INSTRUCTION MANUAL

SINKER EDM

PROGRAMMING

1807-202305V1

PRECAUTIONS

1. Functions not covered by this manual, in general, can not be executed.
2. The notes provided in this manual must be observed.
   (Otherwise, the function may not be executed correctly or the system could be adversely affected.)
3. Specifications are subject to change for improvement without notice.
4. Please contact us if you have any uncertainly about the manual.

CONTENTS

1. PREFACE

This “Programming Manual” describes how to make NC programs to be used for the sinker electrical discharge machine.
The manual is intended for not only beginners who make an NC program for the first time but also those who want to utilize the most of the NC functions.

2. PROGRAM STRUCTURE

An aggregate of a series of operation instructions required for the work by sinker electric discharge machine is called “program”. This program consists of blocks that are instruction units for the sinker electric discharge machine.

One block consists of some “words” and each work consists of an address of alphabetic characters and a numeric value following the address. Each numeric character, alphabetic or code is called “character”.

Fig. 2.1 shows the program structure.

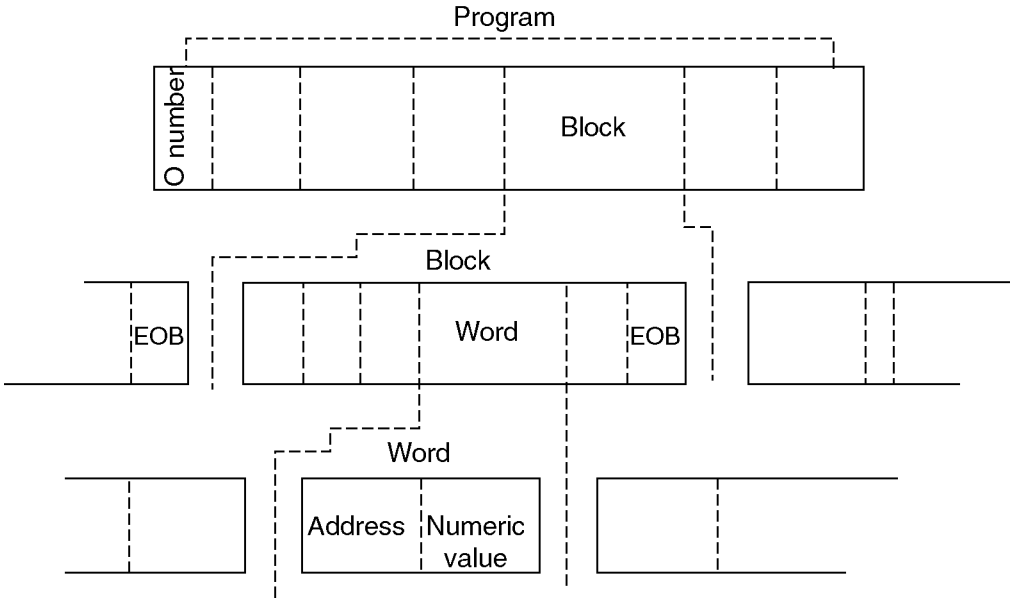


Fig. 2.1 Program structure

Next, program components are shown in Fig. 2.2.

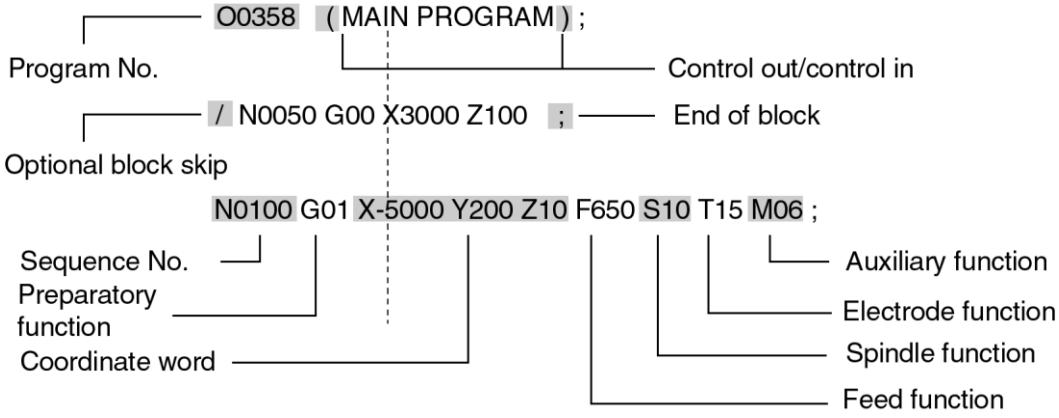


Fig. 2.2 Program component elements

The program is divided into “main program” and “subprogram”.

It is possible to store repetitive patterns and common sequences into the memory as a subprogram, and call them by the main program. This simplifies the program to a great extent.

There is no difference in structure between these programs.

Fig. 2.3 shows a program structure using a subprogram.

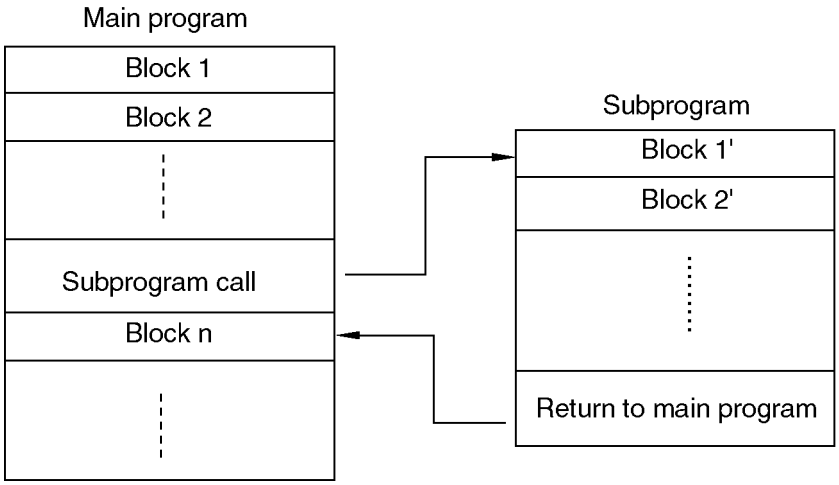


Fig. 2.3 Program structure using a subprogram

2.1 ADDRESS WORD

The “address” is a character to indicate the meaning of each numeric value when this value and the address comprise a word. The word is also the minimum significant unit in giving a command to the NC unit.

The basic address words as shown in Table 2.1, 2.2.

Table 2.1 Address word table (metric)

Function	Address	Metric	
		Standard resolution unit	High resolution unit
Program No.	O	1 - 9999	1 - 9999
Sequence No.	N	1 - 9999	1 - 9999
Preparatory function	G	0 - 999	0 - 999
Coordinates	X, Y, Z, B, C, W, I, J, K, R	±99999.999 mm ±99999.999 deg ±277.7778 rev	±99999.9999 mm ±99999.9990 deg ±277.77778 rev
Feed function	F	1 - 9999 mm/min	0.1 - 9999.9 mm/min
Spindle function	S	0 - 15	0 - 15
Electrode function	T	0 - 9999	0 - 9999
Auxiliary function	M	0 - 999	0 - 999
Calling program No.	P	1 - 9999	1 - 9999
Repeat count	L	0 - 9999	0 - 9999
Offset No.	D, H	0 - 99	0 - 99
Machining condition	E	1 - 9999	1 - 9999

Table 2.2 Address word table (inch)

Function	Address	Inch	
		Standard resolution unit	High resolution unit
Program No.	O	1 - 9999	1 - 9999
Sequence No.	N	1 - 9999	1 - 9999
Preparatory function	G	0 - 999	0 - 999
Coordinates	X, Y, Z, B, C, W, I, J, K, R	±3937.0078 inch ±99999.999 deg ±277.7778 rev	±3937.00787 inch ±99999.9990 deg ±277.77778 rev
Feed function	F	0.1 - 9999.9 inch/min	0.01 - 9999.99 inch/min
Spindle function	S	0 - 15	0 - 15
Electrode function	T	0 - 9999	0 - 9999
Auxiliary function	M	0 - 999	0 - 999
Calling program No.	P	1 - 9999	1 - 9999
Repeat count	L	0 - 9999	0 - 9999
Offset No.	D, H	0 - 99	0 - 99
Machining condition	E	1 - 9999	1 - 9999

Note 1: The resolution of command unit can be switched by the parameter.
Note 2: Up to 11 digits of numeric value including a sign and decimal point can be specified.
Note 3: Program examples in this instruction manual are given on precondition of metric system and standard resolution unit.

Next, a description of each word is given.

2.1.1. Program No. word

The NC unit can store some programs into the memory. To distinguish these stored programs from each other, “Program numbers” are provided.
The program number is used at the top (head) of a program. It is a word consisting of an address “O” and a 4-digit number (1 - 9999) that follows the address.

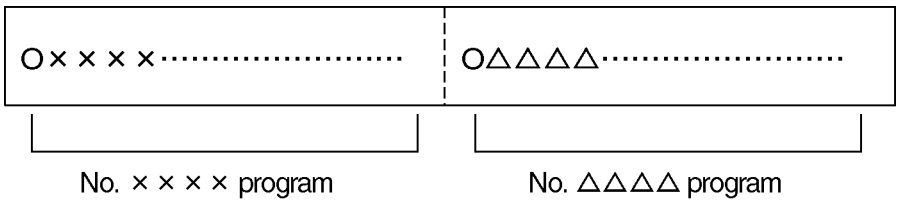


Fig. 2.4 Program No.

2.1.2 Sequence No. word

The sequence number is used as a pointer to search the specific position in a program. A sequence number is provided at the head of a block, being a word consisting of 4 digits (1 - 9999) following address “N”. The order of sequence numbers is free and allowed to skip. Sequence numbers may be provided at all blocks or some blocks.

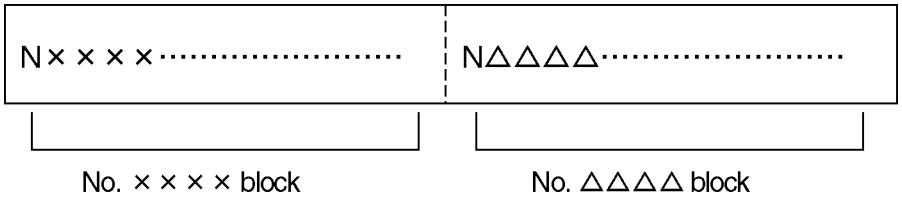


Fig. 2.5 Sequence No.

Note 1: Specify a sequence number at the head of a block.

2.1.3 Coordinate word

The coordinate word indicates a moving amount of each axis of the machine or an electrode position and is also called “dimension word”. Generally, the coordinate word consists of an address indicating an axis and a numeric value indicating an electrode position or a moving direction/amount.

As this coordinate word is used very often in programming, it will be described here in more detail.

(a) Coordinate word address

Coordinate word addresses can be classified as shown in Table 2.3.

Table 2.3 Addresses of coordinate word

Coordinate word address		Meaning of numeric value
Travel axes	X, Y, Z, W, B, C (Note)	With addresses of the basic axes and additional axes, the position/distance corresponding to the axis or an angle of the rotation axis is specified.
Circular interpolation parameters	R	A circular radius is specified.
	I, J, K	A distance from the start point to the circular center (components of the X-, Y- and Z-axes) is specified.

Note: In general, the addresses of standard axes are X, Y, Z, and those of additional axes are B, C, W, etc.

(b) Numeric value setting units

The following setting units are available:

Input unit ..... Minimum unit for inputting a program input, manual data input, electrode compensation value, etc.

Moving unit ..... Minimum moving unit of each axis on the machine side.

Display unit ..... Minimum display unit on the display.

The unit systems can be classified into metric system and inch system, and resolution unit.

Which systems are selected initially has been set by the parameters.

Table 2.4 shows each setting unit.

Table 2.4 Setting Units

Setting unit / Unit	Input/moving/display			
	metric system		inch system	
	Standard resolution	High resolution	Standard resolution	High resolution
Position/distance	0.001 mm	0.0001 mm	0.0001 inch	0.00001 inch
Angle	0.001 deg	0.0001 deg	0.001 deg	0.0001 deg
Rotation	0.0001 rev	0.00001 rev	0.0001 rev	0.00001 rev
Time	0.001 sec	0.001 sec	0.001 sec	0.001 sec

(c) Decimal point input

In setting numeric values of position, distance and time, decimal points can be used. When no decimal value is used, a numeric value indicates a multiple of the minimum setting unit.

When a decimal point is used, the position of the decimal point indicates 1 mm or 1 inch.

The addresses for which a decimal point can be used are as follows:

X, Y, Z, I, J, K, B, C, R, P, Q, W

Example: For position/distance

X15 ..... 0.015 mm or 0.0015 inch (Standard resolution)
0.0015 mm or 0.00015 inch (High resolution)

X15. ..... 15 mm or 15 inch

X15.00 ..... 15 mm or 15 inch

Next, each function word and the words related to it are briefly described.

2.1.4 Preparatory function word

This word is a numeric value of 3 digits (0 - 999) following address “G” and specifies a kind of control function in that block.

This function makes preparations for the execution of the function commanded to the NC unit and is called “preparatory function” or “G function”.

2.1.5 Feed function word

The feed word function specifies a relative rate between the workpiece and electrode and is called “F function”. Though there are various kinds of feed function words, adopts the F direct feed command.

(a) F direct feed command

This is used to specify a direct feed rate following address “F”.

2.1.6 Spindle function word

This word is used to specify a rotational speed of the rotating axis. The rotational speed of the spindle is specified indirectly in a numeric value (0 - 15) following address “S”. This function is called “S function”.

2.1.7 Electrode function word

This word is used for the machines with the electrode change function to specify the electrode number or electrode ID number with 4-digit numeric values (0-9999) following the address “T”.

2.1.8 Auxiliary function word

This word is used to specify various operations for the machine in a numeric value of 2 or 3 digits (0 - 999) following address “M”. The word is also used for a subprogram control command and program end command. This function is called “M function”.

2.1.9 Calling program number word/repeat count word

The calling program number word is used to specify a subprogram number following address “P”. This word is used to call the subprogram from the main program. The repeat count word is used to command a repeat count in a numeric value of 0 - 9999 following address “L”.

2.1.10 Offset number word

To compensate a difference in electrode diameter/length between the electrode assumed in programming and the real electrode, its offset value must be specified.

The offset number word specifies a registered number for offset value with 2-digit number (1 -99) preceded by address “D” or “H”.

Address “D” is specified for electrode diameter compensation and address “H” is specified for electrode length compensation.

Note: Among the function words, there are some that are called Modal commands.

The Modal command is a group of commands that deny one another or commands of similar motion. Among grouped commands, the previous command remains valid until another command in the same group is executed. The Modal commands selected upon power-on or RESET are preset in the NC unit.

2.1.11 Machining condition word

This word is used to specify a machining condition. This is expressed in numeric value (1 -9999) of 4 digits following address “E”.

2.2 BLOCKS

To give a motion command to the NC unit, a block is generated by gathering some words.
The block is an aggregate of continuous words that can be handled as one unit by the NC unit.
Besides the words, some characters having a function are added to it.
The NC unit performs automatic operation reading blocks one after another. The maximum number of characters in one block is limited to 192 characters.
Next, the characters to be added to the block are described.

2.2.1 End of block (EOB) ... ";" (semicolon)

The end of block (EOB) is a character to distinguish one block from the others in the program.
The range from after the EOB till the next EOB is read into the NC unit as one block.

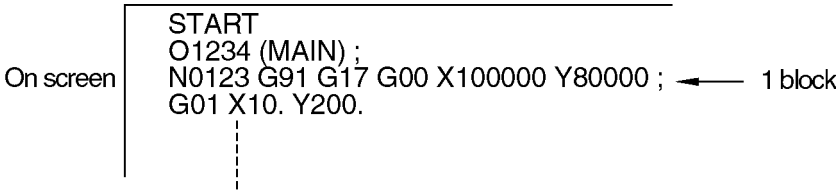


Fig. 2.6 End of block

On the NC unit screen and in this manual, the end of block is expressed with a ";" (semicolon) character.

2.2.2 Optional block skip ... "/" (slash)

In some cases, when a program is made, a check may be made before machining. In other cases, when the program uses a subprogram and this subprogram has been checked, the program check time can be reduced unless the calling block is executed.
In these cases, the optional block skip function can select a block to be executed.
To select the block to be executed by using this function, "/" (slash) is put at the head of the block to be selected and then a program is generated. If the optional block skip function of the operation panel on the machine side is turned on when the tape or memory using the program is operated, the block with this "/" (slash) is not executed. When the function is off, the block with "/" (slash) is executed.

Note 1: In storing the NC program into the memory, the program is read irrespective of "optional block skip".
Note 2: The "optional block skip" at operation is processed in reading into the NC unit and skipped information does not enter the NC unit. At this time, the next block is read at once.
If the optional block skip function is turned on after reading, the stored block is executed.
Note 3: This function is disabled while a sequence number is searched. That is, even the sequence number of the block with "/" (slash) can be searched irrespective of the ON/OFF state of the optional block skip.

2.2.3 Comment input (control out/in) ... “(” / “)”

If a program name is attached to the program number or a comment is put in the course of the program, the program will become very understandable. The comment input function makes this possible.

A comment can be made in a program, consisting of alphanumeric sandwiched with “(” from “)”.

The number of characters is 128 characters including a program block.

Example:

```
OXXXX (MAIN PROGRAM 1) ;
          |
          |
G00 X100. Y200. (START POINT) ;
          |
          |
```

Note 1: Do not enter the following codes in “(” from “)” of the program.
NUL, BS, HT, LF, “%”, “; ” ..... ISO code
NUL, BS, HT, CR, ER, “; ” ..... EIA code

Note 2: Plural comments may be used in one block. Also, “(”, “) ” are valid at any position in a block.

3. COORDINATE SYSTEM

The NC unit can move the electrode to the position that is taught to the NC unit in advance. To teach this position exactly, the coordinate words are used, and the relationship between these coordinate words is defined in the coordinate systems. The following three kinds of coordinate systems are available.

(1) Machine coordinate system
(2) Workpiece coordinate system
(3) Local coordinate system

The position of destination of the electrode is specified by coordinate works of one of these systems as required. In any coordinate system, each coordinate is set in parallel to the travel axis of the machine. The coordinate axes are set according to the standard coordinate system shown in Fig. 2.7.

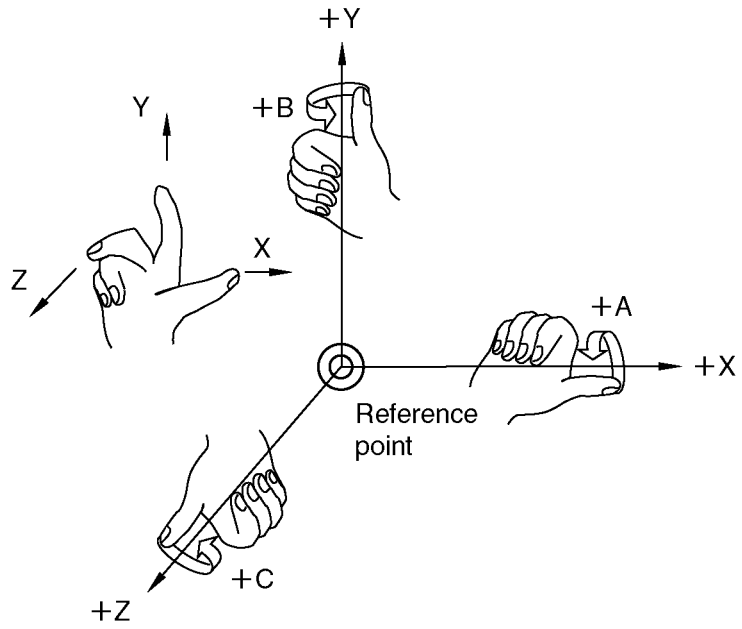


Fig. 3.1 Standard coordinate system

Each coordinate system is described below.

3.1 MACHINE COORDINATE SYSTEM

The reference point on a machine tool is called the “machine zero point” of the machine. The machine zero point depends on the model.

The coordinate system with this machine zero point as a reference point is called “machine coordinate system”.

This coordinate system is a reference coordinate system to set the other coordinate systems. That is, the other coordinate systems are set by specifying one point within this coordinate system as a zero point.

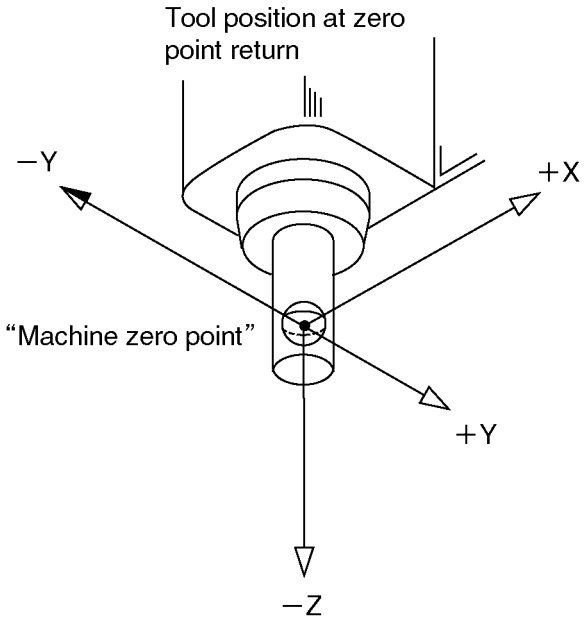


Fig. 3.2 Machine coordinates system

3.2 WORKPIECE COORDINATE SYSTEM

After a program is made in the machine coordinate system fixed to the machine, if the workpiece mounting position is changed, the position programmed in that program may be shifted from the workpiece position, causing a machining failure. To avoid this, programming is performed with the coordinate system with one point on the workpiece as a zero point.

In the workpiece coordinate system, its zero point can be set with coordinates from the machine origin, or at arbitrary position using the local coordinate system setting function in a program.

In such a case, up to 80 work coordinate systems of 8 (G54 – G61) □□10 groups (G500 – G509) can be set, so that the programming can be done by using the coordinate system set at the desired position according to the workpiece position.

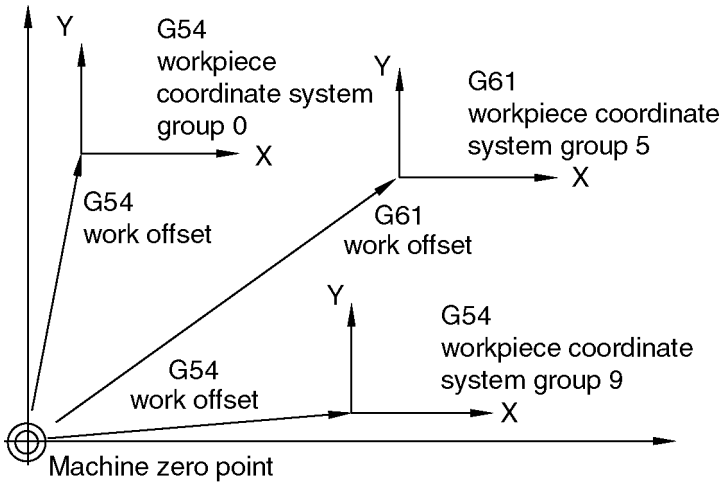


Fig. 3.3 Workpiece coordinate system

3.3 LOCAL COORDINATE SYSTEM

Programming within the workpiece coordinate system may be more easily performed if a coordinate system to be set within the workpiece coordinate system is generated.

This coordinate system having a zero point within the workpiece coordinate system is called “Local coordinate system”.

The local coordinate system can be set by specifying the zero point of the local coordinate sys-tem within the workpiece coordinate system. After the coordinate system is set, specified coordinate words indicate position in the local coordinate system.

Once a local coordinate system has been set, the local coordinate system, which is shifted by the amount of local offset, is set in all work coordinate systems (G54 – G61, G500 – G509).

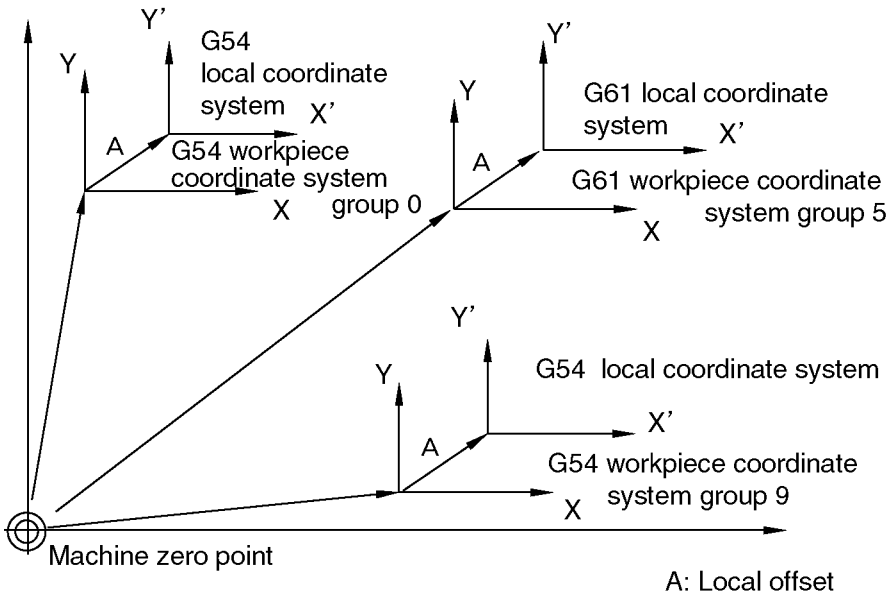


Fig. 3.4 Local coordinate system

3.4 ABSOLUTE AND INCREMENTAL COMMANDS

Regarding the electrode position, the end point of the previously executed block becomes the start point of the next block. Coordinate words are used to specify this electrode position to the NC unit. The following 2 methods are available as methods of specifying numeric values of coordinate words.

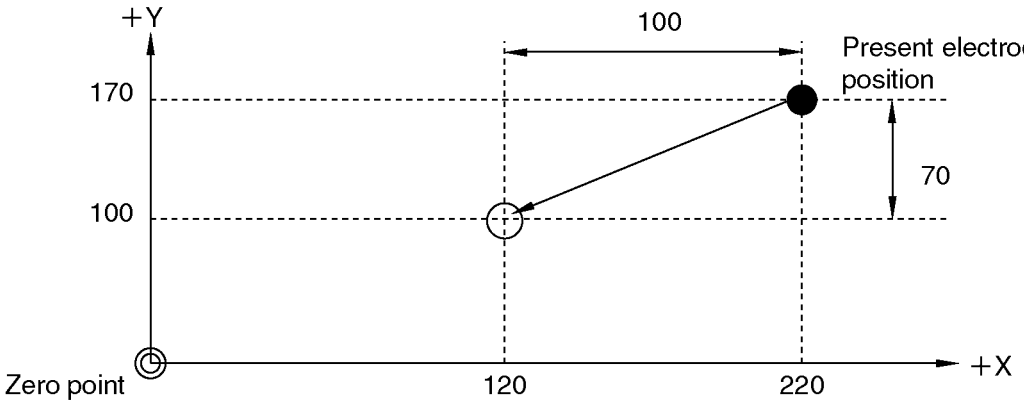
(1) Absolute command

(2) Incremental command

The major difference between these two methods is as follows: the absolute command specifies the position of destination (end point) of the electrode, while the incremental command specifies the moving amount of the electrode.

The state enabling the absolute command is called “absolute mode” and the state enabling the incremental command is called “incremental mode”.

It is possible to use both incremental command and absolute command together. However, if a mode is specified doubly in one block or another mode specification is omitted, the last specified mode is validated.



The movement from " ● " to " ○ " is specified by absolute mode.
X120. Y100. (specification of the position marked " ○ ")

The movement from " ● " to " ○ " is specified by incremental mode.
X-100. Y-70. (specification of the position marked " ○ ")

Fig.3.5 Absolute and incremental

4. FUNCTION WORDS

Function words are classified by operations and each function is described in detail.

4.1 G/M CODE LIST

Function words are classified by operations and each function is described in detail.

The following should be taken into account in reading the description of each function word.

(1) A description is made on the “function word table” shown at the head of each item.

Example:

Word	Initial	Group	Function	Item
G 04		∗	Dwell	
G 17		B	XY plane specification	

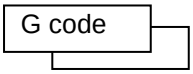
It is shown in the Group column whether it is modal command.

In the case of ∗ sign, it is not a modal command.

In the case of the alphabet, it is a modal command.

As shown in the above example, the function words described in each item, their functions and Modal command groups are mentioned in the table.

4.1.1 G CODE LIST



Word	Initial	Group	Function	Item
G00	00	A	Positioning	4.4.1
01			Linear interpolation	
02			Circular interpolation (CW)	4.4.2
03			Circular interpolation (CCW)	
04		∗	Dwell	4.4.3
09				
10		∗	Preset data change	4.5.1 4.6.2
14		∗	Unidirectional positioning	4.4.4
15				
16				
17	17	B	XY plane specification	4.2.1
18			ZX plane specification	
19			YZ plane specification	
22	22	D	Inhibit area specification	4.2.2
23			Inhibit area specification cancel	
26		∗	Rotation copy	4.10.6
27		∗	Subprogram simple call	5.2.2
28		∗	Reference point return through intermediate point	4.4.5
29		∗	Start point return through intermediate point	
30		∗	2nd - 4th reference point return through intermediate point	4.4.6
31		∗	External skip	4.4.7

G code

Word	Initial	Group	Function	Item
G40	40	E	Electrode diameters compensation cancel	4.6.3
41			Electrode diameter compensation (left)	
42			Electrode diameter compensation (right)	
43	49	F	Electrode length compensation (+ direction)	4.6.4
44			Electrode length compensation (- direction)	
45				
46				
47	47	H	Scaling cancel	4.7
48			Scaling ON	
49	49	F	Electrode length compensation cancel	4.6.4
52			Local coordinate system selection	4.2.3
53		∗	Machine coordinate system selection	4.2.4
54	54	J	Workpiece coordinate system 1 selection	4.2.5
55			Workpiece coordinate system 2 selection	
56			Workpiece coordinate system 3 selection	
57			Workpiece coordinate system 4 selection	
58			Workpiece coordinate system 5 selection	
59			Workpiece coordinate system 6 selection	
60			Workpiece coordinate system 7 selection	
61			Workpiece coordinate system 8 selection	
65		∗	Parameter call	5.2.3
66	67	#	Modal call	
67			Modal call cancel	
68	69	L	Rotation ON	4.8
69			Rotation cancel	
70	70	M	Mirror image cancel	4.9
71			Mirror image set/reset	
74	75	S	Pattern machining ON	4.10.1
75			Pattern machining cancel	
76	77	N	Jump On	4.10.2
77			Jump cancel	
78				
79				
80	80	O	Machining cancel	4.10.3
81			Automatic depth-set machining	
82			Predetermined depth-set machining	
83			Ordinary machining	
84		∗	Edge positioning	4.10.4
85		∗	Inside positioning	
86		∗	Outside positioning	
90	90	P	Absolute command	4.3.1
91			Incremental command	
92		∗	Workpiece coordinate system setting	4.2.6
93		#	Dummy command code cal	

G code

Word	Initial	Group	Function	Item
G98		*	Machining start point return	4.10.5
99		*	Machining interrupt point return	
500	500	Q	Work coordinate system group 0	4.2.5
501			Work coordinate system group 1	
500			Work coordinate system group 0	
501			Work coordinate system group 1	
502			Work coordinate system group 2	
503			Work coordinate system group 3	
504			Work coordinate system group 4	
505			Work coordinate system group 5	
506			Work coordinate system group 6	
507			Work coordinate system group 7	
508			Work coordinate system group 8	
509			Work coordinate system group 9	
958	959	R	Continuous block (profile) M/C OFF	4.10.5
959			Continuous block (profile) M/C ON	

Note 1: The G commands marked A, B, C ... S in the group will be valid until another G command of the same group is specified, once they are specified.
(The G commands like this are called "Modal G commands".)

Note 2: The Modal G command can be specified doubly in the same block.
If a G command belonging to the same group is double specified, the last specified G command is valid.

Note 3: If a G code that is not mentioned in the G code table or a G code without the corresponding option is specified, an alarm is displayed.

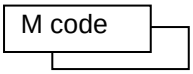
Note 4: Regarding the G commands marked "*" in the group, only the specified block is valid.

Note 5: The commands in group "#" are not described in this manual.
G66 (Modal call)
G67 (Modal call cancel)
G93 (Dummy command code call)

Note 6: Commands G100 - G137, G140, G141 are not described in this manual.

Note 7: "Initial" indicates codes of the Modal group upon power-on.
G00/G01 can be selected by parameter.
G90/G91 can be selected by parameter.

4.1.2 M CODE LIST



Word	Initial	Group	Function	Item
M 00		*	Program stop	4.11.1
01		*	Optional stop	
02		*	End of program	
03	57	A	Electrode rotation start (CW)	4.11.8
05			Electrode rotation stop	
06		*	Electrode change	4.11.7
07		*	Loading	
08		*	Unloading	
09	09	B	Blowing valve OFF	4.11.10
18		*	Electrode multiplane indexing (DH)	4.11.9
19		*	Touch stop disregard function	4.11.2
24	25	C	Polarity reverse ON	4.11.3
25			Polarity reverse OFF	
26	27	D	Machining ON	4.11.5
27			Machining OFF	
28	29	E	Dielectric fluid supply ON	4.11.4
29			Dielectric fluid supply OFF	
30		*	End of tape	4.11.1
40	09	B	10 POS valve No. 1 ON	4.11.10
41			10 POS valve No. 2 ON	
42			10 POS valve No. 3 ON	
43			10 POS valve No. 4 ON	
44			10 POS valve No. 5 ON	
45			10 POS valve No. 6 ON	
46			10 POS valve No. 7 ON	
47			10 POS valve No. 8 ON	
48			10 POS valve No. 9 ON	
49	09	B	10 POS valve No. 10 ON	4.11.10
50			Blowing ON, Low pressure blowing ON	
56	57	A	MA/MR indexing mode selection	4.11.8
57			MA/MR C-axis mode selection	

M code

Word	Initial	Group	Function	Item
M58		∗	Tank descent	4.11.11
59		∗	Tank ascent	
60		∗	Pallet change	4.11.12
70	71	F	SL machining ON	4.11.13
71			SL machining OFF	
72	73	B	High pressure blowing ON	4.11.10
73			High pressure blowing OFF	
78	79	G	Rotary axis brake ON	4.11.14
79			Rotary axis brake OFF	
98		∗	Subprogram call	4.11.6
99		∗	End of subprogram	

Note 1: The M commands marked A, B, C, ... G in the group will be valid until another M command of the same group is specified, once they are specified.
(The commands like this are called "Modal M commands".)

Note 2: The M command can be specified doubly in the same block.

① If the ON-OFF switching M command belonging to the same group is doubly specified, the ON command is executed first and the OFF command is executed last.

② Select commands such as M56 and M57 cannot be specified doubly in the same block.

Note 3: Programming the M code not registered in the M code list or not given with an option causes an alarm.

Note 4: Regarding the M commands marked "∗" in the group, only the specified block is valid.

Note 5: "Initial" indicates codes of the modal group upon power-on or RESET.

4.2 COORDINATE SYSTEM COMMANDS

This section describes the commands related to the coordinate system.

The coordinate system is used as a base to indicate the electrode position when a travel command is given to the electrode.

The commands described in this section are used for setting and change of reference coordinate system, specification of electrode travel planes, setting of an electrode moving range, etc.

Table 4.1 Coordinate Command Table

Word	Modal	Function	Item
G17 G18 G19	○ (B)	XY plane designation ZX plane designation YZ plane designation	4.2.1
G22 G23	○ (D)	Inhibit area designation Inhibit area designation cancel	4.2.2
G52	×	Local coordinate system selection	4.2.3
G53	×	Machine coordinate system selection	4.2.4
G54 G55 G56 G57 G58 G59 G60 G61	○ (J)	Workpiece coordinate system 1 selection Workpiece coordinate system 2 selection Workpiece coordinate system 3 selection Workpiece coordinate system 4 selection Workpiece coordinate system 5 selection Workpiece coordinate system 6 selection Workpiece coordinate system 7 selection Workpiece coordinate system 8 selection	4.2.5
G92	×	Workpiece coordinate reference point shift	4.2.6
G92	○	Workpiece coordinate system rotation	4.2.6
G500 G501 G502 G503 G504 G505 G506 G507 G508 G509	○ (Q)	Workpiece coordinate system group 0 selection Workpiece coordinate system group 1 selection Workpiece coordinate system group 2 selection Workpiece coordinate system group 3 selection Workpiece coordinate system group 4 selection Workpiece coordinate system group 5 selection Workpiece coordinate system group 6 selection Workpiece coordinate system group 7 selection Workpiece coordinate system group 8 selection Workpiece coordinate system group 9 selection	4.2.5

4.2.1 PLANE DESIGNATION (G17/18/19)

FUNCTION: Plane designation
G17 – XY plane designation
G18 – ZX plane designation
G19 – YZ plane designation

FORMAT: G17/G18/G19;

EXAMPLE OF STATEMENT:
G17;
G18;
G19;

EXPLANATION:

◎ This command specifies a plane for the electrode to move on at circular interpolation and electrode diameter compensation.

◎ In any coordinate system, the plane to be specified is such that includes 2 axes. This command, once any plane is specified, does not change until another plane is specified.
In the initial state when power is supplied, G17 (XY plane) is specified.

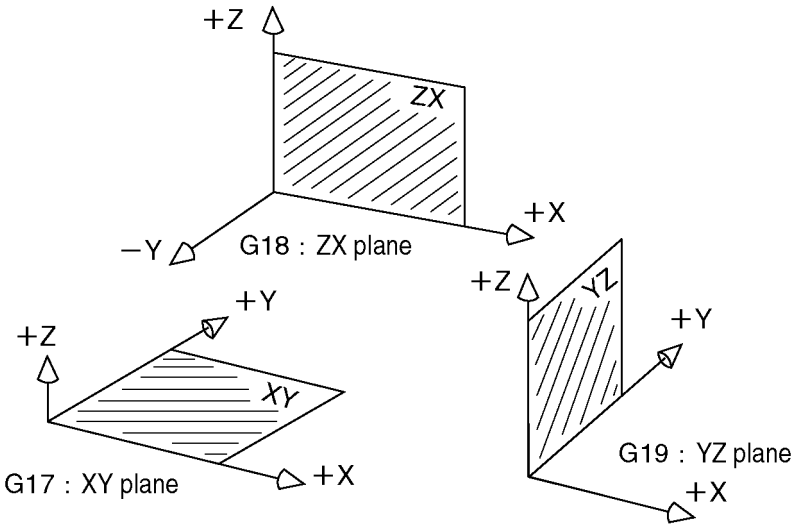


Fig. 4.1

SAMPLE: Note that at circular interpolation, quite different motion is performed if plane designation is different even when the same travel command is specified. The following sample program is an example of this case.

```
G92 X0 Y0;
G90;
G17 G02 X20. I10. ;    .....  ①
G00 X0 Y0;
G18 G02 X20. I10. ;    .....  ②
M30;
```

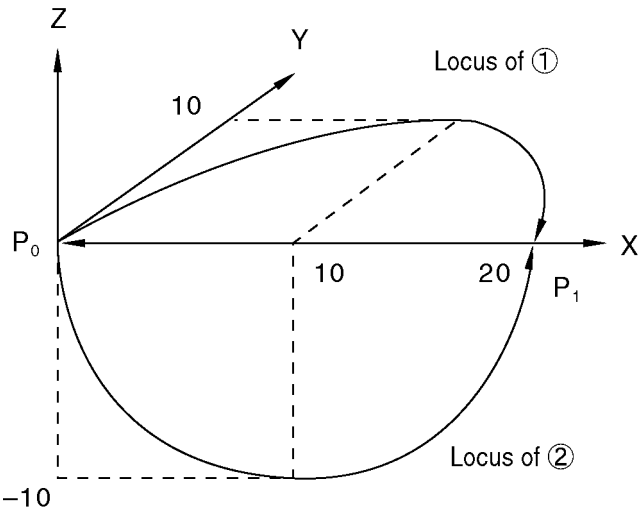


Fig. 4.2

4.2.2 INHIBIT AREA DESIGNATION (G22/23)

FUNCTION: Inhibit area designation
G22 — Inhibit area designation
G23 — Inhibit area designation cancel

FORMAT: G22 | X_Y_Z_ | | I_J_K_ | ;
G23;

EXAMPLE OF STATEMENT:
G22 X2000. Y1000. Z500. I10. J10. K10. ;
G23;

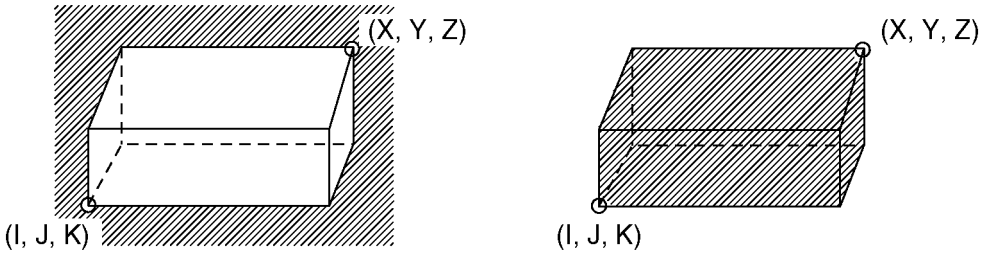
EXPLANATION:

◎ This command can make a check so that the electrode may not operate out of the moving range during automatic operation. * However, the path of a pattern operation is not taken into consideration.

◎ The surface of a rectangular parallelepiped with the 2 points specified by command G22 as opposite angles is set as an inhibit area. Either inside of outside of the rectangular parallel piped is determined as an inhibit area by parameter.

[Inhibit area setting: Parameter No. 0070]

A plane can also specified by specifying two axes in such a way as X_Y_I_J_.
This setting is performed with a coordinate value of the machine coordinate system with the machine zero point as 0.



The outside is an inhibit area. The inside is an inhibit area.

Fig. 4.3

◎ When the end point of the block to be executed enters the inhibit area, the NC unit generates an alarm to the blocks after command G22, and does not execute them. Command G23 releases the electrode entry check to the inhibit area.
When an address other than X, Y, Z, I, J and K is specified, the addresses of X-I, Y-J and Z-K do not match or a zero point return has not been completed, an alarm is caused.

Note 1: In setting the inhibit area, do not perform setting such a way that two points become the same point.

Note 2: If the electrode has entered the inhibit area when command G22 is executed, the inhibit area is determined in the next travel command block.
When the alarm is caused by entering the inhibit area, the electrode can move only in the reverse direction of its coming direction.

Note 3: Specified commands G22 and G23 separately in individual blocks.

Note 4: Specify larger coordinate values following the addresses of X, Y and Z than those for I, J and K.
Otherwise, no inside inhibit area is set, but the whole area is specified as an inhibit area when setting the outside inhibit area.

Note 5: In commands G02 and G03, the inhibit area is determined at the end point of movement. Therefore, the electrode path may enter the inhibit area.

SAMPLE:

Program)

```
G17 G90 G92 X0 Y0 Z0;
G22 X40. Y25. Z25. I0 J0 K0;   [Inhibit area setting]
G00 X20. Y10. ;                (P0 → P1)
G01 X35. ;                     (P1 → P2)
G03 Y30. J10. ;                (P2 → P3)
G01 Y10. ;
G00 X0 Y0;
M30;
```

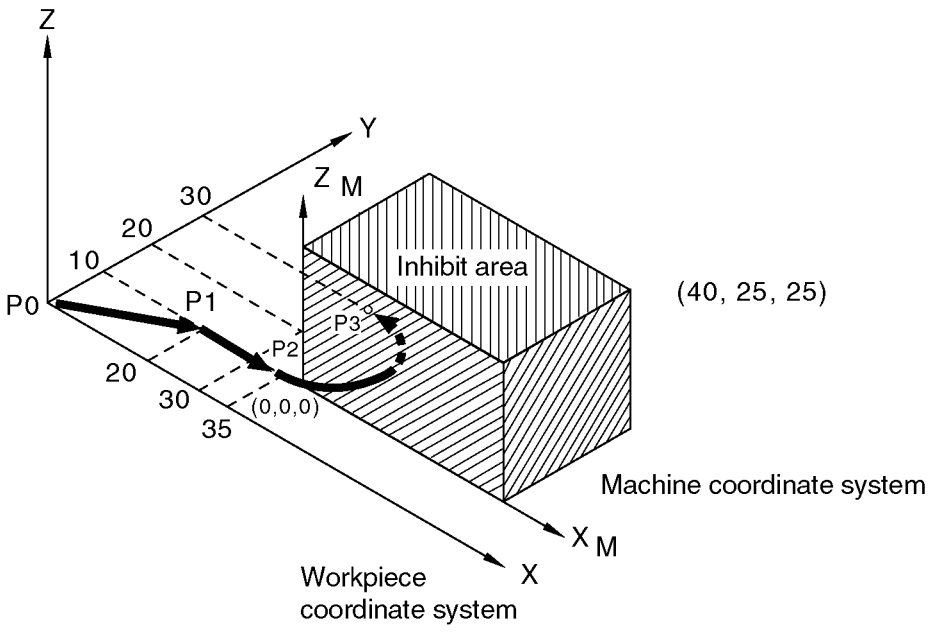


Fig. 4.4

4.2.3 LOCAL COORDINATE SYSTEM SETTING (G52)

FUNCTION: Local coordinate system setting

FORMAT: G52 | X_Y_Z_ | (C_B_W_) | ;

EXAMPLE OF STATEMENT:

G52 X200. Y100. Z10. ;

EXPLANATION:

◎ By this command, a local coordinate system can be set in the workpiece coordinate system.

◎ The coordinate words following G52 indicate one point in the workpiece coordinate system currently selected, and a local coordinate system with this point as a zero point is set.

After that, the travel command in the specified absolute mode has a coordinate value in the local coordinate system.

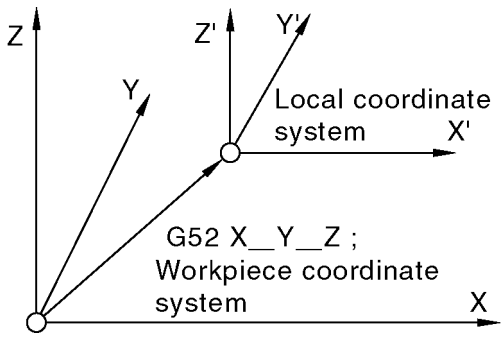


Fig. 3.5

To cancel the local coordinate system, execute one of the following commands.

(1) G52 X0 Y0 Z0 ;

Matches the zero point of the local coordinate system with the zero point of the workpiece coordinate system.

(2) G92 X_Y_Z_ ;

Perform workpiece coordinate system setting.

Note: When command G52 is executed in the incremental mode, its command value is regarded as the coordinate value to indicate one point in the workpiece coordinate system.

SAMPLE:

Program)

```
        G90;
① G92 X0 Y0 ;
        G52 X20. Y10. ;
② G01 X0 Y0 ;                (P0 → P1)
③ X30. ;                     (P1 → P2)
④ X0 Y20. ;                  (P2 → P3)
⑤ Y0;                        (P3 → P4)
⑥ X-20. Y-10. ;              (P1 → P0)
        G52 X0 Y0;
        M30;
```

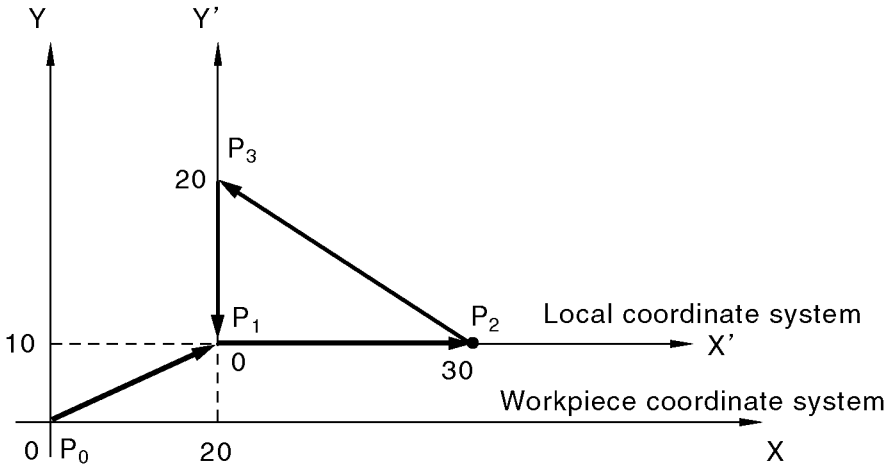


Fig. 4.6

A local coordinate system is set by specifying its zero point from the workpiece coordinate system. (①)

Movement in the local coordinate system (②, ③, ④, ⑤, ⑥).

After moving the reference point of the original workpiece coordinate system, the work coordinate system is set again.

4.2.4 MACHINE COORDINATE SYSTEM COMMAND (G53)

FUNCTION: Machine coordinate system command

FORMAT: G53 | X_Y_Z_ (C_B_W_) | ;

EXAMPLE OF STATEMENT:

G53 X100. Y50. Z200. ;

EXPLANATION:

- ◎ This command specifies a position in the machine coordinate system during program execution in the workpiece coordinate system.
- ◎ This command can be specified in the positioning (G00) and linear interpolation (G01) modes.
- ◎ Upon receipt of this command, the NC unit moves the electrode from the current position in the workpiece coordinate system to the position in the machine coordinate system, indicated by command G53.

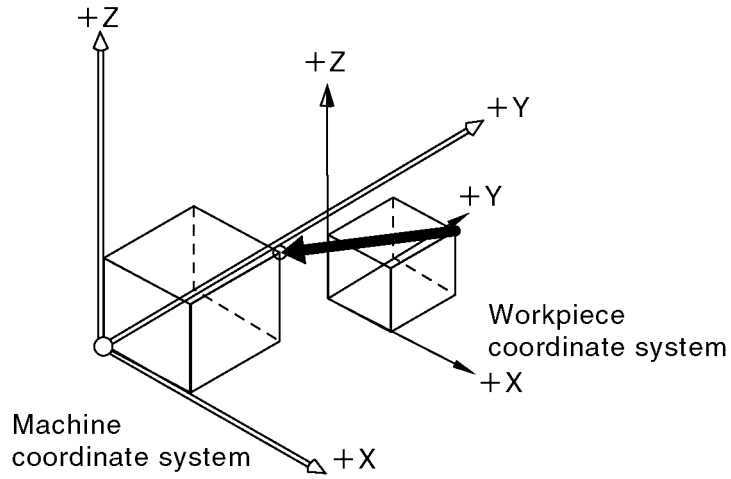


Fig. 4.7

Note 1: If the reference position return operation has not been completed before the G53 command is specified, the alarm will result.

Note 2: Coordinate words on the G53 command are specified by the absolute (G90) mode. Movement from the machine coordinate system to one point in the workpiece coordinate system must be specified in the absolute mode.
When it is returned by the incremental mode, various offset values and compensation values cannot be returned.
The G53, in general, should be executed after various offset values and compensation values are cancelled.

SAMPLE: A movement is made from the current point (P0) in the workpiece coordinate system to position P1 in the machine coordinate system. For a further movement from position P1 in the machine coordinate system to position P2 in the machine coordinate system, command G53 or the incremental command is used.
When the absolute command is executed without G53, a position in the workpiece coordinate system is indicated.

Program)

```
G90 ;
G01 X40. Y50. ;
      :
      :
G53 X15. Y40. ;          (P0 → P1)
G91 X5. Y-10. ;          (P1 → P2)
G90 X30. Y30. ;          (P2 → P3)
M30;
```

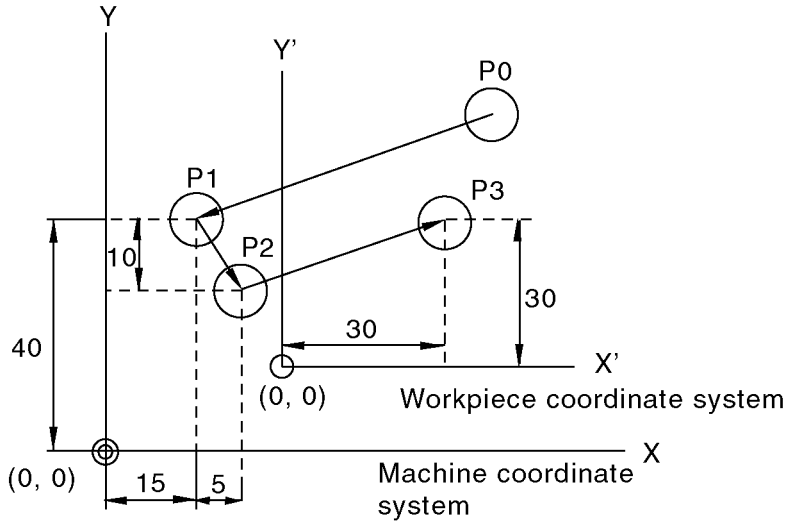


Fig. 4.8

4.2.5 WORKPIECE COORDINATE SYSTEM SELECTION (G54-61/G500-509)

FUNCTION: Workpiece coordinate system selection

G54 – Workpiece coordinate system 1
G55 – Workpiece coordinate system 2
G56 – Workpiece coordinate system 3
G57 – Workpiece coordinate system 4
G58 – Workpiece coordinate system 5
G59 – Workpiece coordinate system 6
G60 – Workpiece coordinate system 7
G61 – Workpiece coordinate system 8
G500 – Workpiece coordinate group 0
G501 – Workpiece coordinate group 1
G502 – Workpiece coordinate group 2
G503 – Workpiece coordinate group 3
G504 – Workpiece coordinate group 4
G505 – Workpiece coordinate group 5
G506 – Workpiece coordinate group 6
G507 – Workpiece coordinate group 7
G508 – Workpiece coordinate group 8
G509 – Workpiece coordinate group 9

FORMAT: G54/55/56/57/58/59/60/61 ;
G500/501/502/503/504/505/506/507/508/509;

EXAMPLE OF STATEMENT:

```
G500 G54;
     G57;
G509 G61;
```

EXPLANATION:

◎ By this command, either one of workpiece coordinate systems 1-8 currently set to the NC unit is selected from the groups 0-9 and used.

◎ A travel command specified in the absolute mode after this command indicates a position in this selected workpiece coordinate system.
The zero point of each workpiece coordinate system in the machine coordinate system is preset in a parameter in the NC unit. The value of this parameter is called “workpiece zero point offset value”.

Note 1: Upon power on, the workpiece coordinate system 1 (G54) group 0 (G500) is selected.

Note 2: This command may be specified in the same block as another command.
Avoid a Modal command except the coordinate command.

Note 3: Although there are G54-G61 in each group, G54-61 modal values do not change even if the group is selected in the range of G500-509 when modal values are in the G54-61 status.
On the contrary, the group modal values do not change even if G54-61 is specified in a certain group.

SAMPLE:

Program using workpiece coordinate systems 1 and 2.

When the workpiece zero point offset value is preset as shown below, the following program moves as shown in Fig. 4.9.

Group	Workpiece coordinate system	Workpiece zero point offset	
		X	Y
0	Workpiece coordinate system 1	50.0	10.0
0	Workpiece coordinate system 2	10.0	10.0

Program)

```
G500;
G54 G00 X0 Y0;          (    → P0)
G90 X20.0 Y20.0;        (P0 → P1)
X10.0 Y50.0;            (P1 → P2)
G55 X10.0 Y10.0;        (P2 → P3)
X0 Y0;                  (P3 → P4)
G54 X0 Y0;              (P4 → P0)
M30;
```

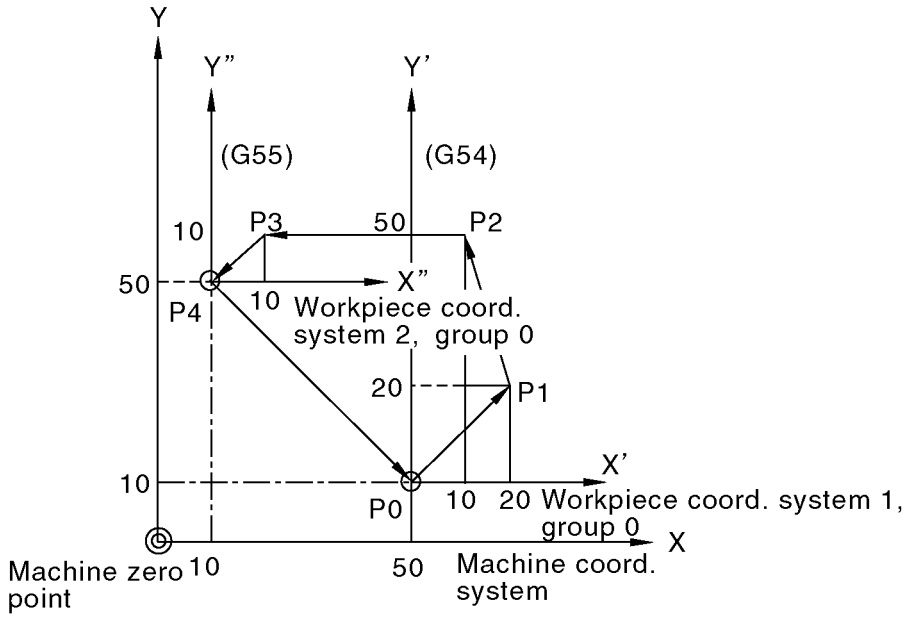


Fig. 4.9

SAMPLE:

Program using work coordinate system 1, group 0 and 1.

When the workpiece zero point offset value is set as shown below, the following program moves as shown in Fig. 4.10.

Group	Workpiece coordinate system	Workpiece zero point offset	
		X	Y
0	Workpiece coordinate system 1	50.0	10.0
1	Workpiece coordinate system 1	10.0	50.0

Program)

```
G500 G54 G00 X0 Y0;        (    → P0)
G90 X20.0 Y20.0;           (P0 → P1)
X10.0 Y50.0;               (P1 → P2)
G501 X10.0 Y10.0;          (P2 → P3)
X0 Y0;                     (P3 → P4)
G500 X0 Y0;                (P4 → P0)
M30;
```

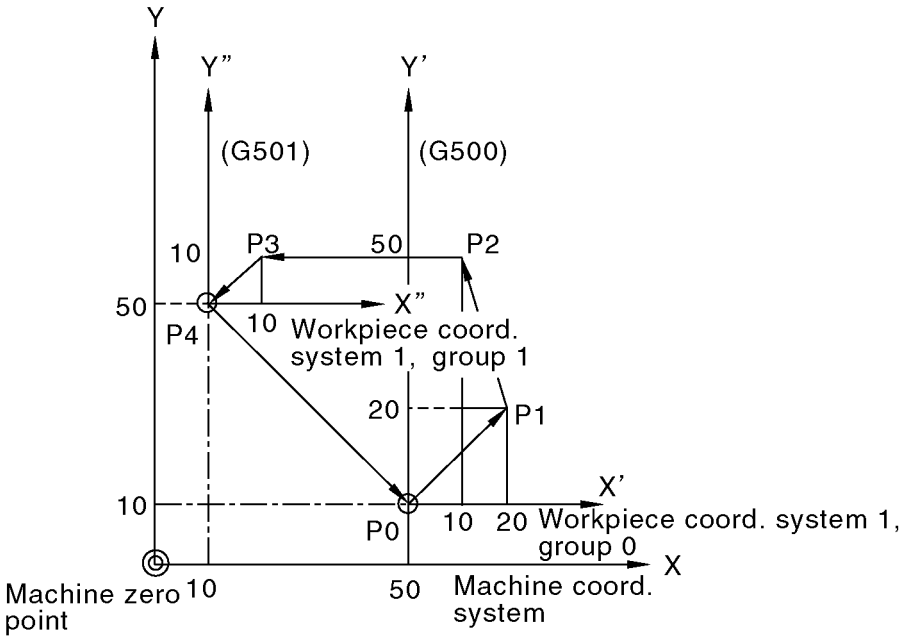


Fig. 4.10

4.2.6 WORKPIECE COORDINATE SYSTEM SETTING (G92)

FUNCTION: Workpiece coordinate system selection
Zero point shift —— ①
Rotation —— ②

FORMAT: Workpiece coordinate system selection

G92 | L_X_Y_Z_ (C_B_W_) | ----- ① Parallel movement of workpiece coordinate zero point and distance from the reference point ;

X_Y_A_ ----- ② X, Y: Coordinate value of rotation center

A0 : Rotation cancel
A1 - 9 : Coordinate rotation angle specification corresponding to parameter No. 0091 - 0099
L0 : Shift of only the workpiece coordinate system currently selected
L1 : Shift of all coordinate systems

EXAMPLE OF STATEMENT:

G92 X10.0 Y20.0 Z30.0;
G92 X20.0 Y20.0 A1;

EXPLANATION:

◎ By this command, the zero point of a work coordinate system can be shifted parallel and rotated.
Each of them is described below.

Zero point shift	G92 X_Y_Z_ ;

◎ By this command, the zero point of a work coordinate system can be changed.
◎ The workpiece coordinate system is set again so that the coordinate value specified by command G92 may be the current electrode position on the workpiece coordinate system currently selected. At this time, a difference between the original workpiece zero point offset and the changed workpiece zero point offset is added to one or all workpiece zero point offsets and then one or all workpiece coordinate systems are shifted by the same amount.

Note 1: When command G92 has been executed in the compensation mode, a coordinate system is set so that the coordinate value before compensation may correspond to the position to be specified by command G92.

Note 2: In the electrode diameter compensation mode, command G92 stops diameter compensation temporarily.

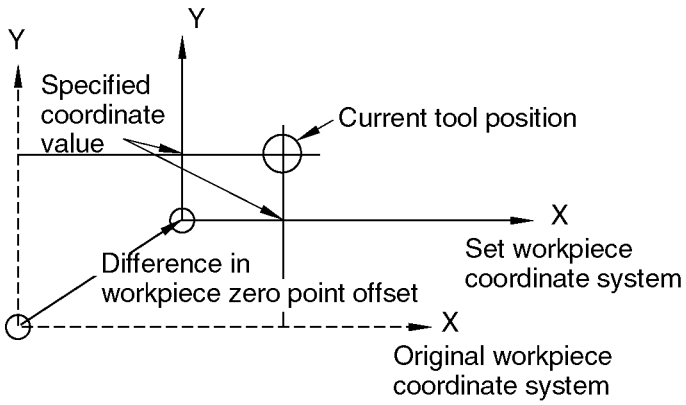


Fig. 4.11

Rotation	G92 X_Y_A_ ;

◎ By this command, the subsequent program is rotated about the specified coordinates by the angle set to the parameter given with “A”.

◎ A1 to A9 correspond to the rotation angle set to the parameter No. 0091 to 0099 respectively. Also, the “A0” is used to cancel the coordinate system rotation.

Note 1: Coordinate system rotation can not be cancelled by the RESET key.

Note 2: Do not use this command together with the model plan.

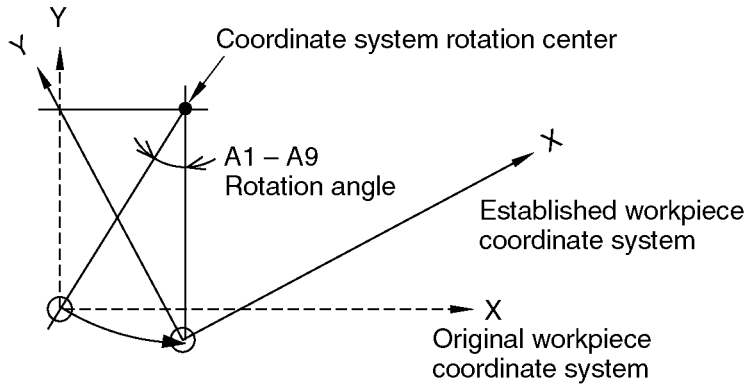


Fig. 4.12

SAMPLE:

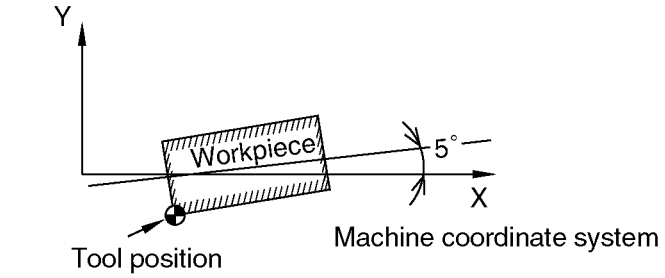
In mounting a workpiece on the machine, it may not be set in parallel to the travel axis of the machine or a fine adjustment may not be made satisfactorily. These problems can be solved by this command G92.

Before starting the program

Register a difference in angle between workpiece coordinate system used in a program and axial direction of machine to the parameter.
Adjust the electrode position to the reference position of a workpiece.

Program)

```
G92 X0 Y0;          [Workpiece coordinate system setting by electrode position]
G92 X0 Y0 A1;       [Coordinate system rotation]
G92 X20.0 Y20.0;    [Setting of the coordinate system corresponding to the program]
G90 G00 X0 Y0;      (    → P0)
X20.0 Y20.0;        (P0 → P1)
X80.0;              (P1 → P2)
Y50.0;              (P2 → P3)
X20.0;              (P3 → P4)
Y20.0;              (P4 → P1)
X0 Y0;              (P1 → P0)
G92 X20. Y20. A0;   [Coordinate system rotation cancel]
M30;
```



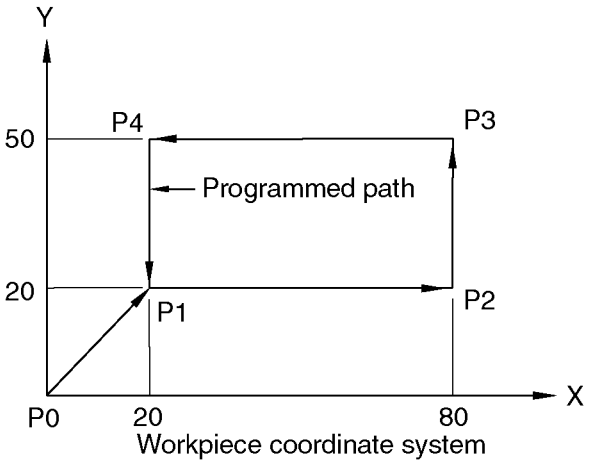


Fig. 4.13

4.3 SELECTION OF COMMAND VALUE INPUT UNIT SYSTEM

This section describes the commands to set the unit systems for numeric values used in the program and its input methods.

Table 4.2 Command Value Input Mode

Word	Modal	Function	Item
G90	○	Absolute mode	4.3.1
G91	(P)	Incremental mode	

4.3.1 ABSOLUTE/INCREMENTAL (G90/91)

FUNCTION: Command mode selection
G90 – Absolute command
G91 – Incremental command

FORMAT: G90/91 | X_Y_Z_ (C_B_W_) | ;

EXAMPLE OF STATEMENT:

G90 X25.0 Y20.0 ;
G91 X-25.0 Y-20.0 ;

EXPLANATION:

◎ The absolute command (G90) specifies all the points to move, by coordinate values from the workpiece coordinate zero point.

◎ The incremental command (G91) specifies the point to move, by an incremental value from the current point.

(1) Absolute mode (G90 X25.0 Y20.0 ;)

(2) Incremental mode (G91 X-25.0 Y-20.0 ;)

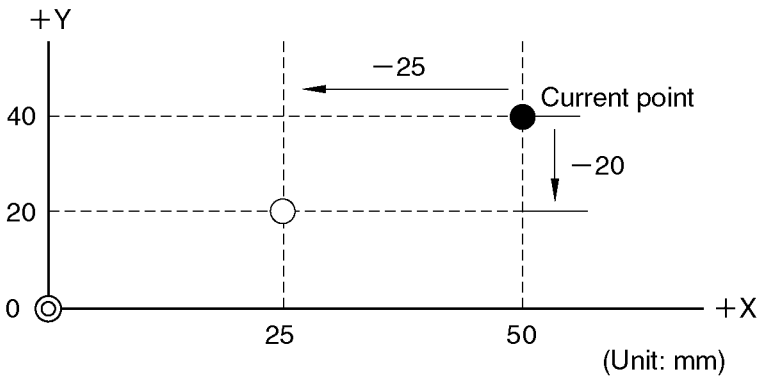


Fig. 4.14

Note: Upon power-on in the NC unit, selection between G90 and G91 is set in a parameter.
[G90/G91 selection: Parameter No. 60]

SAMPLE:

Program	
G92 X0 Y0;	[X'Y' workpiece coordinate system setting]
G90 G01 X10.0 Y20.0 F1 ;	(P0 → P1)
X30.0 Y30.0;	(P1 → P2)
G91 X30.0 Y-10.0;	(P2 → P3)
G92 X20.0 Y10.0;	[X"Y" workpiece coordinate system setting]
X10.0 Y20.0;	(P3 → P4)
G90 X40.0 Y10.0;	(P4 → P5)
M30;	

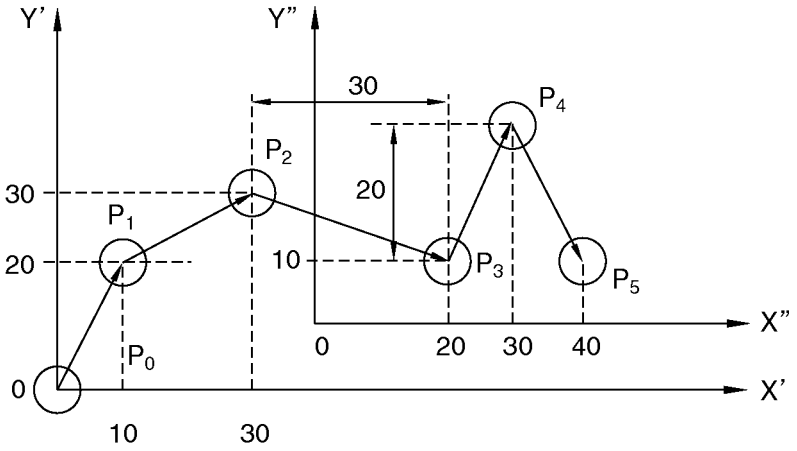


Fig. 4.15

4.4 TRAVEL COMMANDS

This section describes the commands related to movements.

The travel commands are those that control electrode movements such as positioning, linear interpolation, feed rate and dwell and so on.

The function words (commands) described in this section are as follows.

Table 4.3 Travel Command Table

Word	Modal	Function	Item
G00 G01 G02 G03	○ (A)	Linear interpolation (rapid traverse) Linear interpolation (F feed for machining)	4.4.1
		Circular interpolation (CW) Circular interpolation (CCW)	4.4.2
G04	×	Dwell	4.4.3
G14	×	Unidirectional positioning	4.4.4
G28 G29	×	Reference point return through intermediate point Reference point return/start point return through intermediate point	4.4.5
G30	×	2nd-4th reference point return through intermediate point	4.4.6
G31	×	External skip function	4.4.7
F	○	Feed function	4.4.8

4.4.1 LINEAR INTERPOLATION (G00/01)

FUNCTION: G00 – Linear interpolation (rapid traverse)
G01 – Linear interpolation (F feed for machining)

FORMAT: G00 | X_Y_Z_ (C_B_W) | ;

EXAMPLE OF STATEMENT:

G00 X200.0 Y100.0;

EXPLANATION:

◎ By this command, the axis can be moved to the specified position at the maximum feedrate (rapid traverse).

◎ Regarding an axis on which no movement occurs, omission is allowed.

◎ The axes move linearly from start point to end point at rapid traverse. (See Fig.4.16)

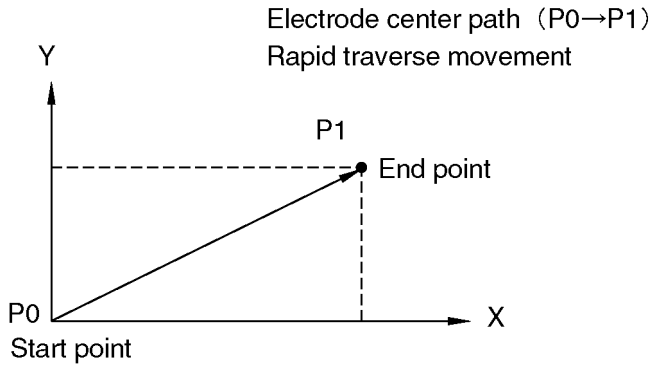


Fig. 4.16

The feedrate depends on the model.

For the rapid traverse command, the plane designation does not need to be taken into account.

SAMPLE:

Program I (Absolute)

```
G92 X0 Y0 Z0;
G90 G00 X30. Y20. ;        (P0 → P1)
Y50. ;                     (P1 → P2)
X70. Z20. ;                (P2 → P3)
X60. Y20. Z10. ;           (P3 → P4)
M30;
```

Program II (Incremental)

G91 G00 X30. Y20;	(P0 → P1)
Y30. ;	(P1 → P2)
X40. Z20. ;	(P2 → P3)
X-10. Y-30. Z-10. ;	(P3 → P4)
M30;	

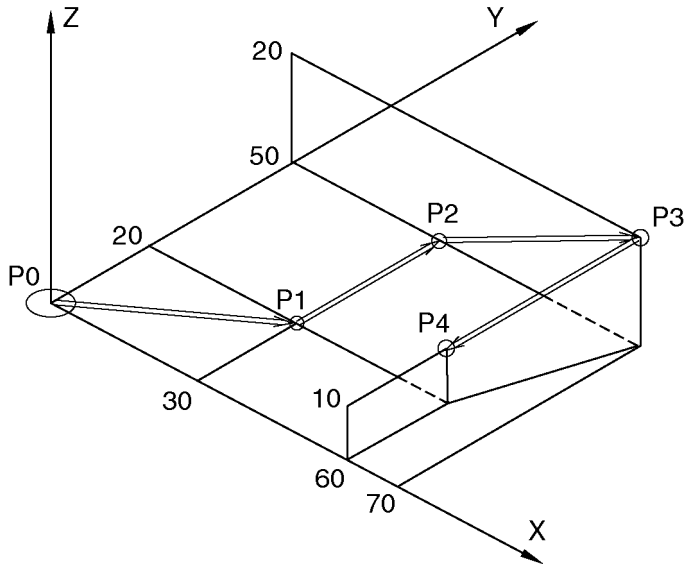


Fig. 4.17

FORMAT:

G01 | X_Y_Z_ (C_B_W_) F_ |; F: Feed function word

EXAMPLE OF STATEMENT:

G01 X50.0 Y30.0 F100;

EXPLANATION:

◎ This command permits the electrode to linearly move from the start point being the current position to the specified point (end point) at the specified feed rate by the feed function. The rate on the way differs the with machining ON (M26) or machining OFF (M27).

◎ For command G01 in the machining OFF mode, the F-feed command is needed in that block or before it. If this command is given without feed rate designation, current modal F is selected. The feed rate is a linear rate to the forward direction of the electrode. (For the feed rate designation, see the F command.)
For command G01, plane designation is not needed.

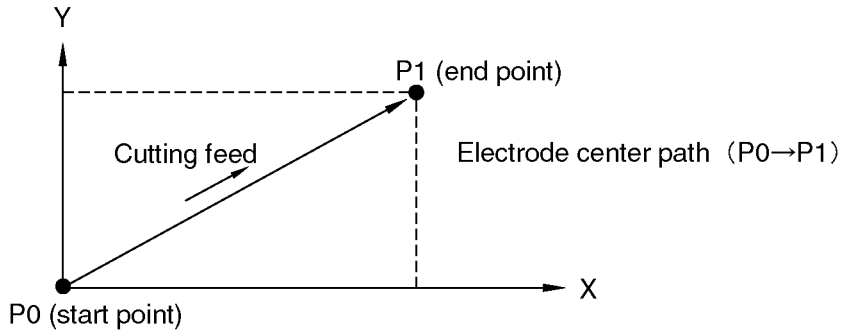
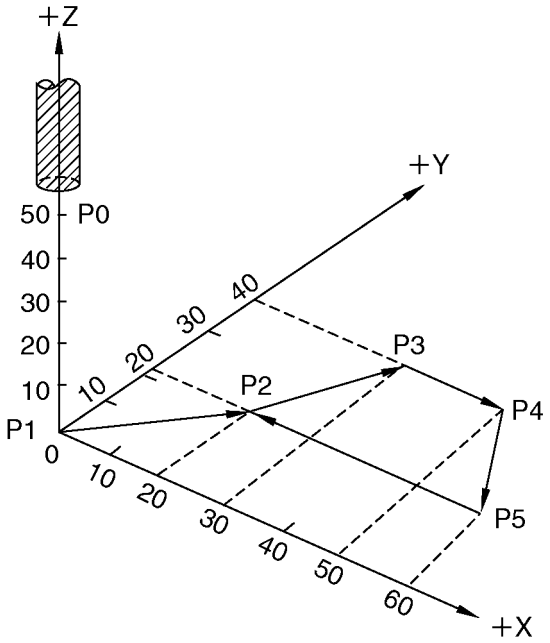


Fig. 4.18

SAMPLE: Program using command G00/G01

Incremental	Absolute	Electrode center path	Feed rate
O0001 (INC);	O0002 (ABS);		
G91;	G90 G92 X0 Y0 Z50.;		
G00 Z-50.0;	G00 Z0;	（P0→P1）	Rapid traverse
X20.0 Y20.0;	X20.0 Y20.0;	（P1→P2）	Rapid traverse
G01 X10.0 Y20.0;	G01 X30.0 Y40.0;	（P2→P3）	F feed
X20.0;	X50.0;	（P3→P4）	F feed
X10.0 Y-20.0;	X60.0 Y20.0;	（P4→P5）	F feed
X-40.0;	X20.0;	（P5→P2）	F feed
G00 X-20.0 Y-20.0;	G00 X0 Y0;	（P2→P1）	Rapid traverse
Z50.0;	Z50.0;	（P1→P0）	Rapid traverse
M30;	M30;		



(Unit : mm)

Fig. 4.19

4.4.2 CIRCULAR INTERPOLATION (G02/03)

FUNCTION: Circular interpolation
G02 – Clockwise (CW)
G03 – Counterclockwise (CCW)

FORMAT:

G02/03	X__ Y__	R__ I__ J__	F__	;
G02/03	X__ Z__	R__ I__ K__	F__	;
G02/03	Y__ Z__	R__ J__ K__	F__	;

R : Radius value
I, J, K : Incremental value from the start point of an arc to its center

EXAMPLE OF STATEMENT:

G02 X20.0 Y-10.0 I5.0 J-15.0 ;
G03 Y50.0 Z20.0 R60.0 F300.0 ;

EXPLANATION:

◎ By this command, the axis can be moved from current point to end point along a circular arc drawn with the specified radius or in the center of specified point.
Clockwise (G02) and counterclockwise (G03) are determined by viewing the negative direction of the Z-axis (Y-axis, X-axis) to the XY plane (ZX plane, YZ plane) in the right-hand orthogonal coordinate system.

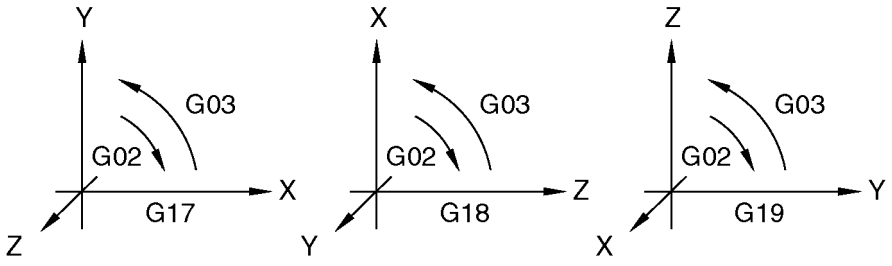


Fig. 4.20

◎ If two axes of X, Y and Z are specified, a plane with an arc can be determined by a combination of them. If designation is made by R, the plane will be specified by the plane selecting command (G17 - G19).
Between the plane specified by the plane designation command and the plane specified by two axes, the latter has priority.
An arc can be specified by the following two methods, i.e., circular center coordinate value (I, J, K) and circular radius (R). These are described below.

Specification by center coordinate value:

(1) By I, J, K (circular center coordinate value)

The circular center is specified by addresses I, J, K corresponding to X-, Y- and Z-axes. The numeric value following I, J, K is a component in the direction of each axis when viewing the circular center from the start point of an arc, and is always specified by an incremental value irrespective of G90/G91.

Example: G02 X20.0 Y-10.0 I5.0 J-15.0;

The operation is shown below. (See Fig.4.21)

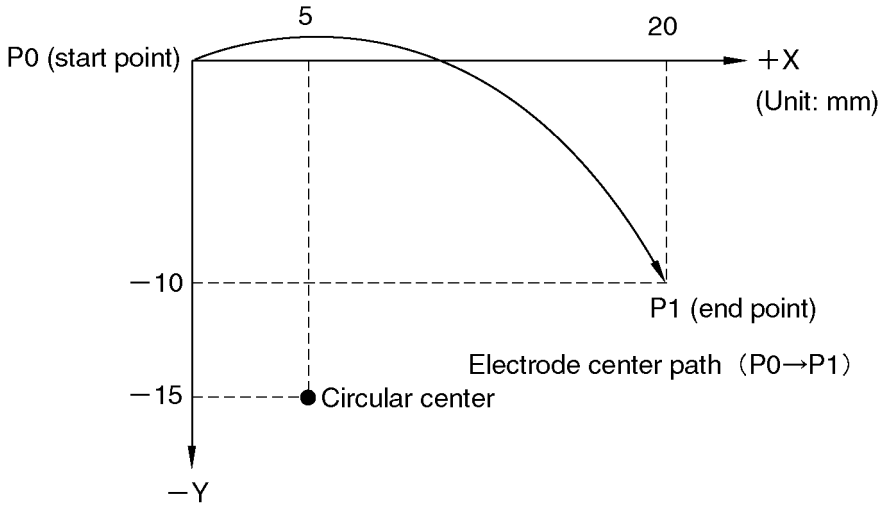


Fig. 4.21

In the case of an arc whose start point coincides with its end point, namely, a circle, coordinates of the end point does not need to be specified.

Specification by circular radius:

(2) To specify circular interpolation by R (circular radius) instead of the center coordinate by I, J, K, two types of arcs are conceivable, i.e., one is that the central angle by start point ->circular center ->end point is 180 degrees or less and the other is that this central angle exceeds 180 degrees.

In the latter type, specify R (radius) by a negative value.

Example: For the arc shown in Fig.3.22.

i) $0 < \theta \leqq 180$

G90 G02 X60.0 Y15.0 R30.0 ; ... Arc ①

ii) $180 < \theta \leqq 360$

G90 G02 X60.0 Y15.0 R-30.0 ; ... Arc ②

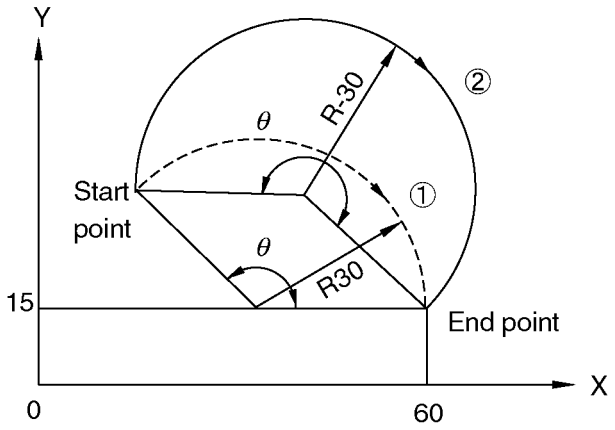


Fig. 4.22

Note 1: When the central angle is 355 degrees or more, an error may occur. So specify in two blocks.

Note 2: In an arc whose central angle is 180 degrees and whose start point and end point are not on the central axis of R (radius), an error may occur. So specify in two blocks.

◎ The feed rate of circular interpolation is the cutting feed rate specified by the F function. Control is provided so that the rate along the arc (in the tangential direction of the arc) may be the specified feed rate.

CAUTION:

In this unit, when there is a program error in the coordinate values of the end point, an axis is forcibly moved to the end point if one axis matches the command value though the other axis differs from the command value but is found within the tolerance set in a parameter. The shape of the workpiece becomes different from the programmed shape. Therefore, try to minimize the program error by increasing the computational precision.

When the program error is larger than the tolerance set in the parameter, an alarm is caused.

[Circular end point error tolerance: Parameter No. 80]

G02 X16.0 Y-7.0 I5.0 J-11.0 F2;

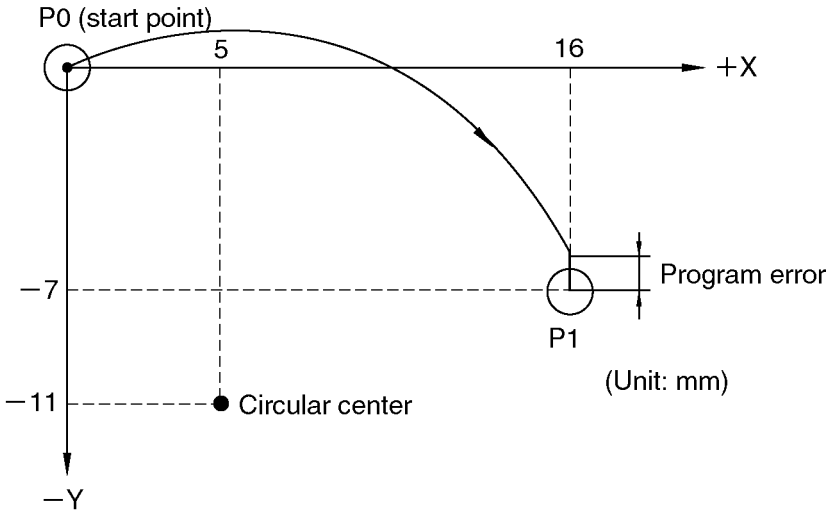


Fig. 4.23

SAMPLE:

The path shown in Fig. 4.24 is programmed by absolute and incremental commands.

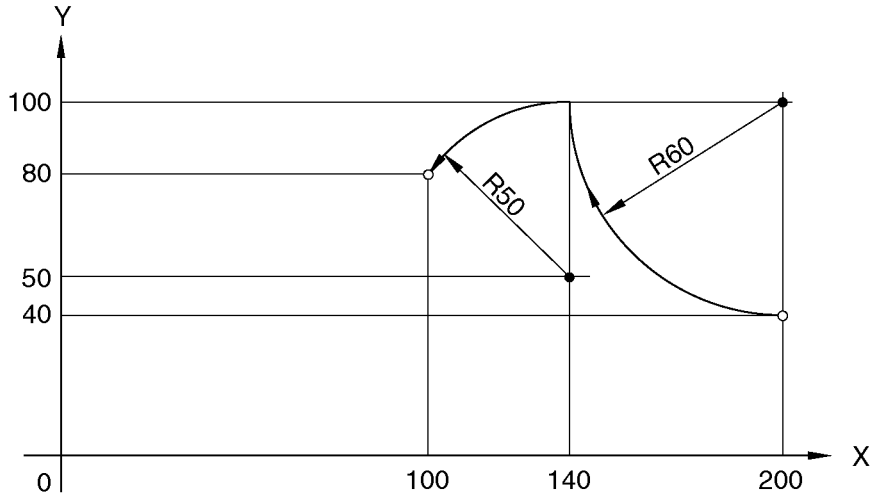


Fig. 4.24

	Absolute	Incremental
Circular Center Coordinate value	G92 X200.0 Y40.0 Z0; G90 G02 X140.0 Y100.0 J60.0 ; G03 X100.0 Y80.0 J-50.0;	G91; G02 X-60.0 Y60.0 J60.0 ; G03 X-40.0 Y-20.0 J-50.0 ;
Circular radius	G92 X200.0 Y40.0 Z0; G90 G02 X140.0 Y100.0 R60.0 ; G03 X100.0 Y80.0 R50.0;	G91; G02 X-60.0 Y60.0 R60.0 ; G03 X-40.0 Y-20.0 R50.0 ;

4.4.3 DWELL (G04)

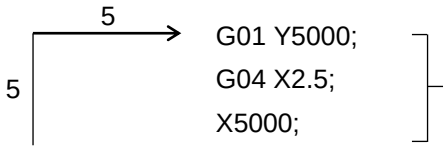
FUNCTION:

Specify the G04 when the axis is to be stopped for certain time during execution of a program. The dwell time is set with the address word X. The least input increment and the command value range are as listed below.

Dwell X	Least input increment	Command value range
Metric	0.001 sec	0 - 99999.999 sec
Inch	0.0001 sec	0 - 9999.9999 sec

FORMAT: G04X (P)__;

EXAMPLE OF STATEMENT:



The Axis stops for 2.5 sec before movement of X5 mm.

EXPLANATION:

◎ A discharge gap will be increased because the axis movement stops under the machining condition, if the machine is under machining in the continuous block (contour) machining ON mode (G959).

◎ Do not program the G04 along with the travel command in the same block.
[Example]
G04 X5000 G01 Y5000; is treated as G01 X5000 Y5000;

◎ Since the G04 is used in the block without movement, it should not affect the function such as diameter compensation that have restriction for the block without movement.

◎ G04 is a one-shot G code.

◎ The timer control unit is 0.05 sec and therefore the time less than this time cannot be set in actual.

4.4.4 UNIDIRECTIONAL POSITIONING (G14)

FUNCTION: Unidirectional positioning

FORMAT: G14 | X_Y_Z_ (C_B_W_) | ;

EXAMPLE OF STATEMENT:

G14 X100.0 Y100.0;

EXPLANATION:

◎ This command carry out precise positioning excluding backlash.

◎ This command is given by command G00/G01, but must be specified each time because it is not a Modal command.

◎ When this command is given, the NC unit performs the following operations depending on moving directions. (See Fig. 4.25.)

(1) If positioning is executed in the opposite direction against the determined direction

After a movement is made by the determined approach distance over the specified end point, positioning is performed from the determined positioning direction.

(2) If positioning is executed in the same direction against the determined direction

After a movement if stopped at the approach distance from the end point, positioning is performed.

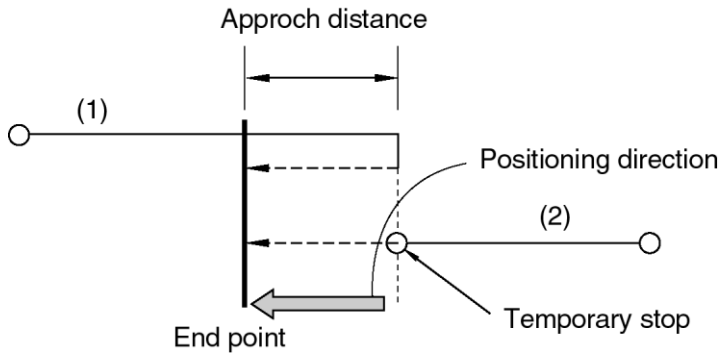


Fig. 4.25

The approach distance and positioning direction are preset in parameters. The direction is expressed by +/- of the approach distance.

[*Axis unidirectional positioning approach distance: Parameter No. *073]

Note 1: During machining (M26), unidirectional positioning is ignored.

Note 2: During electrode diameter compensation, unidirectional positioning (G14) cannot be specified.

SAMPLE:

Example of the case where the approach distance is set at -50 mm on the X and Y axes.

Program)

```
N001 G90;
N002 G14 X100.0 Y100.0;   ……   (    → P0)
N003 G14 X350.0;          ……   (P0 → P1)
N004 G14 Y300.0;          ……   (P1 → P2)
N005 G14 X100.0;          ……   (P2 → P3)
```

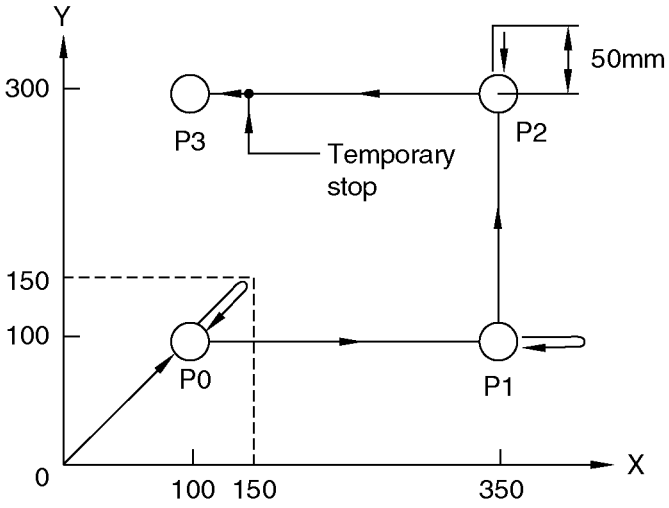


Fig. 4.26

4.4.5 REFERENCE POINT RETURN (G28/29)

FUNCTION: G28 – Reference point return via intermediate point
G29 – Automatic return from reference point

FORMAT: G28 | X_Y_Z_ (C_B_W_) L_ | ;
L0: Reference point return
L1: Reference point search

EXAMPLE OF STATEMENT:
G28 X100. Y20. ;

EXPLANATION:

◎ By this command, the specified axes can be returned to the reference point automatically. (Use this command, for example, if ATC unit is attached.)

◎ The coordinate words used together with G28 specify the position to which the axes are positioned temporarily before they return to the reference point.
The specification is possible in both absolute and incremental modes.
The coordinate value specified in the absolute mode is within the workpiece coordinate system.
The point specified and defined here is called "reference point return intermediate point". However, the intermediate point designation does not function in the first reference point return after the NC power on.

◎ Upon receipt of this command, the NC unit performs positioning at the specified intermediate point at the Modal rate and positioning of the axis on which the intermediate point has been specified between the intermediate point and the reference point, at the Modal speed.

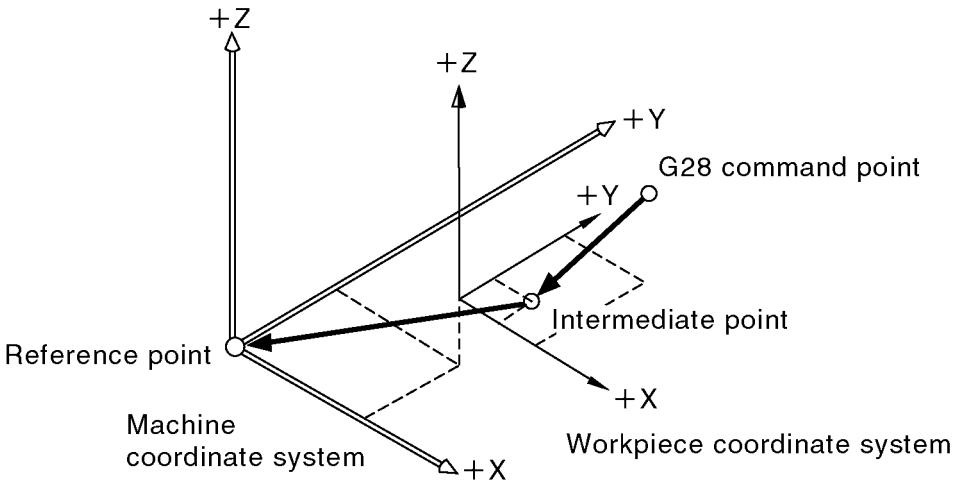


Fig. 4.27

SAMPLE:

The following programs are used to automatically return the electrode that reached a position after machining, to the machine zero point (reference point).

Program I (Path ... ①)	Program I (Path ... ②)
O101;	O202;
|	|
G91;	G91;
G28 X-25.0 Y-35.0;	G28 X0 Y0;
(P0→P1→P2)	(P0→P2)

With program I, after positioning is performed at the intermediate point (P1) indicated by command G28, positioning is performed at the reference point.
With program II, positioning is directly performed at the reference point because the intermediate point indicated by command G28 is the current point (P0).

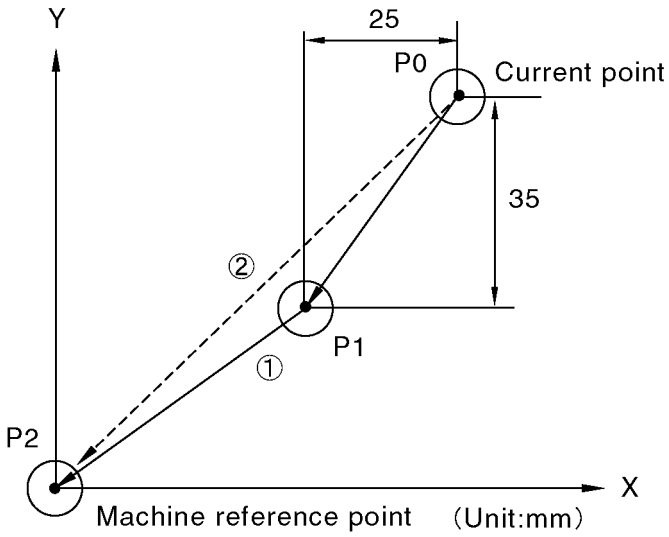


Fig. 4.28

FORMAT: G29 | X_Y_Z_ (C_B_W_) | ;

EXAMPLE OF STATEMENT:

G29 X150.0 Y50.0;

EXPLANATION:

◎ This command is specified immediately after command G28 (automatic return to the reference point) and permits positioning at the specified end point from the reference point through the intermediate point of reference return.

◎ The end point is specified in either absolute or incremental mode.
With the incremental command, the moving amount from the intermediate point of reference point return is specified.

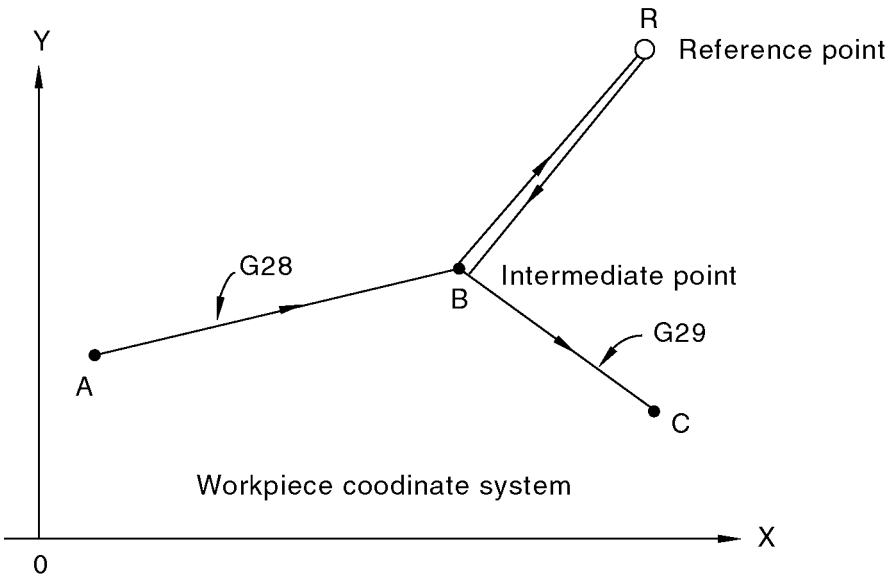


Fig. 4.29

Note: If command G29 is executed without executing command G28, an alarm is caused.

SAMPLE:

A program example of the case where command G29 is executed after the execution of command G28 is shown below.

Program:

```
O303;
G91;
G28 X-35.0 Y-30.0;        (P0 → P1 → P2)
G29 X40.0 Y-20.0;     (P2 → P1 → P3)
```

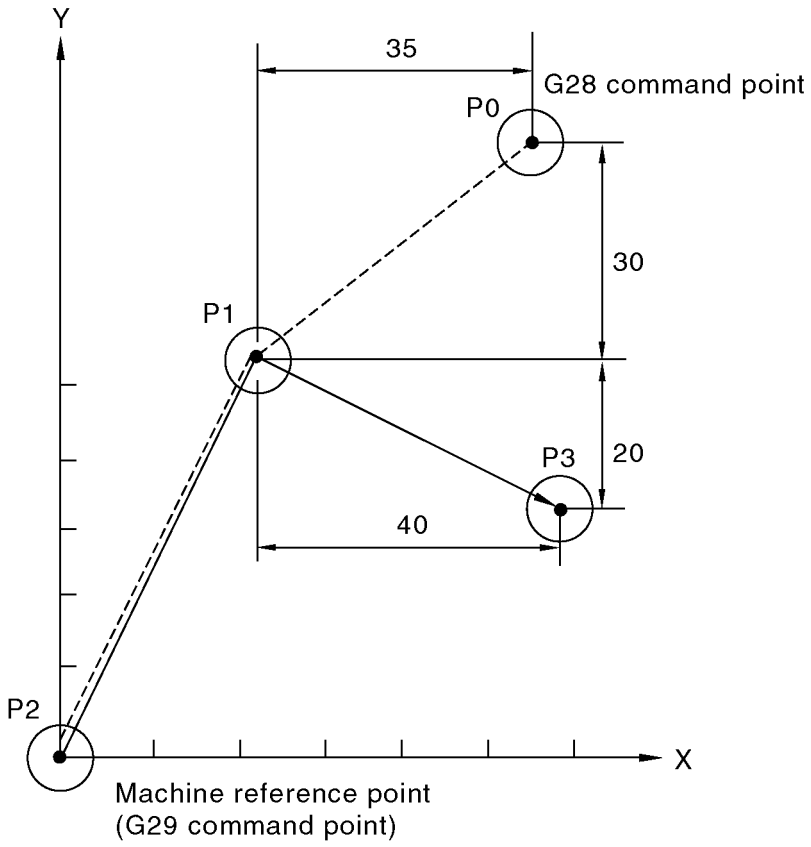

Y
35
G28 command point
P0
30
P1
20
P3
40
P2
X
Machine reference point
(G29 command point)

Fig. 4.30

4.4.6 2ND-4TH REFERENCE POINT RETURN (G30)

FUNCTION: 2nd-4th reference point return through intermediate point

FORMAT: G30 P2/3/4 | X_Y_Z_ (C_B_W_) | ;

Intermediate point

2 ··· 2nd reference point

3 ··· 3nd reference point

4 ··· 4nd reference point

EXAMPLE OF STATEMENT:

G30 P2 X50.0 Y100.0;

EXPLANATION:

◎ By this command, the axes can be returned to either 2nd, 3rd, or 4th reference point via intermediate point automatically. (Use this command, for example, if the pallet changer is attached.)

◎ The coordinate words used together with G30 specify the position to which the axes are positioned (intermediate point) before they return to the reference point. This command can be used in both absolute and incremental modes.
The reference point for return is specified by address “P”.

◎ The NC unit, when receiving this command, positions the axes to the specified intermediate point at rapid traverse, and further positions them to the specified reference point at rapid traverse.

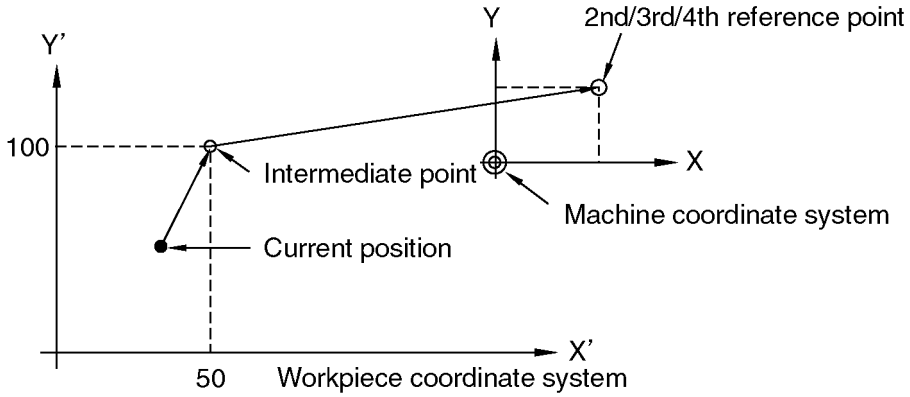


Fig. 4.31

Note: Set the 2nd - 4th reference point in the machine coordinate system by the parameter.
[*Axis 2nd - 4th reference point : Parameter No. *070 - *072]

SAMPLE:

The following is the case where the 2nd reference point is set by the parameter as follows.

No	DATA	COMMENT
1070	10000	X 2ND REF POINT·····X-axis
2070	10000	Y 2ND REF POINT·····Y-axis
3070	0	Z 2ND REF POINT·····Z-axis

Program)

```
G92 X0 Y0;
G90 G00;
      :
      :
G30 P2 X30. Y40. ;      (P0 → P1 → P2)
G29 X50. Y20. ;         (P2 → P1 → P3)
```

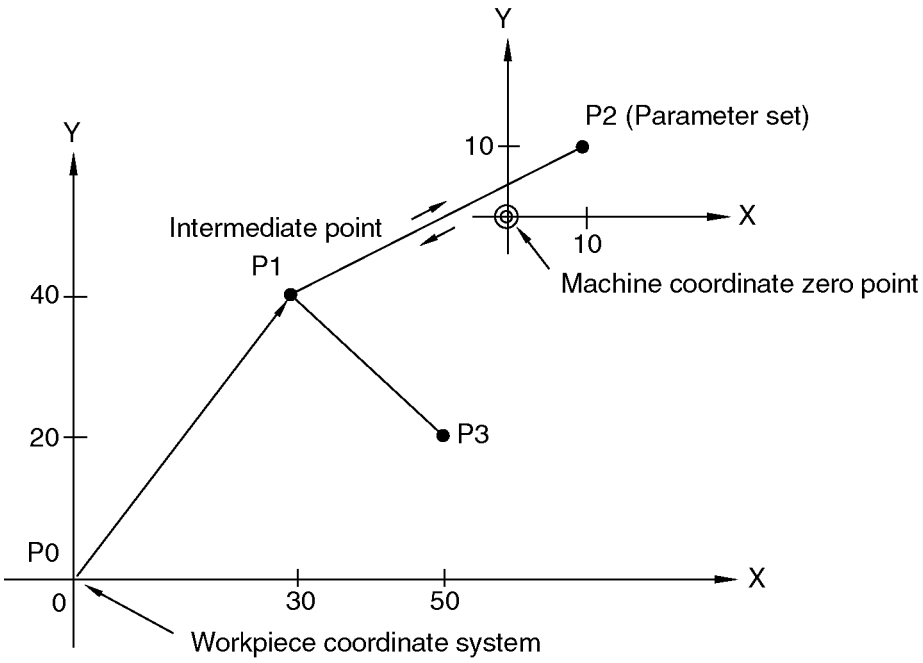


Fig. 4.32

4.4.7 EXTERNAL SKIP FUNCTION (G31)

FUNCTION: External skip function

FORMAT:

G31 | L_ | X_ Y_ Z_ (C_B_W_) F_ ;

L0 or omit	: Touch point search
L1	: Discharge point search
X/Y/Z (/C/B/W)	: Coordinate words (machine travel command)
F	: Feed rate (valid for touch point search only)

EXAMPLE OF STATEMENT

G31 X100.0 Y100.0;

EXPLANATION:

◎ By the G31 command, each axis moves to the position given with coordinate words (X/Y/Z(/C/B/W)). But if a skip signal specified with L is output (two kinds of gap signals have been defined for the L0 and L1 in the NC unit), the axis stops immediately during movement, and goes to the next block without completing current block in a program. However, if a skip signal is not output, each axis moves to the end point, then it goes to the next block in a program.

◎ The coordinate words following the G31 specify the end point coordinates of travel in the absolute mode (G90), or the travel amount in the incremental mode (G91). For the axis not to be traveled, the coordinate word may be omitted.

◎ In the case of G31L0 or G31 (touch point search), the feed rate can be specified by the F feed function in that block or previous blocks. If the F code is omitted, current modal F is selected.

The G31L1 (discharge point search) can be programmed in the modal M26 mode (ma-chining ON) only. Programming the G31L1 in the modal M27 mode (machining OFF) causes an alarm a program error.

In the case of G31L1, as the feedrate, the gap open speed determined by the setting of “07: SERVO No.” and “20: SERVO ADJ” currently selected on the Condition screen is selected (F feed function is ignored).

◎ For the G31 command, as mentioned above, if a skip signal is given during travel to the end point coordinates generated from the set values of coordinate words, the execution of G31 command block is stopped immediately, and the program proceeds to the next block.

The operation after program execution is interrupted by the skip signal varies depending on whether the next block of current program is specified in the absolute mode (G90) or incremental mode (G91).

The following describes a difference in operation between programming modes of the next block of G31 command.

The following describes a difference in operation between programming modes of the next block of G31 command.

(1) If the next block of G31 command is specified in absolute mode (G90)
The axis specified to be moved to the position given in the next block travels up to that position starting from the interrupted position by the skip signal. The axis not specified with the travel position stays at the position where the skip signal was entered.

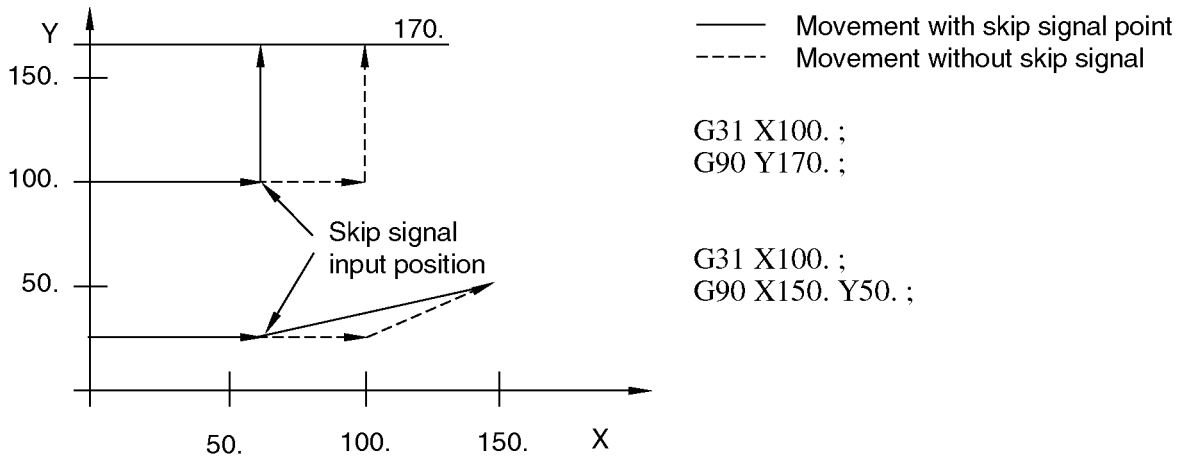


Fig. 4.33

(2) If the next block of G31 command is specified in incremental mode (G91)
The axis specified to be moved by the amount given in the next block travels by the given amount starting from the interrupted position by the skip signal. The axis not specified with the travel amount stays at the position where the skip signal was entered.

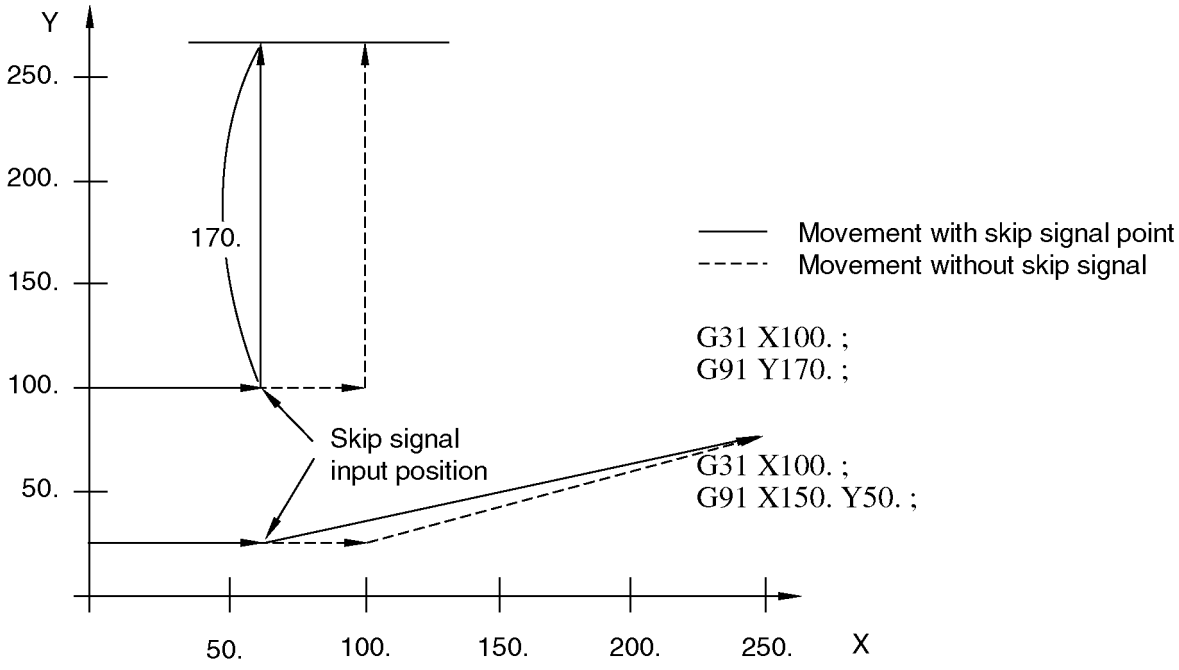


Fig. 4.34

4.4.8 DIRECT FEED RATE DESIGNATION (F)

FUNCTION: Direct feed rate designation

FORMAT: F_;

EXAMPLE OF STATEMENT:

F100;

EXPLANATION:

◎ By this command, the feed rate other than machining feed can be set.

◎ As a programming method, the F direct feed command is available.
This command specifies the feed rate directly following address “F”.
The feed rate range is 1 mm/min to 2000 mm/min.
The feed rate specified by this command can be changed in the range of 0% to 200%.
This function is called “F override”.
This F override can be set in a step of 10%.

4.5 DATA SETTING

This section describes setting or change of numeric values specified in the program.

The setting data includes;

- ○ Setting/change of electrode diameter offset
- ○ Setting/change of workpiece offset
- ○ Setting/change of electrode offset
- ○ Setting/change of machining condition elements
- ○ Setting/change of model data
- ○ Setting/change of parameters
- ○ Registration of machining conditions
- ○ Machine sequence interface

4.5.1 DIRECT DATA WRITING (G10)

FUNCTION: Programming of system variables

① Change of electrode diameter offset
② Change of workpiece position offset
③ Change of electrode offset
④ Change of machining condition elements
⑤ Change of model data
⑥ Change of parameters
⑦ Registration of machining conditions
⑧ Machine sequence interface (started with the ICON)

FORMAT:

① G10 D/H_R_; ··· [Setting]
D/H_Q_ ; ··· [Increment]
D/H : Offset number
R : Offset amount
Q : Additional amount

② G10 L2 R_P_X_Y_Z_ (C_B_W_) ;··· [Setting]
Q_X_Y_Z_ (C_B_W_) ; ··· [Increment]
L2: Changes workpiece offset value.
R: Group number
0 - 9: G500-G509
99: All groups
Omit: Modal value (current)
P: Sets offset value of workpiece offset No.
1 - 8: G54-G61
99: All workpiece offsets
Q: Additional offset amount of workpiece offset No.
1 - 8: G54-G61
99: All workpiece offsets

③ G10 L6 P_X_Y_Z_ (C_B_W_) ;
L6: Changes electrode offset.
P: Electrode offset No.1 - 200

④ G10 L10 ~ 12 R_ ;
L10: Sets directly machining condition element.
L11: Sets additional amount of machining condition element.
L12: Sets incremental reduction value of machining condition element.
R: Selects an element.
1st-3rd digits : Setting value (000-255)
4th-5th digits : Element selection (01-63)

Machining condition elements setting range table

(1) Main elements

Element No.	Setting range	Screen display
01	001 - 099	01: τON
02	001 - 099	02: τOFF
03	000 - 007	03: VH
04	000 - (030)	04: VL
05	000 - 007	05: IPH
06	000 - 062	06: IPL
	070 - 089	
07	001 - 048	07: SERVO No
08	000 - 001	08: POL
10	001 - 099	10: DF
11	000 - 012	11: τOFF ADJ

Note: For the "08": POL", 000 specifies "+" and 001 specifies "-".

(2) Sub elements

Element No.	Setting range	Screen display
20	001 - 032	20: SERVO ADJ
21	001 - 007	21: MODE 1
22	000 - 004	22: MODE 2
23	001 - 005	23: MODE 3
24	000 - 003	24: WR 1
25	000 - 001	25: WR 2
26	000 - 003	26: WR 3
27	000 - 007	27: SRCH 1
28	000 - 007	28: SRCH 2
29	001 - 004	29: SV SIG
30	000 - 007	30: ARC SIG
31	000 - 015	31: C LEVEL
32	000 - 015	32: H LEVEL
33	000 - 015	33: V LEVEL
34	000 - 014	34: R LEVEL
40	000 - 031	40: V TIME
41	000 - 003	41: S TIME
42	000 - 001	42: AC 1-1
43	000 - 001	43: AC 1-2
44	000 - 008	44: AC 1-3
45	000 - 255	45: AC 2-1
48	000 - 255	48: SM
49	001 - 008	49: JUMP_LEVEL
50	000 - 015	50: CLK_S
51	000 - 015	51: ELECTRODE
52	000 - 015	52: MATERIAL
60	000 - 020	60: (WC 1)
61	000 - 007	61: (WC 2)

⑤ G10 L50 I _ J _ K_P_ ; . . . . . [Model data process element]

L50: Model data setting

I: Model No. (0 - 99)

J: Process No. (1 - 15)

K: Process item (1 - 11)

1: Roughness (P : 0000 - 9999)

2: MC Cond. (P : 0000 - 9999)

3: Bottom Remain (P : 0000 - 9999) [µm]

4: Side Remain (P : 0000 - 9999) [µm]

5: Bottom Gap (P : 0000 - 9999) [µm]

6: Side Gap (P : 0000 - 9999) [µm]

7: MC Speed (P : –999 to 999)

8: Elec. Wear (P : 0000 - 9999)

9: Minimum reduction amount 2 mm
(P : 0000 - 9999) [µm]

10: Minimum reduction amount 10 mm
(P : 0000 - 9999) [µm]

11: Minimum reduction amount 20 mm
(P : 0000 - 9999) [µm]

Note 1: Unit of machining speed

-5 9 9 = 0.00099

Significant value (mantissa part)

Index part (10-5)

Note 2: Unit of electrode consumption (0.1%)
1000 = 100%

G10 L50 I _ J20 P_ ; . . . . . [Model data electrode material]

I: Model No. (0 - 99)

P: Material (1 - 9)

1: Cu, 2: Gr1, 3: Gr2,

4: CuW, 5: AgW, 6: Al, 7: Bs,

8: St, 9: WC

G10 L50 I_J21 P_ ; . . . . . [Model data workpiece material]

I: Model No. (0 - 99)

P: Material (1 - 9)

G10L50I_J_K_P_Q_R_ ; . . . . . [Model data comment]

I: Model No. (0 - 99)

J: Comment type

22: For metric

23: For inch

K: Number of comment lines (writing start offset)

(0 - 19)

P: 0 and 1st of K line

Q: 2nd and 3rd of K line

R: 4th and 5th of K line

Note 1: Up to 20 characters can be registered.

Note 2: If all of P, Q and R are used, K varied like

0 → 6 → 12 → 18.

⑥ G10 L999 P_R_ ; . . . . . [Parameter data]

L999: Specify this when changing parameter data.

P: Parameter No.

R: Parameter data

⑦ G10 L20 P1 I_ ; . . . . . [E No.]

L20: Registers of E No.

P1: Format

I: E No.

G10 L21 I_J_K_ ; . . . . . [Main elements]

L21 : Registers elements.

I : E No.

J	8	7	6	5	4	3	2	1

J : τON, τOFF, VH, VL

1st & 2nd digits: τON

3rd & 4th digits: τOFF

5th digit: VL

6th & 7th digits: VH

K	8	7	6	5	4	3	2	1

K : IPH, IPL, SERVO No., POL

1st & 2nd digits: IPH

3rd, 4th, 5th digits: IPL

6th digit: Servo No. 1st digit

7th digit: P OL

8th digit: Servo No. 2st digit

G10 L22 I_P_Q_R_J_ ; . . . . . [Sub elements]

L22 : Registers elements

I : E No.

P

8	7	6	5	4	3	2	1

P : C LEVEL, V LEVEL, V TIME, S TIME, WC 2

1st & 2nd digits: C LEVEL
3rd & 4th digits: V LEVEL
5th & 6th digits: V TIME
7th digit: S TIME
8th digit: WC 2

Q

8	7	6	5	4	3	2	1

Q : WC1, AC 1-1, AC 1-2, AC 1-3

1st & 2nd digits: WC 1
5th digit: AC 1-1
6th digit: AC 1-2
7th & 8th digits: AC 1-3

R

8	7	6	5	4	3	2	1

R : AC 2-1, SM

1st, 2nd & 3rd digits: AC 2-1
5th, 6th & 7th digits: SM

J

8	7	6	5	4	3	2	1

J : WR1, WR2, WR3, MODE1, MODE2, MODE3

1st digit: WR 1
2nd digit: WR 2
3rd digit: WR 3
4th digit: MODE 1
5th digit: MODE 2
6th digit: MODE 3

K

8	7	6	5	4	3	2	1

K : SRCH 1, SRCH 2, SV SIG, ARC SIG, H LEVEL, R LEVEL

1st digit: SRCH 1
2nd digit: SRCH 2
3rd digit: SV SIG
4th digit: ARC SIG
5th & 6th digits: H LEVEL
7th & 8th digits: R LEVE

G10 L23 I_J_K_ ; . . . . . [Sub elements]

L23 : Registers elements

I : E No.

J	8	7	6	5	4	3	2	1

J : JUMP_LEVEL, ELECTRODE, W. MATERIAL

1st & 2nd digits: JUMP_LEVEL

3rd & 4th digits: ELECTRODE

5th & 6th digits: W. MATERIAL

K	8	7	6	5	4	3	2	1

K : CLK_S, SV ADJ

1st & 2nd digits: CLK_S

3rd & 4th digits: SV ADJ

⑧ G10L60P_Q_R_I_J_K_;‥‥ [started with the ICON]

P: Start O number

Q: Start N number

R: Code call parameter 1

I: Code call parameter 2

J: Code call parameter 3

K: Code call parameter 4

EXAMPLE OF STATEMENT:

G10 D01 R10. ;
G10 H12 Q-3. ;
G10 L2 P1 X10. Y20. Z30. ;
G10 L999 P552 R5 ;

EXPLANATION:

◎ This command can change system variables in the program.
Changeable variables are as follows:

① Change of electrode diameter offset
The offset value used for electrode diameter compensation can be changed to the value specified by R, or it can be added by value specified by Q.

② Change of workpiece position offset
The workpiece zero point offset in the workpiece coordinate system specified by P can be changed to the value preceded by the axis address.

③ Change of electrode offset
The offset value associated with electrode offset number specified by P can be changed to the value preceded by the axis address.

④ Change of machining condition element
The machining condition element specified by 4th and 5th digits of R can be changed to the value specified by 1st, 2nd and 3rd digits.

⑤ Change of model data
The model data number specified by I can be changed or registered to the value or data specified by P (Q, R) of each item (Process, Electrode, Workpiece, Comment) specified by J.

⑥ Change of parameter data
The data of parameter specified by P can be changed to the data specified by R.

⑦ Registry of machining condition
The machining condition items (E No., main, sub and auxiliary elements) specified by L of the condition number specified by I can be changed to the values specified by P, Q, R, J, and K.

⑧ Started with the ICON

Note: When registering variables through the above item②to④, then using them, use blocks after the 3rd block.
Also, for the workpiece offset, new variables will become active by specifying the work coordinate system again.

SAMPLE:

◎ The following shows an example in which the workpiece offset is changed.

Program

G10 L2 P2 X10. Y20. ; [G55 workpiece offset: X=10.Y=20.]

M03 ;

G01 F200 ;

G00 G55 X0 Y0 ;

Old workpiece offset is used by buffer processing.

Specify again original work coordinate system
even if it is already specified by G55.

4.6 ELECTRODE COMPENSATION

This section details the electrode compensation function and the function words related to it.

The main items described in this section are as follows:

(1) Offset number

(2) Offset value change by program

(3) Electrode diameter compensation

(4) Electrode length compensation

Table 4.4 Electrode Compensation Command Table

Word	Modal	Function	Item
D/H	○	Offset number specification	4.6.1
G10 (G28)	×	Offset value change	4.6.2
G40 G41 G42	○ (E)	Electrode diameter compensation cancel Electrode diameter compensation, left side Electrode diameter compensation, right side	4.6.3
G43 G44 G49	○ (F)	Electrode length compensation, + direction Electrode length compensation, – direction Electrode length compensation cancel	4.6.4

4.6.1 SELECTION OF ELECTRODE COMPENSATION VALUE (D/H)

FUNCTION: Electrode compensation value selection

FORMAT: D_H_; D/H: 1-99

EXAMPLE OF STATEMENT:

D01 H05;

……D02;

……H07;

EXPLANATION:

◎ This command can specify one of the electrode compensation values stored in the NC unit.

◎ The command specifies the offset number (0-99) where the compensation value was stored, following address D or H.

Specify the offset value used for the electrode diameter compensation with “D”, and the offset value used for the electrode length compensation with “H”.

One offset number can be specified for each of D and H. A same-number address may be specified for D and H.

This is a Modal command. The previous command remains valid until another number is specified by the same address.

Upon power-on, address 00 is specified for both D and H.

◎ The NC unit executes the compensation function from the block where the compensation number was specified by this command, according to the compensation value.

Note: The offset number is common to address “D” and address “H”.

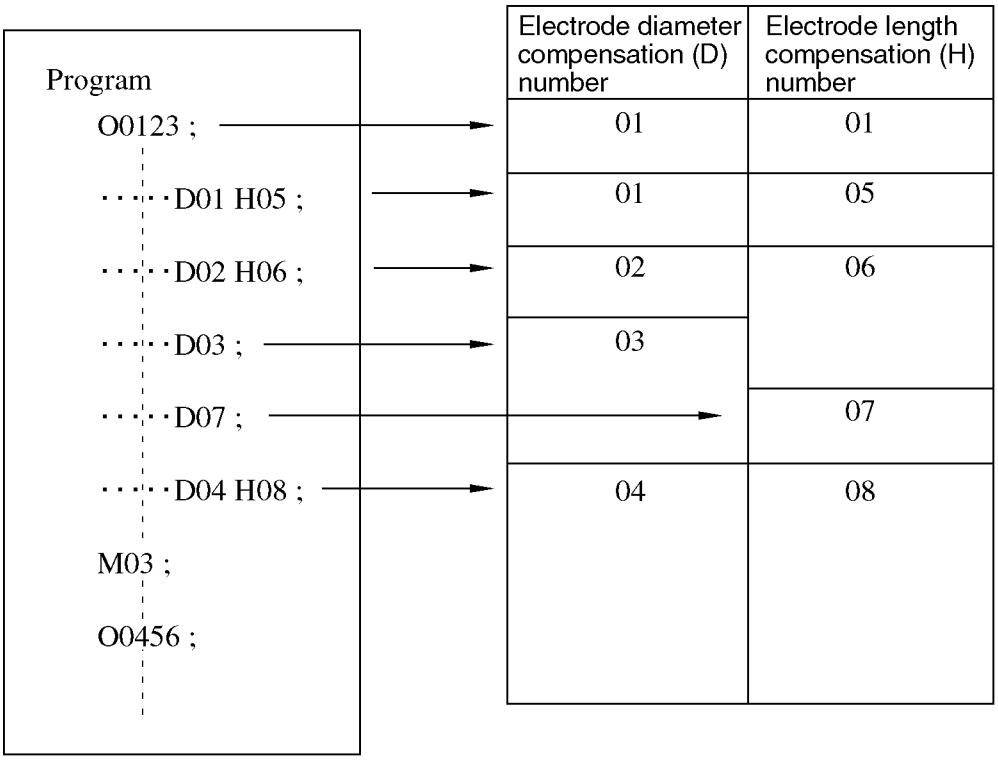
Available numbers are 0-99.

If any numeric number other than these is used for specification, an alarm is caused.

The compensation value of offset number “0” is always 0.

SAMPLE:

When a program containing the following commands is executed, the offset value to be used for electrode compensation changes as shown in the table below.


Program
O0123 ;
D01 H05 ;
D02 H06 ;
D03 ;
D07 ;
D04 H08 ;
M03 ;
O0456 ;
Electrode diameter compensation (D) number
Electrode length compensation (H) number
01
01
01
05
02
06
03
07
04
08

4.6.2 CHANGE OF ELECTRODE COMPENSATION VALUE (G10)

FUNCTION: Electrode compensation value change

FORMAT:

G10 | D/H_R_ / D/H_Q_ | ;

EXAMPLE OF STATEMENT:

G10 D01 R20. ;
G10 H01 R20000;
G10 D10 Q-5. ;

EXPLANATION:

◎ This command changes the compensation value to be used for electrode diameter compensation, electrode length compensation and electrode position offset by program.

(1) The electrode compensation value stored in the NC unit, which is specified by the numeric value following address D and H, is changed to the compensation value specified by R.

(2) The value specified by Q is added to the electrode compensation value specified with value following the address D and H, which has been registered to the NC unit.

SAMPLE:

The compensation value can be changed as follows according to the program.

Before program execution	
Compensation No.	Compensation value
1	10
2	20
3	30

Program
G10 D01 R100. ;
G10 H02 R200. ;
G10 H03 Q-20. ;

After program execution	
Compensation No.	Compensation value
1	100
2	200
3	10

4.6.3 ELECTRODE DIAMETER COMPENSATION (G40/41/42)

FUNCTION: Electrode diameter compensation
G40 – Electrode diameter compensation cancel
G41 – Electrode diameter compensation (left side)
G42 – Electrode diameter compensation (right side)

FORMAT:

G40 ;

G41/42	G17X__Y__ G18Z__X__ G19Y__Z__	D__;

EXAMPLE OF STATEMENT:

G40;
G41 G17 D01; I2 J5;
G18 G42 G01 X100. Z10. D02;
G41 X100. Y200. ;

EXPLANATION:

◎ Command G41/G42 sets the electrode diameter compensation mode. After this command, electrode diameter compensation is executed until the mode is changed to the electrode compensation cancel mode by command G40.
By command G41/G42, “positioning” and “linear interpolation” are performed from the block containing this command, and electrode diameter compensation is performed for the “circular interpolation” command following the next block.
Command G41 generates an offset path on the left side facing the forward direction of the electrode, while command G42 generates it on the right side.

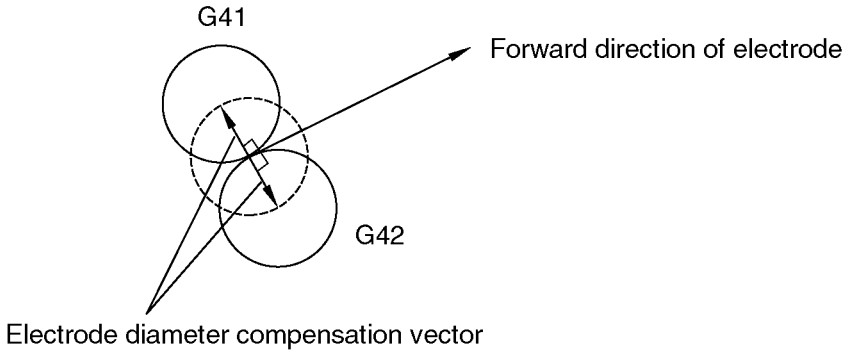
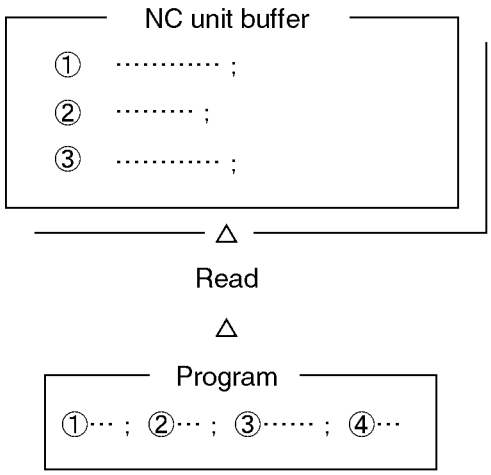


Fig. 4.35

◎ Electrode diameter compensation is enabled only within the plane specified by the plane specification command when the electrode diameter compensation mode is set. If in the electrode diameter compensation mode, a plane other than the compensation plane is specified or non-movement within the compensation plane is specified in two blocks or more running, electrode diameter compensation is temporarily stopped.



If an intersection point cannot be obtained by block ② or ③ for the movement of block ①, the message of "no intersection point" is output.

Fig. 4.36

The electrode diameter compensation mode must be specified in the positioning (G00) or linear interpolation (G01) mode. If specification is made in the circular interpolation (G02/03) mode, the command point and end point are not on the same arc and this leads to a circular end point error.

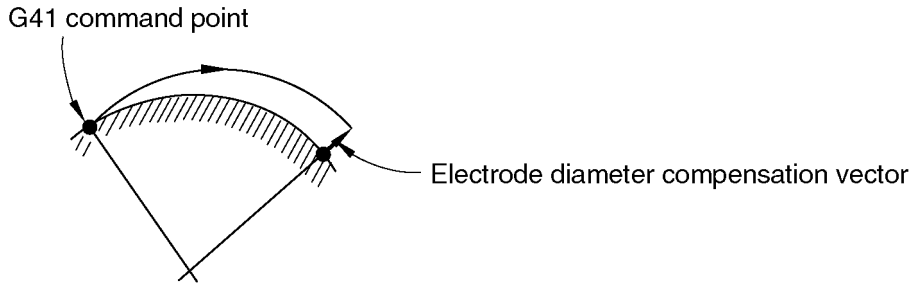


Fig. 4.37

Upon power on or reset, the electrode diameter compensation cancel mode is selected.

◎ Upon receipt of the electrode diameter compensation command in the electrode diameter compensation cancel mode, the NC unit generates an electrode diameter compensation vector for the first time.
After that, the electrode center moves on the path offset by this compensation vector from the programmed path.
The moving operation of the electrode by a travel command immediately following command G41/G42 is called “initial electrode diameter compensation operation” when the electrode is caused to approach workpiece, the electrode is offset by its radius from the electrode path programmed by part outside shape.

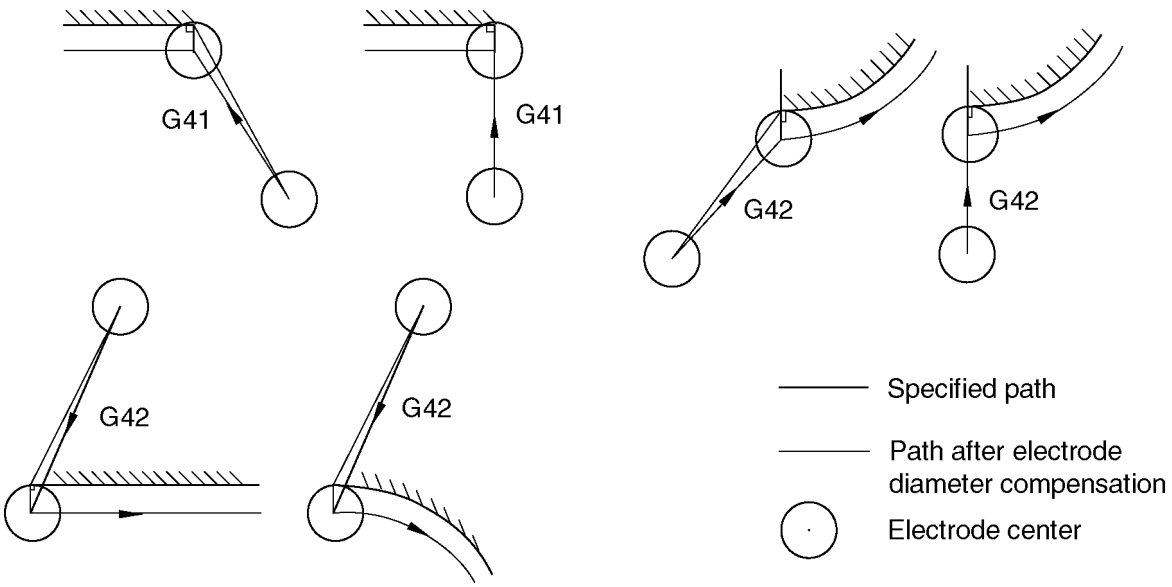


Fig. 4.38

In principle, the electrode center path in the electrode diameter compensation mode can be obtained by the following procedure.

For a command without no directional designation of I, J and K, obtain a pattern offset by the current block and a pattern offset by the next block, and make a movement to their intersection point.

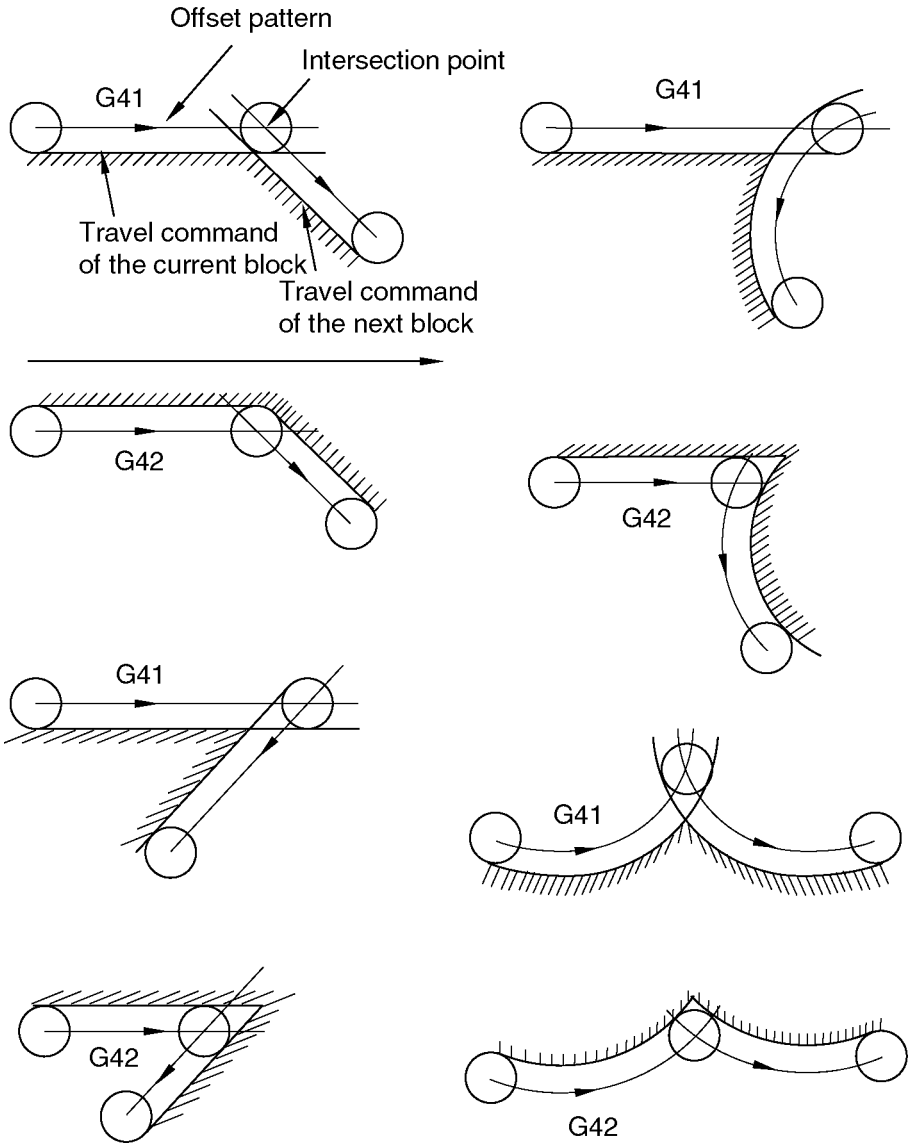

Offset pattern
Intersection point
G41
Travel command
of the current block
Travel command
of the next block
G41
G42
G42
G41
G41
G42
G42

Fig. 4.39

Interference check

The NC unit reads block 3 of the program and computes the offset pattern for the travel command. In this case, if the moving direction of the program differs from that of the offset path, it is presumed that interference has occurred and a message is output.

The NC unit gives different determination depending on operating states of the machine.

(1) For dry run

If interference has occurred upon execution of a dry run, a massage is output and the NC unit is stopped.

In the dry run mode, the NC unit considers that the program is being checked. Therefore, after making an interference check, it is possible to continue the following checks pressing the start button.

(2) Ordinary machining

If interference occurs when the dry run mode is off, it is considered that the worker checks the interference, and machining is continued.

An interference message is output.

Machining the inside of a smaller arc than the electrode diameter compensation value

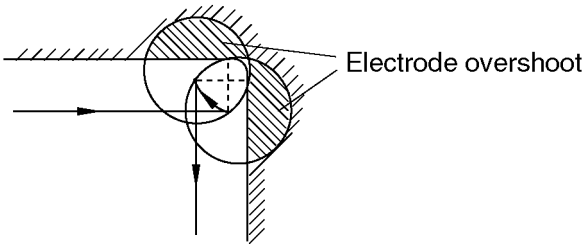


Fig. 4.40a

Where the moving amount is small

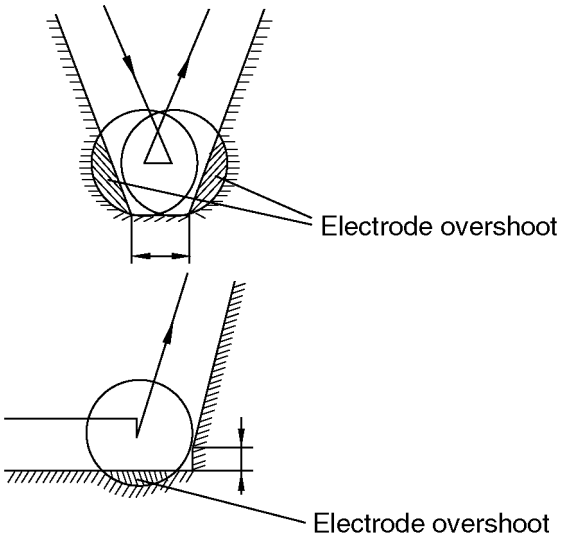


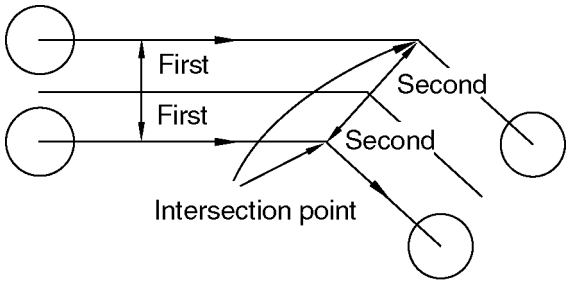
Fig. 4.40b

Compensation value change

In the electrode diameter compensation mode, it is possible to change the compensation value without canceling compensation.

Between straight line and straight line

Where there are intersection points (the intersection points are adopted.)



Where there are no intersection points (the 1st electrode diameter compensation vector is adopted.)

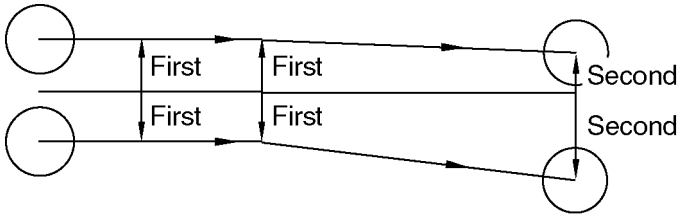
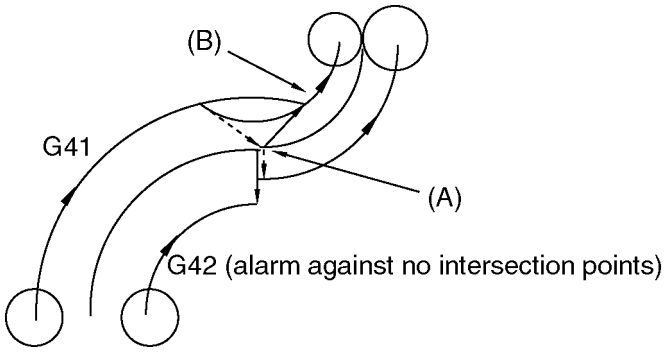


Fig. 4.41a

Between arc and arc (intersection point B after the travel command prior to electrode diameter compensation passes intersection point A is adopted.)



Between straight line and arc (intersection point B after the travel command prior to electrode diameter compensation passes intersection point A is adopted.)

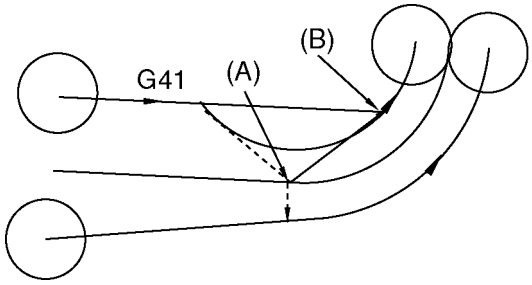


Fig. 4.41b

Cancel

To cancel the electrode diameter compensation mode, specify command G40.
When command G40 is specified, electrode diameter compensation is not performed in and after that block until command G41/42 is specified.
When this command G40 is executed, the electrode moves so that the electrode diameter compensation vector stored in the NC unit may reach "0".
Be sure to terminate the program in the electrode diameter compensation cancel mode.
If the program end (M02/30) command is given in the electrode diameter compensation mode, an alarm is caused. When command G40 is executed in the same block as the axis command out of the compensation plane, an alarm is also caused.

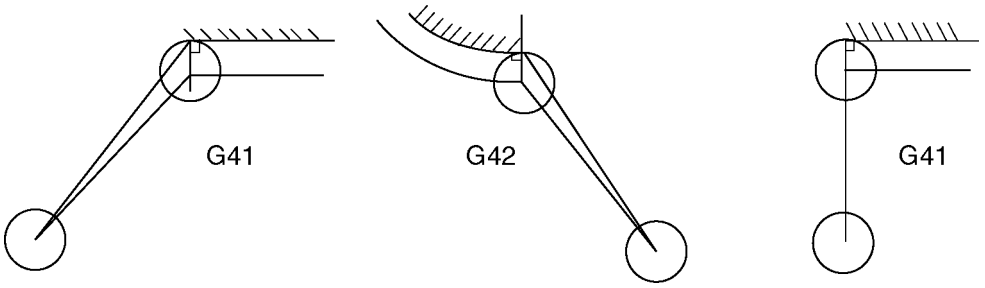


Fig. 4.42

Note:

(1) In the electrode diameter compensation mode, two or more blocks without any moving amount on the electrode diameter compensation plane are specified electrode diameter compensation for a movement within the next electrode diameter compensation plane differs from ordinary compensation.
In the G41 mode, compensation is performed by circular interpolation CW whose radius is the compensation value.
In the G42 mode, compensation is performed by circular interpolation CCW.

```
G17 ;
Y100. 0 ; ····· ①
Z5. 0 ; ····· ②
Z15. 0 ; ····· ③
X100. 0 ; ····· ④
```

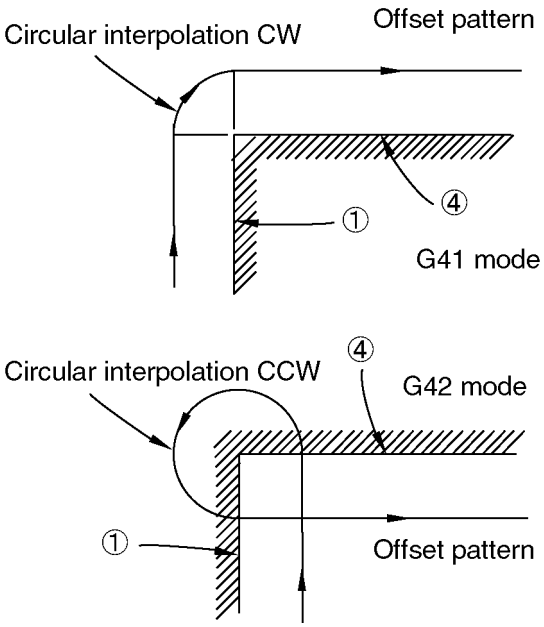


Fig. 4.43

(2) When coordinate system setting (G92), dwell (G04) or automatic reference point return (G28) is specified in the electrode diameter compensation mode, electrode diameter compensation is temporarily stopped but later returned automatically.

(3) When there are not intersection points in a pattern offset from the program path upon out-side compensation, that part is interpolated with 3 straight lines.
If there are no intersection points upon inside compensation, an alarm is caused.
Alarm due to no intersection points

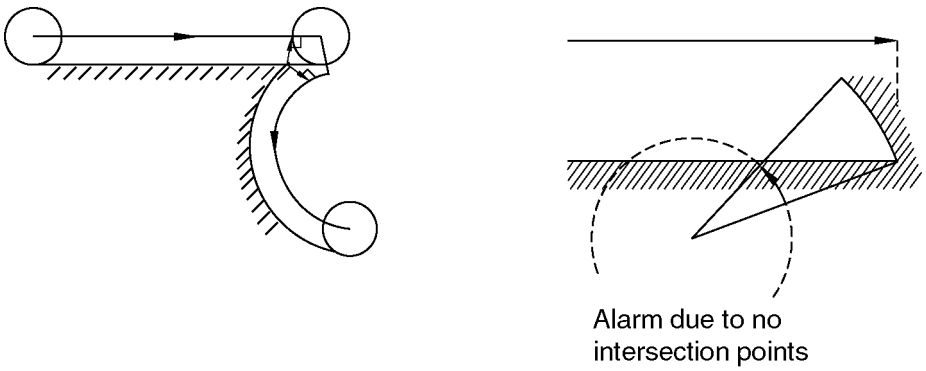


Fig. 4.44

In a pattern offset between straight lines, when the distance between their intersection point and the original pattern (intersection vector) exceeds the tolerance specified in a parameter, interpolation is performed with 3 straight lines.
If such a corner is operated by a single block, the operation is temporarily stopped at the electrode position shown in the figure below.

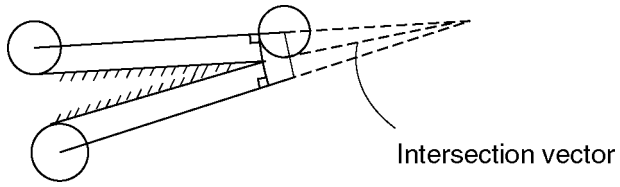
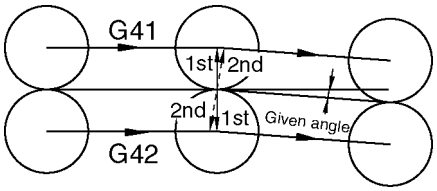


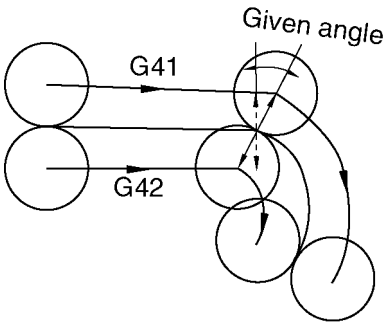
Fig. 4.45

(4) When the angular difference in compensation vector between 2 blocks is within the given angle specified in a parameter, an intersection vector is selected as shown below.

Between straight line and straight line (the 1st vector is an intersection point.)



Between straight line and arc (The arc-side vector is an intersection point.)



Between arc and arc (The 1st vector is an intersection point.)

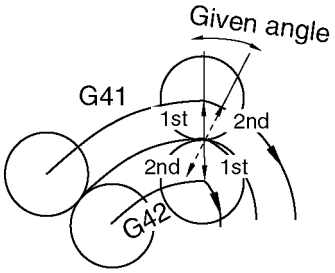


Fig. 4.46

SAMPLE 1:

Machining can be performed by electrode compensation value change (G10) and electrode diameter compensation as shown below. This machining is called “automatic electrode diameter allowance”.

Program)

Main program

```
O1 (MAIN PRO);
G91 G17 G10 D01 R5. ;        [Electrode dia. compensation value writing D01=5mm]
G01 Z-40. F100;
G41 X35. D01;                [Electrode dia. compensation mode, left side]
G02 I10. ;
G01 G40 X-35. ;              [Electrode dia. compensation mode cancel]
M98 P101 L10;                [Subprogram call]
G00 Z140. ;
M02;
```

Subprogram)

```
O101 (SUB PRO) ;
G10 D01 Q3. ;                [Compensation value +3mm]
Z-10. ;
G41 X35. D01;                [Electrode dia. compensation mode, left side]
G02 I10. ;
G01 G40 X-35. ;              [Electrode dia. compensation mode cancel]
M99;                         [Return to main program]
```

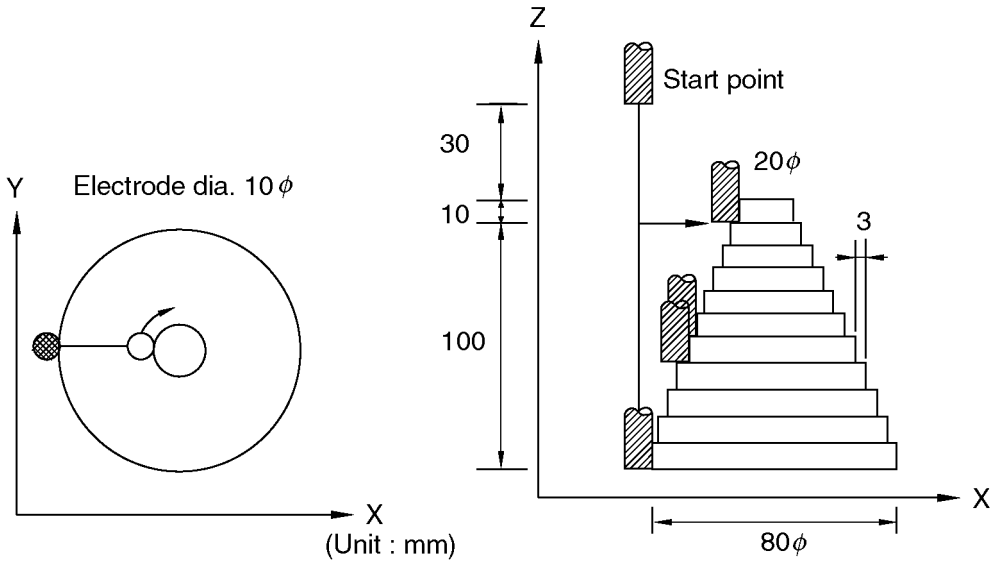


Fig. 4.47

SAMPLE 2:

The following is a program for a pattern without intersection points in the outside compensation.

Program)

G90 G92 X0 Y0 ;	(→ P0)
G01 G41 X20.0 Y20.0 D01 F100 ;	(P0 → P1)
X70.0 Y150.0 ;	(P1 → P2)
X250.0 Y150.0 ;	(P2 → P3)
G03 X200.0 Y40.0 I10.0 J-70.0 ;	(P3 → P4)
G01 X200.0 Y20.0 ;	(P4 → P5)
X20.0 Y20.0 ;	(P5 → P1')
G40 X0. Y0. ;	(P1' → P0)
M30 ;	

Path ① …… Compensation value 20 mm

Path ② …… Compensation value 30 mm

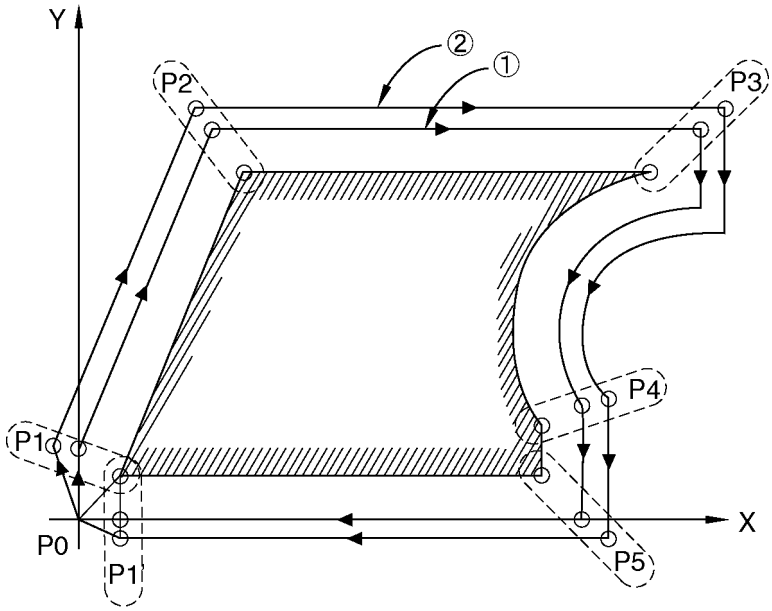


Fig. 4.48

4.6.4 ELECTRODE LENGTH COMPENSATION (G43/44/49)

FUNCTION: Electrode length compensation
G43 – Offset in the + direction
G44 – Offset in the - direction
G49 – Cancel

FORMAT: G43/44 | Z_ | H_;
G49 ;

EXAMPLE OF STATEMENT:
G43 Z10.0 H01;
G44 Z100. H22;
G49;

EXPLANATION:

◎ The end-point position of the Z-axis can be offset by the specified offset value by the + side (G43) or – side (G44) by command G43/G44 until command G49 is executed.

◎ The moving amount prior to compensation is specified by the numeric value following Z, and the offset number is specified by the address H. When the Z-axis command is omitted, movement is made by the offset value.

◎ The NC unit performs compensation by the offset value specified upon execution of command G43/G44. This offset value is stored in the NC unit until the compensation cancel (G49) command or offset value change is executed.
In spite of a change in offset number or offset value program, the offset value to be used for compensation will not be changed unless it is specified again by command G43/44.
To change the offset value, be sure to execute command G43/44.

Note: Command G43/44 operates only for the Z-axis. Even when another axis command is specified in the same block, it does not affect these axes.

SAMPLE:

The following is an example of programming by using this electrode length compensation function when the electrode change function is used.

	Program
①	G90 G92 Z0 ;
②	G43 Z-30. H01 ;
③	G91 G30 P2 Z0 ;
	Electrode change (Electrode 1→ Electrode 2)
④	G90 G43 Z-30. H02 ;
⑤	G91 G30 P2 Z0 ;
	Electrode change (Electrode 2→ Electrode 3)
⑥	G90 G43 Z-30. H03 ;
⑦	G49 Z0 ;

Block	Moving amount	Coordinate value
①	0	0
②	– 40	– 40
③	70	30
④	– 55	– 25
⑤	55	30
⑥	– 65	– 35
⑦	35	0

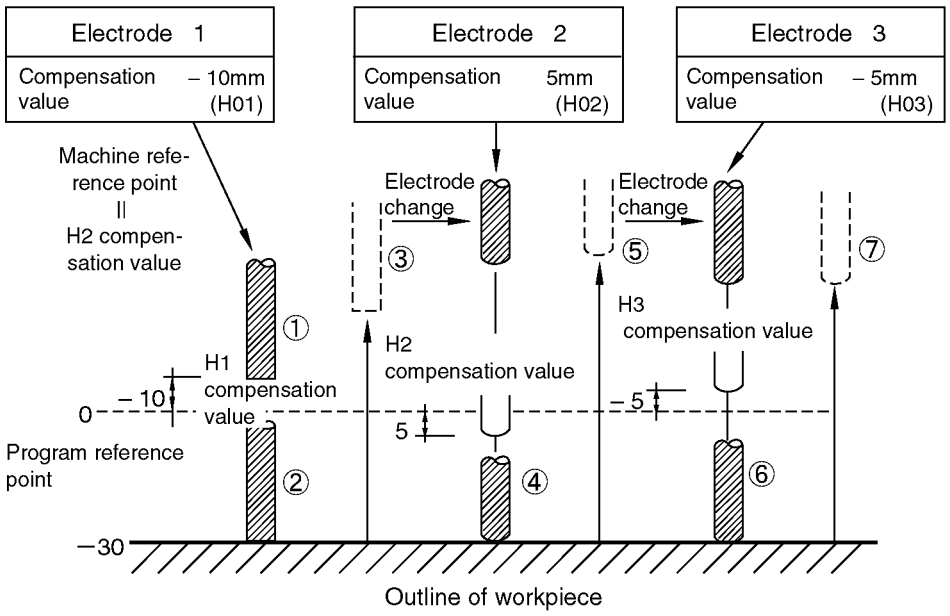


Fig. 4.49

4.7 SCALING (G47/48)

FUNCTION: Scaling (enlargement/reduction)
G47 – Scaling (enlargement/reduction) cancel mode
G48 – Scaling (enlargement/reduction) mode

FORMAT: G47 ;
G48 | X_Y_Z_ (C_B_W_) | P_ ;
P: Magnification factor (Unit: × 1/1000 [standard resolution] or × 1/10000 [high resolution])

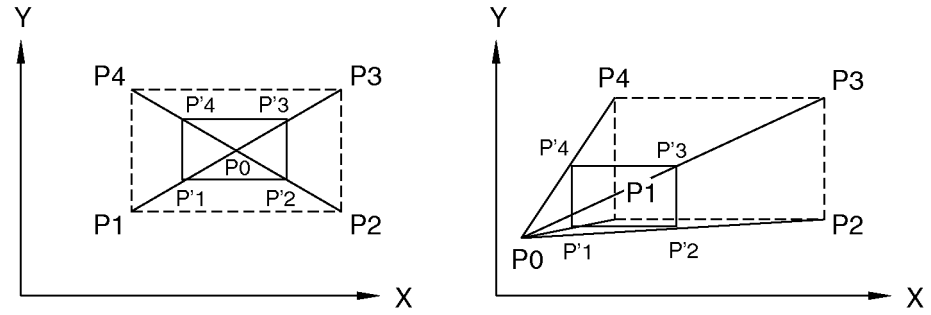
EXAMPLE OF SATEMENT:
G47;
G48 X10.0 Y20.0 Z10.0 P0.5;

EXPLANATION:

◎ Command G48 performs enlargement or reduction around the point specified X, Y, Z (C, B, W) at the magnification factor specified by P.
Command G47 cancels the enlargement/reduction mode.

◎ Settable magnification factor is in a range of 0.001 - 99.999 times (P1 - P99999) for standard resolution, or 0.0001 - 99.9999 times (P1 - P999999) for high resolution.
Specify the G48 command in a single block in the electrode radius compensation cancel mode.
Command 47 can be specified even in the electrode diameter compensation mode.

◎ After command G48, enlargement or reduction is performed on the axis for which the central point has been specified, until command G47 is executed.



P0: Central coordinates for enlargement/reduction

Fig. 4.50

Note 1: A position display is made by the coordinate value posterior to enlargement/ reduction.

Note 2: In the G48 mode, the axis to be enlarged or reduction can be set by the parameter.
In this case, even on an axis that has not been specified by command G48, enlargement/reduction is performed with the position where command G48 was executed as its central point if the axis is specified as valid by the parameter.
(Scaling axis designation: Parameter No. 0517)

Note 3: Radius command R for circular interpolation has no connection with axis designation and parameter setting in command G48.
Whenever the G48 mode is set, the scaling function is enabled.

Note 4: To execute command G48 in the MDI operation mode, change parameter set-ting.
(Enlargement/reduction rotation in MDI enable/disable: Parameter No. 0087)

Note 5: Enlargement/reduction cannot be applied to such offset values as electrode diameter compensation, electrode length compensation.

Note 6: To perform the canned cycle specifying the Z-axis as the enlargement/ reduction mode, set the absolute (G90) mode.

Note 7: When command G68 (rotation) is used together, enlargement/reduction is disabled for rotation angle R.

Note 8: Do not specify G28/29/30 (reference point return) and G92 (workpiece coordinate system setting change) in the G48 mode.

Note 9: No enlargement/reduction is executed if the center coordinate of G48 enlargement/reduction is not specified.

Note 10: Do not use this command together with a machining canned cycle, a positioning canned cycles, or the model plan.

SAMPLE:

The following shows programs that use enlargement/reduction and coordinate rotation. The program I specifies rotation in the enlargement/reduction mode, while the program II specifies enlargement/reduction in the rotation mode.

Program I)

```
G92 X0 Y0 ;                 [Workpiece coordinate system setting]
G48 X55.0 Y40.0 P0.5 ;      [XY plane 1/2 contraction]
G68 X20.0 Y20.0 R-45.0 ;    [Coordinate rotation 45°CW]
G01 G91 X20.0 Y20.0 F1 ;    (P0 → P1)
X70.0 ;                     (P1 → P2)
G03 Y40.0 I-35.0 J20.0 ;    (P2 → P3)
G01 X-70.0 ;                (P3 → P4)
Y-40.0 ;                    (P4 → P1)
G69 ;
G90 G47 X0 Y0 ;             (P1 → P0)
M30 ;
```

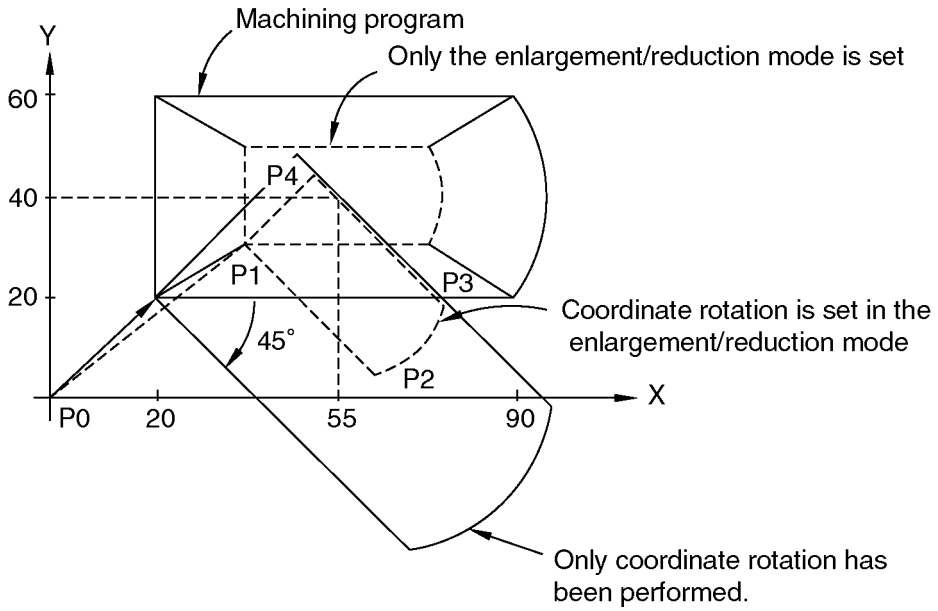


Fig. 4.51

Program II)

```
G92 X0 Y0;                  [Workpiece coordinate system setting]
G68 X20.0 Y20.0 R-45.0;     [Coordinate rotation 45°CW]
G48 X55.0 Y40.0 P0.5;       [XY plane 1/2 reduction]
G01 G91 X20.0 Y20.0 F1;     (P0 → P1)
X70.0;                      (P1 → P2)
G03 Y40.0 I-35.0 J20.0;     (P2 → P3)
G01 X-70.0;                 (P3 → P4)
Y-40.0;                     (P4 → P1)
G47;
G90 G69 X0 Y0;              (P1 → P0)
M30;
```

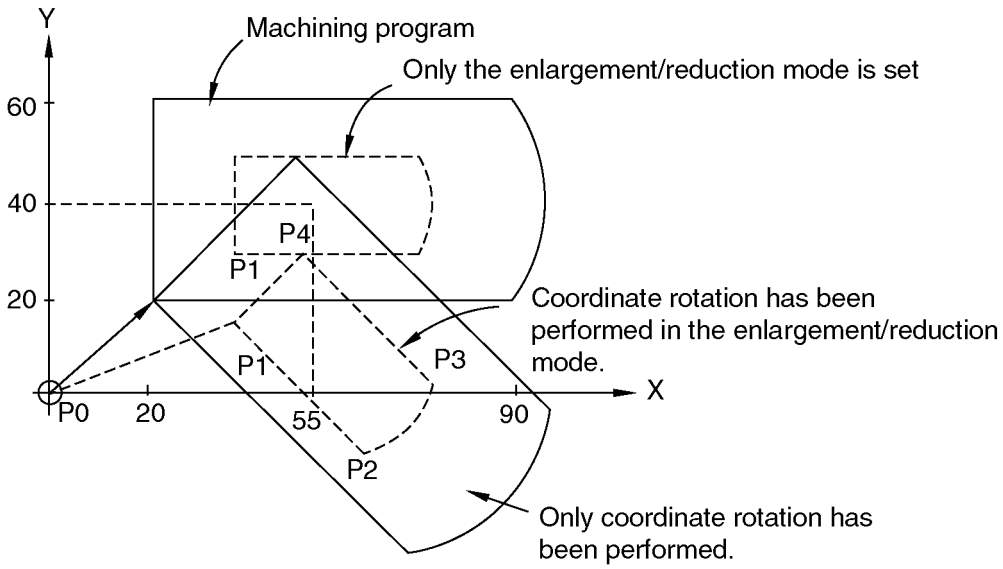


Fig. 4.52

4.8 ROTATION (G68/69)

FUNCTION: Rotation
G68 – Rotation
G69 – Rotation cancel

FORMAT:

G68	X__Y__ Y__Z__ Z__X__	R__;	R: Rotation angle

G69

EXAMPLE OF STATEMENT:

```
G68 X100. Y50. R90. ;
G69;
```

EXPLANATION:

◎ The G68 command rotates the coordinate system about the center coordinates specified by X, Y and Z by the angle specified by R. On the contrary, the G69 command cancel the coordinate rotation.

◎ Specify the rotation angle in the unit of 0.001 degree for standard resolution, or 0.0001 degree for high resolution.
The rotation plane is determined by axis specified along with G68 and the plane already selected at that time (G17, G18, G19).
If they are not same, the axis specified along with G68 is selected.

Note 1: The CCW direction is positive direction and the rotation angle range to be set is 0 - 360°.

Note 2: The G69 command may be specified together with another command in the same block.

Note 3: If rotation center coordinates for G68 are not set, the position where G68 command is present is used as rotation center.

Note 4: Do not use this command together with a machining canned cycle, a positioning canned cycles, or the model plan.

SAMPLE:

The following shows a program in which the tool path is rotated by the rotation command.

Program Coordinate rotation mode ... Electrode path ①

Cancel mode..... Electrode path ②

G92 X0 Y0;	[Coordinate rotation]
G68 X20.0 Y20.0 R-45.0 ;	[Electrode dia. compensation D01 = 5 mm]
G42 G01 G91 X20.0 Y20.0 M26 D01;	(P0 → P1)
X70.0;	(P1 → P2)
Y40.0;	(P2 → P3)
X-70.0;	(P3 → P4)
Y-40.0;	(P4 → P1')
G69 G40 X-20.0 Y-20.0 ;	(P1' → P0) [Electrode dia. comp. & coord.
M30;	rotation cancel]

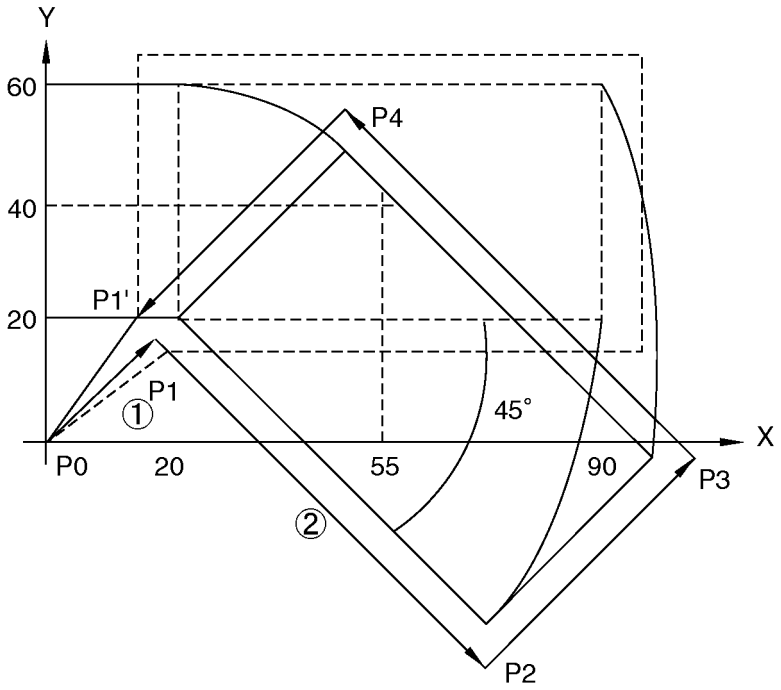


Fig. 4.53

4.9 MIRROR IMAGE (G70/71)

FUNCTION: Mirror image change
G70 – Mirror image cancel
G71 – Mirror image set/reset

FORMAT: G70;
G71 | X_Y_Z_ (C_B_W_) | ;

EXAMPLE OF STATEMENT:
G70;
G71 X1 Y1;

EXPLANATION:

For axially symmetric machining or point symmetric machining, the mirror image function re-verses the moving direction to perform this with one program.
This section describes mirror image commands.
The commands described in this section are shown below.

Table 4.5 Mirror image commands

Word	Modal	Function	Item
G70 G71	○ (M)	Mirror image cancel Mirror image set/reset	3.9

◎ The commands can reverse the mirror image state.
◎ The mirror image state of every axis is cancelled by command G70.
When the numeric value following the axis address is “1”, the mirror image function for that axis is turned on by command G71.
When the said value is “0”, the mirror image function for that axis is turned off by the command.

Note 1: Upon power-on, every axis is in the mirror image cancel mode.
Whether the mirror image cancel mode is to be set or not upon RESET is preset in a parameter.
(Initialization/no initialization of the G Modal command upon RESET: Parameter No. 64)

Note 2: Do not use this command together with a machining canned cycle, a positioning canned cycles, or the model plan.

SAMPLE:

メインプログラム）		サブプログラム）	
G17 G70 G91;		O101;	
M98 P101;	.....①	G01 X20.0 Y10.0 F1;	(P0 → P1)
G71 X1;		X20.0;	(P1 → P2)
M98 P101;	.....②	G03 X-20.0 Y20.0 I-20.0;	(P2 → P3)
G71 Y1;		G01 X-20.0 Y-30.0;	(P3 → P0)
M98 P101;	.....③	M99;	
G71 X0;			
M98 P101;	.....④		
G70;			
M30;			

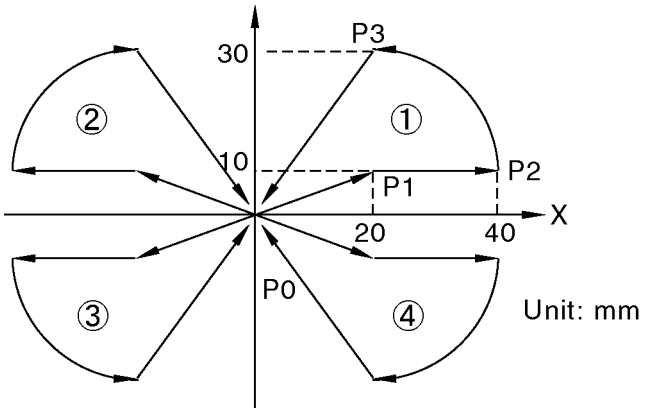


Fig. 4.54

4.10 CANNED CYCLE

Canned cycles are functional operations that are modeled and then stored into the memory. When a canned cycle is specified, the predetermined multiple operation cycle is executed.

Table 4.6 Canned cycle commands

Word	Modal	Function	Item
G74	○ (S)	Pattern canned cycle ON	4.10.1
G75		Pattern canned cycle cancel	4.10.1
G76	○ (N)	Jump canned cycle ON	4.10.2
G77		Jump canned cycle cancel	4.10.2
G80	○ (O)	Machining cycle cancel	4.10.3
G81		Automatic depth-set machining cycle	4.10.3
G82		Predetermined depth-set machining cycle	4.10.3
G83		Ordinary machining cycle	4.10.3
G84	×	Edge positioning	4.10.4
G85	×	Inside positioning	4.10.4
G86	×	Outside positioning	4.10.4

“Pattern” and “jump” are valid only in the moving or canned machining cycle of G01/02/03 in the machining ON (M26) command.
With “jump” and “measurement”, feed-per-minute is performed but machining feed is not performed even in the machining ON mode. (The same operation as machining OFF)
With the canned machining cycle, machining feed if performed but feed-per-minute is not performed even in the machining OFF mode. (The same operation as machining ON)

4.10.1 PATTERN MACHINING (G74/75)

The pattern machining function moves the machining axis and other 2 axes (3 axes) excluding the machining axis, or 2 axes (3 axes) including the machining axis on the plane and path defined for pattern machining, and is turned on/off by canned cycle command G74/75 for use. Once the pattern machining operation is specified, the command becomes a Modal and is enabled in all machining blocks.

FUNCTION: Pattern

G74 - Pattern canned cycle ON

G75 - Pattern canned cycle cancel

FORMAT:

G74 | L_R_P_F_A_ | ;

G75 ;

L : Type of pattern

0 : Basic pattern

1 : Equal radial pattern

2 : Regular polygon pattern

3 : F pattern (optional pattern)

R : Pattern radius

Standard pattern (0.002 - 8.000 [mm])

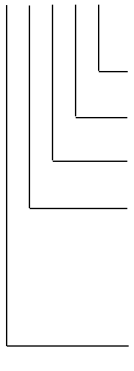
Other than standard pattern (0.002 - 4.000 [mm])

However, regular triangle is (0.002 - 2.309 [mm])

P : Pattern detail (variable depending on L)

1) In the case of L0 (basic pattern)

P ∗ $ $ $ $: ∗ : Pattern mode, $ $ $ $: Pattern machining



4th quadrant

3rd quadrant

2nd quadrant

1st quadrant

0 :Invalid, 1 :1/4 circle, 2 :Rectangle

3 :Triangle, 4 :Oblique line, 5 :Straight line

Orbiting mode

0 :Normal pattern(Orbiting at the constant cycle.)

2 :Expanded pattern.(Expanded orbiting without feed toward servo axis.)

4 :3-axes(Expands orbiting radius depending on MC depth.)

7 :Center return(Decreases the feed and returns orbiting center during unstable.)

2) P : Number of radiations (3 - 10) [at L1 (equal radial pattern)]

3) P : Number of regular polygon (3 - 18) [at L2 (regular polygon pattern)]

4) P : Program number (1 - 9999) [at L3 (optional pattern)]

F : Pattern peripheral speed
(1 - 500 [mm/min]) [Standard resolution]
(10 - 5000 [0.1mm/min]) [High resolution]
※ Standard pattern speed =32 mm/min

A : Pattern rotation angle

EXAMPLE OF STATEMENT:

(1) G74 R100 P01111 F32;
(2) G74 R125;
(3) G74;
(4) G75;

EXPLANATION:

◎ The pattern machining function operates the machining axis and other 2 axes except the machining axis, or 2 axes (3 axes) including the machining axis by the plane and path defined for pattern machining, and is turned on/off by canned cycle command G74/G75. Once specified, the pattern machining operation becomes Modal, being enabled in all machining blocks.

Pattern machining is classified by path definition as follows:

1) Basic pattern machining	(G74 L0 R_ P_ ;)
2) Equal radial pattern machining	(G74 L1 R_ P3-10 ;)
	P: Number of radiations
3) Regular polygonal pattern machining	(G74 L2 R_ P3-18 ;)
	P: Number of regular polygons
4) Optional pattern machining	(G74 L3 R_ P1-9999 ;)
	P: Program No.
5) Pattern machining cancel	(G75 ;)

During the pattern machining canned cycle, scaling, rotation and mirror image are disabled.

In basic pattern machining, the path is defined on a plane by quadrants. The pattern machining function is turned on/off by canned cycle command G74/G75.

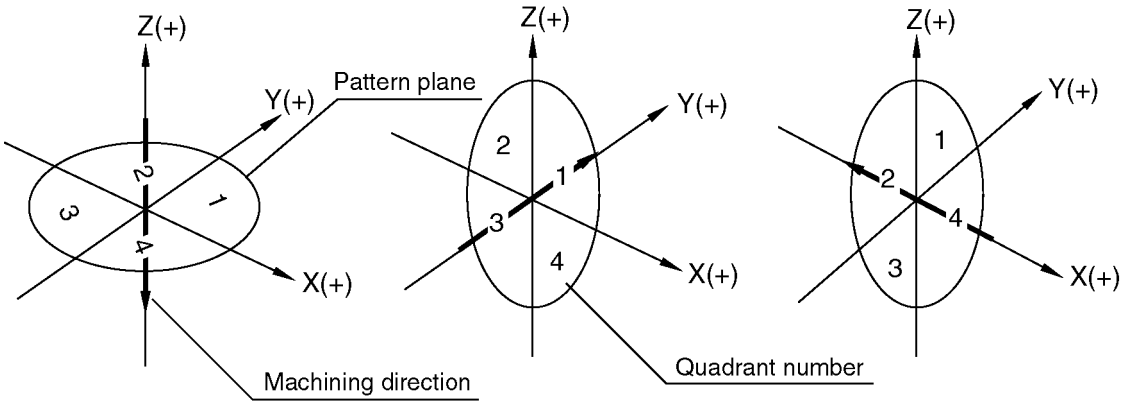
In each quadrant, pattern machining patterns can be used in combination in each quadrant but this is limited to the following pattern group combinations.

1) Combination of patterns 0, 1, 2 and 3
2) Combination of 0, 4 and 5.

Note 1: Be sure to specify the G74 block in a single block.
When another code is used together, L, R, P and F of the G74 block are processed in different senses.

Note 2: Upon RESET (Power On, M02, M30, etc.), initialization is performed as R = 0, P = 0, standard pattern machining rate 32 mm/min, G75 (pattern machining cancel).

・The quadrants vary with the machining direction.



・Quadrants when the machining direction is –Z

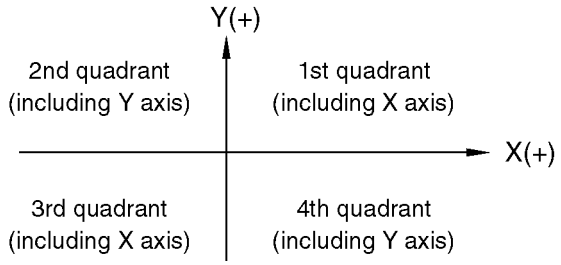
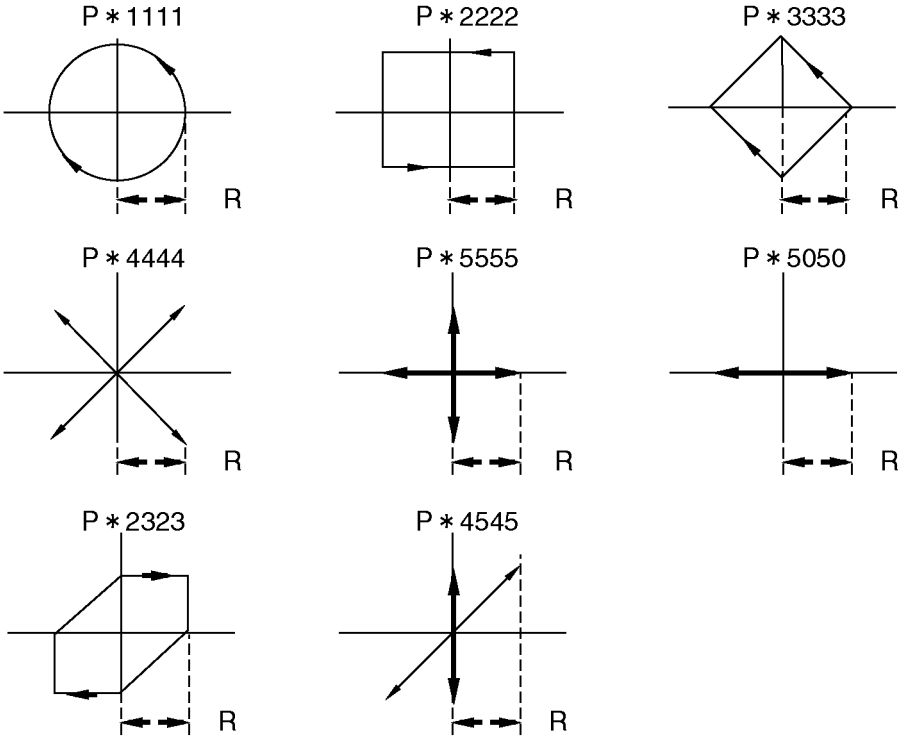


Fig. 4.55 Pattern machining planes and quadrants

Example:

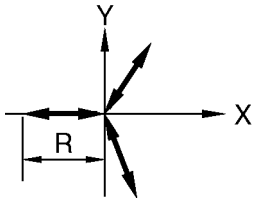


Example 1: Basic pattern machining (ordinary pattern machining, square, radius 1000 mm, rate 20 mm/min)

```
M26;                          (Machining ON)
G74 L0 R1.0 P02222 F20;       (Pattern ON)
G01 Z-1.0;                    (Z axis machining)
      :
      :
      :
```

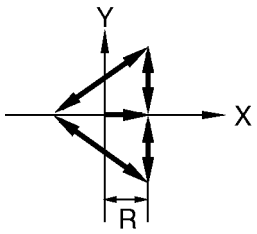
Example 2: Equal radial pattern

```
M26;                          (Machining ON)
G74 L1 R1.0 P3;               (Pattern ON) ..... 3 radiations
G01 Z-1.0;                    (Z axis machining)
      :
      :
```



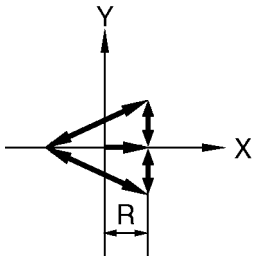
Example 3: Regular polygon pattern

```
M26 ;                         (Machining ON )
G74 L2 R1.0 P3;               (Pattern ON) ..... Regular triangle
G01 Z-1.0;                    (Z axis machining)
      :
      :
      :
```



Example 4: F pattern (Arbitrary pattern)

```
O3;
G41 G91 G01 X5.0;
Y5.0;
X-10.0 Y-5.0 ;
X10.0 Y-5.0;
Y5.0;
G40 X-5.0;
M99 ;
```




```
M26 ;                         (Machining ON )
G74 L3 R1.0 P3;               (Pattern ON) ····· F pattern
G01 Z-1.0;                    (Z axis machining)
      :
      :
```

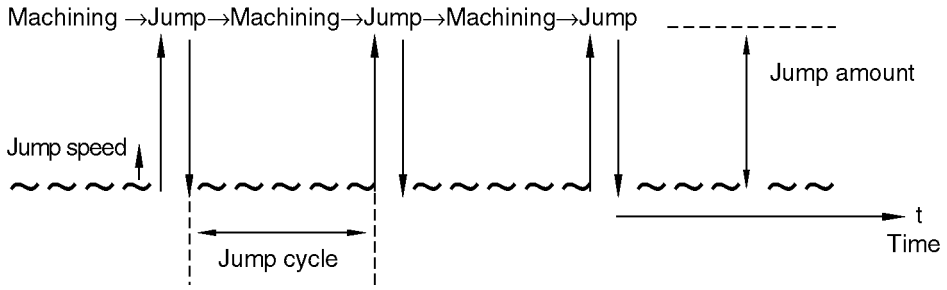
4.10.2 JUMP (G76/G77)

FUNCTION:

The jump function changes the movement on the machining axis to the opposite direction or an optional direction by the machining axis or axis 1, axis 2 or axis 3 except the machining axis. This function is turned on/off by the canned cycle G76/G77.

Once specified, the jump operation becomes Modal and is enabled in all machining blocks.

During machining, the jump operation is performed in reciprocating from in the direction of machining start point at the time interval set by "jump cycle", by the "jump amount" specified and at the "jump speed" specified.



The jump canned cycle can be used together with pattern machining canned cycle.

Jump can be classified by jump operations as follows.

1) Basic jump G76 | L*0S_T_F_ I_K_ |;
2) Variable jump G76 | L*0S_T_F_ I_K_ | A_B_C_D_;
3) Intermittent jump G76 | L*0S_T_F_ I_K_A_B_C_D_ | Q_U_V_W_;
4) Arbitrary point jump G76 | L*0S_T_F_ I_K_A_B_C_D_Q_U_V_W_ | X_Y_Z_ ;
5) Jump cancel G77 ;

Note: During the jump canned cycle, scaling, rotation and mirror image are disabled.

FORMAT:

G77; ----- [Jump OFF]

G78 | L * 0 | | S_T_F_ | ;

L * 0

- 0: Always 0
- 0: Jump manual key enabled
- 1: Jump manual key disabled

S : Jump amount
T : Jump cycle
F : Jump speed
I : Jump clamp amount
K : Servo Jump amount

EXAMPLE OF STATEMENT:

(1) G76 S2 T5 F4; Amount 0.4 mm, cycle 0.5 sec, speed 200 mm/min

(2) G76 S400 T500 F200; (ditto)

(3) G76;

EXPLANATION:

◎ The basic jump function moves the machining axis from its position to the opposite path of the machining direction by jump amount to return to the original position after lapse of the preset jump interval time. The jump function is turned on and off by the canned cycle command G76/77.

* When L is omitted, it is regarded as L00. In this case, ordinary jump is omitted for used.
* S, T and F are omissible. In this case, the Modal value given on the MC COND screen is output as a preset value.

Note: The G76 block must be specified in a single block. If another G code exits together, G76 blocks S, T, F, I, K, A, B, C, D, Q, U, V and W are processed in a different sense.

FUNCTION: Jump amount

EXPLANATION:

◎ The jump amount is specified by S (stroke) in the G76 block. Indirect command and direct command are available for this purpose.

(1) Indirect command (S1 - S16)

Code	Jump amount	Code	Jump amount
S01	200	S09	1800
S02	400	S10	2000
S03	600	S11	2200
S04	800	S12	2400
S05	1000	S13	2600
S06	1200	S14	2800
S07	1400	S15	3000
S08	1600	S16	3200

Unit: µm

* Corresponds to the jump amount scale on the machining condition screen.

(2) Direct command (S200 - S999999) □µm] (Standard resolution)
(S2000 -S9999990) [0.1 µm] (High resolution)

This is given to directly specify the jump amount. Direct command is made with 200 or more. (It is impossible with S17 - S199.)

In the range of 0.2 mm - 999.999 mm

Note: Decimal point input is impossible.

FUNCTION: Jump cycle

EXPLANATION:

◎ The jump cycle is specified by T (time) in the G76 block. Indirect command and direct command are available for this purpose.
(The preset jump cycle time does not include the jump operation time.)

(1) Indirect command (T1 - T16)

Code	Jump speed	Code	Jump speed
T01	100	T09	900
T02	200	T10	1000
T03	300	T11	1200
T04	400	T12	1400
T05	500	T13	1600
T06	600	T14	1800
T07	700	T15	2000
T08	800	T16	2200

Unit: 0.001sec.

* Corresponds to the jump cycle scale on the machining condition screen.

(2) Direct command (T100 - T10000) [msec] (Standard and High resolution)
This is given to directly specify the jump cycle. Direct command is made with 100 or more. (It is impossible with T17 - T99.) In the range of 0.1 - 10 sec.

Notes:

(1) Decimal point input is impossible.

(2) Execution is performed in units of 0.1 sec.
A value below low-order 2 digits (99 msec) is truncated.
Example: With T1234, execution is performed at a jump cycle of 1.2 sec.

FUNCTION: Jump speed

EXPLANATION:

◎ The jump speed is specified by F (feed rate) in the G76 block.
Indirect command and direct command are available for this purpose.

(1) Indirect command (F1 - F10)

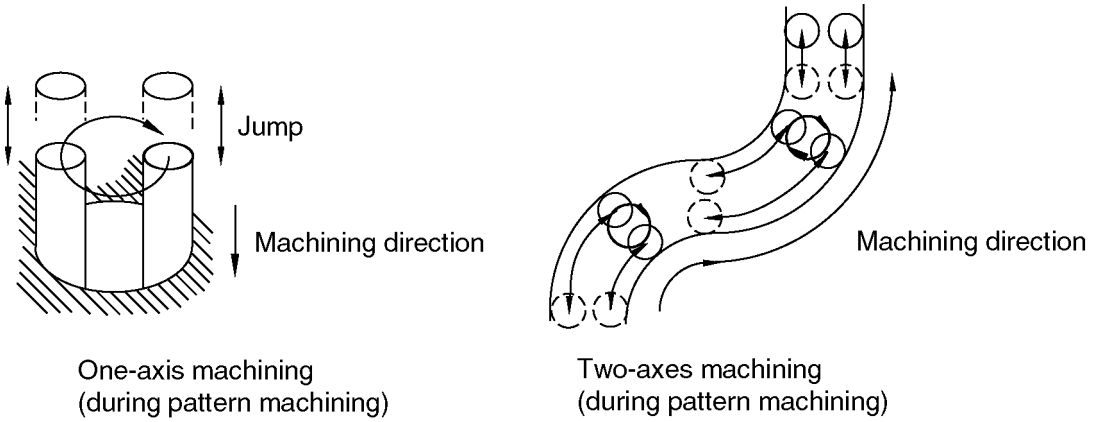
Code	Jump speed
F01	50
F02	100
F03	150
F04	200
F05	250
F06	300
F07	350
F08	400
F09	450
F10	500

Unit: mm/min

* Corresponds to the jump speed scale on the machining condition screen.

(2) Direct command (F50 - F500) [mm/min] (Standard resolution)
(F500 - F5000) [0.1 mm/min] (High resolution)
This is given to directly specify the jump speed. Direct command is made with 50 or more. (It is impossible with F11 - F49.)
In the range of 50 mm/min - 500 mm/min

Note: Decimal point input is impossible.



One-axis machining
(during pattern machining)

Two-axes machining
(during pattern machining)

Fig. 4.56

If the jump amount exceeds the start point of a current machining block, the jump control function operates to return the electrode at the machining start point. However, only when a machining is specified along with the travel in the previous block in the continuous block machining ON mode (G959), the electrode jumps two blocks reversely up to the start point of previous machining block (one block).
Figure 3.58 shows a jump example in the continuous block machining ON mode.

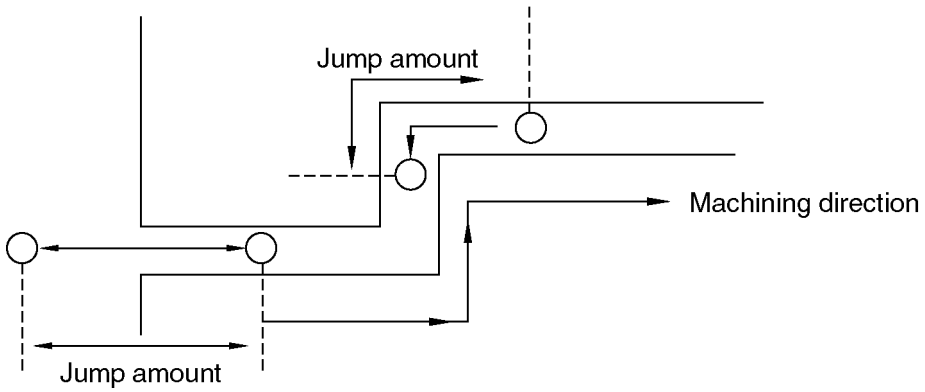


Fig. 4.57

Example 1:

```
M26;                          [Machining ON]
G74 R1000 P02222;             [Pattern ON]
G76 (L0) S1000 T500;          [Jump amount 1 mm, cycle 0.5 sec]
          [S5] [T5]
G01 Z-1.0;
 :
```

Example 2:

```
M26;                          [Machining ON]
G74 R1000 P22222;             [Pattern ON]
G76 S2000 T2000;              [Jump amount 2 mm, cycle 2.0 sec]
     [S10] [T15]
G01 Z-1.0;
 :
```

FUNCTION: Variable jump

FORMAT:

G76 | L*0 S_ T_ F_ I_ K_ | A_ B_ C_ D_ ;

L: Manual switch valid/invalid

S: Jump amount (start)

T: Jump cycle (start)

F: Jump speed

* When S, T and F are omitted, a Modal value is set.

I: Jump clamp amount (1 - 999.999mm)

K: Servo jump amount (1 - 999.999mm)

A: Effective depth (1 - 999.999 mm)

B: Jump renewal step amount (1 - 999.999 mm)

C: Jump amount at the end of machining

(The setting range/unit is the same as that for basic jump S.)

(1 - 16, 200 - 999999) (Standard resolution)

(1 - 16, 2000 - 9999990) (High resolution)

D: Jump interval at the and of machining

(The setting range/unit is the same as that for basic jump T.)

(1 - 16, 100 - 10000) (Standard and High resolution)

EXPLANATION:

◎ The variable jump function renews the jump amount and jump time of basic jump according to the depth and is enabled except for extended pattern machining.
The variable jump function changes the jump condition automatically from the jump amount (S) and jump cycle (T) at the start of machining so that they may be the jump amount (C) and jump cycle (D) at the end of machining, specified by effective depth (A).
The jump renewal frequency depends on the effective depth and jump renewal step.
(Frequency = Effective depth □□Jump renewal step)

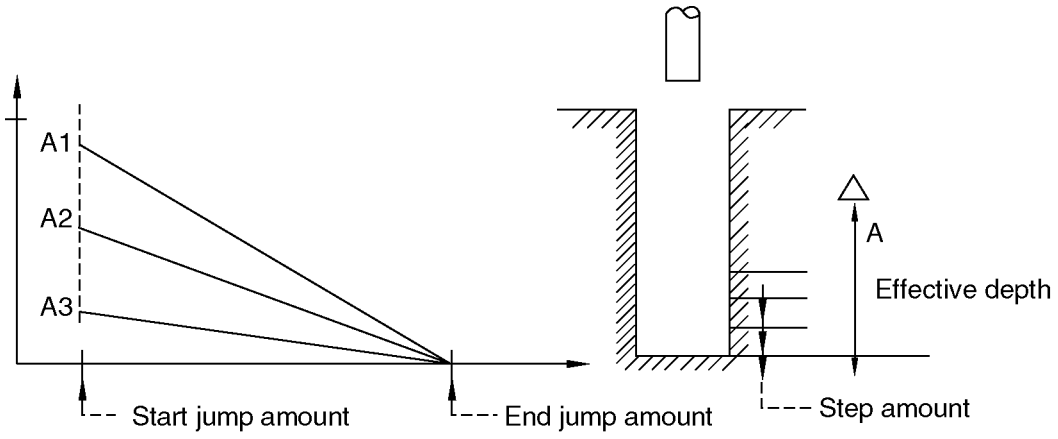


Fig. 4.58 Where the hole depth is arbitrarily set for same machining.

The indirect command of jump amount (S1 - S16) changes from 0.2 mm in 0.2 mm steps. If the renewal frequency for S15 - S16 is set to 10, the jump is renewed in 10 µm steps.
Though the indirect jump cycle (T1 – T16) is not continuous, if T6 – T13 are set to the update times 10, the jump cycle is updated between 0.6 sec and 1.6 sec in a step of 0.1 sec. (A step below 0.1 sec is omitted.)
When the manual switch is turned on at variable jump, the jump condition of S, T and F can be changed manually.
Note: The jump amount/cycle/speed upon execution of the variable jump is not displayed on the machining condition screen.

FUNCTION: Intermittent jump

FORMAT: G76 | L*0 S_ T_ F_ I_ K_ A_ B_ C_ D_ | Q_ U_ V_ W_ ;

L: Manual switch valid/invalid

S, T, F: Jump amount, jump cycle, jump speed

* When they are omitted, a Modal value is set.

I, K: Jump clamp amount, servo jump amount

A, B, C, D: Variable jump

Q: Intermittent start jump frequency (1 - 100) (times)

U: Intermittent jump amount

(The setting range/unit is the same as that for basic jump S.)

(1 - 16, 200 - 999999) (Standard resolution)

(1 - 16, 2000 - 9999990) (High resolution)

V: Intermittent jump cycle

(The setting range/unit is the same as that for basic jump T.)

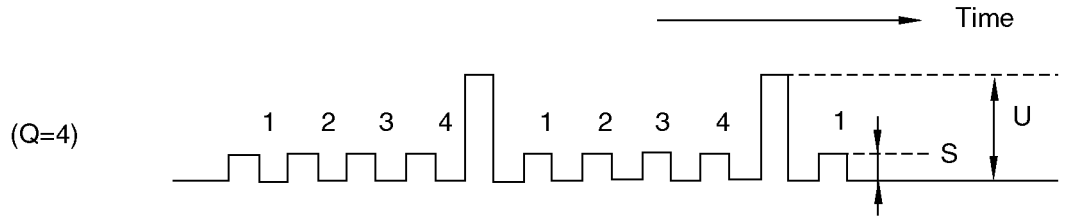
(1 - 16, 100 - 10000) (Standard and High resolution)

W: Intermittent jump frequency (1 - 100) times

* When omitted, it is regarded as W1.

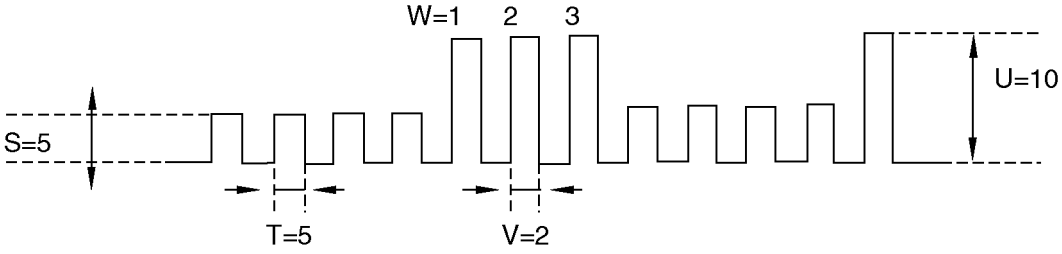
EXPLANATION:

◎ The intermittent jump command repeats ordinary jump according to the amount/cycle/frequency specified by U/V/W after completion of the frequency specified by Q, and can be used together with the variable jump command.
When W is omitted, intermittent jump is performed only once.



Example :

G76 S5 T5 Q4 U10 V2 W3 ;



When manual operations are enabled, S and T can be changed manually.

Note: The 2nd jump execution state by Q, U, V and W is not displayed on the machining condition screen.

FUNCTION: Arbitrary point jump

FORMAT: G76 | L*0 S_ T_ F_ I_ K_ A_ B_ C_ D_ Q_ U_ V_ W_ | X_ Y_ Z_ ;

X: X axis offset from machining start point

Y: Y axis offset from machining start point

Z: Z axis offset from machining start point

EXPLANATION:

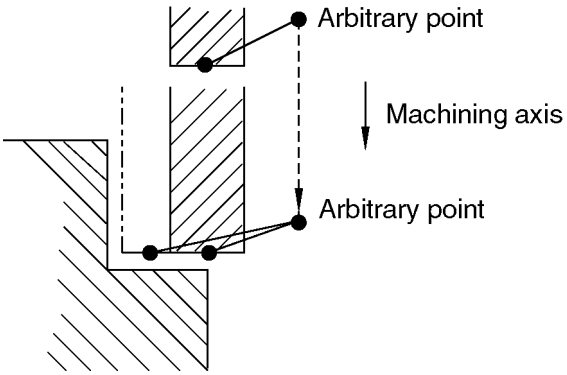


Fig. 4.59

The offset point in the pattern axis is always the predetermined point jumping from the pattern center, and the machining axis varies with constant jump stroke from the jump start point as the machining progresses.

In the arbitrary point jump mode, therefore, the point in the pattern axis is fixed and the machining axis jumps to varying point sequentially.

4.10.3 MACHINING (G80/81/82/83)

Once specified, the canned machining cycle command is executed until it is cancelled, each time a travel command is given.

(Machining is performed irrespective of machining ON/OFF (M26/M27).)

The canned machining cycle is limited to linear interpolation.

The canned machining cycle is effective in multiple plate machining.

The machining cycle is classified as follows:

1) Automatic depth- set machining cycle (G81)
2) Predetermined depth- set machining cycle (G82)
3) Ordinary machining cycle (G83)
4) Machining cycle cancel (G80)

Description of machining cycle mode designation: L**:

L＊＊

[Retract part]

0: Retract (return to the machining start point)

1: No retract (stay at the machining end point)

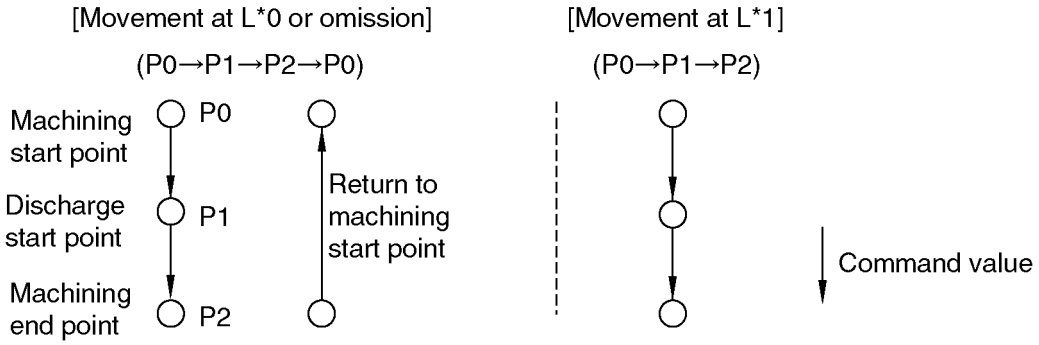
[Canned cycle part]

0: Automatic cancellation of the canned cycle after completion of one-block machining (G80) (In the next travel command block, the canned machining cycle is not executed.)

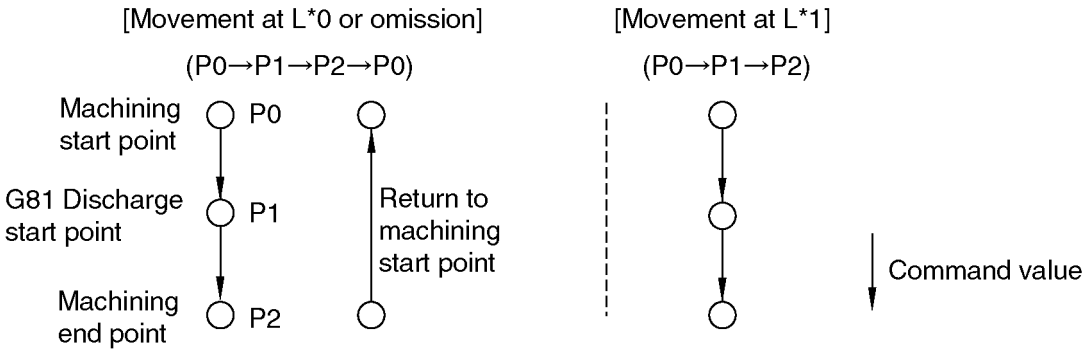
1: Standard canned cycle (Continuation of the canned machining cycle until G80 is specified)

Note: Do not use G81, G82, G83 (machining canned cycles) and G84, G85, G86 (positioning canned cycles) in the G48 (scaling), G68 (rotation), G71 (mirror image), or coordinate system rotation mode.

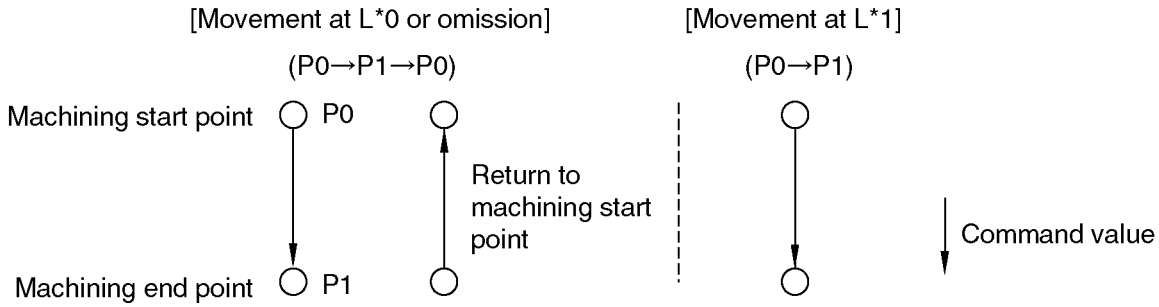
1) G81: Automatic depth-set machining cycle



2) G82: Predetermined depth- set machining cycle



3) G83: Ordinary machining cycle



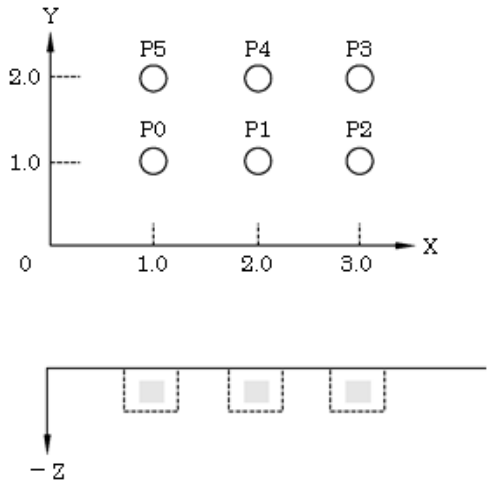
Time- managed machining can be specified for each machining cycle.
Do not use a machining canned cycle in the scaling, rotation, mirror image, or coordinate system rotation mode.

FUNCTION: G81/82/83 canned machining cycle cancel

FORMAT: G80 ;

SAMPLE:

```
M26 G91 ;          (Machining ON)
G83 L10 Z-1.0 ; (P0 machining)
G00 X1.0 ;         (P1 machining)
X1.0 ;             (P2 machining)
Y1.0 ;             (P3 machining)
X-1.0 ;            (P4 machining)
X-1.0 ;            (P5 machining)
G80 ;              (Canned cycle cancel)
```



Note: When 0 is set at L 2nd digit in G83 L00 (omissible) Z-1.0;, only P0 is machined.
(G83 is not a Modal command.).

FUNCTION: Automatic depth-set machining

FORMAT: G81 | L_ P_ X_ Y_ Z_ (C_B_W_) | ;

P: Time-managed machining (Described later)

X/Y/Z(/C/B/W): Machining amount

EXAMPLE OF STATEMENT:

(1) G81 Z-10. ;

(2) E3106 ; ····· (Glazed machining)

G81 P60. Z-12. ;

EXPLANATION:

◎ When this canned cycle is specified, automatic depth-set machining continues a movement in the specified axial direction until discharge is started by G01 interpolation.

◎ When discharge is started, machining is performed from the discharge start point by taking the specified amount as an increment.

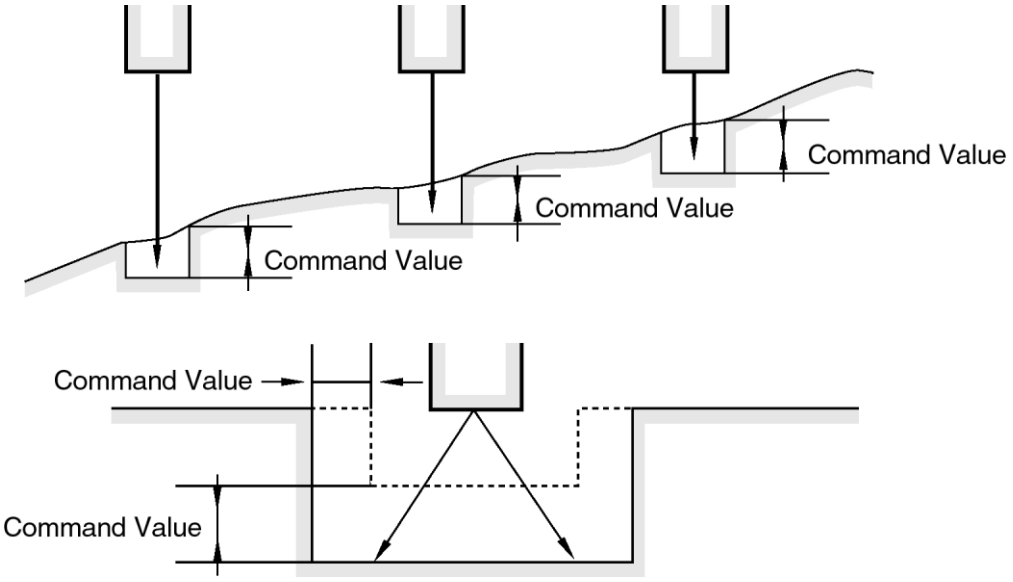


Fig. 4.60

FUNCTION: Predetermined depth-set machining

FORMAT: G82 | L_ P_ X_ Y_ Z_ (C_B_W_) | ;

P: Time-managed machining (Described later)

X/Y/Z(/C/B/W): Machining amount from the point detected by G81

EXAMPLE OF STATEMENT:

G82 Z-10. ;

EXPLANATION:

◎ Predetermined depth-set machining is performed on condition that G81 automatic depth-set machining has been completed before this canned cycle.
Machining is performed from the discharge start point detected at automatic depth-set machining by taking the specified moving amount as an increment.

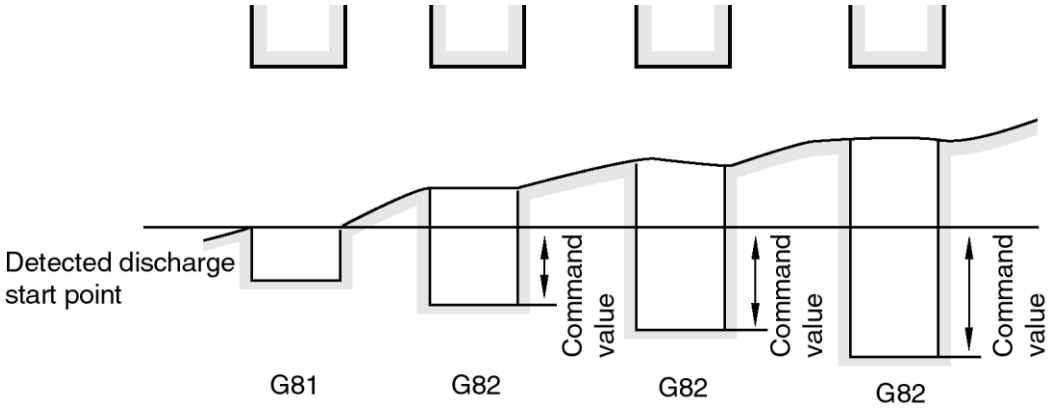


Fig. 4.61

FUNCTION: Ordinary machining

FORMAT: G83 | L_ P_ X_ Y_ Z_ (C_B_W_) | ;

P: Time-managed machining (Described later)

X/Y/Z (/C/B/W): Machining amount

EXAMPLE OF STATEMENT:

G90 ;

G83 Z-10. ;

EXPLANATION:

◎ Unlike automatic depth-set machining and predetermined depth-set machining, ordinary machining is performed in the coordinate system of absolute/incremental commands (G90/G91) according to the command value.

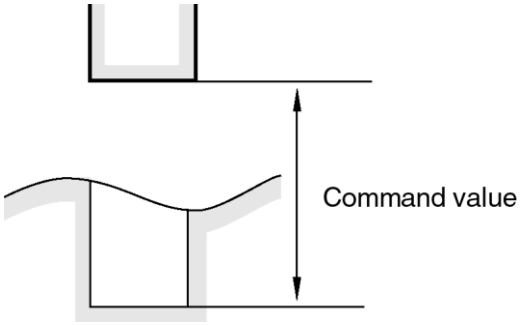


Fig. 4.62

FUNCTION: Time-managed machining

FORMAT: G81/82/83P _ | X/Y/Z (/C/B/W) _ | ;

① Input with decimal point ... unit [minute]
0.1≦P≦10000.

② Input with decimal point
100≦P≦10000000 [0.001 minute] (Standard resolution)
1000≦P≦100000000 [0.0001 minute] (High resolution)

EXPLANATION:

◎ Canned machining cycle time management specification can be made in units of 0.001 min by address P.
(0.1 - 10000 min)
When the command time has elapsed or the specified machining amount has been reached, whichever is earlier, time-managed machining terminates.
With this designation, machining may be performed by the specified time lapse without moving the specified machining amount. Care should be taken in dimension management.
This is an effective function in machining that is difficult to perform machining/ dimension management and to provide a machining termination, e.g., glazed machining.

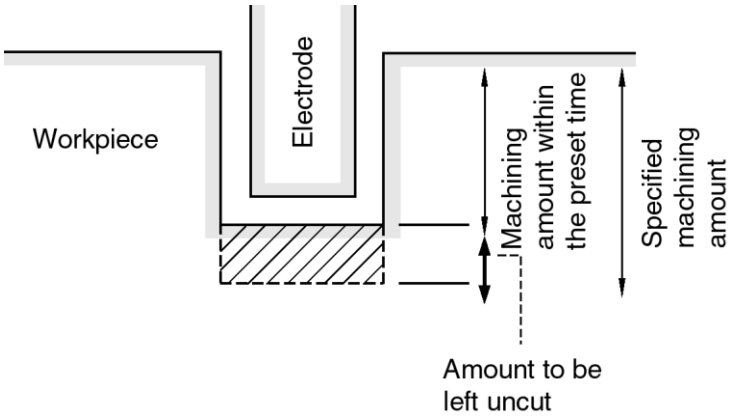


Fig. 4.63

4.10.4 POSITIONING (G84/85/86)

The positioning is a function of measuring the edge, inside and outside of the workpiece, performing positioning at the specified position or center and storing its machine coordinates and measurement results into the memory.

This is enabled only in the specified block and not a Modal command.
Do not use a positioning canned cycle in the scaling, rotation, mirror image, or coordinate system rotation mode.

The canned positioning cycle is classified as follows:

1) Edge positioning (G84)
2) Inside positioning (G85)
3) Outside positioning (G86)

FUNCTION: Edge positioning (L1: Outside edge, L0: Inside edge)

FORMAT: G84 X/Y/Z(/C/B/W) | I/J/K_L_R_F_T_ A_ H_ U_ V_ | ;

X/Y/Z(/C/B/W): Measurement axis (An enough distance between the axis to be measured and the edge is specified by incremental value.) (At one-edge measurement, 4 axes (option: 6 axes) can be specified.)

Note: A maximum of 3 axes (3 edges) can be measured simultaneously. (X-, Y- and Z-axes)

I/J/K: Measurement axis rapid traverse amount (absolute value)
For each of X, Y and Z, 3 axes of I, J, K can be specified at the same time.
Note: The input order of I, J and K is an alphabetic order.

L: Measurement method (0 or omission: Inside; 1: Outside)

R: Positioning after measurement (absolute value)
Note: When omitted, the parameter value (1 mm) is adopted.

F: Measurement rate
Note: When it is omitted, 50 mm/min is set.

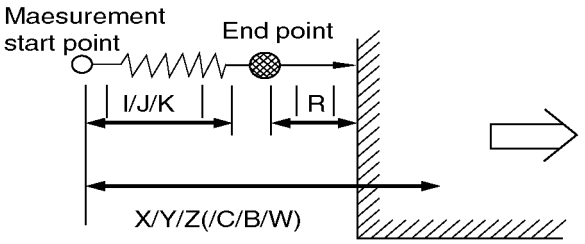
T: Measurement coordinates storing number (1 - 1100)
1 - 200: Stored in electrode offset number (1 - 200)
500 - 999: Stored in macro variable number (#500 - #999)

A: Rotation angle

H: Plane (17/18/19)

U: X-direction shift amount (only for Z-axis measurement)

V: Y-direction shift amount (only for Z-axis measurement)



Measurement coordinate storing No.

Axis	T01	T02	T03
X Y Z (C) (B) (W)			

Fig. 4.64

Note: H_ is needed to specify rotational angle A_ and is valid for up to 2 axes.
(Only H17: X-Y plane is valid.)

The use of rotation angle A_ with a modal value other than G17 (X-Y plane) causes an alarm.

FUNCTION: Inside positioning (groove/hole center)

FORMAT: G85 X/Y/Z(/C/B/W)_ | I/J/K_L_ P_ R_ F_ T_ Q_ A_ H_ | ;

X/Y/Z (/C/B/W): Measurement axis (An enough distance between the axis to be measured and the inside is specified by incremental value.)
(At one-inside measurement, 4 axes (option: 6 axes) can be measured.)
Note: A maximum of 3 axes can be measured simultaneously.
(X-, Y- and Z-axes)

I/J/K: Measurement axis rapid traverse amount (absolute value)
For each of X, Y and Z, 3 axes of I, J, K can be specified at the same time.
Note: The input order of I, J and K is an alphabetic order.

L: Contents of measurement (omission or 0: Central coordinates;
1: Internal diameter)
Note: The P must be specified whenever 1 is selected.

P: Electrode diameter (Used with L1 in a pair. When it is omitted, alarm is caused.)

R: Positioning after measurement (absolute value)
Effective for 3-axis measurement (axis other than plane)
(For example, if R is omitted on the XY plane, the measurement in the Z-axis is not executed.)

F: Measurement rate
Note: When it is omitted, 50 mm/min is set.

T: Measurement coordinates storing number (1 - 1100)
1 - 200: Stored in electrode offset number (1 - 200)
500 - 999: Stored in macro variable number (#500 - #999)

Q: Fall amount at inside diameter measurement

A: Rotation angle

H: Plane (17/18/19)

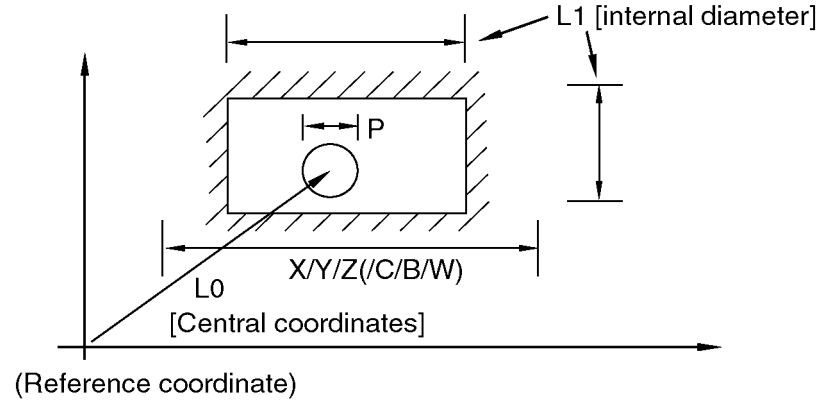


Fig. 4.65

Note: H_ is needed to specify inside measurement of 2 axes or more or A_ (rotational angle).
(Only H17: X-Y plane is valid.)

The use of rotation angle A_ with a modal value other than G17 (X-Y plane) causes an alarm.

FUNCTION: Outside positioning (plate/column center)

FORMAT: G86 X/Y/Z(/C/B/W) _ | I/J/K_ L_ P_ R_ F_ T_ | ;

X/Y/Z(/C/B/W): Measurement axis (An enough distance between the axis to be measured and the outside is specified by incremental value.) (At one-axis outside measurement, 4 axes (option: 6 axes) can be measured.)

Note: A maximum of 3 axes can be measured simultaneously. (X-, Y- and Z- axes)

I/J/K: Measurement axis rapid traverse amount (absolute value)
(For each of X, Y and Z, 3 axes of I, J, K can be specified at the same time.)

Note: The input order of I, J and K is an alphabetic order.

L: Contents of measurement (omission or 0: Central coordinate;
1: External diameter)

Note: The P must be specified whenever I is selected.

P: Electrode diameter (Used with L1 in a pair. When it is omitted, an alarm is caused.)

R: Positioning after measurement (absolute value)
(Effective for 3-axis measurement. If R is omitted, the measurement in the axis other than plane is not executed.)
(For example, if R is omitted on the XY plane, the measurement in the Z-axis is not executed.)

F: Measurement rate

Note: When it is omitted, 50 mm/min is set.

T: Measurement coordinates storing number (1 - 1100)

0: Stored in the number specified by parameter No. 622 “STAN-DARD ELECTRODE No.”.

1 - 200: Stored in electrode offset number (1 - 200)

500 - 999: Stored in macro variable number (#500 - #999)

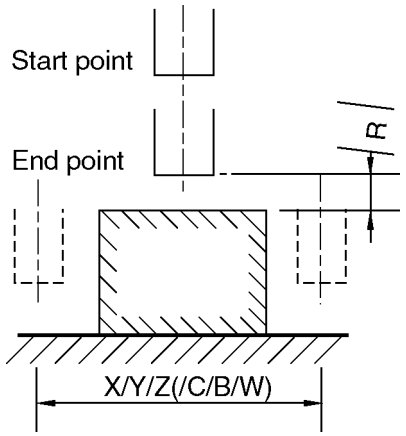


Fig. 4.66

4.10.5 MACHINING START POINT/INTERRUPT POINT RETURN FUNCTION (G98/99) CONTINUOUS BLOCK (CONTOUR) MACHINING OFF/ON (G958/959)

FUNCTION: Machining start point return ─ G98
Machining interrupt point return ─ G99
Continuous block (contour) machining OFF ─ G958
Continuous block (contour) machining ON ─ G959

FORMAT: G98 ;
G99 ;
G958 ;
G959 ;

EXPLANATION OF STATEMENT:

```
M26;            [Machining ON]
G91 G959;    [Continuous block (contour) machining ON]
G01 X2. ;
Y2. ;
X-2. ;
Y-2. ;
G98;          [Machining start point return]
G958;        [Continuous block (contour) machining OFF]
M30;
```

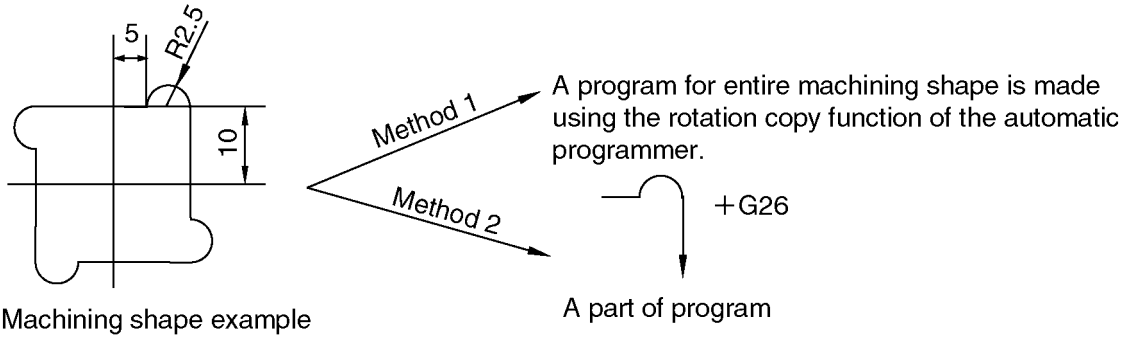
EXPLANATION:

◎ When a contour was machined with G959, the axis can be returned to the machining start point by the G98. Therefore, a T-slot can be machined with G98 without using a retrace program. (However, the maximum number of blocks is 50.)

4.10.6 ROTATION COPY (G26)

FUNCTION:

The rotation copy function repeatedly executes a program for a certain shape as a subprogram by rotating the axis about the specified point as the rotation center. For the sake of simplicity, only a part program in which the same shape is repeated in rotational direction may be used, such as gear and motor core.



Whether the rotation copy function of automatic programmer or G26 is used should be determined depending on the machining shape, repeat count, easiness of programming, program length, easiness of editing, etc.

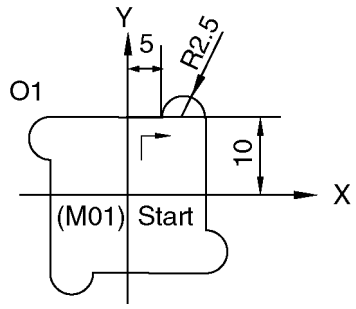
FORMAT: G26 (X_) (Y_) (S_) A_ (O_) (P_) (R_) (M_) (B_) (Q_) ;

( ) Omissible

X, Y: Rotation center (absolute or incremental)
S: Copy start angle (0 - ±360.000)
A: Rotation copy angle (0 - ±360.000)
O: Subprogram No. to be called (1 - 9999)
P: Sequence No. of subprogram to be called (1 - 9999)
R: Copy repeat count (0 - 9999)
M: Specify M01 M00 if a temporary stop is made after one-time repetition by R.
B: Specify how many blocks (1-256 blocks) are executed from the top of program called after copy by the number of times specified by R.
If B is omitted, the subprogram execution is ended after copy by the number of times specified by R.
Normally, B is omitted.
Q: Number of divisions (0 - 999)

The following sample 1 shows a program example using the G26 with the start point in the rotation center.

SAMPLE 1:

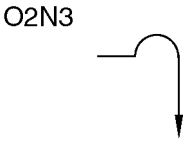


Main program (O1)

```
G92 X0 Y0 ;
M26 G959 ;
G91 G01 G41 Y10000 D1 ;
G26 (X0) Y-10000 (S0) A-90000 O2 (P3) R4 (M01) ;
G40 G01 Y-10000 ;
M30 ;
```

Subprogram (O2)

```
N3 (G01) X5000 ;
G02 X5000 I2500 ;
G01 Y-10000 M99 ;
```



End of subprogram

EXPLANATION:

◎ Rotation center (X, Y)

In the incremental mode, the rotation center is the point as viewed from the point where G26 is specified.

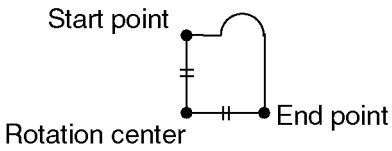
In this example, it is (X0) Y-10000.

In the absolute mode, specify the workpiece coordinate value of rotation center.

In this example, it is G90 X0 Y0.

Note 1: If rotation center entry is omitted, the point where G26 is specified is used as rotation center.

Note 2: The rotation center must be positioned at the same distance from start point and end point of subprogram.

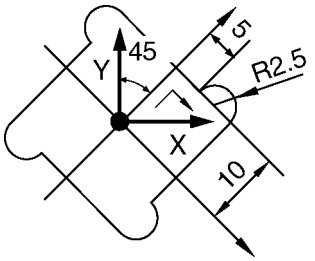


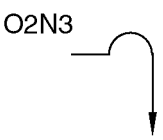
◎ Copy start angle (S)

The copy start angle specifies the first copy start angle with the rotation center designated as the center. The rotational direction CCW is positive, and CW is negative direction. In this example, the first direction is same as subprogram, and therefore the copy start angle is (S0). Usually, a subprogram should be made so that the copy start angle is S0, like this example. If S is omitted, it is same as S0.

If an entire machining shape is to be rotated, in general, do not use S but the rotation function (G68) as shown in sample 2.

SAMPLE 2:

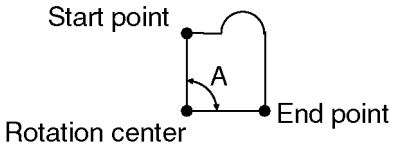




Subprogram (O2)
N3 (G01) X5000;
G02 X5000 I2500;
G01 Y-10000 M99;

Main program
G92 X0 Y0;
G68 (X0) (Y0) R-45000; ⟶ Rotation −45°
M26 G959;
G91 G01 G41 Y10000 D1;
G26 (X0) Y-10000 (S0) A-90000 O2 (P3) R4 (M01) ;
G40 G01 Y-10000;
M69; ⟶ Rotation OFF
M30;

◎ Rotation copy angle (A)
If copy is made several times, specify rotation angle per subprogram. Usually, the angle between start point and end point of subprogram on the basis of rotation center is specified. The rotational direction CCW is positive, and CW is negative direction.

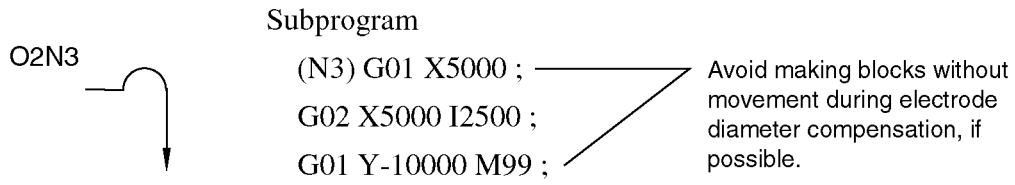


In this example, specify A-90000.

The least input increment is 0.001 degree [standard resolution] or 0.0001 degree [high resolution]. Use Q (division command) if division less than the least input increment is necessary.

Note 1: Omission of A causes an alarm [1057: G26 COMMAND ERROR].
Note 2: Specify of A0 results in repeated copy at the same position.
Note 3: To use Q (division command), see “Division Command”.

◎ Program No. of subprogram (O) and sequence No. (P) to be called Assign the program number (O) or sequence number (N) to the first block of subprogram to be called. For calling, use O (program No.) or P (sequence No.).

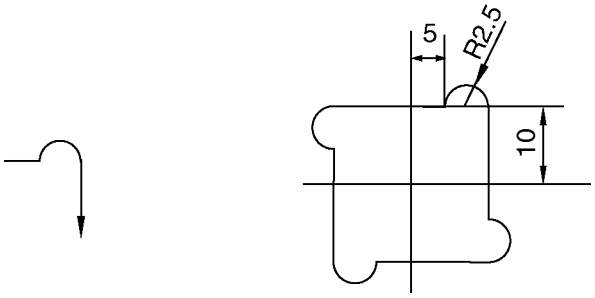


Note 1: If O No. is omitted, the program number of current program (main program) is called.

Note 2: If P is omitted, the top of program specified by O is called.

◎ Copy repeat count (R)

Specify the count by which the rotation copy of subprogram is made. In the program example 1, R4 is specified.



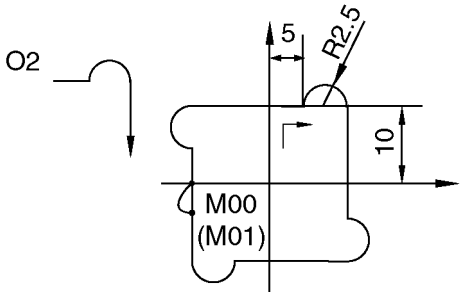
Note 1: If R is omitted, copy is made only once.

Note 2: If R0 is specified, no copy is made.

◎ Temporary stop during copy operation (M01 or M00)

If M01 or M00 is specified in the G26 block, a temporary stop is made after execution of one-time repetition by R. The copy operation continues when the START key is pressed again.

In the program example 1, operation is stopped temporarily by M01 or M00 after execution of copy 3 times (R=4).



(G91) ;

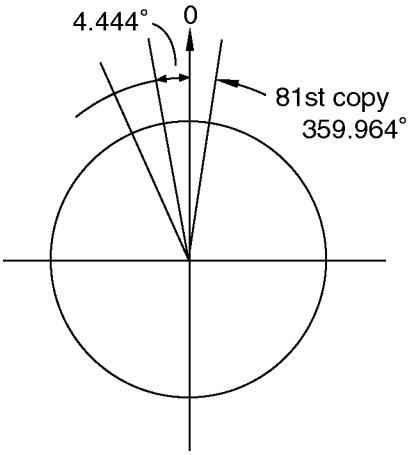
G26 X0 Y-10000 S0 A-90000 O2 R4 M00 ;

Note 1: The M01 (optional stop) can stop operation temporarily only when the OPT. STOP key valid. The M00 always causes temporary stop.

◎ Division command (Q)

As the least input increment of rotation copy angle (A) is 0.001 degree [standard resolution] or 0.0001 degree [high resolution], if higher division precision than the least input increment is required, use the number of division command.

EXAMPLE:



If 360° is divided to 81 divisions, the rotation copy angle per division is 360°/ 81 = 4.4444° ....

If rotation copy angle (A) is 4.444, the angle at the end of 81st rotation copy is 4.44°x 81 = 359.96°, thus causing difference from the desired 360°.

Fig. 4.67

In this case, use of Q (division command)prevents a fraction from being truncated internally by the NC function and minimizes cumulative error.

Q: Number of divisions for angle specified by A (eg. 81)

A: Angle to be divided (e.g. 360.)

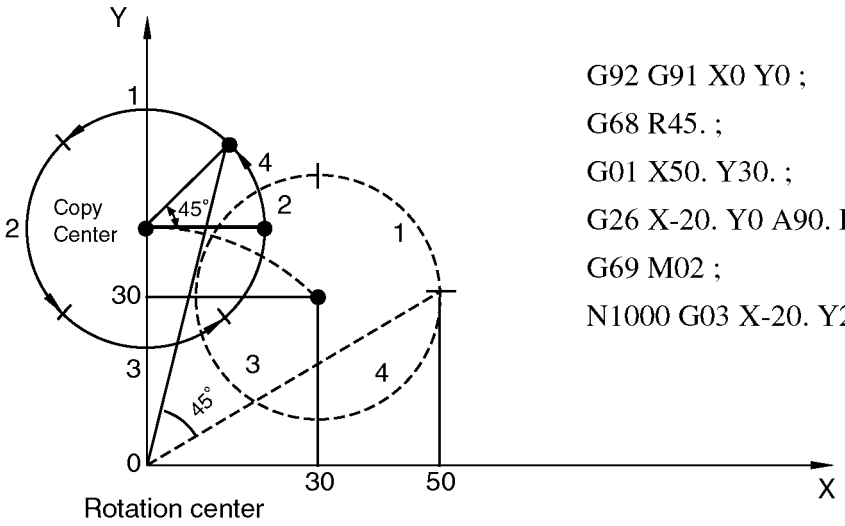
Note 1: When Q (division command) is used, the copy start angle (S) is the copy start division point. Copy start angle = A/Q x S (copy start division point)

◎ Additional precautions on using rotation copy function

(a) Calling another subprogram by G26 or M98 from the subprogram called by G26 causes an alarm [1005: G48, G68, G26 EXCESS APPOINT].

(b) If the coordinate system rotation or G68 (rotation) is specified before G26 is specified, the G26 rotation center and copy start angle rotate about that rotation center.

SAMPLE 3: G68 and G26




```
G92 G91 X0 Y0 ;
G68 R45. ;
G01 X50. Y30. ;
G26 X-20. Y0 A90. P1000 R4 ;
G69 M02 ;
N1000 G03 X-20. Y20. I-20. M99 ;
```

Fig. 4.68

(c) If G48 (scaling) is specified before G26 is specified, the G26 rotation center is scaled against that scaling center.

SAMPLE 4: G48 and G26

```
G92 X0 Y0 ;
G48 X0 Y0 P500 ;
G01 X50. Y30. G91 ;
G26 X-20. Y0 A90. P1000 R4 ;
G47 M02 ;
N1000 G03 X-20. Y20. I-20. M99 ;
```

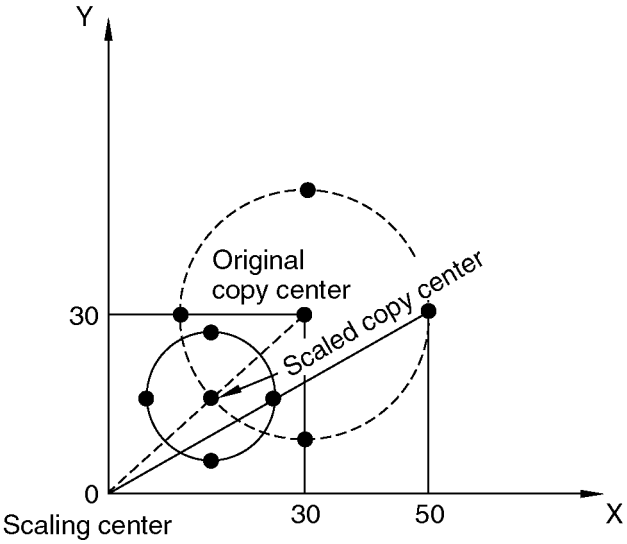


Fig. 4.69

(d) The G26 is a one-shot G code.

Note: Do not specify the rotation copy function together with axis exchange, mirror image or coordinate system switching function.

4.11 AUXILIARY (M)/SPINDLE (S)/ELECTRODE (T) FUNCTIONS

This section describes the commands for auxiliary, spindle, and electrode functions.

The auxiliary function, programmed with a numeral following the address M, specifies the machine auxiliary operation.

The spindle function, programmed with a numeral following the address S, specifies the spindle speed of a rotary unit.

The electrode function, programmed with a numeral following the address T, specifies the electrode No. on the electrode changer.

The functional words (commands) listed in this section are as follows:

Table 4.7 Auxiliary, spindle, and electrode functions

Word	Modal	Function	Item
M00	×	Program stop	4.11.1
M01	×	Optional stop	4.11.1
M02	×	End of program	4.11.1
M30	×	End of tape	4.11.1
M19	×	Touch stop disregard function	4.11.2
M24	○	Polarity reverse ON	4.11.3
M25	(C)	Polarity reverse OFF	4.11.3
M28	○	Dielectric fluid supply ON	4.11.4
M29	(E)	Dielectric fluid supply OFF	4.11.4
M26	○	Machining ON	4.11.5
M27	(D)	Machining OFF	4.11.5
M98	×	Subprogram call	4.11.6
M99	×	End of subprogram	4.11.6
M06	×	Electrode change	4.11.7
M07	×	Loading	4.11.7
M08	×	Unloading	4.11.7
M57		C-axis function	4.11.8
M56	○	Indexing function	4.11.8
M03	(A)	Rotating function	4.11.8
M05	×	Rotation stop	4.11.8
M18	×	DH head function	4.11.9
M50		Blowing ON, Low pressure blowing ON	4.11.10
M09	○	Blowing OFF, All valves blowing OFF, Low pressure blowing ON	4.11.10
M40~49	(B)	Valve 1-10 blowing ON	4.11.10
M72		High pressure blowing ON	4.11.10
M73		High pressure blowing OFF	4.11.10
M58	×	Tank descent	4.11.11
M59	×	Tank ascent	4.11.11
M60	×	Pallet change	4.11.12
M70	○	SL machining ON	4.11.13
M71	(F)	SL machining OFF	4.11.13
M78	○	Rotary axis break ON	4.11.14
M79	(G)	Rotary axis break OFF	4.11.14
S	○	Spindle function	4.11.8
T	○	Electrode function	4.11.7

4.11.2 TOUCH STOP DISREGARD FUNCTION (M19)

FUNCTION: Touch stop disregard function

FORMAT: M19;

EXPLANATION:

◎ In the machining OFF (M27) mode, G00 mode or measurement operation mode, the touch stop function is always enable. (When a touch occurs, an alarm is caused.) Inversely, in the positioning block just after the machining block, positioning block just before a touch, automatic operation stop or halt, ATC operation, retract operation or return operation, the touch stop function is automatically disabled. The touch stop disregard function disables the touch stop function. Even in the above effective range of touch stop, the touch signal is disabled only in the block where M19 was programmed. Therefore, if the electrode comes into contact with the workpiece with this block, movement is not stopped.

Note 1: When a travel command is specified in the same block as M19, M19 is executed before movement.

Note 2: M19 is not a Modal command.

4.11.1 AUTOMATIC OPERATION STOP/TERMINATION (M00/01/02/30)

The M function is used to specify the auxiliary of the machine. Exceptionally, the auxiliary function of the NC inside is provided to perform automatic operation stop/termination processing.

FUNCTION: Program stop
FORMAT: M00;
EXPLANATION:

◎ At automatic operation, when the M00 block is read, reading the next block is stopped and automatic operation is halted.
When a travel command is specified in the same block as this command, M00 is executed after termination of movement.

FUNCTION: Optional stop
FORMAT: M01;
EXPLANATION:

◎ At automatic operation, when the M01 block is read, reading the next block is stopped when the optional stop function is enabled (set on the NC function screen), and automatic operation is halted.
If a travel command is specified in the same block as this command, M01 is executed after termination of movement.

FUNCTION: M02 - End of program
M30 - End of tape
FORMAT: M02/30;
EXPLANATION:

◎ At automatic operation, when the M02/30 block is read, the automatic operation terminates. (Reset status)
When a travel command is specified in the same block as this command, M02/30 is executed after completion of the movement.
At program operation, when the M30 block is read, and the automatic operation terminates.

411.3 POLARITY REVERSE FUNCTION (M24/25)

FUNCTION: M24 - Polarity reverse ON
M25 - Polarity reverse OFF

FORMAT: M24/25;

EXPLANATION:

◎ The polarity state of the spindle set by machining condition polarity (08: POL) is put into the reverse ON/OFF state.
When this command is specified in the same block as a travel command, M24 is executed before a movement and M25 is executed after a movement.
Upon power-on or RESET, M25 is executed.

◎ Use this for a machining with workpiece installed on the spindle and electrode on the table.

Note: The polarity (08: POL) of the machining conditions does not change by the polarity reverse function.

4.11.4 DIELECTRIC FLUID SUPPLY FUNCTION (M28/29)

FUNCTION: M28 - Dielectric fluid supply ON (fluid supply)
M29 - Dielectric fluid supply OFF (fluid drain)

FORMAT: M28/29;

EXPLANATION:

◎ Dielectric fluid ON/OFF can be specified not only by key but also by automatic operation. When the automatic dielectric fluid supply/drain unit is provided, the drain key can be used in the same way as M29. When this command is specified in the same block as a travel command, M28 is executed before a movement and M29 is executed after a movement.
Upon power-on or RESET, M29 is executed.

◎ The dielectric fluid supply is completed when the float is turned on. Therefore, it will take time to go to the next block until the tank is filled with dielectric fluid.

4.11.5 MACHINING FUNCTION (M26/27)

FUNCTION: M26 - Machining ON
M27 - Machining OFF

FORMAT: M26/27;

EXPLANATION:

◎ This command performs machining power ON/OFF switching.
In the machining OFF mode, F feed (feed-per-minute) is performed with G01/G02/G03.
In the machining ON mode, machining feed (servo feed) is performed with G01/G02/G03.
(Even in the machining ON mode, the measurement is performed at the F feed. Even in the machining OFF mode, machining feed (servo feed) is performed with machining canned cycle.)
When this command is specified in the same block as a travel command, M26 is executed before movement and M27 is executed after movement. At power-on or RESET, M27 is executed.

Note 1: If F is changed in the process of machining, only Modal is renewed and enabled in the machining OFF mode.

Note 2: When a touch signal is detected in the machining OFF mode, an alarm is detected.

Note 3: After the machine is operated in the dry run mode, the machining power is not turned on in the machining ON mode. Therefore, F feed and machining feed are performed at the rates set on the portable operation panel. (When step feed is selected, it is performed at low rate.)

Note 4: Machining ON/OFF mean turning on/off the machining power. Gap pulses are output by the machining axis command.

SAMPLE:

```
M27 G91;                (Machining OFF)
G01 X1.0 F3;            (F3 feed)
G83 Z1.5 E1000;         (Machining feed)
G01 X1.0 F500.0;        (F feed 500 mm/min)
M26 E1100;              (Machining ON)
G01 Z1.5;               (Machining feed)
G98;                    (Return to machining start point)
G84 Z2.0;               (Measurement)
  |
  |
```

4.11.6 SUBPROGRAM FUNCTION (M98/99)

FUNCTION: M98 - Subprogram call
M99 - End of subprogram

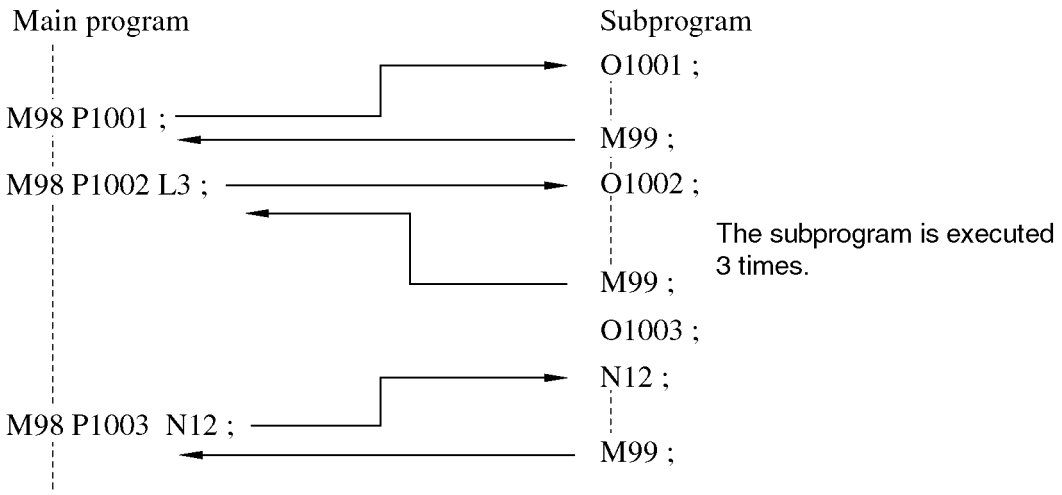
FORMAT: M98 | P/O_ L/R_ N/P_ | ;

P/O: Called program No.
L/R: Calling frequency (When it is omitted, the number of calls is 1.)
N/P: Subprogram start "N" number (When it is omitted, execution is started from the head block.)

M99 | P/N_ L_ | ;

P/N: Program No. to be executed after return to main program (If omitted, execution starts from the next block of called block.)
L: Change of subprogram calling frequency (Normally, omitted)

EXAMPLE OF STATEMENT:



EXPLANATION:

◎ The pattern to be used repeatedly in the program is stored as a subprogram in the memory and can be called by the M function.

◎ Command M98 is used to call the subprogram from the main program. Command M99 indicates the end of the subprogram and returns it to the main program.

Note1: A mixed use of "P_L_N_" combination and "O_R_P_" combination is inhibited.

Note2: It is impossible to use two or more "O_R_P_" for one program.

4.11.7 ELECTRODE CHANGE FUNCTION (M06/07/08/T)

This is an auxiliary function when the electrode changer (ATC)is provided, and its type of change is classified into sequential and random .The 3R electrode changer can also be adopted. For electrode change, the machine must be moved to a position that permits it, but this differs a little with models and options.

When this command is specified in the same block as a travel command, M06 is executed after a movement.

The ATC is executed through the 2nd reference point return (G30P2).

Kind of ATC:

Random ATC	----------	M06T__;	Any electrode selectable
Sequential ATC	-----	M07;	Load magazine electrode to spindle
	----	M06;	Return spindle electrode to magazine and load next electrode after indexing one pot
	----	M08;	Unload spindle electrode to magazine

FUNCTION: Sequential ATC

EXPLANATION:

◎ This is a type of ATC when magazine electrodes are used sequentially.
The electrodes are loaded in the magazine in the order of use.
To set an electrode in the spindle before automatic operation, an empty pot of the magazine must be indexed to store this electrode.
If no electrode is set in the spindle, the first pot to be used must be indexed beforehand.
The head of the program is M07.

SAMPLE:

```
G28 C0 ;                    (If C axis is attached)
G91 G30 P2 X0 Y0 Z0 ;       (ATC position)
M07;
  |
  |
G28 G0;                     (If C axis is attached)
G91 G30 P2 X0 Y0 Z0 ;       (ATC position)
M06;                        (Sequential ATC)
  |
M08;
                            ✻M07 (Loading)
                            ✻M08 (Unloading)
```

FUNCTION: Random ATC

FORMAT: M06 T_ | H1 |;

T: Electrode No. specification (01 - 99)

✻Magazine electrode to be changed (new electrode)

H1: Magazine front index

✻This is effective only in continuous unloading and loading operation.

EXPLANATION:

◎ Magazine pots are numbered. This command is used to optionally use the electrodes in the magazine. Each electrode is stored in the relative pot.
Electrode number (T) and ATC (M06) must be specified in the same block as a rule.

◎ Two digits of T code are used for electrode number specification by random ATC. This T code can be specified in the range of 01 - 99 but this range may be limited by the electrode changer in use.

8-electrode specification …… T01 - T08

16-electrode specification …… T01 - T16

By using the presence/absence of spindle electrode before automatic operation and T00 after automatic operation, electrode loading/unloading can be specified.
(When T00 is used, this is regarded as no electrode specification.)

FUNCTION: Operation from loading when no electrode is in the spindle

EXPLANATION:

◎ When no electrode is in the spindle, the unloading (storing the spindle electrode into the magazine) operation is skipped and only loading (loading the magazine electrode into the spindle) operation is performed.
To perform automatic operation without loading any electrode into the spindle, specify T00; in a single block.

Note 1: In the case of random ATC, if a single block of T is programmed before electrode change, an alarm is caused in the block of M06T_;.

Note 2: Specify T00 in a single block.

SAMPLE:

T00;	(No electrode is loaded in the spindle.)
M06 T03 ;	(Magazine T03 → Spindle loading)
M06 T05 ;	(Spindle T03 → Magazine, Magazine T05 → Spindle)

FUNCTION: Operation in the case where an electrode is provided in the spindle

EXPLANATION:

◎ To perform automatic operation with an electrode loaded in the spindle, it is necessary to index the pot to store the spindle electrode by the next ATC operation.
Specify the spindle electrode storing number at the head of the program (before the next ATC) in a single block by T code.

Note: In the case of random ATC, if a single block of T is not programmed before electrode change an alarm is caused in the block of M06 T_;
(Specify the spindle electrode number in a single block.)

SAMPLE:

```
T05 ;                (Spindle electrode No.)
M06 T03 ;            (Spindle T05 → Magazine, Magazine T03 → Spindle)
M06 T08 ;            (Spindle T08 → Magazine, Magazine T08 → Spindle)
```

FUNCTION: Loading the spindle electrode into the magazine (unloading)

EXPLANATION:

◎ To terminate automatic operation storing the spindle electrode into the magazine, it is possible to perform only the unloading operation. (M06 T00 ;)
No loading (loading the magazine electrode into the spindle is performed.)

SAMPLE:

```
T01 ;                (Spindle electrode No.)
M06 T03 ;            (Spindle T01 → Magazine, Magazine T03 → Spindle)
M06 T05 ;            (Spindle T03 → Magazine, Magazine T05 → Spindle)
M06 T00 ;            (Spindle T05 → Magazine)
M02;
```

FUNCTION: 3R electrode changer

EXPLANATION:

◎ Apart from our ATC, System 3R ATC can be added.
The 3R ATC permits only random ATC. Specify the electrode number (T) and electrode change (M06) in the same block.
In performing automatic operation, where an electrode is provided in the spindle or not, the 3R ATC can identify this state automatically by the sensors provided in the spindle and pot.

Where no electrode is provided in the spindle…… Magazine electrode → Spindle (loading)
Where an electrode is provided in the spindle…… Spindle electrode → Magazine (unloading)
Magazine electrode → Spindle (loading)
Loading the spindle electrode into the magazine…Spindle electrode → Magazine (unloading)

Note 1: To perform automatic operation with an electrode in the spindle, an empty pot of the magazine must be indexed beforehand.

Note 2: If ATC is performed with an electrode in the spindle when the indexed magazine is not an empty pot, an alarm is caused.
On the contrary, an alarm will result if no electrode is present in the spindle and the magazine.

Note 3: In loading an electrode from the magazine to the spindle, if it is not found in the magazine, an alarm is caused.

Note 4: When the C-axis provided, the ATC position of the X-axis is +100 mm apart from the zero point in order to avoid interference with the C-axis upon rotation of the arm.

SAMPLE 1:

```
M06 T03;   (Spindle electrode → Magazine, Magazine T01 → Spindle)
M06 T05;   (Spindle T01 → Magazine, Magazine T05 → Spindle)
M06 T00 ;  (Spindle T05 → Magazine)
M02;
```

SAMPLE 2:

```
G28 C0;                 (C-axis reference point)
G91 G30 P2 X0 Y0 Z0;    (X, Y, Z-axes 2nd reference point)
M06 T03;                (ATC)
   |
```

FUNCTION: EROWA robot electrode changer

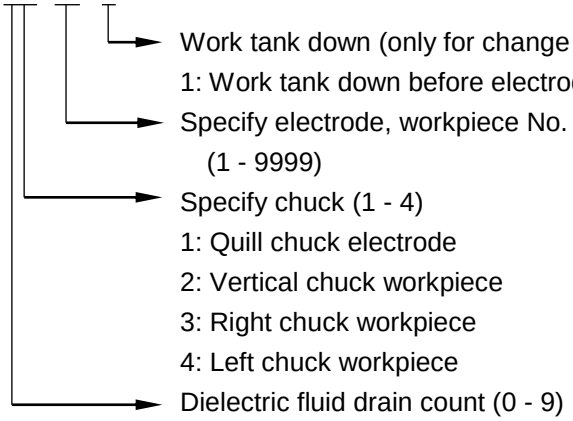
EXPLANATION:

This section describes how to make a program for electrode and workpiece change by using the EROWA robot.

FORMAT:

① Change of electrode and workpiece in magazine and chuck

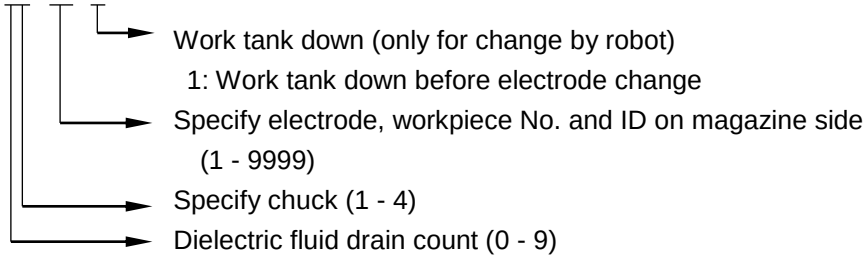
M6 Lxx Txx Dx;



Note: No electrode nor workpiece must be loaded in the chuck at the start of automatic operation.

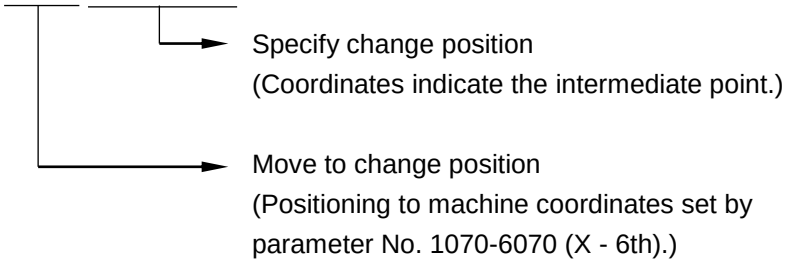
② Change of electrode and workpiece from chuck to magazine (unloading)

M8 Lxx Txx Dx ;



③ Specify to change positioning of electrode and workpiece.

G30 P2 X_Y_Z_ (C)_ ;



SAMPLE)

```
G91 G30 P2 X0 Y0 Z0;        (Move to change position)
M6 L1 T5;                   (Load electrode No.5 from magazine to quill chuck)
    |  (MC program)
    |
G91 G30 P2 X0 Y0 Z0;        (Move to change position)
M6 L1 T10;                  (Unload electrode No.5 from quill chuck to magazine,
    |  (MC program)          and load electrode No.10 from magazine to quill chuck)
    |
G91 G30 P2 X0 Y0 Z0;        (Move to change position)
M8 L1 T10;                  (Unload electrode No.10 from quill chuck to magazine)
M02;
```

4.11.8 MA/MR HEAD FUNCTION (M57/56/03/05)

The spindle can be used for various uses such as C-axis function, indexing function and rotation function.

	Switching code	Command
C-axis function	M57	C_
Indexing function	M56	C_
Rotation function	M03/M05	S_

Upon power- on, the switching code is M57. Be sure to use the MA/MR head after the execution of zero point return.

When the indexing function and rotation function are switched over to the C-axis function to disconnect the C-axis motor from the C-axis unit and then the C-axis motor is connected with the C-axis unit, there will arise some error. Just after switching the other function to the C-axis function, be sure to perform zero point return. (This is required whenever ATC is executed.)

FUNCTION: C-axis function

FORMAT: M57 | C_ |;

EXPLANATION:

◎ Specify this command if the MA/MR head is used as C-axis. After switching the other functions (indexing and rotation functions) to the C-axis function, be sure to perform "zero point return".

The least input increment is 0.001 degree / 0.0001 revolution for standard resolution, or 0.0001 degree / 0.00001 revolution for high resolution. Select appropriately according to the application.

When this command is specified in the same block as a travel command, M57 is executed before a movement.

Note: ① M05 status → M57 ; is OK.

② M03 status → M57 ; causes the warning. Execute M05; once.

SAMPLE: (REV specification)

```
G91 M26 E1000;                  (Machining ON)
M57;                            (C-axis function selection)
G28 L1 C0;                      (C-axis zero point return)
G01 G91 Z-20.000 C-20.0000;     (1-mm-pitch 20-mm-depth right screw machining)
G98;                            (Machining start point return)
  |
```

FUNCTION: Indexing function

FORMAT: M56 | C_ |;

EXPLANATION:

◎ This command is specified to use the MA/MR head as the indexing function. Indexing is performed by the C-axis motor. Then, the MA/MR head is disconnected from the indexing motor and clamped.
As the clamp position is divided into 24 parts in units of 15 degrees, the indexing angle is specified by an integral multiple of 15 degrees.

◎ The indexing rate is operated by rapid traverse. Machining feed is not performed even in the machining ON mode.
Absolute/incremental specification is validated. For switching to the indexing function, it is recommend to switch the C-axis function from the zero point.

◎ Specify M56 | C_ | in a single block. If another travel command is found, an alarm is caused.

Note: The execution of ATC needs the indexing angle of 0 degree or C-axis zero point return.

SAMPLE:

```
M57 G91;              (C-axis function)
G28 L1 C0;            (C-axis zero point return)
M56 C45.000;          (Indexing 45°)
C0;                   (Indexing 0°)
```

FUNCTION: Rotation function

FORMAT: M03/05 | S_ | ;

M03: Forward rotation

M05: Stop

S: Rotation speed (00 - 15)

EXPLANATION:

◎ This command is specified to use the MA/MR head as the rotation function. Rotation are provided by disconnecting the C-axis motor from the C-axis unit and directly connecting the C-axis motor with the electrode holder.

◎ It is recommended to switch to the rotation function from the C-axis zero point.
The rotation function needs that the rotational speed should be set in the same block beforehand.
When this command is specified in the same block as a travel command, this rotation command is executed before movement.

MA head standard parameter value

S code	Rotation speed	S code	Rotation speed	S code	Rotation speed
00	10	06	125	12	480
01	40	07	160	13	630
02	50	08	200	14	800
03	63	09	250	15	1000
04	80	10	315		
05	100	11	400		

Note: For ATC, rotations must be stopped by M05, and the C-axis must be return to the zero point after the rotation function is switched to the C-axis function by M57.

SAMPLE:

```
M57 G91;              (C-axis function)
G28 L1 C0;            (C-axis zero point return)
M26;                  (Machining ON)
M03 S10 G01 Z-1.0;    (Rotation ON)
```

4.11.9 DH HEAD FUNCTION (M18)

This is added to perform only the indexing function without using the MA head. The number of planes to be indexed is 8 (in units of 45 degrees).For ATC, indexing must be performed on one plane (0 degree).

When this command is specified in the same block as a travel command, indexing is performed before a movement.

FUNCTION; DH (8-plane indexing)

FORMAT: M18 C_;

C 0 , C1: 1 plane (0°)

C 45.0, C2: 2 plane (45°)

| |

C270.0, C7: 7 plane (270°)

C315.0, C8: 8 plane (315°)

EXPLANATION:

◎ Plane specification is made by C1 - C8.
Degree specification is directly made by C0, C45.0, - C315.0.

◎ When plane specification is made exceeding 9 planes or 360.0 degrees, an alarm is caused.
If a smaller angle than the current angle is specified by plane or degree specification, indexing is performed by backward rotation.

4.11.10 BLOWING FUNCTION (M09/40 - 49/50/72/73)

This is used to perform machining by blowing the dielectric fluid to the workpiece.
In the blowing unit, there are three types such as (1) electrode blow valve for ATC machine, (2) additional valve unit for a set of 10 valves, and (3) auto variable pressure type blowing unit. The dielectric fluid blowing is turned on and off with the miscellaneous function codes [M] .When this command is specified in the same block as a travel command, it is executed before a movement. Intermittent blowing or alternate blowing can also be specified.

FUNCTION: Electrode blow valve
FORMAT:

M50: Blowing ON
M09: Blowing OFF

FUNCTION: Ten-valve unit

FORMAT: M_ | L_P_ | ;

M40: Valve 1 blowing ON

M41: Valve 2 blowing ON

⋮

M48: Valve 9 blowing ON

M49: Valve 10 blowing ON

M09: All valve blowing OFF (included electrode blowing)

L1: Intermittent blowing

L0: Continuous blowing

P: Valve switching time

EXPLANATION:

◎ A maximum of 3 valves can be used simultaneously for blowing by parameter setting. If the 3rd or 4th valve is specified by parameter setting (No. 0670), the 4th valve backward from it is turned off and the new 3 valves are turned on for blowing.
(① This function is valid in the automatic mode only. Turning on the valves exceeding the value set to the parameter is ignored. ② When 1 is set to the parameter, a new valve is turned on and the previously specified valve is turned off the newest one valve is always turned on for blowing.)

◎ When M09 is executed, all valve are turned off.

Note: The electrode blow valve is turned off, too.

Example:

	Valve 1	Valve 2	Valve 3	Valve 4	Valve 5	Valve 6	Valve 7	Valve 8	Valve 9	Valve 10
M09 ;	×	×	×	×	×	×	×	×	×	×
M40 ;	○	×	×	×	×	×	×	×	×	×
M41 ;	○	○	×	×	×	×	×	×	×	×
M42 ;	○	○	○	×	×	×	×	×	×	×
M43 ;	×	○	○	○	×	×	×	×	×	×
M44 ;	×	×	○	○	○	×	×	×	×	×
M45 ;	×	×	×	○	○	○	×	×	×	×
M46 ;	×	×	×	×	○	○	○	×	×	×
M47 ;	×	×	×	×	×	○	○	○	×	×
M48 ;	×	×	×	×	×	×	○	○	○	×
M49 ;	×	×	×	×	×	×	×	○	○	○

* When up to 3 valves are simultaneously turned on by parameter setting

◎ Multiple valve cannot be turned on by one block.
◎ When multiple valve are used for machining, alternate valve switching may provide higher efficiency in discharging machined sludge through intermittent blowing rather than continuous blowing.
◎ Intermittent blowing is enabled with a model having 10 valve.

Command:

1. Program

M41 L1 P_ ;
M42 L1 P_ ;

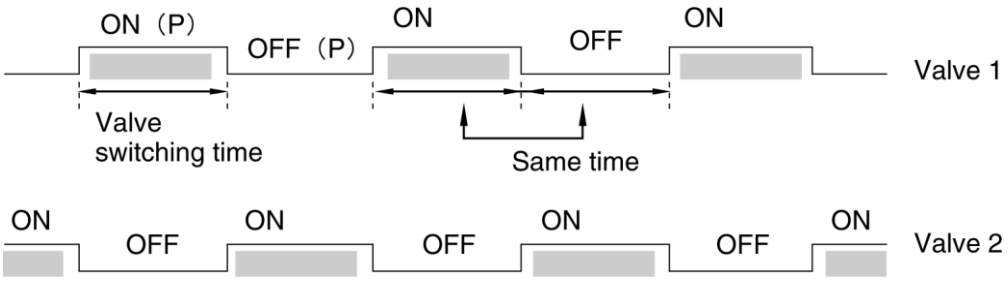
Valve switching time (2.5 - 500 sec)

Omission = P5.0 (ON/OFF at 2.5 sec interval)
1: Intermittent blowing mode
0 or omission: Continuous blowing mode

Valve setting

Note: The P value, specified with value above decimal point, sets the cycle time. (ON time is 1/2 of input value.)

Operation explanation:



FUNCTION: Automatic variable pressure type blowing unit

FORMAT: M72 | L_ |; High pressure blowing ON

L1: High pressure blowing ON, pressure level 1

L2: High pressure blowing ON, pressure level 2

:

L8: High pressure blowing ON, pressure level 8

Default L : High pressure blowing On with the pressure level selected last

M73; High pressure blowing OFF

M50; Low pressure blowing ON

M09; Low pressure blowing OFF

EXPLANATION:

◎ The blowing units added optionally provide dielectric fluid blowing at high pressure (about 10 - 60 kgf/cm^2) or at low pressure (about 0 - 1 kgf/cm^2) from the leading end of pipe electrode. Further, the automatic variable pressure type blowing unit can provide high pressure blowing of up to 8 different pressures set to the parameters. Also, high pressure blowing and low pressure blowing operate exclusively according to the specification.

4.11.11 TANK ELEVATING FUNCTION (M58/59)

FUNCTION: Work tank ascent/descent

FORMAT: M58; Work tank descent
M59; Work tank ascent

EXPLANATION:

◎ This command is used when the automatic tank ascend/descend function is added. This is used in combination with the dielectric fluid supply (M28)/dielectric fluid drain (M29) commands.
If the command is a specified in the same block as a travel command, M58 is executed after a movement and M59 is executed before a movement.

Example:

```
M59;                    (Work tank ascent)
M28;                    (Dielectric fluid ON)
M29;                    (Dielectric fluid OFF)
M58;                    (Work tank descent)
```

4.11.12 PALLET CHANGE FUNCTION (M60)

FUNCTION: Pallet change

EXPLANATION:

◎ This is a start command when the pallet change (workpiece changer) or robot is provided in the outside for loading/unloading workpieces.
When an input is made from the external unit, M60 is completed and the next block is executed.

◎ For loading/unloading workpieces, the 2nd reference point is used at the work change axis position and the tank ascent/descent and automatic dielectric fluid supply/drain commands are used together in order to satisfy the workpiece change condition.

4.11.13 SL FUNCTION (M70/71)

This function specifies the suck quantity and suck ON for the SL unit in the smokeless (SL) machining.

FUNCTION: SL machining ON (setting of large/medium/small suck)
SL machining OFF

FORMAT: M70 L_;····· SL ON (L1: Large, L2: Medium, L3: Small, default = Large)
M71;·····SL OFF

EXAMPLE OF STATEMENT:

```
M70 L1;
G83 E_ Z_;
M71;
```

EXPLANATION:

◎ M70 L_ command makes suction setting and modal display.

◎ M71 command is activated when the RESET key is pressed.

4.11.14 ROTARY AXIS BRAKE FUNCTION (M78/79)

FUNCTION: Rotary axis brake ON
Rotary axis brake OFF

FORMAT: M78; ····· Brake ON
M79; ····· Brake OFF

EXAMPLE OF STATEMENT:

M78;
M79;

EXPLANATION:

◎ In a machining with large electrodes, this function is used to apply a mechanical brake.

◎ The EROWA or 3R rotary axis is furnished with a mechanical brake and this option is applicable to these axes, but not applicable to the MINPULS C axis, MA head and MR head.

4.12 MACHINING CONDITIONS (E/G10)

The machining conditions can be roughly classified into the following to conditions:

① Standard registered machining conditions

② User-registered machining conditions

Standard registered machining conditions

These machining conditions have been registered internally, and the user cannot change the registered data. (Those that are in execution can be changed temporarily.)

User-registered machining conditions ..... 1000 sets (100 groups) max.

These can be freely registered, cancelled or changed by the user.

1) Ten machining conditions (low-order 1 digit: ***0 - ***9) comprise one group.

2) One machining condition consist of machining condition elements such as τON and τOFF.

FUNCTION: Machining condition E number call

FORMAT: E****; (0001 - 9999)

EXAMPLE OF STATEMENT:

(1) E****;

(2) G01 E**** Z-10.;

(3) G83 E**** Z-1.23;

EXPLANATION:

◎ This command remains Modal until the next E number call is received.

Note: The machining conditions do not change even when the RESET key is pressed or the power is turned off and on.

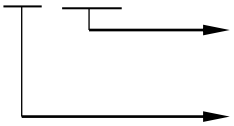
◎ If the specified machining conditions are not valid, an warning (halt state) is caused.

① There is no E number. (No registration)

② E number in the state where machining condition element registration is insufficient.

FUNCTION: Change of the preset value of one machining condition element of execution machining conditions.

FORMAT: G10 L10 R # # ＊ ＊ ＊;



Preset value (Be sure to input a 3haihundigit number without omitting 0.)

Machining condition element number

EXAMPLE OF STATEMENT:

(1) G10 L10 R1016; (τON is set at 16.)

(2) G10 L10 R8001; (The main axis polarity is set to “-”.)

EXPLANATION:

◎ The data are saved same as E number until E number is called (E**** ;) or same element number is changed, even when the RESET key is pressed or the power is turned off and on.

◎ For the machining condition elements, see 3.5.1 DIRECT DATA WRITING (G10).

5. CUSTOM MACRO

5.1 General Description

5.1.1 Features of custom macro

The Custom Macro is a programming language with the “macro command code” for implementing operation functions or judgment functions added to the NC command codes of “Address + Numeric values” of conventional part program.

With the custom macro, a machine builder or user himself can prepare machining program or machine control program requiring complicated calculations or condition judgments.

The custom macro differs greatly from the part program in the following points.

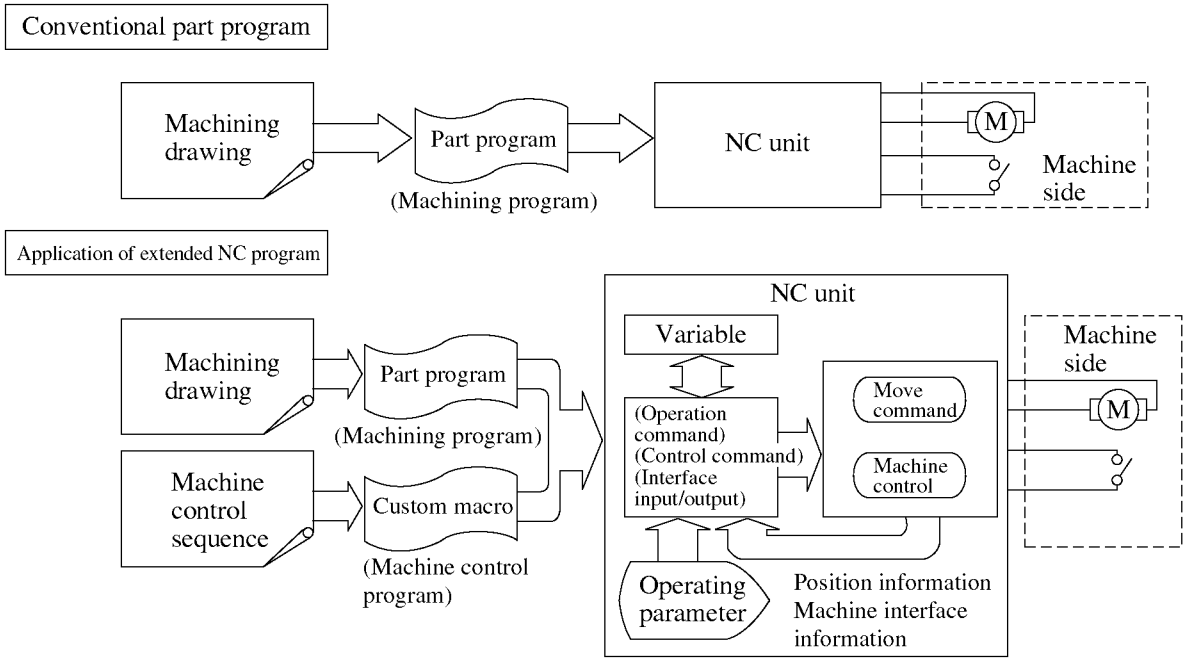
(1) A “variable” can be used for a ward value.

(2) “Calculation” can be made between variables and constants.

(3) The operation parameters of the NC unit and the internal information of the system software can be read as variables.

(4) The “control commands” such as conditional jump or repetition can be used.

(5) A wide variety of subprogram call patterns are available. With these features, program functions can be further enhanced and the program can be applied more extensively.



5.1.2 Custom macro and run mode

In the custom macro, components of all programs can be classified as follows.

(1) NC program
- Main program: Program which starts execution at the run starts.
- Subprogram: Program which starts execution at the run starts. Program which starts execution at the subprogram call command of the main program. The main program can pass any parameter to the subprogram.

(2) Command code
- NC command code: Command code by the address word used for the conventional part programming.
- Macro command code: Command which operates on position information of the NC command block and flow of the NC program.

These components can be used in each run mode of the NC unit under the following limitations.

Table 5.1 Relationship between run mode and custom macro

Component / Rum mode	NC program		Command code	
	Main program	Subprogram	NC command code	Macro command code
Memory run	Possible	Possible	Possible	Possible
MDI run	Possible	Impossible (Note 1)	Possible	Possible

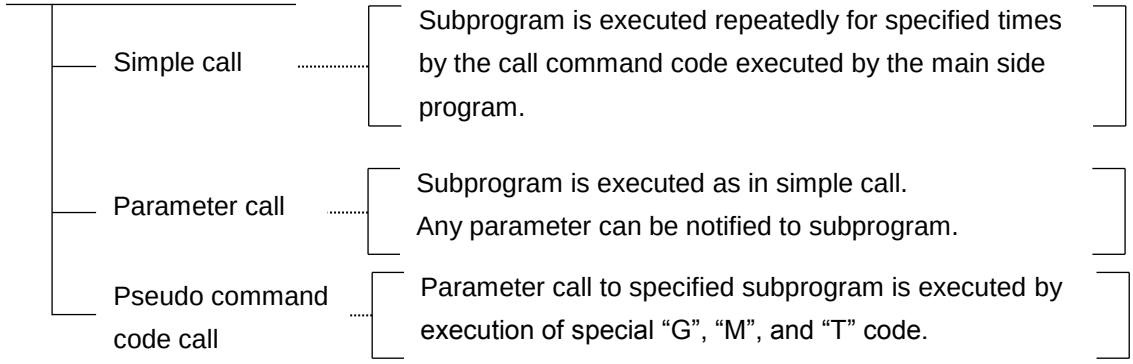
Note 1: The subprogram used in the MDI run must have been registered on the memory by the EDIT screen prior to run start (same as normal part program).

5.2 Subprogram

5.2.1 Types of subprogram calls

There are the following types of the subprogram call patterns which can be used in the custom macro.

Subprogram call patterns

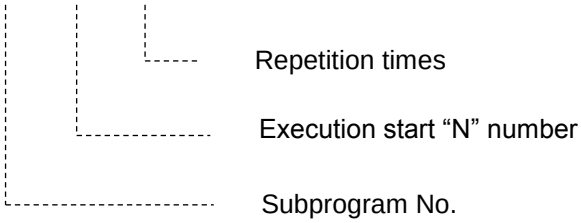


5.2.2 Simple call (M98/G27)

“M98” or “G27” command code is used.

(1) When M98 command is used:

M98 O/P__P/N__R/L__;



Note 1: Mixed use of combination “O_ P_ R_” and combination “P_ N_ L_” is not allowed.
(The wire electric discharge machine is fixed to O_ P_ R_ by parameter 74.)

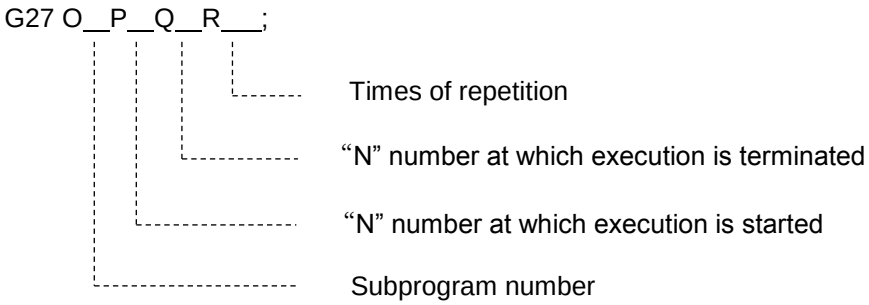
Note 2: Omission of R/L results in single repetition time.

Note 3: Omission of P/N results in execution of the subprogram from its beginning.

Note 4: Commanding with each word value substituted by any macro variable is possible.
This is also true with other subprogram call commands.

Note 5: It is impossible to use two or more “O_R_P_” for one program.

(2) When G27 command is used:



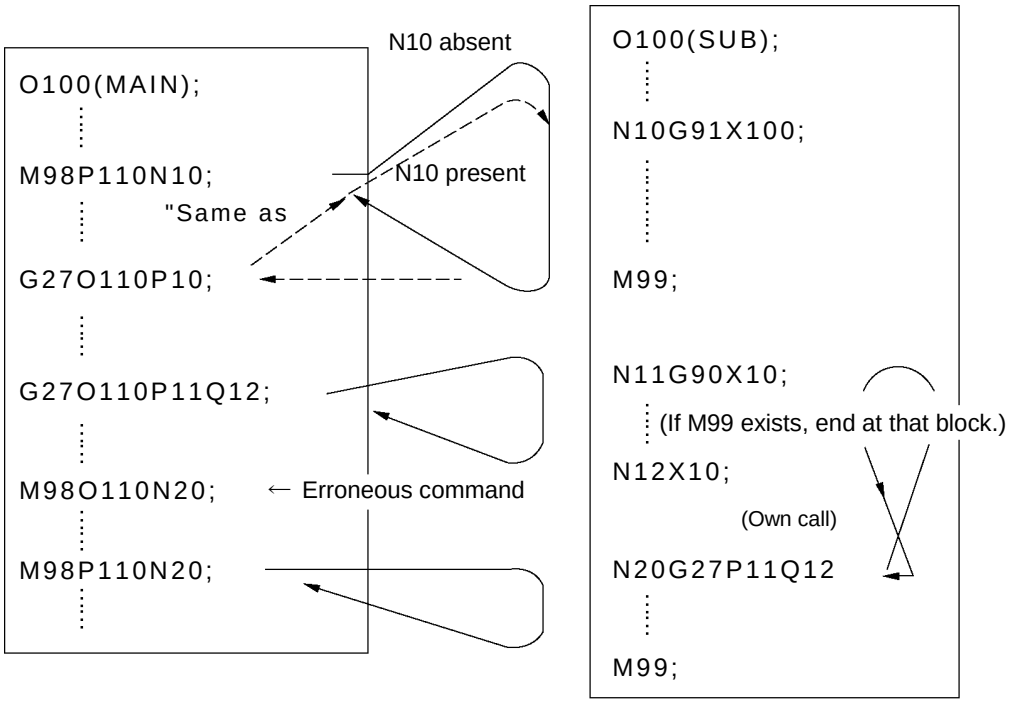
Note 1: Omission of O results in call of calling program itself.

Note 2: Omission of P results in execution of the called program from its beginning.

Note 3: No subprogram call command must appear in the block containing "N" number indicated by Q. Omission of Q results in termination of the sub-program upon execution of "M99" command.

Note 4: Omission of R has the same effect as specifying R1.

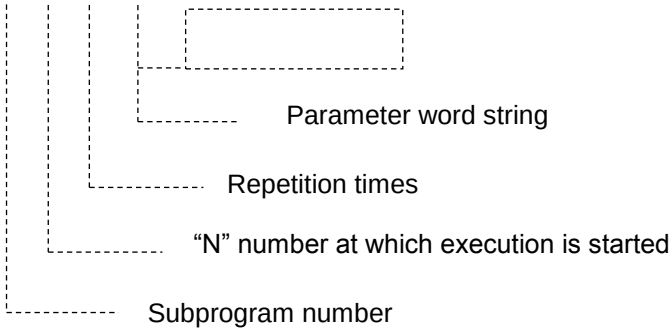
Example: Usage of M98/G27



5.2.3 Parameter call (G65)

"G65" command code is used for this purpose.

G65 P__N__L__X__Y__ ;



Note 1: L may be omitted, in which case the results is the same as specifying L1.

Note 2: N may be omitted, in which case execution is made from the first block of the subprogram.

Note 3: Specify N and L always before parameter word string.

Note 4: In case N or L only is specified for parameter word string without specifying both, it is necessary to add a dummy parameter other than N or L at the beginning of the parameter word string.

Example:

G65 P100 L1; → G65 P100 A0 L1;

Parameter "L" Dummy parameter

5.2.4 Pseudo command code call (Parameter No. 140 - 171)

The pseudo command code call is used to call a subprogram that uses macro command codes having special functions other than the command codes provided by the NC unit, then to execute these command codes in a subprogram. Special “G”, “M”, and “T” codes used to call a subprogram are defined by the parameters.

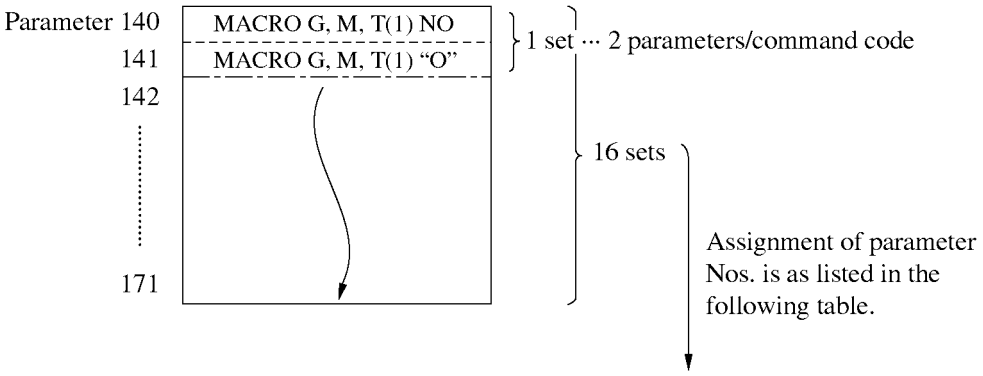
Example 1 of parameter setting

No. 141	20500
No. 140	8100

When parameters are set to values as shown here, subprogram No. “8100” is called by execution of command code “M500”.

(1) Parameter No.

2 parameters per command code must be set for the pseudo command code call. Up to 16 command codes can be set with addresses of “G”, “M” and “T” combined.



Set No.	Parameter No.	Set No.	Parameter No.	Set No.	Parameter No.
1	140 - 141	7	152 - 153	13	164 - 165
2	142 - 143	8	154 - 155	14	166 - 167
3	144 - 145	9	156 - 157	15	168 - 169
4	146 - 147	10	158 - 159	16	170 - 171
5	148 - 149	11	160 - 161		
6	150 - 151	12	162 - 163		

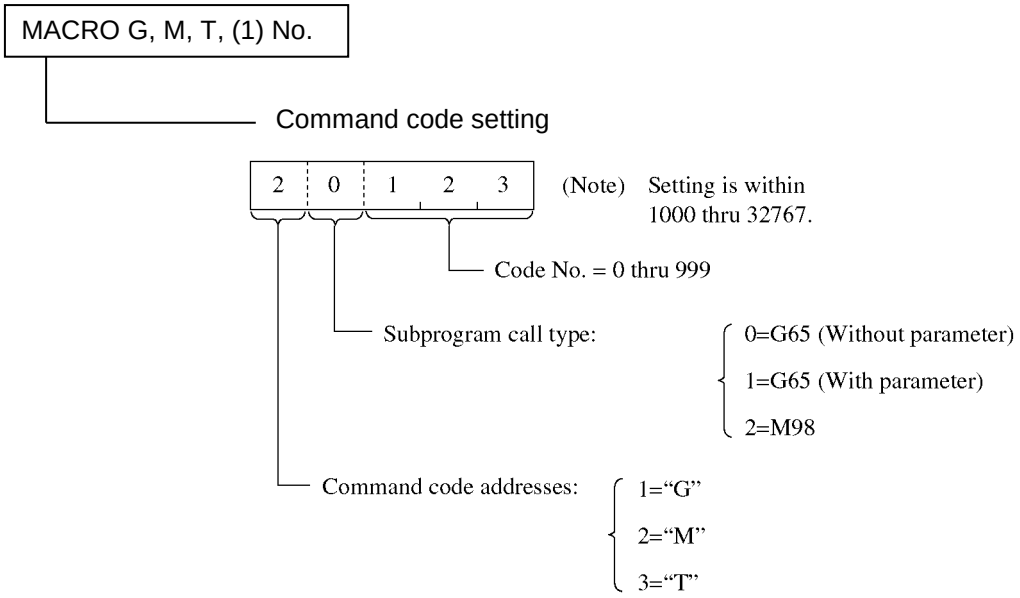
Note 1: Addresses of “G”, “M” and “T” can be specified in any set of these 16 sets.

Note 2: Set the parameters not in use to “0”.

(2) Parameter setting

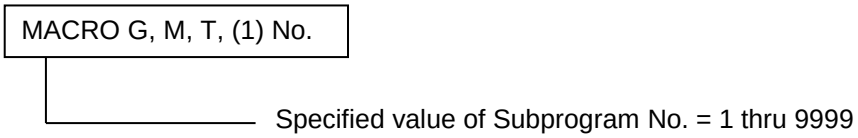
① Specification of command code

Set the code Nos. of “G”, “M” and “T” calling the subprogram and the call types in decimal 5 digits.



② Specification of subprogram No.

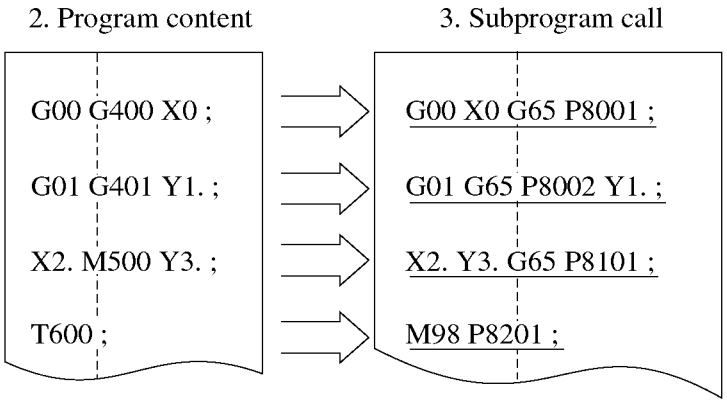
Specify the subprogram No. corresponding to the command code in decimal value.



Example 2 of parameter setting

1. Parameter value

140	10400
141	8001
142	11401
143	8002
144	20500
145	8101
146	32600
147	8201

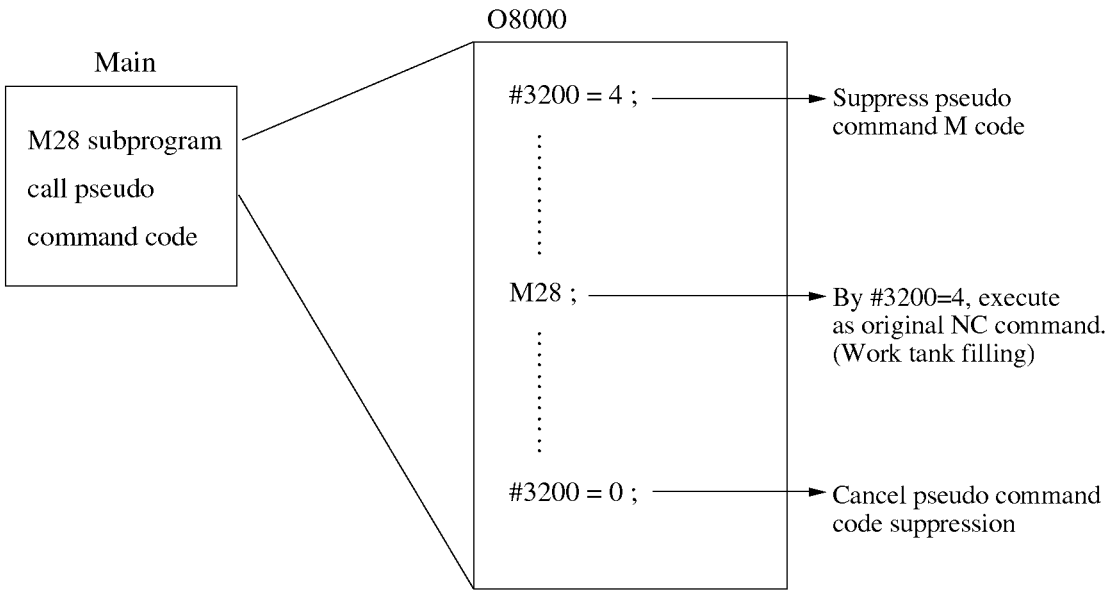


(3) Executing ordinary code in pseudo command code

In a subprogram called by the pseudo command code call, such call by parameter specification can be temporarily suppressed

Example 3 of parameter setting.

No. 141	22028
No. 140	8000



(4) Precautions on use of pseudo command codes

When a pseudo command code is used, attention should be paid to the following.

(a) Blocks containing other subprogram call or pseudo command codes cannot be used.

(b) In case a pseudo command code is used in a block containing other NC command codes, the subprogram is called after completion of operation of all such NC command codes.

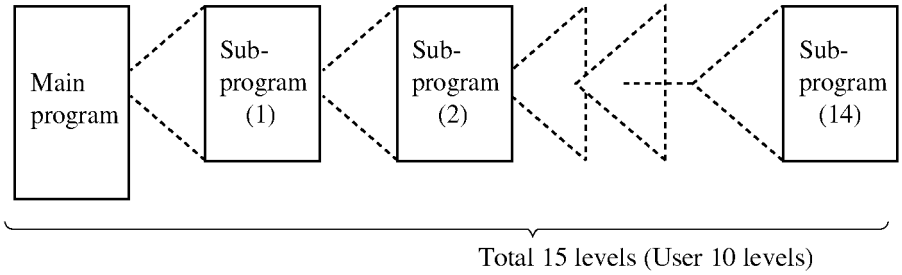
(c) In case same command codes as those provided by NC unit are defined as pseudo command codes, they function as pseudo command codes.

5.2.5 Nesting of subprograms

The subprogram nesting can be counted at two different levels; simple subprogram level and macro level.

(1) Subprogram level

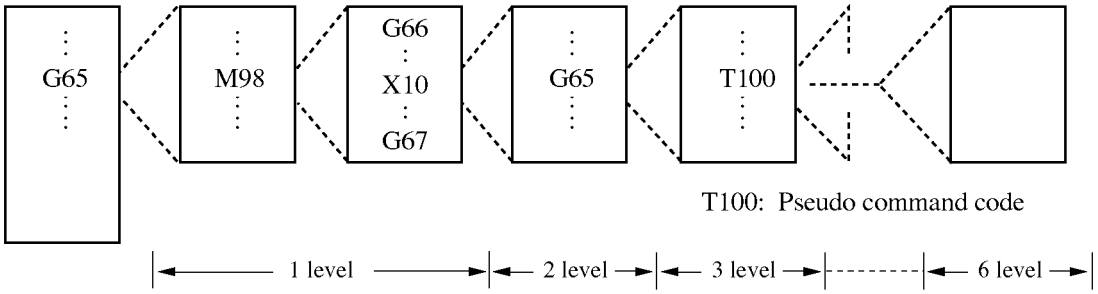
Indicates the nesting level of all subprogram calls by M98, G27, G65, pseudo command code, etc. and up to a total of 15 nesting levels of call are available.



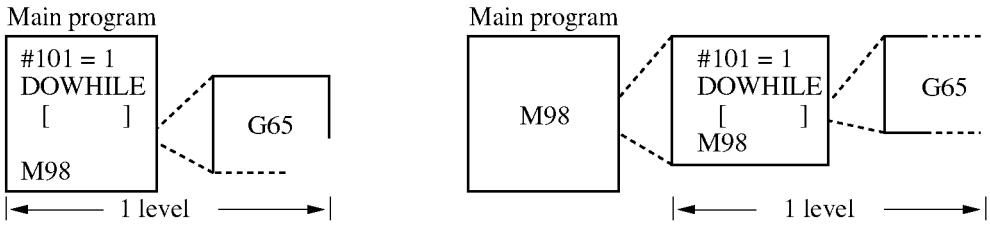
Note: However, the system uses 5 levels, and therefore the user can use up to 10 levels.

(2) Macro level

Indicates the nesting level of subprogram parameter calls by G65, G66, pseudo command code, etc. and up to a total of 6 nesting levels of call are available.



However, when a block containing a macro command code was executed before executing parameter call as shown below, one level is occupied unconditionally.



Note: However, the system uses 2 levels, and therefore the user can use up to 4 levels.

5.3 Parameter and Variable Function

5.3.1 Purpose of variables

In case a subprogram is to be created for general purpose, a function to make the machining scale variable while the workpiece machining shape is fixed is necessary.

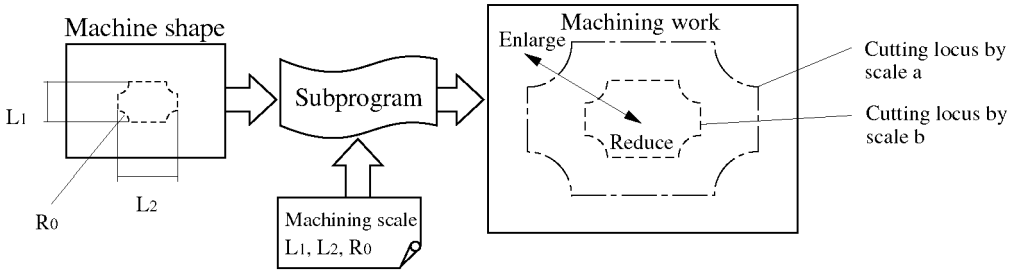


Fig. 5.1 Purpose of subprogram for general use

When such a subprogram is to be implemented, the axis movement amount is an unknown value that cannot be determined by the subprogram itself among various commands that are output to the EDM. On the other hand, the scale information is known to the main program.
In such a case, the following definitions are applied.

- Machining scale information given by the main program: ····· Parameter
- Container of unknown values received by subprogram: ····· Variable

Variables are implemented by the combination of special character "#" and variable number.
A variable can also be used for variable number itself.

Variable expression #10

Variable number

Indirect expression of variable ····· # (#10)

Variables have also the following applications.

(a) Use in substitution calculation

#10 = #10 + 123	Arithmetic expression

(b) Use in judgment condition

IF#10 = #20 – 1	Conditional expression

(c) Substitution of NC command code

G01X#10Y#20 – 1

Variables can also be classified as shown below according to the initialization timing of value, effective range, etc.

Table 5.2 Types of variables

Nomenclature	Q'ty	Effective range	Initialization update	Application
Local variable	33	Within macro level	Subprogram call	1. For input of macro parameter 2. For internal operation
Common variable (non-holding type)	Max. 400	Within operation, mode, cleared upon reset. Other specifications available only for automatic operation system.		1. For internal operation 2. For communication between subprograms
Common variable (holding type)	Max. 900 + 50	Common to all operation modes	All memory clear (at maintenance work)	1. For string machine characteristics 2. For custom macro parameter
System variable	Not open *1	Common to all operation modes	Updated at time of power ON, with operation or with screen operation.	1. System reference and control 2. Reference to operating state 3. Machine control 4. Reference to and update of operation parameters.

*1: The system variables are not open to the users to prevent system malfunction. Please contact us if you want to use them.

Table 5.3 Applications of Common Variables and Nesting Levels

Appli-cation	Manual run system	Model/ edit system	Machine control system				Auto run system *1			Graphic system
			ICON	MSEQ1	MSEQ2	MSEQ3	MEM	MDI	SAUT	
Subprogram nest level (M98)	0 | 5	0 | 9	0 | 9	0 | 9	0 | 9	0 | 9	0 | 14 *2	0 | 14 *2	0 | 14 *2	0 | 14 *2
	---	---	---	---	---	---				
Macro nest level (G65)	0 | 2	0 | 3	0 | 3	0 | 3	0 | 3	0 | 3	0 | 5 *2	0 | 5 *2	0 | 5 *2	0 | 5 *2
	---	---	---	---	---	---				
Non- hold type common variables	---	#100 | #149	#100 | #149	#100 | #149	#100 | #149	#100 | #149	#100 | #499	#100 | #499	#100 | #499	#100 | #499
		---	---	---	---	---				
Hold type common variables	*3 #500 - #999 (real number) #9550 - #9599 (real number) #9600 - #9999 (real number)									#500 - #999 (real number) #9550 - #9599 (real number) #9600 - #9999 (real number)

*1: The MDI and MEM in automatic run system are used for user program execution.

*2: Up to 10 levels of subprograms or 4 levels of macro programs can be used for user programs.

*3: Hold type common variables are common to all run systems. For variables opened to the user, see Table 5.4

Table 5.4 Variable Number List (Auto run system)

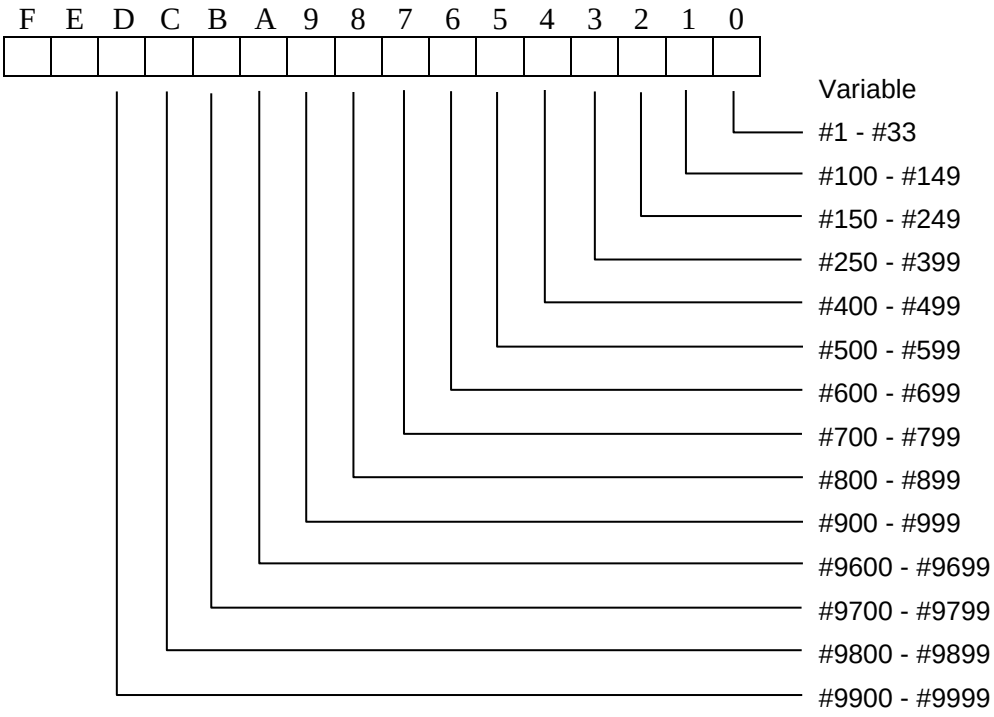
Name	Variable number	Type	Q'ty	Class	Remarks
Local variables	#0 (Reversed value for "empty variable") #1 - #33	A	33	Both user and system	Double accuracy real number type
Non-holding type common variables	#100 - #149	A	50	System	Double accuracy real number type
	#150 - #249	B	100		Single accuracy
	#250 - #399	B	150		
	#400 - #499	B	100	User	
Holding type common variables	#500 - #599	A	100	System	Double accuracy real number type
	#600 - #699	B	100		Single accuracy
	#700 - #799	B	100		
	#800 - #899	B	100		
	#900 - #999	B	100	User	
	#9550 - #9599	A	50	System	Double accuracy real number type
	#9600 - #9699	B	100	System	Single accuracy
	#9700 - #9799	B	100	System	Single accuracy
	#9800 - #9899	B	100	System	Single accuracy
	#9900 - #9999	B	100	System	Single accuracy

Note 1: Never use the variables without permission, except variables with "User" in the Class column to prevent system malfunction.

Table 5.5 Types of Variables

Type	Bit length	Variable expression	Range	Accuracy (mantissa)
A	64	Real number	$4.19\times10^{-307} \leqq \leqq 1.67\times10^{308}$	Significant 16 digits
B	32	Real number	$8.43\times10^{-37} \leqq \leqq 3.37\times10^{38}$	Significant 7 digits
C	16	Integer	–32766 $\leqq \leqq$ 32767	±32766

Note 2: Variables for the system are normally masked, and therefore cancel the masking by parameter No. 133, as necessary.



Setting of each bit 0: Displayed 1: Not displayed

5.3.2 Variable accuracy and constant value (VARMAP)

(1) Variable accuracy

Local variables, non-holding type common variables and holding type common variables of type A have accuracy high enough for normal calculation. The absolute value range of the variables is as shown below.

$4.19 \times 10^{-307} \leqq 1 \times 1 \leqq 1.67 \times 10^{308}$

Note that the accuracy is reduced to significant 7 digits if non-holding type and holding type common variables of type B are used.

$8.43 \times 10^{-37} \leqq 1 \times 1 \leqq 3.37 \times 10^{38}$

Change of variable type

The type B common variables listed in Table 4.4 can be used as double accuracy real number type variables same as type A through the instruction shown below. However, type A and C variables cannot be changed to other type.

Format: VARMAP (a, b, c);

a: Start variable No. for type change (100 - 999)

b: Number of variables for type change (1 - n)

c: Type attribute to be changed

(0: Single accuracy real number 1: Double accuracy real number)

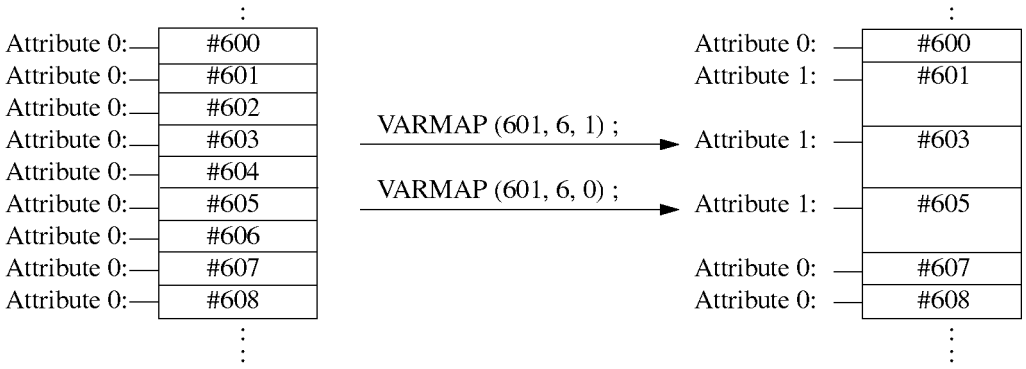
If 1 is specified for type attribute, variables can be used as double accuracy real number.

To return to single accuracy real number, specify 0.

Variables of which type is changed are initialized to original type in the same timing as initialization of variables.

Macro variable number and total number of variables

If type B variables are changed to double accuracy real number type by the VARMAP instruction, continuous two variables are merged into one variable, and as a result, the next variable of changed variable cannot be used and total number of variables is reduced by one.



(2) Contents values

The value is within decimal 8 digits and the digits below decimal are within 4 digits.

The value for NC command codes must be within the limit specified for each command but when they are used as parameter words, they can be specified regardless of word address.

Example:

G65 P1 Q2 M5. T1.2 ;

5.3.3 Characteristics of empty variables (#0)

The local variables and non-holding type variables become “empty” upon initialization. This “empty” state changes to a state other than “empty” when a variable parameter is passed or the calculation result is substituted. A variable in empty state is called an empty variable, which has many features unlike a variable with value.

(1) Judgment of empty variable

“#0” variable is a special variable that has the same value as an empty variable.

Whether a variable is an empty variable is judged by comparison with “#0”.

Example: Conditional expression

IF#1 = #0 …… Empty?”

IF#1 < > #0 …… “Other than empty?”

Note 1:Comparison of largeness of values of empty variables is meaningless.

(2) Calculation of empty variables

Empty variables are equivalent to constant “0” in an arithmetic expression.

Example: Substitution expression

#5 = 5✻ #1…… #1 = when empty, #5 = 0

Example: Conditional expression

IF (#1✻5) = 0…… #1 = when empty, result is “true”.

IF (#1✻5) = #0…… #1 = when empty, result is “false”.

IF #1 = 0…… #1 = when empty, result is “false”.

(3) Setting of empty variable

A variable substituted with “#0” returns to an empty variable.

Example: Substitution expression

#5 = #0…… #5 = Empty variable

(4) Replacement of NC command code

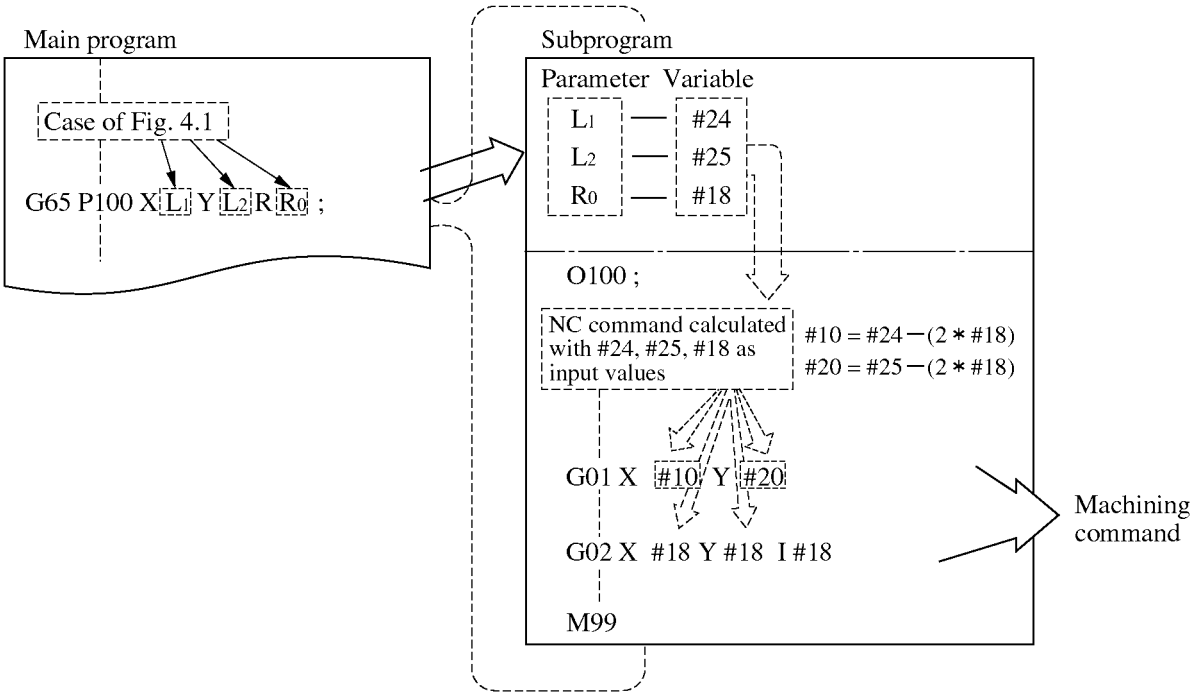
In case the replacement variable of an NC command code is “empty”, the word itself is ignored.

Example: Replacement of NC command code

X#5Y#5✻1…… #5 = when empty, “Y0” only is output.

5.3.4 Parameters and local variables (#1 - #33)

The local variables are initialized by RESET operation and by subprogram parameter call. When the latter is made, the value of the parameter word specified by the command code is stored in the local variable corresponding to each word address.



The correspondence between parameter word address and local variable number is as shown in Table 4.6

In case the constant value of a parameter word is an integer, it is converted to a real value considering the decimal point position determined for each word address before storage in the local variable. The correspondence between word address and decimal point position is shown in Table 4.7 and 4.8

The storage of local variables is processed according to the following conversion equation.

$$\text{(Value of local variable)} = \frac{\text{(Value of parameter word)}}{10^{n}}$$

n: Decimal point position indicated in Table 5.7 and 5.8.

Table 5.6 Parameter word vs. local variable

# No.	Model	
	I	II
#0	Empty	Empty
1	A	A
2	B	B
3	C	C
4	I	I_1
5	J	J_1
6	K	K_1
7	D	I_2
8	E	J_2
9	F	K_2
10		I_3
11	H	J_3
12	L	K_3
13	M	I_4
14	N	J_4
15	O	K_4
16	P	I_5
17	Q	J_5
18	R	K_5
19	S	I_6
20	T	J_6
21	U	K_6
22	V	I_7
23	W	J_7
24	X	K_7
25	Y	I_8
26	Z	J_8
27		K_8
28		I_9
29		J_9
30		K_9
31		I_{10}
32		J_{10}
33		K_{10}

Table 5.7 Word address and decimal point position (Standard resolution)

Word address	mm input	inch input
A, B, C	3	4
D, H, E, F	0	0
I, J, K	3	4
L, M, N, O, S	0	0
P, Q, R	3	4
T	0	0
U, V, W	3	4
X, Y, Z	3	4

Table 5.8 Word address and decimal point position (High resolution)

Word address	mm input	inch input
A, B, C	4	5
D, H, E, F	0	0
I, J, K	4	5
L, M, N, O, S	0	0
P, Q, R	4	5
T	0	0
U, V, W	4	5
X, Y, Z	4	5

Note 1: Parameter word models I and II can be used as mixed.

Note 2: In case variable Nos. are overlapped as a result of mixed parameter word models I and II, the latter parameter word becomes effective.

5.3.5 NC command code substitution

In case the numeric part of NC command code is replaced by a variable, the processing that is exactly opposite to the parameter word processing shown in item 5.3.4 is made. When variable replacement is made, the real number stored in each variables is converted to an integer with decimal point position specified for destination word address taken into account. Then, the function as the NC command code is made. The decimal point position information is in accordance with Table 5.7 and 5.8. The fraction part which is eliminated by conversion to an integer is rounded at the 10th digit.

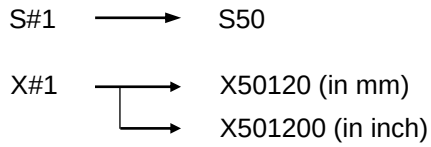
The conversion equation for replacement is as follows:

$$\text{(Replaced value)} = \text{(Variable value)} \times 10^{n}$$

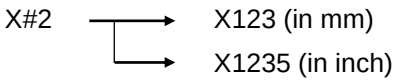
n: Decimal point position indicated in Table 4.7 and 4.8

Some examples of replacement are shown below.

Example 1. When #1 = 50.12



Example 2. When #2 = 0.12345



5.4 Operation Function

In macro command codes, operation can be made freely with variables and constants (numerical values).

(1) Application of operation

The variable becomes is used as it is in the operation. The general form of each is shown below.

a. Substitution

[Variable] = [Expression] Example #10 = #20 + 1 ;

b. Judgment condition

[Function] [Expression] [Relative operator]

Example IF#10 < #20 + 1 ;

c. NC code replacement

[Address] [Expression] Example G01X#10 – 1Y … ;

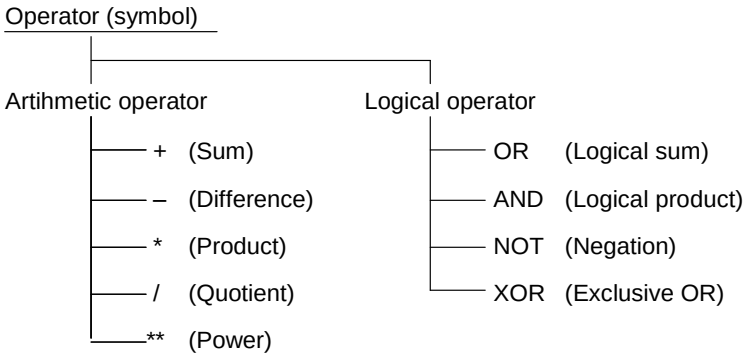
Note: Definition of “expression”

{ Expression [Variable (constant)] [Operator] [Variable (constant value)] [Operator …}

“Repetition”

(2) Kinds of operators

The following operators can be used in an expression excluding relative operator.



(3) Use of function

Functions can be used as variable elements of an expression. In this case, the variable elements are defined as follows.

Variable ... [Function name] ([Expression])

Example) #20 = #10 + SIN (#30 * 4)

There are following functions.

Function name

SIN	(Sine, degree)	ABS	(Absolute)	
COS	(Cosine, degree)	BIN	(BCD □□BIN)	
TAN	(Tangent, degree)	BCD	(BIN □□BCD)	
ATAN	(Arctangent, deg.)	ROUND	(Round off)	
SQRT	(Square root)	FIX	(Fix down)	
		FUP	(Fix up)	
		UMUL	(H reso. multiplication)	(Note 1)
		UDIV	(H reso. division)	(Note 2)
		CMIL	(Converter accuracy)	(Note 3)
		UNAM	(Angle resolution)	(Note 4)

Note 1: The magnification of UMUL varies with the setting of Set Data No.63 bit2 (input resolution of command).

Set Data No.63 bit2 = 0 (Standard resolution)
Metric : ×1000
Inch : ×10000

Set Data No.63 bit2 = 1 (High resolution)
Metric : ×10000
Inch : ×100000

Note 2: The magnification of UDIV varies with the setting of Set Data No.63 bit2 (input resolution of command).

Set Data No.63 bit2 = 0 (Standard resolution)
Metric : $\frac{1}{\times 1000}$
Inch : $\frac{1}{\times 10000}$

Set Data No.63 bit2 = 1 (High resolution)
Metric : $\frac{1}{\times 10000}$
Inch : $\frac{1}{\times 100000}$

Note 3: The CMIL is a function to convert the unit from metric to inch when the magnification in metric mode set in program (Set Data No.63 bit0 = 0) is changed to inch mode (Set Data No.63 bit0 = 1).
Do not use G917 (converter accuracy command) when using the CMIL.
The use of CMIL in the program in which G917 is used causes an error in inch conversion.

Note 4: The magnification of UNAM varies without the setting of Set Data No.63 bit2 (input resolution of command).

Set Data No.63 bit2 = 0 (Standard resolution)
Metric : ×1
Inch : $\frac{1}{\times 10}$

Set Data No.63 bit2 = 1 (High resolution)
Metric : ×1
Inch : $\frac{1}{\times 10}$

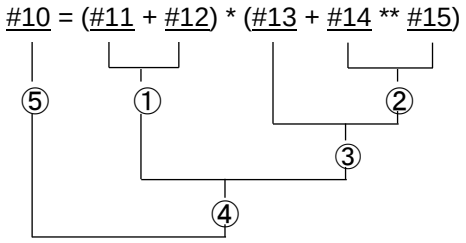
(4) Operation order

To adjust the operation order, parentheses: (–-) can be used.
The priority of operators including parentheses are as shown below.

High
Order
Low

- Expression in parentheses·····(Expression)
- Power ·····#10 ** 2
- Multiplication/division ·····#10 * 2
- Addition/subtraction ····· #10 - 2

The operation sequence is as follows.
#10 = (#11 + #12) * (#13 + #14 ** #15)



5.5 Judgment and Branch Functions (IF, ELSE, GOTO)

Execution of blocks in specific range or change of execution of next block to other specific block according to the result of variable value check can be made.

(1) Structure of judgment and branch commands

Two or more blocks are used for judgment and branch function. The general form is shown below.

```
① IF (Condition expression) ;
- - - - - - - - - - - - - - - - - - - -
② [ Command code ] ;
- - - - - - - - - - - - - - - - - - - -
③ ELSE ;
- - - - - - - - - - - - - - - - - - - -
④ [ Command code ] ;
- - - - - - - - - - - - - - - - - - - -
```

Blocks ① and ② essential

Blocks ③ and ④ used only when necessary

Function

If the judgment result of block ① is “true”, block ② is selected and if it is “false”, blocks ③ and ④ are selected.

(2) Multiple branch function

```
IF < Conditional expression 1 > ;
    [Selective statement string 1] ;
ELSEIF < Conditional expression 2 > ;
    [Selective statement string 2] ;
ELSEIF < Conditional expression 3 > ;
    [Selective statement string 3] ;
              :
              :
ELSEIF < Conditional expression n > ;
    [Selective statement string n] ;
ELSE;
    [Selective statement string n + 1] ;
```

- •Any number of branches may be used.
- •Conditional expressions are evaluated in the ascending order.
- •Compound conditional expression is acceptable
- •Processing if neither of conditions 1 to n is not satisfied.
- •This ELSE statement is omissible.

(3) Conditional expression

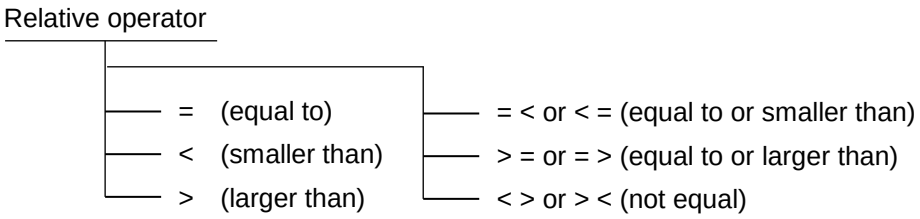
The general form of conditional expression is shown below.

[Expression] [Relative operator] [Expression]

Example 1. IF#10 < #20-1

Example 2. IF (#10AND255) > #20

There are the following kinds of relative operators.



Compound condition judgment can also be made. The general forms are shown below.

(Conditional expression) [Logical operator] (Conditional expression)

[Logical operator] ([–])

• For conditional expression negation

[ ] NOT (Conditional expression n)

Example: IF (NOT (#10 = #20) AND (#10 < > #30))

• A compound conditional expression may be used.

(Compound conditional expression) [Logical operator] ([–])

Example: IF ((#10 = #20) AND (#10 = #30)) OR (#2 = #3)

• There are the following kinds of logical operators.

AND ····· Logical product, OR ····· Logical sum

XOR ····· Exclusive OR

(4) Command code

Any command code permitted in the program can be used in the command code to be selected (and naturally, judgment branch is also possible).

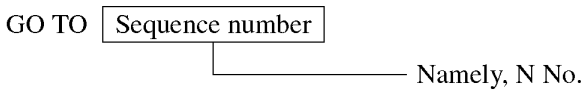
After selection, the command codes in brackets are continuously executed.

```
IF#10 = #20 ;                Macro command code
   [#10 = #20 + 1] ;         NC command code
ELSE ;
   [#10 = #10 + 2 ;          } Command codes may be in multiple blocks.
   G01X#10Y#20] ;
```

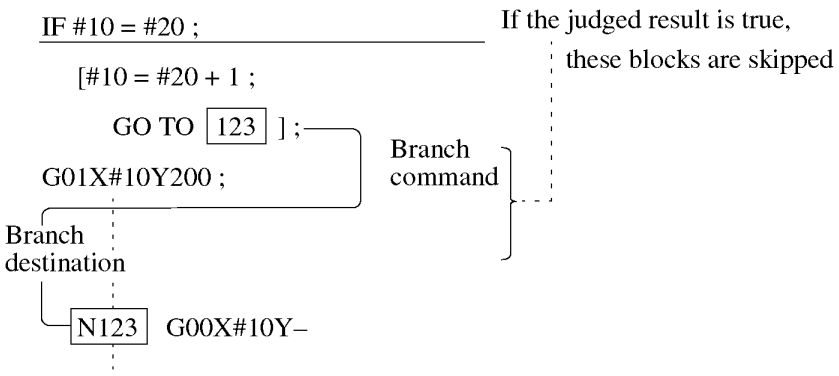
Brackets may be located anywhere.

(5) Branch command code

If the “branch command code GOTO” is specified at the end of the command code to be selected, the block to be next executed can be changed to any block. The general form of the branch command code is shown below.



Example:



5.6 Repetition Function (DOWHILE)

Repeatedly executes the block in the brackets while the judgment of the conditional expression is “true”. The general form is shown below.

DOWHILE (Condition expression) ;
[Command code] ;

The conditional expression and the command code are same as the judgment branch function.

5.7 Suppress Command of External Control Function

In addition to the macro command codes described in paras. 5.3 thru 5.6, suppress command by the G code is possible.

	Command G code	
	Suppress	Release
Single block	G945	G944
Pretreatment suppression	G911	G910

5.8 Registration of Custom Macro

5.8.1 Preparation of program tape

Create a program in the same manner as normal part programs. However, use macro command code special symbols in the program.

Special symbols for macro command code

#	&	’	*	,	<	=	>	?	@	”	[	]

5.8.2 Registration of program

To register an extended NC program containing special characters described above into memory, the “parameter 25” must be set.

The values of parameters are as follows.

Setting of Parameter No. 25

(Binary notation)

7	6	5	4	3	2	1	0

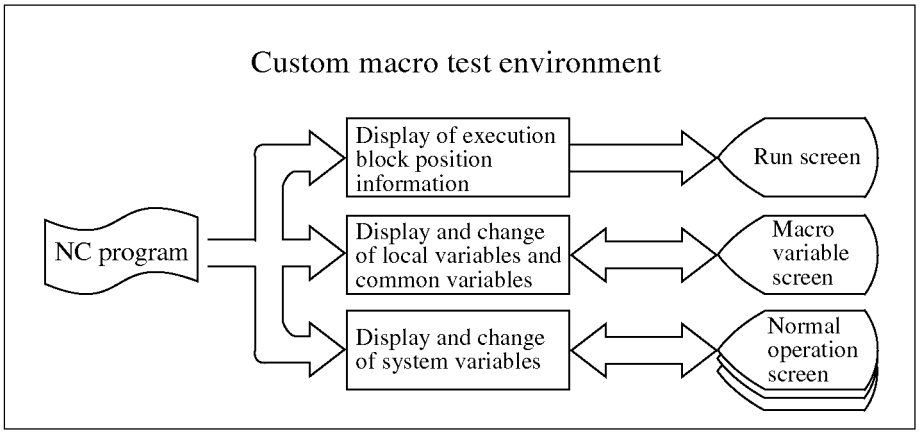
When registering macro command codes, set this bit to “1”.
(It has been already set on shipment.)

5.8.3 Edit of program

Editing of the program is same as the normal editing operation.

5.8.4 Custom Macro Run

The completed custom macro can be operated in the same manner as normal NC programs. However, in testing an incomplete custom macro or using special functions, the execution processes of macro command codes and changes of variables must be checked in addition to a checking of conventional NC command codes. In testing a program containing macro command codes, conduct the test in the environment as shown below:



5.9 System Variable

5.9.1 Offset (#2000-)

These variables are for reading and setting the offset amount to be used in radius compensation.

(1) Variable number

	Sinker EDM
# 2000	Offset No. 0
# 2001	Offset No. 1
:	:
:	:
# 2099	Offset No. 99

(2) Data format

Reading and setting are made by real number values with consideration paid to the machine command unit system set by parameters.

mm unit system	+99999.999 to -99999.999

(3) Precautions on use

#2001 to #2099 are available for use in the case of the sinker EDM.

For the sinker EDM that does not use the offset, this can be used as the hold type variable.

5.9.2 Workpiece offset (#2501-)

Used for reading and setting of the offset amounts of axes that have been set to workpiece coordinate system parameters.

(1) Variable number

Table 5.9 Workpiece offset variable numbers

Offset No.	Workpiece offset variable number				
	X axis	Y axis	3rd axis	–	6th axis
External offset	#2501	#2502	#2503	–	#2506
G54 offset	#2541	#2542	#2543	–	#2546
G55 offset	#2551	#2552	#2553	–	#2556
G56 offset	#2561	#2562	#2563	–	#2566
: :					
G61 offset	#2611	#2612	#2613	–	#2616

(2) Extended work offset variables (sinker EDM)

These variables are used to read and set the work offset values associated with arbitrary “group No.” and “work No.” of 80 pairs of work offsets.

#146 = 0 . . . Designates group No. + Work No.

#147 = Designates group No. (1 - 10)

#148 = Designates work No. (1-8)

Table 5.10 Extended work offset variable numbers

Offset No.	X axis	Y axis	3rd axis	–	6th axis
Extended offset	#2691	#2692	#2693	–	#2696

(3) Data format

Same as the offset.

(4) Precautions on use

As in the case of offset, the work offset must be set again if the offset value changed by substitution of this variable is to be made valid.

5.9.3 System offset (#5501-) (Do not use)

Electrode offset has been used in MGC, MGE, MGF, MGH generator. However these offset are not available in ES100A / ES200A generator.

Thus, to read and set the electrode offset amount, please use # 5891 ~ in 5.9.4.

ES100A/ES200A generator : These variables can't be used.
MGC,MGE,MGF,MGH generator : Electrode offset

5.9.4 Electrode offset (#5891-)

This function reads and sets the electrode offset amount.

(1) Variable numbers

#147	Electrode offset number
#148	Electrode offset access setting
#5891	Electrode offset amount of the 1st axis (X axis)
#5892	Electrode offset amount of the 2nd axis (Y axis)
#5893	Electrode offset amount of the 3rd axis (Z axis)
#5894	Electrode offset amount of the 4th axis (4th axis)
#5895	Electrode offset amount of the 5th axis (5th axis)
#5896	Electrode offset amount of the 6th axis (6th axis)

(2) Data format

#147 : Electrode offset number (1 - 200)
#148 : Electrode offset access setting (specify 0)
#5891- #5896 : Same as offset

< Example >

Current machine coordinates (X - Z) are set to the electrode offset No. 10 (X - Z).

```
#147 = 10;
#148 = 0;
# 5891 = #5101;
# 5892 = #5102;
# 5893 = #5103;
    :
    :
```