Therefore, the EOR code at the end should not be omitted.

3.2.3 CAUTIONS IN PROGRAMMING

- One block ends with an EOB code.
  The EOB code is CR in EIA, and LF/NL in ISO. However, in the example programs in this manual, ";" is used to represent EOB codes to make recognition easy.

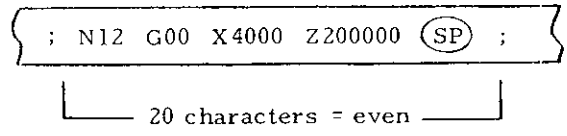
- One part program ends with a block containing M02 (end of program) or M30 (end of tape).

- When an M02 or M30 code is read, the NC resets itself or rewind the tape (or the memory), depending on the design of the machine under control. For this, refer to the manual prepared by the machine tool builder.

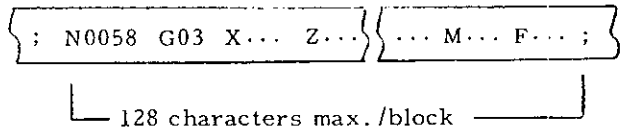
- Do not use any characters other than the address characters and the function code characters specified in 2.1.2 ADDRESS AND FUNCTION CHARACTERS.

- When the tape vertical parity check (TV check) function is to be used, the number of characters in each block must be made even with an SP (space) code.
  Disregard codes such as BS, Tab, SP, UC, LC, and Del should be avoided in the significant data area, if unnecessary.

- The maximum number of characters that can be written in a block is 128, not counting disregard characters such as Del.



(a) Evening up number of characters for TV check (odd is treated as error)



(b) Maximum number of effective characters that can be written in a block

Fig. 3.3

3.2.4 TV CHECK (TAPE VERTICAL RARITY CHECK)

When the tapes are to be checked for vertical parity, programs must be so made that each block (including EOB) contains even number of characters. Normally, SP codes are used to make the number of characters even.

The TV check function is turned on and off by the setting function. While the TV check function is on, all blocks containing odd number of characters are regarded as errors.

Setting No.: $\#6000_{D6} = 0$ ... TV check off

$\#6000_{D6} = 1$ ... TV check on

3.3 PART PROGRAM TAPE PUNCHING

3.3.1 PAPER TAPE SELECT

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.

3.3.2 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 hole part at the leading and the trailing ends. 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.

3.3.3 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

3.4 PART PROGRAM TAPE HANDLING

3.4.1 SPLING NC TAPE

To join part program tapes, the two ends should be placed end to end without overlapping and without a space, and a proper length of splice tape should be pasted on one side. (approx. 0.08 mm in thickness) Splice tapes are available in the fully perforated type and in the type with which only the feed holes are punched, but the former is more convenient. After splicing, the tape should be checked for correct alignment of the feed holes before use. Do not use rigid industrial adhesive, and do not make the joint too thick, as these conditions are conductive to jamming troubles.

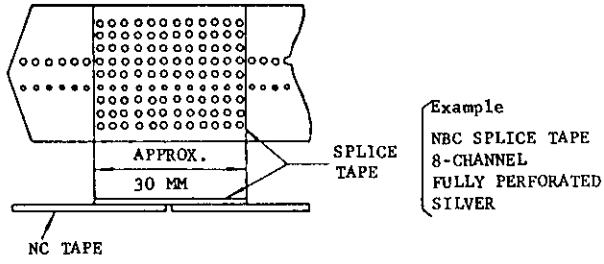


Fig. 3.4 Splicing Part Program Tape

3.4.2 KEEPING NC TAPE

Part program tapes should be stored in a clean area, free of contaminants and humidity. Do not handle part program tapes wearing gloves contaminated with oil or cutting fluid.
Generally, properly maintained part program tapes can last at least 300 cycles, with one cycle consisting of one reading and one rewinding pass.

4. NC OPERATOR'S STATION WITH CRT CHARACTER DISPLAY

4.1 PUSHBUTTONS, LAMPS AND KEYS

Fig. 4.1 shows an overall view of NC operator's panel with CRT display. The names and functions of operator devices are as follows.

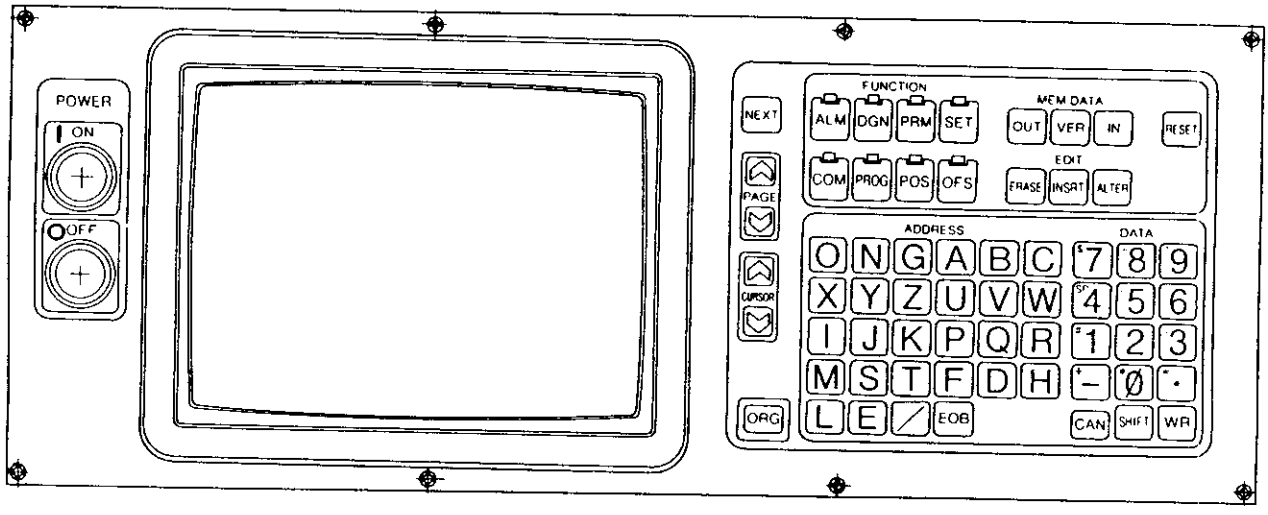


Fig. 4.1 Standard NC Operator's Station with CRT Character Display

4.1.1 POWER ON/OFF PUSHBUTTONS

· POWER ON pushbutton

To turn on the power for the control: Depress the pushbutton first to turn on the control power and depress it again to turn on the servo power. Push this button to recover the servo power after an emergency stop.

· POWER OFF pushbutton

To turn off the power for the control: Depress it to turn off both the servo and control powers.

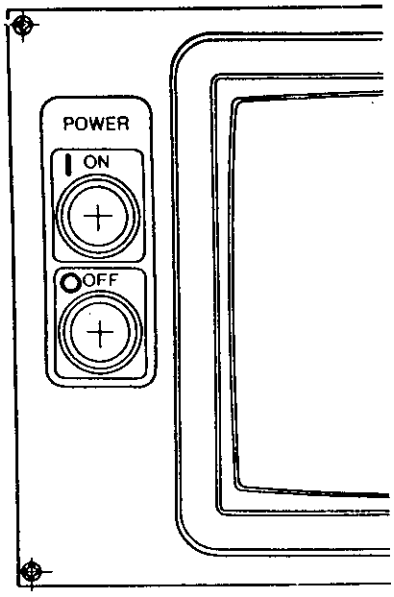


Fig. 4.2

4.1.2 CRT CHARACTER DISPLAY

According to each operation, this display indicates the alpha-numerical data in a regular size (1x1)and, triple-size (3x3) of the regular size.

Braun tube size: 9 inches

Maximum number of characters:
32 characters x 16 lines =
512 characters (at regular size)

Indicating characters:

Numerals - [0] through [9] , [-] , [.]
Alphabetic characters - [A] through [Z]
Special code - [;] (EOB), [/] (slash), etc.

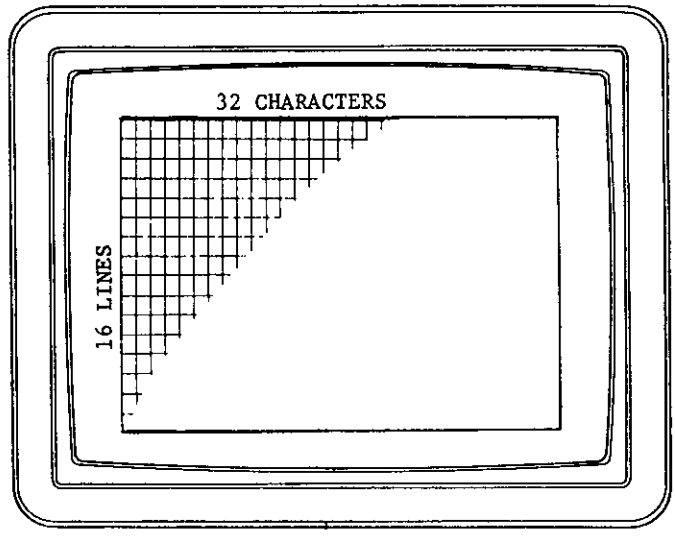


Fig. 4.3 Braun Tube

4.1.3 FUNCTION SELECT KEYS

The key selects one of eight functions for the operation of the display and MDI. Pushing a key makes it effective and light up.

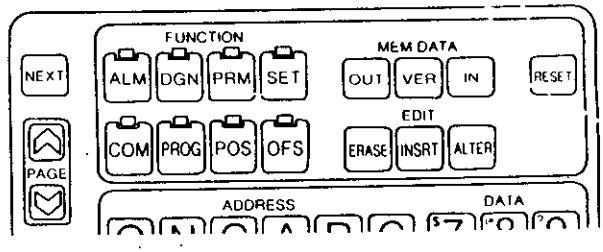


Fig. 4.4

· [ALM] (Alarm) key:

Select this key for display of alarm and status codes. The function becomes effective when the power is turned on or an alarm occurs.

· DGN (Diagnosis) key:

Select this key for display of input/output signal status.

· PRM (Parameter) key:

Select this key for display or writing-in of parameters.

· SET (Setting) key:

Select this key for display or writing-in of setting data.

· COM (Command) key:

Select this key for display or writing-in (MDI) of the command data for automatic operation.

· PROG (Program) key:

Select this key for display or writing-in of a part program.

· POS (Position) key:

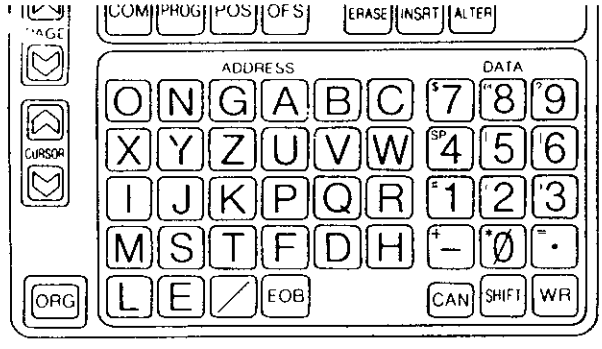
Select this key for display of various current positions.

· OFS (Offset) key:

Select this key for display or writing-in of tool offset values.

4.1.4 ADDRESS KEYS

These keys are to designate an address character when writing in various data.



Note: Special characters

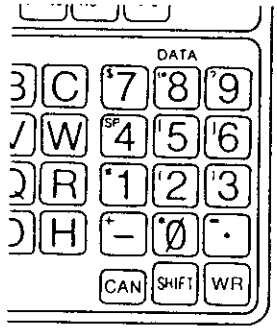
/ (Slash) key: For an optional block skip command.

EOB (EOB) key: For the block end command. On the CRT display, ";" is displayed instead of "EOB."

Fig. 4.5

4.1.5 DATA KEYS

These keys consist of 15 keys in total, such as 0 through 9, - (minus) . CAN SHIFT WR , and can be used for writing-in of such all numeral values as tool offset value setting data, parameter data, and so on, in addition to command value.



Note:

0 to 9 key
- (minus) key : For input of numerical data

. (decimal point) key: For input of decimal poit

CAN (cancellation) key:
For cancellation of the numeric value or address data erroneously keyed.

WR (write) key:
For storing address data by address keys and data keys into buffer storage.

SHIFT (shift) key:
Depressing SHIFT key after depressing 0 to 9, - or . key makes the display turn into * to ? , + , = which are written on the upper left corner of the keys. These special characters are used in user macro.

Fig. 4.6

4.1.6 NEXT KEY

The NEXT key is used for special purpose and expanding function in display or writing data.

· Writing of optional in EDIT mode.

· For other special purpose and expanding function.

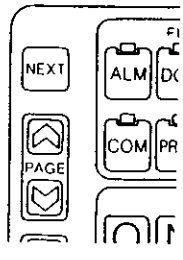


Fig. 4.7

4.1.7 PAGE KEYS

The PAGE key is used for display the next page or the previous page when CRT display is regarded as page.
For example, when a bundle of tool offset values are displayed by OFS key, this key is pushed to display the next bundle of tool offset values, which just looks like opening the pages of a book.

- Depressing [PAGE ↓] key displays the next page.
- Depressing [↑ PAGE] key displays the previous page.
- Keeping the PAGE key depressed makes the page step automatically forward or backward.

4.1.8 CURSOR KEYS

The CURSOR control key is used to move the cursor. For example, when a page of parameter data are displayed by PRM key.

- Depressing [CURSOR ↓] key moves the cursor forward.
- Depressing [↑ CURSOR] key moves the cursor forward.
- Keeping the cursor control key depressed makes the cursor move automatically forward or backward.

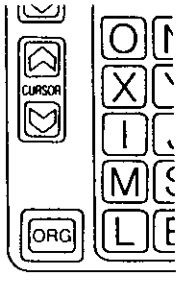


Fig. 4.8

4.1.9 ORG (ORIGIN) KEYS

The ORG key is used to set the current position of the machine tool as the origin of the reference coordinate system.
The origin setting can be made for each axis.
The reference coordinate system means the coordinate system which is set by G92 command or the automatic coordinate system setting.

ORG key is used for the following operation.

- Reset of current position (UNIVERSAL, EXTERNAL)
- Reset of tool offset values
- Reset of operation time

4.1.10 EDIT KEYS

These keys are for editing a stored part program.

ERASE key: Used for erasure of data in storage.

INSRT key: Used for insertion of data in memory.

ALTER key: Used for alteration of data in memory.

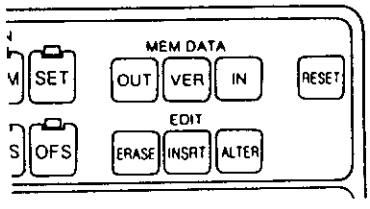


Fig. 4.9

4.1.11 MEM DATA (MEMORY DATA) KEYS

TAPE KEYs are to start the tape operation except in the automatic operation mode. They are effective only in the EDT mode.

- OUT key

  This key is to start outputting various data in memory through data I/O interface.

- IN key

  This key is to start storing various data into memory through tape reader or data I/O interface.

- VER key

  This key is to start verifying between memory data and punched tape data.

4.1.12 RESET KEY

This key resets the control.

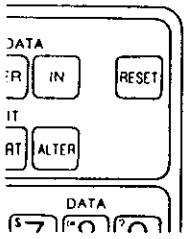


Fig. 4.10

Operations to be executed by this RESET key are:

- Move command cancel
- Buffer register clear
- Alarm code release if the cause is eliminated
- Tool offset cancel
- Auxiliary function cancel
- Label skip function ON
- Memory pointer rewind
- Sequence number reset
- RST signal transmission
- G code of A group

Refer to 2.9.1 LIST OF G CODES AND GROUPS.

The following will not be affected by operating the RESET key.

- Current position values of each axis.
- F commands
- S, T and B commands
- Tool offset values, setting data, parameter data

NOTE: Depressing the RESET key or the remote reset pushbutton is defined as "Reset operation" in this manual.

4.1.13 TAPE FEED AND SYSTEM NO. SWITCHES

These switches are mounted above the tape reader.

- TAPE FEED switch

  This is a switch to wind and rewind the tape manually. Setting the switch to F (forward) causes the tape to feed. To rewind the tape. set the switch to R (reverse). This switch is effective, either manually or automatically.

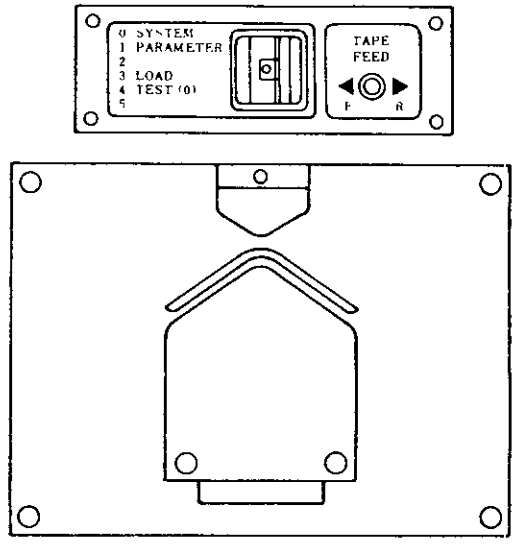


Fig. 4.11

- SYSTEM NO. switch

  Set the switch at "0" during the usual operation. Functions of its each setting are as follows.

  "0": SYSTEM

  For usual operation. Writing parameters is prevented.

  "1": PARAMETER

  To write parameters. At this position, the Cycle Start is prevented.

  "3": LOAD

  To store the maintenance tape into the control.

  "4": TEST (0)

  The usual operation is similar to case of "0" SYSTEM. Self-diagnostics of the memory contents and checking of reference zero return position are omitted.

4.2 POWER ON/OFF OPERATION

4.2.1 TURNING ON POWER

Check the machine before turning on power, referring to the machine tool builder's manual for details. Operations after completion of preinspections are as follows.

- Depress the POWER ON pushbutton to turn on the control power. The internal timer will be read in about two seconds. Then the servo power is ready for turning on, which is shown by alarm code "310."
- Depress the POWER ON pushbutton again to turn on the servo power. The NRD (NC READY) signal is sent out when the NC power is normally supplied.
- When the NRD signal turns on the machine power, and the MRD (MACHINE READY) signal returns back to the control, the READY lamp will be lit.

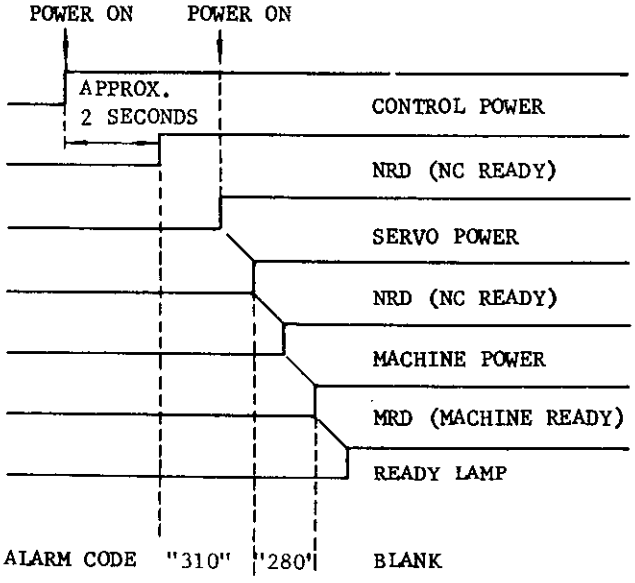


Fig. 4.12 Sequence of Turning on Operation

4.2.2 TURNING OFF POWER

Depressing the POWER OFF pushbutton causes both the servo and control powers to be turned off simultaneously. However, for more stable operation, use the following procedure.

- First depress the EMERGENCY STOP pushbutton to cut off the servo power. The NRD (NC READY) signal is interrupted, which usually results in turning the machine power, too.
- Depress the POWER OFF pushbutton to cut off the control power.

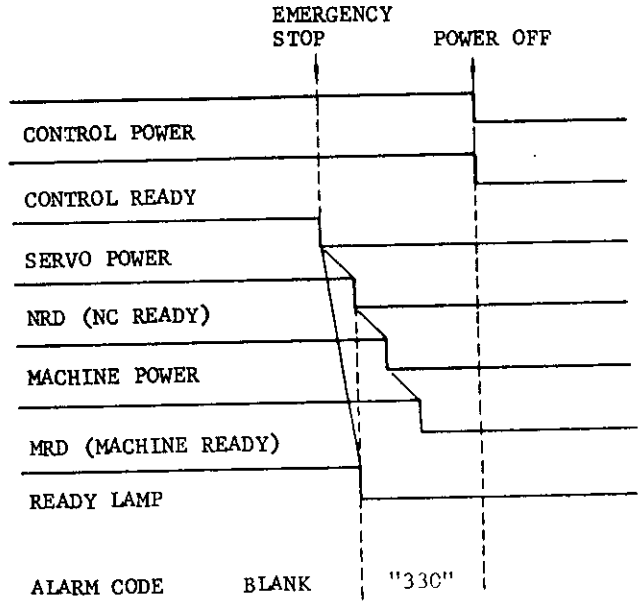


Fig. 4.13 Sequence of Turning off Operation

4.2.3 REMOTE POWER ON/OFF PUSHBUTTONS

Connect the power ON/OFF pushbuttons to EON, EOF and COM terminals on the control panel as shown below. Then the remote turning ON/OFF operation can be made exactly the same as with the POWER ON/OFF pushbuttons.

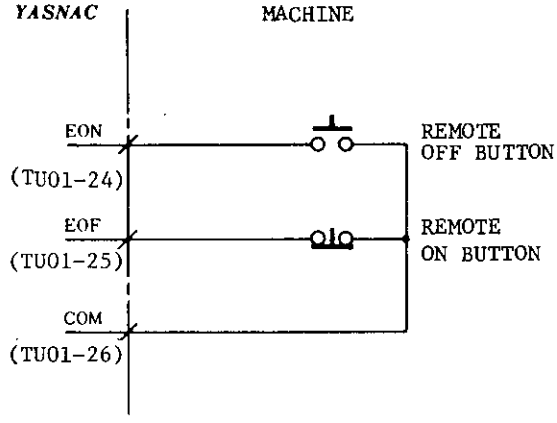


Fig. 4.14 Connections of Remote ON/OFF Pushbuttons

4.3 DISPLAY AND WRITING OPERATION

4.3.1 CONSTANT DISPLAY

The following display is made on both the top and bottom on the displayed picture of CRT, irrespective of the FUNCTION key currently selected.

· Function message

Anyone of the following eight function messages corresponding to the function key is displayed at the top of CRT display.

ALARM	PROGRAM
DIAGNOSIS	POSITION
PARAMETER	OFFSET
SETTING	
COMMAND	

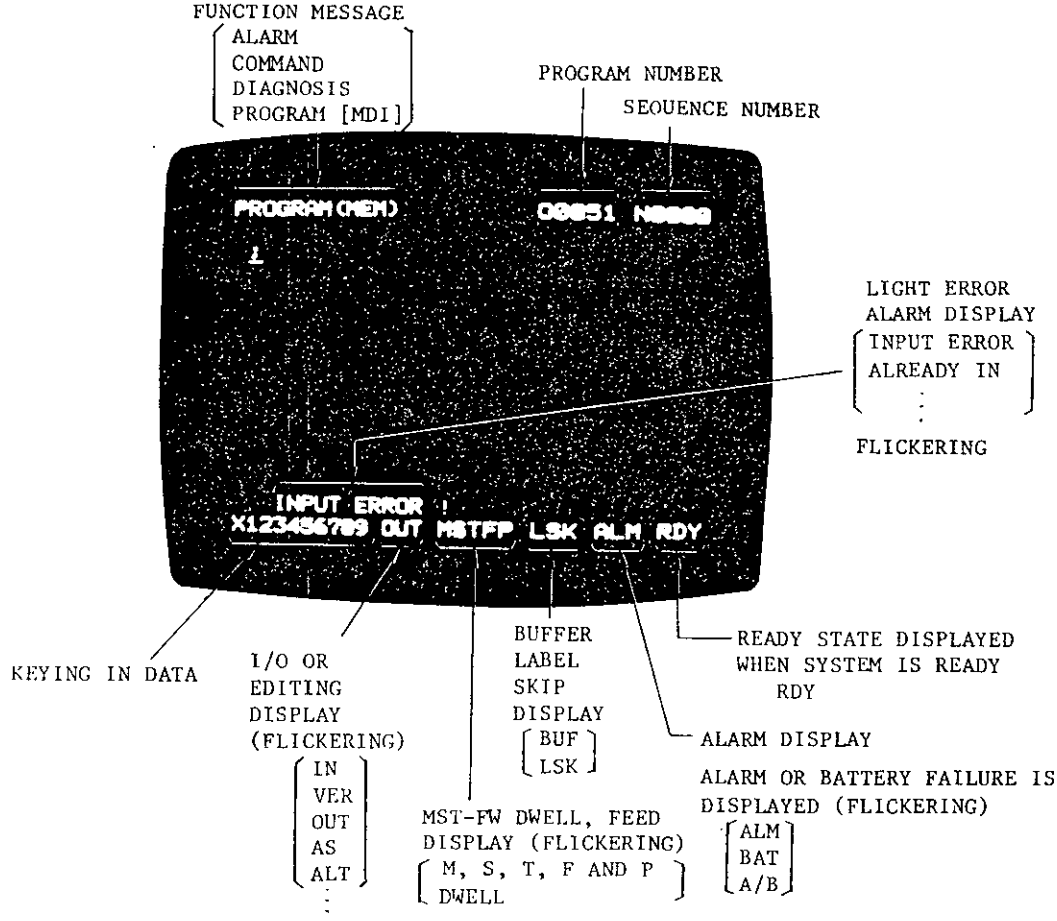


Fig. 4.15

· Program No.

0 and 4 digits of program No. under execution is constantly displayed at the top of CRT display irrespectively of function key.

· Sequence No.

N and 4 digits of program No. under execution is constantly displayed at the top of CRT display irrespectively of function key.

· Display of keying data

Up to 10 characters of keyed in data can be displayed at one time. The data is processed by using ERASE key, INSRT key, ALTER key, etc.

· Display of I/O and editing (flickering)

The following messages are flickerlingly displayed during loading of punched tape, address search or edition.

"IN" ··· loading tape

"VER" ··· verifying tape

"OUT" ··· punching tape out

"AS" ··· searching address

"ALT" ··· altering data in EDIT mode

"INS" ··· insearting data in EDIT mode

"ERS" ··· erasing data in EDIT mode

· Display of MST-FIN signal waiting, dwelling and feeding

"M" ··· waiting FIN signal of M command

"S" ··· waiting FIN signal of S command

"T" ··· waiting FIN signal of T command

"F" ··· feeding
"R" is displayed at rapid traverse

"P" ··· loading tape

"DWELL" · dwelling

M, S, T, F and P are displayed independently each other.

· Display of the state of buffer full and label skip

"BUF" ··· displayed at completion of advanced reading

"LSK" ··· displayed at label skip on

· Display of alarm (flickering)

Alarm continues to be displayed flickeringly until the cause is removed and reset operation is made.

"ALM" ··· indicates alarm state occuring

"BAT" ··· indicates battery alarm occuring

"A/B" ··· indicates both of alarm and battery alarm occuring

· Display of ready state

"RDY" ··· indicates the system is normal and the control is operable

· Display of light errors (flickering)

The messages shown below indicate light errors which occur in keying or searching operation. Differing from the alarm codes, these error messages are cleared by depressing some key. (Generally CAN key)

"INPUT ERROR!" ··· Format error of keyed-in data

"ALREADY IN!" ··· The same number of part program has been stored already.

"EDIT LOCK ON!" ·· Editing operation is made with Edit Lock on.

"MEMORY OVER!" ··· Part program to be stored is beyond memory capacity.

"PROGRAM OVER!" ··· Registered number of part program is beyond 99 (basic) or 199 (option).

"NOT FOUND!" ··· Desired data has not been located.

"BREAK POINT!" ··· Break point occurs

4.3.2 DISPLAY OF COMMAND DATA

1. Depress COM key.

   Anyone of the following three digits appears.

   A. Command data (COMMAND)

   B. Repetition number of subprogram (SUB PROG. NESTING)

   C. State of tool offset (COMMAND ⎩OFFSET⎭)

2. The above display steps forward or backward by depressing [PAGE ↓] or [↑ PAGE] key one by one.

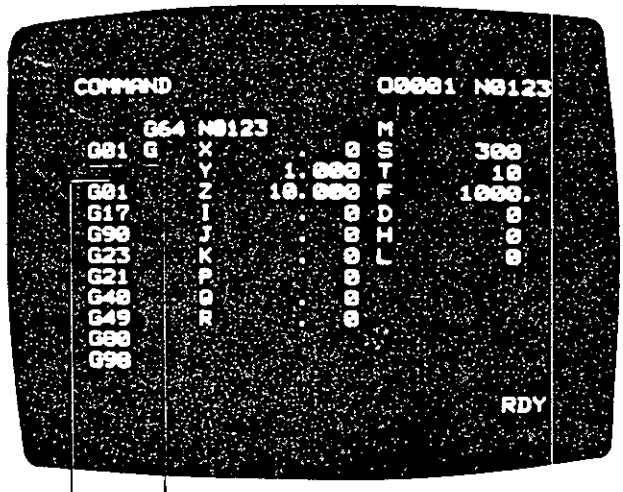
4.3.2.1 COMMAND DATA DISPLAY

The display shows the block data of under execution or just before execution in which compensation calculations have been completed. The conditions of the data to be displayed is as follows.

1. The data shows the contents of the active register during an automatic operation or a feed hold.

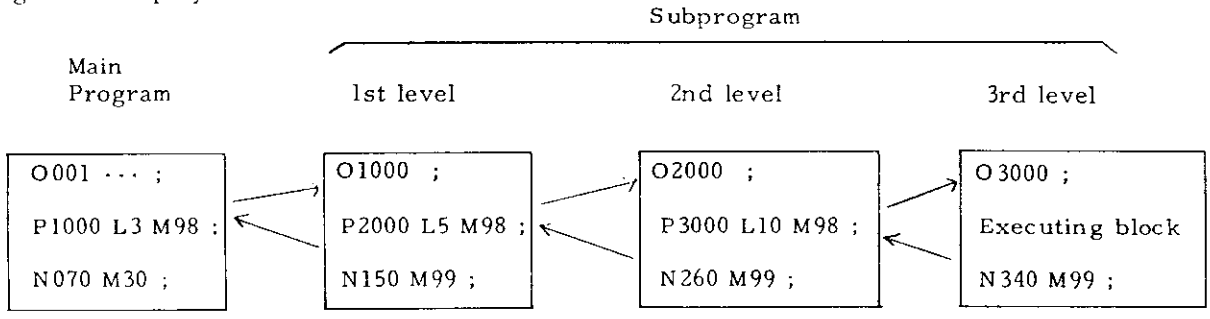
2. While the control is stopped at a block end, the contents of the buffer register are displayed. If the buffer register blank (BUF is not displayed), the contents of the just executed block are displayed.

3. In the MDI operation, the current block data are displayed after cycle is indicated.



4.3.2.2 DISPLAY OF REMAINING MUNBER OF REPETITIONS OF SUBPROGRAM (SUB PROG. NESTING)

The remaining number of repetitions of a subprogram is displayed.



The subprogram has executed the 3rd level twice and entered into the execution of 3rd time of the 3rd level.

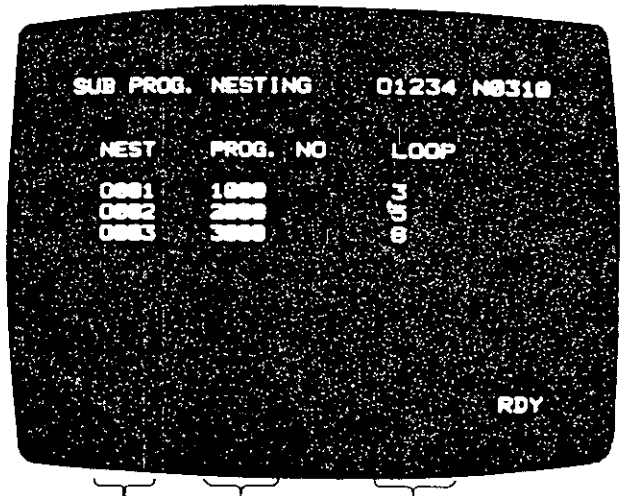
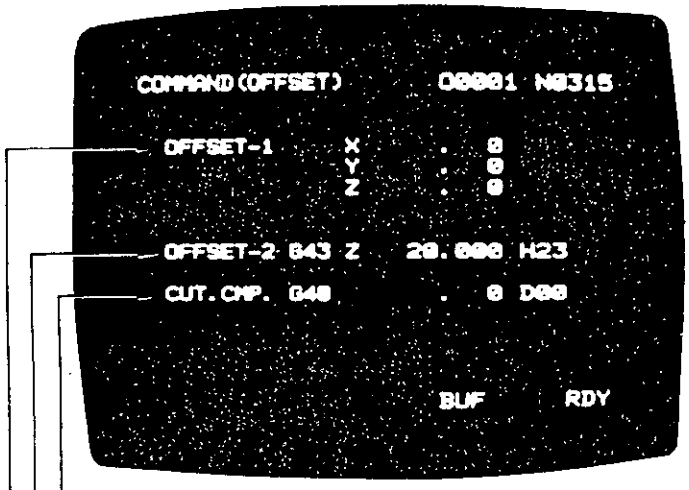


Fig. 4.16 Example of SUB PROG. NESTING

4.3.2.3 DISPLAY OF TOOL OFFSET STATE: COMMAND (OFFSET)

The current state of tool offset is displayed as shown below.



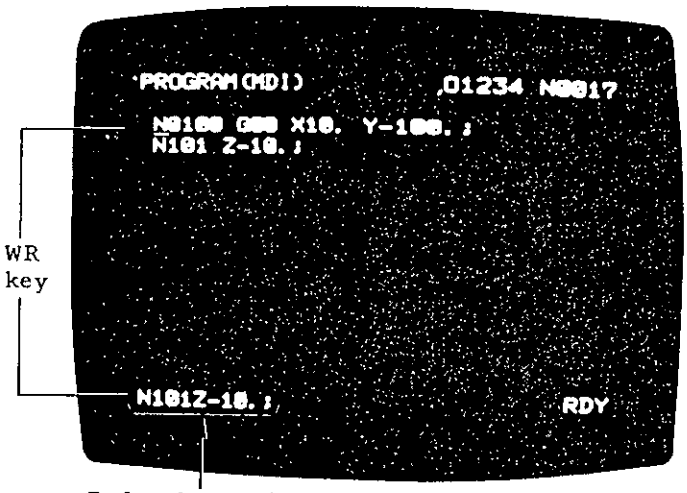
Value of tool position offset

Value of tool length offset and H code

Value of tool radius compensation and D code

NOTE: Function COM is exclusively used for display. Data cannot be written under function COM. Select function PROG. to write block data.

Fig. 4.17 Example of display of tool offset state



Refered to as "the data which has just been entered."

Enter [N], [1], [0], [1], [Z], [-], [1], [0], [•], [EOB] in this order.

Note: The depression of the EOB key appears ";."

Fig. 4.18

4.3.3 WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI

In MDI, EDIT, and MEM modes, it is possible to write data into blocks by MDI and perform operation. (MEM mode permits displaying only.) The following operations are possible when function PROG. is selected.

1. In MDI mode

A. Writing and displaying data

Data entered through the keyboard will appear on the bottom line of the CRT screen, from left to right. Up to 10 characters may be entered at a time.

Depress the WR key, then the data moves to the middle of the CRT and the bottom line becomes blank.

In MDI mode, data of up to 10 lines may be collected on the CRT screen.

B. Editing MDI data

The [CURSOR ↓], [↑ CURSOR], ERASE, INSRT, and ALTER keys permit editing multi-block data written in. Address (word) pointed to by the cursor will be edited. The [CURSOR ↓] and [↑ CURSOR] keys move the cursor forward and backward.

(1) ERASE key: When this key has been depressed, the whole word designated is erased.

(2) INSRT key: This key inserts the data which has just been entered to the location which is next to the word the cursor points to.

(3) ALTER key: This key replaces the word which the cursor points to by the data which has just been entered.

(4) WR key: This key appends the data which has just been entered at the end of the program displayed. In MDI mode, it is possible to edit only the current page. Note: Editing is possible in MDI mode and EDIT mode.

4.3.3 WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI (CONT'D)

C. Operation in MDI mode

Depress the Cycle Start button to let the program of blocks displayed on the CRT to run automatically.

At the end of operation, the multi-block program displayed is cleared from the CRT.

2. In EDIT mode

See 4.6 EDITING OPERATION OF PART PROGRAMS.
Programs written by MDI can be executed repeatedly by M99.

3. In MEM mode

This mode permits to display the program which is running by memory operation. The cursor points to the top of the block which is currently being executed, and it moves to the next block as execution proceeds.

Up to 10 lines may be displayed at a time. When execution of the ninth has been completed, the next page appears with the tenth line of the last screen appearing at the top.

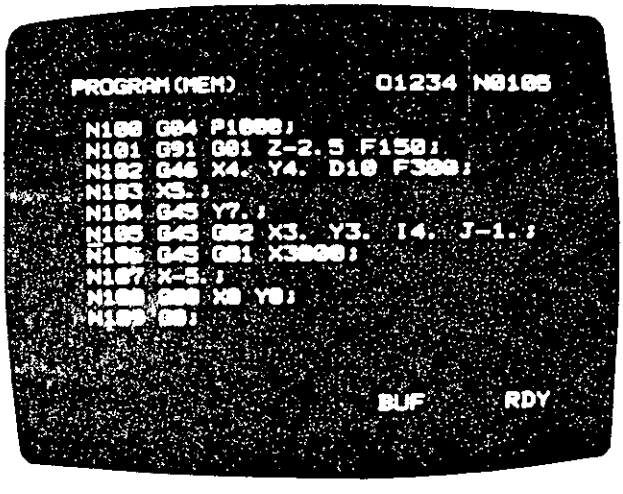


Fig. 4.19

4.3.4 DISPLAYING AND RESETTING CURRENT POSITION

It is possible to observe the current position in any mode. Operate as follows.

1. Depress the POS key. Any of the following screens will appear.

   A. Current position display-universal (POSITION [UNIVERSAL])

   B. Current position display-external (POSITION [EXTERNAL])

   C. Current position display-increment (POSITION [INCREMENT])

   D. Current position display-all (POSITION)

   E. Servo positioning error display (POSITION [ERROR])

2. Depress the [PAGE ↓] or [↑ PAGE] key, and one screen will change to the next.

NOTE: Mode E is possible only when the system No. switch is set at "4."

4.3.4.1 CURRENT POSITION DISPLAY (UNIVERSAL):

POSITION (UNIVERSAL)

The current tool position which is the sum of the parameters of move commands will be displayed. Depending on the value of parameter $\#6005_{D5}$ (G92 display preset), either of the following will appear.

· When parameter $\#6005_{D5} = 1$ (Position in the reference coordinate system)

1. The tool position displayed is based on the coordinate system set up with G92.

2. To reset this screen, depress the ORG key after designating an axis with the ADDRESS key. The current position will be reset to "0." This is possible only during a manual operation mode (RAPID, JOG, STEP, or HANDLE). The depression of the ORC key is ineffective during normal operation and in the "buffer full" state.

3. The coordinate system which is employed for this screen is called the "reference coordinate system." A work coordinate system (option) will be set up in reference to the reference coordinate system.

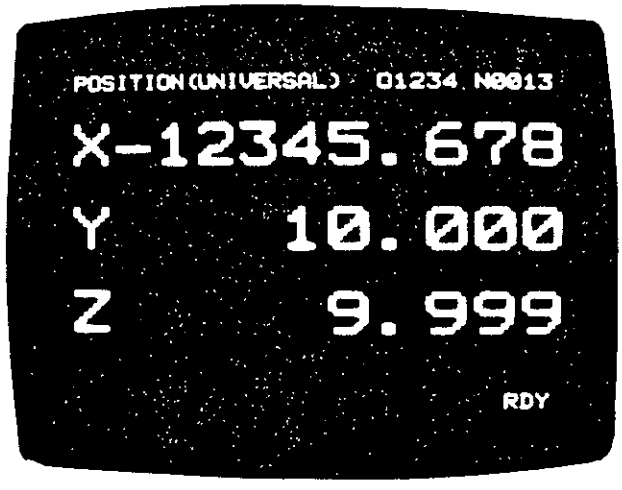


Fig. 4.20 Current Position Display (Universal)-Example

· When parameter #6005$_{D5}$ = 0 (Position obtained by simple summation)

1. G92, even if issued, does not affect the display. Move commands will be summed and displayed.

2. To reset this screen, depress the ORG key after designating an axis with the ADDRESS key. The current position along the designated axis will be reset to "0."
This is possible in any modes and even during operation.

4.3.4.2 CURRENT POSITION DISPLAY (EXTERNAL): POSITION (EXTERNAL)

· Move commands will be summed and displayed. G92, if issued, does not affect the display.

To reset this screen, depress the ORG key after designating an axis with the ADDRESS key. The current position along the designated axis will be reset to "0." This is possible in any modes and even during operation.

These displaying and resetting operations are the same as with the case of POSITION(UNIVERSAL) #6005$_{D5}$ = 0 (Position obtained by simple summation). But the resetting operation is effective only to the displayed screen since there are independent position registers.

The data displayed in this mode are the same as those displayed on the "3-axis/4-axis external position display" (option). You may consider that the coordinate data of POSITION (EXTERNAL) are transmitted to the outside as they are unchanged.

Fig. 4.21 Current Position Display (External)-Example (with 4-axis control)

4.3.4.3 CURRENT POSITION DISPLAY (INCREMENT): POSITION [INCREMENT]

Displayed in this mode are:

· In automatic mode, distance to the end point of the block at every moment

· In manual mode, distance to the position where manual operation is to start.

The increment display in manual mode will be cancelled in automatic mode. (Fig. 4.3.4.3)

4.3.4.4 CURRENT POSITION DISPLAY (ALL): POSITION

· All position data will be displayed.

· 〈MACHINE〉 coordinates indicate the current position in the coordinate system whose origin is the reference point set up by resetting. Data for "stored stroke limit^{+}" and "pitch error compensation^{+}" functions are defined in this coordinate system. (Fig. 4.3.4.4)

4.3.4.4 CURRENT POSITION DISPLAY (ALL): POSITION (CONT'D)

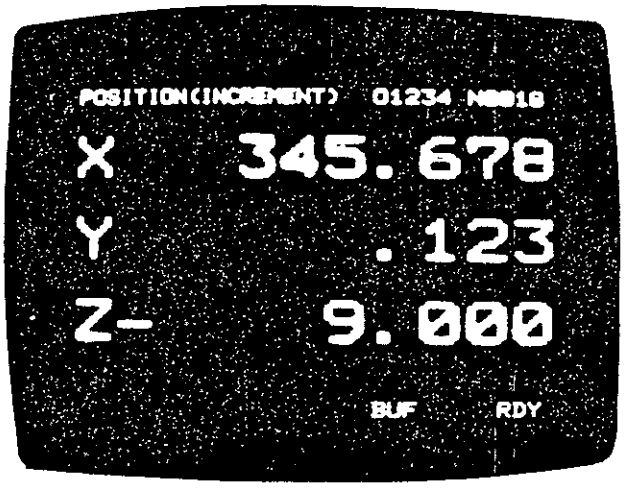


Fig. 4.22 Current Position Display (Increment) - Example

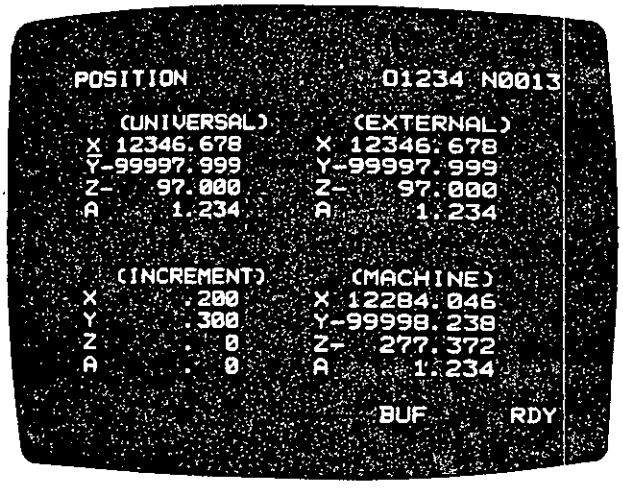


Fig. 4.23 Current Position Display (All) - Example

4.3.4.5 SERVO POSITIONING ERROR DISPLAY: ERROR PULSE

- This screen appears only when the system No. switch is set at "4." This mode will be normally used during maintenance.
- Servo positioning error means the difference between the command position and the current tool position. Error will be displayed in units of pulse.

4.3.5 DISPLAYING AND WRITING TOOL OFFSET DATA

Tool offset data are stored in the memory of the control. These data may be displayed and rewritten in any modes and even during automatic operation.

- Displaying tool offset data.

1. Select the OFS function key.
2. Enter numerals, like 1 and 0, then depress the [CURSOR ↓] or [↑ CURSOR] key. Then ten pairs of tool offset number and tool offset, including the designated pair, will be displayed and the cursor positioned at the designated tool offset number.

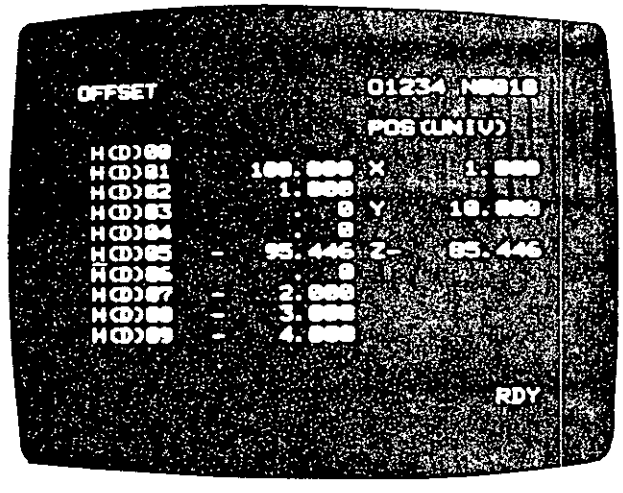


Fig. 4.24

3. Depress the [CURSOR ↓] or [↑ CURSOR] key to move to a smaller or larger tool offset number. If you move the cursor beyond the first or last tool offset number displayed in the current screen, the neighboring ten sets of tool offset number and tool offset will appear automatically.

4. The preceding or following page may be displayed by depressing the [↑ PAGE] or [PAGE ↓] key. The cursor will be positioned at the first tool offset number displayed at this time.

5. Tool offset will be displayed in units of 0.001 mm (or (0.0001") and up to 999.999 mm (or 99.9999").

· Writing tool offset data

To rewrite a tool offset data, specify an increment which is to add arithmetically to a tool offset held in memory.

1. Position the cursor at the tool offset number whose offset data is to be changed.

2. Enter the increment which is to be added to the tool offset.

3. Depress the WR key. Then the specified increment will be added to the old tool offset.

NOTES:

· A new tool offset itself may be input instead of an increment. For this purpose, depress the ORG key first. The tool offset number pointed by the cursor will be reset to "0." Then enter a new tool offset.

· Tool offset data held in the memory of the control are preserved even after power is turned off.

· It is possible to rewrite tool offset data in any modes, even during automatic operation.

· Tool offsets modified during automatic operation become effective when the system starts to read commands for a new block. The old tool offsets remain effective for the current block and the blocks whose data are already read in the buffer for advance reading.

4.3.6 DISPLAYING AND WRITING SETTING DATA

In this system, varying setting data are held in the internal memory and permit to specify mirror image axes, TV check on/off, etc. For details, see Appendix 1, LIST OF SETTING NUMBERS.

It is possible to display and write setting data at any time even during automatic operation.

1, Types of setting

Setting is made in binary mode or decimal mode.

A. Binary mode

Setting numbers #6000-#6004 are associated with setting data of binary mode, that is, 8-bit information (D7-D0). Each bit indicates the on/off state of the associated function. The decimal value of each line is given at the rightmost column.

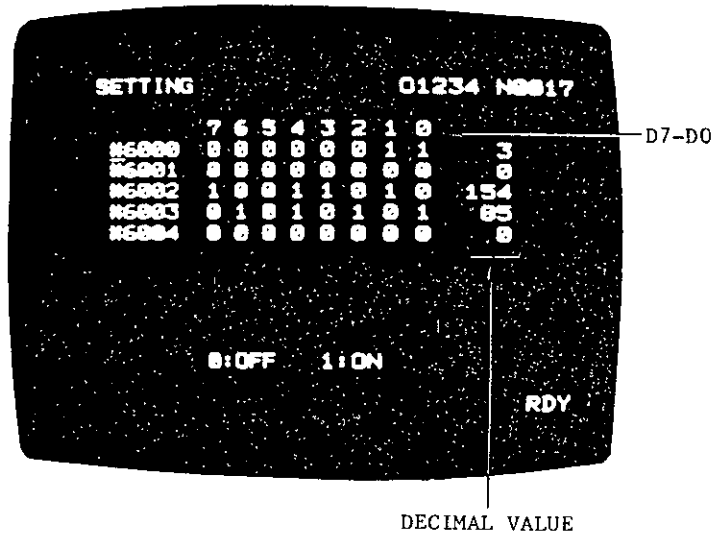


Fig. 4.25 Setting (Deciaml model)-Example

B. Decimal mode

Setting numbers of #6200-#6219 and #6500-#6599 are associated with setting data of decimal mode.

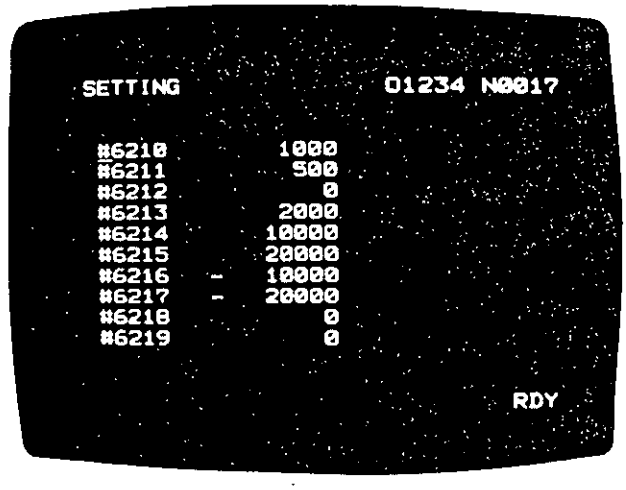


Fig. 4.26 Setting (Decimal mode)-Example

4.3.6 DISPLAYING AND WRITING SETTING DATA (CONT'D)

2. Displaying setting data

· Enter a setting number then depress the [CURSOR ↓] [↑ CURSOR] key. ("#" need not be entered.) Up to 10 groups of setting number and data will be displayed at a time.

· Depress the CURSOR keys to change a setting number and the PAGE keys to change a screen.

3. Writing setting data

A. In binary mode

(1) Designate a desired setting number.

(2) Depress the INSRT key. The cursor moves to the bit data from a setting number. Designate the data of D7.

(3) Depress the [CURSOR ↓] key. Each time the key is depressed, the cursor moves one bit toward D0. Locate the cursor at a desired bit position.

(4) Depress the WR key. The designated bit data reverses (0 to 1 or 1 to 0). If you depress the WR key again, the bit data will reverse again. Normally, "1" designates on state and "0" off state.

(5) To write data in decimal mode, locate the cursor at the rightmost column (decimal data).

EXAMPLE: Writing in decimal mode

Entered data	7	6	5	4	3	2	1	0	
[0] [WR] →	0	0	0	0	0	0	0	0	0
[2] [5] [5] [WR] →	1	1	1	1	1	1	1	1	255

(6) Repeat steps (2) through (5) to write desired data. If you keep the [CURSOR ↓] or [↑ CURSOR] key depressed, the cursor will move column by column in the screen automatically.

(7) When data has been written, depress the INSRT key. Normally, this sequence of operations begins and ends both with the depression of the INSRT key.

B. In decimal mode

(1) Designate a desired setting number.

(2) Enter a data and depress the WR key. The data will be assigned to the setting number which the cursor points to.

(3) Depress the [CURSOR ↓], [↑ CURSOR], [PAGE ↓], or [↑ PAGE] key to change a setting number or the screen.

4.3.7 DISPLAYING AND WRITING PARAMETERS

In this system, varying parameters are stored in the memory and they determine operating conditions such as tape code and feed rate. For details, see Appendix 2, LIST OF PARAMETER NUMBERS. The parameters may be displayed at any time even during automatic operation.

1. Kinds of parameters

Parameters are displayed either in decimal mode in binary mode.

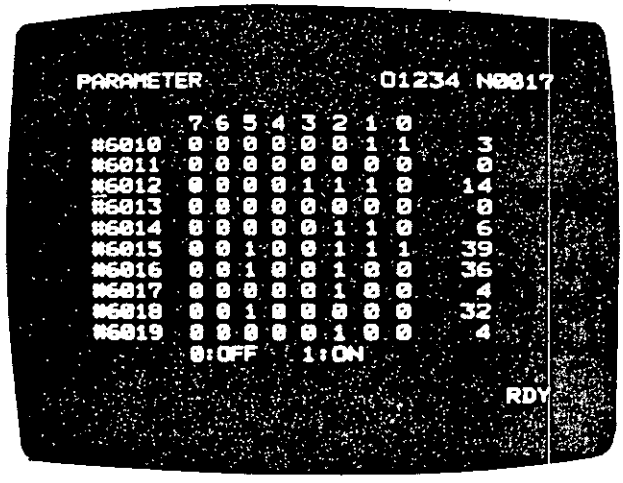


Fig. 4.27 Parameters (in binary mode)-Example

Parameter numbers #6005-#6045 are assigned to binary mode. Those of #6050 and up are assigned to decimal mode.

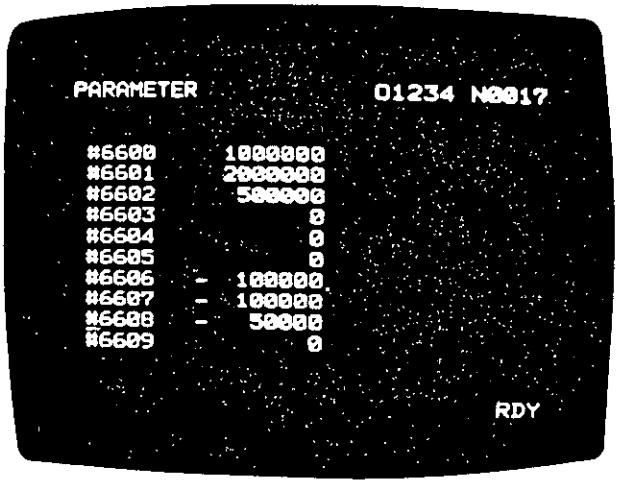


Fig. 4.28 Parameters (in decimal mode)-Example

2. Displaying parameters

   Operation is the same as in displaying setting data except that PRM should be depressed instead of SET. See 4.3.6 DISPLAYING AND WRITING SETTING DATA.

3. Writing parameters

   The parameter values are preset according to the performance of the machine and purposes. Therefore, you should consult the machine tool builder if you want to change parameter settings.

   The parameters are protected with a system No. switch provided on the tape reader so that they should not be destroyed by wrong operation. Normally system No. 0 is selected and, at this time, the parameters cannot be rewritten by any operation.

   A. The operation of writing parameters is the same as of writing setting data but the parameters are protected. See 4.3.6 DISPLAYING AND WRITING SETTING DATA.

   B. Parameters cannot be rewritten unless the system No. switch is set at "1."

      After rewriting parameters, be sure to reset the system No. switch at "0."

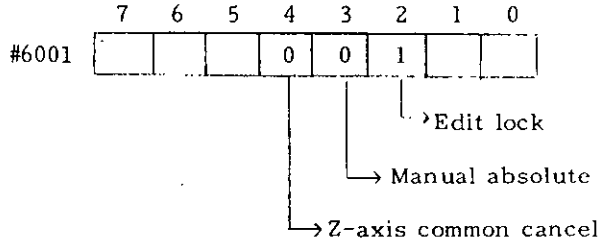
   C. If the following parameters have been changed, be sure to turn off power then turn it on again. Otherwise the system might fail to operate properly.

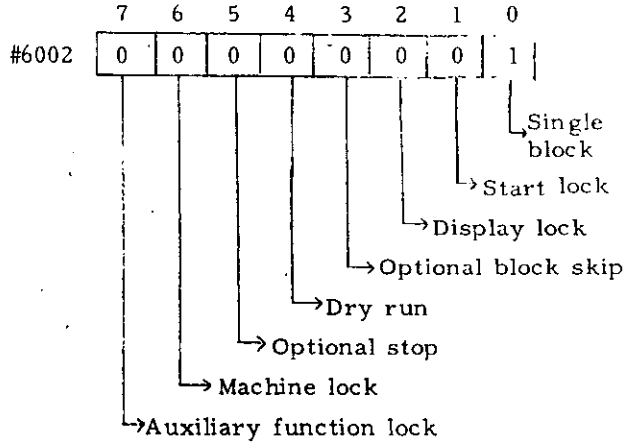
#6009
#6010
#6023
#6032
#6086 to #6091
#6094 to #6099
#6068 to #6071
#6322 to #6337
#6642 to #6645
#8000 to #8511
After reading in parameter tape

4.3.8 INTERNAL TOGGLE SWITCHES

The following switches may be easily turned on and off on the NC operator's station even when they cannot be operated on the machine control station. Setting numbers and their contents are as follows.

"1" = On, "0" = Off





4.3.8 INTERNAL TOGGLE SWITCHES (CONT'D)

If the machine control station is provided with the switches that turn on and off the above functions, the state of a switch on the machine's control station is ORed with that of the NC operator's panel to determine the state finally.

Setting data	Machine's switch	Result on/off
"0" = OFF	OFF	OFF
"0" = OFF	ON	ON
"1" = ON	OFF	ON
"1" = ON	ON	ON

The functions of the internal toggle switches work only when parameter #6006$_{D3}$ = 1 (internal toggle switch function on). If it is off, only the switches of the machine control station work.

4.3.9 OPERATION TIME DISPLAY

The system counts the duration of automatic operation and it may be displayed. This function permits the display of the time it has been taken for a piece of work or the total operational time of the system.

1. Procedure of display

   Depress the ALM key, then select a screen of running time with the [PAGE ↓] key as shown below.

   Three kinds of operation time will be displayed in hours, minutes, and seconds.

   Top: Total operating time after POWER ON

   Middle: Total operating time of CYCLE START

   Bottom: Total operating time of FEED

2. Resetting display

   Each time of operational time may be reset independently by the following procedure.

   When operating times are displayed:

   A. "1" "ORG" POWER ON time at the top will be reset.

   B. "2" "ORG" CYCLE START time in the middle will be reset.

   C. "3" "ORG" FEED time at the bottom will be reset

The timers of operation time preserve data unless they are reset, even after power is turned off.

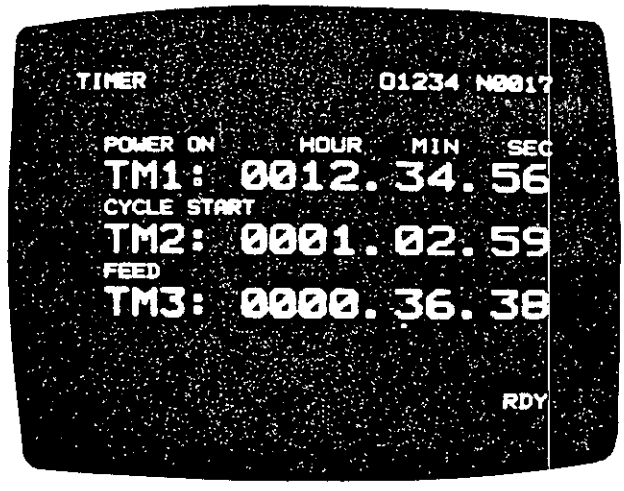


Fig. 4.29

4.3.10 ADDRESS SEARCH

Search continues until a data (character string) held on tape or in the memory which coincides with the data (character string) entered through the NC operator's panel is found. The contents of tape will be searched in TAPE mode and those of the part program memory in MEM or EDIT mode.

1. Operation

   A. Select TAPE, MEM, or EDIT mode.

   B. Depress the PROG function key.

   C. Depress the RESET key. "LSK" appears and the pointer returns to the top of the program number in MEM mode.

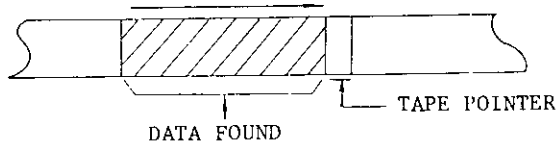
   D. Enter the data (string of not more than 10 characters headed by address) to be searched.

   E. Depress the [CURSOR ↓] key. Search starts. "AS" blinks during search.

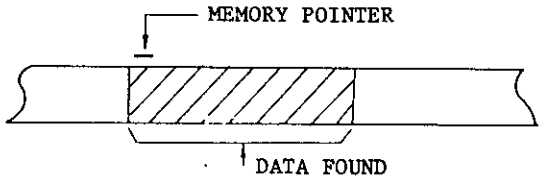
2. End of search

   A. "AS" disappears when search is completed.

   (1) In TAPE mode, the tape pointer points to the character that immediately follows the data found and the tape stops.



(2) In MEM or EDIT mode, the pointer of the part program memory points to the top of the data found (pointed by the cursor). In all cases, only search will be performed but neither BUF display nor advance reading will be performed.



B. "AS" disappears and "NOT FOUND!" appears on the CRT if the desired data is not found. This message will disappear when you depress a key (CAN normally) of the control station.

3. Remarks

- Do not omit leading zeros of the search data. The data itself which has been entered through the keyboard will be compared with those on the tape or in the part program memory.

  When searching a program number cataloged, leading zeros may be omitted.

- Commands encountered during search will be ignored even if they are modal commands.

- On Cycle Start after search, the data of a block which the pointer points to will be read in and executed.

4. Search of program number

The address search function also permits to search a part program out of those stored in the memory.

A. Select MEM or EDIT mode.

B. Depress the PROG function key.

C. Depress the RESET key.

D. Enter the program number "O□□□□."

E. Depress the [CURSOR ↓] key.

The designated program number will be searched. The result of search is as described in 2. In MEM mode, you may depress the CYCLE START button immediately after completion of search to start automatic operation from the beginning of the program.

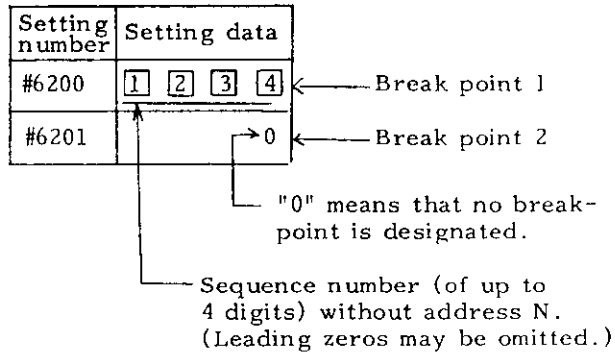
4.3.11 BREAKPOINT FUNCTION

It is possible to suspend operation at the end of a block by designating a sequence number in set function. Location is 6200 and 6201.

1. If the current sequence number is found to be equal to a sequence number designated as setting data during automatic operation, operation will stop after execution of the block like in single block operation.

2. The designated sequence number is called a breakpoint and up to two breakpoints may be designated.

3. Setting numbers are as follows for designating breakpoints.



4. "BREAKPOINT!" appears blinking when operation has stopped at a breakpoint. To restart, depress the CYCLE START button.

NOTE: If the breakpoint function is not used, set the contents of #6200 and #6201 to "0."

4.3.12 ALARM CODE DISPLAY

If an alarm status has happened, ALM" or "A/B" (on battery alarm) blinks on the bottom line of the screen regardless of working mode and function. If this happens, the detailed information of the alarm status may be displayed by the following operation.

1. Depress the ALM key.

   Then up to four pairs of alarm code and message will be displayed, with more serious one on a higher line.

NOTE: The alarm screen will appear during an alarm state and, therefore, it is not needed to operate the [PAGE ↓] key.

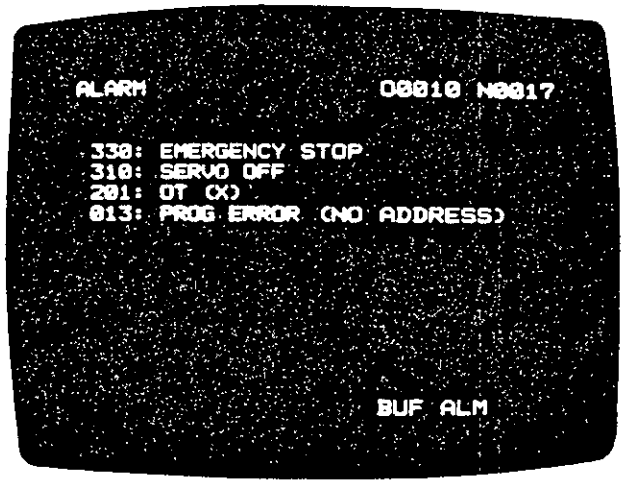


Fig. 4.30 Alarm Codes and Message Displayed-Example

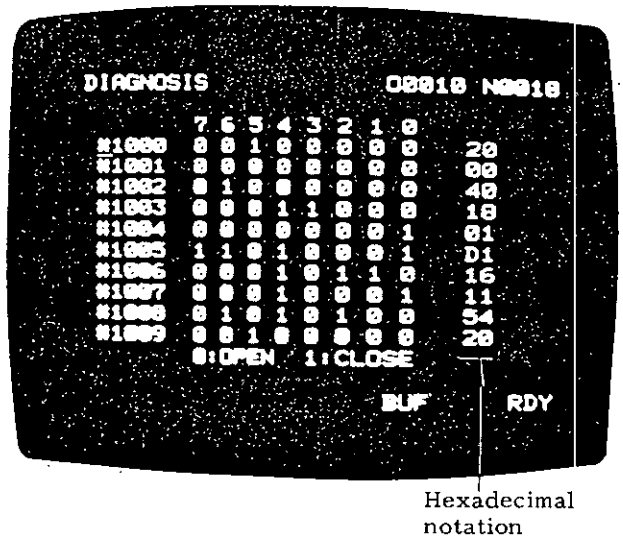


Fig. 4.31 State of I/O Signals Displayed-Exampe

To reset the alarm status and screen, remove the cause of alarm then depress the RESET key.

For the detail of alarm codes, see Appendix 5, LIST OF ALARM CODES.

4.3.13 DISPLAYING ON/OFF INPUT/OUTPUT SIGNALS

Depress the DGN function key, and the state of every input/output signal will be displayed on the CRT. This is possible at any time even during automatic operation.

For more detail of this operation, see 8.6.3 DIAGNOSTICS OF INPUT/OUTPUT SIGNALS.

The state of the input/output signal is also given in the hexadecimal notation at the rightmost column for the ease of maintenance work.

4.4 TAPE INPUT/OUTPUT OPERATIONS OF NC DATA

Such NC data as tool offsets, setting data, and parameter data may be read from and written onto tape. A tape reader will work to read data from tape. To write data onto tape, a data input/output interface (option) is needed.

Here we assume that this option is incorporated.

See 4.7.2 I/O DEVICES AND BAUD RATE SETTING for how to set the type of input/output device (setting #6003) and baud rate (parameter #6026).

4.4.1 INPUTTING TOOL OFFSETS FROM TAPE

Though tool offsets are inputted by MDI operation normally, they may also be entered by means of paper tape.

1. The tape format of tool offsets is as follows.

```
Label  % ;
H01  X   1.000  ;
H02  X   2.000  ;
D03  X -10.089  ;
 .
 .
 .
%
```

Note:
Either H or D may be used for address.

2. The input operation is as follows.

   A. Select EDIT mode.

   B. Depress the RESET key.

   C. Depress the OFS key.

   D. Set the tool offset data tape onto the tape reader.

   E. Depress the IN key.

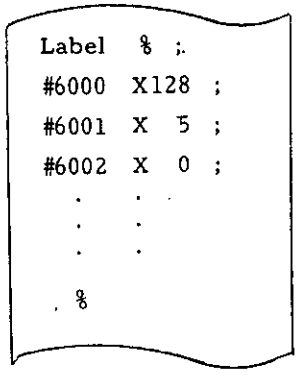
      The tape reader starts to read the tape. "IN" blinks on the CRT while the data are read.

   F. The tape reader stops when it has read "%" (or ER). "IN" disappears from the CRT. Now the tool offset data have been read into memory.

4.4.2 INPUTTING SETTING DATA AND PARAMETER DATA

Though setting data and parameter data are inputted by MDI operation normally, they may also be entered by means of paper tape. Setting data and parameter data may be inputted from a single tape.

1. The tape format is as follows.



Notes:
1. "%" is used in the ISO code and "ER" in the EIA code.
2. "N" is used in the EIA code instead of "#."

2. The input operation is as follows.

A. Select EDIT mode.

B. Depress the RESET key.

C. Deprss the PRM key.

D. Set the setting/parameter data tape onto the tape reader.

E. Depress IN key.

The tape reader starts to read the tape. "IN" blinks on the CRT while the data are read.

F. The tape reader stops when it has read "%" (or "ER"). "IN" disappears from the CRT. Now the setting/parameter data have been read into memory.

Turn on power again because the control is in the HOLT state (key inoperative) at completion of input.

4.4.3 OUTPUTTING TOOL OFFSETS TO PAPER TAPE

The tool offset data set in the system may be outputted to paper tape.

1. The output operation is as follows.

A. Select EDIT mode.

B. Depress the RESET key.

C. Depress the OFS key.

D. Check that the punch is ready for operation.

E. Depress the OUT key.

The paper tape punch punches the tool offset data onto paper tape and stops automatically when all contents of the tool offset memory have been outputted.

F. To suspend the operation, depress the RESET key.

At this time, the output operation cannot be resumed. Restart from the beginning.

2. The tape format is the same as that described in 4.4.1 INPUTTING TOOL OFFSETS FROM TAPE.

4.4.4 OUTPUTTING SETTING DATA AND PARAMETER DATA TO PAPER TAPE

1. The output operation is as follows.

A. Select EDIT mode.

B. Depress the RESET key.

C. Depress the PRM key.

D. Check that the punch is ready for operation.

E. Depress the OUT key.

The paper tape punch punches the setting/parameter data onto paper tape continuously.

F. To suspend the operation, depress the RESET key.

At this time, the output operation cannot be resumed. Restart from the beginning.

2. The tape format is the same as that described in 4.4.2 INPUTTING SETTING DATA AND PARAMETER DATA.

4.4.5 OUTPUTTING PART PROGRAM TO PAPER TAPE

(1) The part program of the designated program number is punched out by the following operations:

a. Connect the external equipment such as the tape puncher to the NC via the data input/output interface.

b. Make the external equipment relay for operation.

c. Power on the NC.

4.4.5 OUTPUTTING A PART PROGRAM TO PAPER TAPE (CONT'D)

d. Select the EDIT mode.

e. Depress the PROG function key.

f. Check to see if the external equipment is ready.

g. Depress the RESET key.

h. Depress "O" key and key in program number.

i. Depress the OUT key.

The part program of the keyed-in program number is outputted to the external equipment. If it is a tape puncher, tape punch is performed. When the output of the part program is completed, the tape puncher stops automatically. During the output of data, "OUT" is flashing.

j. To discontinue the punch out operation, depress the RESET key. However, the discontinued operation cannot be resumed. Go back to f. and repeat the operations all over again.

NOTE: When RESET, OUT are operated without keying in the program number, the part program of the currently displayed program number is outputted.

(2) Punch out of all part programs

All registered part programs may be outputted to the external equipment by the above operations except that 0 - 9 9 9 9 must be keyed in the operation of h.
All part programs stored in the memory are outputted (punched out, etc.) consecutively.

NOTE: The contents of program number O0000 are outputted only when #6231D3 = 1.

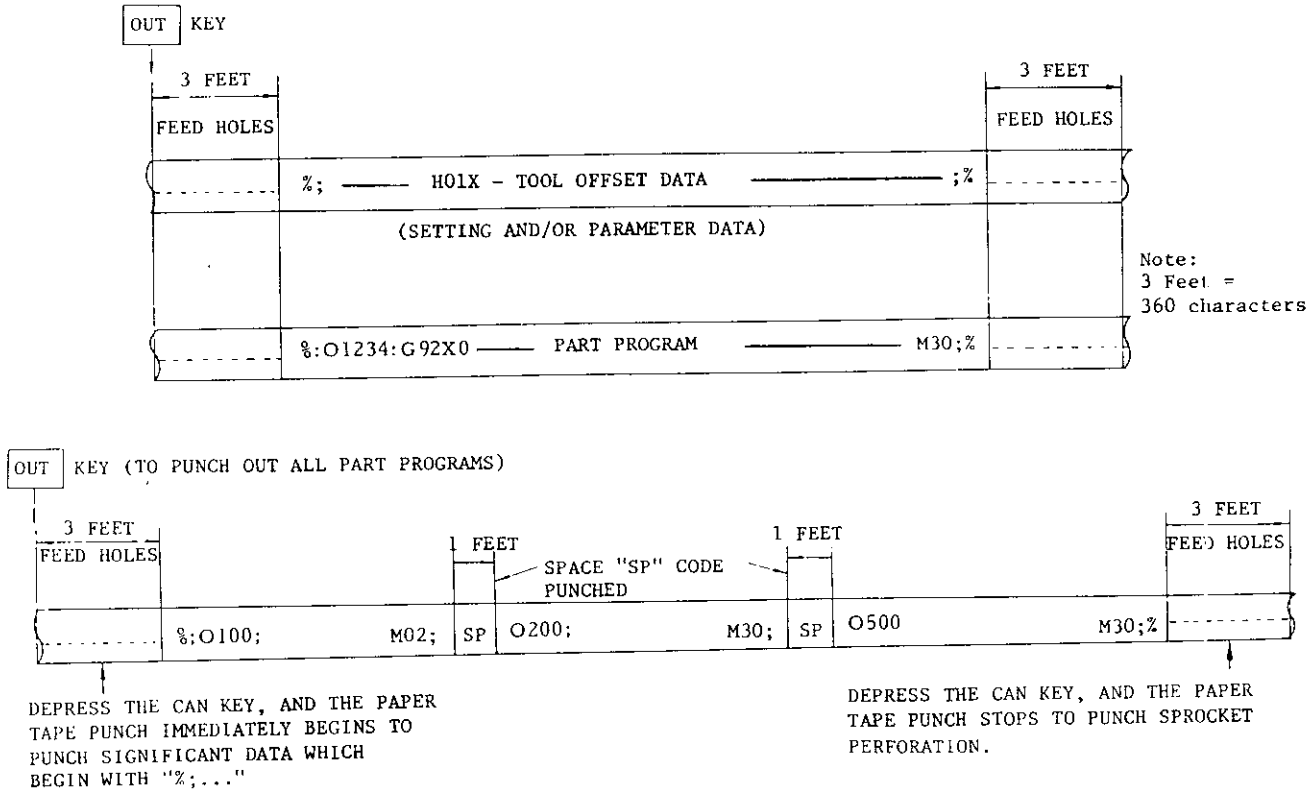


Fig. 4.32 Data and Program Formats on Paper Tape

4.5 LOADING PART PROGRAMS INTO MEMORY

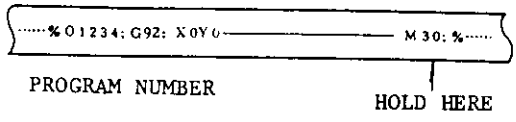
4.5.1 LOADING PART PROGRAM TAPE INTO MEMORY

(1) Loading a part program which has a program number.

a. Select EDIT mode.

b. Depress the PROG key.

c. Load the NC tape to the tape reader or an equivalent external device.



d. Depress the RESET key.

e. Depress the IN key.

Then the system starts to read the tape and enlists the program number punched on the tape as the first record. The system checks for duplicator of program number as in 1. Operation ends with error if the designated program number is not found on the tape.

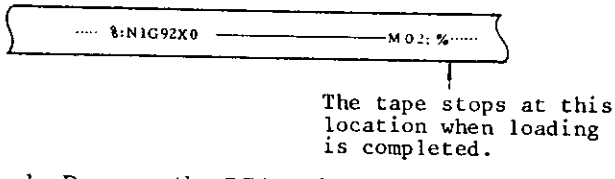
When the tape reader has read "M02 ;," "M03 ;," or "M99 ;," it stops and "IN" disappears from the CRT. Now the part program has been stored in memory.

(2) Loading a part program which has no program number.

a. Select EDIT mode.

b. Depress the PROG keys.

c. Load the NC tape to the tape reader or an equivalent external device.



d. Depress the RESET key.

e. Depress the address O key then enter the program number.

f. Depress the IN key.
The system starts to read the tape. If the keyed-in program number coincides with the registered program number, "ALREADY IN" blinks on the CRT screen. If this happens, delete the program number, then repeat steps a. through f. while the tape is being read, "IN" blinks on the CRT.

g. When the tape reader has read "M02 ;," "M03 ;," or "M99 ;," it stops and "IN" disappears from the CRT. Now the part program has been stored in memory.

NOTES:

1. Program number "O0000" is always in the registered state, so it cannot be erased. This program number should not be used in general.

2. The tape which has no program number may be stored as described before. However, write a program number to the head of the tape, in principle. The operation of "Oxxxx IN" described before causes only program number registration. It does not cause the storing of information of "O0000" into the part program memory. Only the program number on tape is stored into the memory. Assume that a tape having no program number is stored and then all part programs are punched out by depressing "0," "-," "9," "9," "9," "9," and "OUT" keys. Since this tape contains programs with no program number, the correct restoring of all part programs may not be performed by depressing "0," "-," "9," "9," "9," "9," and "IN" keys.

3. Consequently, when a tape having no program number has been stored, write the program number to the head of part program by the EDIT operation.

EXAMPLE:

N1 G92 X0 Z0 ;

When this is in the first block, position the cursor to N and key in as follows (in EDIT and PROG modes):

Oxxxx ; N1 ALTER

(3) Storing a program with program numbers changed I

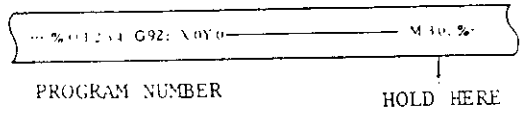
To register a program with a program number different from the one punched on tape, perform the following operations:

a. Select the EDIT mode.

b. Depress the PROG key.

c. Set the NC tape to the tape reader or the external equivalent equipment.

4.5.1 LOADING PART PROGRAM TAPE INTO MEMORY (CONT'D)



d. Depress the RESET key.

e. Key in "O" and PROGRAM NUMBER.

f. Depress the IN key.

The program number entered from the key is registered in preference to the program number punched on the tape. At this time, the program number on the tape is written to the part program memory simply as a label.
M02 ;, M30 ; or M99 ; is read and the storing operation is completed.

NOTE:

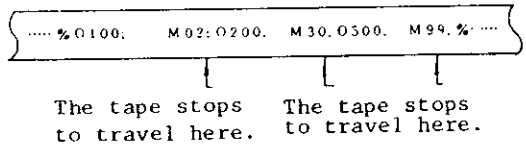
1. If a program is stored with a changed program number as described above, the program number punched on the tape is stored in the part program memory without change. Consequently, to avoid the confusion in the later handling, replace the program number in the part program memory with changed program number by the EDIT operation.

(4) Loading part programs from a tape

a. Select EDIT mode.

b. Depress the PROG key.

c. Load the NC tape to the tape reader or an equivalent external device.



d. Depress the RESET key.

e. Depress the IN key.

Then the system starts to read the tape and enlists the program number punched on the tape as the first record. The system checks for duplication of program number as described in (2).

The tape reader stops each time it has read "M02 ;," "M30 ;," or "M99 ;."

f. Depress the IN key again.

The tape reader resumes to read the tape. Repeat this operation until all programs are loaded.

(5) Storing a program with program numbers changed II

When "O" key is depressed and program number is keyed in before depressing IN key as described in (4) above, the keyed in program number is registered in preference to the program number punched on the NC tape.

(6) Loading programs continuously

Programs existing on a tape as shown in (4) may be loaded continuously without interruption. For this purpose, depress "O," "-," "9," "9," "9," and "9" before the first depression of the IN key. The tape reader stops at the position of "%."

4.5.2 MAKING ADDITION TO A PART PROGRAM

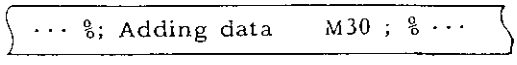
Perform the following operation to add data to a part program which is already loaded.

a. Select EDIT mode.

b. Depress the PROG key

c. Depress the O key then enter the part program number and depress the CURSOR ↓ key.
The system searches the designated program.

d. Load the tape of adding data to the tape reader.



e. Depress the RESET key.

f. Depress the NEXT and IN keys in this order.

The data will be read from the tape into the memory.

NOTE: You cannot add data to a program from the middle of it. If necessary, delete the last part of the program by editing operation and perform this adding.

4.5.3 LOADING PART PROGRAMS BY MDI

Part programs may be loaded not through the tape reader but by MDI operation. Perform the following.

a. Select EDIT mode.

b. Depress the PROG key

c. Depress the RESET key.

d. Depress the O key then enter the part program number and depress the WR key.

   The designated program number will be registered. If this number already exists, "ALREADY IN" blinks and, in this case, it is required to delete the registered program number.

e. Write the part program by operating the address key and the data key. As shown in the figure below, the keyed in data is displayed on the bottom line from left to right sequentially. The maximum number of characters that can be written at a time is 32. Within this limit, data may be keyed in over multiple words or blocks. However, when the 10th character is keyed in, the normal display shown to the right of the line is blanked.

f. Depress the INSRT key.

   The keyed in data is stored in the part program memory.

g. Repeat the operations of e. and f. above to write the part program. The program edit operation is enabled by the use of ERASE, INSRT, and ALTER keys during this program storing operation.

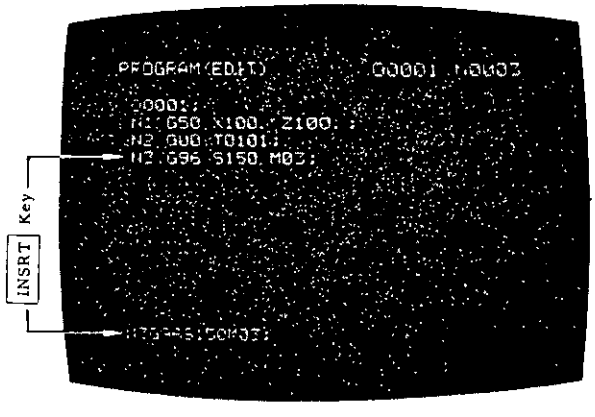


Fig. 4.33

h. Key in M02 ;, M30 ;, or M99 ; and depress INSRT key. This completes the storing of the part program.

4.5.4 DISPLAY OF REGISTERED PROGRAM NUMBER

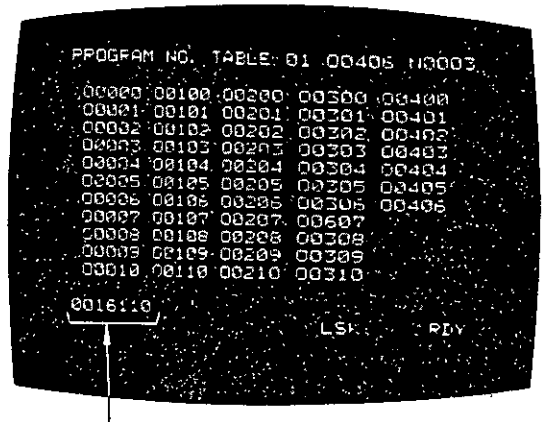
This screen displays all registered program numbers and the number of remaining characters in the part program memory.

(1) The number of program numbers that can be registered depends on options.

Table 4.3.9.1

No.	Max. Number of Programs	Type	Program No. Table
1	99	Basic	01 to 02
2	199	Option 1	01 to 04
3	999	Option 2	01 to 19

(2) All program numbers already registered are displayed. By depressing [PAGE ↓] or [↑ PAGE] key, the page shown below may be obtained.



REMAINING NUMBER OF CHARACTERS IN PART PROGRAM MEMORY

The remaining number of characters in part program memory is displayed in the lower left corner of the screen.

Fig. 4.34

Note: This screen displays only the registered program numbers. A program number is registered by depressing the PROG function key in EDIT mode.

4.6.1 PART PROGRAM DISPLAY

Stored program contents can be displayed, and checked by the operator.

Part Program Call

1. Select the EDIT mode.
2. Depress the RESET and PROG keys.
3. Input the program number with ADDRESS O and depress the [CURSOR ↓] key.

   The specified program number will be searched and the data of 10 lines from the begining of program will be displayed on the CRT. If the program number is not found by searching, "NOT FOUND" will flicker. The display will be reset by depressing the CAN key.

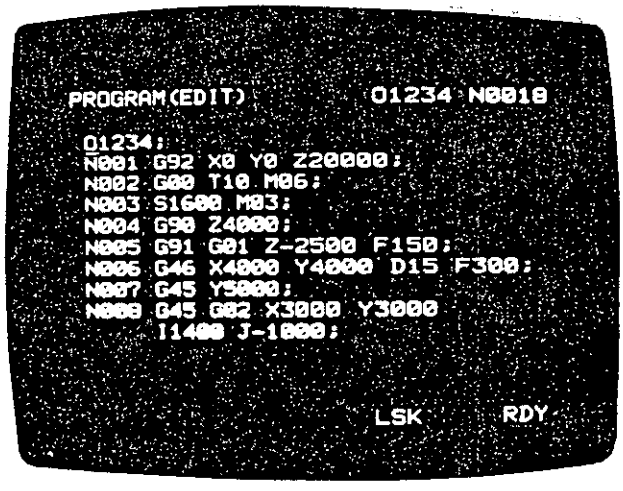


Fig. 4.35

Operation of PAGE and CURSOR key

1. Page keys [↑] and [↓] respectively advances and returns by one page.
2. CURSOR keys [↑] and [↓] respectively moves the CURSOR after and before a word.

Operation in the MEM mode

Searching can be performed in the MEM mode. However, page and cursor cannot be moved by PAGE and CURSOR keys.

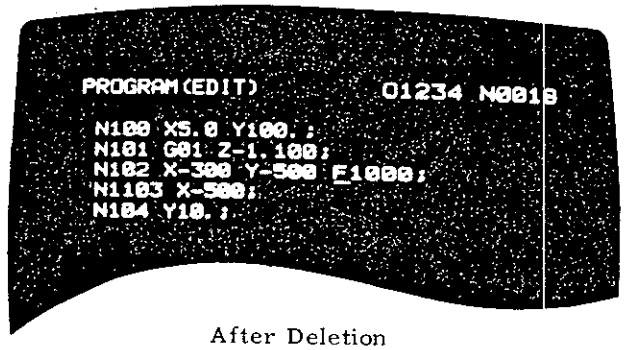
[illegible] PROGRAM BLOCK

Part programs can be deleted using PROG keys in the EDIT mode.

Deletion of Words

Move the CURSOR to the word to be deleted and depress the ERASE key. The CURSOR-indicated word will be deleted.

Before Deletion



After Deletion

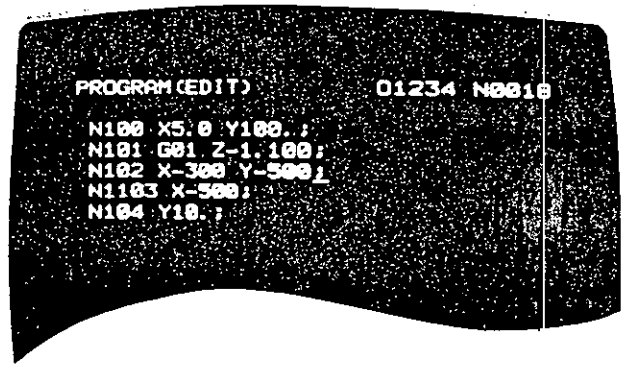


Fig. 4.36

Deletion of Program No.

Enter the program No. with address O and depress the ERASE key. The specified program No. and its part program will be deleted.

Deletion of All the program numbers

Input O -, 9, 9, 9, 9 and depress the ERASE key. All the registered program numbers will be deleted. Program No. "0" is registered newly in the form of EOB.

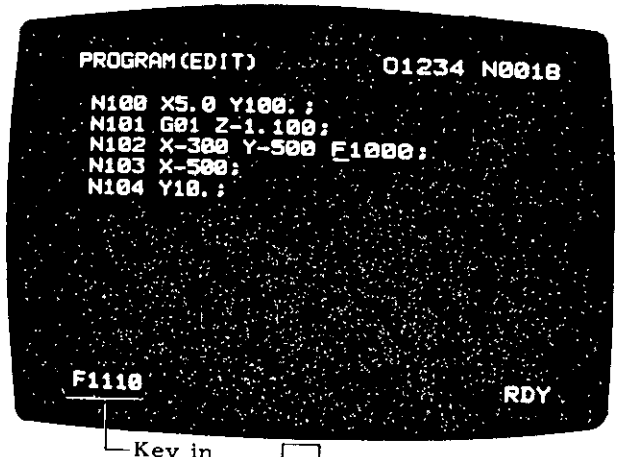
4.6.3 MODIFYING PART PROGRAM BLOCK

Program modification is made using PROG key in the EDIT mode.

Word Modification

Specify the word to be altered with the CURSOR key in the new word, and depress the ALTER key. The new word will replace the CURSOR-indicated word.

Before Modification



After Modification

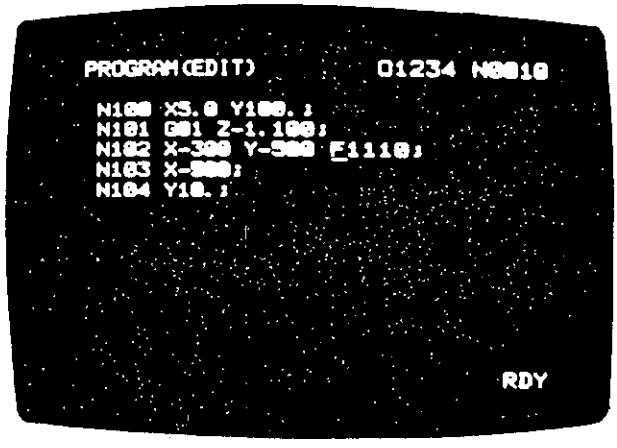


Fig. 4.37

Words less than 32 characters can replace one word specified.

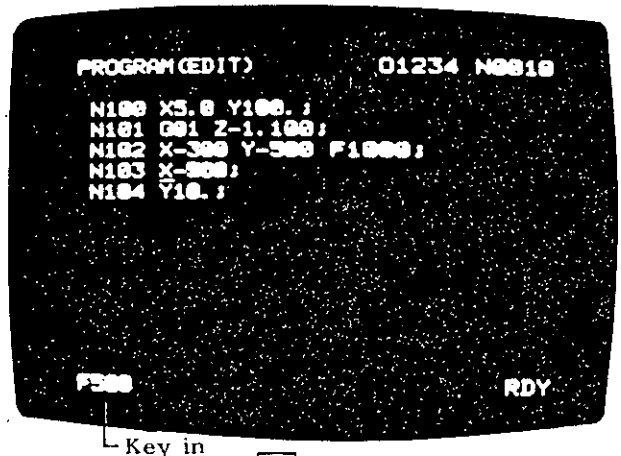
4.6.4 ADDING PART PROGRAM (INSRT KEY)

Programs will be inserted using PROG key in the EDIT mode.

Insertion of words

Specify the word before the word to be added using CURSOR, key in the data to be added, and depress the INSRT key. The new data will be inserted immediately after the word specified by the CURSOR.

Before Insertion



After Insertion

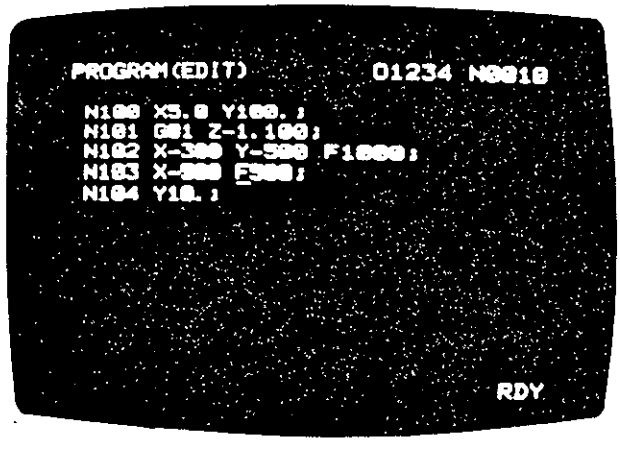


Fig. 4.38

After insertion, CURSOR indicates the last word keyed in words less than 32 characters can be inserted as one group.

4.7 SUPPLEMENT TO DATA INPUT / OUTPUT INTERFACE

The external equipment having the designated input/output interface may be attached to the NC to input/output the following NC information:

(1) Part Programs.
(2) Tool Offset Amount.
(3) Setting Data and Parameters.

4.7.1 TYPES AND FUNCTIONS OF INTERFACE

Table 4.1

	1	2	3	4
Name of interface	FACIT4070 Interface	Current Loop Interface(20mA)	RS232C Interface	RS422 Interface
Type of interface	Parallel·Voltage Interface	Serial·Current Interface	Serial·Voltage Interface	Serial·Balanced Interface
Data transmission speed	70 char/s	Parameter setting Input: #6026 Output: #6028		Input: #6027 Output: #6029
Connector (Note)	MR-20MR		DB-25S	DB-37S
Max cable length	5 m	50 m	15 m	100 m
External equipment	FACIT4070 or equipment having equivalent interface	ASR-33 or equipment having current loop (20 mA) interface	Equipment having RS232C interface	Equipment having RS422 interface
Functions	· Output the NC data to the external equipment (for punching out NC tape)		· Storing NC data into NC memory or collating them. · Outputting NC data to external equipment (punch out) · Automatic operation in the Tape mode	

Note: The types of the connector on the NC side. For the mating connectors to this connector, use the following: MR-20F, DB-25P, DB-37P

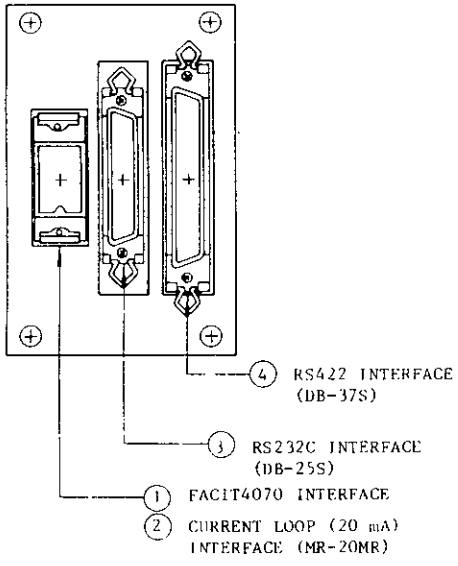


Fig. 4.39 Data I/O Interface Receptacles in Control Cabinet

4.7.2 SETTING OF DATA INPUT/OUTPUT INTERFACE TO BE USED

To use data input/output interface, it is necessary to set which interface is to be used. Make this setting as follows:

(1) Setting of Data Input Interface to Be Used

IDVCE1 (#6003, D1)	IDVCE0 (#6003, D0)	Data Input Interface to be used
0	0	PTR Interface (Note)
0	1	RS232C Interface
1	0	RS422 Interface

Note: PTR interface is for the standard tape reader. Usually, this interface is set.

(2) Setting of Data Output Interface to Be Used

ODVCE1 (#6003, D5)	ODVCE0 (#6003, D4)	Name of Interface
0	0	FACIT4070 Interface
0	1	Current Loop Interface RS232C Interface
1	0	RS422 Interface

4.7.3 SETTING OF BAUD RATE AND OTHERS OF SERIAL INTERFACE

To use serial interface (current loop, RS232C, or RS422), it is necessary to set the baud rate, stop bit length, and control code transmission specification to parameters.

(1) Current Loop or RS232C Interface

As shown below, the data is set for input and output combined or separately.

#6028 D6
- 0 ··· Data is set for input and output combined.
- 1 ··· Data is set for input and output separately.

a. Setting of Baud rate

Input and Output in Common		#6026D3	#6026D2	#6026D1	#6026D0
	Input	#6026D3	#6026D2	#6026D1	#6026D0
	Output	#6028D3	#6028D2	#6028D1	#6028D0
Baud Rate	50	0	0	0	0
	100	0	0	0	1
	110	0	0	1	0
	150	0	0	1	1
	200	0	1	0	0
	300	0	1	0	1
	600	0	1	1	0
	1200	0	1	1	1
	2400	1	0	0	0
	4800	1	0	0	1
	9600	1	0	1	0

b. Setting of stop bit length

Input and Output in Common	#6026D4	= 1: Stop bit as 2 bits = 0: Stop bit as 1 bit
Input	#6026D4	
Output	#6028D4	

c. Setting of control code transmission designation

Input and Output in Common	#6026D5	= 1: Does not send out control code = 0: Sends out control code
Input	#6026D5	
Output	#6028D5	

(2) RS422 Interface

As shown below, the data is set for input and output combined or separately.

#6029 D6
- 0 ··· Data is set for input and output combined.
- 1 ··· Data is set for input and output separately.

a. Setting Baud Rate

Input and Output in Common		#6027D3	#6027D2	#6027D1	#6027D0
Input		#6027D3	#6027D2	#6027D1	#6027D0
Output		#6029D3	#6029D2	#6029D1	#6029D0
Baud Rate	50	0	0	0	0
	100	0	0	0	1
	110	0	0	1	0
	150	0	0	1	1
	200	0	1	0	0
	300	0	1	0	1
	600	0	1	1	0
	1200	0	1	1	1
	2400	1	0	0	0
	4800	1	0	0	1
	9600	1	0	1	0

4.7.3 SETTING OF BAUD RATE AND OTHERS OF SERIAL INTERFACE (CONT'D)

b. Setting Stop Bit

Input and Output in Common	#6027D4	= 1: Stop bit as 2 bits
Input	#6027D4	= 0: Stop bit as 1 bit
Output	#6029D4	

c. Setting Control Code Sending

Input and Output in Common	#6027D5	= 1: Does not send control code
Input	#6027D5	= 0: Sends control code
Output	#6029D5	

Notes:

1. Some controls do not allow the switching between the setting types by #6028D6 but are fixed to the type in which data is set for input and output combined.

2. Set the baud rate and stop bit length according to the specifications of the input/output equipment to be used.

3. The start and stop signals to be send from the NC to the input/output equipment after pressing IN, VER, or OUT key are called "control codes." If the specifications of the input/output equipment do not allow the acceptance of the control codes, set the parameter for control code transmission designation to "1" (not send). In this case, it is necessary to press IN, VER, or OUT key on the NC side then start/stop the input/output equipment manually.

4.7.4 CABLE CONNECTOR SPECIFICATIONS

The specifications of the cable connectors for data input/output interface are as shown in Tables 4.7.10 through 4.7.14. These specifications depend on the external equipment to be used and are therefore listed in this publication for reference purpose only. Refer to the manual of the external equipment.

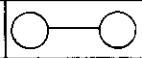
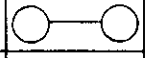
Table 4.2 FACIT4070 Interface Connecting Cable

NC (MR-20F)			Connections	External Equipment (DB-25P)	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
PR	PUNCH READY	1		12	PR
TL	TAPE LOW	2		21	TL
ERR1	ERROR	3		20	ERR1
	Not Used	4			
+6 V	FACIT/ ASR. Auto-selection	5		24	+6 V
	Not Used	6			
	Not Used	7			
0 V	GROUND	8			
0 V	GROUND	9		10	SD
0 V	GROUND	10		25	0 V
CH1	PUNCH DATA 1	11		1	CH1
CH2	PUNCH DATA 2	12		2	CH2
CH3	PUNCH DATA 3	13		3	CH3
CH4	PUNCH DATA 4	14		4	CH4
CH5	PUNCH DATA 5	15		5	CH5
CH6	PUNCH DATA 6	16		6	CH6
CH7	PUNCH DATA 7	17		7	CH7
CH8	PUNCH DATA 8	18		8	CH8
CH9	FEED HOLD	19		9	CH9
PI	PUNCH INSTRUCTION	20		11	PI

↑Note

Note: The pin numbers at the time the external equipment is FACIT 4070 and its plug-in connector is DB-25P.

Table 4.3 Current Loop (20 mA) Interface Connection Cable

NC (MR-20F)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
		1			
	Not Used	⟩			
		4			
+6 V	FACIT/ ASR. Auto-selection	5			
TTY2	Current loop (-)	6			
TTY1	Current loop (+)	7			
0 V	GROUND	8			
		9			
	Not Used	⟩			
		20			

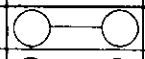
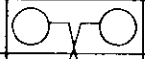
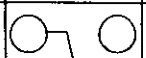
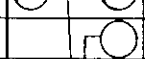
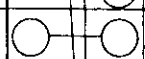
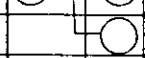
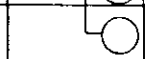
(Note 2)

Notes:

1. The type of connector and pin number are different with external equipment.

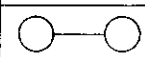
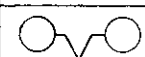
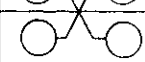
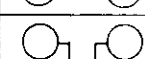
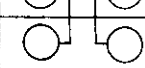
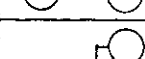
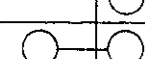
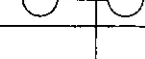
2. When the current loop interface is used, short-circuit pin No. 4 (signal RS) and pin No. 5 (signal CS) of plug connector DB-25P for RS232C. Then connect the plug to the NC receptacle DB-25S.

Table 4.4 RS232C Interface Connecting Cable (A)

NC (DB-25P)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
FG	Frame grounding	1			FG
SD	Sending data	2			SD
RD	Receiving data	3			RD
RS	Request sending	4			RS
CS	Capable of sending	5			CS
	Not used	6			DR
SG	Signal grounding	7			SG
		8			IO BUSY
	Not used	⟩ 25			ER (OR IO ALARM)

Note: When the external equipment does not control the CS (Capable of Sending) signal given to NC, short-circuit pins RS and CS on both ends of the cable as shown in Table 4.7.13.

Table 4.5 RS232C Interface Connecting Cable (B)

NC (DB-25P)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
FG	Frame grounding	1			FG
SD	Sending data	2			SD
RD	Receiving data	3			RD
RS	Request sending	4			RS
CS	Capable of sending	5			CS
	Not used	6			DR
SG	Signal grounding	7			SG
		8			
		⟩			ER (OR IO ALARM)
	Not used	25			

4.7.4 CABLE CONNECTOR SPECIFICATIONS (CONT'D)

Table 4.6 RS422 Interface Connection Cable

NC (DB-37P)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
SHIELD	Shield	1			
	Not used	2			
	Not used	3			
SD	Sending data	4			SD
	Not used	5			
RD	Receiving data	6			RD
RS	Request sending	7			RS
	Not used	8			
CS	Cable of sending	9			CS
	Not used	10			
	Not used	11			
ER	NC ready	12			ER
DR	I/O device ready	13			DR
		14			
	Not used				
		18			
SG	Signal grounding	19			
	Not used	20			
	Not used	21			
*SD	Sending data	22			*SD
	Not used	23			
*RD	Receiving data	24			*RD
*RS	Request sending	25			*RS
	Not used	26			
*CS	Capable of sending	27			*CS
	Not used	28			
	Not used	29			
*ER	NC ready	30			*ER
*DR	I/O device ready	31			*DR
	Not used	32			
		37			

Table 4.7 RS232C Interface Connection Cable (B)

NC (DB-25P			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
FG		1			FG
SD	Sending data	2			SD
RD	Receiving data	3			RD
RS	Request sending	4			RS
CS	Capable of sending	5			CS
	Not used	6			DR
SG	Signal grounding	7			SG
		8			
	Not used				ER (OR IO ALARM)
		25			

4.7.5 OPERATIONS USING DATA INPUT/OUTPUT INTERFACE

The use of data input/output interface allows the following operations and runs:

(1) The input/output operations of tool offset amounts, setting data, and parameter data and the output operations (punch out) of part programs. For details, see 4.4 TAPE INPUT/OUTPUT OPERATIONS OF NC DATA.

(2) The storing of part programs into memory. For details, see 4.5 LOADING PART PROGRAMS INTO MEMORY

(3) Tape-verification of part programs, tool offset amount, setting data, and parameter data. For details, see 4.8 TAPE VERIFYING.

(4) Automatic run in tape mode.

The control may be automatically run in tape mode not via the tape reader on the machine but via the data input/output interface. For details, see 7.6 OPERATION IN TAPE AND MEMORY MODE.

4.8 TAPE VERIFYING

The punched tape of the data shown below may be compared to the contents of the NC internal memory to check if they match.

- Part program
- Tool offset data
- Setting data and parameter

The punched tape is entered through the NC tape reader or the data input/output interface (option). For the methods of setting the input/output equipment (setting #6003) and the baud rate (parameters #6026 through #6029), refer to 4.9 DATA INPUT/OUTPUT INTERFACE. The following description is made assuming that this option is installed.

4.8.1 SETTING AND PARAMETER TAPE VERIFYING

1. Select "1" of system No. switch.
2. Set the MODE SELECT switch to the EDIT.
3. Depress PRM function key.
4. Depress RESET key.
5. Load the NC tape via tape reader.
6. Depress VER key.

   Tape starts, and the contents of tape and the contentns of setting/parameter are verified. "VER" is flickerlingly displayed.
   If disagreement with the tape data is detected, "INPUT ERROR" is flickeringly displayed.
7. After completion of verifying without disagreement, tape reader stops and "VER" disappears.

4.8.2 TOOL OFFSET VALUE TAPE VERIFYING

1. Set the MODE SELECT switch to the EDIT.
2. Depress OFS function key.
3. Depress RESET key.
4. Load the source tape via tape reader.
5. Depress VER key

   Tape starts, and the contents of tape and the contents of tool offset values are verified. If disagreement with the tape data is detected, "INPUT ERROR" is flickeringly displayed.
6. After completion of verifying without disagreement, tape reader stops and "VER" disappears.

4.8.3 VERIFYING PART PROGRAM TAPE

(1) Verifying a part program tape having program number

a. Select the EDIT mode.

b. Depress the PROG function key.

c. Set the part program tape to the tape reader.

d. Depress the RESET key.

e. Depress the VER key.

   The tape is started to compare the contents of the part program memory to the contents of the part program tape. During this operation, "VER" keeps flashing. If a mismatch is found, "INPUT ERROR" is displayed flashing.
   When a match is found and this operation is completed, the tape reader stops, upon which "VER" display is erased.

   Note: By the operation of "RESET, VER," the verifying feature verifies the data from the tape head to % code.

(2) Verifying a part program tape having no program number

a. Select the EDIT mode.

b. Depress the PROG function key.

c. Set the part program to the tape reader.

d. Depress the RESET key.

e. Depress the "O" key and key in program number.

f. Depress VER key.

   The tape starts to compare the contents of the part program memory to the contents of the part program tape. During this operation, "VER" keeps flashing. If a mismatch is found, "INPUT ERROR" is displayed flashing. When a match is found and this operation is completed, the tape reader stops, upon which "VER" display is erased. If the keyed in program number is not found in the memory, "NOT FOUND!" is displayed flashing. In this case, depress the CAN key and start with the operation of d.

NOTES:

1. The operations for verification with a program number different from the program number punched on the tape are the same as those of (2) above. The keyed-in program number is processed in preference to the punched program number.
2. Verification by the operation of "Oxxxx VER" regards the punched information as the information on the keyed-in program number. Hence, when verifying a tape containing program numbers, no program number should be keyed in.

(3) Verifying a tape containing multiple part programs

Multiple part programs punched in a single tape are continuously verified by the following operations:

a. Select the EDIT mode.

b. Depress the PROG function key.

c. Set the part program tape to the tape reader.

d. Depress the RESET key.

e. Depress the VER key.

When M02, M30, or M99 is read, the tape reader does not stop but all the part programs are continuously verified up to % code. When the verification is completed, the tape reader stops at the position of % code.

4.8.4 SUMMARY OF EDITING OPERATION

	Operation			Edit Lock	System No. Switch	Mode	Function	Procedure
Parameter	Storing from NC operator's panel keyboard				1	/	PRM	Parameter number → CURSOR Data → WR
	Storing from tape (Note 4) (Note 6)				1	EDIT		RESET → IN
	Punch out (Note 3)							RESET → OUT
	Matching with tape (Note 4)							RESET → VER
Setting	Storing from NC operator's panel keyboard					/	SET	Setting number → CURSOR Data → WR
	Storing from tape				1	EDIT		RESET → IN
	Punch out							RESET → OUT
	Matching with tape							RESET → VER
Offset	Storing from NC operator's panel keyboard					/	OFS	Offset number → CURSOR Data → WR
	Storing from tape					EDIT		RESET → IN
	Punch out							RESET → OUT
	Matching with tape							RESET → VER
	Clear of all offsets					/		O → -9999 → ORG
Part program	Storing from NC operator's panel keyboard			OFF		EDIT	PROG	O → Program number → WR Repeat of edit opeation "addition of address data"
	Storing from tape	One part program	Tape with number O	OFF				RESET → IN
			Tape without number O	OFF				RESET → O → Program number → IN
		All part programs on tape		OFF				RESET → O → -9999 → IN
		Addition to registered part program		OFF				RESET → NEXT → IN
	Punch out	Designated part program						RESET → O → Program number → OUT
		All part programs						RESET → O → -9999 → OUT
	Matching with tape	One part program	Tape with number O					RESET → VER
			Tape without number O (Note 1)					RESET → O → Program number → VER
		All part programs on tape						RESET → VER
	Edit	Modify of address data (Note 2)		OFF				CURSOR (Set to address data to be modified) → Address data → ALTER
		Add of address data (Note 2)		OFF				CURSOR (Set to address data just before addition) → Address data → INSRT
		Delete of one address data		OFF				CURSOR (Set to address data to be deleted) → ERASE
		Delete of one block (Note 5)		OFF				CURSOR (Set to address data at head of block to be deleted) → EOB → ERASE
	Address search					TAPE MEM EDIT	-	Address data to be searched → CURSOR
	Clear	Designated part program		OFF		EDIT	PROG	O → Program number to be searched → ERASE
		All part programs on tape		OFF				O → -9999 → ERASE

Notes:

1. Storing of a part program having a program number different from program number 0 on tape is performed by the same operation as for "tape without program number 0."
2. Within the limit of 32 characters, addition of multiple address data and the change to one address data are permitted.
3. Setting is punched out at the same time.
4. If the tape contains setting information, it is also stored and matched at the same time.
5. When the cursor to the address data in the middle of a block and EOB and ERASE keys are depressed, the data following the cursor position is deleted.
6. When data has been stored from a parameter tape, turn the power on and off.

5. TAPE READER

5.1 TAPE READER COMPARTMENT

5.1.1 TAPE FEED SWITCH AND SYSTEM NO. SWITCH

These switches are located above the tape reader and are accessible when the tape reader door is opened.

· TAPE FEED switch

This is a spring return type switch for manually feeding the tape forward and backward. When the switch lever is pushed to the F (forward) direction, the tape moves forward, or from right to left, and when it is pushed to the R (reverse) direction, the tape moves in the reverse direction. However, during an automatic or manual operation cycle, and while the tape guide is pushed up, this switch is ineffective. The tape feeder is stopped when the control reads % code.

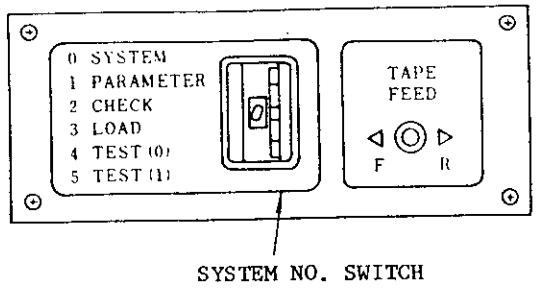


Fig. 5.1

· SYSTEM No. Switch

Normally, this switch is not to be manipulated by the user. During ordinary NC operations, the switch should be in the 0 position. For writing parameters, it should be set at "1." For details, refer to 4.1.13 TAPE FEED AND SYSTEM NO. SWITCHES.

5.1.2 TAPE READER

· Light Source

As the light source, and LED is used. It requires no maintenance except for daily cleaning.

· Tape Reading Head and Tape Feeder

In the tape reading head, phototransistors are built in under a glass window. Since dust and scratches on the glass window are liable to cause reading errors, it should be cleaned periodically, and handled with care. Refer to 8.1 ROUTINE INSPECTION SCHEDULE. Located in the center is the tape feed sprocket, for engaging the feed holes of the part program tape.

· Tape Guide

For feeding a tape through the tape reader, push up the tape guide release magnet under the reading head to raise the tape guide, pass the tape over the reading head, and then, push down the tape guide gently. While the tape guide is lifted, cycle start in the TAPE mode cannot be made, and the TAPE FEED switch is ineffective. The sprocket is freed from the feed holes so that the tape can be pulled freely in both directions.

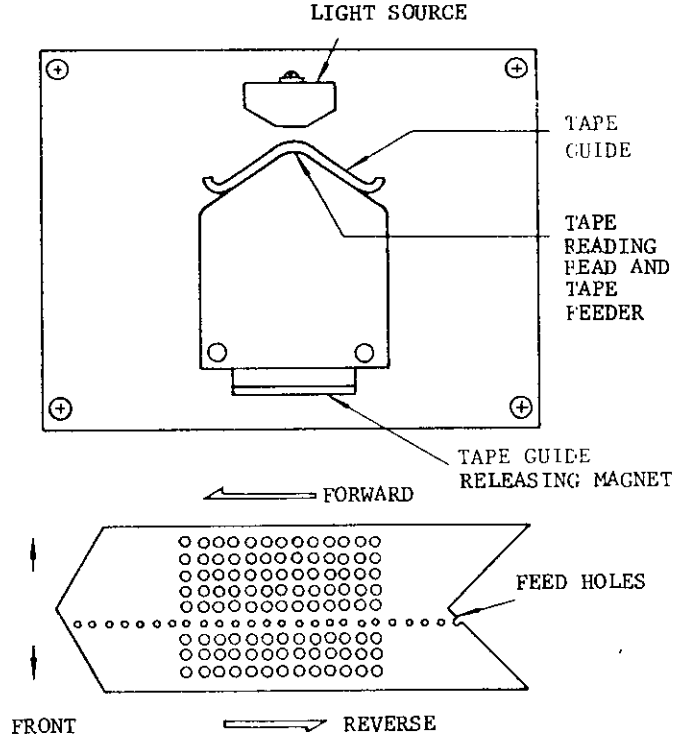


Fig. 5.2

5.1.3 TAPE TUMBLE BOX

The tape tumble box is installed under the tape reader.

To facilitate tape removal from the tumble box, a piece of braided nylon tape is laid in the tumble box covering its entire interior space. As the part program tape is piled up on the nylon tape as it is dropped into the tumble box, the entire part program tape comes out of the tumble box when the nylon tape is pulled up.

When the two screws at the bottom of the tape tumble box are unscrewed, the lower part of the box opens to permit tape removal downward. Clean the interior of the tape tumble box periodically, in accordance with 8.1 ROUTINE INSPECTION SCHEDULE.

NOTE:

The unit type housing is not provided with this tape tumble box.

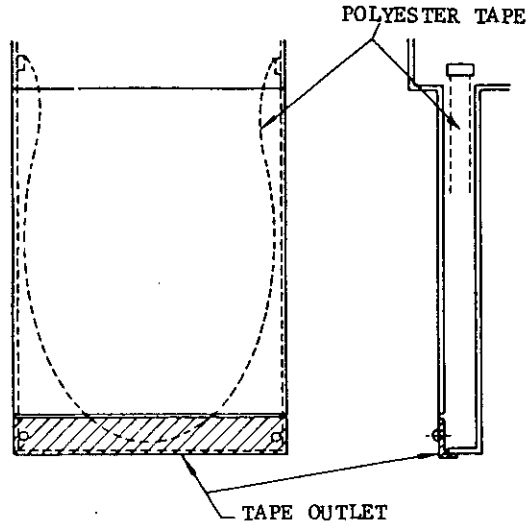


Fig. 5.2. Tape tumble box

5.2 TAPE REELS †

5.2.1 TAPE REELS

One of the following two types of tape readers designed for use with tape reels are available as options.

	Tape reel	Reel dia.	Note
A	6" reel	6"	For 80 m part program tape
B	8" reel	8"	For 180 m part program tape

Note: These tape lengths are for 0.108 mm thick tape.

5.2.2 HANDLING 6-INCH TAPE REELS

When the tape is not in use, and when mounting and dismounting the tape, be sure to arrest the tension arms by the tension arm rest. While the tension arms are in the arrested positions, the reel motor is switched off, and the reels are free.

Mount the tape as follows.

1. Pull the reel lock pawl on the right reel spindle to the horizontal position, and dismount the reel from the spindle.

2. Insert the trailing end of the punched part program tape into the slit in the hub of the right reel, and wind the tape on the reel.

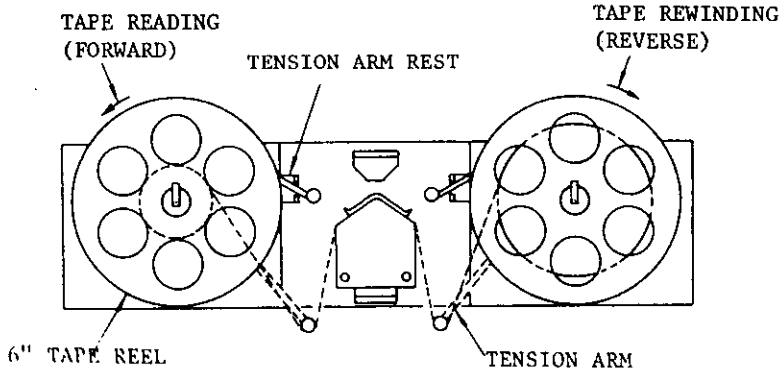


Fig. 5.3

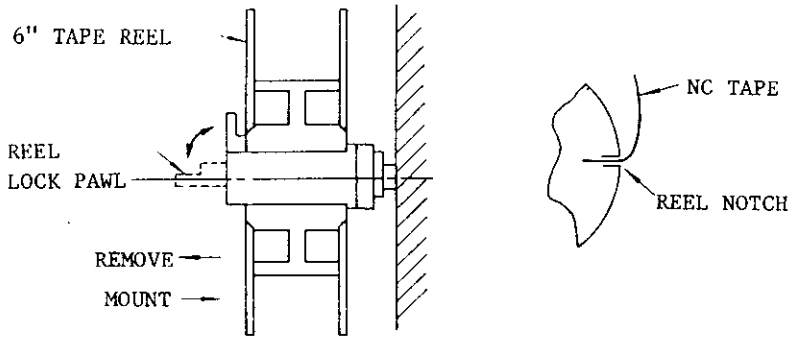


Fig. 5.4

3. Mount a reel with a tape wound on it on the right reel shaft, align the reel lock slot with the reel lock pawl, and push the lock pawl to the vertical position to lock the reel to the shaft.

4. Pull the tape approximately 1 m out of the reel, and pass it through the tape reader.

5. Wind the free end of the tape on the left reel in the same way as 1 through 3, and mount the reel on the left reel shaft. At this time, wind the tape at least 3 turns to eliminate any possibility of slipring loose.

6. Holding the reel with hand to prevent slackening of the tape, free the tension arm from the tension arm rest, and lower it gently. The broken lines in Fig. 5.2.2.1 show the correctly set tape.

Now, the tape can be moved and wound smoothly in either directions in the automatic operation mode or by the manipulation of the TAPE FEED switch.

NOTES:

- When dismounting the tape reel, observe the following two points.
  (1) Hold the reel to be removed by hand, and let the tension arm be arrested by the tension arm rest.
  (2) Push up the tape guide of the tape reading head beforehand.
- Start operation only after making sure that the reel lock pawl is engaged in the lock slot in the reel.

5.2.3 HANDLING 8-INCH TAPE REELS

When the tape is not in use, and when mounting and dismounting the tape, be sure to switch off the power supply switch. In this condition, the reel motor is switched off, and the reels are free.

Names and the functions of members are as follows.

- Power supply switch

  When this switch is turned on, the tape reels are ready to operate.

- Guide rollers

  They guide the tape in rewinding and in forward feeding.

- Tension arm

  In conjunction with the guide rollers, the tension arms give proper tension to the tape and eliminate slack.

- Tension arm stopper

  While no tape is mounted on the tape reader, the tension arms rest on these stoppers.

- 8" tape reel

  In a tape reel 8" in diameter, a part program tape approximately 180 m in length can be wound.

- Reel hub

  The reel hubs are mounted on the reel spindle, and serves to receive and arrest tape reels on them.

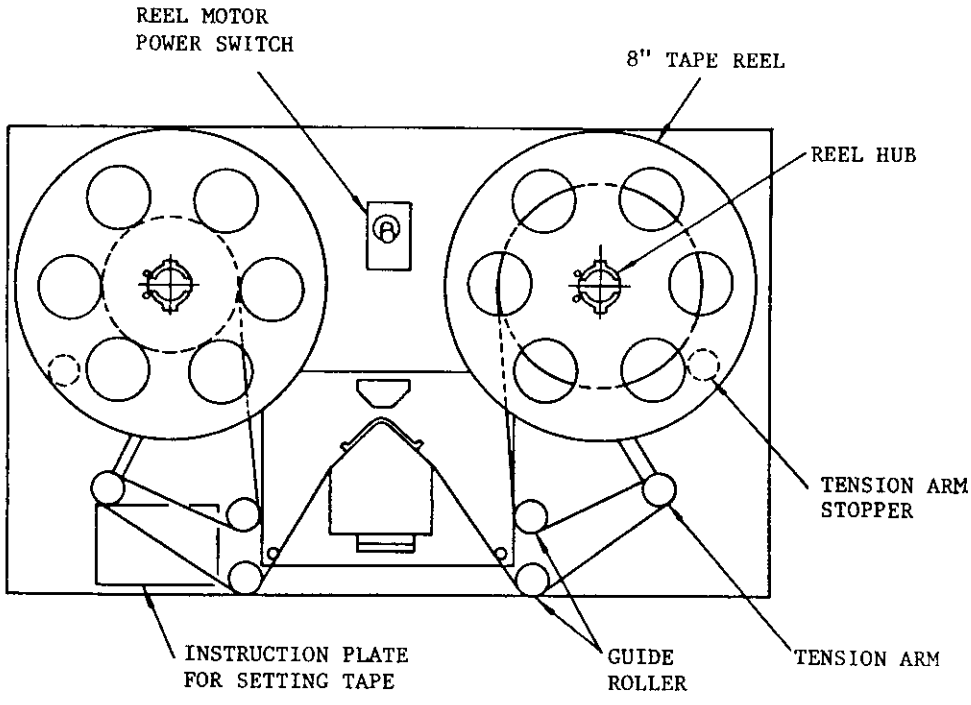


Fig. 5.5

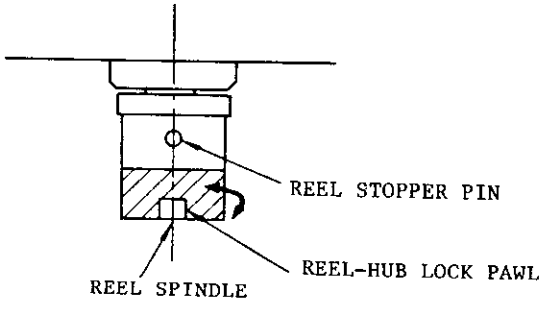


Fig. 5.6

· Tape threading graphic plate

This plate shows the tape threading route graphically.

Thread the part program tape as follows.

1. Holding the right reel with one hand, turn the reel hub until the pawl is aligned with the reel groove, and dismount the outside reel flange, the tape bobbin, and the inside reel flange from the tape hub.
2. Insert the trailing end of the part program tape in the slit in the tape bobbin, and wind the tape on it. (When a tape bobbin without a slit is used, fasten the tape end to the bobbin with adhesive tape.)
3. Turn the reel hub until the lock pawl is aligned with the stop pin. (See Fig. 5.2.3.2)
4. Mount the inside reel flange on the reel hub, aligning the groove in the flange with the lock pawl on the reel hub.
5. Mount the reel bobbin with a part program tape wound on it on the inside reel flange.
6. Mount the outside reel flange, aligning the notch in the flange with the lock pawl, and turn the lock pawl until it is brought between the two notches on the inside reel flange.
7. Unwind the part program tape approximately 1.5 m.
8. Thread the tape through the right guide rollers and the tension arm to the tape reader.
9. Wind the leading end of the tape on the left reel in the same way as 1 through 6.
In this case, wind the tape 2 to 3 turns on the tape bobbin and make sure that the tape does not slip loose when pulled.

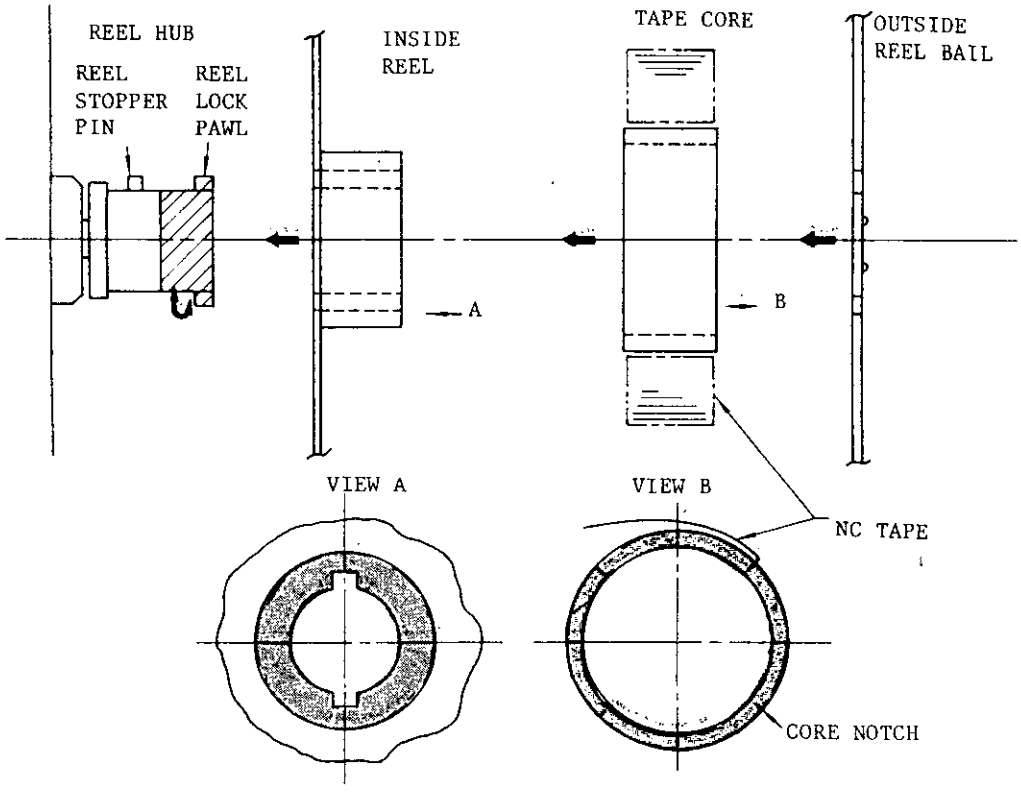


Fig. 5.7 Tape Mounting Procedure

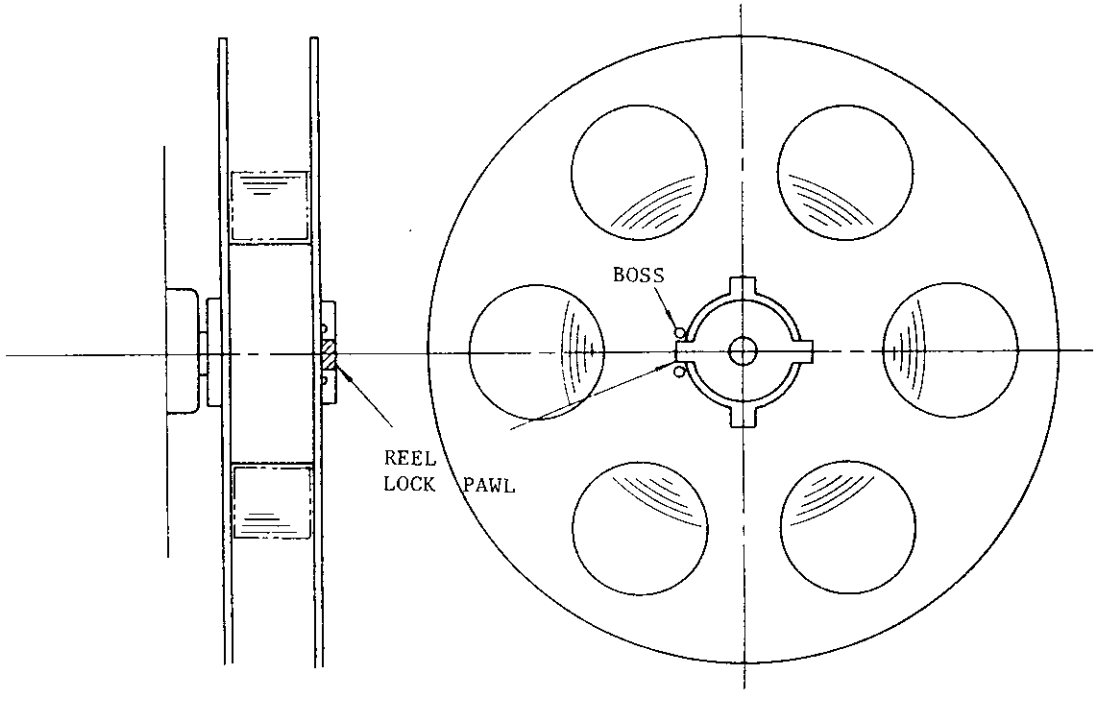


Fig. 5.8 Mounted Tape Reel with Tape

10. Thread the tape through the left guide rollers and the tension arm, as shown in the tape threading graphic plate.

11. Turn the two reels by hand and bring the tension arms to a middle position between the guide rollers and the tension arm stopper. Now, the tape reels and the part program tape have been properly mounted as shown in Fig. 5.2.3.1.

12. Turn the power supply switch on. The tension arms may move momentarily, but become stable soon. Now, the tape can be driven smoothly in both directions in the automatic operation mode and by the manipulation of the TAPE FEED switch.

NOTES:

- While the part program tape is not in use, keep the power supply switch turned off.
- Before dismounting the loaded tape reels, be sure to do the following.
  (1) Turn off the power supply switch.
  (2) Release the tape from the tension arms.
  (3) Push up the tape guide away from the tape reader.
- While the power supply switch is on, do not turn the tape reels by hand, etc.

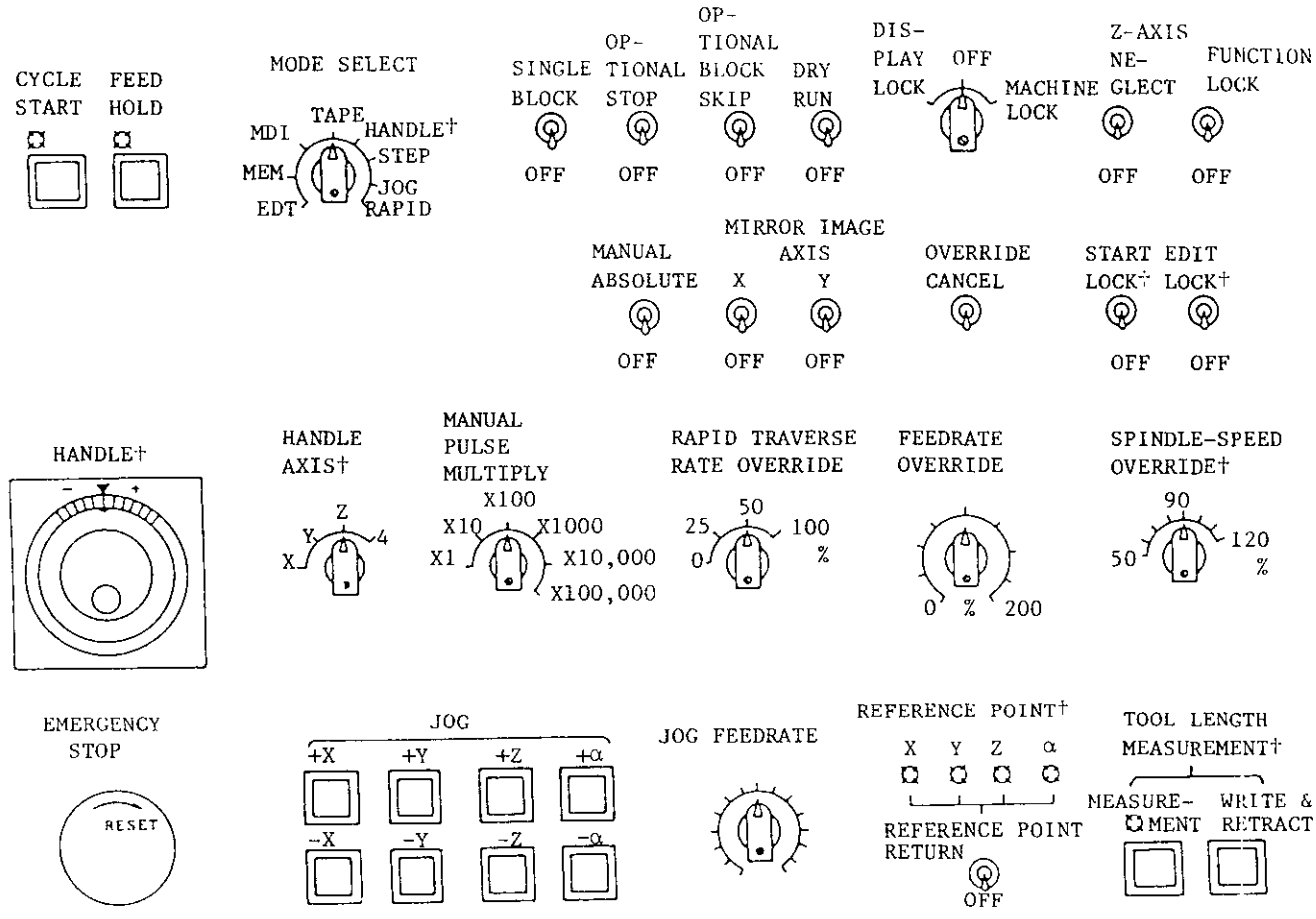


Fig. 6.1 Machine Control Station

6. MACHINE CONTROL STATION

6.1 SWITCHING UNITS ON THE CONTROL STATION

Fig. 6.1.0 shows the layout of switching units on the control station. For details, refer to the machine tool builder's manual.

6.1.1 MODE SELECT SWITCH

This switch gives the operator a choice among the following eight modes of operation (RAPID, JOG, STEP, HANDLE, TAPE, MDI, MEM, EDT). RAPID, JOG, and HANDLE modes are called manual operation mode, and TAPE, MDI, and MEM, automatic-operation mode in this manual.

RAPID: Allows the tool to traverse rapidly or return to reference zero by manual operation.

JOG: Allows the tool to feed continuously by manual operation. Feedrate is set by JOG FEEDRATE switch.

STEP: Allows the tool to feed manually by step each time JOG pushbutton is depressed.

HANDLE†: Allows the tool to feed by operating the manual pulse generator†.

TAPE: Automatically controls the system from NC tape.

MDI: Allows the operator to insert up to 10 blocks of data through the DATA keyboard and control the system automatically with the data.

MEM: Automatically controls the system with the stored part program.

EDT: Stores the part program into memory and edit the part program.

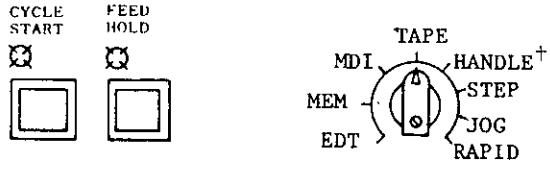


Fig. 6.2

6.1.2 CYCLE START PUSHBUTTON AND LAMP

Depress this pushbutton to start the system in the automatic operation mode (TAPE, MDI and MEM). The CYCLE START indicating lamp lights when automatic operation starts. Depress it again to start the operation after temporary stop by operating FEED HOLD pushbutton or MODE SELECT switch.

6.1.3 FEED HOLD PUSHBUTTON AND LAMP

When the FEED HOLD pushbutton is depressed during automatic operation, the feedrate is decreased immediately and machine motion is stopped. Feedhold is not active during tapping by G84, however, it functions during positioning before tapping. Depressing the FEED HOLD pushbutton during dwell by G04 works on completion of the current block.

If it is depressed while M-, S-, T or B†-function without move command is being executed, the FEED HOLD lamp will light, but these functions will be continued until finished. On completion of the function, the lamp goes off and machine operation is stopped. Depress the CYCLE START pushbutton to resume the operation after temporary stop by operating FEED HOLD pushbutton.

FEED HOLD lamp is automatically illuminated when the machine stops temporarily during canned cycles if SINGLE BLOCK switch is set on.

6.1.4 EMERGENCY STOP PUSHBUTTON

Depress this pushbutton to immediately stop all machine movement in an emergency. The servo power is turned off and the machine is stopped immediately by dynamic brake. The NC ALARM lamp lights and alarm code "330" and "310" are displayed

To recover the system from an emergency stop after the cause has been removed, take the following procedure.

1. Turn the EMERGENCY STOP pushbutton clockwise to release the locking.
2. Depress the RESET key. Alarm code "330" is deleted from page.
3. Turn on the servo power again by depressing POWER ON pushbutton. NC ALARM LAMP is extinguished and READY lamp lights up.

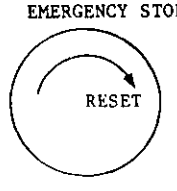


Fig. 6.3

6.1.5 HANDLE DIAL† (MANUAL PULSE GENERATOR)

The dial is used as a manual pulse generator to feed the tool manually with the MODE SELECT switch set to the HANDLE. HANDLE operation is effective for an axis. Procedure of HANDLE operation is as follows:

1. Set the MODE SELECT switch to the HANDLE.
2. Select the axis to be operated with HANDLE AXIS select switch.
3. Set the move amount per graduation of the dial by setting MANUAL PULSE MULTIPLY switch.
4. Rotate the dial to move the selected axis.

   Turning it clockwise causes the axis to move in the plus direction. The axis moves in the minus direction by turning it counterclockwise.

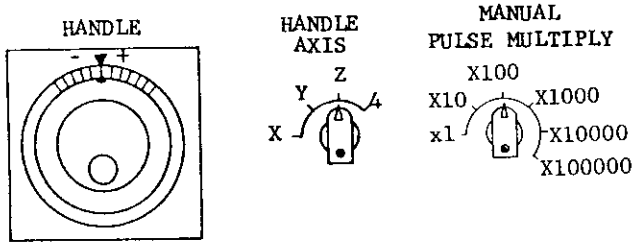


Fig. 6.4

6.1.6 HANDLE AXIS SELECT SWITCH†

This switch is used to select an axis to be operated. For operation of the switch, see 6.1.5 Handle Dial†.

6.1.7 MANUAL PULSE MULTIPLY SELECT SWITCH†

This switch is used to:

- Select the value from Table 6.1. corresponding to a single graduation of the HANDLE dial in the HANDLE mode.
- Select the move amount (1 step) from Table 6.2 corresponding to each depression of JCG pushbutton in the STEP mode.

Table 6.1 Selection of Move Amount in the HANDLE Mode†

	Metric	Inch	Rotating Angle†
x 1	0.001 mm/ graduation	0.0001 inch/ graduation	0.001 deg/ graduation
x 10	0.01 mm/ graduation	0.001 inch/ graduation	0.01 deg/ graduation
x 100 x 1000 x 10000 x 100000	0.1 mm/ graduation	0.01 inch/ graduation	0.1 deg/ graduation

Table 6.2 Selection of Move Amount in the STEP Mode

	Metric	Inch	Rotating Angle†
x 1	0.001 mm/step	0.0001 inch/step	0.001 deg/step
x 10	0.01 mm/step	0.001 inch/step	0.01 deg/step
x 100	0.1 mm/step	0.01 inch/step	0.1 deg/step
x 1000	1.0 mm/step	0.1 inch/step	1.0 deg/step
x 10000	10.0 mm/step	1.0 inch/step	10.0 deg/step
x 100000	100.0 mm/step	10.0 inch/step	100.0 deg/step

6.1.8 HANDLE DIALS FOR SIMULTANEOUS CONTROL OF UP TO THREE AXES†

When a manual pulse generator is connected for each axis, the tool can be manually moved along selected three of the four axes (X, Y, Z and α) simultaneously.

1. The tool move distance per graduation of the HANDLE dial for the manual pulse generator is determined by the MANUAL PULSE MULTIPLY switch (Table 6.1.7.1). This switch is effective on all the three axes.

2. Set the mode select switch to HANDLE, and turn the HANDLE dials for the desired axes in the positive or negative direction.

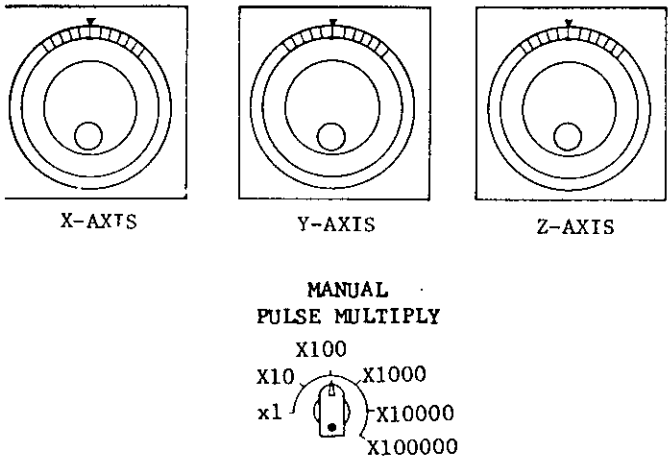


Fig. 6.5

6.1.9 JOG PUSHBUTTONS

This pushbutton is used to feed the tool manually.

- With any of pushbuttons +X, -X, +Y, -Y, +Z, or -Z (+α, -α)† held in the RAPID mode, the axis can be moved rapidly until the button is released.
- These pushbuttons move the tool at the speed set by JOG FEEDRATE switch in the JOG mode.
- Each time the pushbutton is depressed in the STEP mode, the tool is moved by the value per step set by MANUAL PULSE MULTIPLY select switch. Maximum feedrate per step is determined by parameter # "6222."

NOTE: JOG pushbuttons work on all axes.

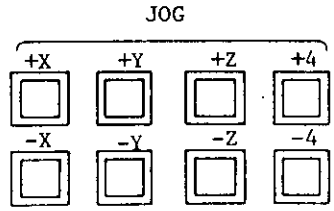


Fig. 6.6

6.1.10 JOG FEEDRATE SWITCH

The JOG FEEDRATE switch is used to select the jog feedrate in the JOG mode. Up to 32 steps of feedrate can be specified. Jog feedrate depends on the machine tool. For definite values, refer to the machine tool builder's manual. See Table 6.1.10. The JOG feedrate can be preset by parameters #6233 to 6264.

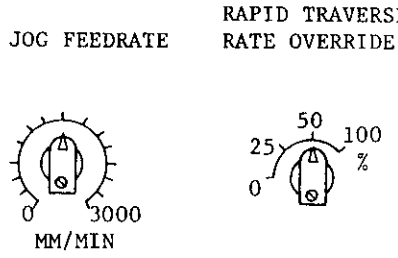


Fig. 6.7

6.1.11 RAPID TRAVERSE RATE OVERRIDE SWITCH

This switch is used to adjust the traverse rate by F0, 25, 50 and 100%. 100% Rate is the rapid traverse rate set by parameter #6280 to 6283. The switch is effective both in automatic operation including G00 command and in manual operation (RAPID mode). F0 is set by parameter #6231.

6.1.12 FEEDRATE OVERRIDE SWITCH

In the automatic operation mode (TAPE, MEM, MDI), this switch is used to adjust the feedrate by 10% from 0 to 200% of the programmed feedrate specified with an F function at whatever position the switch may be set. Feed during tapping by G74 and G33 follows F command. Where OVERRIDE CANCEL switch is set on, the tool will be moved at the programmed feedrate by F code regardless of switch setting.

Table 6.3 Jog Feedrate

Step	Parameter No.	mm/min	Step	Parameter No.	mm/min
0	#6233	0	16	#6249	100
1	#6234	1	17	#6250	120
2	#6235	2	18	#6251	150
3	#6236	4	19	#6252	200
4	#6237	6	20	#6253	250
5	#6238	8	21	#6254	300
6	#6239	10	22	#6255	400
7	#6240	12	23	#6256	500
8	#6241	15	24	#6257	600
9	#6242	20	25	#6258	800
10	#6243	25	26	#6259	1000
11	#6244	30	27	#6260	1200
12	#6245	40	28	#6261	1500
13	#6246	50	29	#6262	2000
14	#6247	60	30	#6263	2500
15	#6248	80	31	#6264	3000

Notes:

- Jog feedrate depends on the machine tool.
  For definite values, refer to the machine tool builder's manual.
- Feedrate of the fourth axis†, if provided, is shown by deg/min.

Table 6.4 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	21	200
10	100		

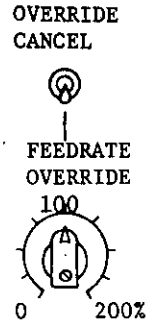


Fig. 6.8

6.1.13 FEEDRATE OVERRIDE CANCEL SWITCH

Turning on the OVERRIDE CANCEL switch prevents the function of FEEDRATE OVERRIDE switch.

6.1.14 SPINDLE SPEED OVERRIDE SWITCH†

1. With this switch, the current spindle speed can be changed to an override speed which is set at 10% intervals between 50 and 120% of the current spindle speed.

2. During the tapping cycle by G74 and G33 under the command of G84, the spindle speed may selectively be made independent of this switch, remaining at the speed set by an S code. This selection is made with the parameter #6007$_{D2}$.

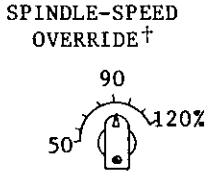


Fig. 6.9

6.1.15 MANUAL REFERENCE POINT RETURN SWITCH†

This switch is for bringing the tool back to the reference point manually.

For its operation method, refer to 6.2.1 MANUAL RETURN TO REFERENCE POINT.

6.1.16 REFERENCE POINT LAMPS†

These lamps indicate that the tool is positioned on the reference point. They light when the tool is brought to the reference point through the manual or automatic return to reference point (G28), or by the reference point return check (G27), and goes out as the tool moves away from the reference point by a subsequent operation.

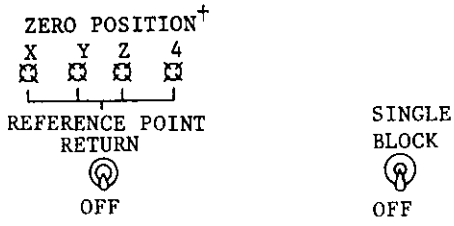


Fig. 6.10

6.1.17 SINGLE BLOCK SWITCH

Turning on this switch permits individual block-by-block operation. Turning on this switch after finishing the current block in the automatic operation mode, the machine stops. A block of data is executed each time the CYCLE START pushbutton is activated.

6.1.18 OPTIONAL STOP SWITCH

This switch is to execute M01 command in automatic operation mode (TAPE, MEM or MDI).

When the switch is on, the program stops on completion of the block including M01 command, while CYCLE START pushbutton remains illuminated. When the control catches FIN signal, the light is extinguished. To restart the program, depress the CYCLE START button. When the switch is off, M01 command is ignored.

Operation of the switch is not effective for the block being executed. During the automatic operation, the switch acts for the next block.

6.1.19 OPTIONAL BLOCK SKIP SWITCH

This switch selects whether the data in blocks including a "/" is disregarded or not.

1. While the switch is on, all the commands in a block programmed after a "/" are neglected. However, block data appearing before the "/" remains effective.

6.1.19 OPTIONAL BLOCK SKIP SWITCH (CONT'D)

2. While this switch is off, blocks including a "/" are executed along with other blocks. This switch is ineffective on the block under execution and blocks stored in the advance-reading buffer. When this switch is turned on during an automatic operation cycle, it works on the block read after the switching on has occured.

NOTES:

a. The two commands "/" and "/1" are equivalent.

b. With the control provided with the optional block skip B function+, 8 independent blocks can be skipped with the switching of the switches corresponding to "/2" through "/9."

6.1.20 DRY RUN SWITCH

Turning on the DRY RUN switch in the TAPE, MDI or MEM mode causes the tool to move at the speed selected by the JOG FEEDRATE switch, ignoring all programmed F-functions. F commands can be displayed as they are programmed. This switch may be used to check the program.

Rapid traverse (G00) rate for dry run operation can be set by setting parameter # "6006D2."

Parameter # "6006D2"	Rapid Traverse at Dry Run Operation
"0"	Rapid traverse rate*
"1"	Jog feedrate

* The tool moves at the traverse rate set by RAPID TRAVERSE RATE OVERRIDE switch if provided.

NOTES:

- Switching the DRY RUN switch during automatic operation becomes effective on the current block. Switching it in mm/rev mode+ or during tapping becomes effective on the next block.
- Rapid traverse rate override is kept effective during dry run operation.
- During tapping, the set speed when tapping starts will be kept. It cannot be changed by JOG operation during dry run operation.

Fig. 6.11

6.1.21 DISPLAY LOCK/MACHINE LOCK SWITCH

This switch functions to stop updating the position display, or to stop move command pulses to the servos. This switch cannot be set unless the machine is stopped at block end or temporarily stopped by FEEDHOLD pushbutton.

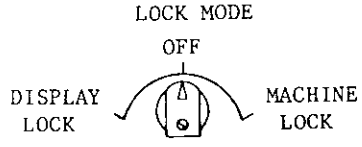


Fig. 6.12

"OFF"

Usual operation is made at "OFF" position in both manual and automatic operation. The machine and the position display operate according to the command by automatic operation or manual operation.

"DISPLAY LOCK"

This position is used to exclude the axis movement value from the position display. Current position display is not updated, though the machine moves.

"MACHINE LOCK"

Setting the switch at MACHINE LOCK inhibits axis movement including Zero Return. The position display is updated. M-, S-, and T-functions are executed. This position is selected to preset the display or to check the tape data.

6.1.22 Z-AXIS FEED NEGLECT SWITCH

The switch is used for dry run operation or drawing-check operation on the X-Y plane. Turning on the switch causes the Z-axis in MACHINE LOCK condition. The Z-axis movement is inhibited, though the position display is updated.

Operate the switch when the machine is stopped. That is, the switch does not function except when the machine is stopped at the block end by SINGLE BLOCK switch or temporarily stopped by FEED HOLD pushbutton.

6.1.23 M-FUNCTION LOCK SWITCH (AUXILIARY FUNCTION LOCK)

- When the M-FUNCTION LOCK switch is on, it ignores the M, S, T, and B† commands. To check the tape data, the operation by the switch is used in combination with MACHINE LOCK function.

Fig. 6.13

- The following M codes are executed even if the switch is set on.
  - M00, M01, M02, M30
    Both its decoded signals and its BCD codes are sent out to the machine.
  - M90 to M99
    BCD code is not sent out.
- Turning on the M-FUNCTION LOCK switch during automatic operation becomes effective on the block after the next block of the current block.

6.1.24 MANUAL ABSOLUTE SWITCH

- When MANUAL ABSOLUTE SWITCH is on.

  When automatic operation is restarted after interrupted by manual operation, the tool performs the rest of the command in the interrupted block from the end point of manual operation. The tool moves in parallel with the path specified by the program.

  When the command of the next block is G00 or G01, the tool moves automatically to the target coordinate specified by the program. Then the operation is performed according to block of data.

  When the command of the next block is G02 or G03 (circular interpolation), the interpolation is performed in parallel with programmed command. The tool automatically returns to the target coordinate when G00 or G01 is commanded after the circular interpolation.

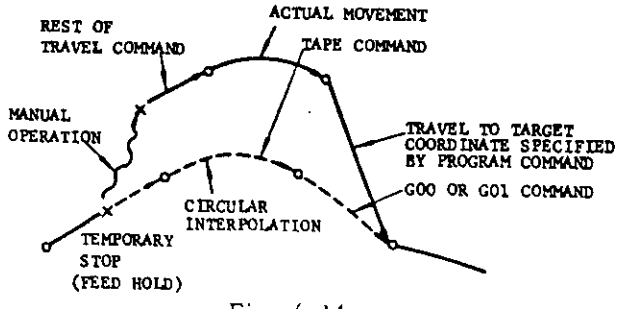


Fig. 6.14

- When MANUAL ABSOLUTE switch is off.

  After the automatic operation is interrupted by manual operation, the coordinate system is shifted, and the tool performs the rest of the travel commands in parallel with programmed moves.

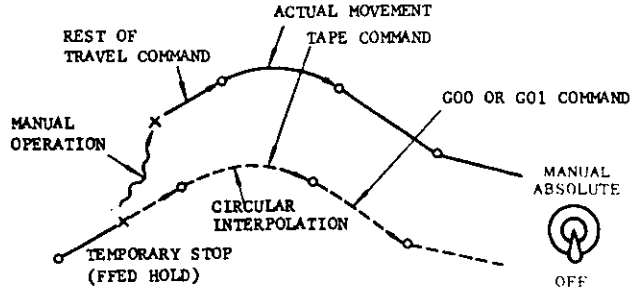


Fig. 6.15 Tool Movement with MANUAL ABSOLUTE Switch Off

The parallel shift is reset by executing Reference Zero Return manually, automatically by G28, or operating the RESET key. The command value is forced to change to the current position. Thus the shift value is reset.

6.1.25 MIRROR IMAGE AXIS SELECTOR SWITCH

MIRROR IMAGE AXIS switch selects the axis whose motion is reversed for programmed operations.

To select the mirror image axis with this switch as well as setting function, set the data of setting # "6000$_{D0-D3}$" to 0.

Turn on the MIRROR IMAGE AXIS switch of the axis to which Mirror Image function is assigned. The motion of the selected mirror image axis is set up at M95 command is given until M94 is commanded. For details, see 2.8.5.

NOTE: During the M95 (Mirror Image ON) mode, never operate the MIRROR IMAGE AXIS switch.

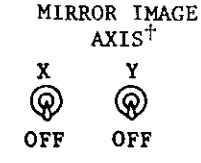


Fig. 6.16

6.1.26 TOOL LENGTH MEASUREMENT PUSH-BUTTON AND LAMP†

Use the WRITE button to automatically store the amount of Z-axis move manually made between "home-position" and "base-position" directly in the tool offset memory. For operating procedure, refer to 6.2.3 Automatic Tool Length Measurement†.

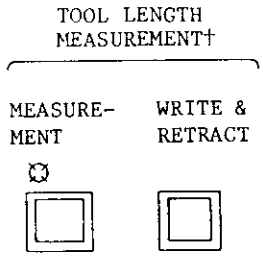


Fig. 6.17

6.1.27 START LOCK INPUT (OR SWITCH)†

When the START LOCK is on, CYCLE START pushbutton does not function. Use the START LOCK input to prevent operating the machine in abnormal condition during automatic operation. The input may be used as on/off switch on control station for machine.

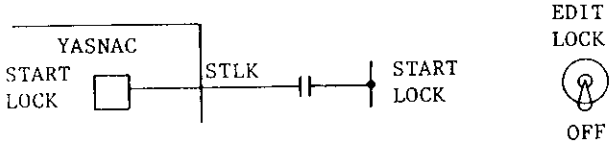


Fig. 6.18

6.1.28 EDIT LOCK SWITCH†

Turning on the EDIT LOCK switch prevents the function of ERS, INS, ALT, and EOB keys, and storing from NC tape. When editing is made with EDIT LOCK switch turned on, "EDIT LOCK" flickers on the CRT display.

6.1.29 AXIS INTERLOCK INPUT

The control is provided with AXIS INTERLOCK input for each axis to prevent axis motion.

Interlocking an axis in feed motion causes the axis to slow down to a stop. When the interlock is released, the axis motion finishes the interrupted block and proceeds to the next.

Interlocking one of the two or three axes being simultaneously interpolated disables the interpolation.

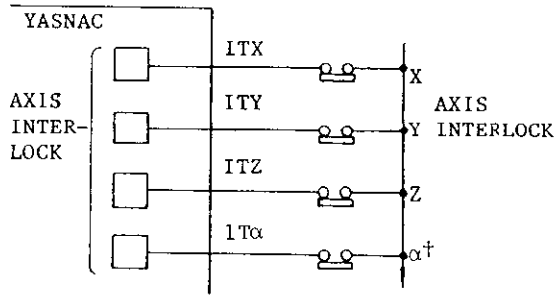


Fig. 6.19

6.1.30 EXTERNAL DECELERATION INPUT SIGNALS

In order to eliminate the danger of high speed contact at speed end caused by erroneous motion commands, limit switches for originating external deceleration input signals may be installed at selected points.

1. During rapid traverse (G00) and manual operation

   When the limit switch is tripped by the tool movement, the traverse speed is decelerated in the tripping direction to a level set by the parameter #6340. In the direction opposite to the tripping direction, the original speed remains unchanged.

2. During motion at feedrate (G94)

   While the limit switch is being tripped, the tool moves at a speed set by the parameter #6341. If the feedrate set by the F command is lower than the rate set by the parameter, the original feedrate remains unchanged.

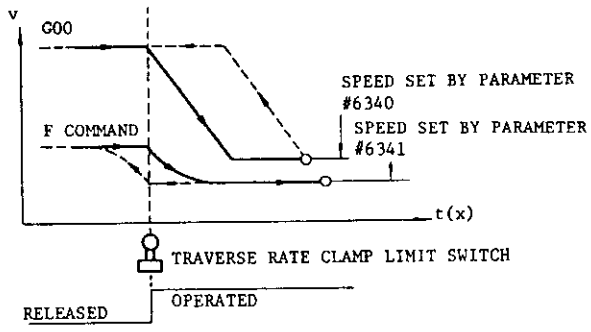


Fig. 6.20

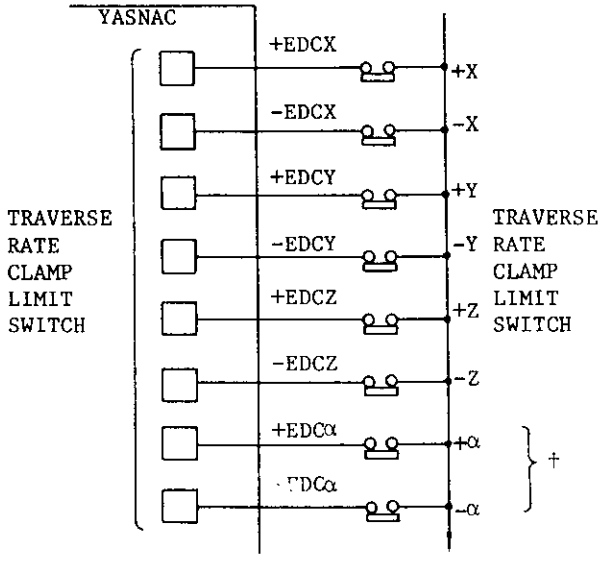


Fig. 6.21

Notes:

i. The external deceleration function is ineffective on feedrate specified in mm per revolution of the spindle (mm/rev).

ii. It is also ineffective on the HANDLE feed.

6.2 OPERATION PROCEDURE

6.2.1 MANUAL RETURN TO REFERENCE POINT

With this function, the tool is returned to the reference point manually. The procedure is as follows.

1. Set the mode select switch to RAPID or JOG.

2. Manually move the tool to a position some distance away from the reference point. When the tool is within the range A shown below, it can be brought back to the reference point in the normal way, as described below.

3. Turn on the REFERENCE POINT RETURN switch.

4. Keep the JOG button for the return direction depressed. The tool starts to move as in the normal manual control, but the speed is decelerated at the deceleration point, and the motion stops automatically at the reference point.

5. Then, the REFERENCE POINT lamp for the relevant axis lights.

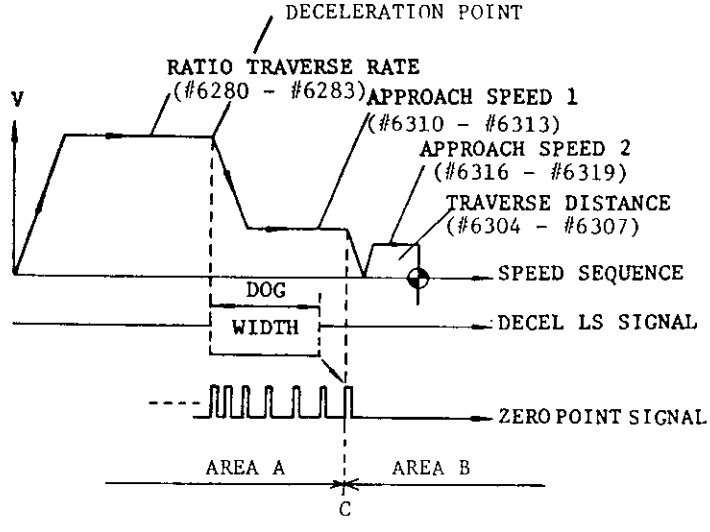


Fig. 6.22

NOTES:

a. As long as the power supply is turned on, either the manual or the automatic return to reference point can be initiated, regardless of the tool position, but the tool will not return to the reference point accurately if the tool is started from a point in the area B. Be sure to bring the tool into the area A before initiating a manual or automatic return motion.

b. Once the tool is returned to the reference point, the point C is stored, and if the reference return motion is initiated from a point in the area B, this is regarded as an error. Start the reference return motion from a position in the area A.

6.2.1 MANUAL RETURN TO REFERENCE POINT (CONT'D)

c. Once the tool is returned to the reference point, it can not be further moved in the same direction unless the REFERENCE POINT RETURN switch is turned off.

d. While the MACHINE LOCK switch is on, the reference point return function is ineffective.

e. Do not return the tool to the reference point by the manual reference point return function, while the buffer is loaded with blocks read in advance of execution, because the stored motion data will be erased by the reference point return motion.

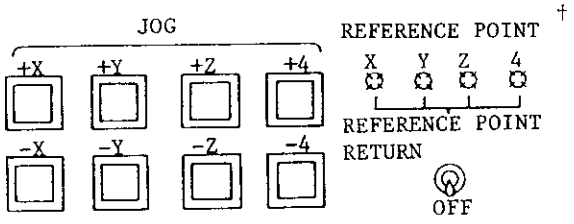


Fig. 6.2.1.2

6.2.2 AUTOMATIC COORDINATE SYSTEM SETTING†

With this function, a new coordinate system is set up automatically upon the return of the tool to the reference point by the manual reference point return function. The coordinates of the new origin are preset with the following parameters. The coordinate system set up by this function is equivalent to the ones set up by G92.

1. Parameters for metric system

Parameter	Meaning
#6636	X coordinate
#6637	Y coordinate
#6638	Z coordinate
#6639	4th coordinate

2. Parameters for inch system

Parameter	Meaning
#6630	X coordinate
#6631	Y coordinate
#6632	Z coordinate
#6633	4th coordinate

3. Axis can be selected by parameter #6015 for both metric and inch systems.

4. Upon the return of the tool to the reference point by the manual reference point return function (G28), when the controlled axis is the 4th or 5th axis of the rotary axis (A, B, or C), a new coordinate system is set up automatically in the same way as mentioned above.

Upon Tool Return to Reference Point	Basic Axis X, Y, Z	4th, 5th Axis U, V, W	4th, 5th Axis A, B, C
Manual	○	○	○
Automatic	×	×	○

○: When parameter #6015 Dn = 1, the automatic coordinate system setting is effective.

×: Even if #6015 setting is 0 or 1 the automatic coordinate system setting will not be performed.

5. Restrictions for automatic coordinate system setting by G28

When automatic coordinate system setting G28 is executed on rotary axis, the tool is returned to the nearest reference point [360° × n(n: integer), if first reference point is 0°]. At this time, the display shows the current value of 360° × n. Operation should not be continued at this value. For operation, current value display should be set up automatically to the desired value.

NOTE:

If G28 command (automatic coordinate system setting) is used during operation on the work coordinate system (G54 to G59 commands), basic coordinate system shifts. As a result, work coordinate system, which is commanded by G54 to G59 and set on the basic coordinate system, α, shifts also. Refer to 2.9.25, Work Coordinate System Setting (G52 to G59).

6.2.3 AUTOMATIC TOOL LENGTH MEASUREMENT†

When a tool mounted on the spindle is manually brought to a position where the tool tip makes contact with the reference surface for Z-axis, and the WRITE & RETRACT button is pushed, the following operations are performed by the control.

a. The distance between the set Z-axis home position and the reference surface is stored automatically in the memory having the currently specified correction number.
The difference between tool touch position and base position can be set by parameter.

b. Increase the correction number by 1, in preparation for the next writing.

c. Return the tool to the Z-axis home position.

1. Measuring method

a. Mount a tool on the spindle, and move it to a Z-axis position which is to be set as the home position. Any position may be set as the home position, but for facilitating tool changing process, the tool changing position may be set up as the home position.

b. Select the manual operation mode (RAPID, JOG, HANDLE or STEP) using MODE SELECT switch.

c. Push the function key [OFS].

The offset number specified previously and related data are displayed.

d. The page covering tool offset values specified the tool offset number keyed in will be shown. The specified number is shown by cursor

e. Push the MEASUREMENT button when the motion stops. MEASUREMENT lamp lights and the current position of Z-axis will be set as home position.
(The button is effective only in the manual operation mode and while the [OFS] key is selected.

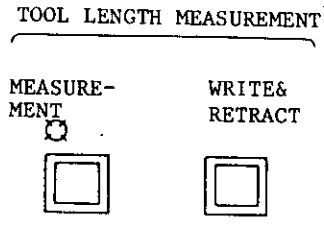


Fig. 6.24

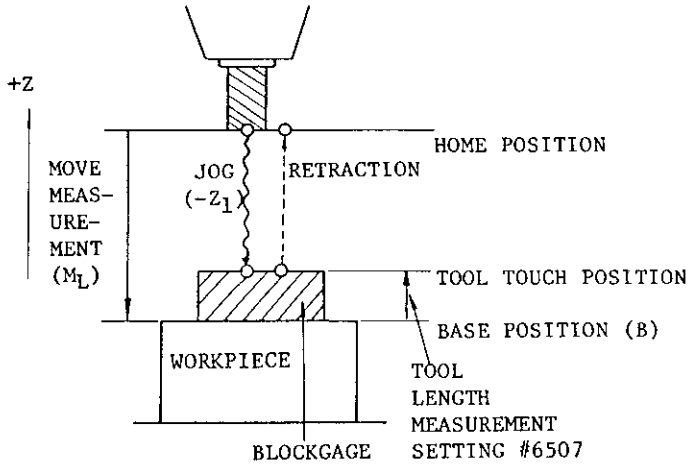


Fig. 6.25

Home position is the point where measurement starts. Measurement is made with the point temporarily determined as coordinate 0.

Note: To display the home position as coordinate 0, reset the position referring to 4.3.4.1. Measurement after resetting will be made in the offset mode.

f. Set the tool to the tool touch position by controlling Z-axis manually. Tool touch position is the position specified arbitrarily based on the base position (machine workpiece surface). The difference between tool touch position and base position is set in advance in setting #6507 ("1" = input unit). The difference is regarded as thickness of block gage. Write it in plus value.

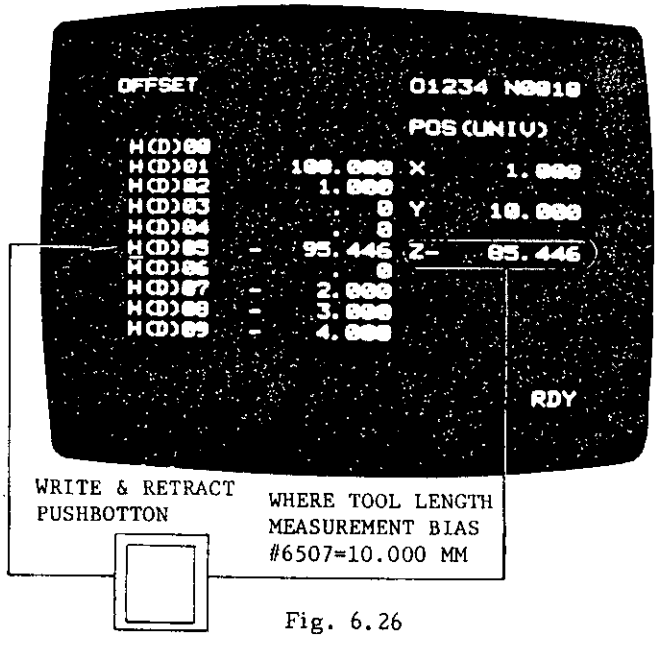


Fig. 6.26

The figure above shows the value in the offset number 05.

6.2.3 AUTOMATIC TOOL LENGTH MEASUREMENT† (CONT'D)

g. Depress the WRITE & RETRACT button. The control executes the following operation.

i. The control stores the difference between home position and base position, that is, move measurement value (ML) as the value specified by the offset number.

$ML = (-Z_1) - (B)$

ii. The tool automatically returns in the Z-direction to the home position in rapid traverse.

iii. The tool offset number is increased by one in preparation for next writing of offset value.
(When it is H99, H01 is designated.)

h. Exchange the tool with a next tool by manual operation or by MDI operation. Even when the MDI mode is switched on, the MEASUREMENT lamp remains lighted. Return to the MANUAL mode afterwards.

i. Repeat the processes f. through h. to store all the required offset values.

j. Push the MEASUREMENT key.

The MEASUREMENT lamp goes out, and the automatic writing function is turned off. The length measuring data also disappears from the CRT.

NOTES:

- In this automatic writing mode, the measured values are stored in the absolute values.
- When the home position is different from the tool change position, the new tool may be brought to the reference surface directly without first returning to the home position. Once the home position has been set up by the use of the MEASUREMENT key, repositioning to the home position is not necessary.
- When the WRITE & RETRACT key is pushed with H00 designated as the tool offset number, the tool offset number is changed to H01 but no writing nor tool return is performed. When the key is pushed again, the tool offset value is written under H01 in the normal manner.

2. Operation for measuring remaining distance (Parameter $\#6039_{D4}$ = "1")

The following three methods are possible in addition to the method described above.

a. Whereas the move measurement value M_L (Fig. 6.2.3.2) is written as a tool offset value, the remaining distance R_L is written when the parameter $\#6039_{D4}$ is set to "1."

The bottom level which is used as a base of calculating remaining distance can be set by setting #6508 (bottom level setting, 1 = input unit). Writing operation is the same as described in step 1.

$R_L = -(A - M_L)$

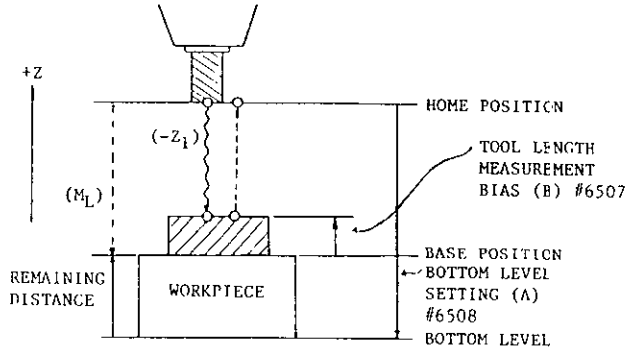


Fig. 6.27

3. The following method is possible with NC control station.

i. Instead of the MEASUREMENT key, depress the POS and PAGE key to select the page of POSITION [UNIVERSAL].

ii. Reset the display by depressing Z and ORG keys. This means that home position is set to "0."

iii. Return the function mode to offset by depressing OFS key.

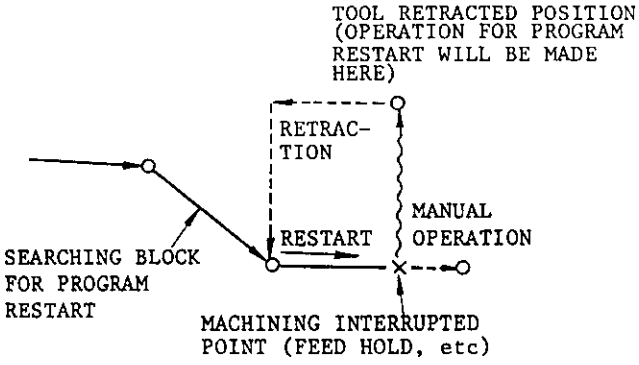
iv. Instead of the WRITE & RETRACT key, depress the NEXT, Z, WR keys in that order.

This executes automatic writing of the same tool offset measured as written in step 1. The Z-axis, however, will not return to home position.

v. In this measurement operation, parameter $\#6039_{D4}$ (move measurement value/remaining distance switching) is effective. Storing either one of the values is determined by parameter.

6.2.4 PROGRAM RESTART†

Machining may be restarted from the block that follows the one for which the sequence number was specified. This restart is useful when replacing the broken tool or taking over the machining operation from the last work shift.



Program restart is of either type P or type Q, depending on whether the change of the coordinate system before or after the restart is permitted.

6.2.4.1 PROGRAM RESTART OF TYPE P

In type P, the program is restarted assuming that the change of coordinate system before and after the restart will not occur. Therefore, this type is used after the replacement of the tool broken during operation, for example.

Operation Procedure

When the tool is broken during machining operation, the automatic operation may be restarted in the following procedure:

(1) Press FEED HOLD button to stop the operation and make the tool escape.

(2) Replace the broken tool and perform the associated chores. Change the tool offset, if required.

(3) Turn on PROGRAM RESTART switch on the machine control station.

(4) Press PROG function key to display the part program being executed. Take note of the sequence number of the block immediately before the block on which the feed-hold operation was performed.

(5) Set the head of the part program as follows:

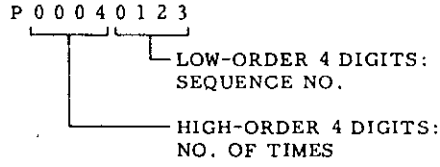
a. In memory mode

Set memory mode, key-in the program number (Oxxxx) to be restarted, and press [CURSOR ↓] key.

b. In tape mode

Set tape mode and set the head of the tape to the tape reader.

(6) Key-in P, sequence number, and [CURSOR ↓] in this order. The sequence number is the one that was taken note of in step (4).
While making the setups for program restart, the control searches for the block of the specified sequence number. If the same sequence number appears repeatedly, the nth sequence number may be specified.



For example, the nth sequence number may be called in a subprogram for which L times of execution is specified. For the sequence number that appears first, the high-order 4 digits may be omitted. In this case, the leading zero of the sequence number may also be omitted.

(7) When the search is completed, the CRT screen automatically displays the program restart information.

```
PROGRAM RETURN        O0001 N0125
 (UNIVERSAL)  M  03  33  10  06  45
X   100.000       63  **  **  **  **
Y-  200.000
Z   300.000
A    45.000

 (INCREMENT)  M COUNT (0006)
X-  258.774
Y   483.123      T 0015  0025
Z-  182.347      S 0500
A    73.442      B 0000


                          PRN       RDY
```

Fig. 6.28 Display of Program Restart Information

a. Position (UNIVERSAL) indicates the position of machining restart. Normally, this position is the start point of the block on which the feed-hold operation was performed.

b. Position (INCREMENT) indicates the distance from the current tool position to the machining restart position.

c. The M codes and the number of M codes designated (M COUNT) from the head of the program to the block of the specified sequence number are displayed. However, if the number of designated M codes exceeds 35, the 35 M codes as counted from the specified block are displayed.

d. The two last T codes specified up to the designated block are displayed.

e. The last S code specified up to the designated block is displayed.

f. The last B code specified up to the designated block is displayed.

Note: The M codes and T codes are displayed in the order in which they were specified. The code displayed last is nearest the designated block.

(8) Turn off PROGRAM RESTART switch on the machine control station.

(9) Look at the displayed program restart information and specify the M, T, S, or B code necessary for the restart as shown below:

a. Set MDI mode.

b. Press PROG function key.

c. Key-in the necessary M, T, S, or B code and depress WR key.

d. Press CYCLE START button.

e. Press POS function key and check the display of program restart information.

(10) Set the original operation mode (memory or tape mode).

(11) Depress CYCLE START button.

The tool moves to the machining restart position, axis by axis; that is, the 4th axis, X-axis, Y-axis, and Z-axis, in this order. Then, the automatic operation restarts from the head of the block (on which the feed-hold operation was performed) that follows the block of the designated sequence number.

If the coordinate system is changed by any of the following operations performed after the interruption automatic operation, use the program restart of type Q:

(1) The machine power was turned off.

(2) G92 is specified by MDI operation.

(3) The setting of work coordinate system is specified.

(4) The automatic setting of work coordinate system was specified by reference-point return.

(5) ORG key was pressed.

Operational Procedure

The automatic operation interrupted by any of the above operations may be resumed using the following procedure:

(1) When the machine power is turned on after the interruption of machining, perform the necessary operations such as reference-point return.

(2) Manually move the tool to the start point (of machining) of the part program. Change the tool offset amount if necessary.

(3) Turn on PROGRAM RESTART switch on the machine control station.

(4) Press PROG function key to display the part program (being executed). Record the sequence number of the block immediately before the block to be restarted.

(5) Set the head of the part program as follows:

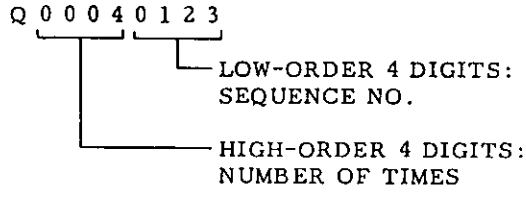
a. In memory mode

Set memory mode, key-in the program number (Oxxxx) to be restarted, and depress CURSOR ↓ key.

b. In tape mode

Set tape mode and set the head of the tape to the tape reader.

(6) Key-in Q, sequence number, and CURSOR ↓ in this order. The sequence number is the one that was taken note of in step (4).
While making the setups for program restart, the machine searches the block of the specified sequence number. If the same sequence number appears repeatedly, the nth sequence number may be specified.



For example, the nth sequence number may be called in a subprogram for which L times of execution was specified. For the sequence number that appears first, the high-order 4 digits may be omitted. In this case, the leading zero of the sequence number may also be omitted.

(7) When the search is completed, the CRT screen automatically displays the program restart information.

(8) Turn off PROGRAM RESTART switch.

(9) Look at the displayed program restart information and specify the M, T, S, or B code required for the restart in MDI mode.

(10) Set the original operation mode (memory or tape mode).

(11) Depress CYCLE START button.

The tool moves to the machining restart position, axis by axis; that is, the 4th axis, X-axis, Y-axis, and Z-axis, in this order. Then, the automatic operation restarts from the head of the block (on which the feed-hold operation was performed) that follows the block of the designated sequence number.

CONSIDERATIONS AND REMARKS

(1) Before depressing CYCLE START button to restart the program execution, check to see if the axis-by-axis tool movement (the 4th axis, X-axis, Y-axis, and Z-axis, in this order) interferes with the work or any part of the machine. If the tool is found interfering, correct the tool position manually.

(2) In both types P and Q, the tool which is moving to the machining restart position axis-by-axis may be single-block-stopped after the completion of the movement for each axis. Even if the incremental shift amount is "0," the single-block stop is performed when SINGLE BLOCK switch is on. When the single-block stop is performed, however, the intervention of MDI operation is not allowed. Manual intervention is possible. But, if a manual intervention is performed on the axis which has already returned, it will not return to the machining restart position again.

(3) During the search operation for program restart, set the switches on the machine control station to the state before commanding program restart. Otherwise, the former position cannot be reached.

(4) When a feed-hold operation was performed during the search operation for program restart, or a reset operation was performed during or after the search operation, perform the operations all over again.

(5) When PROGRAM RESTART switch is on, the operation of CYCLE START button is ignored.

(6) Whether it is before or after machining, each manual operation should be performed with MANUAL ABSOLUTE switch on and MACHINE LOCK switch off.

(7) In any of the following situations, the tool cannot return to the correct position:

a. A manual operation was performed with MANUAL ABSOLUTE switch off.

b. A manual operation was performed with MACHINE LOCK switch on.

c. The search operation for program restart was performed with MACHINE LOCK switch on and then this switch was turned off.

d. A manual intervention was performed during the axis shift to the return position.

(8) In type P, any of the following operations performed in the time between the discontinuation of machining and the search operation for program restart will cause an error:

a. The setting of coordinate system was specified.

ERROR ··· 121 PRTN ERROR (G92)

b. The setting of work coordinate system was specified.

ERROR ··· 122 PRTN ERROR (G54 - G59)

c. The coordinate system was modified by operation ORG key.

ERROR ··· 123 PRTN ERROR (ORG)

In type P, the correct program restart is made possible only for the blocks that follow the one for which the coordinate-system setting was performed last before the discontinuation of machining.

(9) If the designated block is not found, an error will be caused.

ERROR ··· 120 PRTN ERROR (NOT FOUND)

6.2.5 PLAYBACK FUNCTION†

The current axis position during movement can be stored as command value in the part program. This function is permitted in the PLAYBACK mode when PLAYBACK switch is turned on in manual operation mode.

To turn on PLAYBACK mode, activate the PLAYBACK switch on the machine control station or set #6000 D5 at "1." Procedure for storing program in the PLAYBACK mode is as follows.

(1) Set the control in the manual operation mode (RAPID, JOG, STEP, or HANDLE).

(2) Turn on the PLAYBACK switch.

(3) Depress the PROG key of function keys. The CRT screen shows the contents of part program number O plus 4 digits and the current value at the bottom.

(4) Register the new program number after address O. To register 01234, for example, key -in 0, 1, 2, 3, 4 and depress the WR key. The numbers are displayed as they are keyed-in. See Fig.

```
PROG(PLAYBACK)          O0001 N0010
O0001;
N0001   G92 X140 Y100 Z30;
N0002   G00 S1200 T0202 M03;
N0003   G41;
N0004   X0 Z500;
N0005   G01 Z0 F20;
N0006   X2000;
N0007   Z-2000;
N0008   Y2000;
N0009   X3000 Y4000;
 POS    X1234, 567 Y2345, 678
(UNIV)  Z-2345, 789
                                LSK    RDY
```

Fig. 6,29

```
PROG(PLAYBACK)          O1234 N0010
O1234;




G00X1234, 567                   LSK    RDY
```

EDIT BUFFER REGISTER

Fig. 6.30

(5) Move the axis to the position which will be written as command.

(6) Depress the address key for the axis whose position is written.

(7) Depress the WR key.
The displayed current value is the command value of the selected address. Key-in the numerical data before depressing the WR key, and the total amount of keyed-in value plus current value will be the new command value.

NOTE: The current value in POSITION (UNIVERSAL) is displayed by depressing the PROG key in the PLAYBACK mode. It is the same display as that shown in POSITION(UNIVERSAL) by depressing the POS key.

(8) Write-in the value, except for axis, by a block of data in the same way as writing in the EDIT mode. (See 4.6 EDIT.)

(9) Depress the EOB key.

(10) Depress the WSRT key.
A block of data is stored in the part program.

(11) Repeat steps (5) through (10) until the desired data is written.

(12) Key-in M02; or M30; at the end of the part program.

NOTES:

1. The part program stored in the PLAYBACK operation can be edited in both PLAYBACK and EDIT modes. Position data stored in the PLAYBACK operation can be edited (deletion, insertion, alteration).

2. The current value in POSITION (UNIVERSAL) can be written-in using the WR key.

3. Part programs made by PLAYBACK operation and the ones made by writing operation are the same. Maximum capacity of part program is the total of both programs.

4. When the parameter #6006 D5 (ten times the input) is "1," data written through the keyboard will be multiplied by ten.

Ex. At the current position X1.000, keying-in X, 1, 0, and depressing the WR key will store the value X1.1000.

5. MDI operation cannot be intervened during compensation C in the PLAYBACK mode. If the Cycle Start pushbutton is depressed in the PLAYBACK mode, message "PLAYBACK LOCK," will be displayed. With PLAYBACK switch turned off, MDI operation can function.

Writing Operation

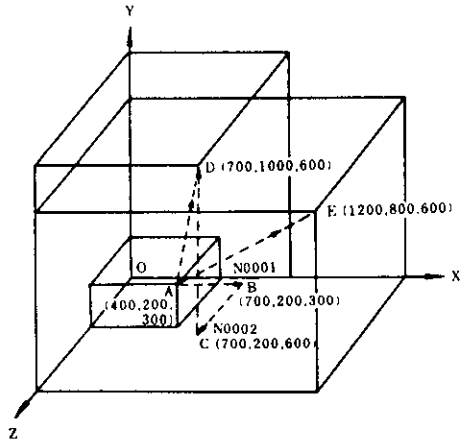


Fig. 6.31

Data of positioning at point A is stored in the part program. The block after point A is stored in the PLAYBACK mode.

A. Procedure of changing current position data using G00 when machine was moved from point A to point B. Sequence No. for this data block is N0001.

(1) Key-in N, 0, 0, 0, 1.
Do not depress the WR key. If depressed, "INPUT ERROR" will appear on the screen.

(2) Key-in G, 0, 0.

(3) Depress WR key after X.

(4) Depress the EOB key.

(5) Depress the INSRT key.

Then, block of N0001 G00 X0.700; will be stored in the part program.

B. Procedure for storing current position data using G01, F02 when machine was moved from point B to point C. Sequence No. for this data block is N0002.

(1) Key-in N, 0, 0, 0, 2.

(2) Key-in G, 0, 1.

(3) Depress the WR key after Z.

(4) Key-in F, 2, 0.

(5) Depress the EOB key.

(6) Depress the INSRT key.

Then block of N0002 G01Z 0.600F20; will be stored in the part program.

C. Procedure for storing current position data using G00 when machine was moved from point C to point D. Sequence No. for this data block is N0003.

(1) Key-in N, 0, 0, 0, 3.

(2) Key-in G, 0, 0.

(3) Depress the WR key after Y.

(4) Depress the EOB key.

(5) Depress the INSRT key.

Then, block of N0003 G00 Y1.000; will be stored in the part program.

D. Procedure of storing current position data for three axes simultaneously when machine was moved from point A to point D with linear interpolation Sequence No. for the data block is N0004.

(1) Key-in N, 0, 0, 0, 4.

(2) Key-in G, 0, 1.

(3) Depress the WR key after X.

(4) Depress the WR key after Y.

(5) Depress the WR key after Z.

(6) Key-in F, 1, 0, 0.

(7) Depress the EOB key.

(8) Depress the INSRT key.

Then block of N0004 G01 X0.700 Y1.000 Z0.600 F100; will be stored in the part program.

E. When the machine current position is at point D (700, 1000, 600) with start at A, follow the procedure to store the current position data using the position data of point D when the machine is moved from point A to point E (1200, 800, 600).

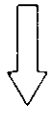
Key-in the incremental value for each axis when machine moves from point D to point E. Sequence No. for this data block is N0005.

```
PROG(PLAYBACK)              O0010N0000
O0010;
G91 G28 X0 Y0 Z0;
G92 X9 Y0 Z0;
G90 G00 X0.4 Y0.2 Z0.3;
N0001 G00 X0.700;
N0002 G01 Z0.600 F20;
N0003 G00 Y1.000;
N0004 G01 X0.700 Y1.000 Z0.600 F100;

 POS      X      .700          Y1.000
(UNIV)    Z      .600

N0005 G01X0.5                    RDY
```

Fig. 6.32



BY DEPRESSING WR KEY, POSITION X IS ADDED TO THE KEYED-IN VALUE.

```
PROG(PLAYBACK)              O0010N0000
O0010;
G91 G28 X0 Y0 Z0;
G92 X0 Y0 Z0;
G90 G00 X0.4 Y0.2 Z0.3;
N0001 G00 X0.700;
N0002 G01 Z0.600 F20;
N0003 G00 Y1.000;
N0004 G01 X0.700 Y1.000 Z0.600 F100;


 POS      X      .700          Y1.000
(UNIV)    Z      .600

N0005 G01 X1.200                 RDY
```

Fig. 6.33

(1) Key-in N, 0, 0, 0, 5.

(2) Key-in G, 0, 1.

(3) Depress the WR key after keying-in X0.5.

(4) Depress the WR key after keying-in Y-0.2

(5) Depress the WR key after keying-in Z.

(6) Key-in F, 1, 0, 0.

(7) Depress the EOB key.

(8) Depress the INSRT key.

Then, block of N0005 G01 X1.200 Y0.800 Z0.600 F0.600 F100; will be stored in the part program.

7. OPERATION PROCEDURE

7.1 INSPECTION BEFORE TURNING ON POWER

Make sure that the front and rear doors of the control are firmly closed. The control employs a totally-enclosed, dustproof enclosure to shut out surrounding air. If the door is open, lock it by turning two door lockers with a large screwdriver. In addition, inspect the machine referring to the machine tool builder's manual.

7.2 TURNING ON POWER

1. Check to see that the main power is supplied for the control.
2. Depress the POWER ON pushbutton on the operator's panel, and the control power is supplied and then the cooling fans will start running. Make sure that air blows out from the exhaust ports of the upper side of the control.
3. Depress the POWER ON pushbutton again to turn on the servo power supply. When the machine is ready to operate, READY lamp lights.
4. If READY lamp does not light, detect and eliminate the cause according to the alarm code displayed. Refer to 4.3.12 DISPLAYING ALARM CODE.

7.3 MANUAL OPERATION

When the MODE SELECT switch on the machine control station is set to RAPID, JOG, STEP or HANDLE position, the machine can be operated manually.

Operation in RAPID Mode

1. Set MODE SELECT switch to RAPID.
2. Select the speed using RAPID TRAVERSE RATE OVERRIDE switch.

   Speed setting range: 100% - 50% - 25% - F0
3. Push JOG button to select the axis and direction of movement. The machine moves at the specified speed while the JOG button depressed.

Manual operation is defined as the operation in RAPID, JOG, STEP, or HANDLE.

Operation in JOG Mode

1. Set MODE SELECT switch to JOG.
2. Adjust the feedrate to the desired setting with JOG FEEDRATE switch (Up to 32 steps).
3. Push JOG button to select the axis and direction of movement. The machine moves at the specified speed while the JOG button depressed.

Operation in STEP Mode

1. Set MODE SELECT switch to STEP.
2. Select the move amount per step using MANUAL PULSE MULTIPLY switch.

   (Move amount setting range)

   Metric: 0.001 - 0.01 - 0.1 - 1.0 - 10.0-100.0 mm/step

   Inch: 0.0001 - 0.001 - 0.01 - 0.1 - 1.0-10.0 inch/step
3. Depress JOG button to select the axis and direction of movement. The machine moves by the move amount per step each time the button is depressed.

Operation in HANDLE Mode†

The control with HANDLE dial† can permit the operation described below.

1. Set MODE SELECT switch to HANDLE.
2. Select the axis with HANDLE AXIS switch.
3. Select the move amount of the machine corresponding to one scale of HANDLE dial using MANUAL PULSE MULTIPLY switch.

   Metric: 0.001 - 0.01 - 0.1 (mm per graduation)

   Inch: 0.0001 - 0.001 - 0.01 (inch per graduation)

   NOTE: "X1000" or "X10000" is regarded as "X100."
4. Rotate HANDLE dial.

   Turning the dial clockwise:
   The machine moves in the positive direction.

   Turning the dial counterclockwise:
   The machine moves in the negative direction.

7.4 PREPARATION FOR STORED LEADSCRE ERROR COMPENSATION AND STORED STROKE LIMIT +

Return to Reference Point

With an NC equipped with the stored leadscrew error compensation or the stored stroke limit functions, either of the following two reference point return motions must be performed after switching on the power supply and before starting automatic operation.

a. Manual return to reference point (See 6.2.1)

b. Execute G91 G28 X0 Y0 Z0 ; in the MDI mode.

   This procedure is to teach the reference point to the control, since doing so is necessary because both pitch error compensation and stored stroke check are performed with reference to the reference point.

   Checking Parameter #6006$_{D1}$

   When the control is equipped with the pitch error compensation function or the stored stroke limit function, set this parameter to "1." With the parameter #6006$_{D1}$ set to "1," a return to the reference point is required before starting cycles, alarm codes (001 - 004 "reference point return incomplete") are displayed, if the CYCLE START key is pushed without making a reference point return immediately after turning on the power supply. Be sure to perform the operation for return to reference point.

7.5 PREPARATION FOR AUTOMATIC OPERATION

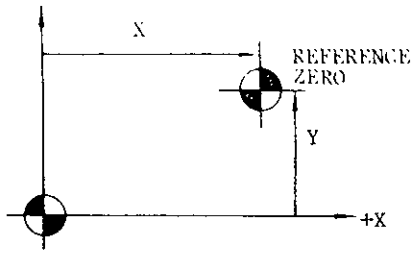
The machine must be positioned properly according to the part program prior to the start of automatic operation. After positioning the absolute coordinate system for the machining must be set properly by manual operation or programming.

1. When G92 is not programmed in a tape or memory.

   · Return the machine manually to the reference point.
   (Refer to 6.2.1 MANUAL RETURN TO REFERENCE POINT.)

   · The G92 command according to the part program should be executed by MDI.

   G92 X··· Y··· Z··· ;



ABSOLUTE ZERO POINT (0, 0)

Fig. 7.1

If "G92 X0 Y0 Z0 ;" setting is required, the coordinate of each axis can be set to "0" easily using ORG key.
Refer to 4.1.9 ORG KEY.

EXAMPLE

```
EOR ;
N1 G00 X··· Y··· Z··· ;
 .
 .
 .
```

Fig. 7.2

2. When G92 is programmed in a tape or memory.

   When the program requires G92 to be executed at the reference zero, return the machine to the reference point by manual return to reference point to reference zero.

EXAMPLE:

```
EOR ;
N1 G92 X··· Y··· Z··· ;
 .
 .
 .
```

Fig. 7.3

3. When G28 and G92 are programmed.

   When the program begins with G28 and with G28 and G92, move the machine manually into the area where return to reference point can be performed.

EXAMPLE:

```
EOR ;
N1 G28 X··· Y··· Z···;
N2 G92 X··· Y··· Z···;
  .
  .
  .
```

Fig. 7.4

7.6 OPERATION IN TAPE AND MEMORY MODE

1. Make sure that NC ALARM lamp is not illuminated. If illuminated, detect and eliminate the cause by the indication of alarm code. Refer to 4.3.12 DISPLAYING ALARM CODE.
2. Check and correct the stored offset values, and then put the machine in the correct start point.
3. Set the switches on the control station of machine to the proper positions.
   - MODE SELECT switch
   - SINGLE BLOCK toggle switch
   - RAPID TRAVERSE RATE OVERRIDE switch
   - MANUAL ABSOLUTE toggle switch
   - OPTIONAL BLOCK SKIP toggle switch
   - OPTIONAL STOP (M01) toggle switch
   - DRY RUN toggle switch
   - FEEDRATE OVERRIDE & JOG FEEDRATE switch
4. Set the punched tape onto the tape reader. In MEM mode, this opration is not required.
5. Depress RESET key on the control station. Then LABEL SKIP lamp will be illuminated and the memory will be rewound.
6. Depress CYCLE START button to give a Cycle Start to the system.
7. When the Feed Hold is required for the machine during the system operation, depress FEED HOLD button.
8. If the unexpected event occurs in the system, immediately depress EMERGENCY STOP pushbutton.

7.7 MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION

1. Stop the automatic operation temporarily by depressing FEED HOLD pushbutton or by setting SINGLE BLOCK switch to ON position.
2. Record the current positions of each axis on a paper using the current position display operation.
3. Set MODE SELECT switch to manual operation mode (HANDLE, JOG or RAPID), and the machine can be manually operated.
4. Return the machine manually to the recorded positions.
5. Set MODE SELECT switch to the interrupted automatic-mode (TAPE, MDI or MEM).
6. Depress CYCLE START pushbutton, and the machine will resume the automatic operation.

NOTES:

- Where MODE SELECT switch is changed without depressing FEED HOLD pushbutton.
  a. When the automatic-mode (TAPE, MDI or MEM) is changed to the manual-mode (HANDLE, JOG or RAPID), the machine rapidly slows down and stops.
  b. When the automatic-modes are changed the machine is stopped at the block end.
- Where the machine is restarted by depressing CYCLE START button, the tool path shifted due to manual operation will be changed by ON-OFF operation of MANUAL ABSOLUTE switch. Refer to 6.1.24 MANUAL ABSOLUTE SWITCH.

  In manual operation mode, when the CYCLE START button is depressed after wiring F, M, S, T or B† code by use of the same procedure as that of MDI operation, the command becomes effective and is executed as soon as written. This procedure is used to add new data to an active buffer. However, M00, M01, M02, M30 and M90 to M99 cannot be written.

7.8 AUTOMATIC OPERATION IN MDI MODE

1. Set MODE SELECT switch to MDI position.
2. Write up to 10 blocks of data by MDI operation, and execute by pressing CYCLE START. Refer to 4.3.3 Writing in Blocks and Displaying Contents by MDI.
3. Depress CYCLE START button, and automatic operation can be executed in MDI mode.

7.9 MDI OPERATION INTERRUPTING AUTOMATIC OPERATION

To modify the block data after interrupting operation in TAPE or MEM mode, the following operation should be done after interrupting the operation.

1. Turn on SINGLE BLOCK switch, and the operation is interrupted after the completion of the block being executed. At the same time, the next blocks of data may be read in advance.

2. Display the data on CRT DISPLAY according to 4.3.2 DISPLAY OF COMMAND DATA, and check it.

3. Set MODE SELECT switch to MDI operation.

4. Write the data referring to 4.3.3 Writing in Blocks and Displaying Contents by MDI. Execute the data by depressing the CYCLE START button.

5. Set back MODE SELECT switch to the interrupted automatic mode (TAPE or MEM).

6. Return SINGLE BLOCK switch to OFF position.

7. Depress CYCLE START button, and TAPE or MEM operation can be continued.

NOTES:

- Writing data by MDI cannot be executed in tool radius compensation modes (G41, G42) because two – four blocks are read ahead.

- In manual operation mode, when the CYCLE START button is depressed after writing F, M, S, T or B† code by use of the same procedure as that of MDI operation, the command becomes effective and is executed as soon as written. This procedure is used to add a new data to an active buffer. However, M00, M01, M02, M30 and M90 to M99 cannot be written.

- Writing data by MDI should not be performed in canned cycle modes (G73, G74, G76, G77, G81 to G89). The machine may not operate properly.

- Excepting in tool radius compensation and canned cycle modes, MDI operation is possible.

7.10 PREPARATION FOR TURNING OFF POWER

1. Make sure that the machine is at standby and CYCLE START lamp is extinguished.

2. Check to see that NC ALARM is not indicated on CRT. If alarm is displayed, detect the causes of displayed alarm code and eliminate them. Refer to 4.3.12 DISPLAYING ALARM CODE.

3. Inspect the machine referring to the machine tool builder's manual.

7.11 TURNING OFF POWER

1. Depress EMERGENCY STOP pushbutton to turn off the servo power supply.

2. Depress POWER OFF pushbutton on the operator's panel to turn off the control power supply.

3. Cut off the main power supply from the control.

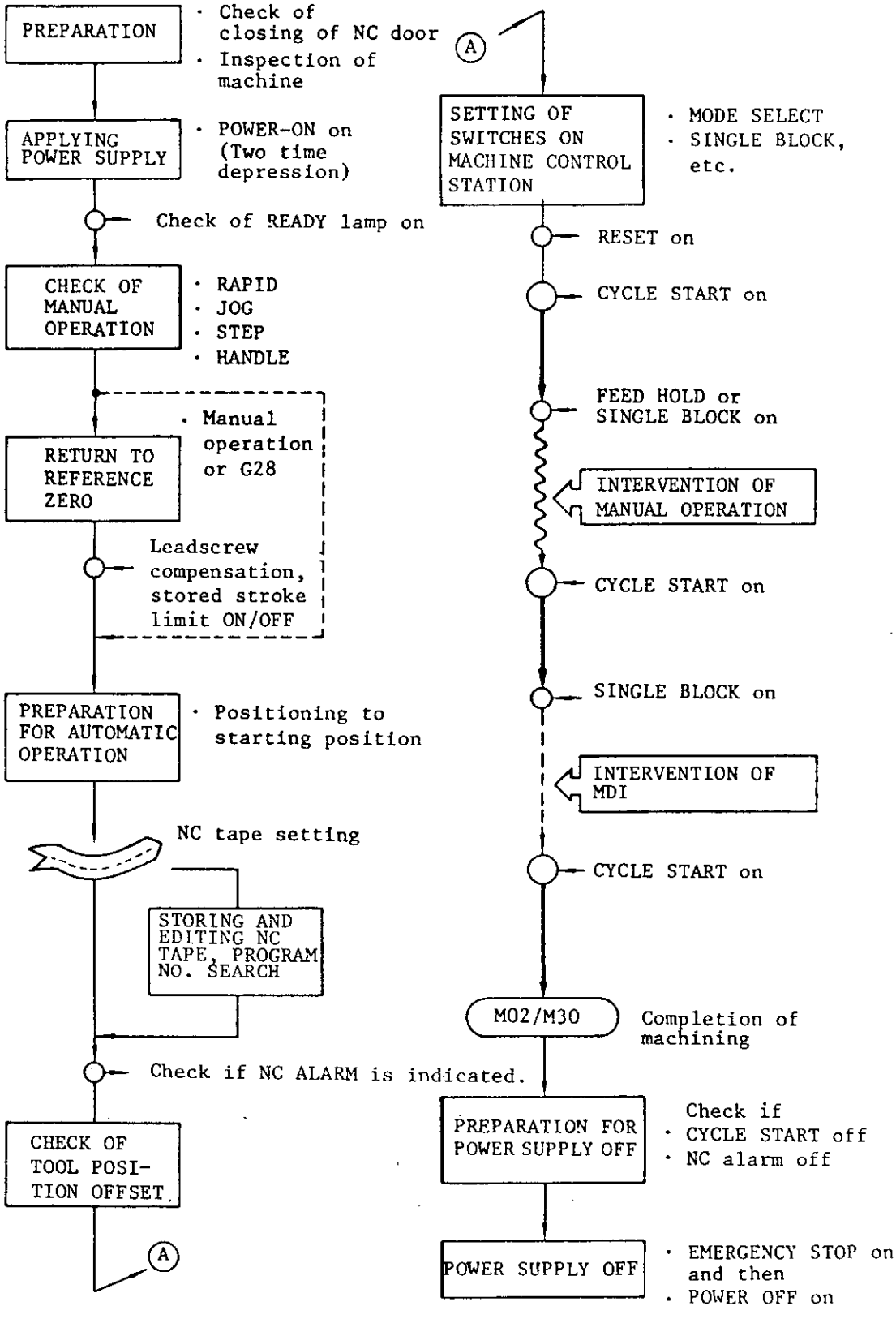


Fig. 7.4 Operating Procedure

8. MAINTENANCE

8.1 ROUTINE INSPECTION SCHEDULE

The following table shows the minimum requirements to be observed for maintenance according to time in order to keep the equipment in optimum condition for extended period.

Table 8.1.0 Inspection Schedule

Items		Frequency	With the system-off	With the system-on	Remarks
Tape reader	Cleaning of reading head	Daily	○		Including light source part.
	Cleaning of tape tumble box	Weekly	○		
	Lubricating of tension arm shaft end	As required	○		
Control panel	Tight closing the doors	Daily	○		
	Checking for loose fit and gaps of side plates and worn door gaskets	Monthly	○		
Servomotor and DC motor for spindle	Vibration and noise	Daily		○	Feel by hand, and do the audible inspection.
	Motor contamination and breakage	Daily or as required	○	○	Inspect visually.
	Clearance of ventilation openings		○	○	Inspect mainly spindle DC motor.
	Burned spots, cracks, wear, and pressure of brushes	Every three months	○		Check the length of brushes.
	Roughened commutator surface		○		Check dark bar, threading and grooving of commutator.
	Dirt in interior of motor		○		Clean with compressed air.
Battery		Daily	○	○	See if alarm for BATTERY is displayed on CRT screen.

Except for those checks which can be and made with the NC energized, such as checks for external cleanliness for vibration and for noise, be sure to turn off the power supply to the NC before starting to undertake routine maintainance service.

For this, turning off the power supply by pushing the POWER OFF button on the NC operator's panel is not sufficient, because after this button is pushed, still several areas in the housing are energized, and are potentially dangerous.

8.1.1 TAPE READER

1. Cleaning head of tape reader

   A. Remove tape rubbish and dust on the glass with a blower brush. If the glass is stained with oil or oily dust, wipe it using a gauze or soft cloth with absolute alcohol. Also clean the tape guide and the tape retainer.

   B. Remove the dust, if any, on LED (light source) with a blower brush.

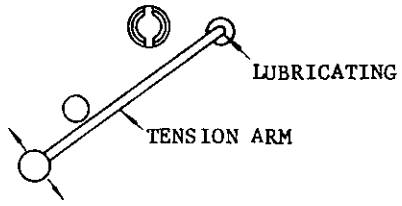
2. Cleaning of tape tumble box

   A. Clean the polyester leading tape with a clean, soft cloth.

   B. Remove the tape outlet cover (See Fig. 5.1.3) by loosening two mounting screws and clean the bottom of the tape tumble box with cloth or brush.

3. Lubricating of tension arm shaft†

   For the control with 6 inch or 8 inch diameter reels, lubricate the shaft end of tension arm, when the tension arm does not move smoothly.



(In the case of 8 inches diameter reel)

Fig. 8.1

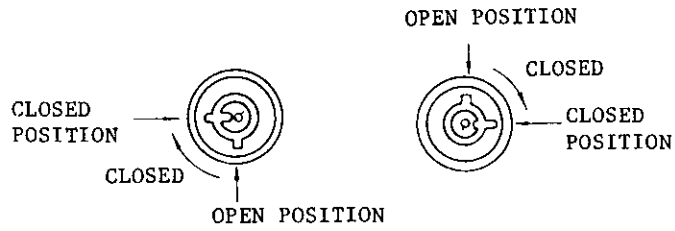
NOTE: When trouble occurs in feeding or winding tape with 8 inch diameter reels, open the front door and brush away dust around the photo-coupler by using a blower brush.

8.1.2 CONTROL PANEL

The control panel is a dustproof, sheet-steel enclosure with gasketed doors.

A. Front and rear doors of the control should be shut tightly, even if the control is not operating.

B. When inspecting the control with the door open, upon completion of inspection, lock the door by turning two door locks with the key attached to the control panel. Turning direction of door locks is as follows.



With the door lock on right side of the door.

With the door lock on left side of the door.

Fig. 8.2

NOTE: If the optional door interlocking switch is provided, opening the door shuts off the main power supply and stops all operations.

C. Check gaskets on the rims of front and rear doors.

D. See if the inside of enclosure is dusty. Clean it, if necessary.

E. Check for any opening in the door base with the doors shut tightly.

8.1.3 SERVOMOTOR AND DCMOTOR FOR SPINDLE

1. Vibration and noise.

   Vibration can be checked by resting the hand on the motors, and for noise, using a listening stick is recommended. If any abnormality is found, contact maintenance personnel immediately.

2. Motor contamination and impairment.

   Check the motor exterior visually. If dirt or damage should be observed, inspect the motor by removing the machine cover. Refer to the machine tool builder's manual.

8.1.3 SERVOMOTOR AND DC MOTOR FOR SPINDLE (CONT'D)

3. Clearance of ventilation window blockage

   Check the ventilation window of DC spindle motor. If it is clogged with dust or dirt, inspect DC spindle motor removing the machine cover. Refer to the machine tool builder's manual.

Inspection of commutators and brushes is essential for maintaining the excellent performance of the control. Inspection work to be executed is described in the following three items.

Quarterly Inspection of Commutators and Brushes

The carbon dust from brushes, accumulated around the commutator, inside the motor, may cause motor troubles such as the layer short of armature and the flashover of commutator. In the worst case, it may lead to fatal damage. To avoid this, be sure to have an inspection on the commutators and brushes at least every three months.

Be sure to turn off the power supply to the NC before starting to check the brushes and motor interior.

For this, turning the circuit breaker on the power supply unit (DCP UNIT) for the servo control unit (CPCR-MR-K) off is not sufficient. To prevent electric shocks and shorting, be sure to shut off the supply of power to the NC.

4. Carbon brushes

   A. Under normal operating conditions, brush wears by 2 to 4 mm per 1000 operating hours. If wear is excessive, check to see if oil has contaminated commutator surface, or if abnormal overcurrent flow through motor circuit.

   B. When brush length becomes shorter than those shown below, replace the brush with a new one.

      Minertia motor junior series: 6 mm or below

      DC motor for spindle: 17 mm or below

   C. If either of brush, or pigtail is broken, brush assembly must be replaced as a whole unit.

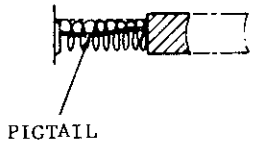


Fig. 8.3

NOTE: When replacing the brush assembly, consult the company.

5. Commutator surface

   A. Visually check surface roughness of the commutator through inspection window.

      After 100 to 200 operating hours, the commutator should take on a polished light brown or chocolate color. The motor has developed an ideal commutator film and needs no attention other than to be kept clean.

   B. See if a blackened bar, threading (or grooving) is on the commutator. If any of the above is observed, investigate the cause of trouble.

      Threading or grooving on the commutator surface may be due to too small motor load. Blackened bar is a result of carbon dust in commutator slots, or accidentally produced sparkings. If the carbon dust is a cause of blackened bar, wipe the commutator with a clean dry cloth to smooth the surface. If sparking occurs, contact the maintenance representative.

6. Motor inside (dirty)

   A. Visually check the motor interior through inspection window.

      The dried carbon dust will not affect motor running, but it is recommended that the inner parts such as commutator, brushholders and brushes be cleaned with a dry compressed air (air pressure: 2-4 kg/cm^2).

   B. Where oily carbon dust exists inside the motor due to poor oil seal or defective enclosure, contact Yaskawa.

7. Servomotor with oil seal

   As the life expectancy of oil seal and brush is 5000 hours (about five years), the inspection and maintenance by the company should be done every 5 years. If possible, yearly inspection taking less than 8 hours is recommended.

8.1.4 BATTERY

Make sure that "BAT" or "A/B" on the right-low position of CRT screen is not displayed. If it is displayed, inform maintenance personnel. The battery must be replaced with a new one within a month.

The control with a bubble memory board (optional) does not require a battery.

8.2 BATTERY REPLACEMENT

The battery is used as power source for memory in order to prevent programming data stored in memory, such as parameter, tool offset and part program from erasing.

When the battery is discharged after a long period of use, "BAT" or "A/B" is blinked on CRT screen to give warning for replacement. On such occasions, the battery must be replaced within 30 days. When replacing, never remove the old battery with power off, otherwise the data stored in memory may be cleared.

Replacing Procedure

1. Depress POWER OFF pushbutton on the operator's station.

2. Open the front door of the control. The battery of the memory(printed circuit) board can be seen on the CPU module which is mounted on rear of the front door.

3. Depress POWER ON pushbutton. Where the control is equipped with a door interlock switch, pull it out by hand. The power can be turned on, with the door open.

4. Check to see if 1LED on memory board is illuminated. Fig. 8.2.1 shows the arrangement of LED and the battery. If illuminated, replace the battery "660S" with new one.

   660S ··· Type: JZNC-GBA02

5. With the control power turned on, connect the receptacles of the new battery in to the plugs (1CN or 2CN) on memory circuit board, and LED will be turned off. See Fig. 8.2.1. If LED is still illuminated, it is due to the improper insertion of battery connectors, or defective battery.

   IMPORTANT: Two plug stations, 1CN and 2CN (or 2CN and 3CN) are connected together with common leads. When an old battery is replaced with a new one, connect the new battery first to the plug station not occupied, then remove the receptacles of the old battery.

6. Depress POWER OFF pushbutton.

7. Remove the mounting screw of old battery, and then replace the battery with new one. In this case, pass the battery lead through gaps between the battery and the battery clamp, and use care not to contact the lead with memory circuit board.

   Where the control is provided with a door interlock switch, push it back in place since power cannot be turned on with the door open.

8. Tightly close the front door.

9. Depress POWER ON pushbutton.

10. Make sure that "BAT" or "A/B" on CRT screen goes off.

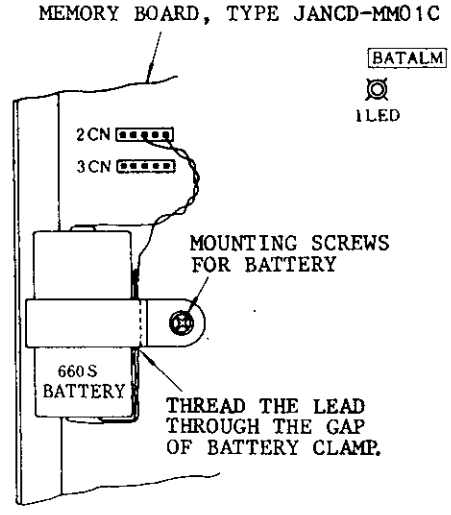


Fig. 8.4 Arrangement of LED and Battery

NOTES:

1. While battery is being replaced, exercise utmost care to prevent contaminants from entering the control, and accomplish the work as quickly as possible.
2. Use special care to prevent water, oil, or dust, to adhere to the devices (printed circuit board, connectors, cables, etc.) inside the control.
3. Never leave any screws or washers in the control.

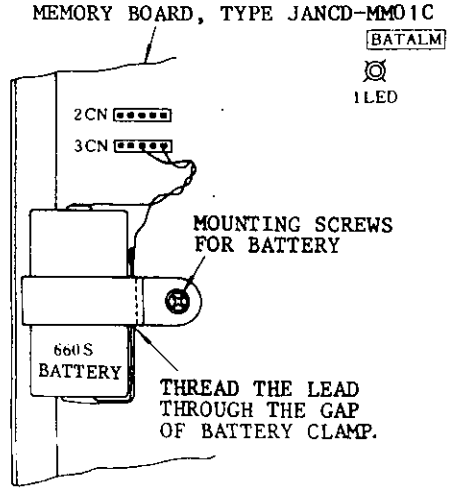


Fig. 8.5 New Battery Mounted

8.3 POWER CIRCUIT BREAKER AND FUSES

The power circuit breaker and fuses are provided for the safety of the control: a power circuit breaker and four fuses are in the composite control power supply unit; two fuses in the power input unit; a fuse and circuit breaker in the servo control power unit. If the breaker trips or the fuses are blown off, consult the maintenance personnel.

8.3.1 CIRCUIT BREAKER OF COMPOSITE CONTROL POWER SUPPLY UNIT

The breaker trips due to overload of composite control power supply unit. Reset the control according to the following procedure. See Fig. 8.3.2 Breaker and Fuses of Composite Control Power Supply Unit.

1. Inspect for the cause of overload, and remove, if any.
2. When the breaker trips, ON-OFF lever of the breaker is in OFF position. Put the lever in ON position.
3. Depress POWER OFF pushbutton on the operator's station. Reset the control and then turn on the power.
4. If the breaker trips again, contact Yaskawa.

8.3.2 FUSES OF COMPOSITE CONTROL POWER SUPPLY UNIT

If any of the fuses are blown, all power supplies are turned off. Reset the control according to the following procedure.

1. Remove the blown fuse that is indicated with white mark coming on the fuse casing window.
2. Inspect for the cause of failure in the control power supply, I/O interface and servo unit, and remove, if any.
3. Replace the blown fuse with new one.
4. Depress POWER OFF pushbutton on the operator's station to reset the control and then turn on the power.
5. If the fuse is blown again, contact Yaskawa.

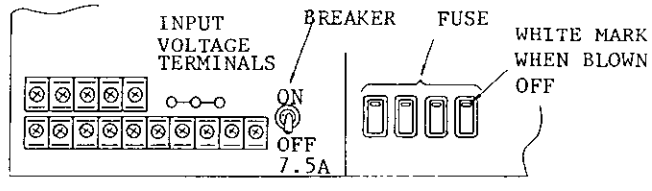


Fig. 8.6 Breaker and Fuses of Composite Control Power Supply Unit

8.3.3 FUSES OF POWER INPUT UNIT

If any of the glass-tube fuses are blown, all power supplies are turned off and 1LED of the power closing unit goes off. To reset the control, use the following procedure.

1. Turn off all power supplies by putting molded-case circuit breaker in OFF position. Never touch the unit until the breaker is turned off.
2. Inspect for the cause of trouble, and remove, if any.
3. Replace the blown fuse with new one.
4. Make sure that 1LED goes on when the breaker is turned on.
5. Turn on the power.
6. If the fuse is blown again, contact Yaskawa.

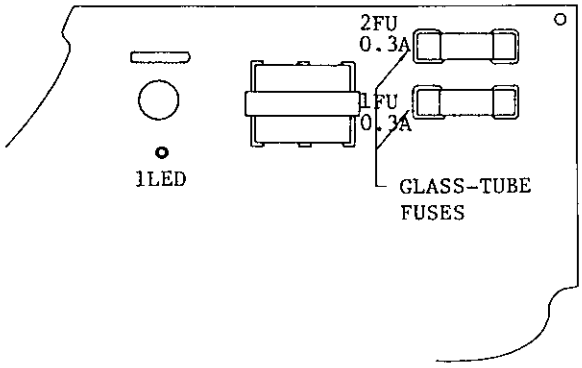


Fig. 8.7 Fuses of Power Closing Unit

8.3.4 FUSE FOR SERVO CONTROL POWER UNIT

When the servo control power unit (called DCP unit hereinafter) is overloaded due to short-circuiting of control unit power line, etc., the fuse for DCP is blown-out. The "POWER" Led will be extinguished to indicate that the fuse is blown. Take the following measure for the blown-out fuse.

(1) Depress the POWER OFF button to turn off the power supply, and then, depress it again to turn off the power to NC.

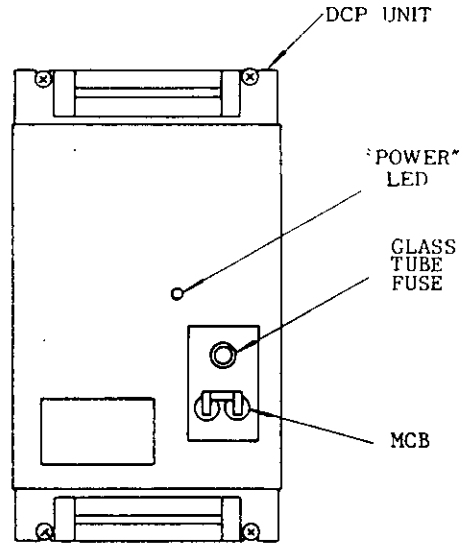
(2) Find the cause of the fuse blowing, and eliminate it.

(3) Replace the blown fuse with the new one of the same rating.

(4) Depress the POWER ON button to turn on the power for NC and depress it again to turn on the servo power. Check to be sure that "POWER" Led lights up.

(5) If the fuse is blown off again, contact our service personnel.

NOTE: DCP unit size will differ with the capacity of 10 A, 15 A, 30 A, 60 A.



8.3.5 CIRCUIT BREAKER FOR SERVO CONTROL

When the DCP unit is overloaded through the shorting of the output circuit, faults of the DCP unit itself, etc., this circuit breaker trips to disconnect the main circuit.

In this case, an alarm "350:OL" is displayed on the CRT. (350:OL may be displayed also by other causes.) When this circuit breaker is tripped to the OFF state, take the following measure.

(1) Push the POWER OFF button to turn off the power supply, and then, stop the supply of power to NC.

(2) Find the cause of the tripping of the circuit breaker, and eliminate it.

(3) Push up the breaker lever to the ON position.

(4) Supply power to NC, and then, push the POWER ON button twice.
The servo circuit is energized, and is ready for operation.

(5) If the circuit breaker is tripped again, notify our service department.

NOTE: When the circuit breaker is manually turned off under power, not turning off automatically by overloading, the alarm "310:SERVO OFF" is displayed on the CRT. The DCP unit can be brought to the operating conditions by the same measures as above, also in this case.

8.4 THERMAL OVERLOAD RELAY OF SERVO UNIT

The servo control unit has the function of detecting the following alarm states.

	Alarm No.
. Fuse (main circuit) blown	331(X) 332(Y) 333(Z) 334(4)
. Overload	351(X) 352(Y) 353(Z) 354(4)
. TG error	391(X) 392(Y) 393(Z) 394(4)

8.4.1 FUSE BLOWING (Alarm No. 331, 332)

When the main circuit of the servo control unit is shorted or when the servo control unit itself becomes faulty, the fuse in the unit will be blown, and the following alarm Nos. will be displayed on the CRT.

331: FUSE(X) ··· for X-axis

332: FUSE(Y) ... for Y-axis

333: FUSE(Z) ... for Z-axis

334: FUSE(4) ... for 4Th axis

When a fuse is blown, and the alarm No. 331 or 332 is displayed, do not attempt to take measures, but the user should immediately notify our service department.

8.4.2 OVERLOAD (ALARM NO. 351, 352)

The servo control unit is provided with electronic thermal relays respectively and independently for the X and Z axes, and they trip under the following conditions.

- Programs involving excessively heavy cuts are executed.
- Programs involving excessively frequent speed changes are executed.
- Frictions in the machine system become abnormally large.

When the electronic thermal relay trips, the servo power supply is turned off, and the following alarm Nos. are displayed on the CRT.

351: OL(X) ··· X-axis overload

352: OL(Y) ... Y-axis overload

353: OL(Z) ... Z-axis overload

354: OL(4) ... 4Th axis overload

When this is the case, take the following measures.

(1) Push the POWER OFF button to turn off the power supply, and then, stop the supply of power to NC.

(2) Find the cause of the overloading. For example, the cause may be eliminated through modifications of the part program, or by the elimination of abnormally large load on the machine.

(3) Supply power to the NC, and push the POWER ON button to turn on the power supply and make the system ready for operation. However, since the servo motor requires approximately 30 minutes to cool down after being overloaded to the extent of tripping the electronic thermal relay, wait at least 30 minutes before starting to operate NC.

(4) If the electronic thermal relay trips, notify our service department.

8.4.3 TG ERROR (ALARM NO. 391, 392)

The servo control unit can detect the following alarm states.

- Wire breaking in the tachometer generator (TG) or overspeeding
- Main circuit overcurrent
- Main circuit overvoltage

When any of these faults occurs, the following alarm Nos. will be displayed on the CRT.

391: TG ERROR (X)

392: TG ERROR (Z)

393: TG ERROR (Z)
394: TG ERROR (4)

When this is the case, take the following measures.

(1) Push the POWER OFF button to turn off the power supply, and then,stop the supply of power to NC.

(2) Find the cause of the alarms, and eliminate it. In this case, if the fault conditions are reported to our service department, we will be able to give advice on troubleshooting.

(3) Resupply power to NC, and then, push the POWER ON button to make the unit ready for operation.

(4) If TG ERROR is displayed again, notify our service department.

8.5 OTHERS

Those special housing type controls, with which all the power sequence control circuits are converted to the NC area, are sometimes provided with MCBs which can be turned on and off externally.

Generally, when these MCBs are turned off, the power supply to the NC is stopped. For details, refer to the manual of the machine tool builders.

8.6 TROUBLE CAUSES AND REMEDIES

8.6.1 ON-LINE DIAGNOSTICS

On-line diagnostics are implemented to locate a trouble quickly and protect the machine against malfunctions. Shown below are the displaying functions executed by the control being on-line and machining.

- Display of three-digit alarm code including a code showing an axis in error.
- Display of four-digit status code including a function code showing M, S, T, V, DWL.
- Input/output signal display

These displays can be made at any time, while the machine is in automatic operation, or at standby.

8.6.2 ALARM CODES AND REMEDIES

Where "ALM" or "A/B" on CRT screen is blinking and the machine stops, depress the ALM key. Then alarm code and message will be displayed on CRT screen. Alarm codes "800," "810," "820," "830" and "840" are displayed as soon as the corresponding error occurs.

For the remedies for trouble causes represented by alarm codes, see APPENDIX 5 LIST OF ALARM CODES on the last part of this manual.

8.6.3 INPUT/OUTPUT SIGNALS

To clear up the causes indicated by alarm codes, check the input/output signals on the CRT screen.

Input/output signals are divided into standard and custom-built ones, and displayed by specifying the corresponding diagnostic number with keys on the operator's station.

Standard signals are included in every type of YASNAC MX1. Custom-built signals are provided for optional machine interface equipped with some type of YASNAC controls.

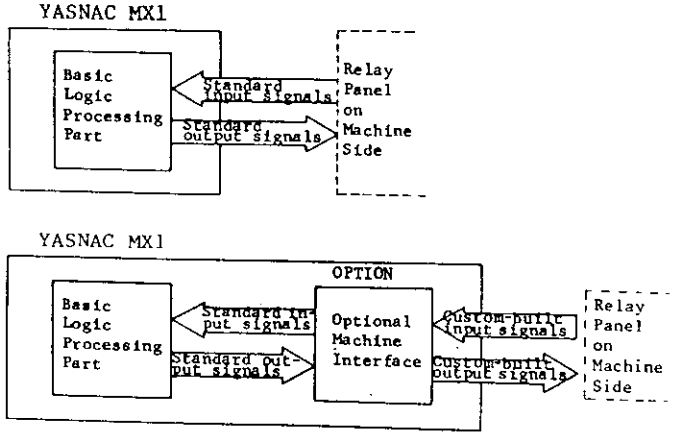


Fig. 8.8

8.6.3 INPUT/OUTPUT SIGNALS (CONT'D)

To display input/output signals, proceed as follows.

1. Depress the DGN key.

   A page containing the diagnostic number specified previously occurs on the CRT screen. The input/output signals are shown in "1," "0" and hexadecimal digit.

   "1": contact close
   "0": contact open

2. Key-in the diagnostic number to be displayed.

3. Depress the cursor [↑] or [↓] key to page the keyed-in diagnostic number on the screen.

   · By depressing the cursor [CURSOR ↓] key, a cursor on the screen moves to the next diagnostic number (line). When down to the last lower line, the next page is displayed on the screen.

   · By depressing the cursor [↑ CURSOR] key, the cursor moves to the previous line. When up to the most upper line, the previous page is displayed.

   · By depressing the page [PAGE ↓] key, the next page appears on the screen.

   · By depressing the page [↑ PAGE] key, the previous page appears.

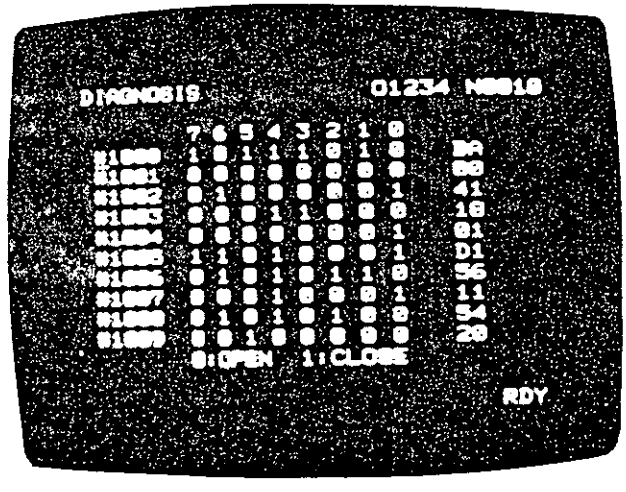


Fig. 8.9 Example of Input/Output Signal Display

8.6.4 BEFORE MAINTENANCE CALL

If the cause of trouble cannot be found by using alarm codes or I/O signals (described in 8.6.1 to 8.6.3), or correct action for the trouble cannot be taken, record the following items, and notify the company as soon as possible.

- Alarm codes and the accompanying data with them.
- The types and characteristics of the troubles.
- The operational procedures just before the trouble occured and number of applied tape.
- Whether the trouble recurs each time, the operation is repeated after depressing the RESET key.
- Date and time when the trouble occured.
- Name of the discoverer of the trouble and the operator.

If trouble occurs, keep the control in the same condition until it can be checked by your Yaskawa representative. If the situation permits, avoid turning off control power, or depressing POWER OFF button.

APPENDIX – 1 LIST OF SETTING NUMBERS

Setting numbers are classified in the following three groups:

Table 1.1

Setting number	Group
No. 6000 to 6004	Setting by bit
No. 6200 to 6219	Setting by word
No. 6500 to 6599	Setting by double words

Appendix table 1-2 lists setting numbers and their functions.

Table 1.2 List of Setting Numbers

SET NO.	D7	D6	D5	D4	D3	D2	D1	D0	
#6000	"1":ISO "0":EIA TAPE CODE	"1":ON "0":OFF TV CHECK			"1":INV "0":NORM 4	"1":INV "0":NORM Z	"1":INV "0":NORM Y	"1":INV "0":NORM X	
						MIRROR IMAGE			
#6001	"1":ON "0":OFF BUZZER ON/OFF	"1":ON "0":OFF EIA/ISO AUTO-RECOGNITION	"1":ON "0":OFF STORED STROKE AT MLK	"1":ON "0":OFF Z-AXIS CANCEL	"1":ON "0":OFF ABS	"1":ON "0":OFF EDT LOCK	"1":ON "0":OFF SECOND STORED STROKE	"1":INCH "0":MM INPUT UNIT	Refer to $D_{2,3,4}$= $\#6006_{D3}$.
				INTERNAL SWITCH					
#6002	"1":ON "0":OFF AFL	"1":ON "0":OFF MLK	"1":ON "0":OFF OPT	"1":ON "0":OFF DRN	"1":ON "0":OFF BDT	"1":ON "0":OFF DLK	"1":ON "0":OFF START LOCK	"1":ON "0":OFF SBK	Refer to $\#6006_{D3}$.
				INTERNAL SWITCHES					
#6003	O_8	O_4	O_2	O_1	I_8	I_4	I_2	I_1	
		"0000": "0001":	FACIT SIO-1			"0000": "0001":	PTR SIO-1		
#6004								"1":OK "0":ERROR NO SKIP SIGNALS AT G31	

SET

SET NO.	Set value	Functions	SET NO	Set value	Functions
#6200		BD-1 ⎫ BREAK POINT			
#6201		-2 ⎭ SEQUENCE NO.			
#6202					
#6203					
#6204					
#6205					
#6206					
#6207					
#6208					
#6209					
#6210		G76/77 DWELL TIME "1"=1 msec			
#6211		G73(δ)=0.001mm (0.001inch)			
#6212					
#6213		G83(δ)=0.001mm (0.0001inch)			
#6214					
#6215					
#6216					
#6217					
#6218					
#6219					

Table 1.2 List of Setting Numbers (Cont'd)

SET NO.	Set value	Functions	SET NO.	Set value	Functions
#6500		SCALLING MULTI numerator	#6540		X
#6501		SCALING MULTI denominator	#6541		Y WORK COORDINATE SYSTEM 5
#6502			#6542		Z G58
#6503			#6543		4
#6504			#6544		
#6505			#6545		
#6506		G76/77 ANGLE DESIGNATION "1"=0.001deg	#6546		X
#6507		TLM BIAS VALUE	#6547		Y WORK COORDINATE SYSTEM 6
#6508		TLM DISTANCE TO ZERO POINT	#6548		Z G59
#6509			#6549		4
#6510		X STORED STROKE LIMIT 2	#6550		
#6511		Y G22	#6551		
#6512		Z (+) SIDE BOUNDARY VALUE	#6552		X
#6513		X	#6553		Y COORDINATE VALUES WHEN SKIP
#6514		Y G22	#6554		Z SIGNAL IS ON
#6515		Z (-) SIDE BOUNDARY VALUE	#6555		4
#6516		X	#6556		
#6517		Y WORK COORDINATE SYSTEM 1	#6557		
#6518		Z G54	#6558		
#6519		4	#6559		
#6520			#6560		F0 FEED
#6521			#6561		F1 FEED
#6522		X	#6562		F2 FEED
#6523		Y WORK COORDINATE SYSTEM 2	#6563		F3 FEED
#6524		Z G55	#6564		F4 FEED F1-DIGIT COMMAND
#6525		4	#6565		F5 FEED "1" = 0.1 mm/min
#6526			#6566		F6 FEED
#6527			#6567		F7 FEED
#6528		X	#6568		F8 FEED
#6529		Y WORK COORDINATE SYSTEM 3	#6569		F9 FEED
#6530		Z G56	#6570		
#6531		4	#6571		
#6532			#6572		
#6533			#6573		
#6534		X	#6574		
#6535		Y WORK COORDINATE SYSTEM 4	#6575		
#6536		Z G57	#6576		
#6537		4	#6577		
#6538			#6578		
#6539			#6579		

SET

SET NO.	Set value	Functions	SET NO.	Set value	Functions
#6580					
#6581					
#6582					
#6583					
#6584					
#6585					
#6586					
#6587					
#6588					
#6589					
#6590					
#6591					
#6592					
#6593					
#6594					
#6595					
#6596					
#6597					
#6598					
#6599					

APPENDIX - 2 LIST OF PARAMETER NUMBERS

Parameter numbers are classified in the following five groups:

Table 2.1

Parameter number	Group
#6005 to #6049	Setting by bit
#6050 to #6199	Setting by byte
#6220 to #6399	Setting by word
#6600 to #6699	Setting by double words
#7000 to #7095	Setting by byte for sequencer

Optimum data of parameters have been set according to machine performance and applications. For any modification of parameter data, consult the machine tool builder.

Data pertaining to parameters #6033 to #6049 must not be modified, for they have been incorporated as part of the system.

#6005	D_7	D_6	D_5	D_4	D_3	D_2	D_1	

D_7 1: Stores the H code for reset, G92 and G28.

0: Sets the H code to H00 for reset, G92 and G28.

D_6 1: Stores the G code in the 01 group for reset.

0: Sets the G code in the 01 group to G00 for reset.

D_5 1: Allows the current value display (universal) to be preset by the coordinate system seeeing command G92.

At this time, the [ORG] key is capable of Zero Setup.

0: Keeps the current value display (universal) from being preset by the coordinate system setting command G92.

At this time, the [ORG] key is incapable of Zero Setup.

D_4, D_3: Status of G codes at power on.

D_4	D_3	Initial status
1	0	Sets the G code in the 08 group to G44 on power application.
0	1	Sets the G code in the 08 group to G43 on power application.
0	0	Sets the G code in the 08 group to G49 on power application.

D_2 1: Sets the G code in the 01 group to G01 on power application.

0: Sets the G code in the 01 group to G00 on power application.

D_1 1: Sets the G code in the 05 group to G95 on power application.

0: Sets the G code in the 05 group to G94 on power application.

D_0 1: Sets the G code in the 05 group to G91 on power application.

0: Sets the G code in the 05 group to G90 on power application.

#6006	D7	D_6	D_5	D_4	D_3	D_2		D_0

D_7, D_6 Signs of S4-digit analog (SDA) output

D7	D_6	Sign	
1	1	Minus	Plus
1	0	Plus	Minus
0	1	Minus	Minus
0	0	Plus	Plus
		SINV signal off	SINV signal on

D5 1: Sets the least input increment to 0.01 mm (or 0.001 in., 0.01 deg.).

0: Sets the least input increment to 0.001 mm (or 0.0001 in., 0.001 deg.).

D_4 1: Checks to see if the spindle speed match signal (SAGR) is off upon transition from a rapid traverse block to a cutting feed block.

0: Provides no check on the spindle speed match signal (SAGR).

D_3 1: Enables the internal toggle switches.

0: Disables the internal toggle switches.

D_2 1: Enables dry run in response to the rapid traverse command.

0: Disables dry run in response to the rapid traverse command.

D_1 1: Causes an alarm ("001-004") upon cycle start when reference point return is not made after power application.

0: Causes no alarm.

NOTE: Set "1" when pitch error compensation or stored stroke limit is provided.

#6007		D_6		D_4	D_3	D_2		D_0

D_6 1: Employs the newly entered tool compensation value in place of the old value.

0: Adds the newly entered tool compensation value to the soterd value to establish another offset.

D_4 1: Provides output during rewinding.

0: Provides no output during rewinding.

D_3 1: Sets the least increment to 0.0001 in.

0: Sets the least increment to 0.001 mm.

D_2 1: Makes the spindle override 100% during tapping.

0: Does not make the spindle override 100% during tapping.

D_0 1: Establishes the 2nd prohibited area of the stored stroke limit outside the boundary.

0: Establishes the 2nd prohibited area of the stored stroke limit inside the boundary.

#6008							D1	

D1 1: Does not clear the common variables of #100 through #149.

0: Clears the common variables of #100 through #149.

Note: Some controls are not provided with the parameter #6008.

#6009					D_3	D_2	D_1	D_0

D_3, D_2, D_1, D_0

Specify the start direction of backlash compensation on the 4th-, Z-, Y- and X-axes, respectively, upon power application.

1: Minus direction

0: Plus direction

#6010					D_3	D_2	D_1	D_0

D_3, D_2, D_1, D_0

Specify the direction of reference point return on the 4th-, Z-, Y- and X-axes, respectively.

1: Minus direction

0: Plus direction

NOTE: The specification is effective for an axis with #6016 at "1."

#6012					D_3	D_2	D_1	D_0

D_3, D_2, D_1, D_0

Specify whether or not the plus-direction external deceleration signal is effective on the 4th-, Z-, Y- and X-axes, respectively.

1: Makes the plus-direction external deceleration signal effective.

0: Makes the plus-direction external deceleration signal ineffective.

#6013					D_3	D_2	D_1	D_0

D_3, D_2, D_1, D_0

Specify whether or not the minus-direction external deceleration signal is effective on the 4th-, Z-, Y- and X-axes, respectively.

1: Makes the minus direction external deceleration signal effective.

0: Makes the minus direction external deceleration signal ineffective.

#6014					D_3	D_2	D_1	D_0

D_3, D_2, D_1, D_0

Specify the direction of the G60 unidirection approach upper limit on the 4th-, Z-, Y- and X axes, respectively.

1: Minus direction

0: Plus direction

NOTE: The approach upper limit is set with #6062 to #6065.

#6015					D3	D2	D1	D0

D_3, D_2, D_1, D_0

Specify whether or not the automatic coordinate system setting is effective on the 4th-, Z-, Y- and X-axes, respectively.

1: Effective

0: Ineffective

NOTE: The automatic coordinate system is established with the following parameters:

Metric system; #6636 - #6639

Inch system; #6630 - #6633

#6016					D3	D2	D1	D0

D_3, D_2, D_1, D_0

Specify whether or not reference point return is effective on the 4th-, Z-, Y- and X-axes, respectively.

1: Makes reference point return effective.

0: Makes reference point return ineffective.

#6017	D7	D6	D5	D4	D3	D2	D1	D0

D_7 - D_0

Specify whether or not a hole is to be made on channels 8-1, respectively, in a code corresponding to symbol "#" (used with user macro) in the EIA code.

1: Hole

0: No hole

Example:
D_7 - D_0 = 01001001

The code with holes on channels 7, 4 and 1 is considered equivalent to symbol "#" in the EIA code. No code for use by the unit can be set.

NOTE: The specification of D_7 - D_0 =00000000 assumes that symbol "#" is not used in the EIA code.

#6018	D7	D6	D5	D4		D2	D1	D0

D_7 1: Provides dwell at hole bottom in the canned cycles of G76 and G77.

0: Does not provide dwell at hole bottom in the canned cycles of G76 and G77.

NOTE: The dwell time is set with #6210.

D_6 1: Establishes M03 for G74 and M04 for G84 as the M code for output at hole bottom in the canned cycles of G74 and G84.

0: Reverses the M code in effect before the canned cycles for output at hole bottom in the canned cycle of G74 and G84.

Before canned cycle	Hole bottom
M03	M04
M04	M03

NOTE: This specification is effective when D_4 = 0 in #6018.

D_5 1: Rotates the spindle forward and in reverse, outputting M05 at hole bottom in the canned cycles of G74 and G84.

0: Rotates the spindle forward and in reverse, not outputting M05 at hole bottom in the canned cycles of G74 and G84.

NOTE: This specification is effective when D_4 = 0 in #6018.

D_4 1: Outputs a read-only signal (SSP, SRV, OSS) in the canned cycles.

0: Outputs the M code in the canned cycles.

D_2 1: Causes the X and Y movement in the canned cycles to follow the G code in group 01 (G02 and G03 regarded as G01).

0: Causes the X and Y movement in the canned cycles to stay in rapid traverse.

D_1 1: Outputs the FMF signal twice in a canned cycle.

0: Outputs the FMF signal once in a canned cycle.

D_0 1: Outputs the external operation signal EF at the end of positioning by G81 (G81 being external operation function).

0: Does not output the external operation signal EF at the end of positioning by G81 (G81 being canned cycle).

PARAMETER NUMBERS AND THEIR CONTENTS (CONT'D)

#6019			D5	D4		D2	D1	D0

D_5 1: Enables dry run at thread-cutting (G33).

0: Disables dry run at thread-cutting (G33).

D_4 1: Employs the feedrate set in parameter #6232 for the skip function command (G31).

0: Employs the F code command as the feedrate for the skip function command (G31).

D_2, D_1

Specify the tool shift direction in the canned cycles of G76 and G77 (effective when #6019 = 0).

D2	D1	Shift direction
1	1	-Y
1	0	+Y
0	1	-X
0	0	+X

D_0 1: Allows the tool shift direction in the canned cycles of G76 and G77 to be specified in setting #6506 (specifiable in the +X direction in increments of 0.001; the shift being cutting feed).

0: Allows the tool shift direction in the canned cycles of G76 and G77 to be specified in D_2 and D_1 of parameter #6019 (specifiable only axially).

#6020				D3	D2	D1	D0

D_3 1: Sets F14 (in/rev.) for the feed per revolution in the inch system.

0: Sets F13 (in/rev.) for the feed per revolution in the inch system.

D_2 1: Sets F23 (mm/rev.) for the feed per revolution in the metric system.

0: Sets F22 (mm/rev.) for the feed per revolution in the metric system.

D_1 1: Sets F32 (mm/min.) for the feed per minute in the metric system.

0: Sets F31 (mm/min.) for the feed per minute in the inch system.

D_0 1: Sets F51 (mm/min.) for the feed per minute in the metric system.

0: Sets F50 (mm/min.) for the feed per minute in the metric system.

#6021						D1	D0

D_1 1: Employs the value following address O or N as the program number (speciable in one block).

0: Employs the value following address O as the program number.

D_0 1: Considers M02, M30 and M99 as the program end when machining data is stored into memory.

0: Does not consider M02, M30 and M99 as the program end when machining data is stored into memory.

#6023		D6		D4	D3	D2	D1	D0

D_6 1: Enables the additional axis to ignore signal 4NG.

0: Disables the additional axis to ignore signal 4NG.

D_4-D_0

Set the address for pan-out and CRT display on the additional axis.

Address	D4	D3	D2	D1	D0
A	0	0	0	0	1
B	0	0	0	1	0
C	0	0	0	1	1
U	1	0	1	0	1
V	1	0	1	1	0
W	1	0	1	1	1

#6026			D_5	D_4	D_3	D_2	D_1	D_0
#6027			D_5	D_4	D_3	D_2	D_1	D_0

#6026, #6027:

D_5 1: Does not allow the control code (DC1 - DC4) to be used on the I/O device.

0: Allows the control code (DC1 - DC4) to be used on the I/O device.

D_4 1: Employs 2 stop bits on the I/O device.

0: Employs 1 stop bit on the I/O device.

D_3 - D_0

Baud rate setting

Baud rate	D_3	D_2	D_1	D_0
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

NOTE: #6026 provides the setting on I/O device 1 (SIO-1) and #6027 on I/O device 2 (SIO-2). For #6028, #6029, see YASNAC MX1 OPERATOR'S MANUAL ADDENDUM section 6.3.

#6030	D_7	D_6		D_4			D_0

D_7 1: Provides an additional axis control module.

0: Does not provide an additional axis control module.

D_6 1: Causes the system to filter the spindle PG reference point signal before reading it.

0: Allows the system to read the spindle PG reference point signal as it is.

D_4 1: Enables the axis interlock function.

0: Disables the axis interlock function.

D_0 1: Enables data output with DIAGNOSE.

0: Disables data output with DIAGNOSE.

#6031						D_2	D_1	D_0

D_2, D_1, D_0

Specify whether or not the direct-in signals IN2, IN1 and IN0 are effective, respectively.

1: Effective with signal "0"

0: Effective with signal "1"

#6032						D_2	D_1	D_0

D_2, D_1, D_0

Specify whether or not the direct-in signals IN2, IN1 and IN0 are effective, respectively.

1: Ineffective

0: Effective

#6050	(X-axis)
#6051	(Y-axis)
#6052	(Z-axis)
#6053	(4th-axis)

#6050 to #6053:

Specify the backlash compensation, respectively, on the X-, Y-, Z- and 4th-axes (setting range: 0-255; "1" = least output increment).

#6056	(X-axis)
#6057	(Y-axis)
#6058	(Z-axis)
#6059	(4th-axis)

#6056 to #6059:

Specify the position error offset, respectively, on the X-, Y-, Z- and 4th-axes (setting range: 0 - 255; "1" = least output increment). The standard setting is 32.

#6068	(X-axis)
#6069	(Y-axis)
#6070	(Z-axis)
#6071	(4th-axis)

#6068 to 6071:

Output in pitch error offset pulses the pitch error compensation times each magnification specification, respectively, for the X-, Y-, Z- and 4th axes. The setting range is 1 to 3, and "1" represents a magnification of 1.

#6062	(X-axis)
#6063	(Y-axis)
#6064	(Z-axis)
#6065	(4th-axis)

#6062 to #6065:

Specify the overtravel, respectively, on the X-, Y-, Z- and 4th-axes in unidirectional approach (G60; setting range: 0 - 255; "1" = least input increment).

#6074	(X-axis)
#6075	(Y-axis)
#6076	(Z-axis)
#6077	(4th-axis)

#6074 to #6077:

Causes an alarm "34*" when a position deviation exceeding the critical servo error value is detected respectively, on the X-, Y-, Z- and 4th axes.

Setting formula: $n = 16 \times \frac{c}{b}$

Standard setting: 16 (b = c)

b; D/A saturation value (pulse count) set in #6080 - #6083

c; Critical servo error value (pulse count)

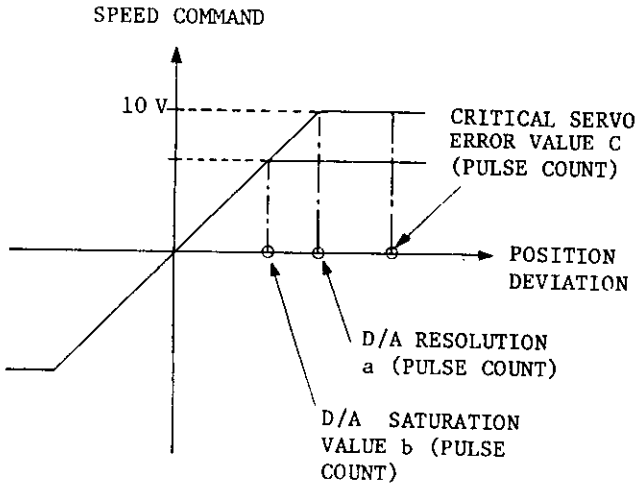


Fig. 3.3.6.1

#6080	(X-axis)
#6081	(Y-axis)
#6082	(Z-axis)
#6083	(4th-axis)

#6080 to #6083:

Specify the D/A saturation value, respectively, for the X-, Y-, Z- and 4th axes.

Setting formula: $2^{(7+n)} = b$

b: D/A saturation value (pulse count)

NOTE: "n" is set.

Standard setting: 6 (b = 8192 pulses)

NOTE: Make the setting so that b = a.

a: D/A resolution (pulse count)

#6086	(X-axis)
#6087	(Y-axis)
#6088	(Z-axis)
#6089	(4th-axis)

#6086 to #6089:

Specify the PG pulse magnification and D/A resolution, respectively, for the X-, Y-, Z- and 4th-axes.
Setting formula: n1 + n2

(1) PG pulse magnification value n1

PG pulse magnification	n1
x 1	64
x 2	80
x 4	112
x 8	48

(2) D/A resolution value n2

D/A resolution a (pulse count)	n2
32768 pulses	14
16384 pulses	13
8192 pulses	12
4096 pulses	11

Standard value: n2 = 12

#6092

Specifies the exponential function acceleration/deceleration time constant for cutting feed (common to all axes).

The setting formula is: "n" = $\frac{t}{4}$ - 1

where, t: time constant(ms), specifiable in units of 4 ms

#6093

Specifies the exponential function acceleration/deceleration speed bias for cutting feed (common to all axes).

Setting: "1" = 2 kpps.

#6094 (X-axis)

#6095 (Y-axis)

#6096 (Z-axis)

#6097 (4th-axis)

#6094 to #6097:

Specify the reference point return method, respectively, for the X-, Y-, Z- and 4th-axes.

"0" of NZ signal enabled		○	
NZ signal employed		○	○
Reference point pulse used	○		
Parameter setting	64	48	32

Standard setting: 64

#6106

Specifies the "rapid traverse section" for a "returning semicircle" by the proportionate semicircle radius in circle cutting (G12, G13).

Setting range: 0 - 10 (x 10%)

Examples:

1. A setting of 0 creates a rapid traverse section automatically computed by the program command values.
2. A setting of 10 (= 100%) makes the entire "returning semicircle" into a "rapid traverse section."

#6107

Specifies the number of manual pulse generators.

Setting range: 1 - 3

#6120 G-1

#6121 G-2

#6122 G-3

#6123 G-4

#6124 G-5

#6125 G-6

#6126 G-7

#6127 G-8

#6128 G-9

#6129 G-10

#6120 to #6129:

Specify up to 10 G codes for calling user macros.

#6130	M-1
#6131	M-2
#6132	M-3
#6133	M-4

#6130 to #6133:
Specify up to 4 M codes for calling user macros.

Setting range: 03 - 29, 31 - 89

NOTE: M00, M01, M02, M30 and M90 - M99 cannot be called by user macros.

#6134

1: Allows the T code to call a user macro.

0: Does not allow the T code to call a user macro.

#6141	
#6142	
#6143	
#6144	
#6145	
#6146	
#6147	
#6148	
#6149	

#6141 to #6149:
Specify the feedrate change for one increment on a manual pulse generator, for F1 to F9, respectively, of F1-digit designation.

Setting: "1" = 0.1 mm/min.

#6220

Specifies the interval from the time, M, S, T and B codes are transmitted until the time MF, SF, TF and BF are transmitted.

Setting: "1" = 1 ms

#6221

Specifies the interval from gear output (GRH, GRL) unit SF transmission when an S5-digit designation is added.

Setting: "1" = 1 ms

#6222

Specify the maximum handle feedrate, which is common to the linear axes (X, Y, Z, U, V, W).

Setting: "1" = 7.5 mm/min.

NOTE: The settings for the rotary axes (A, B, C) are made with #6348.

#6223

Specifies the tool shift speed for canned cycles of G76 and G77.

Setting: "1" = 1 mm/min.

NOTE: This specification is effective when $\#6019D_0 = 1$.
If $\#6019D_0 = 0$, rapid traverse is is effective regardless of this parameter specification.

#6224

Specifies the delay time for checking the spindle speed reaching signal (SAGR).

Setting: "1" = 1 ms

#6225

Specifies the feedrate for the rapid traverse section in circle cutting (G12, G13).

Setting: "1" = 1 mm/min.

#6226

#6227

#6226, #6227

Specify the maximum feedrate for F1-digit designation.

Setting: "1" = 1 mm/min.

NOTE: The maximum feedrate for F1-F4 commands is set in #6226 and that for F5-F9 commands in #6227. Any feedrates increased on manual pulse generators are bunched into these settings.

#6228

Specifies the maximum cutting feedrate for the linear axes (X, Y, Z, U, V, W).

Setting: "1" = 1 mm/min.

#6229

Specifies the maximum cutting feedrate for the rotary axes (A, B, C).

Setting: "1" = 1 mm/min.

NOTE: Any cutting feedrates greater than those set in #6228 and #6229 are bunched into those settings.

#6230

When a circular path is drawn in tool radius compensation outside a corner approaching 180°, the movement follows on a very small circular arc. In this, arc movement is considered to affect the workpiece surface machining, this parameter is used to set the critical arc value.

Setting: "1" = 0.001 mm (metric system)

"1" = 0.0001 in. (inch system)

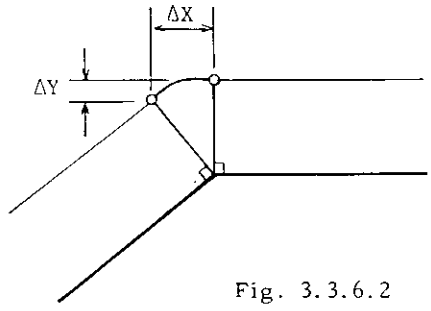


Fig. 3.3.6.2

The corner arc setting is ignored when:

$\Delta X \leqq$ #6230

$\Delta Y \leqq$ #6230

Standard setting = 5

#6231

Specifies the F_0 speed for rapid traverse override.

Setting: "1" = 7.5 mm/min.

#6232

Specifies the feedrate in the skip function (G31).

Setting: "1" = 1 mm/min.

NOTE: This setting is effective when parameter #6019D4 = 1.

#6233

⋮

#6264

#6233 to #6264

Specify the feedrate for the respective positions on the jog feedrate select switch.

Setting: "1" = 1 mm/min.

Typical settings

Table 3.3.6 Typical Settings mm/min

Switch position	Parameter		Continuous manual feedrate	
	Number	Setting	#6250 = 0	#6265 = 10
0	#6233	0	0	0
1	#6234	1	1	0.1
2	#6235	2	2	0.2
3	#6236	4	4	0.4
4	#6237	6	6	0.6
5	#6238	8	8	0.8
6	#6239	10	10	1.0
7	#6240	12	12	1.2
8	#6241	15	15	1.5
9	#6242	20	20	2.0
10	#6243	25	25	25
11	#6244	30	30	30
12	#6245	40	40	40
13	#6246	50	50	50
14	#6247	60	60	60
15	#6248	80	80	80
16	#6249	100	100	100
17	#6250	120	120	120
18	#6251	150	150	150
19	#6252	200	200	200
20	#6253	250	250	250
21	#6254	300	300	300
22	#6255	400	400	400
23	#6256	500	500	500
24	#6257	600	600	600
25	#6258	800	800	800
26	#6259	1000	1000	1000
27	#6260	1200	1200	1200
28	#6261	1500	1500	1500
29	#6262	2000	2000	2000
30	#6263	2500	2500	2500
31	#6264	3000	3000	3000

#6265

The manual feedrates set in parameters #6233 to #6264 can each be reduced to a tenth of the original setting. This applies to the settings on all switch positions lower than the value specified in this parameter #6265.

Setting: 0 - 31 (switch position)

#6266

#6267

#6268

#6269

#6266 to #6269:

Specify the spindle speed upper limit, respectively, for gears 1, 2, 3 and 4 (specifiable only in S5-digit designation).

Setting: 0 - 9999 (rpm)

#6270

Specifies the speed command output value to the spindle motor when a gear shift (GR0) input is entered (specifiable only in S5-digit designation). The setting formula is:

$$\frac{\text{Gear shift spindle motor speed}}{\text{Maximum speed of spindle motor (command = 10 V)}} \times 32767$$

Setting range: 0 - 32767

#6271	
#6272	
#6273	
#6274	

#6271 to #6274:

Specify the maximum speed of the spindle, respectively, for gears 1, 2, 3 and 4 each selected by an input signal (specifiable in S5-digit designation). Set the spindle speed applicable when the speed command voltage is 10 V.

Setting range: 0 - 99999 (rpm)

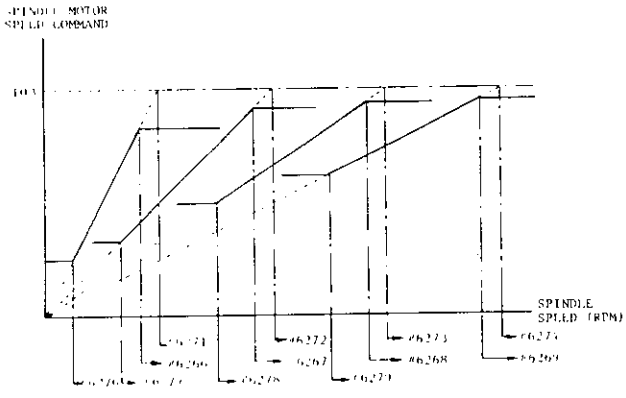


Fig. 3.3.6.3

#6275	

Specifies the spindle motor speed in effect when a spindle orientation (SOR) input is entered (specifiable in S5-digit designation).

Setting range: 0 - 99999 (rpm)

#6276	
#6277	
#6278	
#6279	

#6276 to #6279:

Specify the minimum speed of the spindle, respectively, for gears 1, 2, 3 and 4 each selected by an input signal (specifiable in S5-digit designation).

Setting range: 0 - 99999 (rpm)

#6280	(X-axis)
#6281	(Y-axis)
#6282	(Z-axis)
#6283	(4th-axis)

#6280 to #6283

Specify the rapid traverse rate, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 7.5 mm/min.

#6286	(X-axis)
#6287	(Y-axis)
#6288	(Z-axis)
#6289	(4th-axis)

#6286 to #6289:

Specify the first-stage time constant in linear acceleration/deceleration, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 125/8 (mm/sec^2)

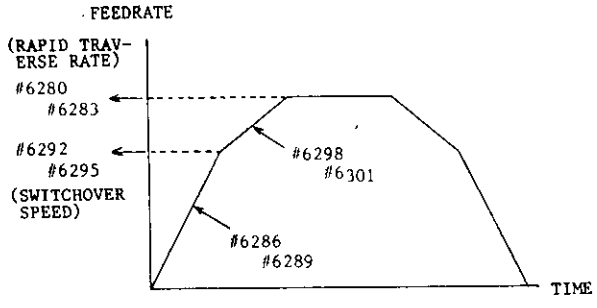


Fig. 3.3.6.4

#6292	(X-axis)
#6293	(Y-axis)
#6294	(Z-axis)
#6295	(4th-axis)

#6292 to #6295:

Specify the second-stage time constant switchover speed in linear acceleration/deceleration, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 7.5 mm/min.

#6298	(X-axis)
#6299	(Y-axis)
#6300	(Z-axis)
#6301	(4th-axis)

#6298 to #6301:

Specify the second-stage time constant in linear acceleration/deceleration, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 125/8 (mm/sec^2)

#6304	(X-axis)
#6305	(Y-axis)
#6306	(Z-axis)
#6307	(4th-axis)

#6304 to #6307:

Specify the traverse distance for reference point return, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 0.001 mm

#6310	(X-axis)
#6311	(Y-axis)
#6312	(Z-axis)
#6313	(4th-axis)

#6310 to #6313:

Specify the approach speed 1 for reference point return, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 7.5 mm/min.

#6316	(X-axis)
#6317	(Y-axis)
#6318	(Z-axis)
#6319	(4th-axis)

#6316 to #6319:

Specify the approach speed 2 for reference point return, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 7.5 mm/min.

NOTE: The parameters associated with reference point return operations are as follows:

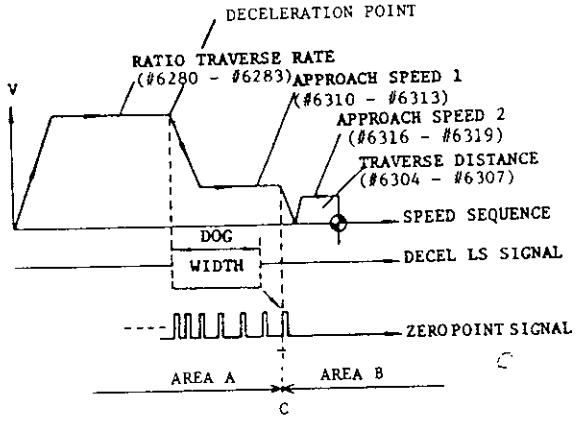


Fig. 3.3.6.5

- Reference point return direction: #6010 D_0 - D_3
- Reference point return enabled/disabled: #6016 D_6 - D_3

#6322	(X-axis)
#6323	(Y-axis)
#6324	(Z-axis)
#6325	(4th-axis)

#6322 to #6325:

Specify the number of the start point for pitch error compensation, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: 0 - 511

#6328	(X-axis)
#6329	(Y-axis)
#6330	(Z-axis)
#6331	(4th-axis)

#6328 to #6331:

Specify the number of the start point for pitch error compensation, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: 0 - 511

#6334	
#6335	
#6336	
#6337	

#6334 to #6337:

Specify the reference point for pitch error compensation, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: 0 - 511

#6340	

Specifies the external deceleration speed for rapid traverse.

Setting: "1" = 7.5 mm/min. (common to all axes)

#6341	

Specifies the external deceleration speed for cutting feed.

Setting: "1" = 1 mm/min. (common to all axes)

#6342	(X-axis)
#6343	(Y-axis)
#6344	(Z-axis)
#6345	(4th-axis)

#6342 to #6345:

Specify the offset in external workpiece coordinate system shift, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: 1 = 0.001 mm

NOTE: Usually, these parameters are automatically set from the machine tool side through the external data input function.

#6348	

Specifies the maximum speed for handle feed on the rotary axes (A, B, C).

Setting: "1" = 7.5 mm/min.

#6600	(X-axis)
#6601	(Y-axis)
#6602	(Z-axis)
#6603	(4th-axis)

#6600 to #6603:

Specify the plus direction boundary value for stored stroke limit 1, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 0.001 mm

#6606	(X-axis)
#6607	(Y-axis)
#6608	(Z-axis)
#6609	(4th-axis)

#6606 to #6609:

Specify the minus direction boundary value for stored stroke limit 1, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 0.001 mm

#6612	(X-axis)
#6613	(Y-axis)
#6614	(Z-axis)
#6615	(4th-axis)

#6612 to #6615:

Specify the distance between the first and the second reference point, respectively, on the X-, Y-, Z- and 4th-axes.

#6618	(X-axis)
#6619	(Y-axis)
#6620	(Z-axis)
#6621	(4th-axis)

#6618 to #6621:

Specify the distance between the first and the third reference point, respectively, on the X-, Y-, Z- and 4th-axes.

#6624	
#6625	
#6626	
#6627	

#6624 to #6627

Specify the distance between the first and the fourth reference point, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 0.001 mm

#6630	(X-axis)
#6631	(Y-axis)
#6632	(Z-axis)
#6633	(4th-axis)

#6630 to #6633

Specify the value for automatic coordinate system setting at the time of inch input, respectively, on the X-, Y-, Z-, and 4th-axes. A desired value should be set in inches for the distance between the first reference point and the reference point of the coordinate system to be established.

Setting: "1" = 0.0001 in.

#6636	(X-axis)
#6637	(Y-axis)
#6638	(Z-axis)
#6639	(4th-axis)

#6636 to #6639

Specify the value for automatic coordinate system setting at the time of metric input, respectively, on the X-, Y-, Z- and 4th-axes. A desired value should be set in millimeters for the distance between the first reference point and the reference point of the coordinate system to be established.

Setting: "1" = 0.001 mm

NOTE: Each setting is effective only for an axis with parameter #6015 at "1."

#6642	(X-axis)
#6643	(Y-axis)
#6644	(Z-axis)
#6645	(4th-axis)

#6642 to #6645

Specify the compensation interval in pitch error compensation, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 0.001 mm (metric output)
"1" = 0.0001 in. (inch output)

#8000	(number 0)
⎫	
#8511	(number 511)

#8000 to #8511

Specify the respective values of pitch error compensation.

Setting: 0 - ±15 (output increment)

NOTE: Parameters #6322 to #6339 determine specific combinations of settings and axes.

APPENDIX - 3 STORED LEADSCREW ERROR COMPENSATION

This function automatically compensates for leadscrew error on each axis according to the compensation data set by parameter and is effective after completion of reference point return. The compensation data are made on the distances between the reference point on each axis and specified points.

Compensation axes: X, Y, Z and 4th axes (including rotary axis)

No. of correction points: 512 Max.

Compensation base point: Reference point

Compensation interval: 6000 Pulses or more

Data setting system: Absolute/incremental (Set by Parameter #6039$_{D2}$)

Compensation value:

Minimum compensation unit: 1 pulse (least output increment)
Compensation multiplication: X13 max.
One-time-compensation value: 15 pulses max.
(Compensation multiplication)

Note 1:

Regardless of absolute/incremental setting, the difference between neighboring compensation values should be (15 pulses x compensation multiplication) and below.

Note 2:

Maximum set value in case of absolute setting is ±127 pulses. Compensation multiplication is taken on this value.

Note 3:

No. of correction points on each axis can be arbitrary as far as the total compensation points are within 512.

Note 4:

Where the 4th axis is a rotary axis, operation is possible within ±200 revolutions maximum.

Table 3.1

	Axis	Parameter #	Functions
Compensation interval	X to α	#6642 to #6645	6000 OR MORE "1" = 1 pulse
Absolute/incremental setting switchable		#6039$_{D2}$	"0" = Incremental setting "1" = Absolute setting
Compensation reference no.	X to α	#6334 to #6337	Value of parameter # of compensation on each point minus 8000 will be written
Compensation max point	X to α	#6322 to #6325	
Compensation min point	X to α	#6328 to #6331	
Compensation value on each point	X to α	#8000 to #8511	-15 to +15 (Incremental setting) "1" = 1 pulse
Compensation multiplication	X to α	#6068 to #6071	0 to 3 "1" = 1X

The figure below shows the example of writing the data for X axis.

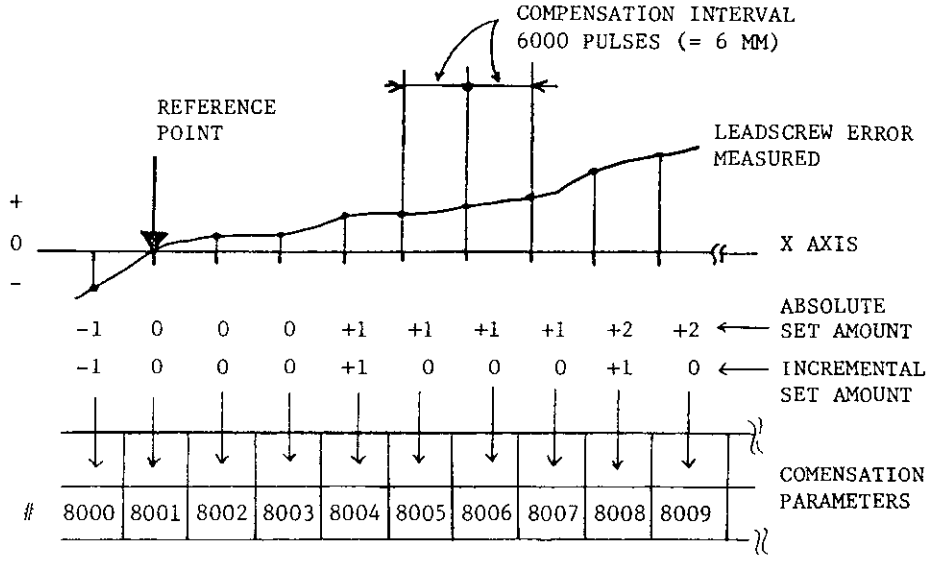


Fig. 3.1

In the above figure,

Compensation interval: 6000 pulses

Absolute/incremental: Incremental

Compensation multiplication: x 1

Compensation point on X axis: 100 points

Each parameter is set as follows:

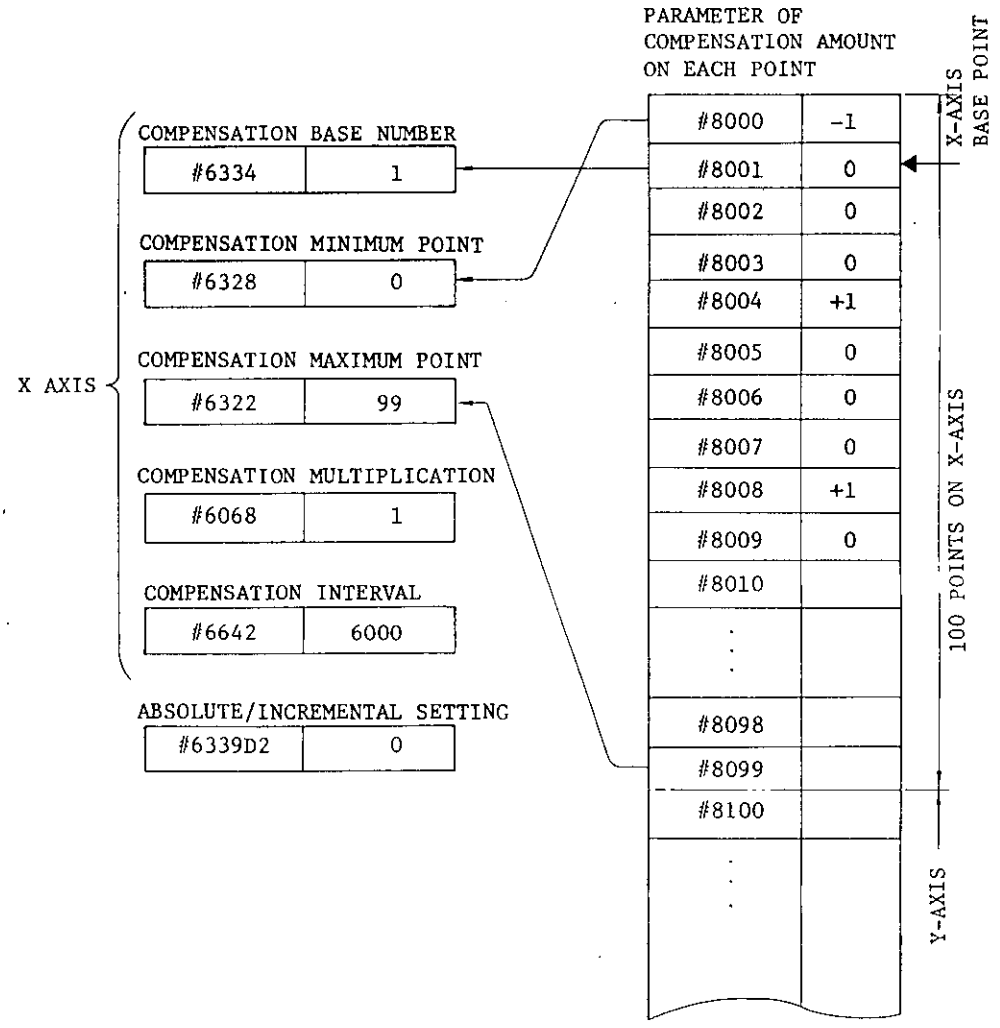


Fig. 3.2

To use the 4th axis as the rotary axis, follow the rules shown below in addition to the rules for setting X-, Y-, and Z-axes.

(1) Compensation Interval

The compensation interval should be more than 6000 pulses and the quotient obtained by dividing 360000 by the compensation interval become a positive integer.

(2) Compensation Amount at Reference Point

The compensation amount to be set to the reference point should be as follows:

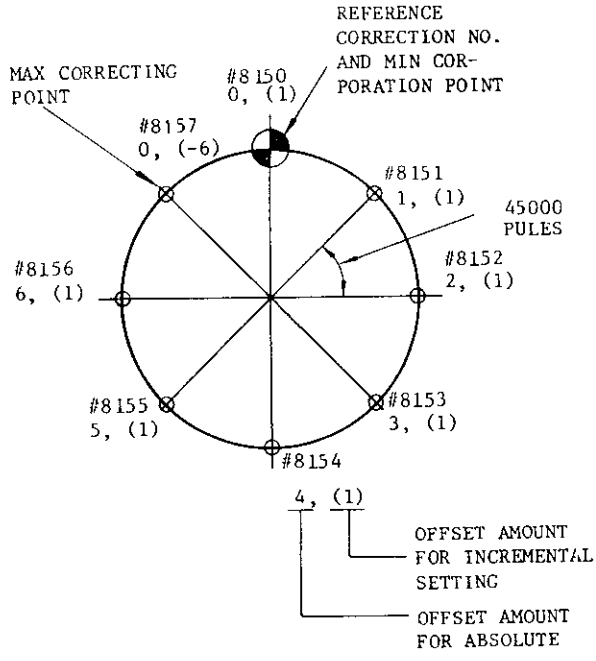
a. Absolute setting ··· "0"

b. Incremental setting ··· "0"

In the case of incremental setting, set such a value for the compensation amount at compensation maximum point that the sum of the compensation amount of each point becomes "0."

(3) Sample Writing

If the compensation interval is 45000 pulses (one rotation divided by 8) as shown below, set the parameters as follows.



4th axis of rotary axis

Offset reference No.

#6337	150

Offset min point

#6331	150

Offset max point

#6325	157

Offset multiplication factor

#6071	1

Offset point

#6645	45000

Offset parameter at each point

Parameter	Absolute setting	Incremental setting
#8150	0	0
#8151	1	1
#8152	2	1
#8153	3	1
#8154	4	1
#8155	5	1
#8156	6	1
#8157	0	-6

APPENDIX – 4 LIST OF STANDARD INPUT/OUTPUT SIGNALS

Table 4.1 shows standard input/output signals. For custom-built signals depending on the system, refer to the list of I/O signals provided on the system.

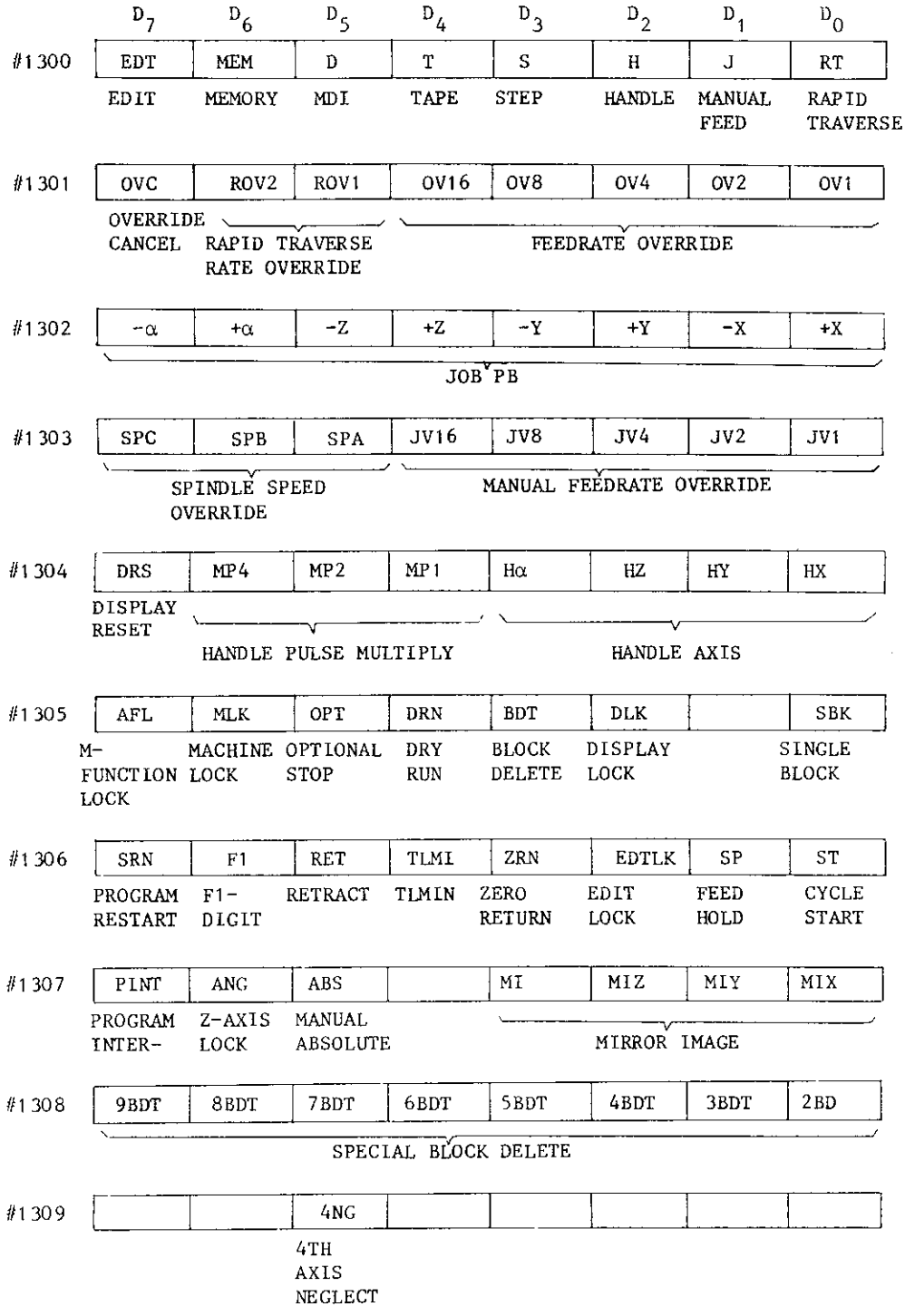
DISPLAY

D7	D6	D5	D4	D3	D2	D1	D0
0	0	0	0	1	1	1	1

Contact open

Contact closed

Table 4-1 List of Standard Input Signals



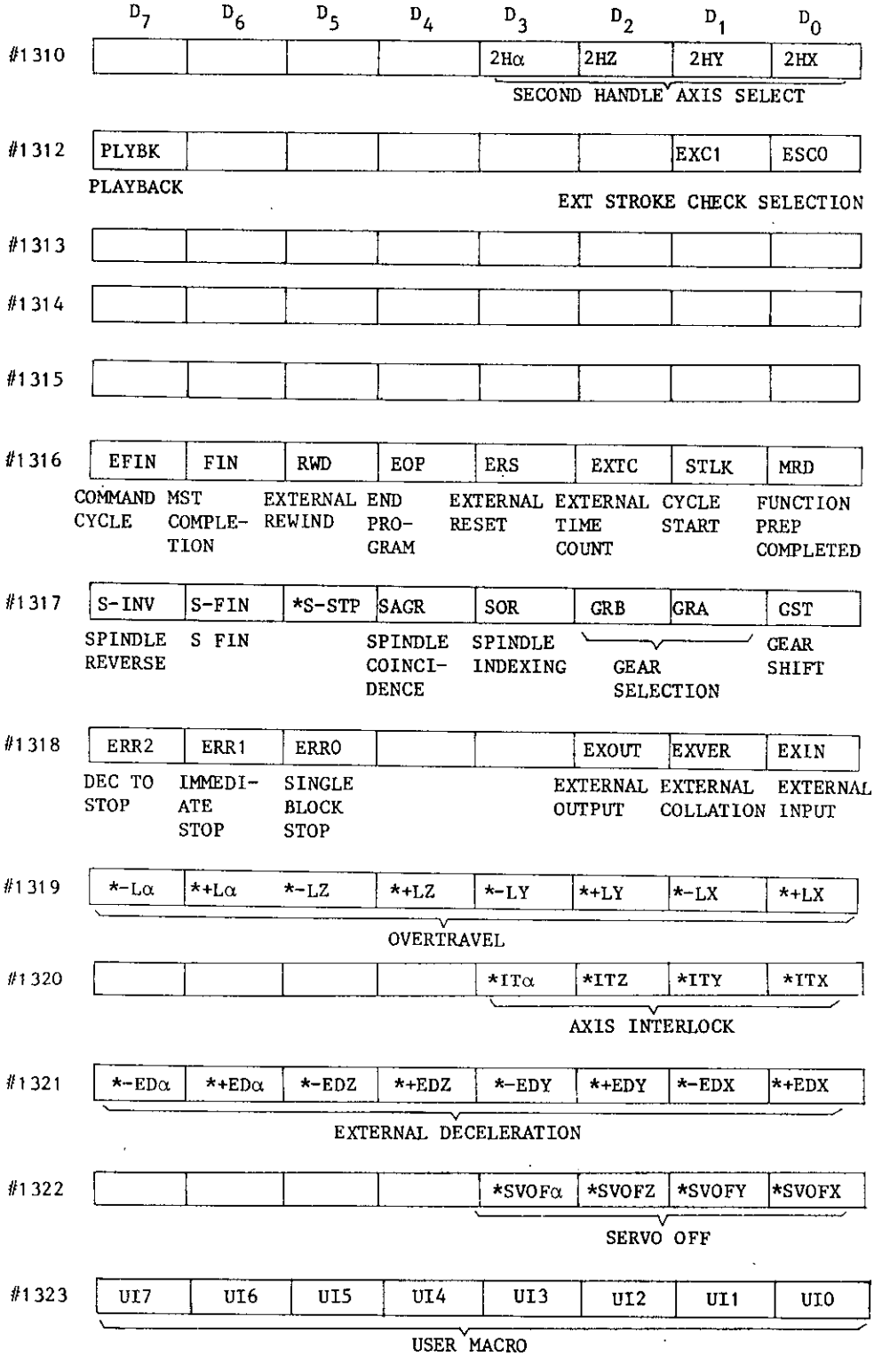
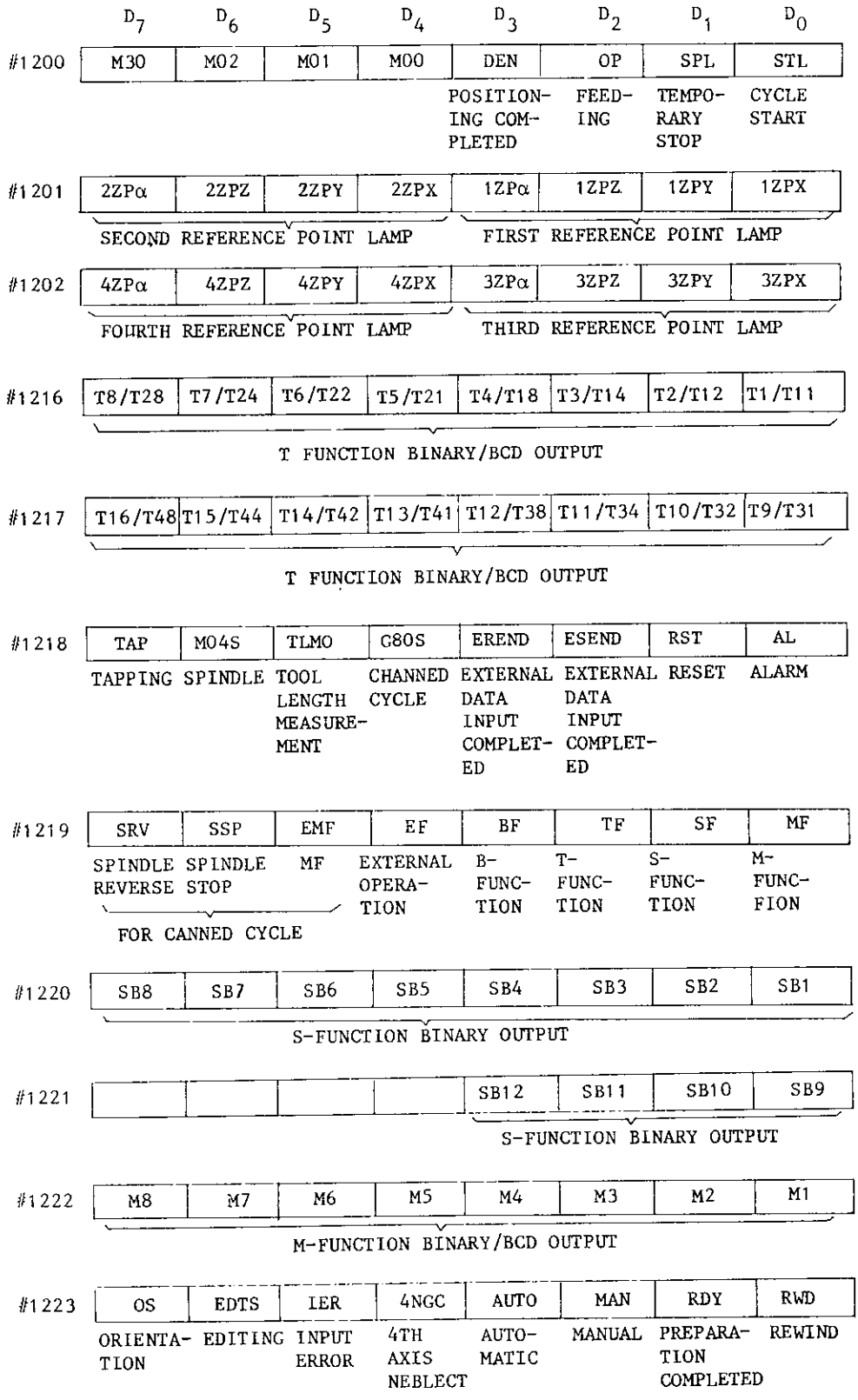

D7 D6 D5 D4 D3 D2 D1 D0
#1310
2Hα 2HZ 2HY 2HX
SECOND HANDLE AXIS SELECT
#1312
PLYBK EXC1 ESC0
PLAYBACK
EXT STROKE CHECK SELECTION
#1313
#1314
#1315
#1316
EFIN FIN RWD EOP ERS EXTC STLK MRD
COMMAND CYCLE
MST COMPLE-TION
EXTERNAL REWIND
END PRO-GRAM
EXTERNAL RESET
EXTERNAL TIME COUNT
CYCLE START
FUNCTION PREP COMPLETED
#1317
S-INV S-FIN *S-STP SAGR SOR GRB GRA GST
SPINDLE REVERSE
S FIN
SPINDLE COINCI-DENCE
SPINDLE INDEXING
GEAR SELECTION
GEAR SHIFT
#1318
ERR2 ERR1 ERR0 EXOUT EXVER EXIN
DEC TO STOP
IMMEDI-ATE STOP
SINGLE BLOCK STOP
EXTERNAL OUTPUT
EXTERNAL COLLATION
EXTERNAL INPUT
#1319
*-Lα *+Lα *-LZ *+LZ *-LY *+LY *-LX *+LX
OVERTRAVEL
#1320
*ITα *ITZ *ITY *ITX
AXIS INTERLOCK
#1321
*-EDα *+EDα *-EDZ *+EDZ *-EDY *+EDY *-EDX *+EDX
EXTERNAL DECELERATION
#1322
*SVOFα *SVOFZ *SVOFY *SVOFX
SERVO OFF
#1323
UI7 UI6 UI5 UI4 UI3 UI2 UI1 UI0
USER MACRO

Table 4.2 List of Standard Output Signals



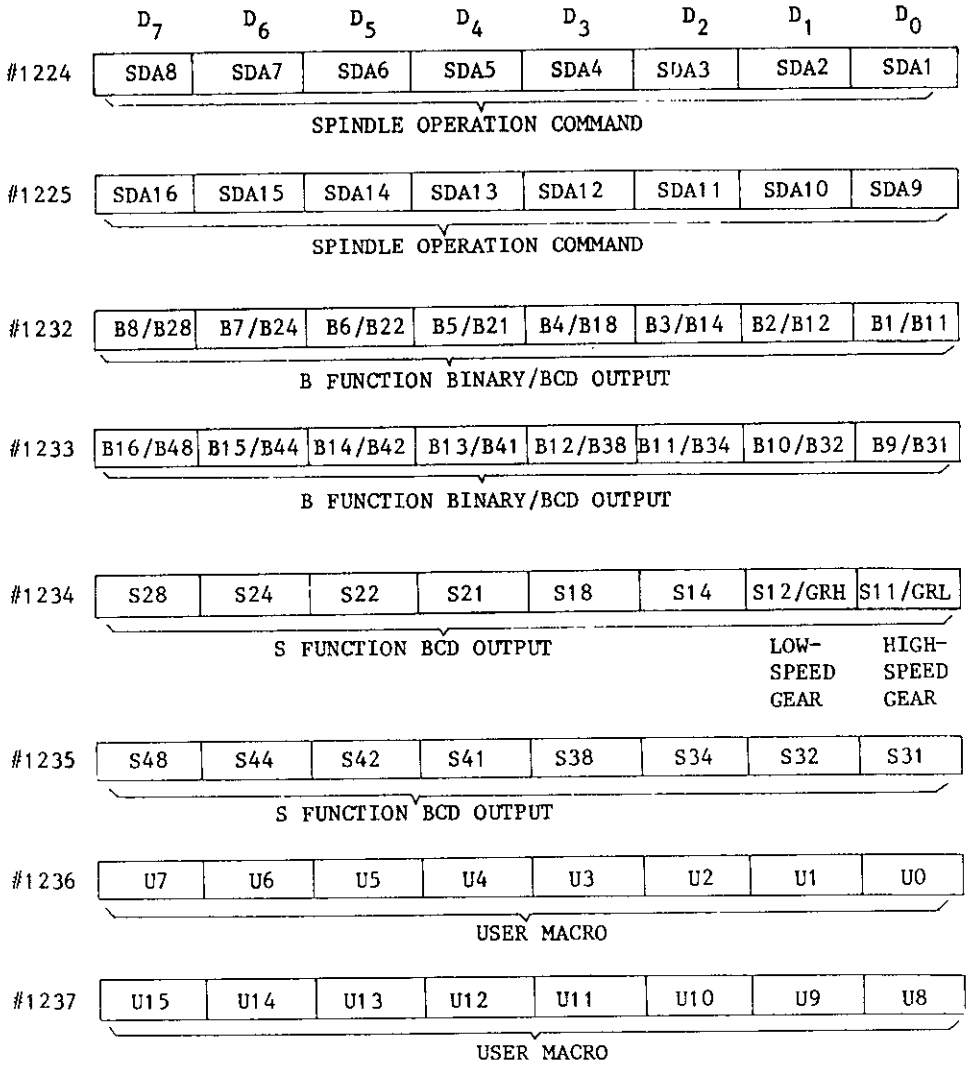
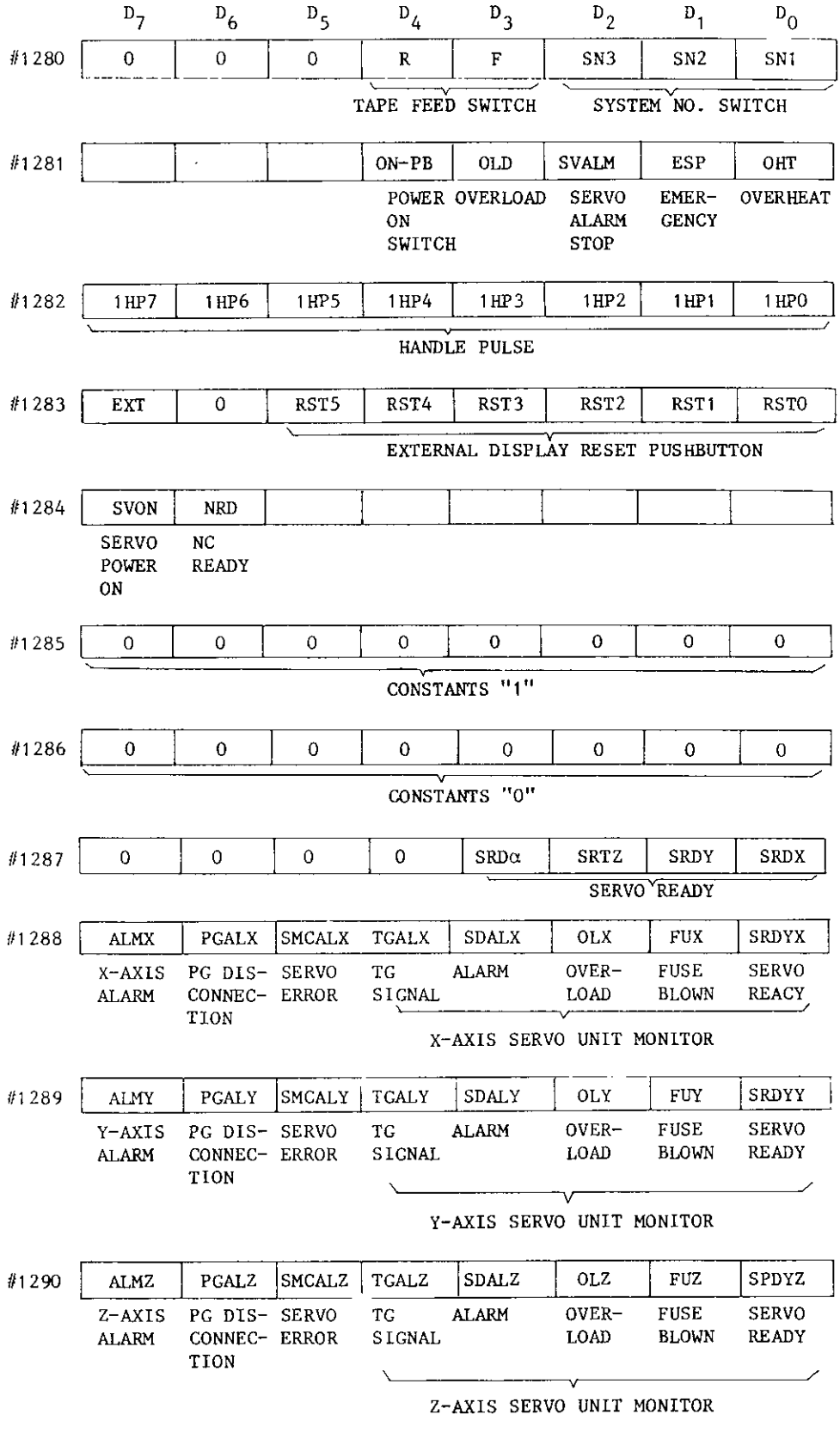
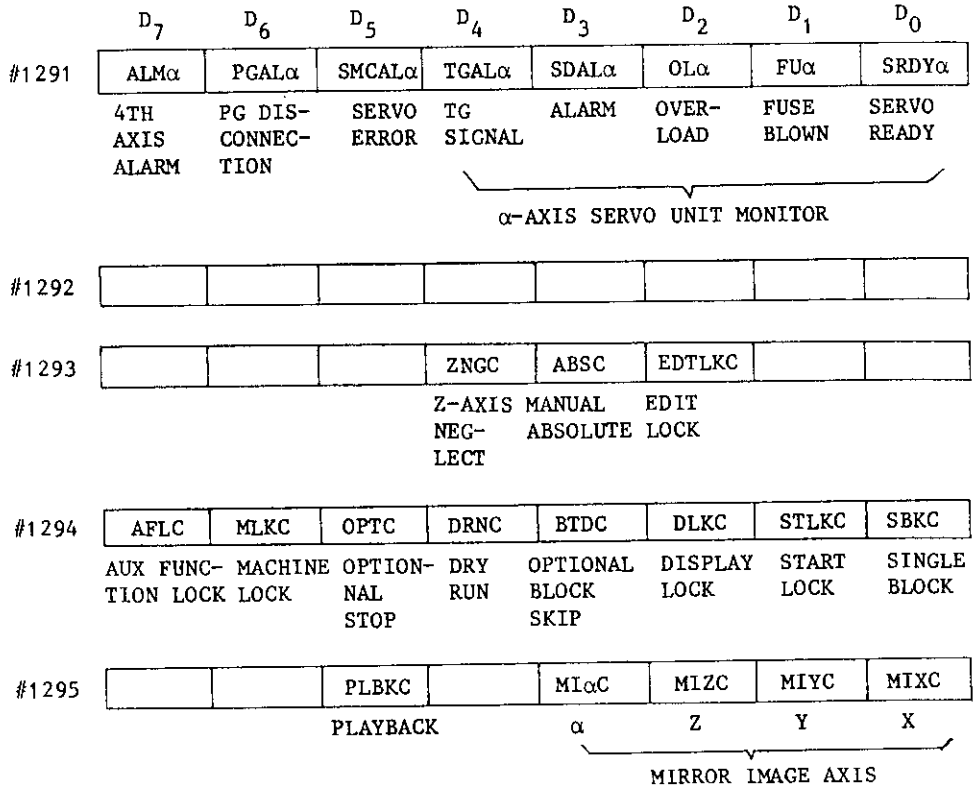

D7 D6 D5 D4 D3 D2 D1 D0
#1224
SDA8 SDA7 SDA6 SDA5 SDA4 SDA3 SDA2 SDA1
SPINDLE OPERATION COMMAND
#1225
SDA16 SDA15 SDA14 SDA13 SDA12 SDA11 SDA10 SDA9
SPINDLE OPERATION COMMAND
#1232
B8/B28 B7/B24 B6/B22 B5/B21 B4/B18 B3/B14 B2/B12 B1/B11
B FUNCTION BINARY/BCD OUTPUT
#1233
B16/B48 B15/B44 B14/B42 B13/B41 B12/B38 B11/B34 B10/B32 B9/B31
B FUNCTION BINARY/BCD OUTPUT
#1234
S28 S24 S22 S21 S18 S14 S12/GRH S11/GRL
S FUNCTION BCD OUTPUT
LOW-SPEED GEAR
HIGH-SPEED GEAR
#1235
S48 S44 S42 S41 S38 S34 S32 S31
S FUNCTION BCD OUTPUT
#1236
U7 U6 U5 U4 U3 U2 U1 U0
USER MACRO
#1237
U15 U14 U13 U12 U11 U10 U9 U8
USER MACRO

Table 4.2 List of Standard Output Signals (Cont'd)




D7 D6 D5 D4 D3 D2 D1 D0
#1291
ALMα PGALα SMCALα TGALα SDALα OLα FUα SRDYα
4TH AXIS ALARM
PG DIS-CONNEC-TION
SERVO ERROR
TG SIGNAL
ALARM
OVER-LOAD
FUSE BLOWN
SERVO READY
α-AXIS SERVO UNIT MONITOR
#1292
#1293
ZNGC ABSC EDTLKC
Z-AXIS NEG-LECT
MANUAL ABSOLUTE
EDIT LOCK
#1294
AFLC MLKC OPTC DRNC BTDC DLKC STLKC SBKC
AUX FUNC-TION LOCK
MACHINE LOCK
OPTION-NAL STOP
DRY RUN
OPTIONAL BLOCK SKIP
DISPLAY LOCK
START LOCK
SINGLE BLOCK
#1295
PLBKC MIαC MIZC MIYC MIXC
PLAYBACK
α Z Y X
MIRROR IMAGE AXIS

APPENDIX – 5 LIST OF ALARM CODES

Table 5.1

Alarm No.	Spindle Operation	Type of Alarm
000 to 099	Stop at block end	Tape format error alarm
100 to 199	Stop at block end	Macro, operation, external input/ output error, sequence error (1)
200 to 299	Decelerate to stop	Overtravel, reference point return, positioning, machine ready
300 to 399	Decelerate to stop	Servo, emergency stop, overload FG, RPG
400 to 499	Decelerate to stop	Sequence error (2)
500 to 599	Immediate stop	Sequence error (3)
600 to 699		Sequencer message
700 to 799		
800 to 899	NC system stop	CPU error, RAM error, ROM error
900 to 999	———	Off-line error

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
000	POWER OFF	014	PROG ERROR ("-," "0")
	SETTING THE PARAMETER REQUIRING TURNING OFF POWER		SIGN "-," "0" NOT CORRECTLY USED
001	ZR UNREADY (X)	015	PROG ERROR (UNUSABLE CH)
	REFERENCE POINT RETURN NOT COMPLETED X		UNUSABLE CHARACTER PROGRAMMED IN INSIGNIFICANT DATA AREA
002	ZR UNREADY (Y)	016	PROG ERROR (UNUSABLE AXIS)
	REFERENCE POINT RETURN NOT COMPLETED Y		DEFINED AS ADDITIONAL AXIS OR B-FUNCTION
003	ZR UNREADY (Z)	017	PROG ERROR (8 DIGITS)
	REFERENCE POINT RETURN NOT COMPLETED Z		INPUT DATA OVERFLOW (MODE THAN 8 CHARACTERS)
004	ZR UNREADY (4)	020	PROG ERROR (G)
	REFERENCE POINT RETURN NOT COMPLETED.		UNUSABLE G CODE OR G CODE NOT INCLUDED IN OPTIONS PROGRAMMED
005	ZR UNREADY (5)	021	PROG ERROR (G)
	REFERENCE POINT RETURN NOT COMPLETED.		G CODE IN 1, 4, * GROUP PROGRAMMED SIMULTANEOUSLY IN A BLOCK
010	TH ERROR	022	PROG ERROR (G02/03, G43/44)
	TAPE HORIZONTAL PARTY ERROR		G43, G44 COMMANDED IN CIRCULAR INTERPOLATION MODE (G02, G030
011	TV ERROR	023	PROG ERROR (G)
	TAPE VERTICAL PARITY ERROR		UNUSABLE G CODE COMMANDED IN CANNED CYCLE
012	OVERFLOW (128CH)	024	PROG ERROR (G, G41/42)
	BUFFER CAPACITY OVERFLOW IN A BLOCK (128 CHARACTERS)		UNUSABLE CODE COMMANDED DURING COMPENSATION MODE
013	PROG ERROR (NO ADDRESS)	025	PROG ERROR (G70/71/72)
	ADDRESS PLUS NO DATA AND NEXT ADDRESS COMMANDED. OR NO ADDRESS PLUS DATA		G70 TO G72 COMMANDED EXCEPT IN CANNED CYCLES

NOTES:
1. No move command in three blocks in series at G41 (G42) command.
2. M00 commanded when rise.
3. Rise at circular interpolation block.

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
026	PROG ERROR (G41/42)	038	PROG ERROR (P, G10)
	RISE ERROR AT COMPENSATION C (COMMAND WHICH CANNOT BE ACCOMODATED CORRECTLY IN COMPENSATION C MODE) SEE NOTES		TOOL LARGE P WHEN WORK COORDINATE SYSTEM IS PROGRAM-INPUT
027	PROG ERROR (G41/42)	040	PROG ERROR (M98, G65/66)
	ERROR AT COMPENSATION C (ERROR IN CIRCULAR INTERPOLATION MODE)		• P NOT PROGRAMMED IN M98, G65, G66, G25. • P DIFFERENT FROM Q IN NO. IN G25MODE. • G25 AND M98/M99 PROGRAMMED SIMULTANEOUSLY.
030	PROG ERROR (F)	041	PROG ERROR (M08/09, G65/66)
	NO F-COMMAND IN FEED COMMAND		PROGRAM NO. (SEQUENCE NO.) NOT FOUND WHEN PROGRAM IS CALLED BY M98, G65, G66, G25, G, M, T
031	PROG ERROR (R = 0)	042	PROG ERROR (M98, NEST)
	CIRCLE WITH RADIUS 0 COMMANDED IN CIRCULAR ARC COMMAND		SUBPROGRAM (G25) OR MACRO CALL FIVE-NESTED.
032	PROG ERROR (G02/03)	043	PROG ERROR (G75)
	COMMANDS ON THREE AXES IN CIRULAR ARC COMMAND WITHOUT HELICAL OPTION		UNUSABLE ADDRESS COMMANDED IN G75.
033	PROG ERROR (G02/03)	044	PROG ERROR (G12/13)
	COMMANDS ON MORE THAN FOUR AXES IN CIRCULAR ARC PLANE WHOSE ARC CANNOT BE SELECTED FROM THE COMMAND		IN CIRCLE CUTTING, PROGRAMMED RADIUS R IS SMALLER THAN COMPENSATION D.
034	PROG ERROR (G02/03)	045	CAL ERROR (G41/42)
	CIRCULAR ARC R DESIGNATION ERROR		———
035	PROG ERROR (D, H)	046	PROG ERROR (G41/42)
	TOO LARGE NO. OF H OR D CODE FOR TOOL RADIUS COMPENSATION AND TOOL LENGTH COMPENSATION		IN COMPENSATION C MODE, CIRCULAR ARC OUTSIDE OF COMPENSATION PLANE PROGRAMMED.
036	PROG ERROR (P, G10)	047	PROG ERROR (G41/42)
	TOO LARGE P (NUMBER DESIGNATION) WHEN OFFSET IS PROGRAM -INPUT		COMPENSATION PLANE CHANGED DURING COMPENSATION C MODE
037	PROG ERROR (P, G10)	048	PROG ERROR (G41/42)
	TOOL LARGE R WHEN WORK COORDINATE SYSTEM IS PROGRAM-INPUT		INTERSECTION POINT NOT OBTAINED BY INTERSECTION COMPUTATION

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
049	PROG ERROR (G41/42)	070	PROG ERROR (M02/30/99)
	REVERSE OR ALMOST REVERSE COMMANDED IN M97 MODE		MEMORY OPERATION FINISH COMMAND NOT GIVEN
050	SCALING ERROR	075	RS232C
	UNUSABLE G CODE (G92, G28 TO G30, G36 TO G38, G70 TO G72) IN SCALING MODE.		RS232C INTERFACE DISAGREEMENT OF NO. OF BITS AND NO. OF BAUD RATES
051	SCALING ERROR	076	RS232C
	ERROR IN G51 AND G50 BLOCK FORMAT. SCALING FACTOR ZERO.		DATA TRANSMISSION FAILURE THROUGH RS232C INTERFACE
055	PROG ERROR (M, S, T, B)	077	RS232C
	M, S, T, B COMMANDS IN THE BLOCK IN WHICH M, S, T, B CODE CANNOT BE COMMANDED		10 CHARACTERA MORE HAVE BEEN READ IN AFTER STOP CODE HAS BEEN TRANSMITTED THROUGH RS232C INTERFACE
056.	PROG ERROR (AXIS)	080	PROG ERROR (G10, G22/23)
	AXIS COMMAND IN G04, G20, AND G21 BLOCKS		G10, G22, AND G23 COMMANDED WITH AXIS DATA
058	MIRROR IMAGE (G28)	084	MIRROR IMAGE (G36/37/38)
	G28 COMMANDED DURING MIRROR IMAGE		MIRROR IMAGE IS ON WITH G36 TO G38
059	ZR UNREADY	085	PROG ERROR (G36/37)
	G28 NOT COMPLETED ON THE AXIS WHICH HAS G29 COMMAND OR REFERENCE POINT RETURN NOT COMPLETED ON THE AXIS		COMMAND OF I (J) ON MORE THAN ONE AXIS AT G36 (G37)
066	RESET UN READY (AFTER EDITING)	086	PROG ERROR (G38)
			COMMAND OTHER THAN K AT G38
067		087	PROG ERROR (G31/36/37/38)
	POWER TURNED OFF DURING WRITING MEMORY		TOUCH SWITCH NOT ON WHEN MOTION REACHES AT END POINT BY G31, G36 TO G38 COMMANDS
068		088	PROG ERROR (G36/37/38)
	EDITING BEING EXECUTED IN THE EDIT INHIBIT AREA		TOUCH SWITCH CALCULATION ERROR AT G36 TO G38 COMMANDS

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
100	CAL ERROR (FIXED POINT)	111	MACRO ERROR (MOVE G66-M99)
	MAGNITUDE OF FIXED POINT DATA EXCEEDING UPPER LIMIT		MOVE COMMAND IN M99 FINISHING COMMAND OF MACRO CALLED BY G66.
101	CAL ERROR (FLOATING)	114	MACRO ERROR (DO-FORMAT)
	EXPONENT OF FLOATING POINT DATA EXCEEDING ALLOWABLE RANGE		NO. OF DOs AND ENDs NOT THE SAME.
102	CAL ERROR (DIVISION)	115	MACRO ERROR ([ ] UNMACH)
	CALCULATION DIVISOR ZERO OR OVERFLOW ERROR.		NO. OF LEFT BRACKETS AND RIGHT BRACKETS NOT THE SAME.
103	CAL ERROR (SQUARE ROOT)	116	MACRO ERROR (DO-END NO.)
	ROOT VALUE IS A NEGATIVE (−)		CONDITION $1 \leq n \leq 3$ NOT ESTABLISHED IN DOn.
104	PROG ERROR (DOUBLE ADD)	117	
	SAME ADDRESS REPEATED IN A BLOCK.		
105	MACRO ERROR (CONSTANT)	118	MACRO ERROR (GOTO N)
	CONSTANTS USABLE IN USER MACRO EXCEEDING THE LIMIT.		CONDITION $0 < n \leq 9999$ NOT ESTABLISHED OR NO SEQUENCE NO. IN GO TO n.
107	MACRO ERROR (FORMAT)	120	PRTN ERROR (NOT FOUND)
	ERROR IN THE FORMAT AND EQUATION.		SEQUENCE NO. NOT FOUND IN PART PROGRAM.
108	MACRO ERROR (UNDEFIN # NO)	121	PRTN ERROR (G92)
	UNDEFINED VARIABLE NO. DESIGNATED.		G92 COMMANDED THROUGH MDI OPERATION DURING PROGRAM RESTART.
109	MACRO ERROR (#NO NOT LEFT)	122	PRTN ERROR (G54-G59)
	COMMANDED PROHIBITED VARIABLE AS SUBSTITUTION IN LEFT-HAND SIDE OF THE EQUATION.		G54 TO G59 COMMANDED THROUGH MDI OPERATION DURING PROGRAM RESTART.
110	MACRO ERROR ([ ]5 LIMIT)	123	PRTN ERROR (ORG)
	MULTIPLE LAYERS OF PARENTHESES EXCEEDING THE UPPER LIMIT (5).		COORDINATE SYSTEM CHANGED BY DEPRESSING THE ORG BUTTON DURING PROGRAM RESTART.

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
124	PRTN ERROR (MDI MOVE)	145	
	AXIS OPERATED BY MDI AFTER PROGRAM RESTART		
130	EXT DATA	146	PROG ERROR (G100)
	DATA ERROR IN A GROUP DATA		HIGH-SPEED CUTTING COMMAND G101 OR G102 NOT CANCELLED BY G100.
131	EXT MESSAGE	147	PROG ERROR (PARAMETER ON)
	NO ALARM NUMBER CORRESPONDING TO EXTERNAL ALARM MESSAGE TO BE CLEARED.		PARAMETER #6008 (D7) SET TO 1 AT HIGH-SPEED CUTTING COMMAND.
132	EXT MESSAGE	170	MEM ERROR (OFS)
	NO CORRESPONDING ALARM NO. WHEN EXTERNAL ALARM MESSAGE IS CLEARED.		TOOL OFFSET TOATAL CHECK ERROR
133	EXT MESSAGE	172	MEM ERROR (SET)
	NO CORRESPONDING ALARM NO. WHEN EXTERNAL ALARM MESSAGE IS CLEARED.		SETTING AREA TOTAL CHECK ERROR
140	PROG ERRROR (P NG)	173	MEM ERROR (PRM)
	NO AXIS COMMAND IN HIGH-SPEED CUTTING PROGRAM AT HIGH-SPEED CUTTING COMMAND.		PARAMETER AREA TOTAL CHECK ERROR
141	PROG ERROR (FILE OVER)	179	OVER TEMP
	NO. OF BLOCKS IN REGISTERED PROGRAM EXCEEDING THE SPECIFIED VALUE IN HIGH-SPEED CUTTING.		PANEL INSIDE TEMPERATURE TOO HIGH
142	PROG ERROR (G00/G01/G02)	180	SEQ ERROR
	G CODE OTHER THAN G00, G01, G02 AND G03 OR M, S, T CODE COMMANDED IN REGISTERED PROGRAM AT HIGH-SPEED CUTTING COMMAND.		SEQUENCE ERROR (1)
143		181	SEQ ERROR
			SEQUENCE ERROR (1)
144	PROG ERROR (G101/G100)	182	SEQ ERROR
	ADDRESS OTHER THAN P COMMANDED IN G100, G101, G102 BLOCK.		SEQUENCE ERROR (1)

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
183	SEQ ERROR	193	SEQ ERROR
	SEQUENCE ERROR (1)		SEQUENCE ERROR (1)
184	SEQ ERROR	194	SEQ ERROR
	SEQUENCE ERROR (1)		SEQUENCE ERROR (1)
185	SEQ ERROR	195	SEQ ERROR
	SEQUENCE ERROR (1)		SEQUENCE ERROR (1)
186	SEQ ERROR	196	SEQ ERROR
	SEQUENCE ERROR (1)		SEQUENCE ERROR (1)
187	SEQ ERROR	197	SEQ ERROR
	SEQUENCE ERROR (1)		SEQUENCE ERROR (1)
188	SEQ ERROR	198	SEQ ERROR
	SEQUENCE ERROR (1)		SEQUENCE ERROR (1)
189	SEQ ERROR	199	SEQ ERROR
	SEQUENCE ERROR (1)		SEQUENCE ERROR (1)
190	SEQ ERROR	201	OT (X)
	SEQUENCE ERROR (1)		OVERTRAVEL X
191	SEQ ERROR	202	OT (Y)
	SEQUENCE ERROR (1)		OVERTRAVEL Y
192	SEQ ERROR	203	OT (Z)
	SEQUENCE ERROR (1)		OVERTRAVEL Z

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
204	OT (4)	223	S-OT2 (Z)
	OVERTRAVEL 4		STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) Z
205	OT (5)	224	S-OT2 (4)
	OVERTRAVEL 5		STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) 4
211	S-OT1 (X)	230	TOOL BROKEN
	STORED STROKE LIMIT FIRST AREA X		AT BROKEN TOOL DETECTION BY G32, G33, Z-AXIS MOVES TO THE SET POSITION BUT SENS (T) INPUT NOT TURNED ON.
212	S-OT1 (Y)	231	ZR ERROR-AREA (X)
	STORED STROKE LIMIT FIRST AREA Y		REFERENCE POINT RETURN AREA ERROR X
213	S-OT1 (Z)	232	ZR ERROR-AREA (Y)
	STORED STROKE LIMIT FIRST AREA Z		REFERENCE POINT RETURN AREA ERROR Y
214	S-OT1 (4)	233	ZR ERROR-AREA (Z)
	STORED STROKE LIMIT FIRST AREA 4		REFERENCE POINT RETURN AREA ERROR Z
215	S-OT1 (5)	234	ZR ERROR-AREA (4)
	STORED STROKE LIMIT 1ST AREA 5		REFERENCE POINT RETURN AREA ERROR 4
220	S-OT2 (INSIDE)	235	ZR ERROR-AREA (5)
	STORED STROKE LIMIT SECOND AREA (INSIDE INHIBIT)		REFERENCE POINT RETURN ERROR 5
221	S-OT2 (X)	241	ZR ERROR-POS (X)
	STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) X		REFERENCE POINT RETURN AREA ERROR X
222	S-OT2 (Y)	242	ZR ERROR-POS (Y)
	STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) Y		REFERENCE POINT RETURN AREA ERROR Y

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
243	ZR ERROR-POS (Z)	274	P-SET ERROR (4)
	REFERENCE POINT RETURN AREA ERROR Z		P SET ERROR 4
244	ZR ERROR-POS (4)	275	P-SET ERROR (5)
	REFERENCE POINT RETURN AREA ERROR 4		PROGRAM SET ERROR 5
245	ZR ERROR-POS (5)	280	MACH UNREADY
	REFERENCE POINT RETURN POSITION ERROR 5.		MACH RDY OFF
250	S-OT3-5 (INSIDE)	310	SERVO POWER NOT SUPPLIED
	STORED STROKE LIMIT 3RD, 4TH, 5TH AREA INSIDE PROHIBITED.		SERVO POWER NOT SUPPLIED
251	S-OT3-5 (X)	320	NC UNREADY
	STORED STROKE LIMIT 3RD, 4TH, 5TH AREA OUTSIDE PROHIBITED X.		NC UNREADY P SET UNREADY
252	S-OT3-5 (Y)	330	EMERGENCY STOP
	STORED STROKE LIMIT 3RD, 4TH, 5TH AREA OUTSIDE PROHIBITED Y.		EMERGENCY STOP
253	S-OT3-5 (Z)	331	FUSE (X)
	STORED STROKE LIMIT 3RD, 4TH, 5TH AREA OUTSIDE PROHIBITED Z.		FUSE BLOWN X
271	P-SET ERROR (X)	332	FUSE (Y)
	P SET ERROR X		FUSE BLOWN Y
272	P-SET ERROR (Y)	333	FUSE (Z)
	P SET ERROR Y		FUSE BLOWN Z
273	P-SET ERROR (Z)	334	FUSE (4)
	P SET ERROR Z		FUSE BLOWN 4

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
335	FUSE (5)	355	OL (5)
	FUSE BLOWN 5		OVERLOAD (1) 5
341	SERVO ERROR (X)	357	OL (OTHERS)
	SERVO ERROR X		OVERLOAD (2)
342	SERVO ERROR (Y)	361	PG ERROR (X)
	SERVO ERROR Y		PG ERROR X
343	SERVO ERROR (Z)	362	PG ERROR (Y)
	SERVO ERROR Z		PG ERROR Y
344	SERVO ERROR (4)	363	PG ERROR (Z)
	SERVO ERROR 4		PG ERROR Z
345	SERVO ERROR (5)	364	PG ERROR (4)
	SERVO ERROR 5		PG ERROR 4
351	OL (X)	365	PG ERROR (5)
	OVERLOAD (1) X		PG ERROR 5
352	OL (Y)	371	FG ERROR (X)
	OVERLOAD (1) Y		FG ERROR X
353	OL (Z)	372	FG ERROR (Y)
	OVERLOAD (1) Z		FG ERROR Y
354	OL (4)	373	FG ERROR (Z)
	OVERLOAD (1) 4		FG ERROR Z

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
374	FG ERROR (4)	394	TG ERROR (4)
	FG ERROR 4		TG LEAD DISCONNECTION
375	FG ERROR (5)	395	TG ERROR (5)
	FG ERROR 5		TG LEAD DISCONNECTION
381	RPG ERROR (X)	400	SEQ ERROR
	RPG ERROR X		SEQUENCE ERROR (2)
382	RPG ERROR (Y)	419	SEQ ERROR
	RPG ERROR Y		SEQUENCE ERROR (2)
383	RPG ERROR (Z)	300	SEQ ERROR
	RPG ERROR Z		SEQUENCE ERROR (3)
384	RPG ERROR (4)	519	SEQ ERROR
	RPG ERROR 4		SEQUENCE ERROR (3)
385	RPG ERROR 5	800	MEM ERROR
	PROGRAM ERROR 5		BUBBLE FAILURE INPUT/OUTPUT FAILURE
391	TG ERROR (X)	801	MEM ERROR
	TG LEAD DISCONNECTION		BUBBLE FAILURE INITIAL ERROR
392	TG ERROR (Y)	802	MEM ERROR
	TG LEAD DISCONNECTION		BUBBLE FAILURE UNDEFINED COMMAND
393	TG ERROR (Z)	803	MEM ERROR
	TG LEAD DISCONNECTION		BUBBLE FAILURE TRANSFER MISSING

Table 5-2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
804	MEM ERROR		
	BUBBLE FAILURE PARITY ERROR		
805	MEM ERROR		
	BUBBLE FAILURE NO MARKER		
806	MEM ERROR		
	BUBBLE FAILURE MANY DEFECT LOOPS		
810	RAM ERROR		
	RAM CHECK ERROR		
820	ROM ERROR		
	ROM CHECK ERROR		
830	CPU ERROR		
	CPU ERROR (1)		
840	CPU ERROR		
	CPU ERROR (2)		
910	TAPE MEM ERROR		
	MEMORY VERIFYING ERROR (OFF-LINE)		
920	TAPE ERROR		
	TAPE READING-IN ERROR (OFF-LINE)		

APPENDIX – 6 LIST OF ADDRESS CHARACTERS

Table 6.1 List of Address Characters

Address Characters	Meanings	B: Basic O: Optional
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	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	O, B
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	O
X	X-coordinate	B
Y	Y-coordinate	B
Z	Z-coordinate	B

Table 6.2 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, User macro operator	
/	/	Optional block skip	
Del	DEL	Disregarded (Including All Mark)	
.	.	Decimal point	
Parameter setting	#	Sharp (Variable)	EIA: Special code
*	*	Astrisk (Multiplication operator)	
=	=	Equal mark	
[	[	Left bracket	
]	]	Right bracket	
:	:	User macro comment	
$	$	User macro comment	
@	@	User macro comment	
?	?	User macro comment	

Notes:

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

Table 6.3 Tape Code

EIA CODE 8	7	6	5	4	3	2	1	CHARACTERS	ISO CODE 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 —								
								— (								
								—)								○
								+							○	○
								—								○
								o :							○	
							○	/								
	○	○	○	○	○	○	○	Del DEL								
○	○	○	○	○	○	○	○	All Mark								
See Note 2.								#								
								*								
								=								
								[								
								]								
								$								
								@								
								?								
	○	○		○			○	.								

Notes:

1. For characters from # to ?, EIA codes have not been agreed upon. In the present system, for the time being, the above provisional codes are used.

2. EIA code of character # can be designated by the parameter #6017.

Table 6.4 Tape Format

No.	Address		Metric output		Inch input		B: Basic O: Optional
			Metric input	Inch input	Metric input	Inch input	
1	Program No.		O4		O4		B
2	Sequence No.		N4		N4		B
3	G function		G3		G3		B
4	Cordinate Word	Linear axis	a + 43	a + 34	a + 53	a + 34	B
		Rotary axis	b + 43	b + 43	b + 43	b + 43	O
5	Feed/min		F40	F31	F50	F31	B
6	Feed/min 1/10		F41	F32	F51	F32	B
7	Feed/rev		F22	F13	F22	F13	O
8	Feed/rev 1/10		F23	F14	F23	F14	O
9	S-function		S2		S2		B
			S5		S5		O
10	T-function		T2		T2		B
			T4		T4		O
11	M-function		M3		M3		B
12	Tool Offset No.		H2 or D2		H2 or D2		B
13	B-function		B3		B3		O
14	Dwell		P53		P53		B
15	Program No. designation		P4		P4		B
16	Sequence No. designation		P4		P4		B
17	No. of repititions		L8		L8		B

Table 6.5 Range of Program Commands

No.	Address		Metric Output		Inch Output	
			Metric Input	Inch Input	Metric Input	Inch Input
1	Program Number	O	1 to 9999	1 to 9999	1 to 9999	1 to 9999
2	Sequence Number	N	1 to 9999	1 to 9999	1 to 9999	1 to 9999
3	G-function	G	0 to 199	0 to 199	0 to 199	0 to 199
4	Coordinate address Linear axis Rotary axis Max cumulative value		± 8388.607 mm ± 8388.607 deg ± 99999.999 mm	± 330.2601 in. ± 8388.607 deg ± 9999.99999 in.	± 21307.061 mm ± 8388.607 deg ± 99999.999 mm	± 838.8607 in. ± 8388.607 deg ± 9999.99999 in.
5	Feed per minute	F	1 to 8100 mm/min	0.1 to 313.0 in./min	1 to 20574.0 mm/min	0.1 to 810.0 in./min
6	Feed per minute 1/10		0.1 to 8100.0 mm/min	0.01 to 313.00 in./min	0.1 to 20574.0 mm/min	0.01 to 810.00 in./min
7	Feed per revolution		0.01 to 99.99 mm/rev	0.0001 to 3.936 in./rev	0.01 to 99.99 mm/rev	0.0001 to 3.936 in./rev
8	Feed per revolution 1/10		0.01 to 99.999 mm/rev	0.001 to 3.9366 in./rev	0.001 to 99.999 mm/rev	0.0001 to 3.9366 in./rev
9	S-function	S2	0 to 99	0 to 99	0 to 99	0 to 99
		S5	0 to 99999	0 to 99999	0 to 99999	0 to 99999
10	T-function	T2	0 to 99	0 to 99	0 to 99	0 to 99
		T4	0 to 9999	0 to 9999	0 to 9999	0 to 9999
11	M-function	M	0 to 199	0 to 199	0 to 199	0 to 199
12	Tool offset No.	H	0 to 99	0 to 99	0 to 99	0 to 99
		D	0 to 99	0 to 99	0 to 99	0 to 99
13	B-function	B	0 to 999	0 to 999	0 to 999	0 to 999
14	Dwell	P	0 to 99999.999 sec	0 to 99999.999 sec	0 to 99999.999 sec	0 to 99999.999 sec
15	Program No. designation	P	1 to 9999	1 to 9999	1 to 9999	1 to 9999
16	Sequence No. designation	P	1 to 9999	1 to 9999	1 to 9999	1 to 9999
17	No. of repetitions	L	99999999	99999999	99999999	99999999

Table 6.6 Range of Data Setting

Address		Metric Output		Inch Output	
		Metric Input	Inch Input	Metric Input	Inch Input
Least input increment		0.001 or 0.01 mm	0.0001 or 0.001 in.	0.001 or 0.01 mm	0.0001 or 0.001 in.
Max stroke (Distance from reference point)		± 99999.999 mm		± 3937.0078 in.	± 9999.9999 in.
Tool offset amount & Tool radius value		0 to ± 999.999 mm	0 to ± 99.9999 in.	0 to ± 999.999 mm	0 to ± 99.9999 in.
Min. feed amount at STEP/HANDLE		0.001 mm	0.0001 in.	0.001 mm	0.0001 in.
Unit of area setting for stored stroke limit	Program designation	0.001 mm	0.0001 in.	0.001 mm	0.0001 in.
	Parameter & setting	0.001 mm		0.0001 in.	
Rapid traverse rate		(7.5 to) 1 to 24,000 mm/min		(0.75 to) 0.1 to 2400.0 in./min	
Manual jog speed					
Seed at Fo					
Max. feedrate		1 to 8,100 mm/min		0.1 to 810.0 in./min	
Dry run speed					
Stored leadscrew compensation, stored stroke limit, and setting area for 2nd to 4th reference points		0 to ± 9999.999 mm		0 to ± 3937.0078 in.	0 to ± 9999.9999 in.
Backlash compensation		0 to 255 pulses		0 to 255 pulses	
Stored leadscrew Compensation	Incremental	0 to 15 pulses		0 to 15 pulses	
	Absolute	0 to 127 pulses		0 to 127 pulses	

Note: 1 pulse = least output increment

Table 6.7 List of G Codes

G code	Group	Function	B: Basic O: Optional
G00	01	Positioning	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	O
G21		Metric input designation	O
G22	04	Stored stroke limit ON	O
G23		Stored stroke limit OFF	O
G27	*	Reference point check	O
G28		Automatic return to reference point	O
G29		Return from reference point	O
G30		Return to 2nd, 3rd, 4th reference point	O
G31		Skip function	O
G33	01	Thread cutting	O
G36	*	Automatic centering	O
G37		Automatic centering	O
G38		Z-axis reference surface offset	O
G40	07	Tool radius compensation cancel	O
G41		Tool radius compensation, left	O
G42		Tool radius compensation, right	O
G43	08	Tool length compensation, plus direction	B
G44		Tool length compensation, minus direction	B
G49		Tool length compensation, 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	O
G53	*	Temporary shift to machine coordinate system	O
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
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	*	Programming of absolute zero point	B
G94	05	Feed per minute (mm/min) designation	O
G95		Feed per revolution (mm/rev.) designation	O
G98	10	Return to initial point for canned cycles	O
G99		Return to point R for canned cycles	O
G100	16	High-speed cutting cancel	O
G101		High-speed cutting in sequential processing mode ON	O
G102		High-speed cutting in processing mode ON	O

Notes:

1. The G codes in the * group are non-modal, and the effective only for the block in which they are commanded. They cannot be programmed twice or more in a block. They must be programmed only once in a block of its own.

2. The codes marked with ◥ is automatically selected at power on or reset.

YASNAC® MX1 OPERATOR'S MANUAL

A Better Tomorrow for Industry through Automation

YASKAWA Electric Mfg. Co., Ltd.

TOKYO OFFICE Ohtemachi Bldg., Chiyoda-ku, Tokyo 100 Japan
Phone (03) 284-9111 Telex YASKAWA J33530 Fax (03) 284 9034

TAIPEI OFFICE TECO ELC & MACH Bldg. 8F, 156-2 Sung Chiang Road, Taipei
Phone (2) 531-7732, 551-7065 Telex (785) 23591 TAIAN Fax (2) 396 2397 314 7818

YASKAWA ELECTRIC AMERICA, INC.: SUBSIDIARY
Chicago Office: *YASNAC America*
305 Era Drive, Northbrook, Illinois 60062, U.S.A.
Phone (312) 564-0770 Telex (230) 270197 YSKW YSNC NBRK Fax (312) 564 3276

Los Angeles Office
14811 Myford Road, Tustin, California 92680, U.S.A.
Phone (714) 731-6841 Telex (230) 678396 YASKAWAUS TSTN Fax (714) 730 8294

YASKAWA ELECTRIC EUROPE GmbH: SUBSIDIARY
Monschauerstrasse 1, 4000 Dusseldorf 11, West Germany
Phone (0211) 501127 Telex (41) 8588673 YASD D Fax (0211) 507737

Printed in Japan September 1984 83-6 1.7 TA ⑤