CNC EDM
POWER SUPPLY CODE INSTRUCTION

DIMON BEIJING CNC TECHNOLOGY CO., LTD.

2017

TABLE OF CONTENTS

PREFACE

Note the following in reading this instruction manual.

1. This instruction manual has been prepared with the intention of covering everything about codes for use in NC programs to operate the power supply unit. However, there are too many items of "what is not possible" nod "what must not be done" to be covered on account of space consideration. Therefore, what is not specifically described as "possible" in this manual should be interpreted as "impossible".
2. As you read the instruction manual in sequence, you may find words and codes that have yet to be explained. When you find such words and codes. We suggest that you first read through the manual in a general way, skipping over these words and codes, and then read it over again to better understand the words and codes that you skipped over in the first reading.

CHAPTER 1. OUTLINE

1) Word

The EDM power supply unit is programmed by combining many various command sentences, each of which is created by combining many various words.

These words, which provide preparatory function, feed function, auxiliary function and other functions, are structured as shown in the following;

Word= Address + [Code / Date]

2) Address

The address is represented by an alphabetical letter (A to Z) to specify the meaning of the code and data following the letter.

The addresses which can be used by the dies inking EDM and their meanings are as follows.

Address	Meanings
N, O	Sequence No.
G	Preparatory function
X,Y,Z,U,V,W	Dimension, angle, etc. of axis travel
I, J, K	Circular arc center coordinate (Incremental coordinate system)
T	Items related to machine control
D, H	Offset amount/value
P	Sub-program No.
L	Number of repetitions of sub-program
C	Machining conditions
M	Auxiliary function
Q	File call-up
RI	Coordinate of center of pattern rotation and cordinate rotation
RJ	Coordinate of pattern and coordiante rotation center
RX	Input of figure and coordinate rotation angle (X axis)
RY	Input of figure and coordinate rotation angle (Y axis)
RA	Input of figure and coordinate rotation angle
R	Specification of corner "R" radius
B	Change of individual machining conditions
CRT, IF, PRINT, KEYIN, JUMP	User Macro Function

3) Code and Data

The code and data input formats are as shown below.

3)-1 N, O (Sequence No.)

Input the sequence No. in four digits.

[Example] N0001, 00002

Input of the sequence No. as N1 or N2 causes an error to occur when a sub-program is called up.

The sequence No. can be specified in the command range of 10,000 numbers from 0000 to 9999.

3)-2 G (Specification of preparatory function such as linear or circular interpolation)

Input the preparatory function in three digits or less. However, when the code number is of two digits or less (00 to 99), the preparatory function can be input in two digits or less for its decoding without causing any error.

[Example] G054, G001 (G54, G1)

3)-3 X, Y, Z, U, V, W, UU, VV (Specification of coordinate travel)

The coordinate travel can be specified in the data range of +/-999999.999mm and +/-99999.9999 inch ("DIGIT" in SETTING ~ OPERATION sub mode= OFF) or +/-99999.9999

mm and +/-9999.99999 inch ("DIGIT" in SETTING-OPERATION sub mode = 1).

Input of a decimal point is also possible for the specification of the coordinate travel.

[Example] 15.0 → l5mm, 15 → 15µm

("DIGIT" in SETTING-OPERATION sub mode = OFF)

3)-4 I, J, K (Specification of circular arc center coordinate)

The circular arc center coordinate can be specified in data range of +/-999999.999 mm and +/-99999.9999 inch ("DIGIT" in SETTING-OPERATION sub mode = OFF) or +/-99999.9999mm and+/-9999.99999 inch ("DIGIT" in SETTING .OPERATION

Sub mode = 1).

Input of a decimal point is also possible for the specification of the circular arc center coordinate.

3)-5 T (Specification of items related to machine control)

Specify the items related to machine control by input in two digits or less.

[Example] T82, T83 (DRAIN ON/OFF)

3)-6 P (Specification of sub-program No.)

Specify the sub-program No. in four digits as in the case of N and O.

3)-7 L (Specification of number of repetitions of sub-program)

The number of repetitions of a sub-program can be specified in the range of 1 to 99999.

The specification can be made by input of L10, L3 and so on.

Note, however, that the input of 1.0 causes an operation format error to occur.

3 -8 C (Specification of machining conditions)

Specify the machining conditions by input in four digits or less.

[Example] C000, C001

The specified machining condition is first searched for in the private file and then in the CONDITION FILE if it is not found in the private file.

For the machining condition input procedure, refer to "CONDITION FILE" in the section of SYSTEM FILE in a separately available instruction manual "CONTROl. SYSTEM".

3)-9 M (Auxiliary function)

Specify the program proceeding and ON/OFF output to the machine by input in three digits or less.

3)-10 Q (Specification of file call-up)

This function is used for call-up of a program in the floppy disk on a file basis during the machining for execution of the called-up program.

3)-11 F (Specification of feed speed)

Specify the machining response speed (SF).

Input the data in the range of 200 to 400.

(1) The data input ill F represents the feed speed itself.

[Example]When F50 is input, the machining proceeds at- a feed speed of 5u per minute.

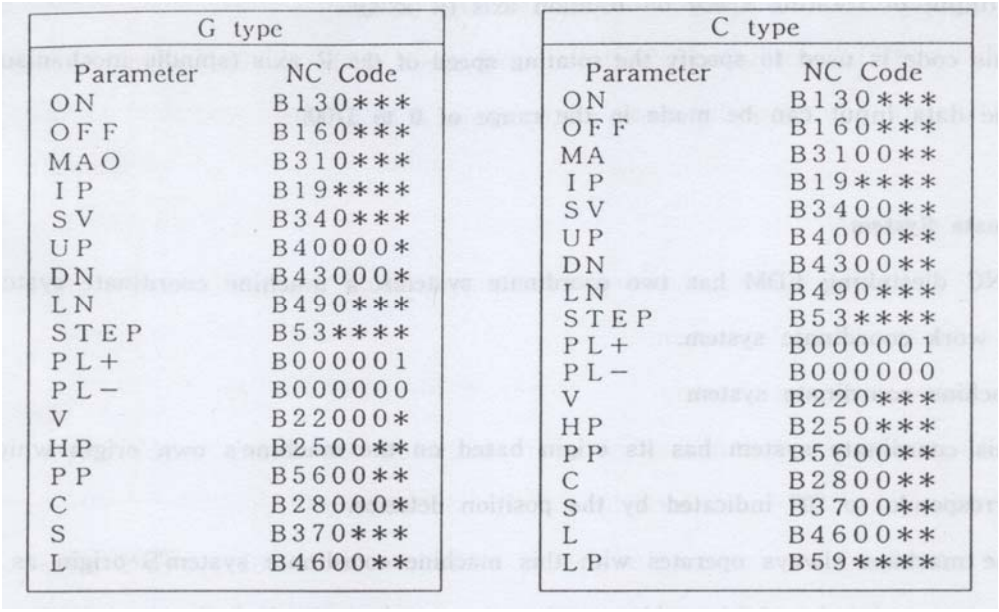
(2) F is not a modal, but only effective only in the block, in which its input has been made.

(3)When the machining conditions are changed during execution of the block in which the F code has been input, the F code data become invalid, while the value set in machining condition SF becomes valid.

3)-12 B (Change of machining condition parameters)

Machining condition parameters now being displayed and outputted (ON, OFF, IP, etc.) can be changed using B code No.

The B codes are as follows:



G type

Parameter	NC Code
ON	B130***
OFF	B160***
MAO	B310***
IP	B19****
SV	B340***
UP	B40000*
DN	B43000*
LN	B490***
STEP	B53****
PL+	B000001
PL−	B000000
V	B22000*
HP	B2500**
PP	B5600**
C	B28000*
S	B370***
L	B4600**

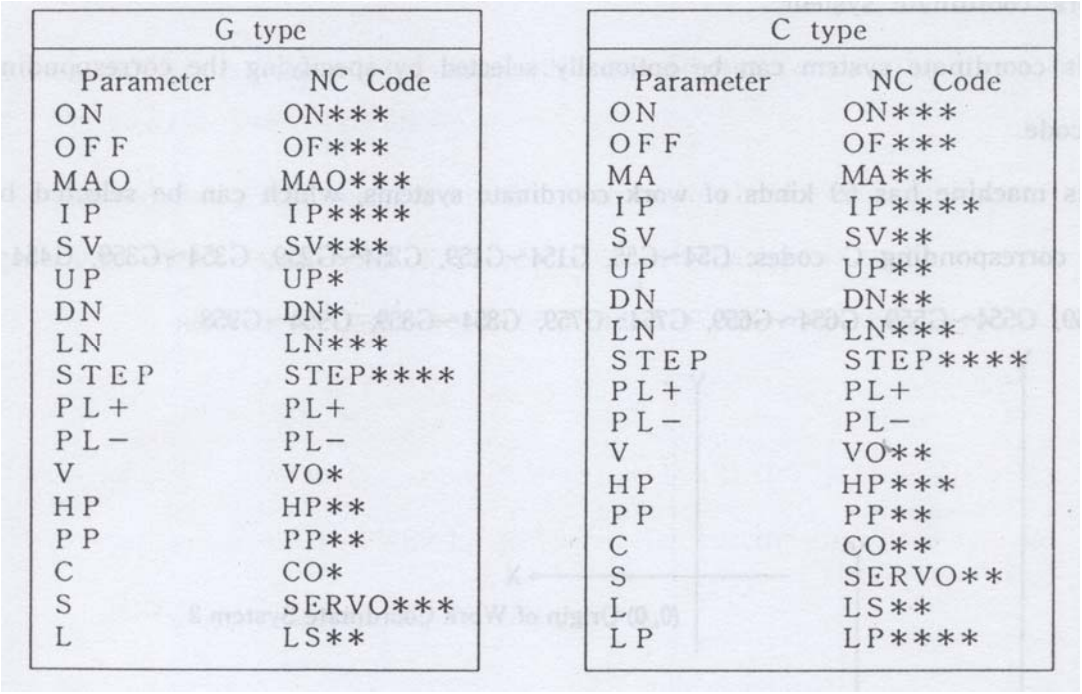
C type

Parameter	NC Code
ON	B130***
OFF	B160***
MA	B3100**
IP	B19****
SV	B3400**
UP	B4000**
DN	B4300**
LN	B490***
STEP	B53****
PL+	B000001
PL−	B000000
V	B220***
HP	B250***
PP	B5600**
C	B2800**
S	B3700**
L	B4600**
LP	B59****

Input data in places marked *.

3)-13 ON, OFF, etc. (change of machining condition parameters)

Machining condition parameters now being displayed and outputted (ON, OFF, IPS, etc.) can be changed using the corresponding machining condition parameter names.



G type

Parameter	NC Code
ON	ON***
OFF	OF***
MAO	MAO***
IP	IP****
SV	SV***
UP	UP*
DN	DN*
LN	LN***
STEP	STEP****
PL+	PL+
PL−	PL−
V	VO*
HP	HP**
PP	PP**
C	CO*
S	SERVO***
L	LS**

C type

Parameter	NC Code
ON	ON***
OFF	OF***
MA	MA**
IP	IP****
SV	SV**
UP	UP**
DN	DN**
LN	LN***
STEP	STEP****
PL+	PL+
PL−	PL−
V	VO**
HP	HP***
PP	PP**
C	CO**
S	SERVO**
L	LS**
LP	LP****

I

Input data in places .marked *.

3)-14 S (Input of rotating speed on rotation axis (R axis))

This code is used to specify the rotating speed of the R axis (spindle mechanism).

The data input can be made in the range of 0 to 1700.

Coordinate System

4) This NC dies inking EDM has two coordinate systems: a machine coordinate system and a work coordinate system.

1. Machine coordinate system

   This coordinate system has its origin based on the machine's own origin which corresponds to "0" indicated by the position detector.

   The machine always operates with this machine coordinate system's origin as a reference point. In addition, this coordinate system is a physical absolute coordinate

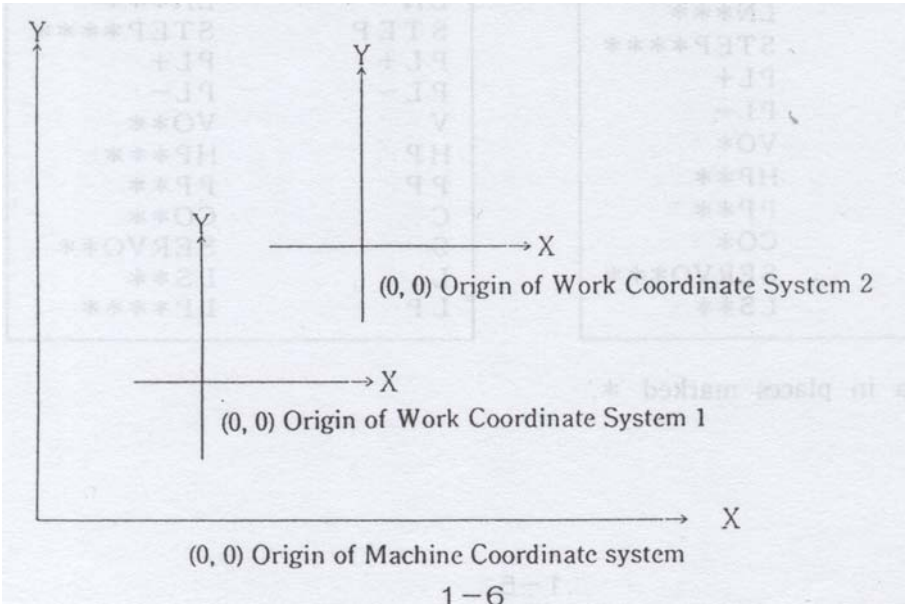
   system, the coordinates of which are not to be negative.

2. Work coordinate system

   This coordinate system can be optionally selected by specifying the corresponding

   G code.

This machine has 59 kinds of work coordinate systems, which can be selected by the corresponding G codes:G54~G55, G154~G159, G254~G259, G354~G359, G454~G459,G554~G559, G654~G659, G754~G759, G854~G8.59, G954~G958.



5) Comment

Any character string enclosed by '(' and ')' is treated as a "Comment", not as a NC program to be executed.

<Example>

```
(MAIN PROGRAM);
G90G92XOY0;
M98 P 0001;
G05                    (X MIRROR IMAGE ON);
M98 P 0001;
G06                    (Y MIRROR  IMAGE  ON);
M98 P 0001;
G09                    ( MIRROR IMAGE CANCEL);
M02;
(SUB PROGRAM)
N0001;
```

```
G01X5.0Y10.0;
    X15.0
G03X 23 . 6602 Y 25 .0118. 66 ,15. 0;
G01XS.0;
    Y10.0;
```

M99 :

Note'('and')'used by Q Command (described in CHAPTER17) or User Macro Function (described in CHAPTER 19) for parameter specification does not constitute such a comment as mentioned above.

CHAPTER 2. BLOCK

Each program is created by several blocks.

Each block consists of more than one word and a character indicating the end of tile block.

Zl Series uses ";" as a character indicating the block end.

1) Rules set for each block

(1) When the X, Y, Z, U, V, W, UU and VV axes are specified in one block, the multiple axes are simultaneously processed by code.

[Example] G91 G00 X7.0 Y5.0 Z10.0

(The X, Y and Z axes simultaneously travel 7mm, 5mm and 10mm, respectively.)

If you wish the three axes to travel in the order of Z, Y and X, specify them in separate blocks as shown in the following example:

[Example] G00 Z10.0;
Y 5.0;
X 7.0;

(2) When two contradictory NC codes exist in one block, an error occurs or the latter code takes priority over the former.

[Example] Case where an error occurs

G00 Xl0.0 G01 Y-10.0

[Example] Case where the latter code takes priority over the former

G00 Xl0.0 Y5.0 Xl5.0

This block provides the same operation as the following block: G00 Xl5.0 Y5.0.

2) Order in which NC codes are processed in one Block

The NC codes are classified as shown in the following table:

Classification	Code
Class A	N. O (Sequence No.)
Class B	A, C, D, F, H, L, P, Q, T, M03~M999, G codes other than those classified as C, H***=*assignment statement IF, JUMP, KEYIN, CRT, PRINT
Class C	G00~G04, G80, G81, G82, G83, G85, G92, G97
Class D	I, J, K, R, U~Z, UU (only G type), VV (only G type), RA, RI, RJ, RX, RY, AY, AZ, BX, BY, BZ, CX, CY, CZ, DP
Class E	(Not in use for diesinking EDM)
Class F	(Not in use for diesinking EDM)
Class G	(Not in use for diesinking EDM)
Class H	M00~M02

1. NC codes in one block are executed in the order of Classes A to F.
2. NC codes of Class H are executed as described below:

   M00.....When this code exists with other codes in one block, it is executed before execution of the other codes.

   M02 ..... Codes before and after this code are not executed.
3. No more than one code of this class in one block can be executed.

   The presence of more than one code of this class in one block causes an error to occur.

4. The presence of any NC code other than specified by Sodick in a block causes an error to occur.
5. G83 causes "XYZUVWT" following this code to be an address for specification of the compensation term.

CHAPTER 3. SEQUENCE NO.

A sequence No. is a number assigned to each block and can be omitted.
The number is represented by a figure of four digits starting with N or O.
Sequence Nos. are conveniently used if they are put in important positions of programs
in increasing order in accordance with the order the programs are executed.
A sequence No. also serves as a mark for call-up of a sub-program.

[Example]

```
N000 …..... Sequence No. (main program)
G90  G54  G92  X-5.  Y10.  Z0
T89
G29
T84
C120
M98  P0010 ..... Call-up sub-program N0010
T85
C220
M98  P0020 ..... Call-up sub-program N0020
 T88
 C320
 M98  P0010
 C420
 M98  P0020
 C510
```

```
M98    P0010
C610


M98 P0020
M02
N0010 .....…Sequence No. (sub-program counterclockwise)
G01    Y0
          X-15.
          Y-30.
          X15.
          Y0
          X5.
M99
N0020 .....    Sequence No. (sub-program clockwise)
G01 Y0
           X15.
           Y-30.
           X-15.
           YO
           X-5
M99
```

CHAPTER 4. OPTIONAL BLOCK SKIP ('/')

If, with "SKIP" of the SETTING mode set in ON state or input of G11, '/' exists at the head of the block in the program to be executed, the specific block delimited by '/' and ";" will not be executed.

Note: *Input '/' at the head of the block.

*X If it is input in the middle of tile block, it is regarded as division sign (÷).

[Example 1]

```
N0000
G90 G54   G92   X0   Y-10.   Z0
G59 G92   X0      Y-10.   Z0
Gl2   .....   SKIP OFF
M98 P0030 ..This causes a block with slash (/) in sub-program N0030 to be executed.
G11.. ..... SKIP ON, which causes the subsequent block with slash (/) not to
            be executed.
M98 P0030..This causes N0030G59, although found, not to be executed, not changing
            the coordinate system to G59 system.
M02 .....   End of program execution
/ N0030G59
    G01 Y0
        X - 15.
        Y - 30.
        X - 15.
        Y0
        X0
        G54
        M99
```

[Example 2]

```
N0000 G92 X0 Y0;
/N0001   G0   G90   Xl0.; .....When N STOP of SETTING ~ FI.AG mode has been
                              set in ON state with N DATA set at 0001. the program
                              execution will not stop at this block
N002 GOG90 Y10.;
```

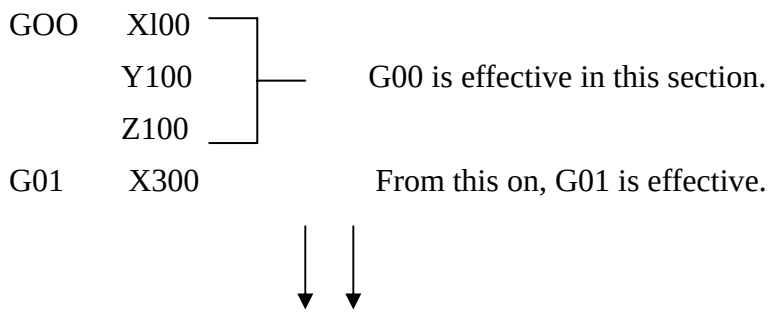
CHAPTER 5. G CODE

G codes are basically classified into two types:

(1)G codes whose functions are limited to blocks to which they are assigned.

(2)G codes whose functions are effective until another G code of the same group appears.(Such G codes are referred to as "Modal Codes".

[Example] Modal codes



The following pages show G codes by groups.

Table 5-1 Setting of Codes by Groups

	G Code	Function		POWER ON	OFF STOP RESET	M02 Execution
✓	G00	Positioning	M	G00	G00	G00
✓	G01	Straight line cut	M			
✓	G02	Circular cut (clockwise)	M			
✓	G03	Circular cut (counterclockwise)	M			
	G04	Dwell				
	G05	X mirror image	M	G09	Note 1	
	G06	Y mirror image	M			
	G07	Z mirror image	M			
	G08	X-Y exchange	M			
	G09	Mirror image and X-Y exchange cancel	M			
	G11	Skip ON	M	G12	Note 1	
	G12	Skip OFF	M			
	G15	U axis origin return				
	G17	XY plane specification	M	G17	G17	
	G18	ZX plane specification	M			
	G19	YZ plane specification	M			
	G20	Inch set ON	M	Note 2	Note 1	
	G21	Inch set OFF	M			
	G22	Software Limit ON	M	G23	No change	G23
	G23	Software Limit OFF	M			
	G24	High speed AJC operation command				
	G25	Ordinary AJC operation command				
	G26	Figure rotation ON	M	G27	G27	G27
	G27	Figure rotation OFF	M			
	G28	Main reference point reset				
	G29	Main reference point setting				
	G30	G92 point return				

	G Code	Function		POWER ON	OFF STOP RESET	M02 Execution
	G48	Edge control ON	M	G49	G49	G49
	G49	Edge control OFF	M			
✓	G54	Work coordinate system 0	M	G54	Note 1	
✓	G55	Work coordinate system 1	M			
✓	G56	Work coordinate system 2	M			
✓	G57	Work coordinate system 3	M			
✓	G58	Work coordinate system 4	M			
✓	G59	Work coordinate system 5	M			
	G60	Main Reference point return				
✓	G80	Travel to ST stop.				
	G81	Travel to machine system limit.				
	G82	Travel half way between current position and origin				
✓	G83	Read current value in assigned compensation term				
✓	G85	Timer machining				
✓	G90	Absolute command	M	Note 2	Note 1	
✓	G91	Incremental command	M			
✓	G92	Coordinate origin setting command				
	G93	X mirror image (Y mirror OFF)	M	G96	Note 1	
	G94	Y mirror image (X mirror OFF)	M			
	G95	X, Y simultaneous mirror image	M			
	G96	Mirror image cancel	M			
	G97	All coordinate system coordinate origin setting command				
	G104	Corner dwell ON	M	G105	G105	G105
	G105	Corner dwell OFF	M			
	G126	Coordinate rotation ON	M		Note 1	
	G127	Coordinate rotation OFF	M			
	G128	1st sub-reference point return				
	G129	1st sub-reference point setting				
	G130	Interference check ON	M	G131	G131	G131
	G131	Interference check OFF	M			

G Code	Function		POWER ON	OFF STOP RESET	M02 Execution
G132	Interference Avoiding ON	M	G133	G133	G133
G133	Interference Avoiding OFF	M			
G136	Interference error ON	M	G137	G137	G137
G137	Interference error OFF	M			
G154	Work coordinate system 10	M	G54	Note 1	
G155	Work coordinate system 11	M			
G156	Work coordinate system 12	M			
G157	Work coordinate system 13	M			
G158	Work coordinate system 14	M			
G159	Work coordinate system 15	M			
G160	Three-dimensional rotation cancel	M	G160	G160	G160
G161	Three-dimensional rotation 1	M			
G162	Three-dimensional rotation 2	M			
G163	Three-dimensional rotation 3	M			
G164	Three-dimensional rotation 4	M			
G165	Three-dimensional rotation 5	M			
G166	Three-dimensional rotation 6	M			
G228	2nd sub-reference point return				
G229	2nd sub-reference point setting				
G254	Work coordinate system 20	M	G54	Note 1	
G255	Work coordinate system 21	M			
G256	Work coordinate system 22	M			
G257	Work coordinate system 23	M			
G258	Work coordinate system 24	M			
G259	Work coordinate system 25	M			
G328	3rd sub-reference point return				
G329	3rd sub-reference point setting				

M : Modal code

	G Code	Function		POWER ON	OFF STOP RESET	M02 Execution
	G354	Work coordinate system 30	M	G54	Note 1	
	G355	Work coordinate system 31	M			
	G356	Work coordinate system 32	M			
	G357	Work coordinate system 33	M			
	G358	Work coordinate system 34	M			
	G359	Work coordinate system 35	M			
	G428	4th sub-reference point return				
	G429	4th sub-reference point setting				
	G454	Work coordinate system 40	M	G54	Note 1	
	G455	Work coordinate system 41	M			
	G456	Work coordinate system 42	M			
	G457	Work coordinate system 43	M			
	G458	Work coordinate system 44	M			
	G459	Work coordinate system 45	M			
	G528	5th sub-reference point return				
	G529	5th sub-reference point setting				
	G554	Work coordinate system 50	M	G54	Note 1	
	G555	Work coordinate system 51	M			
	G556	Work coordinate system 52	M			
	G557	Work coordinate system 53	M			
	G558	Work coordinate system 54	M			
	G559	Work coordinate system 55	M			
	G628	6th sub-reference point return				
	G629	6th sub-reference point setting				

G Code	Function		POWER ON	OFF STOP RESET	M02 Execution
G654	Work coordinate system 60	M	G54	Note 1	
G655	Work coordinate system 61	M			
G656	Work coordinate system 62	M			
G657	Work coordinate system 63	M			
G658	Work coordinate system 64	M			
G659	Work coordinate system 65	M			
G728	7th sub-reference point return				
G729	7th sub-reference point setting				
G754	Work coordinate system 70	M	G54	Note 1	
G755	Work coordinate system 71	M			
G756	Work coordinate system 72	M			
G757	Work coordinate system 73	M			
G758	Work coordinate system 74	M			
G759	Work coordinate system 75	M			
G828	8th sub-reference point return				
G829	8th sub-reference point setting				
G854	Work coordinate system 80	M	G54	Note 1	
G855	Work coordinate system 81	M			
G856	Work coordinate system 82	M			
G857	Work coordinate system 83	M			
G858	Work coordinate system 84	M			
G859	Work coordinate system 85	M			
G928	9th sub-reference point return				
G929	9th sub-reference point setting				

	G Code	Function		POWER ON	OFF STOP RESET	M02 Execution
	G954	Work coordinate system 90	M	G54	Note 1	
	G955	Work coordinate system 91	M			
	G956	Work coordinate system 92	M			
	G957	Work coordinate system 93	M			
	G958	Work coordinate system 94	M			
	G959	Work coordinate system 95	M			

Note l: The setting is made by Cancel Flag.

See CHAPTER22 "Flags Closely Related to Codes".

Note 2: The setting is achieved by SET-TRAVEL.

1) G00 (positioning)

G00 code commands the specified axis to travel without machining. The command format

is as follows:

All the axes can be specified for simultaneous travel.

G00 {Axis specification} _+ {Data}

[Example] G91 GOO X+100 Y+200~

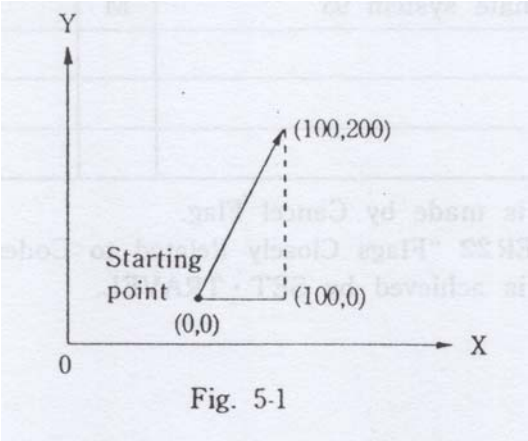


Fig. 5-1

2) G01 (Straight Line Cut)

G01 code commands straight line cutting on the specified axis. The command format is as follows:

All the axes can be specified for simultaneous travel.

G01 {Axis specification} ± {Data}

[Example] G91 G01 X+100 Y+200;

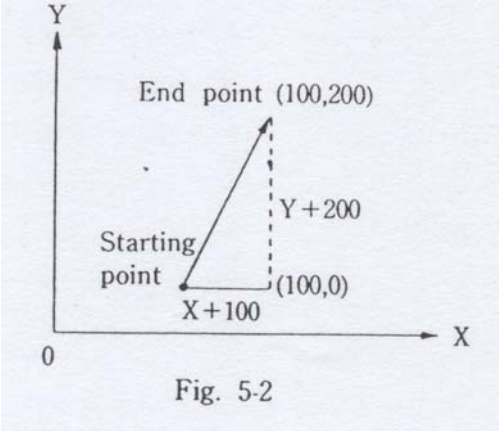


Fig. 5-2

3) G02, G03 (Circular Cut)

(1) Ordinary circular cut

The input format is as follows:

{ G02 / G03 } { Position of end point of circular arc } { Position of center of circular arc relative to its start point }

Circular cut on XY plane (G17)

{ G02 / G03 } { X_____ Y_____ } { I_____ J_____ }

Cricular cut on XZ plane (18)

{ G02 / G03 } { X_____ Z_____ } { I_____ K_____ }

Cricular cut on YZ plane (G19)

{ G02 / G03 } { X_____ Z_____ } { I_____ K_____ }

G02 commands circular cutting in clockwise direction.

G03 commands circular cutting counterclockwise direction.

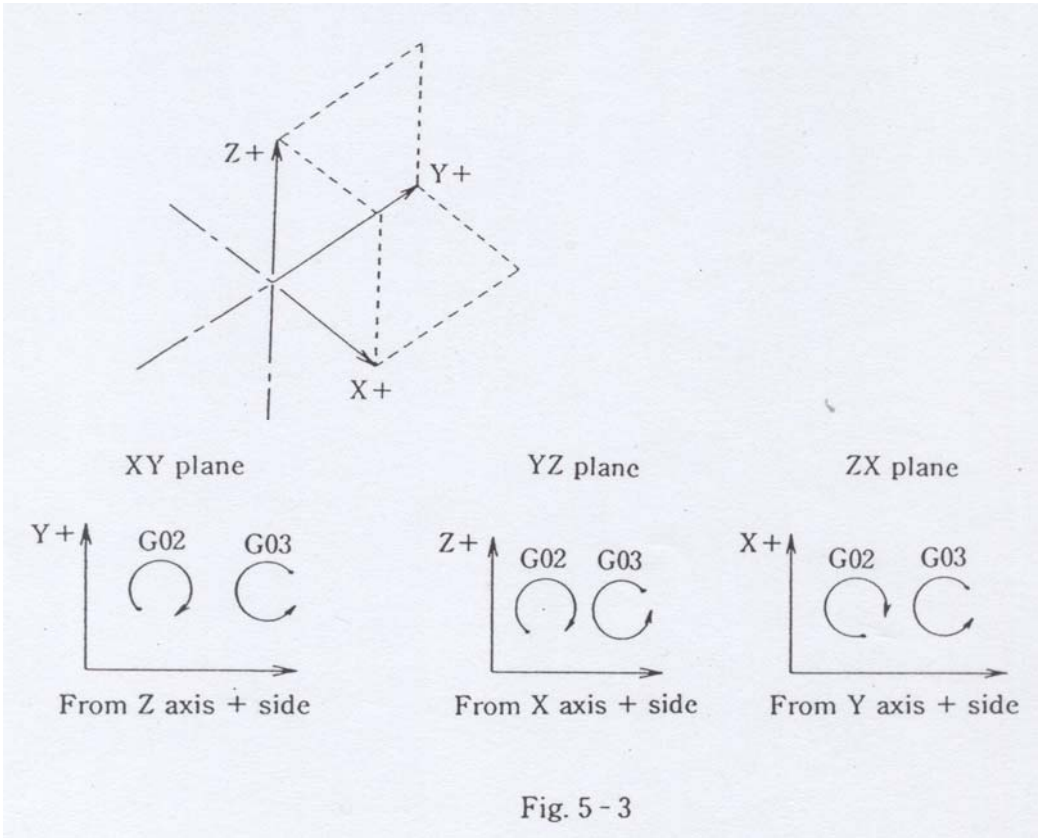


Fig. 5 - 3

A CIRCLE POINT

Upon execution of G02 or G03 for circular cutting, A CIRCLE POINT compares the radii of the circular arc at its start and end points to check whether or not the difference between the two radii is within the value set ill this parameter so that, if the difference is out of the set value, it judges this to be an error to stop the operation, and if the difference is within the setting, it causes the operation to proceed. The unit system used by the parameter is as shown in the following table.

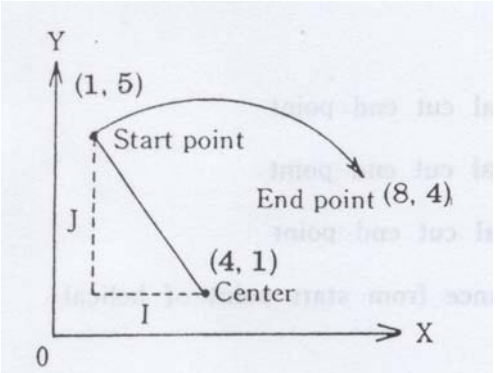
A CIRCLE POINT is set in the SETTING-OPERATION mode, normally at 20, although it can be set in the range of 0 to 999.

Table 5-2

Digit \ Unit system	Metric	Inch
0	0.1 μm	$\frac{0.1}{10000}$ inch
1	0.01 μm	$\frac{0.01}{10000}$ inch

The end point of the circular are is specified by X,Y and Z, while its center is specified by I,J and K, which, respectively, correspond to X, Y and Z.

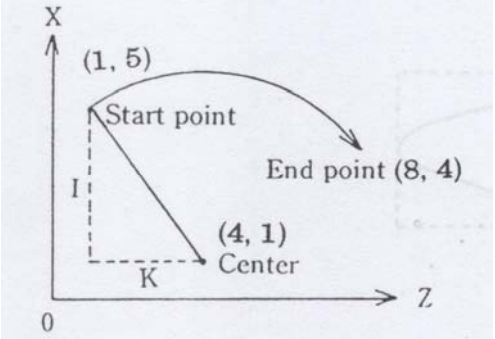
Note: The center of the circular are is given as a relative position viewed from its start point.



G17

G90 G92 X1 Y5

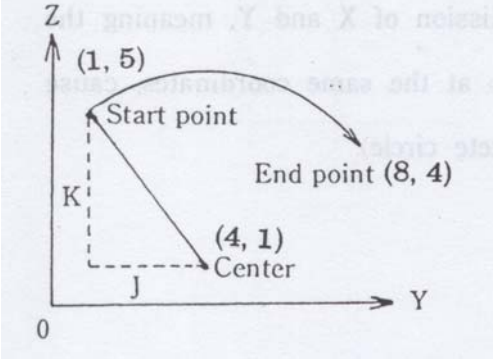
G02 X8 Y4 I3 J-4



G18

G90 G92 Z1 X5

G02 Z8 X4 K3 I-4



G19

G90 G92 Y1 Z5

G02 Y8 Z4 J3 K-4

(2) Helical cut

The "Circular Cut" code (G02, G03) can also be used for helical cutting.

The input format is as follows:

[G02 / G03] X ____Y____I ____J ____Z

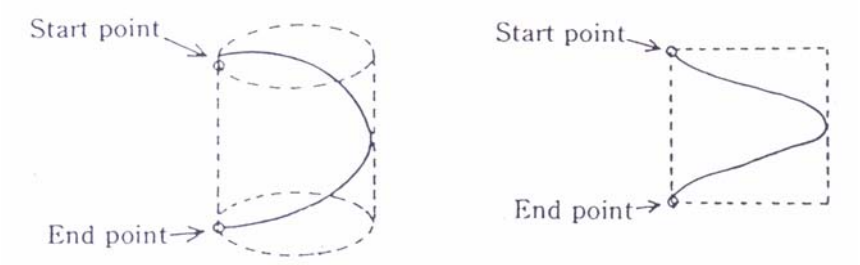
X : X coordinate of helical cut end point

Y : Y coordinate of helical cut end point

Z : Z coordinate of helical cut end point

I : X component of distance from start point of helical cut to its center

J : Y component of distance from start point of helical cut to its center



[Note]

*10 and JO can be omitted on an individual basis, but omission of I and J oil a collective basis results in occurrence of an error.

*The execution of the circular cut code with omission of X and Y, meaning the setting of the circular cut end and start points at the same coordinates, cause the cutting to occur in a 360~degree arc (complete circle).

(3) Oblique circular cut

As in the case of helical cutting, the "Circular Cut" code (G02, G03) can be used for oblique circular cutting.

The input format is as follows'

[G02] X ____Y____I ____J ____BZ
[G03]

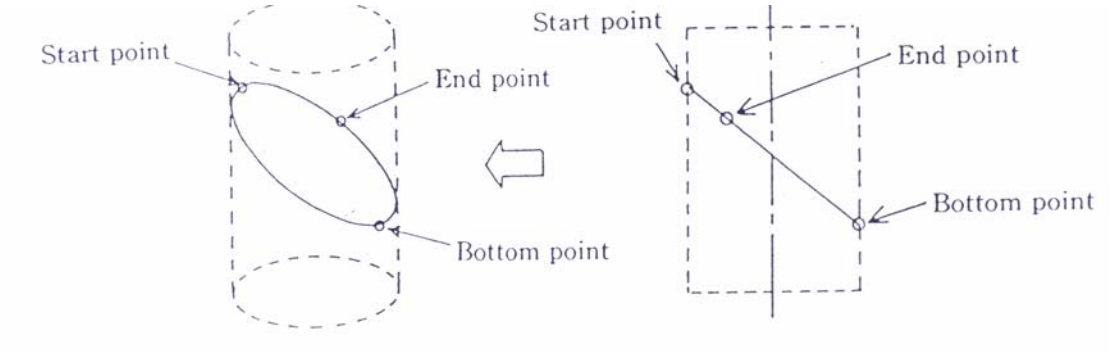
X : X coordinate of oblique circular cut end point

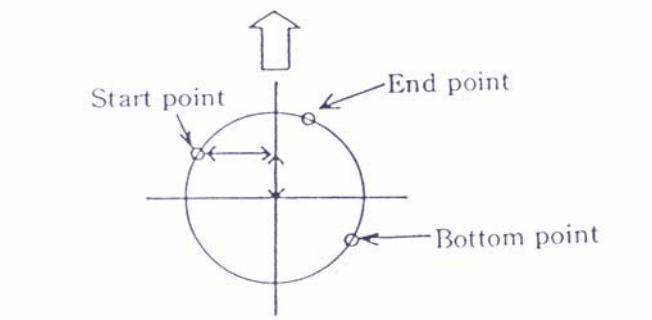
Y : Y coordinate of oblique circular cut end point

I : X coordinate of oblique circular cut center

J : Y coordinate of oblique circular cut center

BZ : Z coordinate of oblique circular cut bottom point





Note

* I0 and J0 call be omitted on an individual basis; however, omission of both causes all error to occur.

*55 A program for circular cutting in which X and Y specification is omitted, meaning that the end and start points of the circular cutting are the same, is interpreted as specifying a 360~ circular arc (or complete circle).

4) G04 (Dwell)

The execution of G04 X————

causes a pause of the specified time to be put between the block preceding this command

and the block following it without stopping the discharge. The dwell time can be specified

up to 99999.999 seconds on a 0.001 second basis with DIGIT=OFF and INCH=OFF.

[Example] Dwell of 3.5 sec

G04 X 3.5; or

G04 X 3.500;

At this point, the system can respond to OFF key.

The program for dwell of 3.5 seconds with DIGIT=ON and INCH=OFF is as follows:

G04 X 3.5; or

G04 X 350000;

5) G05,G06,G07,G09,G93,G94,G95,G96,(Mirror Image and Cancel)

These G codes are used to reverse the +/- direction of the axes during their travel Or machining as shown in the following table.

G05	X direction mirror image
G06	Y direction mirror image
G07	Z direction mirror image
G09	Mirror image and X-Y exchange cancel

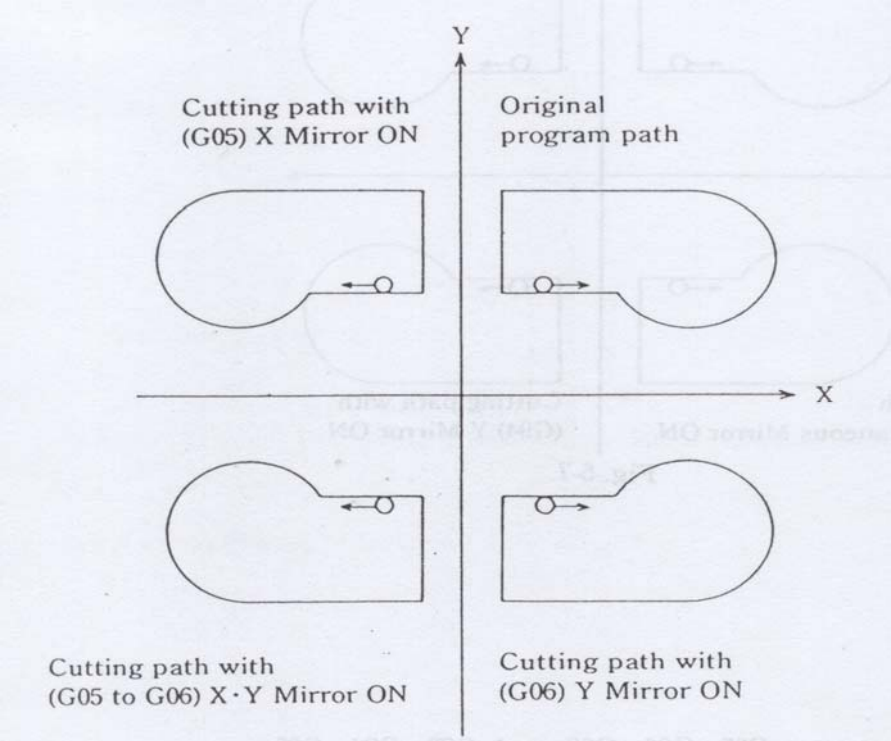


Fig. 5-6

G93	X direction mirror image
G94	Y direction mirror image
G95	X.Y simultaneous mirror image
G96	Mirror image cancel

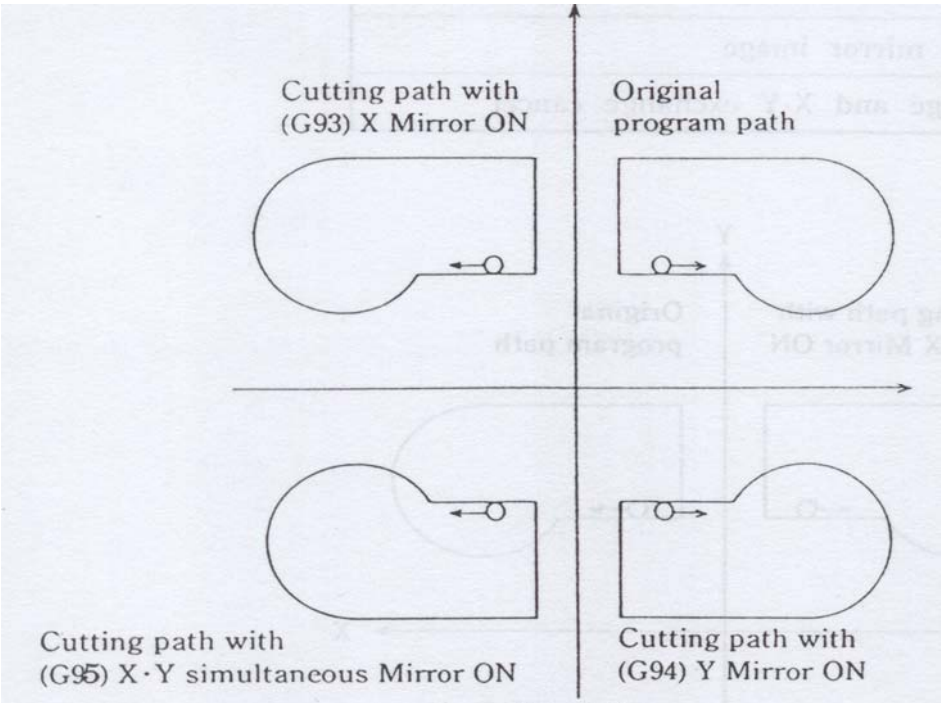


Fig. 5-7

Note: Differences between G05, G06, G09 and G93, G94, G95

X and Y mirror images can be executed simultaneously Offset direction can be changed.	G05	G93	Only X mirror image call be executed.
	G06	G94	0nly Y mirror image can be executed.
Both mirror image and X-Y exchange can be cancelled.	G09	G95	0nly .mirror image can be cancelled

Example :G05; G93;

G06 ;--X.Y simultaneous mirror G94 ;--Only Y mirror.

G05; G93;

G94 ;--Only Y-mirror. G06 ;--X-Y simultaneous mirror

6) G08, G09 (X-Y Exchange and Cancel)

G08 code, when the X axis is specified for machining, causes the machining to occur on the Y axis and, when the Y axis is specified for machining, causes the machining to occur on the X axis.

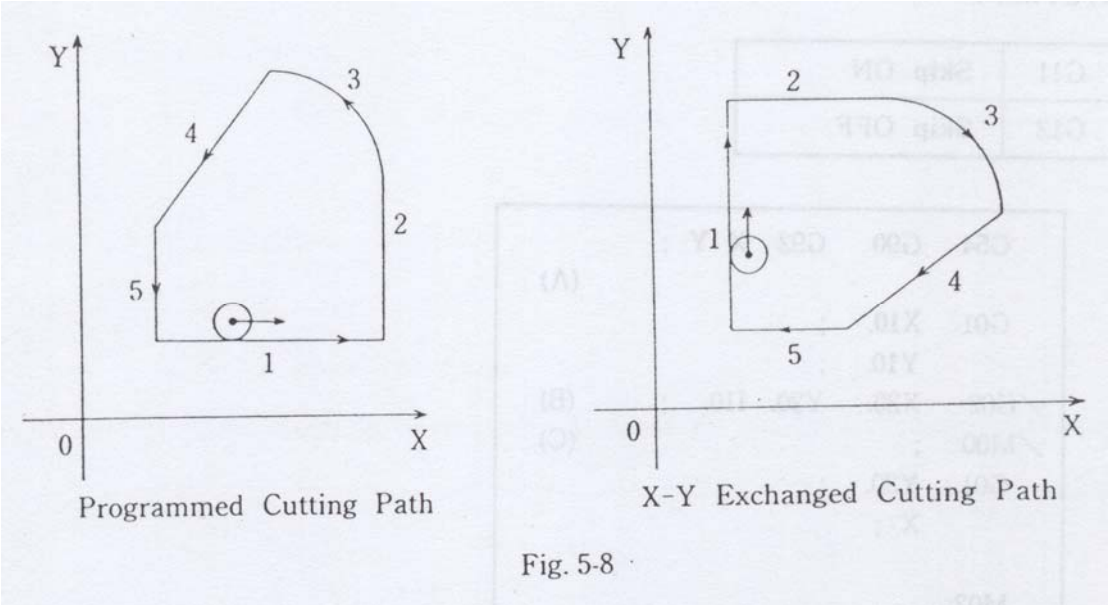


Fig. 5-8

G09 cancels the X-Y exchange function.

7) Gl1, Gl2 (Skip ON/OFF)

Like SKIP, a parameter provided by the SETTING mode, Gll and Gl2 are used to determine whether or not to ignore the block where '/' exists. For details, see CHAPTER4. BI.OCK SKIP ('/') of this instruction manual and Section 7 "Setting of Flags for Program Execution" of CHAPTER 4 in separately-available instruction manual "CONTROL.SYSTEM".

G11	Skip ON
G12	Skip OFF

```
 G54   G90   G92    X   Y ;
                                    (A)
 G01   Xl0.   ;
       Y10.   ;
/G02      X20.   Y20. Il0.   ;          (B)
/MOO   ;                          (C)
G01   Y30.   ;
      X ;
M02;
```

The above program, when executed with input of GII (SKIP ON), skips over blocks (A), (B) and (C).

8) Gl5 (U Axis Machine Origin Return)

The execution of this code causes the electrode to return to the U axis machine origin with the subsequent setting of the U axis machine coordinate at "0". In addition, when Gl5 is executed with U AXIS 0 SET at "0" or "2", the current coordinates of the U axis in G54 G959 coordinate systems are set at "0".

9) Gl7, Gl8, Gl9 (Plane Selection)

This codes are used to select the plane for circular cutting

Gl7 XY plane selection

Gl8 XZ plane selection

Gl9 XZ plane selection

When POWER ON, the system is loaded with Gl7 set for plane selection.

10) These G codes are used to determine whether or not to input data on length in inch.

G20	Inch input ON
G21	Inch input OFF (Metric system input)

When POWER ON, the system is loaded with either G20 or G21 set according to INITINCH in the SETTING' OPERATION sub mode.

N9te' This code must be executed at the head of NC program.

11) G22, G23 (Software Limit)

These G codes detention whether or not to cause the cutting operation to occur within the travel range set for each axis.

G22	Software limit ON
G23	Software limit OFF

The data on the travel range for each axis are to be set in SETTING-MACHINE mode beforehand.

Note: *24 The execution of G22 with the travel range set at 0 causes the whole region

of the specified axis to become a prohibited area, where its travel and cutting operation cannot be executed.

The travel range is specified by the machine coordinate. When DIGIT=ON, raise the coordinate value by one digit.

For details, refer to 2.3. 1) "How to Use Software limit".

12) G24, G25 (AJC Operation Command)

G24	High speed AJC operation Command
G25	Ordinary AJC operation command

These codes are used to select the mode of the AJC operation, which is executed to accelerate removal of chips produced by EDM cutting, thereby stabilizing the cutting operation.

G24 is a command for high speed AJC operation, which occurs at a speed about three times as high as that of AJC operation provided by G25. It should be noted that the AJC operation executed by this code takes place only on the Z axis, but not on the X, Y and other axes. Therefore, G24 code is to be used for single-axis machining on the Z axis.

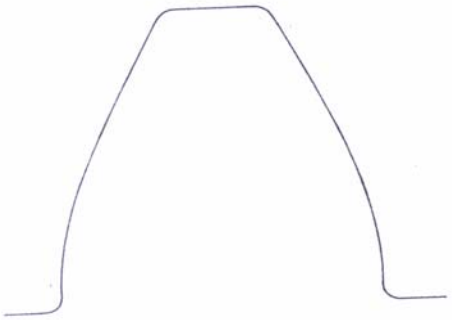
G25 is a command for ordinary AJC operation, which occurs according to the cutting path. Therefore, this code can also be used for coaxial machining on more than one axis and contour machining.

When POWER ON, the system is loaded with G25 set for AJC operation. Once G24 or G25 code is executed, the executed state is held as it is Until G24/G25 code is input again.

12) G26, (;27 (Figure Rotation ON/OFF)

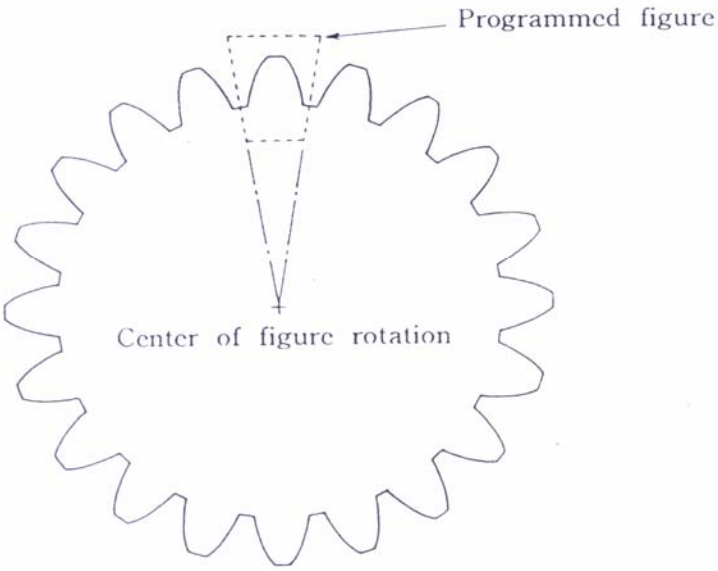
These G codes are used to determine whether or not to perform figure rotation.

G26	Figure rotation ON
G27	Figure rotation OFF



(Sub-programmed figure)

Create such a figure as illustrated in tile drawing at left as a sub-program. Then, select the center, around which the figure is rotated, and rotate the figure to machine such a shape as illustrated ill the drawing at lower left.



* For details, refer to the Figure Rotation section of CHAPTER13

14) G28, G29, G60 (blain Reference Point)

G29	Setting of main reference point
G28, G60	Return to main reference point

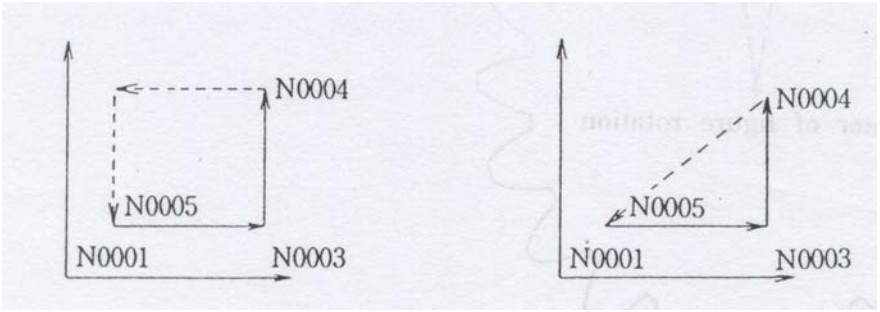
G28 and G60 codes feed the specified axis to the G29 set position with response to "ST Detection", "Limit Detection" and "HALT Key".

G29 (Setting of main reference point)

G29 specifies the current machine coordinates of all axes as the main reference point. The position memorized by G29 is backed up by a battery for data protection against power failure caused for some reason or other. The specified axes travel to the set position simultaneously in the G type and individually in the S type.

```
N0001   G54 G92 Xl0   Y10;
N0002 G29;
NOO03 GOO Xl00;
N0004 GOO        Y100;
N0005 G28;
```

Electrode movement in S type Electrode movement in G type



The specified axes travel individually.

Tile specified axes travel simultaneously.

- G29 offset term memory function

The specification of an offset term after G29 in the same block allows the contents of the offset term to be stored in the memory.

G29 H1 HOO8 H17 H995;

(1) The offset term data given following G29 is memorized and reset when it is to be executed again.

(2) A series of "0" numbers immediately after H can be omitted.

[Note]

i) For setting of the offset term data, the private file takes priority over the OFFSET FII.E, which can only be used for such setting when no offset term data are available in the private file header section.

ii) Offset term updating takes place both in the private file and OFFSET FILE depending on the OFFSET CHANGE of the SETTING.

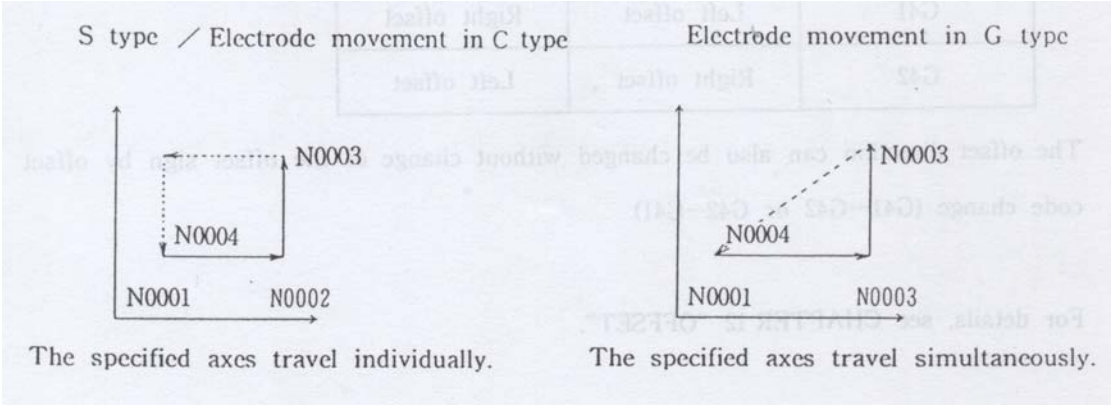
iii) G29 must be executed in an independent block.

15) G30 (Return to Coordinate Set Point)

G30 code is used to cause the specified axis to travel to the machine coordinates specified in the G92/G97 command block.

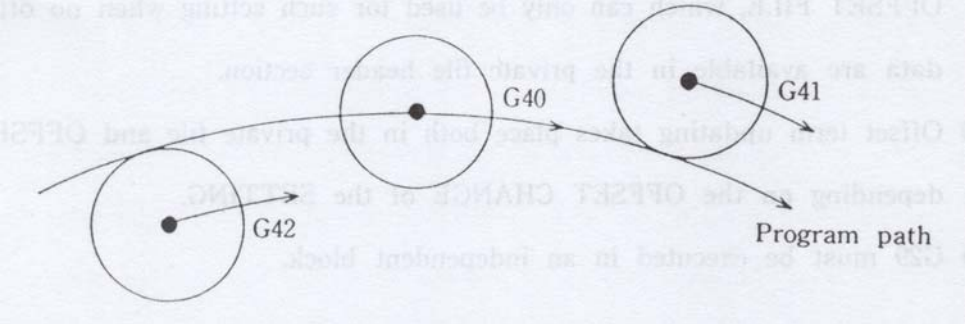
If the program contains more than one G92/G97 code, the machine coordinates specified by the last G92 executed block will be stored in the memory. With the G type, all the axes travel simultaneously. With the S/G type, each axis travels independently. The travel occurs on all the axes without reference to the G92/G97 specified axes.

```
N0001   G54   G92   Xl0   Y10;
N0003   G00   Xl00;
N0004   G00         Y100;
N0005   G30;
```



16) G40, G41, G42 (Electrode Diameter Offset)

The electrode diameter offset function is provided to achieve the programmed dimension son the basis of the electrode diameter by offsetting the electrode center path to the left (G41) or to the right (G42) with regard to the program path.



G40	Electrode diameter offset cancel
G41	Electrode diameter offset to left (for offset of electrode to left with regard to the direction of its advance)
G42	Electrode diameter offset to right (for offset of electrode to right with regard to the direction of its advance)

G41 and G42 are commands for selection of the offset mode, while G40 is a command
for cancellation of this mode.
The offset direction can be determined by the electrode diameter offset codes (G41, G42) combined with the offset sign (+)/(-) as shown in the following table'

Offset sign	Plus (+)	Minus .(-)
G41	left offset	Right offset
G42	Right offset	Left offset

The offset direction can also be changed without change of the offset sign by offset code change (G41--G42 or G42--G41).

For details, see CHAPTER 12 "OFFSET".

17) G48, G49(Edge Control ON/OFF)

O These G codes are used to determine whether or not to execute the edge control function

The edge control amount is set by the following parameters in SETTING OPERATION mode :

(a) OUT CORNER OVER
(b) OUT CORNER RETURN
(c) IN CORNER OVER
(d) IN CORNER RETURN

O Requirements for edge control

- G48 Edge Control ON.
- Offset ON (The edge control function will not operate if the offset is set at "0".)
- Straight line-straight line offset path.
- Comer R OFF.

The edge control function is achieved only if the above requirements are met.

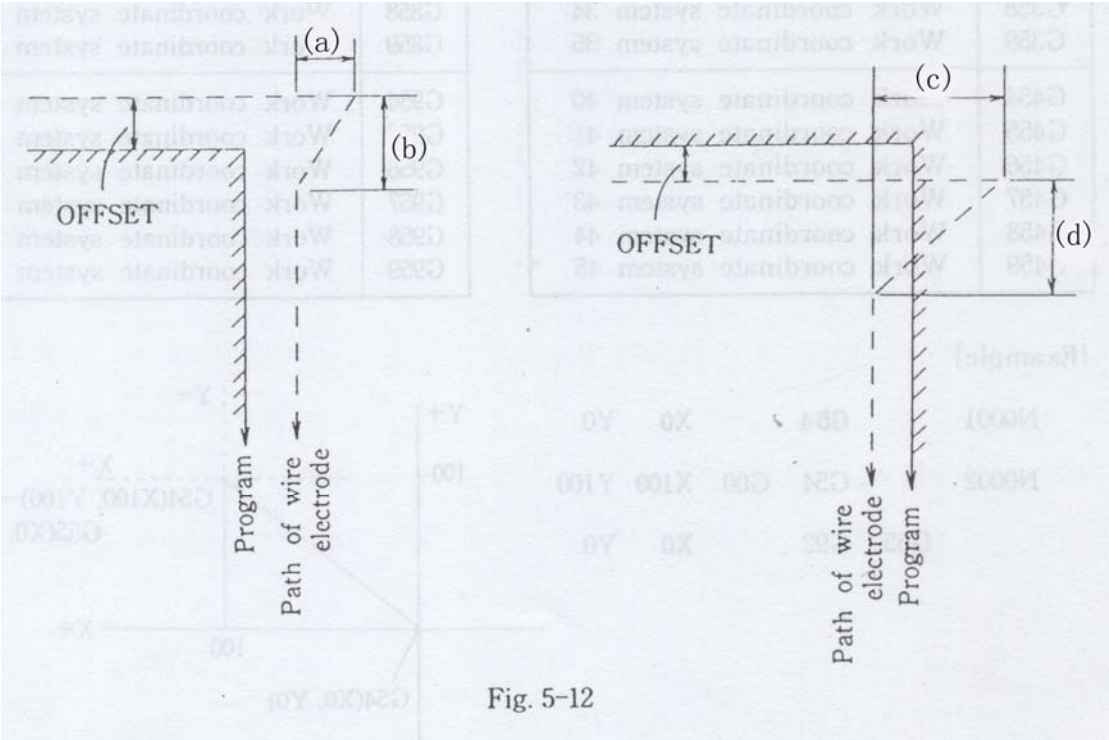


Fig. 5-12

18) G54 to G959 (Work Coordinate 0 to 95)

These G codes are used for selection of the work coordinate system from the following:

0~5, 10~15, 20~25, 30~35, 40~4.5, 50~55, 60~65, 70~75, 80~85 and 90~95.

G54	Work coordinate system 0	G554	Work coordinate system 50
G55	Work coordinate system 1	G555	Work coordinate system 51
G56	Work coordinate system 2	G556	Work coordinate system 52
G57	Work coordinate system 3	G557	Work coordinate system 53
G58	Work coordinate system 4	G558	Work coordinate system 54
G59	Work coordinate system 5	G559	Work coordinate system 55
G154	Work coordinate system 10	G654	Work coordinate system 60
G155	Work coordinate system 11	G655	Work coordinate system 61
G156	Work coordinate system 12	G656	Work coordinate system 62
G157	Work coordinate system 13	G657	Work coordinate system 63
G158	Work coordinate system 14	G658	Work coordinate system 64
G159	Work coordinate system 15	G659	Work coordinate system 65
G254	Work coordinate system 20	G754	Work coordinate system 70
G255	Work coordinate system 21	G755	Work coordinate system 71
G256	Work coordinate system 22	G756	Work coordinate system 72
G257	Work coordinate system 23	G757	Work coordinate system 73
G258	Work coordinate system 24	G758	Work coordinate system 74
G259	Work coordinate system 25	G759	Work coordinate system 75
G354	Work coordinate system 30	G854	Work coordinate system 80
G355	Work coordinate system 31	G855	Work coordinate system 81
G356	Work coordinate system 32	G856	Work coordinate system 82
G357	Work coordinate system 33	G857	Work coordinate system 83
G358	Work coordinate system 34	G858	Work coordinate system 84
G359	Work coordinate system 35	G859	Work coordinate system 85
G454	Work coordinate system 40	G954	Work coordinate system 90
G455	Work coordinate system 41	G955	Work coordinate system 91
G456	Work coordinate system 42	G956	Work coordinate system 92
G457	Work coordinate system 43	G957	Work coordinate system 93
G458	Work coordinate system 44	G958	Work coordinate system 94
G459	Work coordinate system 45	G959	Work coordinate system 95

[Example]

```
N0001        G54          X0      Y0
N0002        G54   G00    X100    Y100
       G55   G92          X0      Y0
```

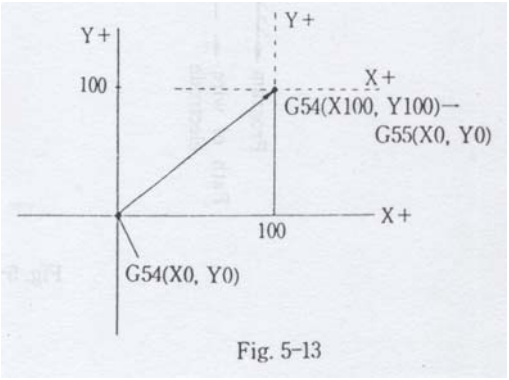


Fig. 5-13

In the above example, N0001 sets the current values (X, Y coordinates) ill the work coordinate system 0 at "0".

On the other hand, N0002 feeds the X and Y coordinates in the work coordinate system

0 by 100urn for each, setting the current values (X, Y coordinates) in the work coordinate

system 1 at "0".

Note: * Work coordinate system 0 (GYM) is selected when POWER ON.

The work coordinate system 95, which is a machine coordinate system, cannot be processed by G92/97 (which will be described later) for coordinate setting. The execution of these codes for coordinate setting of this coordinate system causes and error to occur.

19) G80 (Travel to ST Stop)

The execution of this code causes the electrode to travel from its current position until it comes into contact with the workpiece.

G80{Axis specified for travel)-+ {Specified travel amount}or G80{Axis specified for travel}-+ The specification of an amount, by which the axis travels, causes the electrode to travel from its current position by the specified amount. If, during its travel by execution of this code, the electrode comes into contact with the workpiece, it performs ST operation to stop its travel.

[Example 1] G90 G80 X-10000;

The above program causes the electrode to travel 10mm on the X axis. If the electrode comes into contact with the workpiece in 'the middle of this travel, it performs ST operation to stop its travel. The input of the travel amount can be omitted. The omission of the travel amount input causes the electrode to travel on the specified axis in the specified direction until it comes into contact with the work piece.

[Example 2] G80 X-;

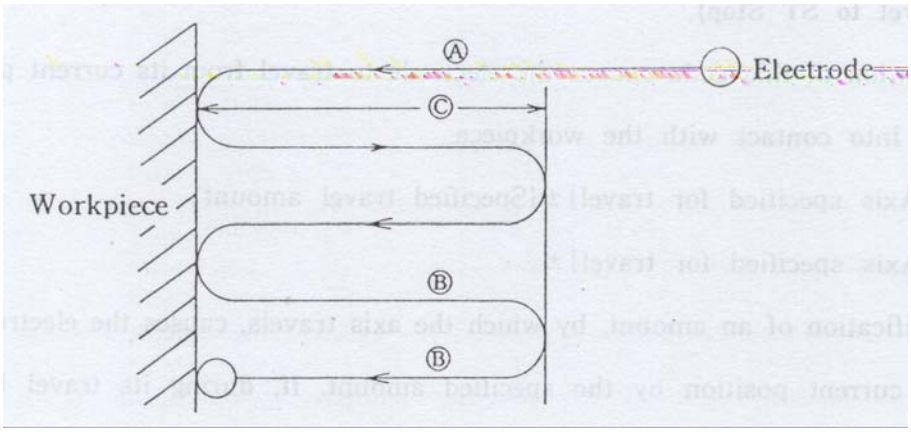
This program causes the electrode to travel on the X axis in the (-) direction until it comes into contact with the workpiece. If such contact takes place, the electrode performs ST operation to stop its travel.

Note

(1) The numerical data input following the axis and its travel direction represents the coordinate value when "Absolute" coordinate system (G90) is selected and the travel amount when "Incremental" coordinate system (G91) is selected.

(2) G80 does not allow more than one axis to travel simultaneously. The execution of the code for travel of more than one axis results in operation of only one axis input first.

The following describes the ST operation performed by the electrode when it comes into contact with the work piece.



The ST operation condition can be controlled by setting the following parameters provided

in SETTING-MOTOR sub mode:

G80 SPEED RATE :Sets the speed of (A) (the speed at which the electrode travels until its first contact with the work piece).

G80 FRONT SPEED: Sets the speed of (B) (the speed at which the electrode travels after its contact with the work piece).

GS0 AFTER RETURN : Sets the distance of (C) (the amount by which the electrode travels back from: the position of its first contact with the work piece).

G80 PICK FREQUENCY: Sets the frequency at which the electrode comes into contact with the work piece.

20) G81 (Travel to Machine System Limit)

The axis specification following G81 code causes the electrode to travel on the specified
axis to the machine system limit.

When LIMIT of SETTING' AXIS is set at "0", the electrode will not travel with the machine coordinate value set at "0".

[Example 1] GS1 X+;

This program causes the electrode to travel on the X axis to the (+) limit.

G81 Y+ X+

This program causes the electrode to travel on the X axis to the (+) limit and then on the Y axis to the (-) limit.

Note: The execution of G81 for more than one axis in one block causes the electrode to travel on one axis at a time in the order of X, Y, Z, U, V, W, UU and VV.

The execution of GS1 in the G type with a motor CPU when the backup has been cleared causes the machine coordinate to be set at '0" after the end of electrode travel to the (-) limit and at the maximum stroke of the axis after the end of electrode travel to the (+) limit.

When this code is used with both X-Y CHANGE: and MIRROR IMAGE set in ON state, MIRROR SWAP REVERCE, a parameter in the SETTING' OPERATION, can be set to select the order in which the two commands are executed.

When MIRROR SWAP REVERCE=ON,

MIRROR IMAGE and X-Y CHANGE are executed in this order.

When MIRROR SWAP REVERCE=OFF,

21) G82 (Travel Halfway between Current Position and Origin)

This code causes the electrode to travel to the midpoint between the current position and the origin on the specified axis.

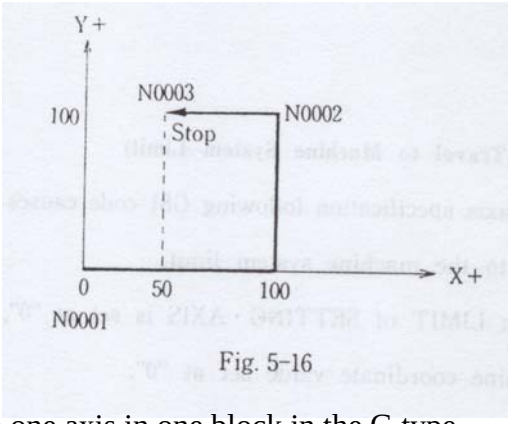


Fig. 5-16

[Example]

```
N0001   G54    G92   X0     Y0;
N0002   GOO          Xl00   Y1OO-
N0003   G82          X;
```

Note: The execution of this code for more than one axis in one block in the G type with a motor CPU causes the electrode to travel in a straight line.

22) G83 (Read Present Value in Specified Offset Term)

The use of G83 allows the value of the item specified by an address following this code to be read into the offset term specified by the number following the address.

G83 {Address} {Offset term No.}

A : Machining time for each block

S : Stop state, which leads to M03 searching

T : Machining time elapsed from start of program execution

X, Y, Z, U, V, W, UU, VV - Coordinates of the axes in the current coordinate system

The offset number is read in "OFFSET FILE" for 100'-,999 and in the private file for 000-~999.

The execution of this code requires the presence of H* * * = ±00000OO00 in the "OFFSET
FILE" or the private file.

The execution of this code causes the offset modes to be canceled on a temporary basis.

[Example 1]

G83 X000.....Reads the current position of X axis in H000.

G83 Y001.....Reads the current position of Y axis in H001.

G83 T002.....Reads the machining time in H002

(if the time required for cutting to the current position is one hour and 23minutes and 45seconds, H002 = +01: 23: 45.)

Example of use of G83

G~ G92 XY

GOO Xl0.

G01 Y10.

G83 T000..... (A)

G01 X0

G83 T001..... (B)

G01 Y20.

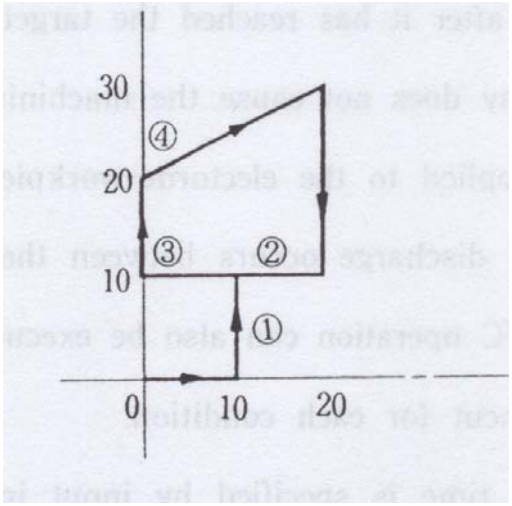
G83 T002 .....(C)

G01 X20. Y30.

G01 Y10.

G01 Xl0.

M02



(A):H000=Time for cutting from (1) to (2)

(B):H001=Time for cutting from (1) to (3)

(C):H002=Time for cutting from (1) to (4)

23) G85 (Timer Machining)

This code is used to execute the machining for a specified period of time.

The input format is as follows:

G&5 {Address} {Specified machining time}

The address specifies X, Y, Z, U, V, W or T.

The specification of X, Y, Z, U, V or W in the address causes the machining to end when it has been performed in excess of the specified time even if it has not reached the target position.

The specification of T in the address causes the machining to continue for the specified period of time after it has reached the target position. However, the execution of this code in this way does not cause the machining to proceed beyond the target position with voltage applied to the electrode-workpiece gap for the specified period of time whether or not discharge occurs between the gap. During this period of time, servo and I.ORAN/AJC operation can also be executed to minimize the volume of workpiece material left uncut for each condition.

The machining time is specified by input in hour/minute/second up to 99999 hours 99 minutes 99 seconds as shown in the following:

G85 X _*****_ _**_ _**_

Hr. Min. Sec.

The block following G85 is to be used for execution of G01.

[Example 1]

G90;

G85 Z1000; The machining will end if it has been performed in excess, of 10 minutes or it has reached the target position of -10mm.

G01 Z-10.;

[Example 2]

G85 T100;

G01 Z-10.; The machining will continue for one minute after it has reached the target position of -10mm. However, the machining will not proceed more than -1mm beyond the position specified by G01.

Note: Use G85 in an independent block. It is impossible to use this code in the following way, which causes an error to occur.

G85 Xl000 G01 Z-5.;

The block following G8,5 is to be used for execution of G01.

(G85 cannot be used for circular cutting by G02 or G03.)

It is impossible to use G85 for LOCK LORAN.

24) G90 (Absolute Coordinate Command), G91(incremental Coordinate Command)

Gg0: Commands the electrode to travel on the specified axis on the basis of its coordinates in the current coordinate system. (Absolute)

G91 : Specifies the amount by which the electrode travels on the specified axis with its current position as origin. (Incremental)

There are two methods for specifying tile amount by which the electrode travels on the specified axis is to travel: Incremental and Absolute commands.

<incremental command>

This command is based on G91 for direct programming of the travel amount itself. This method specifies the amount by which the electrode travels from its current position in the direction of the specified axis.

<Absolute command>

This command is based on Gg0 with the end point of the electrode travel expressed as a coordinate value in the work coordinate system for programming of the coordinate value.

This method specifies time destination of tile travel with the origin assumed as the Center.

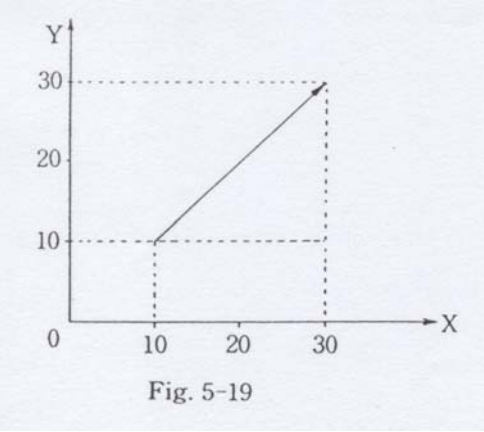


Fig. 5-19

<(incremental command >

G54 G91 G92X'0 Y0;

G01 X20. Y20.;

<Absolute Command>

G54 G90 G92 Xl0. Y10.;

G01 X30. Y30.;

Both of tile above two commands cause time electrode to travel to tile same position.

25) G92, G97 Ga0 (Coordinate Origin Setting and G92 Point Return)

G92 code specifies the current position of the electrode as coordinates ill the current work coordinate system.

G97 code specifies the current position of the electrode as coordinates in all the coordinate

[Example] G92 Xl00 Y100;

The above command causes the current position of tile electrode to be set at (X,Y)-100, 100)

Note: Coordinate setting of the electrode is impossible in work coordinate system 95.

26) Gl04, Gl05 (Corner Dwell ON/OFF)

O These G codes are used to determine whether or not to execute automatic insertion of dwell between blocks of Travel or Cutting code.

G92 X Y
G41 H000
GOO Xl0. ..... (1)
Gl04 X5. .....Corner Dwell ON
G01 Y10. .....(2)
G01 X20. .....(3)
G105 .....Comer Dwell OFF
G01 Y .....(4)
G40 G01 Xl0. .....(5)
M02

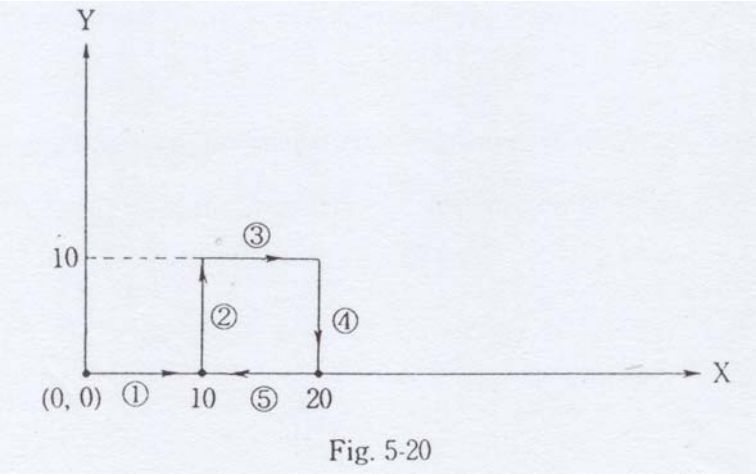


Fig. 5-20

This program causes dwell to be inserted between blocks ~-~ and ~-O for five seconds each.

* 1. OFFSET ON is required for execution of the Corner Dwell function.

*2. The Corner Dwell function cannot be executed on any geometry identified as tangential by I.-l. ANGI.E (described in the section "OPERATION" of CHAPTER 8. SETTING in instruction manual "CONTROL SYSTEM").

*3. The Corner Dwell function is automatically completed if the discharge gap is in good state. The monitoring of the discharge gap state is set in the SETTING ~ DISCHARGE mode as described in the following table:

Discharge Gap Monitoring Parameter

Setting parameter	Description
Gl04 SENSE LEVEL	Discharge gap detection sensitivity low High 1 ——— 99
G104 ESC I.EVEL	Discharge gap detection voltage for escape from Comer Dwell Low High 0 ——— 9(S/C type) 0 ——— 225(G type)

27) G126, G127 (Coordinate Rotation ON/OFF)

These G codes are used to select whether or not to execute coordinate rotation.

G126	Coordinate Rotation ON
G127	Coordinate Rotation OFF

For details, refer to CHAPTER 16 "FIGURE ROTATION AND COORDINATE ROTATION".

28) G128, G129~G928, G929

G128 commands the specified axis to travel to the position specified by G129 with response to "ST Detection", "Limit Detection" and "HALT Key". When the system responds to the "HAI.T Key", it does not allow execution of JOG operation.

G129 specifies the current position of the electrode as a reference point.
The G129 specified position is backed up by a battery for data protection against power failure caused for some reason or other.
The same applies to G228, G229-G928, G929.

Reference point	Return	Setting
1st sub-reference point	G128	G129
2nd sub-reference point	G228	G229
3rd sub-reference point	G328	G329
4th sub-reference point	G428	G429
5th sub-reference point	G528	G529
6th sub-reference point	G628	G629
7th sub-reference point	G728	G729
8th sub-reference point	G828	G829
9th sub-reference point	G928	G929

Note : The execution of the "sub-reference point return" function ill the G type with a motor CPU causes the electrode to travel in a straight line.

29) Gl30, Gl31, G132, G133, G136, G137

(Interference Check ON/OFF, Interference Avoiding ON/OFF, Interference Error ON/OFF)

Gl30	Interference check	ON
Gl31	Interference check	OFF
G132	Interference Avoiding	ON
G133	Interference Avoiding	OFF
G136	Interference error	ON
G137	Interference error	OFF

The execution of the Electrode Diameter offset function in machining of a groove or other parts smaller than the offset amount may result in a difference between the offset and program paths, causing the electrode to cut into the workpiece. These G codes are used to prevent such a trouble.

Note : G132 and G136 codes cannot be executed without execution of G130 code.

When G132 and G136 codes are executed simultaneously, the latter takes priority over the former. For details, see CHAPTER 13 "INTERFERENCE CHECK".

30) Gl60, Gl61, G162, G163, G164, G165, G166 (Three-dimensional Rotation)

The input format is as follows:

Gl60 — Three-dimensional rotation cancel

Gl61
G162
G163 AX ___AY___ AZ ___
G164
G165
G166

AX : Angle of rotation on X axis
AY : Angle of rotation on Y axis
AZ : Angle of rotation on Z axis

G161~G166 differ in the priority order of the three axes on which the electrode rotates as shown ill tile following:

	X axis	Y axis	Z axis
G161	1	2	3
G162	1	3	2
G163	2	1	3
G164	3	1	2
G165	2	3	1
G166	3	2	1

CHAPTER 6. X, Y, Z, U, V, W, UU, VV (I, J, K) COORDINATE AXES

This function executes tile travel of tile table and electrode by input of tile address to specify the axis, on which the table or electrode travels, and numerical data to specify the direction and the amount of the travel.

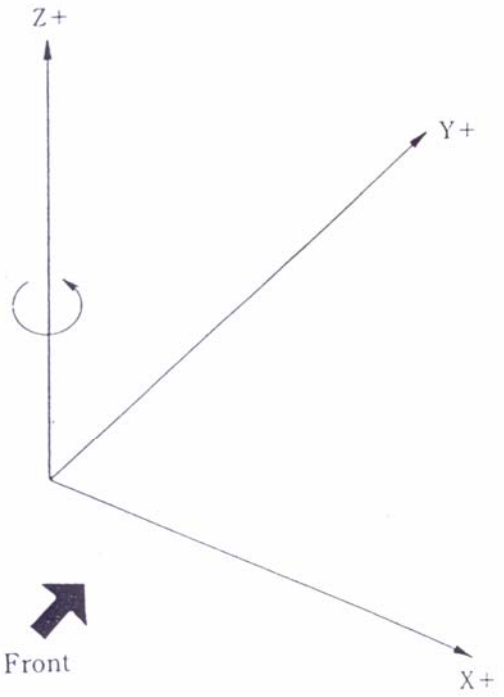


Fig. 6-1

(When viewed toward the machine front)
The left-to-right direction represents X axis.
Its left direction is defined as X(-).
Its right direction is defined as X(+).
The front-to-back direction represents Y axis.
Its direction toward you is defined as Y(-).
Its direction far from you is defined as Y(+).
The vertical direction represents Z axis.
Its upward direction is defined as Z(+).
Its downward direction is defined as Z(-).
Tile axis rotating on tile Z axis viewed from above represents U axis, which is optional.
Its counterclockwise direction is defined as U(+).
Its clockwise direction is defined as U(-).

Note: V, W, UU and VV axes are optionally provided according to the machine structure.

I. J and K are circular cut parameters to specify the relative position of the center of the circular arc viewed from its starting point. The maximum number of simultaneously

controllable axes is 8 for G type and 6 for C type.

The input set units and maximum command values are as shown below.

DIGIT=0

	Input set unit	Maximumu command value	Minimum output unit
Metric system	0.001mm	999999.999mm	0.001mm
Inch unit system	0.0001 inch	99999.9999 inch	0.0001 inch

DIGIT= 1

	Input set unit	Maximumu command value	Minimum output unit
Metric system	0. 0001mm	99999. 9999mm	0. 0001mm
Inch unit system	0. 00001 inch	9999. 99999 inch	0. 00001 inch

Note: The minimum output unit is a minimum unit of output of tile NC power supply unit.

The actual driving unit depends on the machine and driver.

CHAPTER 7. T CODE

1) T82 AUTO DRAIN ON (Option), T83 AUTO DRAIN OFF

"AUTO DRAIN ON" causes drainage of the machining tank not to occur.

"AUTO DRAIN OFF" causes drainage of the machining tank to occur.

2) T84 Pump ON, T85 Pump OFF

"PUMP ON" causes the feed pump to operate to feed dielectric fluid to the machining tank.

"PUMP OFF" causes the feed pump to stop its operation of feeding dielectric fluid to the machining tank.

3) T86 Flush ON (Option), T87 Flush OFF

"FLUSH ON" causes flushing to occur.

"FLUSH OFF" causes flushing not to occur.

4)T88 OIL/WATER OIL, T89 OIL/WATER WATER,

T88 and T89 are used to select oil and water machining, respectively, while T94 is provided for a machine with a "BATH" switch to allow.

5) T01~T24

These codes are used for electrode change by ATC.

CHAPTER 8. M CODE

1)M02 (End of Program)

This code indicates the end of the main program, causing the program written following it not to be executed.

Also, when the <M02> code is executed, the modal code prior to the execution of <M02> is changed by numerical data in * * * CANCEL, a parameter provided by SETTING ~ OPERATION sub mode.

If "NO MAN" has been set in ON state, <M02> causes the power to be automatically turned off after it is read.

In DRY RUN mode, however, <M02> will not cause the power to be automatically turned off after it is read even with "NO MAN" set in ON state.

For instructions on setting "NO MAN", refer to Section 4 of CHAPTER 5 and Section 4 of CHAPTER 8 in instruction manual "CONTROL SYSTEM".

2) M00 (Program Stop)

<M00> causes the execution of the program to be stopped after it .is executed. As in the case of Single Block Stop, modal information before the execution of this code will be saved.

Pressing the RST key causes the program following "M00" code to be executed again. CHAPTER 21 will give a detailed description of the operating function of the machine during the execution of M00 (M01) Program Stop.

3) M01 (Optional Program Stop)

"M01" provides the same function as "MOO". It should be noted, however, that time execution of this function requires "OPTIONAL STOP" in SETTING mode to be set in ON state.

If OPTIONAL STOP=OFF, the execution of this code will be ignored.

4) M98 (Call-up of Sub-program)

This code is used for call-up of a sub-program.

A detailed description of the code will be given in CHAPTER 10 "SUB-PROGRAM".

5) M99 (End of Sub-program)

This code commands the end of a sub-program. The execution of ~M99> code causes the program to be returned to the main program for its execution.

6) M05 (ST Cancel)

This code causes the ST function to be ignored.

When "M05" code is executed, only one operation after its execution is performed with the ST function ignored.

7) M06 (No Discharge)

M06 G01 X_____Y_____; (Executes straight cut to X___ , Y___ position without discharge.)

M06 G02/G03 X_____Y_____I_____ J_____ ; (Executes circular cut to X_____ Y_____ position without discharge.)

Note: *During this operation, the ST function is operating. If you wish this function to be ignored, use "M05" together with this code.

8) M03

When, with "M03" in SETTING mode set at 1 or more, ARC STOP, LIMIT STOP or ST STOP occurs, GOO, G02, G03, G04, G28, G30, G60, G80, G81 and G82 following the block involved in the stop will be ignored to Searle M03 code for execution of the program following this code if it is found. For details, see CHAPTER 15 "M03."

9) M10 to M47 (Special Code)

M10 to M47 codes are used to command the start and stop of various options and attachments.

M10 to M17, M30 to M37, Mll0~Mll7, M130~M137	ON
M20 to M27, M40 to M47, M120~M127, M140~M147	OFF

These codes are paired in terms of ON and OFF as exemplified by such a pail- M15 (ON) M25 (OFF)

10)) M code Input

MS0~M87, M150~M187 (special codes)

MS0~M57	Waits for input voltage to rise above the set level.
M60~M67	Waits for input voltage to lower below the set level.
M70~M77	Waits for input voltage to rise above the set level.
Mg0~M87	Waits for input voltage to lower below tile set level

M150~M157	Waits for input voltage to rise above the set level.
M160~M167	Waits for input voltage to lower below tile set level.
M170~M177	Waits for input voltage to rise above tile set level.
Mlg0~M]87	Waits for input voltage to lower below the set level.

11) M0l

"AI.I. BI.OCK M04" ill the SETTING' OPERATION submode and M04 code can be used to select whether to return the machining to its start point.

The setting of "ALI. BI.OCK M04" in ON state causes the machining in any mode to return to its start point throughout all tile blocks involved in this machining.

The use of M04 causes only the machining involved in the block containing M0<I to return to its start point after its completion.

I)o not use M04 with "OFFSET" set in ON state.

For control only by M04, it is necessary to set "ALI. B1.OCK M04," ill OFF state.

M04 and "AI.I. BI.OCK M04" are executable only for straight cutting (G01) and ignored for circular cutting (G02, G03).

12)M07,M08,M09 (for use in turning unit)

M07	Turning unit LOCK
M08	Turning unit ON
M09	Turning unit OFF

13) M95 (ET)

M95 is used to operate the electrode feeder (ET).The execution of this code causes The electrode feeder to feed the electrode until it comes into contact with the woke piece. Note: This function is optionally available.

CHAPTER 9. B'C CODE

There are two kinds of command codes used to change the machining condition: B and C codes.

1) B Code

B codes are used to change the machining condition setting parameters (ON, OFF, IP etc.) on an individual basis.

There are two command formats for execution of parameter change' one by specifying the corresponding B No. and the other by specifying the corresponding parameter name.

G type

Parameter	Command format	
	B No.	Parameter name
ON	B130***	ON***
OFF	B160***	OF***
MA	B3100**	MA**
IP	B19****	IP****
SV	B340***	SV***
UP	B40000*	UP*
DN	B43000*	DN*
LN	B490***	LN***
STEP	B53****	STEP****
PL+	B000001	PL+
PL−	B000000	PL−
V	V2200**	VO**
HP	B2500**	HP**
PP	N5600**	PP**
C	B2800**	CO**
S	B370***	SERVO***
L	B4600**	LS**

C type

Parameter	Command format	
	B No.	Parameter name
ON	B130***	ON***
OFF	B160***	OF***
MA	B3100**	MA**
IP	B19****	IP****
SV	B3400**	SV**
UP	B40000*	UP**
DN	B43000*	DN**
LN	B490***	LN***
STEP	B53****	STEP****
PL+	B000001	PL+
PL−	B000000	PL−
V	V2200**	VO***
HP	B2500**	HP***
PP	N5600**	PP**
C	B2800**	CO**
S	B3700**	SERVO**
L	B4600**	LS**
LP	B59****	LP****

Input data in place marked with *.

2) C Code

C codes are used for selection of machining conditions.

This can be executed by input the appropriate numerical data in three digits or less following address 'C" as shown in the following example:

[Ex am pie]

C000;

The execution of the above example causes the contents of C000 to be displayed on the machining condition display area of the CRT screen. The actual data of C000 must be set before the program execution. The searching for the machining condition No. is first made in the private file and then in the COND file if it is not found in the private file. The absence of the No. from both files causes an error to occur.

C codes are classified by the type of machining conditions to be selected into the following three groups:

C000~C099 : Private machining conditions

C100~C899 : Public machining conditions

C901~C999 : PIKA machining conditions

The private machining conditions can also be selected by C code data of one or two digits.

CHAPTER 10. LORAN FUNCTION

1) Types of LORAN

The I.ORAN function is mainly used for single-axis machining, in which two axes other than the servo axis are operated for contouring of a simple shape, in order to accelerate removal of chips produced by EDM operation, thereby maintaining the discharge in stable condition, or to cut the side of a workpiece with regard to the direction in which the machining of the workpiece advances. This function, in addition to the above single-axis machining, is also used for two-axis machining for circular cut (G02, G03), as well as for multi-axis machining.

LORAN is classified by LORAN operation into three types and by I. ORAN pattern selection into two types.

FREE LORAN

This is the most commonly used type of LORAN. FREE LORAN allows the electrode to contour a shape specified independently of the servo axis. This operation causes the side gap parallel to the spindle to open and close alternately with the I.ORAN operation, producing the effect of removing chips from the gap.

This type of I.ORAN is used for machining of a workpiece of such a shape as makes its flushing difficult or uneven or machining of a blind hole.

HS I.ORAN

HS I.ORAN is used for machining, where the movement of the servo axis is limited to allow preferential execution of the LORAN operation for contouring on the plane (cycle motion in the horizontal direction).

During the I.ORAN operation in each quadrant, the servo axis only performs its servo operation within the preset range. After completion of the I.ORAN operation in one quadrant, the state of the discharge gap is judged to extend the servo range if it is good. If the discharge gap is in bad state, the servo axis is caused to wait or retract.

HS I.ORAN is applied for finishing of a roughed side face from its top in sequence

in order to secure the target surface roughness and dimension.

LOCK LORAN

LOCK LORAN is used for machining, where the movement of the servo is stopped to cause only I.ORAN operation to occur with a gradual increase in STEP (which will be described later) until it reaches that specified to complete the machining. Each time one cycle of LORAN operation is completed, the discharge gap state is judged to increase STEP if it is good. If the discharge gap is in bad state, the STEP is maintained at its present level or lowered from this level to remove the undercut on the side face.

This type of LORAN is therefore used to remove undercut and swell on the side face produced by roughing so as to secure the target dimensional accuracy.

LOCK LORAN is special machining which causes the electrode to operate with the movement of the servo axis stopped and therefore requires the amount of operation of the servo axis to be '0". Accordingly, to execute LOCK LORAN, it is necessary to feed the electrode to the machining start point by other machining or I.ORAN operations.

[Example]

```
G54 G90;
G01 Z-10.0;
LN201    STEP200;
G01    Z-10.0;          --I.OCK I.ORAN starts at position of Z-10.0.
M02;                        (The amount of operation of Z axis is 0.)
```

LOCK LORAN will no occur in a proper manner if the gap between the electrode bottom and the workpiece is narrow. Also, contact between the electrode and workpiece

at the start of the machining will cause the LORAN operation to occur without any increase in STEP.

Standard I.ORAN and Quadrant LORAN

There are six I.ORAN operation patterns available for selection, including "No LORAN operation".

It is also possible to select different LORAN operation patterns for each quadrant.

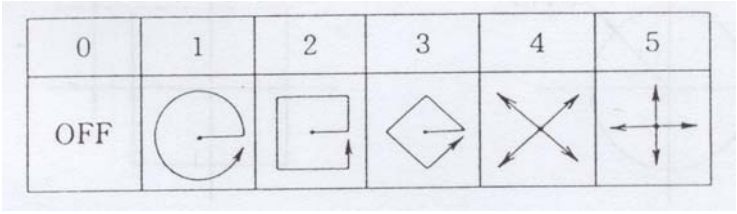
2) Machining Condition of LORAN

LN : Sets the type of LORAN operation, servo system, operation plane and pattern.

LN * * *

└LORAN operation pattern

0: No LORAN operation



The specification of quadrant LORAN makes this setting invalid.

The quadrant LORAN operation pattern is specified by input in LP.

└LORAN operation plane and servo system

0 :XY plane, no servo

1 :ZY plane, no servo

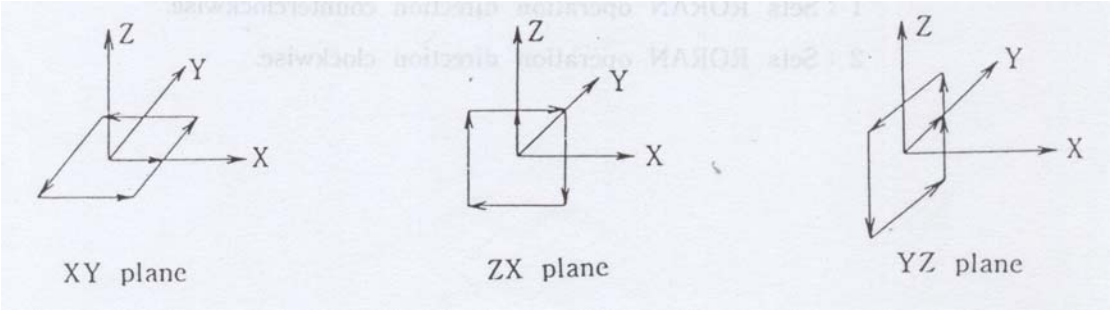
2 :YZ plane, no servo

3:XY plane, servo 2

(Causes the electrode to return to center at time of its retraction.)

4:ZX plane, servo 2

5:YZ plane, servo 2



└LORAN operation type

0 :FREE LORAN 5 :Quadrant FREE I.ORAN

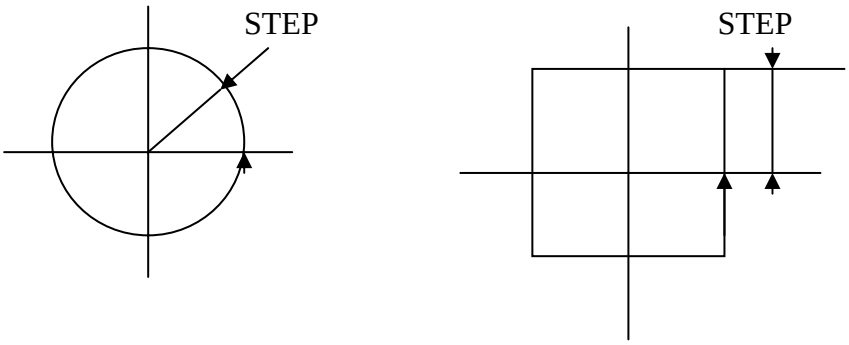
1 :HS I.ORAN 6 :Quadrant HS LORAN

2 :I.OCK LORAN 7 :Quadrant LOCK LORAN

STEP : Sets the radius from the center Of RORAN operation.

STEP * * * *

Sets the radius of LORAN operation in um within the range of 5 to 99.99 am



L : Sets the direction and speed of LORAN operation.

L * *

LORAN speed

Input 0 to 9 to set the speed of LORAN operation.

Input of "0" causes the LORAN speed to be set at its maximum level.

The larger the input value, the lower the LORAN speed.

This setting represents the distance to be covered by LORAN operation for a given period of time.

Therefore, an increase in STEP with the LORAN speed set at the same level results in a longer period of time required for one cycle of I.ORAN operation.

LORAN operation direction

0 :Reverse RORAN operation direction each time two cycles of RORAN operation are completed.

1 :Sets RORAN operation direction counterclockwise-

2 :Sets RORAN operation direction clockwise.

LP : Sets the quadrant I.ORAN operation pattern.

I.P * * * *

- 4th quadrant LORAN operation pattern
- 3rd quadrant LORAN operation pattern
- 2nd quadrant LORAN operation pattern
- 1st quadrant LORAN operation pattern

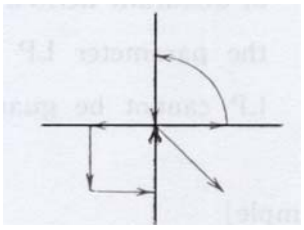
There are six numbers from 0 to 5 for input to select the LORAN operation pattern.

[Example]

The setting of LN and I.P as in the following:

I.N500;

LP1024;



causes quadrant LORAN to give such an operation pattern as illustrated in the drawing at right.

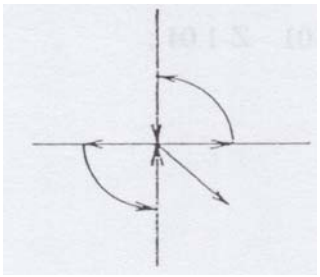
If quadrant I.ORAN operation is executed with a difference between the end point of its pattern in the current quadrant and the start point of its pattern in the next quadrant, the electrode will automatically travel in straight line to the operation start point in the next quadrant after the end of the operation in the current quadrant.

[Example]

The setting of LN and LP as in the following:

LN500;

LP1014;



causes quadrant I.ORAN to give such an operation pattern as illustrated ill the drawing at right.

Parameter I.P is set by HF key or B code.

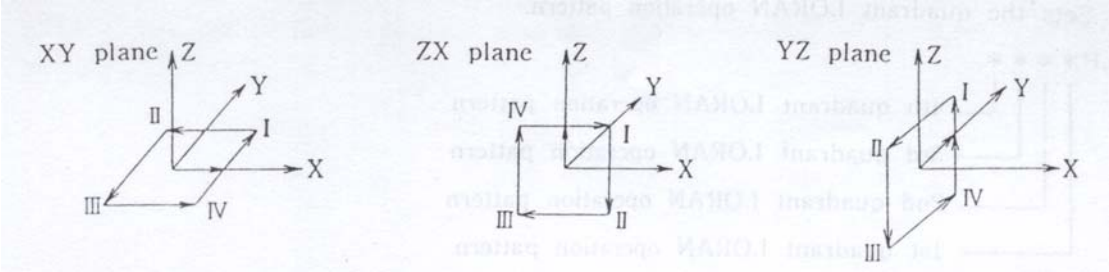
[Exam pie]

LP4321; (Sets LP at 4321.)

R594321 ;(Sets I.P at the same value as above.)

Quadrant setting made according to plane selection

The selection of the XY, ZX and YZ planes causes the quadrant setting to be made as shown in the following respective drawings.



Note: Parameter I.P is not contained in machining condition file. Therefore, the execution of quadrant LORAN, when the machining condition is changed by C code, requires the parameter LP to be set accordingly. Without such setting, the contents of LP cannot be guaranteed.

[Example]

C140;

I.NSO0 I.P4321; --C140 machining condition is selected with quadrant LORAN operation pattern of 4321.

G01 Z-1.0;

C120;

LP4321; --As the machining condition is changed, the parameter LP is to be set accordingly.

G01 Z-1.04;

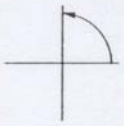
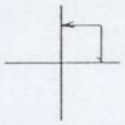
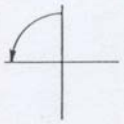
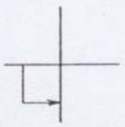
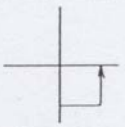
LN

Plane/Servo \ Pattern		OFF					
FREE LORAN	X-Y plane	000	001	002	003	004	005
	X-Z plane	010	011	012	013	014	015
	Y-Z plane	020	021	022	023	024	025
	X-Y plane Servo 2	060	061	062	063	064	065
	Z-X plane Servo 2	070	071	072	073	074	075
	Y-Z plane Servo 2	080	081	082	083	084	085
HS LORAN	X-Y plane	100	101	102	103	104	105
	Z-X plane	110	111	112	113	114	115
	Y-Z plane	120	121	122	123	124	125
	X-Y plane Servo 2	160	161	162	163	164	165
	Z-X plane Servo 2	170	171	172	173	174	175
	Y-Z plane Servo 2	180	181	182	183	184	185

Plane/Servo \ Pattern		OFF					
ROCK LORAN	X-Y plane	200	201	202	203	204	205
	X-Z plane	210	211	212	213	214	215
	Y-Z plane	220	221	222	223	224	225
	X-Y plane Servo 2	260	261	262	263	264	265
	Z-X plane Servo 2	270	271	272	273	274	275
	Y-Z plane Servo 2	280	281	282	283	284	285

Servo 1 : Causes the electrode to return along pattern at its retraction.
Servo 2 : Causes the electrode to return to center at its retraction.

List of Quadrant LORAN Patterns

Pattern No.		0	1	2	3	4	5
Srandard LORAN							
Quadrant LORAN	1st quadrant						
	2st quadrant						
	3rd quadrant						
	4th quadrant						

CHAPTER 11. SUB-PROGRAM

1) How to Use Sub-Program

Some programs are to be created with many repetitions of an identical program. Collection of these identical programs into one fixed program will eliminate program complexity and reduce trouble in program creation, as well as shorten program length. One fixed program thus created is called a sub-program. A program from which a sub-program is called up, is a main program.

[Main program]

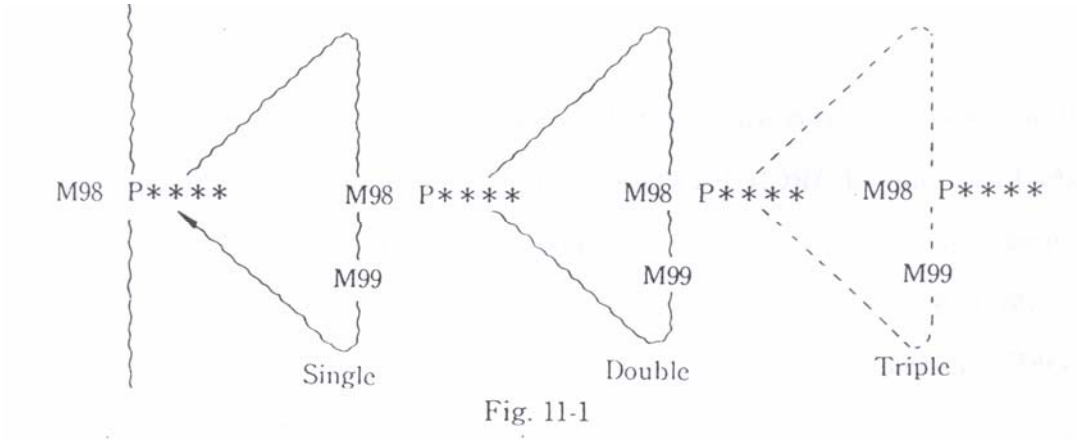


Fig. 11-1

When a sub-program is called up from a main program, it is called a single sub program. This sub-program can be called up to 50 times.

One command for call-up of a sub-program allows the execution of the sub-program to be continuously repeated up to 9.9999 times.

A sub-program must be created with M99 code written at the end of the pro/ram.

This M99 code commands return of the sub program to a main program.

N * * * *;

…….

…….

…….

M99;

A sub-program can be called up in the following format:

M98 P* * * * L* * * *

P* * * * Sequence No. of sub-program

L* * * * Number of repetitions of sub-program

Note： * When I. code is omitted, the sub-program will be called up only once.

*When LO is written in the above format, the CRT screen will display an error message "NC program format is incorrect".

A sub-program can be called up from a sub-program in the same way as when a sub-program is called up from a main program.

A sub-program can be called up from a sub-program up to 50 times.

2) Setting Parameters concerning Sub-Program

RAM LINK and SEARCH PATTERN, parameters provided in SETTING. OPERATION submode, are used to select the method of searching for the sequence No.

(1) RAM LINK (This parameter can only be used in RAM RUN mode.)

OFF: Searches only private file for sequence No.

ON :Searches not only private file, but also file loaded in RAM for sequence No.

(2) SEARCH PATTERN

OFF: Searches for sequence No. without String Exchange.

ON :Searches for sequence No. with String Exchange.

3) Special Way of Using Sub-Program at Its Return to Main Program

(1) When a sequence No. is specified at the end of a sub-program, the sub-program will not return to the main program, but the program specified by the sequence

No. will be executed.

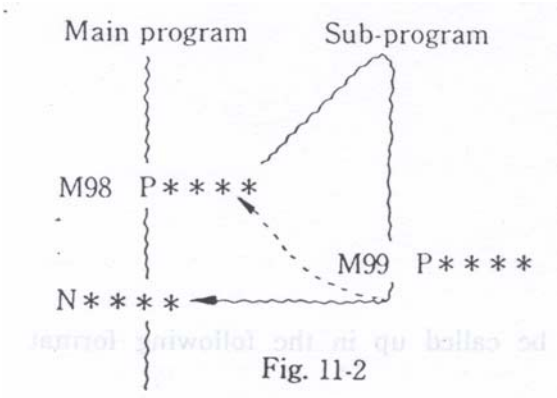


Fig. 11-2

(2) The number of repetitions of a sub-program can be changed in tile sub-program called up.

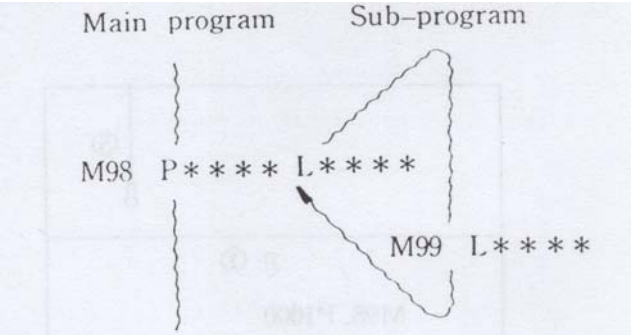


Fig. 11-3

This sub-program will be repeated as many times as specified before returning to a main program.

4) Sequence No. Searching Order

a) The sequence No. searching order is as shown in Fig. 11-4.

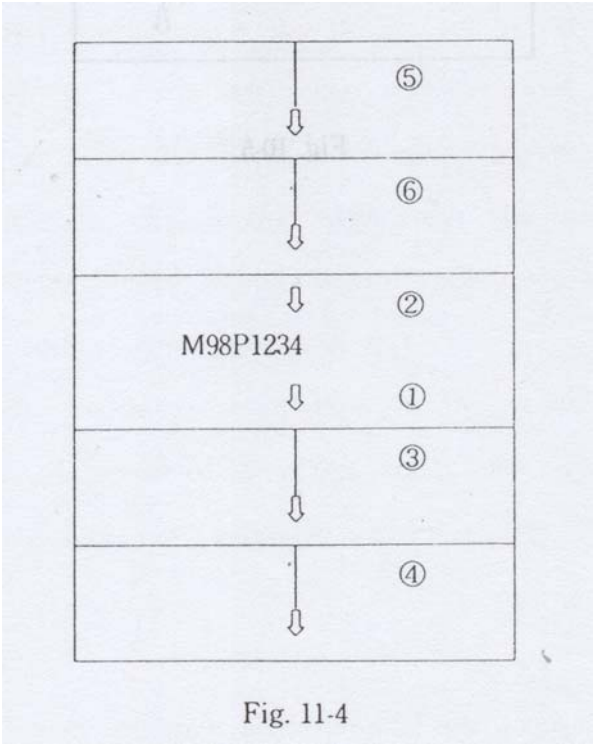


Fig. 11-4

Represents a file.

The searching order shown in the above figure is that given with RAM LINK set in ON state. If RAM LINK is set in OFF state, only (1) and (2) are executed.

b) When M99 P* * * * is programmed, the sequence No. searching order is as shown in Fig. 11-5.

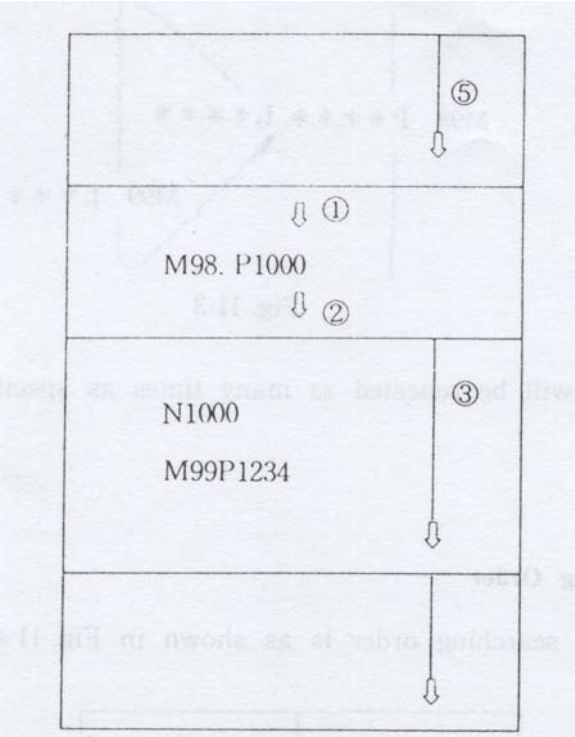


Fig. 10-5

CHAPTER 12. OFFSET

This function is not included in this machine.

1) Offset amount (D, H)

"OFFSET" is executed by input of numerical data of three digits or less (offset number) following D or H code.

The offset amount is written in the [OFFSET FILE] or individual files.

The offset amount can be set in the following range:

+/-999999.999mm (+/-99999.9999mm when DIGIT=ON)

+/-99999.9999inch(+/-9999.99999mm when DIGIT=ON)

The offset amount indicates the value input in

HO00= ±********

in FOFFSET FILE] or individual files.

Note : * The OFFSET mode will change over to CANCEL. mode either when the power is switched on or the [M02] code is executed.

The path of the electrode center coincides with the programmed path.

*The program in the Offset mode must end in the Cancel mode.

*If the program ends in the Offset mode, the system cannot position the electrode at the end point, which, therefore, will not make its final travel to this point, stopping short of it.

*When the following codes exist in the program, the offset mode is automatically canceled on a temporary basis.

(G11, Gl2, G28, G29, G60, G80, G81, G82, G83, G92)

2) Start of offset mode

The start of the offset mode requires positioning command <G00>> or straight cut command <G01~> . The use of circular cut command (G02, G03) for this purpose causes the CRT screen to display the following error message:

"OFFSET approach (IN/OUT) is impossible with ARC instruction."

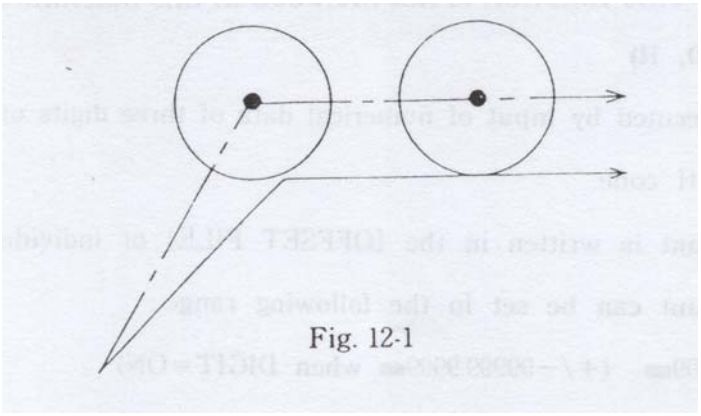


Fig. 12-1

The input of 1 in APPROACH PATTERN in the SETTING-OPERATION mode causes the offset mode to start in such a manner as illustrated below:

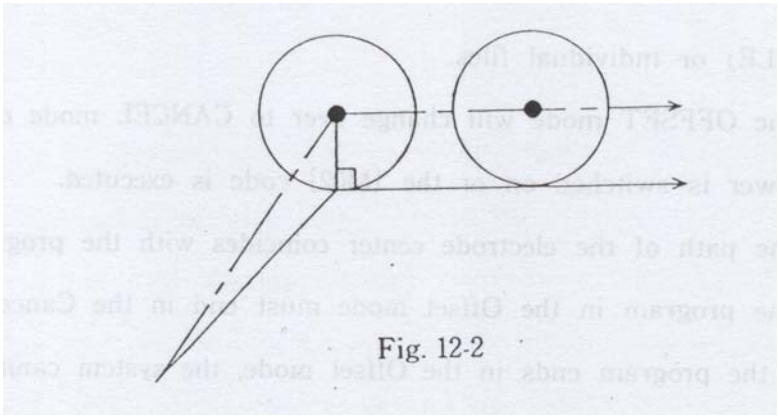


Fig. 12-2

3) End of offset mode

The execution of G40 in the offset mode causes the offset mode to be canceled.

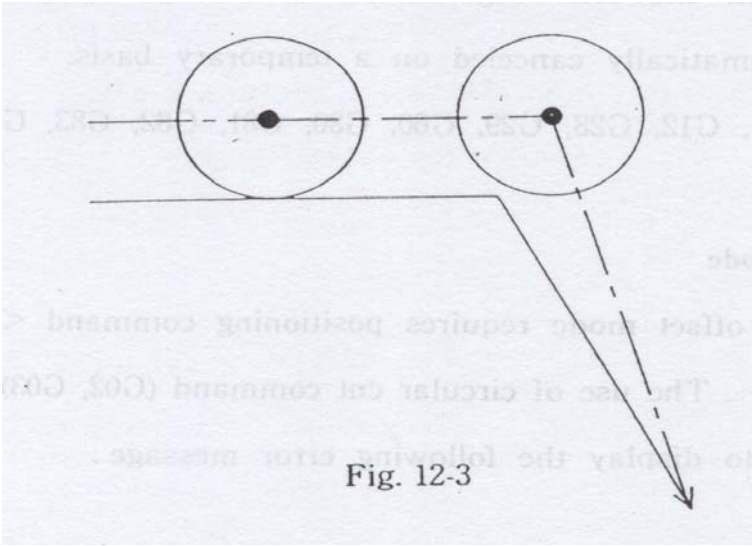


Fig. 12-3

The 'input of 1 in the APPROACH PATTERN causes the offset mode to end in such a manner as illustrated below:

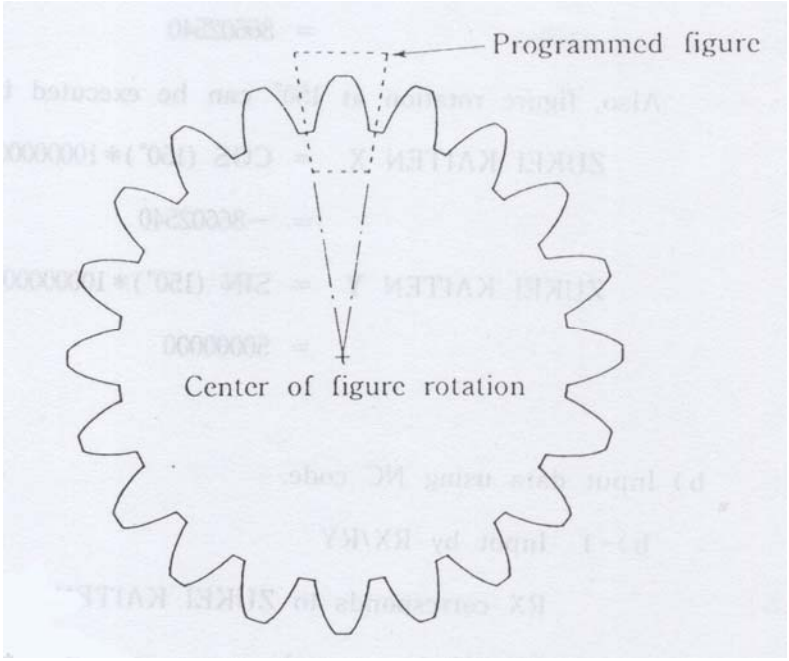
CHAPTER 13. FIGURE ROTATION AND COORDINATE ROTATION

1) Figure Rotation

(1) General description

Figure Rotation is a function to rotate a programmed shape on the specified center at a specified angle.

In order to machine such a shape as a gear, which is made up of an identical figure rotating on a given point, only the repeated figure can be programmed for rotation of this figure to program the whole shape of the gear.



(2) Codes

G26 Figure rotation ON

G27 Figure rotation OFF

RA Figure rotation angle for direct input

RX X component of figure rotation angle

RY Y component of figure rotation angle

RI X coordinate of figure rotation center

RJ Y coordinate of figure rotation center

(3) Execution of figure rotation

The execution of G26 causes the figure rotation function to be executed.

On the other hand, the execution of G27 causes this function to be canceled.

(4) The figure rotation angle can be set by the following three methods'

a) Input data in ZUKEI KAITEN X and Y in SETTING mode using the following formats '

ZUKEI KAITEN X = COS (figure rotation angle) *100000000

ZEIKEI KAITEN Y = SIN (figure rotation angle) *100000000

For example, figure rotation at 60° can be executed by the following data input'

ZUKEI KAITFEN X = COS (60°)*100000000

= 50000000

ZUKEI KAITEN Y = SIN (60°)*100000000

= 86602540

Also. figure rotation at 150° can be executed by the following data input'

ZUKEI KAITEN X = COS (150°)'100000000

=-86602540

ZUKEI KAITEN Y =SIN (150°)*100000000

= 50000000

b) Input data using NC code.

b)-1 Input by RX/RY

RX corresponds to ZUKEI KAITEN X and RY to ZUKEI I(AITEN Y.

RX/RY data are the same as determined by method a) above.

b)-2 Input by RA

RA can be used for direct input of the intended figure rotation angle. The angle input can only be made by the decimal system, which needs consideration to the fact that the input data, if input without use of a decimal point, is affected by the setting of DIGIT in the SETTING mode.

(For imput of 2.5°)

DIGIT	Notation
0	2500 or 2.5
1	125000 or 2.5

If the rotation angle has not been specified by RA or RX/RY ill the program, it is to be set by ZUKEI KAITEN X and ZUKEI KAITEN Y in the SETTING mode.

(5) Setting of figure rotation center coordinate

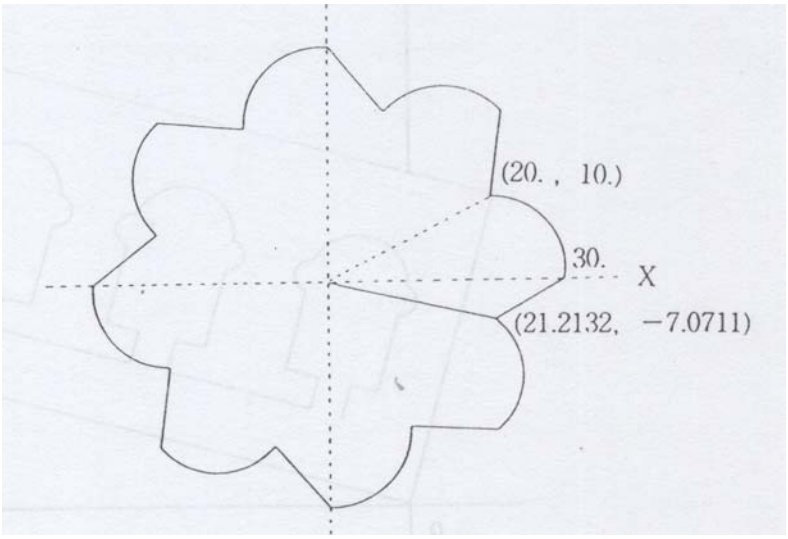
The coordinate of the figure rotation center is specified by RI and RJ.

If the center coordinate has not been specified in the program, it is to be set at the origin in the coordinate system where G26 is executed.

It should also be noted that even if the program has been created by the "Incremental" input system, the input of the rotation center coordinate (RI,RJ) must be based on the coordinate system where G26 is executed.

(6) Example of programming

```
G54  G90  G92 X0.  Y0.  Z0.'
RA45.;    (This is the same as RX70710678 RY70710678.)
RI0. R J0.'
G01 X21.2132 Y-7.0711;
M98 P0001 1.8;
G27;
M02;
N0001;
G01 X30. Y0.;
G03 X20. Y10. I-10.;
G26;
M18;
```



The above program executes the sub-program (N0001) for figure rotation at 45~ with center at the origin in G54 coordinate system.

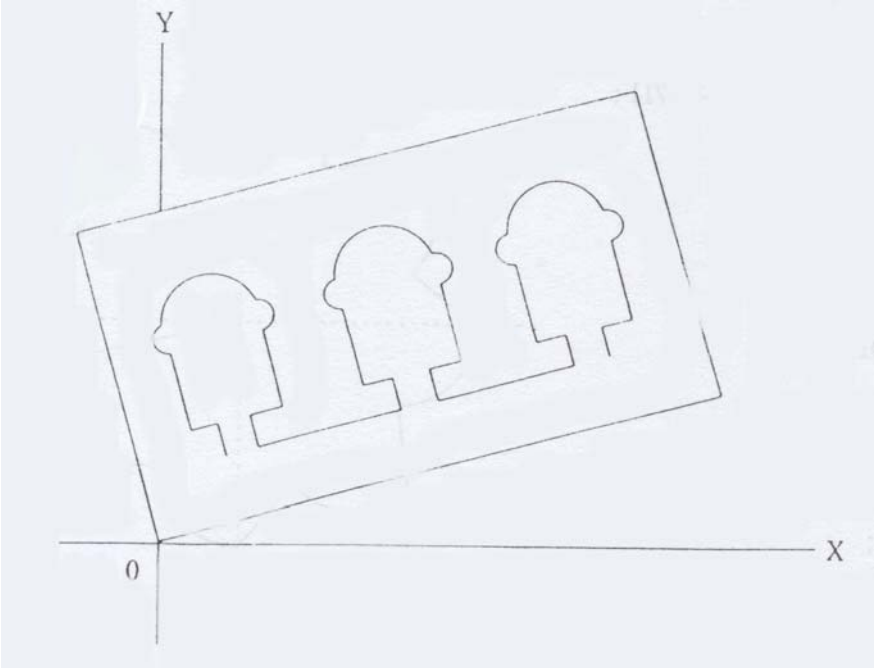
(7) Note

a) The execution of the figure rotation function requires the rotation angle (RA or RX/RY) and rotation center coordinate (RI/RJ) to be set before G26 is executed.
b) During the execution of the figure rotation function, the rotation angle and rotation center coordinate cannot be changed.
c) During the execution of the figure rotation function, G126 (coordinate rotation) cannot be executed. This causes an error to occur.
d) During tile execution of G26 for rotation of a figure under coordinate rotation, the cancellation of the coordinate rotation (G127) causes an error to occur.

2) Coordinate Rotation

(1) General description

Coordinate Rotation is a function to rotate the workpiece, set in an inclined state, on the specified center at the specified angle for offset of the inclination before execution of the program.



(2) Codes

G126 Coordinate rotation ON

G127 Coordinate rotation OFF

KA Coordinate rotation angle for direct input

KX X component of coordinate rotation angle

KY Y component of coordinate rotation angle

KI X coordinate of coordinate rotation center

KJ Y coordinate of coordinate rotation center

(3) Execution of coordinate rotation

The execution of G126 causes the coordinate rotation function to be executed. On the other hand, the execution of G127 causes this function to be canceled.

(4) The coordinate rotation angle, can be set by the following three methods:

a) Input data in ZAHYO KAITEN X and Y in SETTING mode using the following formats:

ZAHYO KAITEN X = COS (coordinate rotation angle) *100000000

ZAHYO KAITEN Y = SIN (coordinate rotation angle) *100000000

For example, coordinate rotation at 30° can be executed by the following data input:

ZAHYO KAITEN X = COS (30°)*100000000

= 86602540

ZAHYO KAITEN Y = SIN (30°)*100000000

= 50000000

Also, coordinate rotation at 110° can be executed by the following data input:

ZAHYO KAITEN X = COS (110°)*100000000

= -34202014

ZAHYO KAITEN Y = SIN (ll0°)*100000000

= 93969262

b) Input data using NC code.

b)-1 Input by KX/KY

KX corresponds to ZAHYO KAITEN X and KY to ZAHYO KAITEN Y. KX/KY data are the same as determined by method a) above.

b)-2 Input by KA

KA can be used for direct input of the intended coordmate rotation angle.

The angle input can only be made by the decimal system, which needs consideration to the fact that the input data, if input without use of a decimal point, is affected by the setting of DIGIT in the SETTING mode.

(For imput of 4.6°)

DIGIT	Notation
0	4600 or 4.6
1	46000 or 4.6

(5) Specification of coordinate rotation center coordinate

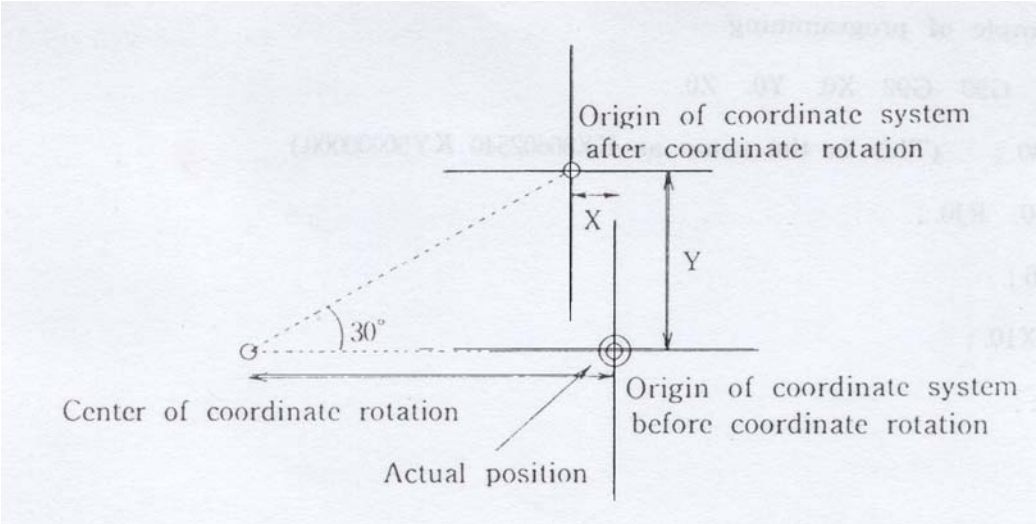
The specification of the coordinate rotation center (KI,KJ) is based on the coordinate system where G126 is executed.

If the center coordinate has not been specified in the program, it is to be set at the origin in the coordinate system where G126 is executed. It should also be noted that even if the program has been created' by the Incremental input system, the input of the rotation center coordinate (KI,KJ) must be based on the coordinate system where G126 is executed.

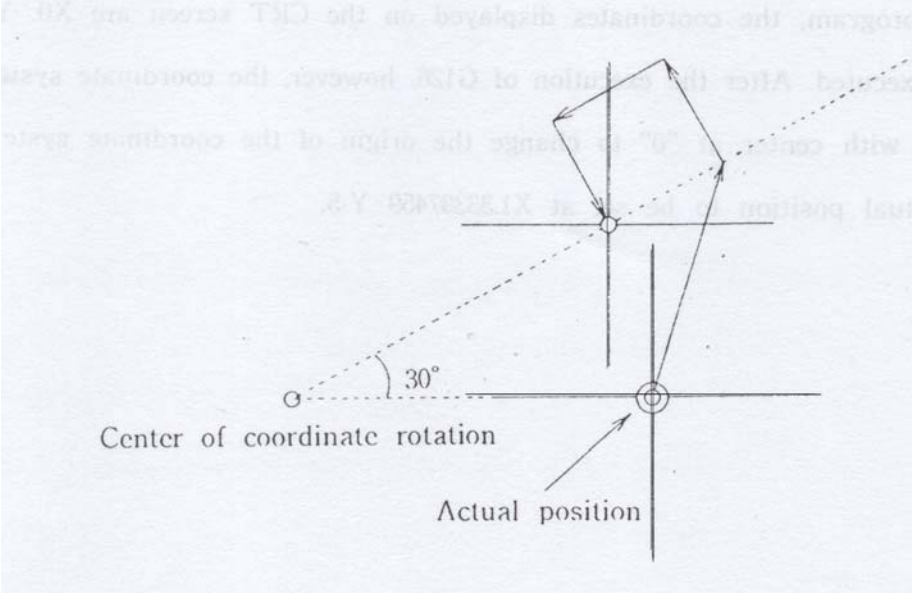
(6) Example of programming

```
G54 G90 G92 X0. Y0. Z0.;
RA30.;      (This is the same as KX86602540 KY50000000.)
RI-10. R J0.;
G126;
G01X10.;
Y10.;
X;
Y;
G127;
M02;
```

In the above program, the coordinates displayed on the CRT screen are X0. Y0. until (;126 is executed. After the execution of G126, however, tile coordinate system rotates by 30~ with center at "0" to change the origin of the coordinate %,stem, causing the actual position to be set at X1.33397459 Y-5.



The program causes the electrode to actually travel as illustrated in tile following drawing.



(7) Note

a) The coordinate rotation angle (KA or KX/KY) and coordinate rotation center coordinate (KI/KJ) must be' set before G126 is executed. The setting of these parameters after the execution of G126 causes an error to occur.

b) The execution of G126 during coordinate rotation causes an error to occur.

CHAPTER 14. INTERFERENCE CHECK

1) General Description

Machining of a groove smaller than the electrode diameter offset amount or electrode travel smaller than the electrode diameter offset amount ma>, cause the electrode to cut into the workpiece. When there is a possibility that such trouble inlay occur, the use of Interference Check function allows such interference to be avoided or to be detected as an error to stop the electrode operation.

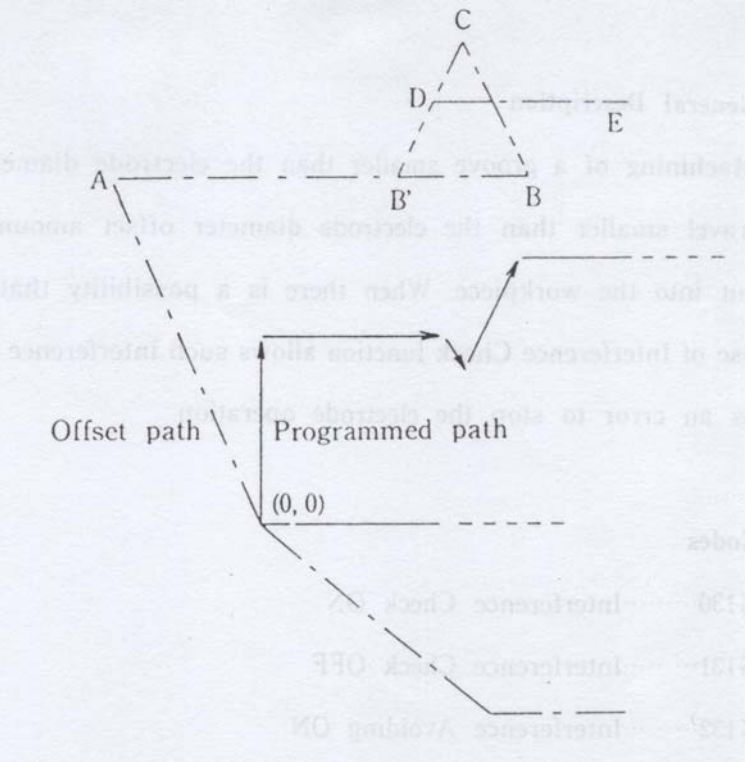
2) Codes

G130 ..... Interference Check ON
G131 ..... Interference Check OFF
G132 ..... Interference Avoiding ON
G133 ..... Interference Avoiding OFF
G136 ..... Interference Error ON
G137 ..... Interference Error OFF

3) Example of programming

Example 1

G54 G90 G92 X Y U V
G41 Ii000 (H000=4.)
G 130
G132 (or G136)
G01 Y5. ..... A
G01 X5. …..... B
G01 X5.5 Y4.5 C
G01 X7. Y7. ..... D
G01 Xl0. ..... E
G01 XS.
G40
G133(or G137)
G131
G01 X0
M02



Electrode travel path taken when INTERFERENCE AVOIDING is OFF

0--A --B--C--D--E--

Electrode travel path taken when INTERFERENCE AVOIDING is ON

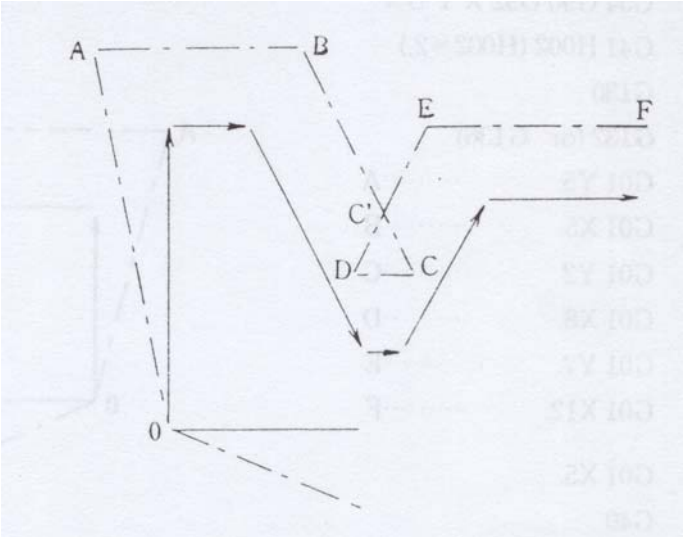
0--A--B'--D--E--

Electrode travel path taken when INTERFERENCE ERROR is ON

0--A, at which error is detected to stop electrode operation

When the offset path in the next block is in opposite direction to the programmed path, the system will skip this block for calculation of the intersection.

Example 2




```
G54 G90 G92 X Y U V
G41 H001 (H001 =2.)
Gl30
G132 (or G137)
G01 Y8.          .....  A
G01 X2.          .....  B
G01 X5. Y2.      .....  C
G01 X6.          .....  D
G01 X8. Y6.      .....  E
G01 Xl2.         .....  F
G01 X5.
G40
Gl33 (or G137)
Gl31
G01 X0.
M02
```

Electrode travel path taken when INTERFERENCE AVOIDING is OFF

0--A--B--C--D--E--F--

Electrode travel path taken when INTERFERENCE AVOIDING is ON

0--A--B--C'--E--F--

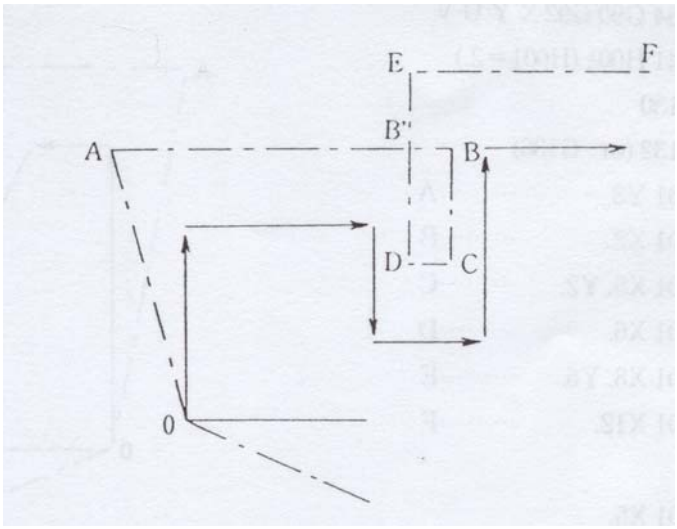
Electrode travel path taken when INTERFERENCE ERROR is ON

O- A- B, at which error is detected to stop electrode operation

When the offset path in the block to be executed and the offset path two blocks ahead of this intersect each other, the electrode travels to the intersection, from which it travels

along the offset path two blocks ahead.

Example 3




```
G54 G90G92 X Y U V
G41 H002 (H002=2.)
Gl30
G132 (or G136)
G01 Y5.          ..... A
G01 X5.          ..... B
G01 Y2.          ..... C
G01 XS.          ..... D
G01 Y7.          ..... E
G01 Xl2.         ..... F
G01 X5.
G40
G132(or G136)
Gl31
G01 X0.
M02
```

Electrode travel path taken when INTERFERENCE AVOIDING is OFF

0--A--B--C--D--E--F--

Electrode travel path taken when INTERFERENCE AVOIDING is ON

0--A--B'--E--F--

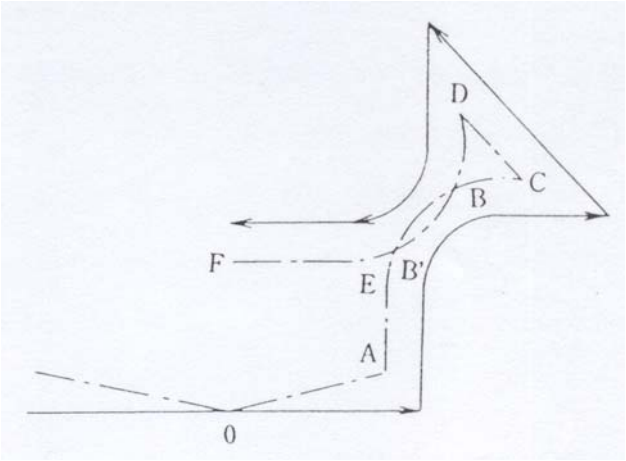
Electrode travel path taken when INTERFERENCE ERROR is ON

0--A. at which error is detected to stop electrode operation

When the offset path in the block to be executed and the offset path three blocks ahead of this intersect each other, the electrode travels to the intersection, from which it travels along the offset path three blocks ahead.

Example 4

```
G54 G90 G92 X Y U V
G41 H004 (H004 = 1.)
Gl31
G132 (or G136)
G01 XS.          .....  A
G01 Y5. R2. R2. .....  B
G01 Xl0.         .....  C
G01X5. Y10.      .....  D
G01 Y5. R2. R2. .....  E
G01 X0           .....  F
G01 X-5.
G40
G133 (or G137)
Gl31
G01 X0
M02
```



Electrode travel path taken when INTERFERENCE AVOIDING is OFF

0--A--B--C--D--E--F--

Electrode travel path taken when INTERFERENCE AVOIDING is ON

0--A--B'--E--F--

Electrode travel path taken when INTERFERENCE ERROR is ON

0--A, at which error is detected to stop electrode operation

When electrode path interference occurs at Comer R, the interference avoiding function

can also be operated in a similar manner.

4) Note

Use the Intervention Check function in a program consisting of not less than four blocks.

CHAPTER 15. M03

1) General Description

When arcing is detected during machining with 15000 pulses produced from tile electrode tip, automatic cleaning is executed as many times as specified in AUTOMATIC CLEANING, a parameter provided in the SETTING' DISCHARGE submode. However, if arcing is detected again, the system will continue to search for M03 without executing the program (when "M03" in the SETTING-FLAG submode has been set at 1 or more.)

[Example]

```
G01 Z-10.000; (roughing)    .....    (a)
GOO M05 Z1.OO0;
G01 Z-10.100; (finishing)
M03;                        .....    (b)
T01;
Z10.;
GOO X20.000Y50.OO0; (travel to the next machining point)
G01 Z-10.000; (roughing)
```

Note: When arcing occurs during the execution of (a), the system will skip the subsequent program blocks without executing them until it finds (b) to execute the program starting the block following M03.

Ill this way, the use of <M03> can stop the machining ill the middle of program execution to make it proceed to the next process corresponding to the program block following this code. However, considering that there should be the cause of arcing, it is recommended that <M03> be used 'in combination with ATC or the like.

If arcing takes place, the cursor will automatically moves downwards to search for M00, M02 or M03. At this time, set the electrode coordinate at "0" and feed it to the limit.

If M00 or M02 is found, the system returns to the program block, where the

arcing took place, causing the CRT screen to display "Arc Stop".
If the program has been edited using G91 (“Incremental” input system), avoid the use of <M03> code, which results in positioning failure.
As in the case of "Arc Stop", "Limit Stop" or "ST Stop" also causes the system to search for M03 to make it execute the program starting with the block following M03.

The M03 searching pattern can be specified by setting "M03", a parameter in the SETTING' FI.AG submode as shown below.

[Setting of M03 ill "SET' FI.AG" submode]

0 : Docs not search for M03 in any case.
1 : Searches for M03 when Arc Stop occurs.
2 : Searches for M03 when Arc Stop or Limit Stop occurs.
3 : Searches for M03 when Arc Stop or ST Stop occurs.
4 : Searches for M03 when Arc Stop or Limit Stop/ST Stop occurs.
5~9 : Not ill use.

M03	ARC (EDM) STOP	LIMIT STOP	ST STOP
0	×	×	×
1	○	○	×
2	○	×	×
3	○	×	○
4	○	○	○

This function, even if such an error or trouble as normally resulting ill temporary stop of the program occurs, causes its execution not to be discontinued, but continued from the block following M03, and thus can be effectively used ill programs such as one for multi-cavity machining.
While the system searches for M03, the comment to this effect will be displayed on the screen, disappearing when a M03 block is read and an error occurs.
When the M03 block is not found, an error occurs, causing the program execution

to be discontinued.

When such an error takes place, the cursor points to the block, in which a trouble resulting in M03 search occurred.

2) Identification of trouble resulting in M03 search

When Arc Stop, LIMIT STOP, ST STOP or other stop state resulting ill M03 search occurs, the NC unit memorizes the stop state. There fore. the input of G83 "S" code in the block following M03 code allows the cause for M03 search to be identified. The stop states resulting in M03 search which are to be read in the compensation term are as follows:

Data in compensation tern	Stop state resulting in M03 search
161	LIMIT STOP
162	ST STOP
164	ARC STOP
165	SOFTWARE LIMIT STOP
170	STROKE STOP

[Example]

M03

G83 S001

IF H001 = 161 (1000, 1001)

The above example causes the system to skip the program to the block of M03 code and record the stop state in H001 for judgment to specify the JUMP destination. This function allows the program to be processed according to the stop state.

Note :The stop state of M03 memorized ill the NC unit is cleared to "0" if the following occurs :

1. If the program ends with M02, %
2. If the OFF key is pressed
3. If an critical' error takes place
4. If the stop state is read once in the compensation term using G83 "S" code

CHAPTER 16.HOW TO USE SOFTWARE LIMIT

1) SOFTWARE LIMIT ON/OFF

G22 ON

G22 OFF

2) Input of numerical data

Move the cursor to input numerical data in "SOFTWARE I.IMIT", a parameter in

SETTING-OPERATION submode.

SOFTWARE I.IMIT (X+) =+010000000

SOFTWARE I.IMIT (X-) =-010000000

SOFTWARE LIMIT (Y+) =+010000000

SOFTWARE I.IMIT (Y-) =010000000

SOFTWARE LIMIT (Z+) =+010000000

SOFTWARF. IAMIT (Z-) =-010000000

SOFTWARE I.IMIT (U+) =+010000000

SOFTWARE I.IMIT (U-) =-010000000

SOFTWARE I.IMIT (V+) =+010000000

SOFTWAI;~E I.IMIT (V-) =-010000000

SOFTWAI~,E I.IMIT (W+) =+010000000

SOFTWARE I.IMIT (W-) =-010000000

SOFTWARE I.IMIT (UU+)=+010000000

SOFTWARE I.IMIT (UU-)=-010000000

SOFTWARE I.IMIT (VV+)=+010000000

SOFTWARE I.IMIT (VV-)=-010000000

(3) If tile numerical data input is completed, press ENT key.

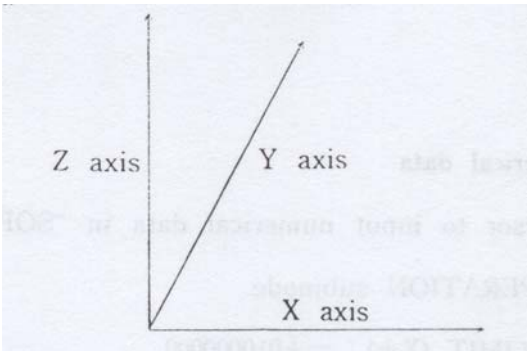
(4) Select another mode.

(5) At the time o[the mode change, the "SOFTWARE LIMIT" is set to the input numerical data.

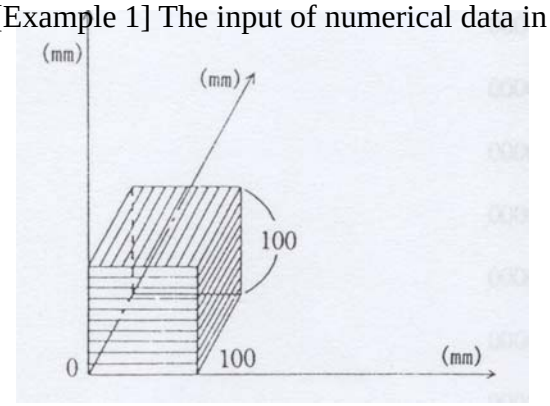
3) How to select numerical data input in SOFTWARE LIMIT

The SOFTWARE I.IMIT is set in the machine coordinate for each axis, therefore

requiring input of numerical data in positive form. which are, however, initialized to +1000000 and -10000000 in the NC unit.



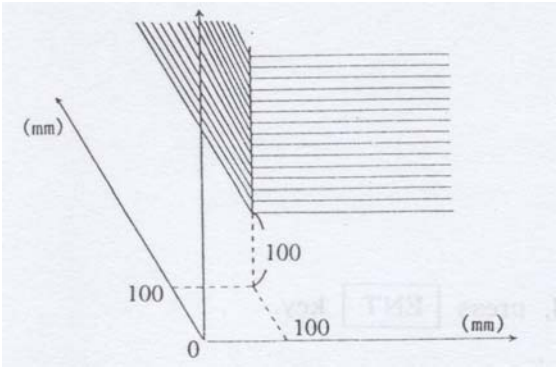
[Example 1] The input of numerical data in SOFTWARE LIMIT as in the following:



SOFTWARE LIMIT X (+)=+1000000
SOFTWARE I.IMIT X(-)=+0
SOFTWARE I.IMIT Y (+)= +10000()o
SOFTWARE LIMIT Y(-)=+0
SOFTWARE I.IMIT Z (+)=+1000000
SOFTWARE LIMIT Z (-)=+0

causes only the inside of the cube to be defined as an electrode travel area. The execution of G22 in this area allows the electrode to travel only in the cube, but stop with STROKE STOP when it will travel out of the area.

[Example 2](mm) The input of numerical data in SOFTWARE I.IMIT as in the following:



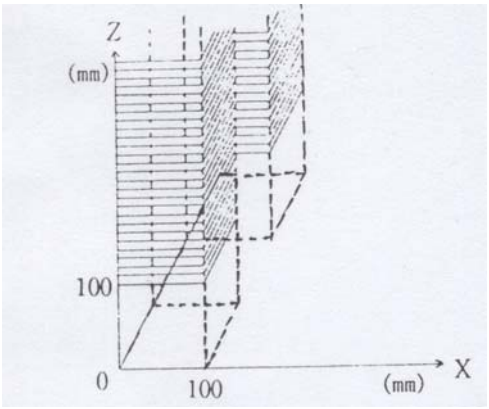
SOFTWARE LIMIT X(+)=+0
SOFTWARE LIMIT X (-)= + 1000000
SOFTWARE LIMIT Y(+)=+0
SOFTWARE LIMIT Y (-)=+1000000
SOFTWARE LIMIT Z(+)=+0
SOFTWARE LIMIT Z (-)= + 1000000

causes only the shaded portion in the drawing at left to be defined as an electrode travel area under the following condition:

X (+) < X (-)
Y (+) < Y (-)

Z (+) < Z (-)

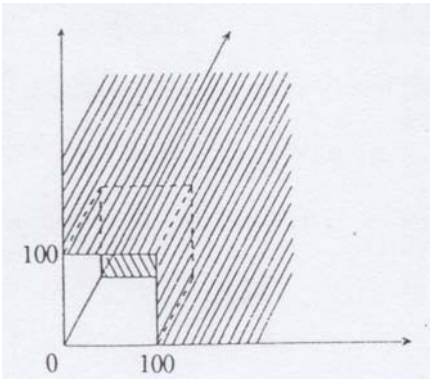
[Example3] The input of numerical data in SOFTWARE LIMIT as in the following:



SOFTWARE LIMIT X (+)= +1000000
SOFTWARE LIMIT X(-)=+0
SOFTWARE LIMIT Y (+)= +100000~)
SOFTWARE LIMIT Y(-)=+2000000
SOFTWARE LIMIT Z(+)=+0
SOFTWARE LIMIT Z(-)=+1000000

causes the electrode travel area to be changed as in the drawing at left.

[Example 4] Complete exchange of the electrode travel area with the electrode non-travel area can be achieved by data exchange between X (+) and X (-), Y (+) and Y (-), and Z (+) and Z (-) with SOFT LIMIT INOUT of SETTING ~ OPERATION submode set at "1". For complete exchange of the electrode travel area with the electrode non-travel area, SOFTWARE LIMIT can be set as follows:



SOFTWARE LIMIT X(+)=+0
SOFTWARE I.LIMIT X (-):+1000[)00
SOFTWARE I.LIMIT Y(+)=+0
SOFTWARE LIMIT Y (-)= + 100000O
SOFTWARE I.LIMIT Z(+)=+0
SOFTWARE I.LIMIT Z(-)=+1000000

*To allow the electrode to escape from the non-travel area, operate the JOG key while pressing the ACK key.

Note: The G type, which is designed so that "SOFT LIMIT INOUT" in the SETTING ~ OPERATION submode cannot be used, only allows the "SOFTWARE I.LIMIT" function to I)e used in such a way as illustrated in Examples 1, 2 and 3 above.

CHAPTER 17. Q COMMAND

Q command allows tile specified file to be executed.

The format for Q command is as follows:

Q {File name} (Parameter)

(1) (2) (3)

(1) When Q command is executed in one block, it must be input at the beginning of the block.

(2) The file name can be input using a maximum of 8 characters. If more than 8 characters are used for the input, the ninth character and the following will be ignored. ,

If the specified file name is not found in the system, an error occurs, causing the CRT screen to display the comment indicating the occurrence of the error.

(3) Transfer of parameters is made with data enclosed in parentheses as parameters.

(,) is used for delimitation between the parameters.

The parameters are input in sequence starting with the first H code in the specified file, irrespective of the magnitude of the H code number. Method of return from Q file The execution of the last block of the Q file causes the system to return from the Q file to the program, from which it was called up.

M02 and M99 cannot be used for this purpose. Therefore, use such a method as described in the following example or input a JUMP statement (see CHAPTER 25) for jump to the last block of the program.

[Example]

:

JUMP .34.56

:

:

N3456

[Comments]

a) "Q command nesting exceed seven levels."

The above error message is displayed when the multi-fold nesting of the Q command leaches eight level.

b) "Q command format is wrong."

The above error message is displayed when Q command is used ill a wrong way.

c) "Q command file is wrong."

The above error message is displayed when the Q command-specified file is not found in the system.

[Example of Use of Q Command]

(1610) Alignment of workpiece comer with electrode approach face (lst quadrant)
(1610);

H000 = +00020000 I1001 = +00010000 11002 = +00000000
H003 = +00000000 11004 = -00000000 11005 = +00001000
M98 P1610:
M98 P1611;
N1610 G90 G59;
G80 Z-;
G92 XYZ;
M05 G00 Z+H005;
X + H000;
Z - H001;
G80 X- ;
G58 G92 X;
M05 X+II005;
G59 Z+II005;
X;
Y+II000;
Z-It001;
G80 Y-;
G58 G92 Y;
M05 Y+II005;
G59 Z +It005;
G58 X-II002Y-H003;
G59 G92 X Y'
M99;
N1611 M99 PI612-
N1612;

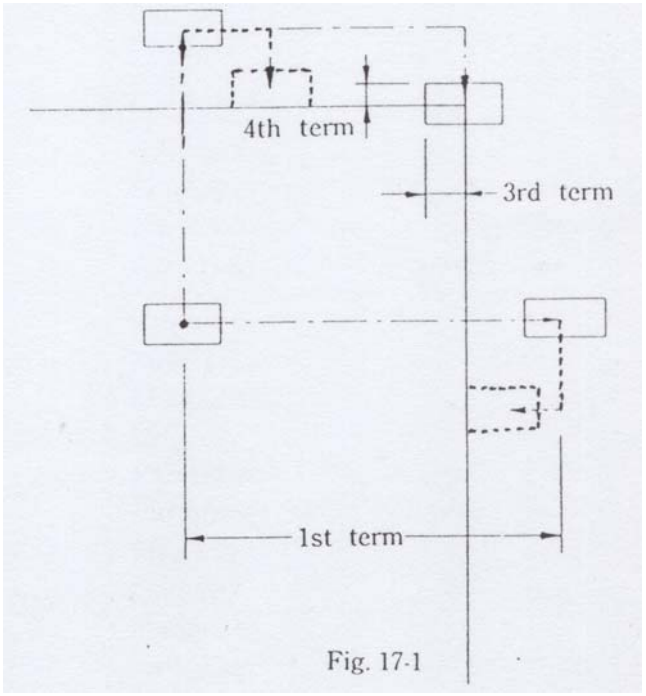


Fig. 17-1

1st term li0(g) : Stroke for escape out of X+/Y+ side workpiece approach face
2nd term Ii001 : Downward stroke for detection of Z axis approach face
3rd term 1-1002 : Stroke for alignment with approach face, followed by X axis (-) feed stroke
4th term 11003 : Stroke for alignment with approach face, followed by Y axis (-) feed stroke
11005 : X/Y/Z axis inversion

1610 is a program to cause the workpiece XY reference surface to come into contact with tile electrode approach face in order to detect the origin for positioning of the electrode on tile workpiece.

(1) The electrode is positioned on the workpiece at the comer of the 1st quadrant so that both X and Y axes are out of the workpiece approach face within the stroke range of the 1st term.
(2) When the electrode starts to operate, it is fed in the Z axis (-) direction for

Detection of its contact with the workpiece.

(3) After the electrode is fed in the X axis direction for detection of its contact with The workpiece, it travels into the workpiece as specified in 3rd term X axis and 4th term Y axis at the height of Z axis H005 (1mm) for G09 (A5) origin setting.

There fore, if the 3rd and 4th terms remain at “0” as in the basic mode of the Q memory list, the origin is to be set at the tangent common to both approach faces.

CHAPTER 18. CALCULATION

Code and data following address can be represented by using an expression.

1) Kinds of calculation

Operators and functions that can be used in such an expression vary according to the kind of word used.

A	N, O	Expression cannot be used.	
B	B, C, D, G, H, L, M, P, T	+ − [ ] H, D	Addition Subtraction Parenthesis Parameter using compensation term
C	A, F, I, J, K, R, S, U, V, W, X, Y, Z UU, VV Right side of assignment statement	+ − * / [ ] H, D SIN COS TAN ASIN ACOS ATAN SQRT ROUND	Addition Subtraction Multiplication Division Parenthesis Parameter using compensation term Sine Cosine Tangent Arc sine Arc cosine Arc tangent Square root Rounding off

[Example] G90 G01 X1000+Hi23;

If the value of H123 is .3000, the X axis travels to 4000.

2) Order of priority in calculation

1. Parenthesis [ ]
2. Parameter H, D
3. Sign +, -
4. Function SIN, COS, TAN, ASIN, ACOS, ATAN, SQRT, ROUND
5. Multiplication/division *, /
6. Addition/subtraction +, -

3) Format of calculation

1. The expression length must be within one block.
2. The parenthesized nesting must not exceed 19 times.

4) Unit system of calculation

1. If "NEW CALCUI.ATION' in SETTING. OPERATION submode is OFF (with TAN,ASIN and ACOS not supported)

(1) For calculation classified as "B" Only natural numbers are supported.
Decimal notation causes an error to occur.

(2) For calculation classified as "C"

1) SIN, COS, ROUND function
Integer is converted into data using decimal notation for calculation.
1 is equivalent to 0.001. (DIGIT=OFF, Meter, Degree)

2) Function other than SIN, COS, ROUND
Data input using decimal notation is converted into integer for processing.
1.0 is equivalent to 1000. (DIGIT=OFF, Meter)
1.0,1.0 is equivalent to 1000.0. (DIGIT=OFF, Meter)
It should be noted, however, that internal calculation is based oil decimal notation.[100/3]+[100/3] is not equivalent to 66, but to 67.

2. If "NEW CAI.CUI.ATION" in SETTING-OPERATION submode is ON

(1) For calculation classified as "B"
Only natural numbers are supported.
Decimal notation causes an error to occur.

(2) For calculation classified as "C"
Input numerical data is divided into data with units such as distances and angles and data without units such as coefficients and ratios for processing to give significant values.
Numerical data with units (distance/angle):
Data input using decimal notation are equivalent in scale to 1000 times data input using integral notation. (DIGIT=OFF, Meter)

Numerical data without units (coefficient/ratio)

Data input using decimal notation are equivalent in scale to data input using integral notation.

(1) Tile multiplicand and dividend of multiplication and division at tile lowest level are with units.

1.0 is 1mm. (Meter)

1 is 1xm. (DIGIT=OFF, Meter)

This means that 1.0+1 equals 1.0~+1~, which is 1.001mm.

1.0 is 1 degree. (Degree)

1 is 0.001 degree. (DIGIT=OFF, Degree)

This means that 1.0+1 equals 1.0 degree +0.001 degree, which is 1.001 degree.

(2) The multiplier and divisor of multiplication and division at the lowest level are coefficients without units.

1.0 is equivalent to 1.

This means that 1.0+1 is equivalent to 2.0 and 2.

1/5 is equivalent to 0.2.

(3) 1)and 0 above show that the value obtained from the expression involving these calculations is with units.

15.0.0.5 is equivalent to 7.5,

where 15.0 is recognized as 15.0mm (Multiplicand : with unit), 0.5 is recognized as 0.5 times (Multiplier : coefficient) and 7.5 naturally indicates 7.5mm.

(4) SIN, COS and TAN are angles with units.

SIN 30.0 is equivalent to SIN 30000. (DIGIT=OFF, Degree) SIN 30.0 is equivalent to SIN 300000. (DIGIT=ON, Degree)

(5)ASIN, ACOS and ATAN are ratios without units.

ATAN 1.0 is equivalent to ATAN 1.

[Example] (DIGIT=OFF, Meter, Degree)

2.*3.0 -- 6tm	2*3.0 -- 6u~
2.*3 --.6tm	2*3 -- 6~
1.5*80 --120tm	80*1.5 -- 120~

[0.8+0.7] *80 -- 120mm 80* [0.8+0.7] -- 120urn

In the following example, [0.8+2] in the 1st term is a coefficient and [0.8+2] in the 2nd term is with a unit.

1000. [0.8+2] + [0.8+2] -- 1000urn* [2.8] +0.802mm

in the following example, [25.0+5000] is with a unit.

40.0 * SIN [25.0 + 5000] -- 40.0. SIN [30.0~]

In the following example, [25.0+5000] is a ratio.

ATAN [25.0 +5000] -- ATAN [5025.0]

(6) Input data with units is automatically identified from the program for division into distances and angles, although, if unidentifiable in this way, the data is processed as distances.

[Example] (DIGIT=OFF, Meter, Degree/Minute/Second)

X [10.0+ 10] -- X [10.010] Distance

SIN [10.0+10] ~ SIN [10.0010] Angle

H000= [10.0+ 10] -- H000= [10.010] Unidentifiable

(7) The rules in (!) and O above hold for input of SIN, COS and TAN.

[Example] (DIGIT=OFF, Meter, Degree)

SIN [5.0.30] -- SIN [150.0]

SIN [5 * 30.0] ~ SIN [0.15]

5) Factors affecting results of calculation

1. Setting of number of digits below decimal point

The number of digits below the decimal point is set by "DIGIT" and "INCH/METER".

This setting affects the difference in scale between data with units input by decimal and integral notation.

3 dig is :Data input by decimal notation has a scale 1000 times that of data input by integral notation.

4 dig is :Data input by decimal notation has a scale 1000 times that of data input by integral' notation.

5 dig is :Data input by decimal notation has a scale 100000 times that of data input

by integral notation.

2. Unit system of angle for input notation The unit system of angle for input notation is set by "DISPLAY".

It should be noted, however, that this setting for data of address A is made I)5"' Degree/minute/second

:Angle data input by decimal notation has a scale 10000 times that of data input by integral notation, irrespective of the above-mentioned setting of the number of digits below the decimal point.

Degree :The scale of input angle data is affected by the above-mentioned setting of the number of digits below the decimal point.

Radian :The scale of input angle data is affected by the above-mentioned setting of the number of digits below the decimal point.

Degree :The scale of input angle data is affected by the above-mentioned setting of the number of digits below the decimal point.

No. of revolutions

:The scale of input angle data is affected by the above-mentioned setting of the number of digits below the decimal point.

No. of pulses

:The scale of input angle data is affected by the above-mentioned setting of the number of digits below the decimal point.

6) Errors concerning calculation

1. If the parenthesized nesting exceeds 19 times, the following error message will appear on the screen'

"Parenthesized nesting exceeds 19 times."

2. If improper parentheses opening/closing or other improper calculation formals are detected, the following error message will appear on the screen'

"NC program format is incorrect."

3. If division is executed with divisor as '0", the following error message will appear on the screen:

"Division by zero."

4. If ally digit omission occurs, the following error message will appear on tile screen:
   "Numerical data are too large (small)."
5. If calculation data exceed the specified limit, the following error message will appear on the screen:
   "Data exceed limit during processing."

7) Note

Rounding of to a unit or ROUND operation is to be generally considered to correspond to "counting fractions over ~ as a unit and disregarding the rest".

The reason for the use of expression "generally" in the above statement is that the ROUND operation does not always provide for complete agreement with calculation of "counting fractions over ~ as a unit and disregarding the rest" made in the human brain.

Take the following operation as an example of the above:

SIN [30.0] (the unit of 30.0 is ..... .)

sin 30~ is 0.5, which, if rounded off to the nearest whole unit, becomes 1. In the NC unit, however, this calculation is made by expressing 30~ in terms of radian.

When 30~ is expressed in terms of radian, it gives a number impossible to round off, producing a very small error due to its rounding-off. Accordingly, sin,30~, when calculated on a radian basis, may become 0.4999999 ..... . This value, when rounded off to the nearest whole unit, becomes 0.

For the above reason, the operation result may cause a NC program for control of the NC program flow to be created in an unexpected direction. There is also a possibility that the same NC program may flow in a different direction depending on the NC version. For it is not guaranteed that the rounding-off of a very small error produced in the NC unit is controlled in the same manner in all the NC versions. Not only from the above, but also from the standpoint of future achievement of NC program interchangeability, a NC program for control of the NC program flow based on the operation results should be created with good care.

CHAPTER 19. USER MACRO FUNCTION

1) General Description

This function is used to change the order in which NC programs are executed, to make the system stand ready for operator's key input and to cause the CRT screen to display a selected character string.

2) Command changing program flow

(1) IF statement

This command causes the NC program flow to change according to a certain condition if this condition is met. The command format is as follows:

IF H*** < H*** (destination for YES to the IF statement, destination for NO to the IF statement)

(Example)

```
G91
IF  H010 ~  H011  (1111,  2222)
N111
G01 Xl0.
M02
N2222
G01 Y10.0;
M02
```

The program jumps to the block of Nl111 for cutting to Xl0.0 if H010=<H011 and to the block of N2222 for cutting to Y10.0.

(2) JUMP statement

This command changes the program flow unconditionally. The command format is as follows'

JUMP 4-digit sequence No. of destination

(Example)

G91..... (1)
N1111..... (2)
JUMP3333..... (3)
N2222..... (4)
GO8..... (5)
N3333..... (6)
G01 Xl0..... (7)
M02..... (8)

The program skips over (~) and ~ to execute the program ill the order of (1),(2),(3),(6),(7),(8).

(3) KEYIN statement

This command causes the system to stand ready for operator's key input, bringing the input data into the specified parameter. The command format is as follows'

KEYIN (H***); (***indicates any given value of three digits or less.)

The execution of the command causes the CRT screen to display "KEYIN H***", indicating that the system stands ready for operator's key input.

Therefore, operate the keyboard to input data and then press the [ENT] key to bring the data into H*** for restart of the program execution.

(Example)

```
H011 = +00000011
G91
KEYIN (H010)
IF H010 ~> H011 (1111, 2222)
N1111
G01X10.0
M02
N2222
G01Y10.
M02
```

The program jumps to the block of Nl111 for cutting to Xl0.0 if a number larger than 11 is input in KEYIN H010= and to the block of N2222 for cutting to Y10.0 if a number not larger than 11 is input in this parameter.

(4) CRT statement

This command causes the CRT screen to display a selected character string of 24 characters or less~ The command format is as follows:

CRT (Character String);

The character string is displayed on the screen until the next CRT statement command is executed. The displayed character string can also be cleared when M02 is executed or ACK key is input as an error occurs.

CRT statement of more than 24 characters is ignored.

(Example)

```
G91
N1111
CRT (N1111)
G04 XS.0
M02
```

The program causes N1111 to be displayed on the screen.

(5) PRINT statement

This command causes the specified data to be output to a printer through a serial interface.

The output data is specified by the following three formats'

Character String-Outputs the specified character string directly.

H***-Outputs contents of H***.

(*** indicates character string of eight digits or less.)

:***-Outputs NC program with a file name ***- ~

(,, * indicates character string of eight digits or less.)

PRINT (145); Outputs 145.

PRINT (H145); Outputs contents of H145.

PRINT (' 145); Outputs NC program with a file name of 145.

CHAPTER 20. INITIAL SETTING OF G AND T CODES

When the following operations are executed, the program codes are set according to the
patterns shown in the table below:

(I) When program is executed to file end

(II) When "%" or "M02" is read

(iii) When "OFF" key is pressed or critical error occurs, which requires pressing of "ACK"
key, during program execution

Pattern / Operation	A	B	C	D	D'	E
Execution of program to file end	○	×	×	×	×	Error if not canceled
OFF/Error	○	○	×	△	▲	○
M02/%	○	○	○	△	▲	Error if not canceled
	G74	☆G00	T80	G05	G20	G40
	☆G75	G01	T81	G06	G21	G41
	G104	G02	T82	G07	G90	G42
	☆G105	G03	T83	G08	G91	☆G50
		G13	T84	☆G09		G51
		☆G14	T85	G11		G52
		G15	T86	☆G12		☆G130
		G26	T87	G22		G131
		☆G27		☆G23		☆G132
		G126		Work coordinate system selection		G133
		☆G127				☆G136
				G93		G137
				G94		☆G140
				G95		G141
				☆G96		G142

(☆ : Default)

x..... Retain set state (IN).

O..... Return to initially set state (CANCEL or OFF).

● ..... Initial state (FLAG)

△ ..... Reset by .CANCEL FLAG

▲ ..... Initial state (FI.AG) by CANCEL FI.AG

CHAPTER 21. OPERATING FUNCTION DURING MOO (M01) PROGRAM ST01)

During MOO (M01) program stop, the following operations are set in OFF state. (At tile time of restart, they are restarted to their original states to execute the machining again.)

DISCHARGE

FI.USH

SUCTION

During MOO (M01) stop, the following operations can be executed.

@ FLUSH ON/OFF

@ SUCTION ON/OFF

@ PUMP ON/OFF

@ AUTO DRAIN ON/OFF

@ JOG Operation

The JOG operation during M00(M01) stop requires the following points to be noted:

@ All the motor axes except the Z axis return to their respective positions, at which they stopped due to MOO (M01), before restarting the machining.'

@ The operation of the Z axis at the time of restart varies according to the setting of Z RETURN in SETTING-OPERATION submode.

Operation of Z Axis at Restart

Z RETURN	HALT	M00	Others
0	○	○	○
1	○	×	○
2	×	○	○
3	×	×	○

○ : The axis returns to the position, at which it stopped due to M00 (M01), before restarting the machining.

× : Restarting machining from position where moved by JOG after M00(M01) stop.

@At restart after SINGLE STOP or STOP, all the axis travel to their respective positions specified by JOG operation before restarting the machining.

@The operation of the axis after its temporary stop state except the above is the same as in the case of HALT STOP.

CHAPTER 22. FLAGS CLOSELY RELATED TO CODES

(Description of flag data)

(1) N-STOP, N. DATA

These flags are used to determine whether or not to stop the machining on a temporary basis when the specified sequence No. is found. The temporary stop of the machining is selected by N-STOP (0=OFF, I=ON), while the sequence No. specification is made by N. DATA.

(2) SINGLE

This flag is used to determine whether to stop the machining on a temporary basis for each NC program block. When SINGLE is set in ON state, the machining stops temporarily on a block basis.

(3) X-Y CHANGE

Setting this flag in ON state causes the command for the X axis to be processed for the Y axis and the command for the Y axis to be processed for the X axis.

This flag causes the electrode to operate in the same way as when G08 is executed.

(4) MIRROR (X, Y, Z)

This flag, when set in ON state for an axis, causes the electrode to travel on tile axis in the direction opposite to that specified. For example, when the flag is set as follows:

MIRROR X--OFF
Y =ON
Z—OFF

the execution of G01. Xl0. Y20. Z30.

causes the electrode to travel in the direction of Xl0. Y-20 Z30. The above program causes the electrode to operate in the same way as when G06 is executed.

(5) SCALE

This flag is used to set the scale factor for multiplication of the travel specified by a NC program.

The scale factor is set at "1" when 1000 is input in the flag and at "1/2" when 500 is input in the flag.

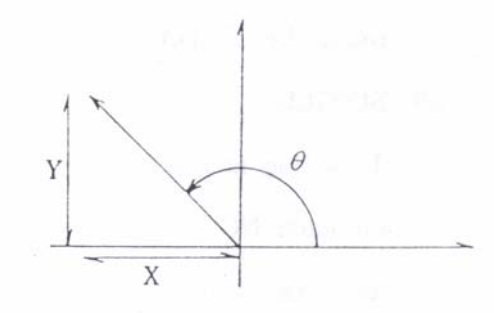
(6) FIGURF. (X, Y)

This flag is used to set the angle of figure rotation.

The relationship between this rotation angle and (X, Y) is as follows:

=tan ' ("/~)

It should be noted, however, that ~ and Y may require addition of +/- sign depending on the rotation angle.



In the example given in the drawing at right, X=-x and Y=y. The execution of G26 causes the Figure Rotation mode to be set in ON state, while the execution of (;27 causes the Figure Rotation mode to be set in OFF state.

(7) COORDINATE (X, Y)

This flag is used to set the angle of coordinate rotation.

The setting of the rotation angle can be made in the same manner as in the case of (6) FIG[JRE.

(8) OPTIONAL STOP

This flag is used to determine whether or not 'to stop the machining on a temporary basis by M01 code as in the case of MOO code. The execution of MO] code with OPTIONAL STOP set in ON state causes the machining to be stopped on a temporary basis.

(9) TAI'F.R IGNOI~E

This flag is used for cancellation of Plane Selection function.

0 'The function is canceled only when Cancel code (Gl7) is executed.

1 :The function is canceled when Cancel code (Gl7) and M02 are executed.

2 :The function is canceled when Cancel code (Gl7) and M02 are executed Ol- all error takes place.

(10) AWT REFERENCE

This flag is used for cancellation of the Coordinate Rotation function.

0 :The function is canceled only when Cancel code (G127) is executed.

1 :The function is canceled when Cancel code (G127) and M02 are executed.

2 :The function is canceled when Cancel code (G127) and M02 are executed or an error takes place.

(11) EDGE CONTROL CANCEL

This flag is used for cancellation of the Edge Control function.

0 :The function is canceled only when Cancel code (G49) is executed.

1 :The function is canceled when Cancel code (G49) and M02 are executed.

2 'The function is canceled when Cancel code (G49) and M02 are executed or all error takes place.

(12) COORDINATE CANCEL

This flag is used for cancellation of the work coordinate system.

0 :The system is canceled only when 'Cancel code (G54) is executed.

1 :The system is canceled when Cancel code (G54) and M02 are executed.

2 :The system is canceled when Cancel code (G54) and M02 are executed or all error takes place.

(13) MIRROR CANCEL

This flag is used for cancellation of the Mirror function.

0 :The function is canceled only when Cancel code (G09, G96) is executed.

1:The function is canceled when Cancel code (G09, G96) and M02 are executed.

2:The function is canceled when Cancel code (G09, G96) and M02 are executed or an error takes place.

(14) X-Y CHANGE CHANCEL

This flag is used for cancellation of the XY Exchange function.

0:The function is canceled only when Cancel code (G09) is executed.

1:The function is canceled when Cancel code (G09) and M02 are executed.

2 :The function is canceled when Cancel code (G09) and M02 are executed or an error takes place.

(15) SKIP CANCEL

This flug is used for cancellation of the Skip function.

0 :The function is canceled only when Cancel code (Gl2) is executed.

1 :The function is canceled when Cancel code (Gl2) and M02 are executed.

2 :The function is canceled when Cancel code (Gl2) and M02 are executed or an error takes place.

(16) ABS/INC CANCEL

This flag is used to determine whether or not to bring the state of (ABS/INC) into the state set by INIT G91 when an error occurs or M02 is executed.

0 :The present state is maintained until (ABS/INC) selection code (G90, G91) is executed.

1 : The present state is canceled when (ABS/INC) selection code and M02 are executed.

2 : The present state is canceled when (ABS/INC selection code and M02 are executed or an error takes place.

(17) SOFTWARE I.IMIT CANCEL

This flag is used for cancellation of the Software Limit function.

0 :The function is not canceled unless Cancel Code is executed.

1 :The function is canceled if an error takes place.

2 :The function is canceled if M02 is executed.

(18) A CIRCI.E POINT

This flag is used to set the allowable range of difference between the radius of the programmed circular arc at its start point and that at its end point.

For example, if A CIRCLE POINT' is set at 20, the allowable difference range is 2urn when DIGIT=0 and 0.2'wn when DIGIT=1.

(19) DIGIT

This flag is used to set the number of digits below the decimal point.

DIGIT	INCH	METRIC
0	4 digits	3 digits
1	5 digits	4 digits

(20) STRING PATTERN

This flag is used to set the String conversion pattern.

STRING PATTERN		
0	No string exchange	
1	(String A):(String B)	Processes (String B) as (String A).
2	(String A)$(String B):(String C)	Puts data held between (String A) and (String B) behind (String C) for processing.

(21) INIT INCH $^{METER}/_{INCH}$

This flag is used to set the state of METER/INCH (OFF: METER, ON: INCH) at POWER ON and determine its state on the basis of this setting at METER/INCH CANCEL.

In addition, after the change of this flag, METER/INCH can be immediately changed.

(22) INIT G91

This flag is used to set the state of ABS/INC (OFF: ABS, ON: INC) at POWER ON and determine its state on the basis of this setting at ABS/INC CANCEL.

In addition, after the change of this flag, ABS/INC can be immediately changed.

(23) SEARCH PATTERN

When this flag is set in OFF state, the. character string must be exactly the same and the sequence No. must be of four digits.

When the flag is set in ON state, the searching is performed with string exchange.

(24) FORMAT STOP

This flag, a simple error processing function, is used to specify how to deal with an error when it is detected in the program. If FORMAT STOP is set in OFF state, it causes the error to occur. If the flag is set in ON state, it causes the error to be located and rectified before the program execution is continued.

(25) RAM LINK

This flag is used to determine whether or not, if M98 or the sequence No. of the JUMP statement destination is not found in the private file when the program is executed in RAM RUN, to search other loaded files for it.

0 :Does not search other loaded files.

1 :Searches other loaded files.

(26) NOT USE CODE 0

1

2

3

4

This flag is used to set a maximum of five not-in-use codes (G, M, T) so that the code, if present in a program, is regarded as an error to stop the program.

The setting of the flag is made as follows:

NOT USE CODE 0 = +00000000

0:G

1:M Code No.

2:'T

(27) MIRR SWAP REVERCE 1

This flag is used to set the order in which MIRROR and X-Y CHANGE are executed when both are set in ON state during execution of GOO, G01, G02 and G03.

OFF : MIRROR and X-Y CHANGE are executed in this order.

ON : X-Y CHANGE and MIRROR are executed in this order.

(28) MIRR SWAP REVERCE 2

This flag is used to set the order in which MIRROR and X-Y CHANGE are executed when both are set in ON state during execution of G80, G81 and G92.

OFF : MIRROR and X-Y CHANGE are executed in this order.

ON : X-Y CHANGE and MIRROR are executed in this order.

CHAPTER 23. MESSAGAES

1) Error Messages

(1) Reading/Writing from user disk error. Press the ACK key.

This message is displayed when read or write of data from the user disk is impossible. Use another user disk. (In case such a trouble occurs, make a backup copy of the user disk on a periodic basis.)

(2) File ** * can not be found. Press the ACK key.

This message is displayed if, when the User Format function is executed, OFFSET FILE, CONDITION FILE or other system files are not found in the hard disk. Create the files and then repeat the same operation over again. File name * * * will be displayed.

(3) File * * * data error. Press the ACK key.

This message is displayed when file in the disk is broken. Use another file. File name * ** will be displayed.

(4) File *** format invalid. Press the ACK key.

This message is displayed when the format of file in the disk is not correct. Correct the format of the file and then execute it. File name * * * will be displayed.

(5) System version is wrong. Press the ACK key.

This message is displayed when the version of the system is wrong. Use a proper system version.

(6) Read error from outer peripherals. Press the ACK key.

This message is displayed when data input from the external device will not be transmitted properly.

(7) Error in graphic file input/output have occurred. Press the ACK key.

This message is displayed when an error occurs in input or output of graphic file.

(8) Time out error between manual controller. Press the ACK key. '

This message is displayed when an error occurs in communication with the manual controller. Check the cable, connector and other parts for proper connection.

(9) Parity error exists in horizontal direction of the tape. Press the ACK key.

This message is displayed when an error occurs in reading of PTR.

(10) Parity error exists in vertical direction of the tape. Press the ACK key.

This message is displayed when an error occurs in reading of PTR.

(11) Parity error has occurred. Press the ACK key.

This message is displayed when an error occurs in communication with the external device.

(12) Data read error from outer peripherals. Press the ACK key.

This message is displayed when data input from the external device has been destroyed.

(13) Transmission error between manual controllers. Press the ACK key.

This message is displayed when an error occurs in communication with the manual controller.

(14) Parity error or Over run between manual controllers. Press the ACK key.

This message is displayed when an error occurs in communication with the manual controller.

(15)

(16) Program end.

This message is displayed when M02 is executed during the execution of DISK RUN, RAM RUN, etc. Select another mode to clear the message.

(17) OFF STOP. Press the ACK key.

This message is displayed when the program execution is completely interrupted by activation of the OFF key. Press the ACK key to clear this state.

(18) Invalid characters exist in NC program. Press the ACK key.

This message is displayed when any character invalid in “DMEC” NC program is used.

19)

(20) Processing format invalid. Press the ACK key.

This message is displayed when a program block created using an incorrect calculation format will be executed.

(21)

(22) Data exceed limit during processing. Press the ACK key.

This message is displayed when data used for calculation in NC program and data obtained from the calculation exceed the specified limit.

(23) The specified OFFSET No. is not found. Press the ACK key.

This message is displayed when the offset No. used in NC program is not found ill the private file or OFFSET FILE.

(24) NC program error. Press the ACK key.

This message is displayed when a program block created using an incorrect fommt will be executed.

(25) Too long calculation. Press the ACK key.

This message is displayed if the calculation expression is created by input of more than a total of 48 alphanumeric characters (up to ?digits), operators (+, -, ~ , /, [.]) and H compensation terms when each of the first two is counted as one and each of the last as four.

(26) This character is not available. Press the ACK key.

This message is displayed when any character not available in: “DMEC” NC program is used.

(27) This B code is not available. Press the ACK key.

This message is displayed when B code is used in any format other than specified for it.

(28) This C code is not available. Press the ACK key. This message is displayed when C code is used in any format other than specified for it.

(29) This D code is not available. Press the ACK key. This message is displayed when D code is used in any format other than specified for it.

(30) This G code is not available. Press the ACK key. This message is displayed when G code is used in any format other than specified for it.

(31) This ft code is not available. Press the ACK key. This message is displayed when H code is used in any format other than specified for it.

(32) This L code is not available. Press the ACK key. This message is displayed when L code is used in any format other than specified for it.

(33) This M code is not available. Press the ACK key. This message is displayed when M code is used in any format other than specified for it.

(34) This N code is not available. Press the ACK key. This message is displayed when N code is used in any format other than specified for it.

(35) This O code is not available. Press the ACK key. This message is displayed when O code is used in any format other than specified for it.

(36) This P code is not available. Press the ACK key. This message is displayed when P code is used in any format other than specified for it.

(37) This Q code is not available. Press the ACK key. This message is displayed when Q code is used in any format other than specified for it.

(38) This T code is not available. Press the ACK. key. This message is displayed when T code is used in any format other than specified for it.

(39) Q command format is incorrect. Press the ACK key.
This message is displayed when Q command is used in an incorrect format.

(40) Q command nesting exceeds seven levels. Press the ACK key.
This message is displayed when Q command will be called eight times.

(41) The specified Q command file cannot be found. Press the ACK key.
This message is displayed when Q command-specified file is not found in the user diskette.

(42) Program nesting exceeds 50 levels. Press the ACK key.
This message is displayed when subprogram Will be called 51 runes using M98.

(43) The specified sequence No. is not found. Press the ACK key.
This message is displayed when M98 and M99 are used without the JUMP destination being found.

(44) M99 code has been used in the main program. Press the ACK key. This message is displayed when M99 is present in the main part of NC program.

(45) Plane selection is incorrect. Press the ACK key.

This message is displayed when the wrong plane is specified for circular cutting.

(46) OFFSET approach (IN/OUT) is impossible with Arc instruction. Press the ACK key.

This message is displayed when Offset/Taper approach will be executed by circular cut code.

(47) Circular interpolation command format is in error. Press the ACK key.

This message is displayed when G02/G03 code is executed without input of I or J/K.

(48) Linear interpolation command format is in error. Press the ACK key.

This message is displayed when G00/G01 code is executed with input of I/J or K.

(49) An axis has not been specified. Press the ACK key.

This message is displayed when ASSIGN is not set for axis specification.

(50) OFFSET value is larger than the radius of the circle. Press the ACK key.

This message is displayed when the electrode path of an circular arc is offset inside the arc with the offset amount larger than its radius.

(51) There is different radius between the starting point. Press the ACK key.

This message is displayed when circular cut code is executed without proper data input.

(52) Inserted 'VALUE' is too small. Press' the ACK key.

This message is displayed when insertion of Comer R is executed with its radius set much smaller than that of the circular arc.

(53)

(54)

(55) This code can not be used by setting. Press the ACK key.

This message is displayed when any code set in NOT USE CODE, a parameter in SETTING OPERATION submode, is executed.

(56) "0" is programmed in Pitch or revolution figure. Press the ACK key.

This message is displayed when SCREW PITCH or REVOLUTION in SETTING OPERATION submode for any axis is set at "0". Input a proper value either in SCREW PITCH or REVOLUTION.

(57) The offset will create interference. Press the ACK key.

This message is displayed if any interference occurs when INTERFERENCE CHECK is set in ON state with KANSHO ERR in SETTING ~ OPERATION submode set at "1".

(58)

(59) OFFSET/TAPER mode was not cancelled by the G40/G50 code. Press the ACK key. This message is displayed when program executed with OFFSET/TAPER set in ON state will be ended without cancellation of the mode.

(60) Anh-interpolation code is over 15. Press the ACK key.

This message is displayed when the number of 'non- interpolation codes exceeds 15.

(61)

(62)

(63)

(64) Division by zero. Press the ACK key.

This message is displayed when calculation in NC program is executed for division with the divisor as "0".

(65) Machining condition error of the header. Press the ACK key.

This message is displayed when the machining condition format is different from that specified.

(66) Specified compensation term error of the header. Press the ACK key.

This message is displayed when the compensation term format is different fi-om that specified.

(67) String file error. Press the ACK key.

This message is displayed when the string file format is different from that specified.

(68) Local conditions are over 100. Press the ACK key.

This message is displayed when the number of local machining conditions set exceeds 100, which is the maximum allowable limit.

(69) Local specified compensation terms are over 100. Press the ACK key.

This message is displayed when the number of local compensation terms set exceeds 100, which is the maximum allowable limit.

(70) Value too large (small). Press the ACK key.

This message is displayed when numerical data processed for calculation in NC program or results obtained from the calculation exceed the specified digit or result in division by zero within NC.

(71) limit check error has occurred. Press the ACK kc5'.

This message is displayed when graphic program executed with Limit Check in Graphic data, a parameter m GRAPHIC mode, set m ON state exceeds the machining coordinate limit.

(72)

(73)Touching Limit Switch on stroke end (over travel).Press the ACK key.

This message is displayed when the table touches the limit switch of the machine system and stops. Press the [ACK] key to clear the message.

(74) The electrode and workpiece touched. Press the ACK key.

This message is displayed when the electrode comes into touch with the workpiece. Press the [ACK] key to clear the message.

(75)

(76)

(77) Software limit stop. Press the ACK key.

This message is displayed when. with SOFTWARE LIMIT ON, the travels out of the range set in SOFTWARE LIMIT. Press the [ACK] key to clear the message.

(78) Dielectric change to oil is missed. Change manually please. Press the ACK key.

This message is displayed when dielectric change from water to oil fails.

(79) This code is not available. Press the ACK key.

This message is displayed when code input is ii, put in any format other than specified.

(80) The fire extinguisher has been activated. Press the ACK key.

This message is displayed when the dielectric fluid temperature rises to such an extent that fire may occur, causing the fire extinguisher to be activated with the power to the machine concurrently turned off. Press tile [ACK] key to clear the message.

(81) M03 call not be found. Press the ACK key.

This message is displayed when, with M03 in SETTING mode set in ON stale, AWT

failure occurs, which causes M03 to be searched without the code being found.

(82) Machining has stopped due to a SHORT circuit. Press the OFF key to restore the state to normal.

This message is displayed when the electrode has touched the workpiece and will not operate any further for machining.

(83)Machine is locked. Press the ACK key.

This message is displayed when ATC is executed with MACHINE LOCK set in ON state.

(84)The specified CONDITION No. is not found. Press the ACK key.

This message is displayed if, when a machining condition is used in NC program, its corresponding No. is not found either in the private file or CONDITION FILE.

(85)The specified OFFSET No. is not found. Press tile ACK key.

This message is displayed if, when an offset term is used in NC program, its corresponding No. is not found either in the private file or OFFSET FILE.

(86)

(87)

(88)

(89)

(90)