TOE-C843-7 20B
INSTRUCTIONS

CNC SYSTEM FOR TURNING APPLICATIONS

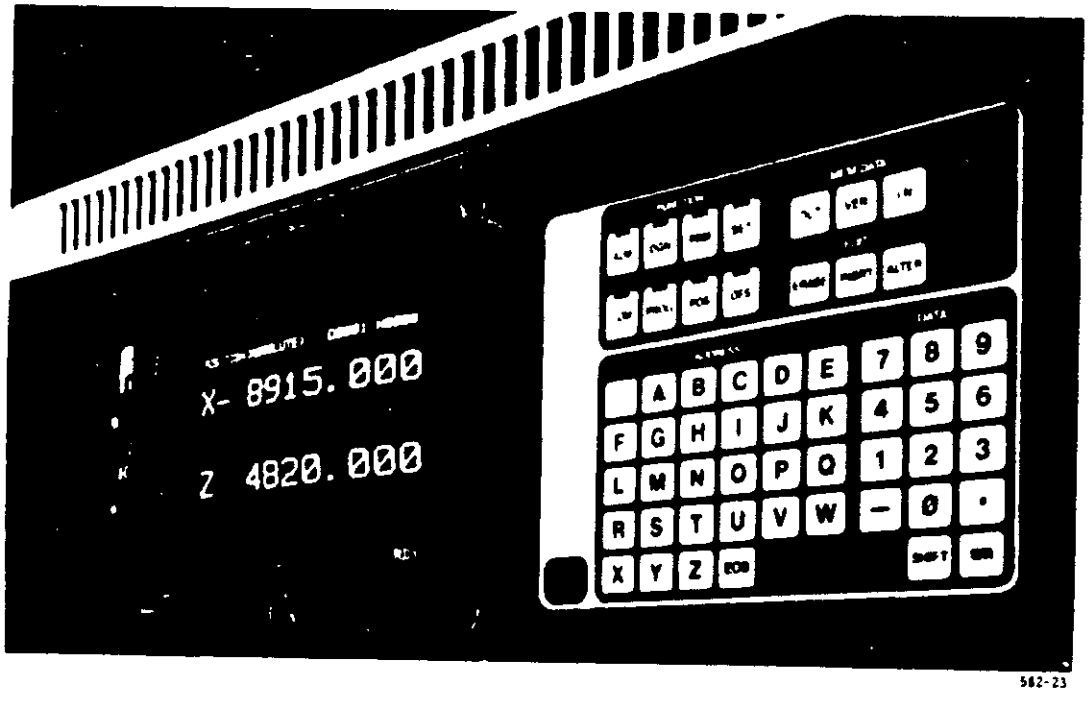
YASNAC LX1

OPERATOR'S MANUAL

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

This manual is primarily intended to give operator's instructions for YASNAC LX1 programming, operation and maintenance.

This manual applies to the basic and optional features of YASNAC LX1. The optional features are marked with a dagger. For the specifications of your YASNAC LX1, refer to the machine tool builder's manual.



582-23

YASNAC LX1 OPERATOR'S STATION

PREFACE

This manual is primarily intended to give operators instructions for YASNAC LX1 programming, operation and maintenance.

This manual applies to the basic and optional features of YASNAC LX1. The optional features are marked with a dagger(-). For the specifications of your YASNAC LX1, refer to the machine tool builder's manual.

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

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

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

- Feed Function Selection: G99 (mm/rev)
- Reference Zero Point
  (Return to reference zero by manual and automatic return). - - - - - - - - - - -
- Absolute Zero Point. - - - - -
- Work Coordinate Zero Point. - - - -

TABLE OF CONTENTS

INDEX

INDEX

INDEX

INDEX

INDEX

INDEX

INDEX

1. INTRODUCTION

YASNAC LX1, "Ultraspeed dual processor CNC" is a combination of two high-performance 16-bit microprocessors running in parallel. Incorporating our modern system technique, it is designed to provide the highest lathe performance.

- The dual processor CNC system drastically reduces the data processing time to meet high-speed cutting. Block-to-block stop time decreased by the use of high-speed buffer function and buffering function.
- Enhanced cutting capability includes a maximum of 24 meters/min feed command, precise feed E command, 500-millimeter lead thread cutting, continuous thread cutting, multiple thread cutting, and variable pitch thread cutting.
- To meet FMS trends, program interrupt function, tool life control, user macro, tool set error correction, stored stroke limit per tool, and other functions can be installed.
- Part program memory can be extended to a maximum of 320 meters. Its data input/output interface is available with FACIT, RS232C and, in addition, RS422 serial interface capable of high-speed long distance transmission.
- Programming is further facilitated by improved tool radius compensation function, G50-work coordinate system setting, angle-specified linear interpolation, and combined beveling/rounding function.
- The servo function uses a drastically miniaturized and low-noise, newly transistorized PWM control unit and a high-performance DC servo motor.

  The position feedback is available with the standard pulse generator (PG) system and, the inductosyn-applied complete closed loop system.

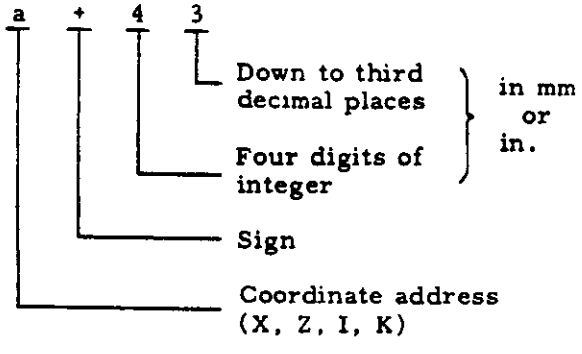
2. PROGRAMMING

2.1 TAPE FORMAT

2.1.1 TAPE FORMAT

A variable block format conforming to JIS# B 6313 is used for YASNAC LX1.
Table 2.1 shows the tape format. Numerals following the address characters in Table 2.1 indicate the programmable number of digits.

EXAMPLE



Japanese Industrial Standard

Note: The decimal point may be omitted in actual programming. For making a program including decimal points, refer to 2.1.3 Decimal Point Programming.

The leading zeros can be suppressed for all address codes. Plus signs need not be programmed, but all minus signs must be programmed. In the manual, EOB code in a program example is represented by a semicolon (;). In actual programming, CR (EIA code) or LF/NL (ISO code) should be used instead of the semicolon (;).

Table 2.1 Tape Format

No.	Address	Metric output		Inch output		B: Basic O: Option
		Metric Input	Inch input	Metric Input	Inch input	
1	Program No.	O4		O4		B
2	Sequence No.	N4		N4		B
3	G-Function	G3		G3		B
4	Coordinate Word a. X, Z, I, K, U, W, R	a + 43 (a + 53)‡	a + 34 (a + 44)‡	a + 53 (a + 53)‡	a + 34 (a + 44)‡	B
5	Feed/min	F50	F32	F50	F42	B
6	Feed/rev and Thread Lead	F32	F24	F42	F24	B
		E34	E26	E44	E26	B
7	S-Function	S2		S2		B
		S4		S4		O
8	T-Function	T(2 + 1)		T(2 + 1)		B
		T(2 + 2)		T(2 + 2)		O
9	M-Function	M3		M3		B
10	Dwell	U(P)53		U(P)53		B
11	Program No. Designation	P4		P4		B
12	Sequence No. Designation	Q(P)4		Q(P)4		B, O
13	No. of Repetitions	L8		L8		B
14	Angle Designation for Straight Line	A(B)33		A(B)33		O
15	Angle Designation for Multiple Thread	B3		B3		O

Notes.

1. Data with ‡ indicates maximum cumulative value.
2. Inch/Metric output is set by setting parameter #6007D3.
3. Inch/Metric input is set by setting (#6001D0).
4. F codes for feedrate/min or feedrate/rev can be switched by G98, G99.

Table 2.2 List of Program Commands

Address		Metric output		Inch output	
		Metric input	Inch input	Metric input	Inch input
Program No. O		1 - 9999		1 - 9999	
Sequence No. N		1 - 9999		1 - 9999	
G function G		0 - 199		0 - 199	
Coordinate Address[1] X, Z, I, K, U, W, R		±8388.607 mm (±99999.999 mm)	±330.2601 in. (±9999.9999 in.)	±21307.061 mm (±99999.999 mm)	±838.8607 in. (±9999 9999 mm)
Feed/min	F	1 - 24000 mm/min	0.01 - 944.88 in/min	1 - 60960 mm/min	0.01 - 2400.00 in/min
Feed/rev and Thread Lead	F	0.01 - 500.00 mm/rev	0.0001-19.6850 in/rev	0.01 - 1270.00 mm/rev	0.0001 - 50.0000 in/rev
	E	0.0001 - 500.0000 mm/rev	0.000004 - 19.685000 in/rev	0.0003 - 1270.0000 mm/rev	0.000010 - 50.000000 in/rev
S-function	S2	0 - 99		0 - 99	
	S4	0 - 9999		0 - 9999	
T-function	T3	0 - 99		0 - 99	
	T4	0 - 9999		0 - 9999	
M-function		0 - 999		0 - 999	
Dwell U, P		0.001 - 99999.999 sec		0.001 - 99999.999 sec	
Program No. Designation		1 - 9999		1 - 9999	
Sequence No. Designation		1 - 9999		1 - 9999	
No. of Repetitions		0 - 99999999		0 - 99999999	
Angle Designation for Straight Line[2]		0 - ±360.000°		0 - ±360.000°	
Angle Designation for Multiple Thread		0 - 360°		0 - 360°	

1 Parenthesized data indicates maximum cumulative value.
2 For angle designation of included angle for G76, see 2.8.26.8 Automatic Threading Cycle (G76).

2.1.2 LIST OF ADDRESS CHARACTERS AND FUNCTION CHARACTERS

Table 2.3 Address Characters

Address	Meaning	B: Basic O: Optional
A	Angle designation for G01 and G111, included angle for G76	O
B	Spindle shift angle 01 multiple thread, angle designation for multiple cornering	O
C	User macro character	O
D	Depth of cut and number of cutting cycles for G71 to G76	O
E	Specifications for precise feed and precise lead for cutting	B
F	Specifications for normal feed and normal lead for cutting	B
G	Preparatory function (G-function)	B
H	User macro character	O
I	X-component of arc center, canned cycle parameter, beveling value (radius value)	B, O
J	User macro character	O
K	Z-component of arc center, canned cycle parameter, beveling value	B, O
	Incremental value of variable lead thread	O
L	Number of subprogram repetition, G13 to G16 angle and coordinate	B, O
M	Miscellaneous function (M-function)	B
N	Sequence number	B
O	Program number	B
P	Dwell, canned cycle starting sequence number, program number, user macro number	B, O
Q	Subprogram starting sequence number, canned cycle ending sequence number	B, O
R	Radius of arc, rounding value, tool radius value	B, O
S	Spindle function (S-function), maximum spindle revolution	B
T	Tool function (T-function), tool coordinate memory number	B, O
U	X-axis incremental command value, dwell, canned cycle parameter	B, O
V	User macro character	O
W	Z-axis incremental command value, canned cycle parameter	B, O
X	X-axis coordinate value	B
Y	User macro character	O
Z	Z-axis coordinate value	B

Table 2.4 Function Characters

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

Notes:

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

2.1.3 DECIMAL POINT PROGRAMMING

Numerals containing a decimal point may be used as the dimensional data of addresses related to coordinates (distance), angle, time and speed. They can be inputted from punched tape or MDI.

Decimal points can be used in the following address words.

Coordinate words. X, Z, I, K, R

Angle words: A, B

Feedrate word: F, E

Time words: U, P

EXAMPLE

	[mm]	[inch]
X15. ———	X15.000 mm	or X15.0000 in.
Y20.5 ———	Y20.500 mm	or Y20.5000 in.
(G99)F.2 —	F0.20 mm/rev (for F32)	or F0.2000 in/rev (for F24)
(G98)F25.6 —	F25 mm/rev (for F50)	or F25.60 in/min (for F32)
G04P1. ———	Dwell 1.000 sec	

Normally, when data without a decimal point is inputted, the control regards "1" as 0.001 mm (or 0.0001 inch or 1 deg.), but with a parameter setting, the control may be made to interpret "1" as 1 mm (or 1 inch or 0.001 deg.). Refer to parameter #6019D_6.

2.1.4 LABEL SKIP FUNCTION

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

- When the power supply is turned on.
- When the RESET operation is executed.

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

2.1.5 BUFFER REGISTER

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

In the tool radius compensation† mode, two blocks of data or up to 4 blocks of data are read in advance and compensation computing required for the next operation is executed. One block can contain up to 128 characters including EOB.

The blocks including the following M codes are not read in advance.

- M00, M01, M02, M30
- M codes (6 maximum) set by parameter commanding to stop advance-reading.

2.1.6 HIGH-SPEED BUFFER REGISTER

A high-speed buffer register is installed as the standard to serve for high-speed cutting. If a consecutive group of blocks is specified in thread cutting (G32) or linear interpolation (G01), the stop time between blocks is reduced to zero by this function (Note 2). This permits in continuous thread cutting a smooth cutting with shortened stop time between blocks.

NOTES:

1. This function is effective for G22 and G23 where the control is provided with Radius Programming for Circular Interpolation option
2. Block-to-block stop time due to the time required to compute tool radius compensation is not eliminated or remains. To reduce this stopping time, use 2.7.4 Buffering Function (M93, M92) (optional). When operation of consecutive blocks up to 5 in M93 mode, inter-block stoppage time is reduced zero.

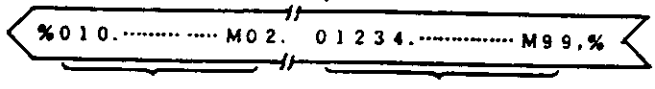
2.2 PROGRAM NUMBER AND SEQUENCE NUMBER

2.2.1 PROGRAM NUMBER

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

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

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



PROGRAM WITH PROGRAM NO. 10 — PROGRAM WITH PROGRAM NO. 1234

ER (or % at ISO code) is punched on both end parts of the tape.

NOTES:

1. The blocks for optional block skip such as /M02;, /M30;, /M99; are not regarded as enc of programs.

2. It is possible with a parameter change (#6201D0), to make the reading of M02, M30, and M99 ineffective as a program end, and to make the succeeding ER (EIA) or % (ISO) as a sign of program end.

2.2.2 SEQUENCE NUMBER

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

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

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

NOTES:

1. Five or more digits must not be written as a sequence number.
2. When two or more blocks have the same sequence number, only one is retrieved and read, and no more searching is performed.
3. Blocks without sequence numbers can also be searched for with respect to the address data contained in the blocks.

2.2.3 OPTIONAL BLOCK SKIP (/1 - /9)(+)

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

EXAMPLE

```
/2 N1234 G01 X100 /3 Z200;
```

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

```
N 1234 G01 X100,.
```

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

NOTES:

1. The optional block skipping process is executed while the blocks are being read into the buffer resister. Once the blocks have been read, subsequent switching on is ineffective to skip the blocks.
2. While reading or punching out programs, this function is ineffective.
3. The block skip /2 - /9 is an option function, and /1 is a basic one.

2.3 COORDINATE WORDS

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

2.3.1 COORDINATE WORDS

Address of Coordinate Words		Meaning
Main Axis	X, Z	Absolute coordinate position of target position
	U, W	Incremental distance (U: Direction in X-axis, W: Direction in Z-axis)
Radius Value for Circular Interpolation	I, K	Incremental distance between start point and center of circular arc. (I: X-axis component, K: Z-axis component)
	R†	Radius value of circular arc

Note: When G90 and G91 are used, addresses X and Z are not fixed as absolute value and follow according to G90/G91 designation. For details, refer to 2.3.5 Absolute and Incremental Inputs.

2.3.2 SIMULTANEOUS CONTROLLABLE AXES

The control provides two-axis control for X- and Z-axis. Number of simultaneously controllable axes, when commanded in the same block, is two axes, X and Z. For the axis without commands, movement will not occur

2.3.3 LEAST INPUT INCREMENT AND LEAST OUTPUT INCREMENT

2.3.3.1 Least Input Increment

The minimum input units that can be commanded by punched tape or MDI are shown below.

Least Input Increment

	1 x	10 x (10 times input unit)
Metric system	0.001 mm	0.01 mm
Inch system	0.0001 in.	0.001 in

X-axis is specified for diameter.

2.3.3.1 Least Input Increment (Cont'd)

Inch/MM input is selected by setting #6001D_0. Inch/MM input selection by G20/G21 is optional. Selection of multiplication factor x1/x10 is made by parameter #6006D_5.

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

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

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

NOTES:

1. If NC tape programmed by 0.001 mm is fed into or stored in an equipment set by 0.01 mm increment, the machine will move ten times the intended dimensions.
2. If the increment system is switched when the contents of NC tape are stored in memory, the machine will move by ten times or one tenth of the commanded dimensions.
3. When the stored program is punched out on the tape, the stored figures are punched out "as stored" regardless of switching of the increment system.
4. Multiplication factor 10X (10 times the input unit) is effective for distance command only. It does not function on the designation of time, angle, etc When multiplication factor 10X is set as effective (#6006D_5 = 1), the same address word is multiplied by 10 or not depending on type of G command.

   EXAMPLE

   G04 U··· ;←—Not multiplied by 10 (Time)
   G00 U··· ;← Multiplied by 10 (Distance)

2.3.3.2 Least Output Increment

Least output increment is the minimum unit of tool motion. Selection of metric system or inch system is made by parameter (#6007D_3).

· Least Output Increment

	X-axis (Radius value)	Z-axis
Metric output	0.0005 mm	0.001 mm
Inch output	0.00005 in.	0.0001 in.

Inch or metric output is selected by parameter #6007D_3.

2.3.4 MAXIMUM PROGRAMMABLE DIMENSIONS

Maximum programmable values of move command are shown below.

Maximum Programmable Values

Metric Output	Metric input	±8388.607 mm
	Inch input	±320.2601 in
Inch Output	Metric input	±21307.061 mm
	Inch input	±838.8607 in

In absolute programming, move amount of each axis must not exceed the maximum programmable value. THE MACHINE MAY NOT FUNCTION PROPERLY IF A MOVE COMMAND OVER THE MAXIMUM PROGRAMMABLE VALUE IS GIVEN.

The above maximum programmable values also apply to distance command addresses I, K, R in addition to move command addresses X, U, W.

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

Maximum Accumulative Values

Metric system	±99999.999 mm
Inch system	±9999.9999 in

Note: These values are not determined by least output increment.

2.3.5 ABSOLUTE AND INCREMENTAL INPUTS

Both absolute input and incremental input can be used for the control.

- Absolute input is specified by the addresses X and Z.

  EXAMPLE: X··· Z··· ;

- Incremental input is specified by the addresses U and W.

  EXAMPLE: U··· W··· ;

- Absolute input and incremental input can be used in one block mixedly.

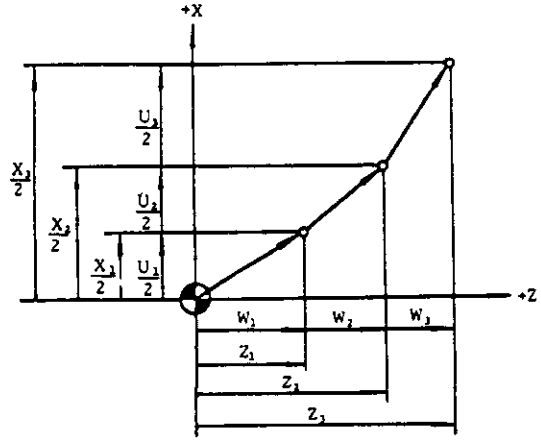
  EXAMPLE: X··· W··· ;
  U··· Z··· ;

NOTE: When addresses X and U or addresses Z and W are used in one block, the latter is effective.

The addresses I and K for designation of arc-center must be specified by the incremental dimension.

Table 2.5

Address	Increment System	Designation	Meaning
X	Absolute Input	Diameter	Position in X-axis direction (Note)
Z		-	Position in Z-axis direction
U	Incremental Input	Diameter	Move amount in X-axis direction (Note)
W		-	Move amount in Z-axis direction
I	Incremental Input	Radius	Distance in X axis-direction from starting point of arc to center
K		-	Distance in Z-axis direction from starting point of arc to center
R^{+}	Incremental Input	-	Direct programming of circular arc



X and Z: Absolute Input

U and W: Incremental Input

Note: Since X and U are designated by the values in diameter, the actual movement is the half of the values.

Fig. 2.1 Absolute Coordinate Values and Incremental Coordinate Values

Cases where G90 and G91 (absolute and incremental commands) are used.

- When special G code I (basic) or II (option) is selected, G90 and G91 codes can be used.

G code	Meaning
G90	Absolute command
G91	Incremental command

As shown below, G90 and G91 commands are effective only to addresses X and Z.

	Addresses	G90 command	G91 command
TAPE, MEM modes	X, Z	Absolute	Incremental
	U, W	Incremental	Incremental
MDI mode	X, Z	Absolute	Not influenced by G90 and G91.
	U, W	Incremental	

EXAMPLE: G91 G00 X40. Z50.; .....

Incremental move command

- Auxiliary data, I, K, R, etc., of circular interpolation are always incremental commands.

NOTE: G90 and G91 can not be programmed together in the same block. If they are written in the same block, the one written later only is effective.

EXAMPLE. G01 G90 X80. G91 Z60.;

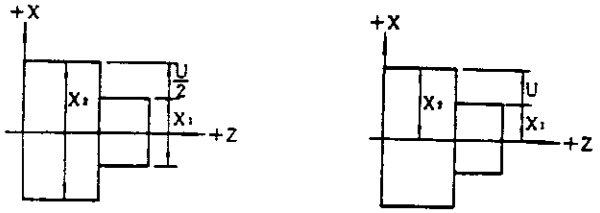
G91 is effective, and in this block, commands become incremental in both the X and Z axes.

2.3.6 X-AXIS DIAMETER/RADIUS SWITCHING†

Addresses X and U for X-axis coordinate words are specified by diameter value. This is called diameter designation. When the control is equipped with DIAMETER/RADIUS switching option, the addresses X can be used for designation of both diameter and radius. The switching is made by the setting of parameter #6006D3.

0: Diameter designation

1: Radius designation



(a) In the case of Diameter Designation

(b) In the case of radius Designation

Fig. 2.2

Table 2.6

	Diameter Programming	Radius Programming
Address X command	Diameter value	Radius value
Address U command	Diameter incremental value	Radius value incremental value
X-axis position display	Diameter value	
Tool position offset value	Diameter value	
Tool coordinate data for work coordinate system	Diameter value	
Nose radius R	Radius value	
Feedrate F, E in X-axis direction	Radius value/rev Radius value/min	
Radius data I, R for circular interpolation	Radius value	
G90 - G94, G70 - G76, Parameters for cornering, and multiple cornering, D, I, K, P, Q, R	Radius value	

2.4 RAPID TRAVERSE RATE

2.4.1 RAPID TRAVERSE RATE

2.4.1.1 Rapid Traverse Rate

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

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

2.4.1.2 Range of Rapid Traverse Rate

(1) For each axis, rapid traverse rates can be set at some suitable multiple of 125 p/sec. (p: Least output increment)

(2) The rapid traverse rate can be set to the upper limit shown below.

Metric Input	24,000 mm/min
Inch Input	2,400 in/min

The upper limit for X-axis speed is half the listed values. The optimum value of upper limit is set according to the machine. Refer to the machine tool builder's manual, for the definite value.

2.4.2 FEED FUNCTION (F- AND E-FUNCTION)

G code listed below must be designated before F, and E function is commanded.

G code	Function
G99	Designation of feedrate in mm/rev.
G98	Designation of feedrate in mm/min.

Note: For the details, refer to 2.8.29 Feed Function Designation.

Since F, E codes are modal, these codes are effective until next F, E codes are given. However, when G98/G99 are switched, new F code must be designated.

2.4.2.1 Feed Per Revolution (G99 Mode)

(1) Tool feed per revolution of the spindle can be specified with F (normal feed) or E (fine feed).

(2) The feed ranges that can be specified by the F and E codes are as follows.

G99 Mode, F and E Feed Ranges

		Format	Range of feed/revolution
Metric output	Metric input	F32	F0.01 - F500.00 mm/rev
		E34	E0.0001 - E500.0000 mm/rev
	Inch input	F24	F0.0001 - F19.6850 in/rev
		E26	E0.000004 - E19.685000 in/rev
Inch output	Metric input	F32	F0.01 - F1270.00 mm/rev
		E34	E0.0003 - E1270.0000 mm/rev
	Inch input	F24	F0.001 - F50.0000 in/rev
		E26	F0.000010 - E50.000000 in/rev

These feed ranges are subject to the following restrictions depending on the spindle speed S.

Metric output	F(E) x S = 24,000 mm/min
Inch output	F(E) x S = 2,4000 in/min

Notes.

1. Program feed per revolution within such a range that the X-axis component remains below 12,000 mm/min or 1,200 in/min.
2. This upper limit may still be reduced by the performance limit of the machine. Refer to the machine tool builder's manual.

NOTES:

1. A command "F0" causes data errors.
2. Any minus value should not be specified for F commands. If specified, the machine will not operate properly.

   EXAMPLE

   F-250 ; ····· Wrong

3. Feedrate commands in the direction of the X-axis must be given in radius.

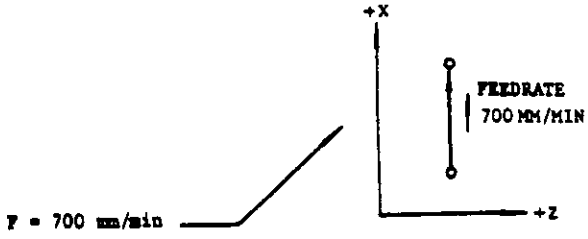
EXAMPLE

G99 S350 (rpm) ;

G01 U10000 F200 ; ··· In case of F32.

In the above case, the feedrate is:

F x S = 2.0 mm/rev. x 350 rpm



4. Values of F command at linear or circular interpolation represent the tangential feedrate when two axes are simultaneously controlled.

EXAMPLE 1

G99 S100 (rpm) ;

G01 U60. W40. F50 ;

In the above case, the feedrate is

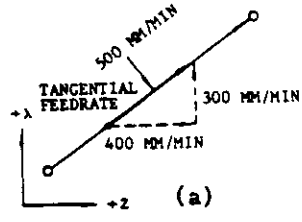
F x S = 0.5 mm/rev x 1000 rpm

= 500 mm/min

$= \sqrt{300^2 + 400^2}$

Z-axis feedrate component

X-axis feedrate component



EXAMPLE 2

G99 S1000 (rpm) ;

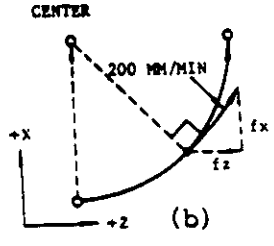
G03 U··· W··· I··· F20 ;

In the above case, the feedrate is:

F x S = 0.2 (mm/rev) x 1000 (rpm)

= 200 mm/min

$= \sqrt{fx^2 + fz^2}$



2.4.2.2 Feed Per Minute (G98 Mode)

(1) Tool feed can be specified in mm/min or in/min with F codes.

(2) The feed range that can be programmed with F codes is as follows.

G98 Mode F Code Feed Range

		Format	Range of feed per minute
Metric Output	Metric Input	F50	F1. - F24000. mm/min
	Inch Input	F32	F0.01 - F944.88 in/min
Inch Output	Metric Input	F50	F1. - F60960. mm/min
	Inch Input	F42	F0.01 - F24000.00 in/min

Notes.

1. Program feed-per-minute values so that the X-axis speed component will not exceed half the above upper limit feedrates.

   EXAMPLE

   G98 G01 U300. F12000 ,
   (Metric output, metric input)

2. The upper limit value is further subject to the limitation imposed by the machine performance. Refer to of the machine tool builder's manual. This upper limit value is to be set in parameter #6228.

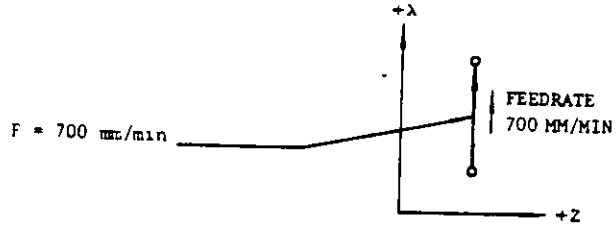
NOTES

1. Do not write F command in FO or negative values
2. Commands in the X-axis direction indicate speeds in radius.

EXAMPLE

G98

G01 X20000 F700 ;



Values of F command at linear or circular interpolation represent the tangential feedrate when two axes are simultaneously controlled.

EXAMPLE 1

G98 ,

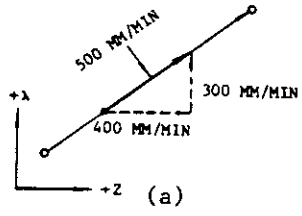
G01 U3000 W4000 F500 ;

In this case,

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

$\sqrt{300^2}$: Y-axis component

400^2 : X-axis component



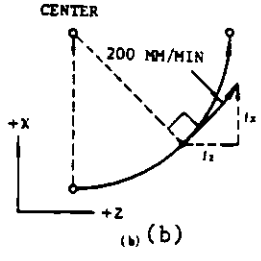
EXAMPLE 2

G98 ;

G03 X··· Z··· I··· F200 ;

In this case,

$F = 200 = \sqrt{fx^2 + fz^2}$



2.4.3 AUTOMATIC ACCELERATION AND DECELERATION

Acceleration and deceleration for rapid traverse and for cutting feed are automatically performed without programming.

2.4.3.1 Acceleration And Deceleration of Rapid Traverse And Manual Feed

In the following operation, the pattern of automatic acceleration and deceleration is linear. (See Fig. 2.3.)

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

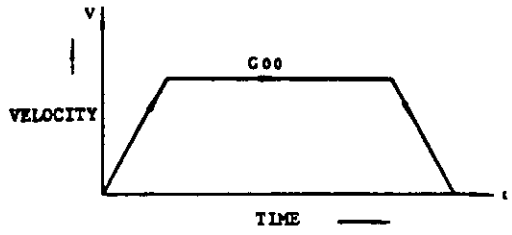


Fig. 2.3

Rapid traverse rate and the acceleration/deceleration constant of rapid traverse rate can be set by parameter. (#6280 to #6287)

2.4.3.2 Acceleration And Deceleration of Cutting Feed

In the following operation, the pattern of automatic acceleration and deceleration is of expotential curve. (See Fig. 2.4.)

· Cutting feed (G01 to G03)

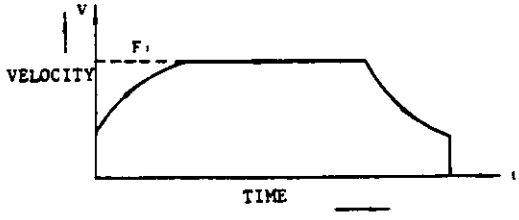


Fig. 2.4

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

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

2.5 SPINDLE-SPEED FUNCTION (S-FUNCTION)

2.5.1 S 2-DIGIT PROGRAMMING

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

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

When a move command and an S code are issued in a block, execution will depend on the machine tool design and construction (Whether the S command is executed together with the move command or after the completion of tool movement). Refer to the machine tool builder's manual.

EXAMPLE

```
G00 S11 M03 ;
        ··· S command
            Spindle CW
    X··· Z··· ;
G01 Z··· F··· ;
    .
    .
    .
```

S11: Effective

```
G00 X··· Z··· M05 ;
        ··· Spindle stop
    ··· M03 ;
    X··· Z··· ;
G01 Z··· F··· ;
```

S11: Effective

```
    S22 ;
    X··· Z··· F··· ;
    .
    .
```

S22: Effective

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

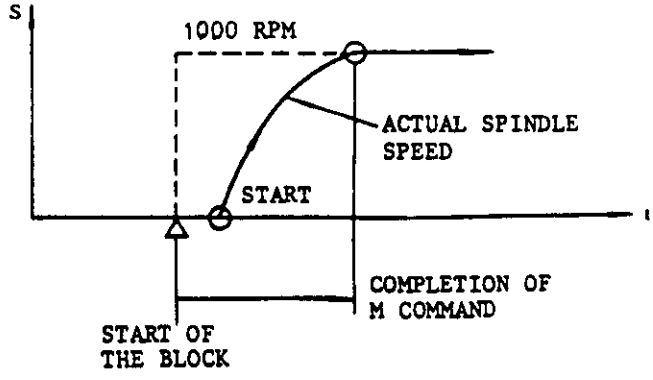
2.5.2 S 4-DIGIT PROGRAMMING A+

(1) Four digits following S (S□□□□) are used to specify the spindle speed in rpm.

(2) When S command is given in a block together with M03 (spindle forward running) or the M04 (reverse running), the control proceeds to the next block after the spindle speed reaches the speed given by the S code. For details, refer to the machine tool builder's manual.

EXAMPLE

S1000 M03 ;



2.5.2 S 4-DIGIT PROGRAMMING A^{-} (CONT'D)

(3) S commands are modal. Although the spindle stops at the M05 command, the S command is retained. Therefore, when M03 (or M04) is given, the spindle runs according to the S command.

(4) When S command is changed after the spindle start by M03 or M04, S command should be given within the range of spindle speed selected by spindle gear.

NOTES:

1. The lower limit of the spindle speed depends on the spindle drive. Refer to the machine tool builder's manual for the low-speed limit. Negative S commands must not be programmed.
2. When the control is provided with the S 4-digit command function, the "Spindle speed override" option can be built into it.
3. With machine tools with which the main spindle gear ratio changes can be specified by M codes, first write the applicable M code to preselect the desired gear ratio, and then, write the S command. Refer to the data of the machine tool builder for the number of gear ratios, the speeds at various gear ratios, and other details.
4. When the control is provided with this function, the spindle maximum speed commanding function with the instruction "G50 S··· ;" can be used

2.5.3 S 4-DIGIT PROGRAMMING B†

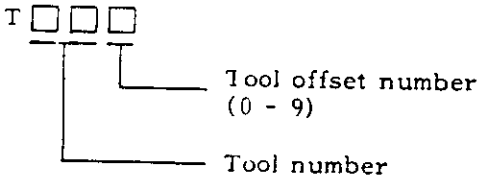
(1) This function is to modify the S 4-digit command A output freely through the programmable machine interface.

(2) Basically, this function is used in the same way as the S 4-digit command A function, but it is normally used to set the manually controlled spindle speeds controlled by the rotary switch on the machine control station corresponding to S command speeds. For the details of S command speeds, refer to the machine tool builder's manual.

2.6 TOOL FUNCTION (T-FUNCTION)

2.6.1 T 3-DIGIT PROGRAMMING

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



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

When a move command and a T code are issued simultaneously, execution will depend on the machine tool design and construction.

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

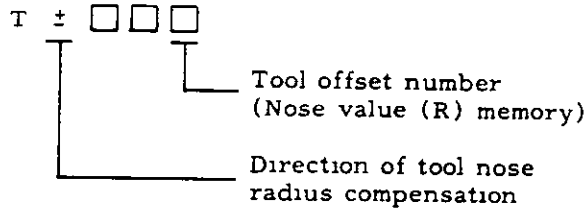
For this, refer to the machine builder's manual.

Tool offset number designation specifies tool offset memory number and executes tool position offset.

- T codes are modal, and therefore, once they are given, they remain effective until another T command is given.

NOTES:

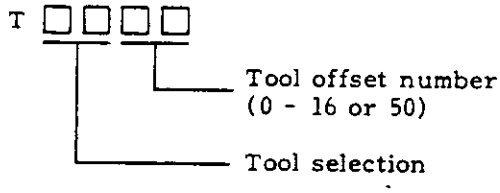
- When the Tool Nose Radius Compensation† option is provided, the T code must be programmed with sign (+ or -). For the details, refer to 2.8.20 Tool Nose Radius Compensation.



- For T code designation for work coordinate system shift (G50 T□□□)†, refer to 2.8.23 Work Coordinate Multi-Shift.

2.6.2 T 4-DIGIT PROGRAMMING

(1) Four digits following the address T specifies the tool number.



(2) Programming T 4-digit command is the same as that for T 3-digit command except for two-digit tool offset number designation.

(3) For applicable tool number to be specified, refer to the machine tool builder's manual.

Specifications	Tool Offset No.
a	0 - 16
b	0 - 50

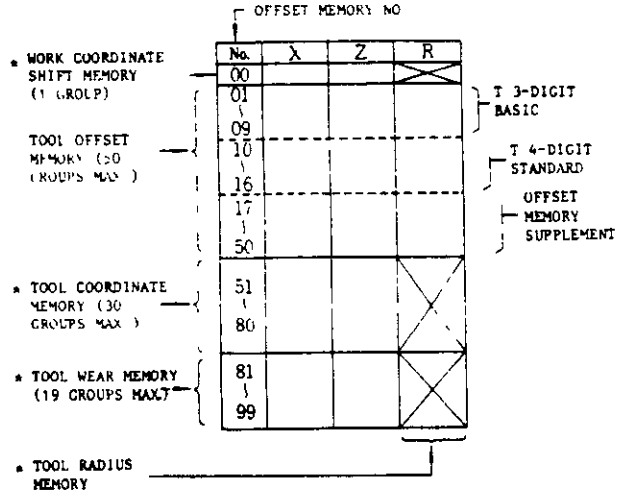
NOTES:

1. When the tool number is changed by the T command, a turret lathe begins to index the tool instantaneously. Therefore, the turret should be removed, before the command, from the area where an accidental collision might occur.
2. Tool offset number 0 or 00 cancels the tool offset.

2.6.3 TOOL OFFSET MEMORY †

The area in which tool position offset values, tool radius compensation values, and other compensation data are stored is called Offset Memory.

(1) The entire memory areas of Offset Memory including the options are as shown below.



NOTE: For the actually usable range within the above Offset Memory, refer to the machine tool builder's manual.

(2) The "tool offset Nos." specified by the T function directly correspond to the "offset memory Nos.," and their contents are used for various compensations. However, the tool coordinate memory Nos. (for setting the work coordinate system) correspond to the tool selection Nos. in the T function. The work coordinate shift memory is an independent function, not related to the T function.)

(3) Write these data in the memory, before starting to operate the machine under automatic control. For the writing procedure, refer to 4.3.5 Displaying and Writing Tool Offset Values. For writing into Tool Coordinate Memory, follow the procedure described in 6.2.3 Work Measurement Value Direct Input†.

2.6.4 TOOL POSITION OFFSETS

When the tool offset number is specified, the offset value corresponding to the tool offset number is added algebraically to the command value in the program and the tool is moved to the offset position. Therefore, the difference between the coordinate values of the programmed tool tip and the actual tool tip must be stored into tool offset memory in advance as the offset value.

When the coordinate value of the actual tool tip has changed due to tool wear or some other reasons, the tool position offset values should be set again. Thus, the programmed machining is attained without correcting the program.

(1) Range of tool position offset value

The programmable range of tool offset value is shown below.

Output	Input	Setting Range
Metric Output	Metric input	0 - ±8388.607 mm
	Inch input	0 - ±330 2601 in
Inch Output	Metric input	0 - ±9999.999 mm
	Inch input	0 - ±838.8607 in

(2) Sign of tool position offset values

a. Store the tool position offset values in the Offset Memory. The offset value is the deviation from the tool tip position of the reference tool which is determined as zero.

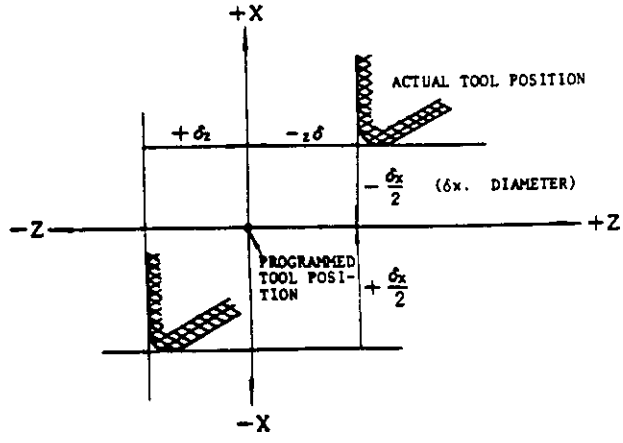


Fig. 2.5

2.6.4 TOOL POSITION OFFSETS (CONT'D)

(3) Description of tool position offset motion

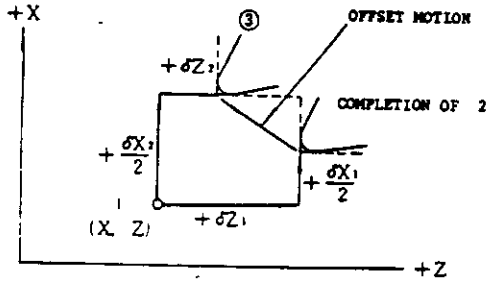
As mentioned above, when the tool specified by the address T and 4 digits is moved, the offset value corresponding to the tool offset number is added to the command value in the program algebraically and the tool tip is moved to the offset position.

When there is no move command in the block, the tool moves only by the offset value. Once, the tool offset number is designated, the tool moves always to the offset position until another number is designated. When the other offset number is designated or the offset value is changed, the offset value is compensated for by the amount of the difference between the old and new offset values.

OFFSET VALUE

T101 → $(+\delta X_1, +\delta Z_1)$

T115 → $(+\delta X_2, +\delta Z_2)$



EXAMPLE

```
T101 ; ..................... 1
  .
  .
  .
G01 X ·· Z··· F(E)... ; ..... 2
T115 , ..................... 3
                    (Block of the
                    offset motion)
```

(4) Move speed with tool offset

The move speed of tool offset is determined by the feedrate command that is effective in the block. Therefore, the feedrate command (G00 or G01 F···) should be issued before or in the block containing the tool offset number.

EXAMPLE

```
G50 X··· Z··· ;
G00 S··· M03 T0108 ;   Offset motion is
    X··· Z··· ;        made at the rapid
  .                    traverse rate.
  .
  .
```

(5) Instructions for commanding tool position offset

Tool position offset is executed by designating the tool offset number corresponding to the actual tool must be designated.

a. Tool offset starts at the block in which the T-code is commanded. When T-code is read, the tool selection signal (BCD) is fed and the tool starts to move by the offset value corresponding to the tool offset number. Since T code is modal, it is retained until the other T code is designated.

EXAMPLE

```
G00 T0202 ; ···  The tool number N02 is
                 selected. Tool offset
                 motion is made accord-
                 ing to the contents of
                 the tool offset number 02.
```

b. When the tool offset value must be changed, the T-code whose tool offset number is rewritten should be commanded again.

EXAMPLE

```
G00 T0202 ;
G01 X··· Z··· F... ;
  .
  .
  .
G01 T0216 ;    Tool offset number 02
               is replaced with 16.
               Tool offset motion is made
               at the cutting feedrate.
```

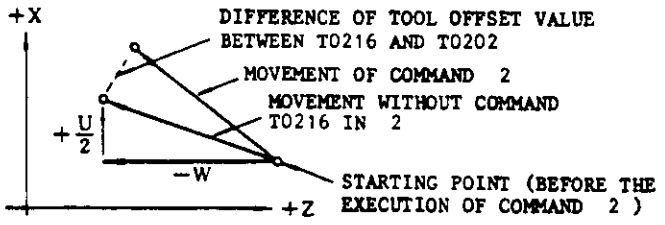
Note that if the tool number is changed in this case, the tool indexing motion starts.

c. The angle of taper cutting can be changed by the following procedure.

T code for change of tool offset number should be commanded in the block together with cutting feed command.

EXAMPLE

(1) G00 T0202 ;
G01 X··· Z··· F··· ;
.
.
.
(2) G01 U+··· W-··· F··· T0216 ;



When the T command and the move command are issued in the same block, the tool nose moves to the offset position. Therefore, in the above case, the taper angle is corrected by the difference of the offset value between T0202 and T0216.

d. When the tool position offset is required to cancel, the T code with the tool offset number 0 or 00 (T □□ 00) must be commanded. The tool position offset is instantaneously cancelled.

EXAMPLE

G00 T0202
G01 X··· Z··· F··· ;
.
.
.
G01 U+··· W-··· F··· T0216 ;
.
.
.
(3) G00 X··· Z··· T0200 ; ···· The offset motion is cancelled. Tool moves according to the position specified by X and Z.

The block (3) of EXAMPLE can be divided into two blocks.

G00 X··· Z··· ;
T0200 ; ····· Only cancel motion is made at rapid traverse rate.

NOTES:

1. Tool position offset is cancelled by RESET operation.
2. The tool offset must be cancelled before M02 or M30 is commanded.
3. The tool offset should be cancelled also before Automatic Zero Return (G28) is commanded.
4. When the control is reset by M02 or M30 command or by executing RESET operation, the tool offset number becomes 0 (or 00).
5. When the Zero Return (auto or manual) is executed, the tool offset is cancelled automatically.
6. The tool offset must be also cancelled before Zero Return Check (G27) is commanded If the G27 is commanded at the state where the tool offset is effective, the control will be the state of Zero Return check error, because the tool offset value is added to the programmed position.

2.6.5 WORK COORDINATE SYSTEM SHIFT†

With this function, coordinate systems set by G50, the Work Coordinate System Setting function, etc. can be shifted through desired distances.

(1) Shift values in the X and Z axes can be written into the Work Coordinate System Shift Memory (one group) with which the offset memory No. is "00," by the same procedure as for writing tool offset values.

(2) The written shift values become effective from the moment described below.

a. When G40 coordinate system is set
b. When G40T work coordinate system is set
c. When automatic coordinate system is set
d. Position Absolute display is reset by ORG key

That is, when these coordinate systems listed above are set, the shift values are simply added. Tools are not shifted.

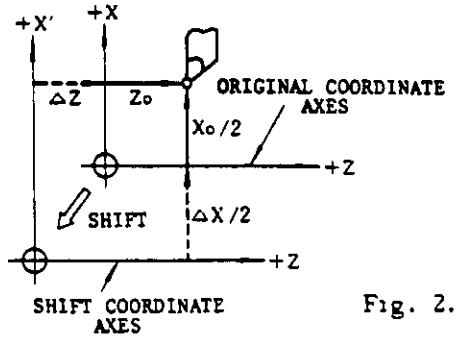


Fig. 2.6

2.6 5 WORK COORDINATE SYSTEM SHIFT (CONT'D)

For positive shift values ΔX and ΔZ, the coordinate axes are shifted in the direction shown above. Xo and Zo are original coordinate system setting values.

(3) This shift function is executed at each time any of the conditions described in a, b, c, and d is met.

(4) When the contents of Work Coordinate System Shift Memory are rewritten, the new shift values become effective from the moment the operation a, b, c, or d above is subsequently executed.

(5) The procedure of "6.2.3 WORK MEASUREMENT VALUE DIRECT INPUT" is effective for the Work Coordinate Shift Memory with an offset memory No. "00."

NOTES

1. The shift command by the Work Coordinate Shift function can not be cancelled unless the setting value is changed to "0." No reset operation is effective in cancelling it.

2. T □□00 ; ····· Tool position offset cancel

   G50 T□□00 ; ··· Work coordinate system setting

   The tool offset No 00 in these instructions has nothing to do with the contents of Work Coordinate Shift Memory.

2.6.6 TOOL WEAR COMPENSATION (T90ΔΔ)†

With certain tool wear value preset in Tool Offset Memory, the preset value can be added to or deducted from the values of any desired tool offset No. in the part program by means of some specific T code command, in order to automatically compensate for tool wear.

(1) With the following T-command, the contents of tool offset Nos. 01 - 19 are changed.

T90 ΔΔ

- ΔΔ — Tool offset No. (01 - 19)
- T90 — Tool wear compensation command

(2) This change is effected as follows. The contents of the Tool Wear Value Memory No. (e.g., 81) corresponding to the specified tool offset No. (e.g., 01) are added to or deducted from the contents of Tool Offset Memory (e g., 01) for both the X and Z axes.

The correspondence between tool offset memory Nos and tool wear value memory Nos. is as shown below.

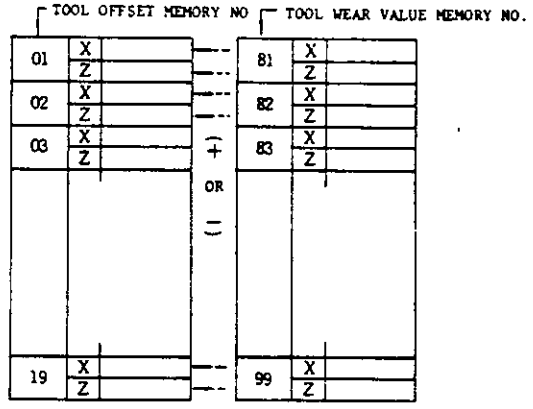


Fig. 2.7

When input signal WOP is on ··· addition

When input signal WOM is on ··· subtraction

EXAMPLE

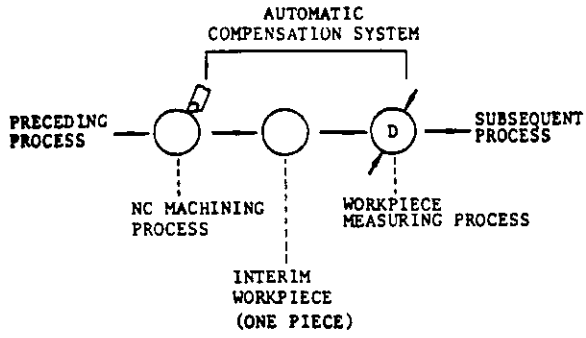
(When WOP is on)

T9012 ; ····· The content of the tool wear value memory No. 92 is added to the content of the tool offset memory No. 12.

NOTES.

1. The offset value change function takes effect when the command T90ΔΔ ; is executed. If both the input signals WOP and WOM are off when this command is executed, no change takes place.

2. While parameter #6023D4 = 1, if a WOP or WOM signal is turned on (closed) twice in succession, the second offset addition or subtraction is not executed. While D4 = 0, the addition or subtraction is executed also in the second time. This function is useful when forming an automatic tool wear compensation system in which the work size measurement by an external measuring instrument is utilized.



For example, with a measuring system in which one interim workpiece is present between the machining process and the measuring process as shown above, #6023D_4 should be set to 1 for proper automatic compensation of tool wear. With a system in which there is no interim workpiece, #6023D_4 should be set to "0." For details, refer to "Connection Manual."

2.7 MISCELLANEOUS FUNCTIONS (M-FUNCTION)

The miscellaneous function is specified with the address M and a maximum 3 digits. The function of each M code (M00 to M99) is determined by the machine, except for several M codes. Refer to the machine tool builder's manual for the function of M codes except for the following M codes concerned with the control.

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

To stop the NC control and machine, the following codes are provided.

M00. Program stop

M01: Optional stop

M02: End of program

M30: End of tape

These commands stop the advance reading of the control. For these M codes, M 2-digit BCD code and their respective decoded signals are outputted.

2.7.2 M CODES FOR INTERNAL PROCESSING (M90 TO M109)

M90 through M109 are for internal processing. Even when they are programmed, no external output signal (BCD and decoded output) is sent.

M90 †: Program interrupt off

M91 †: Program interrupt on

M92 †: Buffering of 1 block

M93 †: Buffering of 4 blocks

M94 †: Remote tool compensation for X-axis

M95 †: Remote tool compensation for Z-axis

M96 †: Tool radius compensation: circular path mode

M97 †: Tool radius compensation: intersection computing mode

M98: Subroutine program call

M99: Subroutine program end

M100 to 109: Not used (for special application)

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

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

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

Note: When power is applied, the current M code is changed to the M code marked with ◥. However, it is not changed by RESET operation.

· M91 P··· ;

During the time from this command to an M90 command, whenever a program interruption signal is received, the program under execution is interrupted (if the machine is in motion, it is stopped after deceleration), and jumped to the program whose number is specified by P.

· M90 ;

With this command, the program interrupt function is cancelled.

2.7.4 BUFFERING FUNCTION (M93, M92)†

(1) The following M codes are issued for buffering function.

M code	Meaning
M92	1-block buffering
M93	4-block buffering

Note: When power is applied, the current M code is changed to the M code marked with ◥. However, it is not changed by RESET operation.

2.7.4 BUFFERING FUNCTION (M93, M92) (CONT'D)

(2) 4-block buffering (M93)

When M93 ; command is given, the control enters the 4-block buffering mode, which remains until M92 is commanded subsequently. In this mode, up to 4 blocks of data are read in advance for subsequent operation. With programs in which the operation time for the 4 blocks read in advance is longer than the reading and processing time of the subsequent 4 blocks, interruption between blocks can be eliminated. This function is effective in avoiding a shiny streak on the workpiece caused by feed stop between blocks.

(3) 1-block buffering (M92)

When M92 command is given, the 4-block buffering mode is cancelled, and the 1 block buffering mode is restored.

NOTE. While the tool radius is being compensated for with the M93 function, up to two blocks not containing move commands are permitted, and as the result, up to 6 blocks may be read in advance.

EXAMPLE

```
N51 M93 ; ——— Start of 4-block advance reading.
N52 G01 U··· F··· ;  ⎫
N53 X··· Z··· ;      ⎬ Stop between blocks for tool radius compensation or other calculation can be avoided.
N54                  ⎭
 .
 .
 .
M58 M92 ; ——— Cancelling 4-block advance reading.
```

2.7.5 REMOTE TOOL OFFSET MODIFICATION (M94, M95)†

With this function, the contents of a specified tool offset No. can be modified by the machining error data obtained by an external measuring instrument and fed back to the control. With this function, an automatic tool wear compensation system can be formed.

(1) For this function, the following M codes are used.

M code	Meaning
M94	X-axis remote tool offset modification
M95	Z-axis remote tool offset modification

These are non-modal M codes.

(2) Compensation in X-axis

M94 U··· ;

With this command, the control outputs a data request signal to the outside, receives signed 3-digit BCD data, adds it to the X-axis offset value of the specified tool offset No., and makes the sum value as the new X-axis tool offset value.

(3) Compensation in Z-axis

M95 Z··· ;

With this command, the same operation is performed as above with Z axis.

(4) At this time, if "input command x 10 signal(DIX)" of the control is on, the input data is multiplied by 10 before being added to the data stored in Tool Offset Memory.

Range of remote tool offset modification data

	Command 10 times input off	Command 10 times input on
Metric input	0 - ±0.999 mm	0 - ±9.990 mm
Inch input	0 - ±0.0999 in.	0 - ±0.9990 in

(5) This function differs in detail with the type of the formed compensation system. For details, refer to the machine tool builder's manual.

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

These M codes are effective when the control is provided with the tool nose radius compensation option.

(1) The following M codes are used.

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

Note: When power is applied, the current M code is changed to the M code marked with ◥. However, it is not changed by RESET operation.

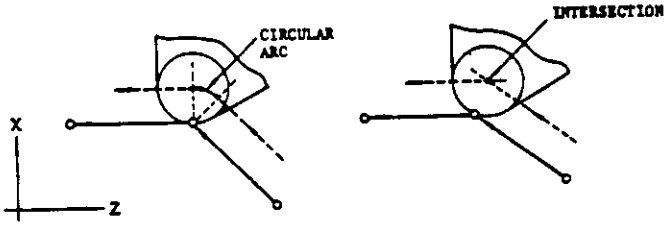
(2) With the tool radius compensation mode by G41 to G44, the locus of the tool (center of tool radius) for commanded workpiece contour lines with the angle between tangents larger than 180° is in the following two categories.

a. M96 mode

The center of the tool nose radius describes a circular arc around the perimeter in the contour line.

b. M97 mode

The center of the tool nose radius moves along the locus that is formed by straight lines shifted from the contour line by the distance equal to the tool radius.



M96 mode (circular arc) M97 mode (calculation of intersection)

Fig. 2.8

(3) Commands of M96 and M97 become effective from the edge in the following command blocks.

a. G01 X··· Z··· F··· ;
(G01) X··· Z··· M96 (or M97) ;
From the move around the edge in this block.

b. G01 X··· Z··· F··· ;
M96 (or M97) ;
(G01) X··· Z··· ;
From the move around the edge in this block.

2.7.7 SOUBROUTINE PROGRAM (M98, M99)

With this function, subroutine programs which have been numbered and stored in advance are called and executed as many times as desired.

(1) The following M codes are used for this function.

M code	Meaning
M98	Call of subroutine program
M99	End of subroutine program

(2) Call of subroutine program (M98)

M98 P··· Q··· L··· ;

With this command, the subroutine program starting with a sequence No. following Q in the part program with the program No. specified by P is called and is executed L times.

However, when

- P is omitted:
subroutine program following the sequence No. Q in the main program is called.
- Q is omitted:
subroutine program starting at the leading end of the program No. specified by P is called.
- L is omitted: execution is only once.

Subroutine programs can be nested up to 4 times.

(3) End of subroutine program (G99)

M99 ; is written at the end of subroutine program to end it.
When this code is written, the operation returns to the block immediately following the main block in which the subroutine program was called after the execution of the subroutine program.

M99 P··· ;

When this is written at the end of a subroutine program, the operation returns to the sequence No. specified by P in the main program.

(4) Simple jump command

M99 P ··· ;

When this command is used in the main program, the operation simply jumps to the sequence No. specified by Q in the main program. If Q is omitted, the program simply jumps to the leading end of the main program.

```
N1 G50 X0 Z0 ;
N2 G00 ··· ;
    .
    .
    .
N9 M99 ;
```

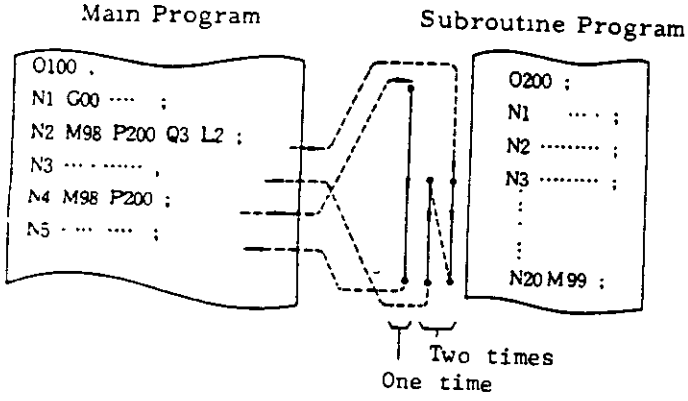
Writing multi blocks (10 lines maximum) of this program and executing cycle start make endless operation.

2.7.7 SOUBROUTINE PROGRAM (M98, M99) (CONT'D)

NOTES.

1. When the program No. specified by address P and the sequence No. specified by Q are not found, alarm code 041 is displayed.
2. While command L for the number of repetitions is under execution, the remaining number of repititions can be displayed. For details refer to 4.3.2.2.
3. This function can be used when subroutine programs are stored in the part program memory. Main programs can be commanded through NC tapes or the part program memory.
4. When subroutine programs are nested more than 4 times, alarm code "042" is displayed.

EXAMPLE



2.7.8 OTHER M CODES

(1) How to use the other M codes other than the above depends upon the machine. Refer to the machine tool builder's manual.

Table 2.7 Typical Example of M Codes for Machine

M code	Meaning	Remarks
M03	Spindle forward running	Direct switching from M03 to M04 cannot be done. M05 must be inserted between them.
M04	Spindle reverse running	
M05	Spindle stop	
M08	Coolant on	
M09	Coolant off	

(2) When these M codes are commanded in the same block with move command, execution will depend on the machine tool design and construction. (Whether the M commands are executed simultaneously with or after completion of move command.)

(3) For these M-code commands, the control outputs M 2-digit BCD codes.

2.7.9 M 3-DIGIT BCD OUTPUT†

When the control is provided with the M 3-digit BCD output option, it can command M 3-digit codes between M00 and M999.

(1) M codes between M00 and M89, and between M110 and M999 are output in 3-digit BCD codes.

(2) M90 through M109 are internal processing M codes, and no BCD code for them is output. See 2.7.2 M CODES FOR INTERNAL PROCESSING.

(3) With M00, M01, and M30, decode signals are output in addition to the BCD output. See 2.7.1 M CODES FOR STOP.

(4) The specific usages of the M 3-digit codes depends on machine tool design. Refer to machine tool builder's manual.

2.8 PREPARATORY FUNCTIONS (G-FUNCTION)

2.8.1 LIST OF G CODES

Address G, plus up to 3 digits specify the meaning of the block. Table 2.8.1 gives G codes and their groups.

(1) G codes are broadly classified into the following two types.

	Meaning
Modal G-code	G-code effective until the other G-code of the same group is commanded.
Non-modal G-code	G-code effective only in the commanded block.

(2) G codes in groups from 01 through 11 are modal. When the control is energized with the power switch, the G codes marked with ◤ in Table 2.8.1 are automatically selected.

(3) G codes of * group in the Table 2.8.1 are non-modal. They should not be commanded together with the other G codes in one block.

(4) The modal G codes can be commanded mixedly in a block.

(5) G codes in Class B are basic, and those in Class O are options. The use of optional G codes is determined by the machine tool design. See the machine tool builder's manual.

(6) Standard G codes can be coverted to special G codes I by parameters. This is a basic feature, and, when parameter #6005D7 is set to 1, standard G codes are converted to special G code I.

(7) When the special G code II option is incorporated in the control, the setting of parameter #6005D7 to 1 will convert G codes to special G codes II. Setting the parameter to 0 will reconvert the G codes to the standard G codes.

2.8.2 POSITIONING (G00, G06)

2.8.2.1 Positioning (G00)

(1) G00 X(U)··· Z(W)··· ;

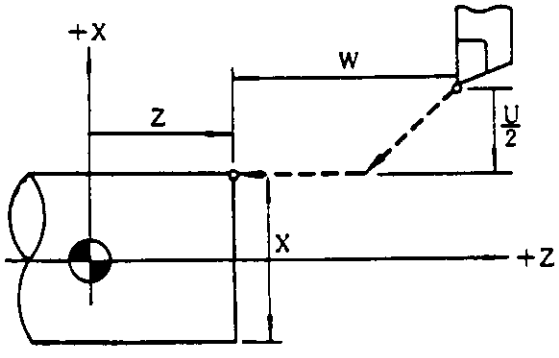
This command moves a tool at rapid traverse rate to the point (X, Z) in the coordinate system set by the G50 command or moves it away by (U, W) from the present point for each axis independently.

(2) For the rapid traverse rate, as it depends upon the machine, refer to the machine tool builder's manual.

EXAMPLE

X-Axis: 12 m/min

Z-Axis: 6 m/min



(3) Along the axes specified by G00, the machine slide moves in rapid traverse rates, independently of each other. The resultant tool locus may not be a straight line, and when working out the program, care must be taken to avoid fouling between the tool and the workpiece.

(4) G00 is a modal G code in the 01 group. When it is commanded, it remains effective until other G codes in the 01 group are commanded.

(5) For the positioning with G00, the pulse distribution is started only after the ERROR DETECT state is turned on, and the program advances to the next block only upon the activation of the ERROR DETECT state after the completion of the pulse distribution. When this G code is used, therefore, the workpiece edges are machined true, and rounding is avoided.

(6) NOTES

a. The ERROR DETECT ON state means the decrease of the servo lag pulses to the permissible level after the pulse distribution for move command.

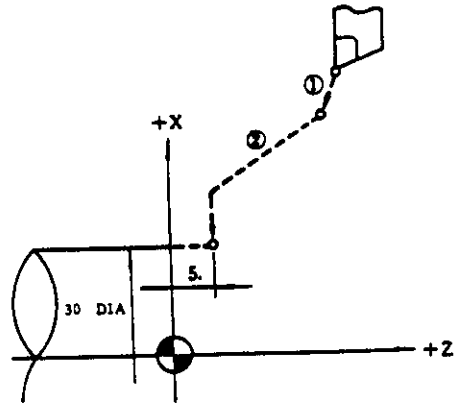
b. When T code is commanded, G00 should be put in the T-code block. G00 is required for designation of tool traverse rate for tool offset motion using T code.

EXAMPLE

G50 X150. Z100. ;

① G00 T0101 S1000 M03 ; ···
··· G00 for designation of traverse rate for tool offset motion

② (G00) X30. Z5. ; ··· G00 can be omitted in positioning.



2.8 PREPARATORY FUNCTIONS (G-FUNCTION) (CONT'D)

Table 2.8 List of G Codes

B Basic
O. Optional

G Code	Special G Code I	Special G Code II	Group	Function		Section
◤ G00	◤ G00	◤ G00	01	Positioning (rapid traverse feed)		B
◤ G01	◤ G01	◤ G01		Linear interpolation, angle programming for linear interpolation		B, O
G02	G02	G02		Circular interpolation CW,(radius R designation)		B, O
G03	G03	G03		Circular interpolation CCW, (radius R designation)		B, O
G04	G04	G04	*	Dwell		B
G06	G06	G06		ERROR DETECT OFF positioning		B
G10	G10	G10		Tool offset value setup		O
G11	G11	G11	01	Beveling	Cornering	O
G12	G12	G12		Rounding		O
G20	G20	G70	05	Inch input specification		O
G21	G21	G71		Metric input specification		O
G22	G22	G22	01	Radius programming for circular interpolation CW		O
G23	G23	G23		Radius programming for circular interpolation CCW		O
G27	G27	G27	*	Reference point return check		B
G28	G28	G28		Automatic return to reference point		B
G29	G29	G29		Return from reference point		B
G30	G30	G30		Return to 2nd reference point		O
G31	G31	G31		Skip function		O
G32	G33	G33	01	Threadcutting, continuous threadcutting, multi-start threadcutting		B, O
G34	G34	G34		Variable lead threadcutting		O
G35	G35	G35	*	Tool set error compensation		O
G36	G36	G36	07	Stored stroke limit 2nd area ON		O
G37	G37	G37		Stored stroke limit 2nd area OFF		O
G38	G38	G38	08	Stored stroke limit 3rd area ON		O
G39	G39	G39		Stored stroke limit 3rd area OFF		O
◤ G40	◤ G40	◤ G40	06	Tool radius compensation cancel		O
G41	G41	G41		Tool radius compensation No. 1		O
G42	G42	G42		Tool radius compensation No 2		O
G43	G43	G43		Tool radius compensation No. 3		O
G44	G44	G44		Tool radius compensation No. 4		O

◤ shows the G codes selected when the control is powered or reset

NOTES

1. The following G codes for initial state when power is applied can be set by parameters

Group	G code	Parameter
01	G00 or G01	#6005D_2
04	G98 or G99	#6005D_1
03	G90 or G91	#6005D_0

2. When the control is reset, whether G code of 01 group should be G00 or kept as the current one can be set by parameter #6005D_6
3. Radius programming for circular interpolation can be made by G02 and G03 instead of G22 G23 respectively.
4. Cornering can be programmed by G01 instead of G11 and G12. Refer to 2.8.7 Cornering.
5. Initial states of G codes of 06, 08, 09 groups when power is applied are determined by their respective setting data (#6001D_0, D_1, D_2)

B Basic
O Optional

Table 2.8 List of G Codes (Cont'd)

G Code	Special G Code I	Special G Code II	Group	Function		Section
G50	G92	G92	*	Coordinate system setup		B
				Maximum spindle revolution setup, work coordinate system setup		O
G51	G51	G51	*	Return of current display value to origin		O
G65	G65	G65		User macro simple call		O
G66	G66	G66	09	User macro modal call		O
◤G67	◤G67	◤G67		User macro modal call cancel		O
G68	G68	G68	10	Mirror image by programming ON		O
◤G69	◤G69	◤G69		Mirror image by programming OFF		O
G70	G70	G72	*	Finishing cycle	Multiple repetitive cycles	O
G71	G71	G73		Stock removal in turning		O
G72	G72	G74		Stock removal in facing		O
G73	G73	G75		Pattern repeating		O
G74	G74	G76		Peck drilling in Z-axis		O
G75	G75	G77		Grooving in X-axis		O
G76	G76	G78		Automatic threadcutting cycle		O
G90	G77	G20	01	Turning cycle A		B
G92	G78	G21		Threading cycle		B
G94	G79	G24		Facing cycle B		B
G96	G96	G96	02	Constant surface speed control		O
◤G97	◤G97	◤G97		Constant surface speed control cancel		O
◤G98	◤G94	◤G94	04	Feed per minute (mm/min)		B
◤G99	◤G95	◤G95		Feed per revolution (mm/rev)		B
	◤G90	◤G90	03	Absolute command		B
	◤G91	◤G91		Incremental command		B
G122	G122	G122	11	Tool registration start	Tool life control	O
◤G123	◤G123	◤G123		Tool registration end		O
G111	G111	G111	*	Taper multiple beveling/rounding		O
G112	G112	G112		Arc multiple beveling/rounding		O

◤ shows the G codes selected when the control is powered or reset

NOTES:

1. The following G codes for initial state when power is applied can be set by parameters.

Group	G code	Parameter
01	G00 or G01	#6005D2
04	G98 or G99	#6005D1
03	G90 or G91	#6005D0

2. When the control is reset, whether G code of 01 group should be G00 or kept as the current one can be set by parameter #6005D6
3. Radius programming for circular interpolation can be made by G02 and G03 instead of G22 G23, respectively.
4. Cornering can be programmed by G01 instead of G11 and G12. Refer to 2.8.7 Cornering.
5. Initial states of G codes of 06, 08, 09 groups when power is applied are determined by their respective setting dat (#6001D6 D5 D2)

2.8.2.2 ERROR DETECT OFF Positioning (G06)

(1) G06 X(U)··· Z(W)··· ;

With this command the positioning process is identical to that of G00 except for the following aspects.

a. G06 is a non-modal G code in the * group. It is effective only in the programmed block.

EXAMPLE

G00 X··· Z··· ;
G06 X··· Z··· ; —— Move by G06
X··· Z··· ; —— Move by G00

b. With the positioning of G06, the positioning pulse distribution is immediately started on the completion of the pulse distribution for the preceding block, after making an ERROR DETECT check, and the program advances to the next block after the completion of the pulse distribution process. For this reason, wrokpiece edges are rounded to the extent of servo lag pulses.

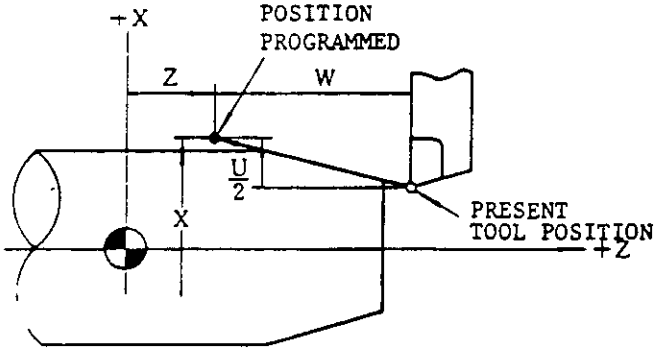
NOTE The ERROR DETECT ON/OFF signals (SM2) are effective only for cutting feeds, and have no influence on the motion under G00 and G06.

2.8.3 LINEAR INTERPOLATION (G01)†

· G01 X(U)··· Z(W)··· F(E)··· ;

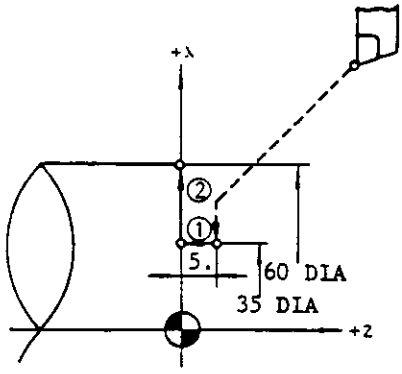
A tool is moved to the point (X, Z) on a straight line at the traverse rate designated by the F or E code in the coordinate system set by G50 moved away by (U, W) from the present point.

F or E code must be specified in the block containing the G01 or in the previous block. If not, it causes a format error. Feedrate designated by the F or E code is the tangential feedrate.



EXAMPLE

G50 X100. Z60. ;
G00 T0202 S600 M03 ;
X35. Z5. ;
① G01 Z0 F1. ;
② X60. F0.2 ; ---- Executed by linear interpolation G01.



· Angle programming for linear interpolation†

With the control equipped with this option, linear interpolation can be commanded at specified angles.

G01 X(U)··· A··· F(E)··· ; (a)

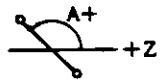
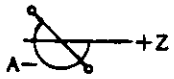
or

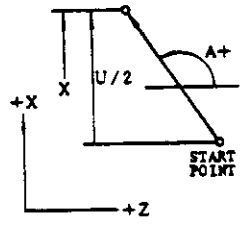
G01 Z(W)··· A··· F(E)··· ; (b)

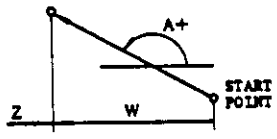
With these commands, a linear interpolation will be executed by specifying angle A in the + direction of the Z-axis and distance either in X- or Z-axis direction. The feedrate in the tangential direction is specified by the F or E code. The range of angle specifiable with address A is as follows.

	Programmable range of angle A
Metric Input	0 - ±360.000°
Inch Input	

Table 2.9

Sign	Meaning	
A+	Angle counterclock-clockwise from +Z-axis.	
A-	Angle clockwise from +Z-axis.	





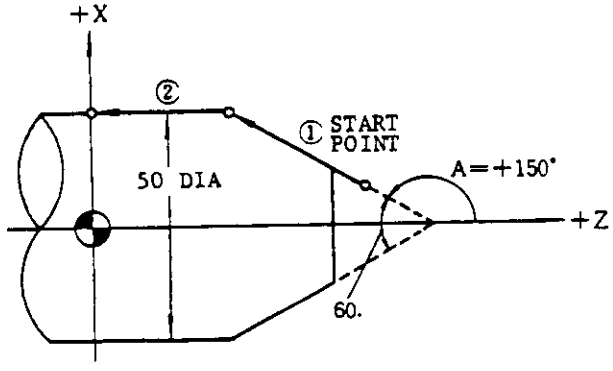
EXAMPLE

.

.

(1) G01 X50. A+150. F0.3 ;

(2) G01 Z0. A-180. ,



2.8.4 CIRCULAR INTERPOLATION (G02, G03)

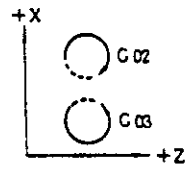
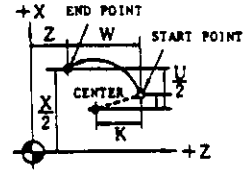
- G02(G03) X(U)··· Z(W)··· I··· K··· F(E)··· ;

  A tool is moved on the circular arc whose center is away from the present position by (I, K). The end point of the arc is (X, Z) in the coordinate system set by G50 or away from the present position by (U, W)

  A tool moves along a circular arc at the traverse rate specified by the F or E code.

- The meanings of G02, G03 and each address are shown below.

Table 2.10

	Meanings	
G02	Circular interpolation, clockwise	
G03	Circular interpolation, counterclockwise	
X(U)	End point of arc on X-axis (Diameter value)	
Z(W)	End point of arc on Z-axis	
I	Distance from start point of arc to arc center on X-axis (Radius value)	
K	Distance from start point of arc to arc-center on Z-axis	

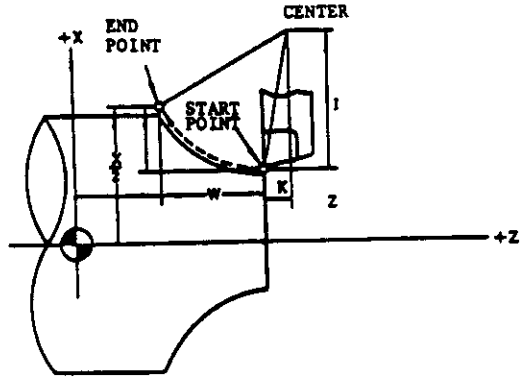
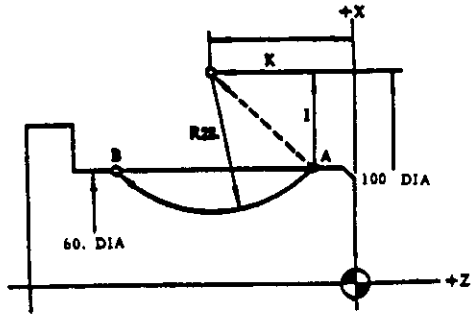


Fig. 2.9

Circular interpolation of an arc on multiquadrant can be programmed in a single block.

EXAMPLE



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

Table 2.11

Arc center coordinate	(10000, -2700)
I	$\frac{100 - 60}{2}$ = 20 mm
K	$-\sqrt{28^2 - 20^2} = \sqrt{-384}$ = -19.596 → -19 60 mm

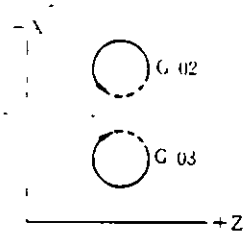
The above case can be programmed as follows.

```
.
.
G01 Z-·· F··· ;
G02 X60. Z-46.6 I20. K-19.6 F··· ;
.
.
```

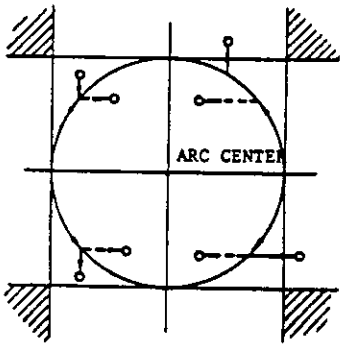
· The feedrate commanded by the F code is a tangential feedrate.

NOTES:

1. The direction of the arc of G02 for Clockwise is defined as follows.
   "When viewing the X.Z plane in -Y direction in the right-hand coordinate system, the tool moves clockwise from the beginning point of the arc."
   Therefore, the direction of rotation in the plane (-X.Z plane) Fig. 2.21 is presented inversely.



2. When the end point of arc is not designated on the circumference specified by the radius, the alarm is not displayed and the tool path is as follows. The mark o indicates the end point of arc.



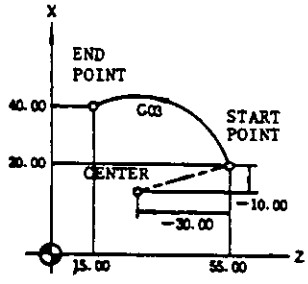
Note that if the end point is designated in the shaded area, the alarm is not displayed and the tool will continue to move endlessly.

The end point coordinate should be precisely commanded when the circular interpolation is applied to the tool nose radius compensation, or the tool may not move properly. Generally, it is recommendable to calculate up to the next digit of least input increment and count fractions over 1/2 as one and disregard the rest.

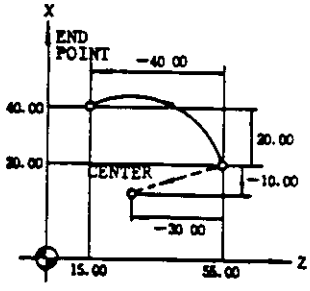
3. When the control is provided with Radius Programming for Circular Interpolation, radius value can be commanded by G02, G03 instead of G22, G23.

EXAMPLE

a. G03 X80.0 Z15.0 I-10.0 K-30.0 F150 ;



b. G03 U40.0 W-40 0 I-10.0 K-30.0 F150 ;



2.8.5 DWELL (G04)

(1) G04 U(P)··· ;

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

(2) Dwell is programmed as an independent block.

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

Dwell time:	0.001 to 9999.99 seconds

Dwell time is not influenced by input/output increment.

EXAMPLE:

G04 U3.5 ··· 3.5-second dwell

G04 P3500 ··· 3.5-second dwell

NOTES:

1. G04 is a non-modal G code.
2. The counting of dwell time is started from the instant the control enters the ERROR DETECT ON state upon completion of the move command block before G04. Therefore, with G04 U0 ;, the control advances to the next block immediately after detecting the ERROR DETECT ON state.

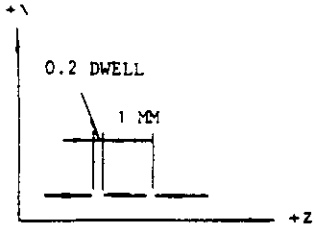
EXAMPLE

G01 W-1. F25 ; — 1 mm feed

G04 U0.2 ; — 0.2 second dwell time

G01 W-1. ; — 1 mm feed

With the above program, chip cutting feed is obtained.



2.8.6 TOOL OFFSET VALUE (G10)+

With G10 command, tool offset values can be set and corrected.

(1) G10 P··· X(U)··· Z(W)··· R··· ;

With this command, tool offset values are set or corrected in part programs.

Table 2.12

	Meaning
P	For specifying tool offset No.
X Z	For changing the tool offset value to the specified value.
U W	For adding the specified value to the original tool offset values.
R	For changing the tool radius to the specified value.

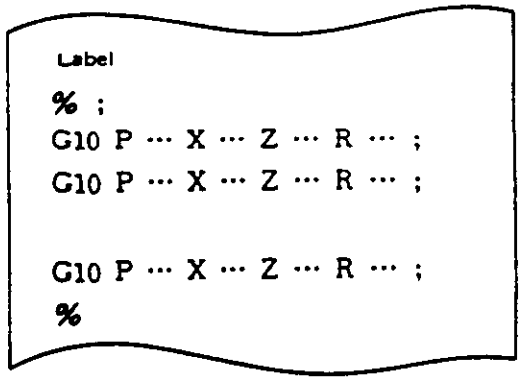
The offset values for which no address is programmed are not changed.

EXAMPLE

G10 P16 X32.5 W0.05 ;

- W0.05: 0.05 mm is added to Z-axis value
- X32.5: X-axis value is set to 32.5 m
- P16: Contents of tool offset No. 16 is changed

(2) The above format is used to make offset value tapes, and to store the values in Offset Memory at once.
The tape format is as follows.



2.8.7 CORNERING (G11, G12)±

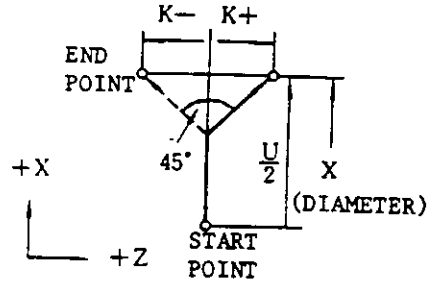
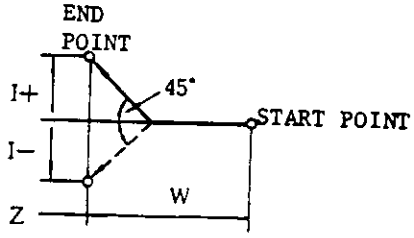
(1) Beveling (G11)

$$G11 \left\{ \begin{matrix} X(U)\cdots\ K\cdots \\ Z(W)\cdots\ I\cdots \end{matrix} \right\} F(E)\cdots\ ;$$

This command removes the sharp corner of workpiece. Addresses X and Z cannot be specified simultaneously in a block.

Meaning of each address is shown below.

Table 2.13

Beveling for X-axis	Beveling for Z-axis
G11 X(U)··· K··· F(E)··· ;	G11 Z(W)··· I··· F(E)··· ;
	
K ± ··· Beveling Value Beveling Direction	I ± ··· Beveling Value (Radius Value) Beveling Direction

Beveling values (K and I) are limited within the following values.

$$|K| < |U/2| \quad , \quad |I| < |W|$$

The command exceeding the above value causes format error.

```
G00 X30. Z0 ,
① G11 Z-20. I8. F30 ;
② (G11) X80 K-7. ;
```

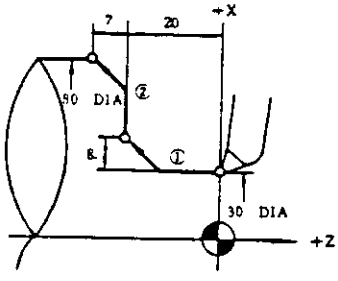


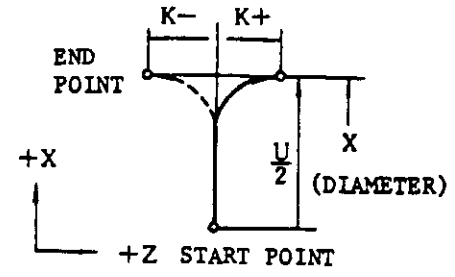
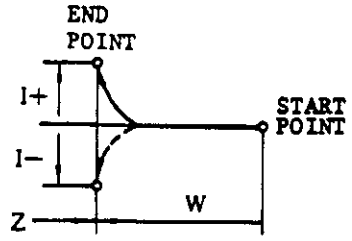
Fig. 2.10

(2) Rounding (G12)

$$G12 \left\{ \begin{matrix} X(U)\cdots\ K\cdots \\ Z(W)\cdots\ I\cdots \end{matrix} \right\} F(E)\cdots\ ,$$

This command performs the rounding of the corner. Addresses X and Z cannot be specified simultaneously in a block. The corner is formed as a quarter-round. Meaning of each address is shown below.

Table 2.14

Rounding for X axis	Rounding for Z axis
G12 X(U)··· K··· F(E)··· ;	G12 Z(W)··· I··· F(E)··· ;
	
K ± ··· Rounding value Rounding direction	K ± ··· Rounding value Rounding direction

Rounding values (K and I) are limited within the following values.

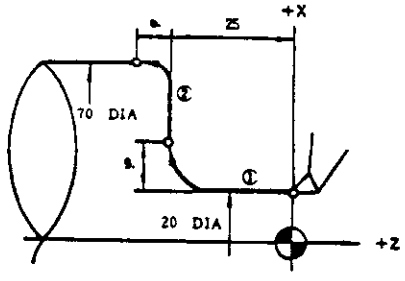
$$|K| < |U/2| \quad , \quad |I| < |W|$$

The command exceeding the above value causes format error.

```
G00 X20. Z0 ;
```

① `G12 Z-25. I9. F30 ;`

② `(G12) X70 K-6 F20 ;`



NOTES:

1. G11 and G12 are modal G codes in the A group. They remain effective until other G codes in the group A are commanded.
2. G11 and G12 are for one axis only. If they are commanded for both axes in the same block, they constitute a format error.

   EXAMPLE

   G12 X··· W··· K··· ; Error "050"
3. In the G10 or G12 modes, no block without I and K nor block in which I and K are 0 can be commanded. If such a block is commanded, correct tool movement can not be assured.
4. Tool radius compensation function^{+} is effective to the blocks containing G11 or G12.
5. In the finish form commands G70 through G73 of the special canned cycle^{+}, blocks containing G11 or G12 can be commanded.
6. G01 code can be used instead of G11 to specify identical beveling.

   G01 { X(U)··· K··· / Z(W)··· I··· } F(E)··· ;
7. G01 code can be used instead of G12 to specify rounding. However, in this case, R must be used instead of I and K.

   G01 { X(U)··· R··· / Z(W)··· R··· } F(E)··· ;

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

Unit of measurement (metric or inch) of input data is selectively specified by the following G codes.

G code	Input unit
G20	Inch input
G21	Metric input

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

a. Subsequent part programs

b. Tool offset values

c. Part of settings and parameters

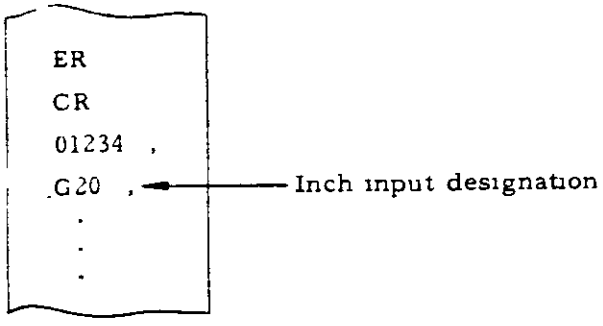
d. Part of manual movements

e. Displays

NOTES:

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

EXAMPLE



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

a Cancel work coordinate system (G50T), if used.

b. Cancel tool position offset, and tool radius compensation (G41 - G44).

3 Take the following procedure after the command of G20/G21 selection.

a. Program absolute zero point for all axes before move command (G50).

b. In principle, make the display reset operation when current position display (external) is used.

4. The tool offset values are processed differently in the G20 mode and the G21 mode. G20/G21 must be commanded after modifying the tool offset values.

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

2.8.9 RADIUS PROGRAMMING FOR CIRCULAR INTERPOLATION (G22, G23)†

In programming circular interpolation (G02, G03), the control requires the data of the arc-center coordinates. Normally, they are given by using the addresses I and K.

(1) In programming of G22 or G23, the control automatically calculates the arc center coordinates (I, K) from the radius value designated by the address R and performs circular interpolation.

$$\left.\begin{matrix} G22 \\ (G23) \end{matrix}\right\} X(U)\cdots\ Z(W)\cdots\ R\cdots\ F(E)\cdots\ ;$$

A tool moves along the circular arc whose center is radius R away from the present position. The end point of arc is at coordinates (X, Z) set by G50 or is away from the present position by (U, W). Tool moves along the circular arc at feedrate designated by F code.

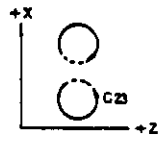
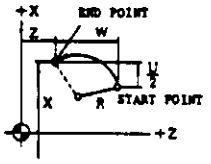
(2) The meanings of G22, G23 and each address are shown below.

(3) Designation of radius value R

Radius value R is commanded by incremental value with a sign of radius programming.

In this case,
When radius value R > 0, an arc, describing less than 180°, and when R < 0, an arc describing more than 180° are specified.

Table 2.15

	Meaning	
G22	Circular interpolation by radius for CW	
G23	Circular interpolation by radius for CCW	
X(U)	The X-coordinate of the end of the arc (Diameter value)	
Z(W)	The Z-coordinate of the end of the arc	
R	Distance from the start point of arc to arc center (Incremental value with sign)	

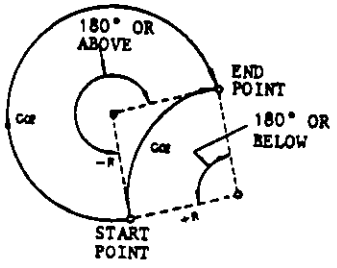


Fig. 2.11

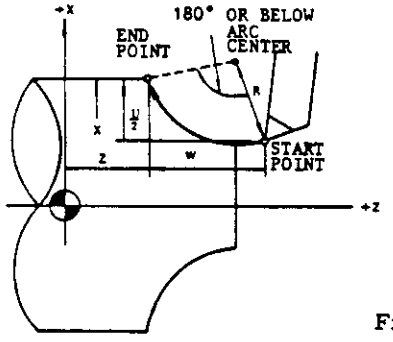
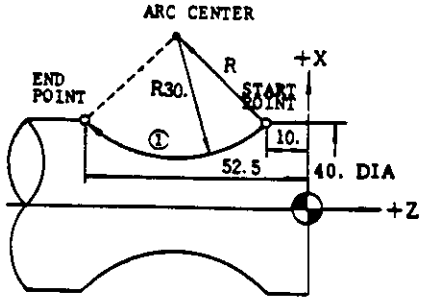


Fig. 2.12

EXAMPLE

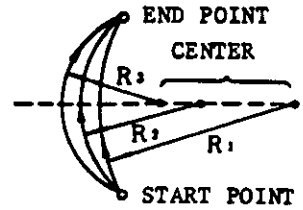
G01 X40. Z-10. F20 ;

① G02 (X40.) Z-52.5 R30. (F20) ;



NOTES:

1. G22 and G23 codes are modal. They are kept until other G code of 01 group is commanded.
2. In the G22 or G23 mode, the block in which R is not contained or R is designated as zero should not be commanded. Radius cannot be designated by I and K.
3. When R is varied with both start and end points fixed, the tool will move along the following circular arc.



Therefore, in the following case, the arc center does not exist which causes data error.

$$R < \frac{\text{(Distance between start point and end point)}}{2}$$

4. Tool nose radius compensation is effective for the block containing G22 or G23.
5. The block containing G22 or G23 can be designated in finishing shape commands of special canned cycles (G70 to G73).
6. When the control is provided with radius programming option, circular interpolation by radius (R) programming can be made by G02, G03 instead of G22, G23.

2.8.10 REFERENCE POINT CHECK (G27)

(1) G27 X(U)··· Z(W)··· ;

With this command, the tool is positioned to the absolute coordinate point (X, Z) or incremental coordinate point (U, W) by moving along the two axes simultaneously, and then, the position is checked for conformance with the reference point. For the axis for which no command is given, positioning and checking are not executed.

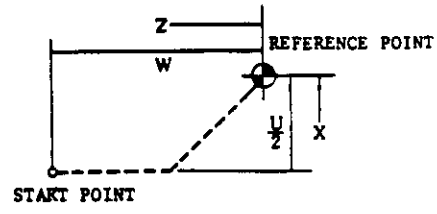


Fig. 2.13

2.8.10 REFERENCE POINT CHECK (G27) (CONT'D)

(2) If the position is the reference point, the return-to-reference lamp lights. The position is the reference point in all the axial directions specified, the automatic operation is continued further. If the position is not the reference point even along one axis, this constitutes the return-position-error, and the automatic operation is interrupted. (Cycle start lamp goes off.)

NOTES.

1. The reference point is an fixed point on the machine tool to which the tool can return by the motion under the control of the automatic reference point return or G28 automatic reference point return function. See 6.2.1 Automatic Reference Point Return.
2. If G27 is commanded in the tool position offset mode, the tool returns to the position displaced from the reference point by the tool offset value. Positioning cannot be made at the reference point. Before commanding G27, cancel the tool offset mode.

   EXAMPLE

   a. Cancelling tool offset in the block preceding G27.

      T □ □ 00 ;

      G27 U··· W··· ;

   b. Cancelling in the block containing G27.

      G27 U··· W··· T □ □ 00 ;

3. The mirror image function is effective with the motion commanded by G27. To avoid the return position error, command G27 in the G69 mode (Opposite tool post mirror image off).

2.8.11 AUTOMATIC RETURN TO REFERENCE POINT (G28)

(1) G28 X(U)··· Z(W)··· ;

With this command, the tool can be brought back to the reference point automatically after passing through an interim point. In other words, the tool positions to the commanded absolute coordinate position (X, Z) or incremental position (U, W) by moving simultaneously along the two axes, and then automatically returns to the reference point by the reference point return function. The specified point (X, Z) or (U, W) is known as "INTERIM POSITIONING POINT," or "INTERIM POINT."

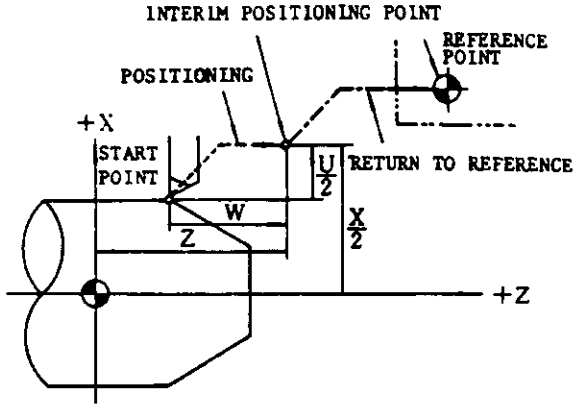


Fig. 2.14

The tool does not move along the axis for which instruction is omitted.

(2) When the return to reference motion is completed, Reference Point Return lamp for the returned axis lights. When the tool returns to the reference point in both axes, the automatic operation is resumed.

(3) The series of RETURN TO REFERENCE motions are as follows. With initial power application, the return motions to the reference point are as shown below in the low-speed mode as shown below.

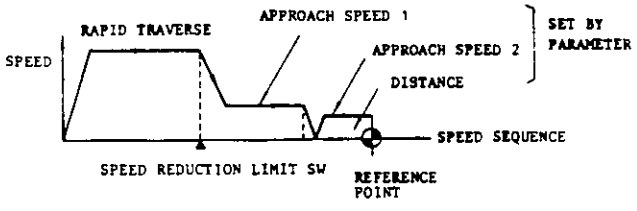


Fig. 2.15

Thereafter, the RETURN TO REFERENCE POINT motion is in rapid traverse as under the command of G00.

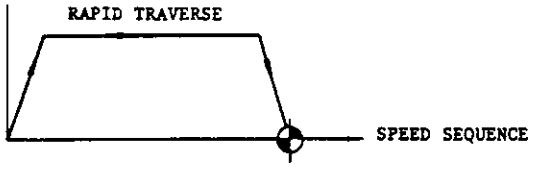
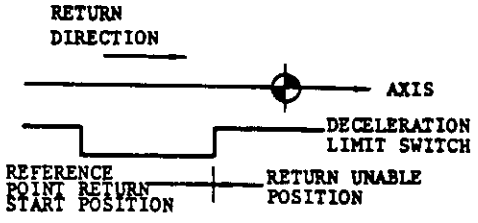


Fig. 2.16

(4) However, when low traverse speed is specified by parameter #6010D$_5$ set to 1, the same low motion speed as in the 1st time is obtained.

NOTES:

1. For parameter setting and other details of the low traverse speed return motion shown in Fig. 2.8.11.2, refer to 6.2.1 Manual Return to Reference Point.
2. The starting point for RETURN TO REFERENCE POINT motion must be in the area shown in Fig. 2.8.11.4. Fig. 2.8.11.3 can be started from any position.



3. Before writing G28 in the program, cancel the tool position offset as shown below.

   a. Cancelling in the preceding block

   T □□00 ;

   G28 X··· Z··· ;

   b. Cancelling in the block containg G28

   G28 X··· Z··· T □□00 ;

4. When G28 is written with the tool position offset or tool radius compensation on, the offset or compensation is automatically cancelled.

2.8.12 RETURN FROM REFERENCE POINT (G29)

(1) With this function, the tool is positioned to a specified point via the interim point, after it has been once returned to the reference zero point by the AUTOMATIC RETURN TO REFERENCE ZERO COMMAND (G28).

G28 X··· Z··· ; Point A → B → C (reference zero point)

Point B

G29 X··· Z··· ; Point C → B → D

Point D

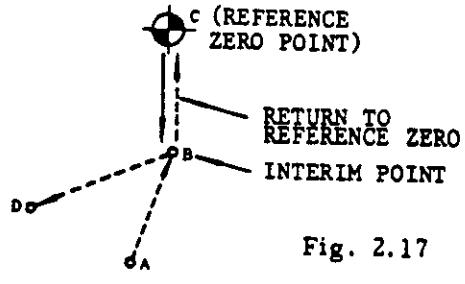


Fig. 2.17

(2) When G29 is used, consideration on the distance between points B and C is unnecessary in programming. Especially when incremental instructions are used, this function is useful to return the tool to the original coordinate system after returning to the reference zero.

(3) Motions C → B and B → D are made simultaneously along the two axes in rapid traverse. However, the tool will not move in the direction for which instruction is omitted.

(4) Where G28 is programmed several times, the point B created by the latest G28 instruction is effective for the motion by G29.

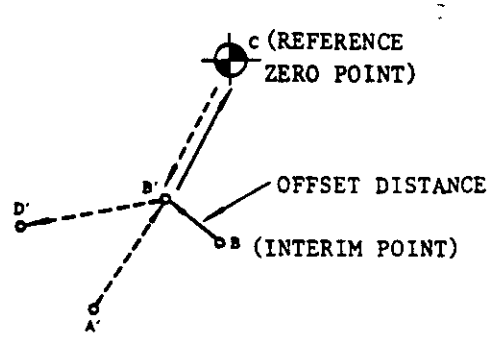
EXAMPLE (absolute input)

Coordinates of interim point is equivalent to these two blocks.

```
                                    X     Z
N20 G28 X10.  Z20.  ; ——►(10.   20.)
 .
 .
N25 G28 X30.  ; ————►(30.   20.)
N26 G29 X-40. Z-50. ;
    G00 X30.  Z20.  ;
    G00 X-40. Z-50. ;
```

NOTES:

1. Commanding G29 without the execution of G28 after turning on the control constitutes an error "059."
2. In principle, cancel tool offset before programming G28 or G29. If they are programmed while offset is effective, the interim point B will be offset, and the tool will pass point B'.



2.8.12 RETURN FROM REFERENCE POINT (G29) (CONT'D)

NOTES:

3. Commanding G29 in the TOOL RADIUS COMPENSATION mode (G40 - G44) or in CANNED CYCLES (G70 - G76, G90, G92 and G94) constitutes an error.

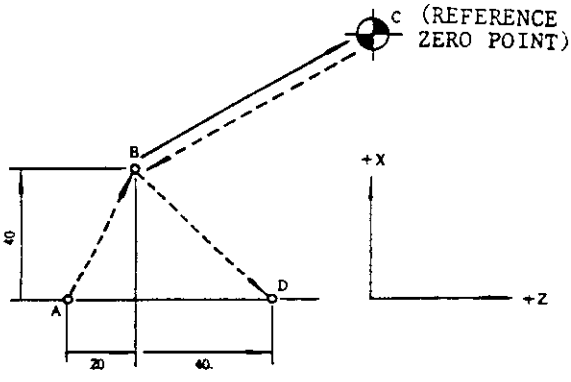
   EXAMPLE

   N50 T0300 ,

   N51 G28 U80. W20. ;

   N52 T0400 ,

   N53 G29 U-80. W40. ;



2.8.13 2ND REFERENCE POINT RETURN (G30)+

(1) G30 X(U) · Z(W)··· ,

With this command, the tool first moves to an interim positioning point (X, Z) or (U, W) in two axial directions simultaneously, and then, moves to the 2nd reference point. The tool does not move along the axis for which no coordinate position is specified.

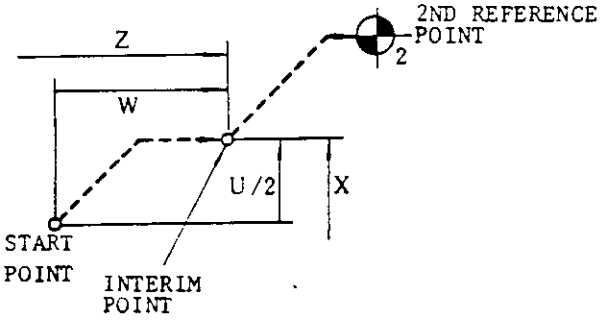
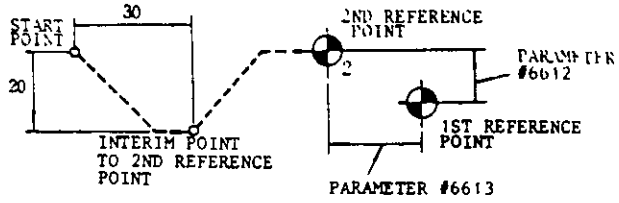


Fig. 2.18

(2) The 2nd reference point is specified in advance in terms of the distance from the 1st reference point commanded by G28, by parameters #6612 and #6613.

EXAMPLE

G30 U-40. W30 .



NOTES:

1. Before commanding G30, after the energization of the control, G28 or MANUAL RETURN TO REFERENCE POINT must be executed.
2. For the 2nd REFERENCE POINT RETURN motion, there is no area from where returning is impossible, and the tool can be return from any position
3. The same notes 3. and 4. of (5) for 2.8.11 Automatic Return to Reference Point apply to G30 command.
4. When G29 is commanded after G30, the tool moves via the interim point specified by G30 to the position specified by G29. However, the interim point is renewed only in the axis specified by G30.

2.8.14 SKIP FUNCTION (G31)+

(1) G31 X(U)··· Z(W)··· (F(E)···) ;

With this command, a special linear interpolation is executed. During the interpolation movement under this instruction, the tool interrupts the interpolation motion immediately, and proceeds to the next block, when a skip signal is received.

(2) The motion after the receipt of a skip signal varies with the instruction of the next block.

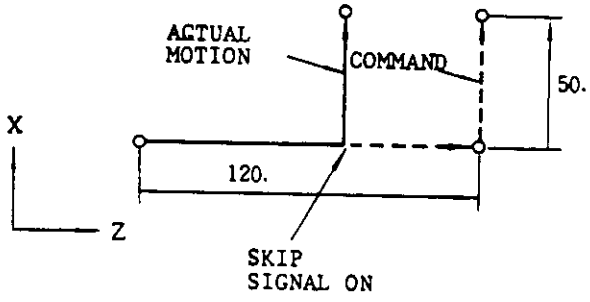
a. When the next block is programmed in incremental values:

The tool moves incrementally in accordance with the next block from the point where the interpolation is interrupted.

EXAMPLE

G31 W120. ;

G01 U100. ;



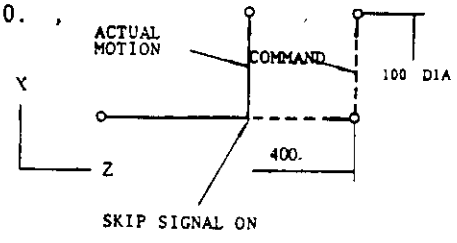
b. When the next block is programmed in absolute values only for one axis:

The tool moves to the specified coordinate position in the specified axis. It remains at the position where the skip signal is received, if axis is not specified.

EXAMPLE

G31 Z400. ;

G01 X100. ,



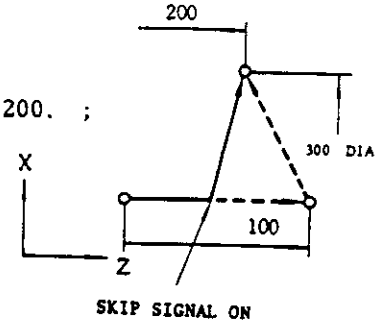
c. When the next block is programmed in absolute values along two axes:

The tool moves to the commanded position from the point at which a skip signal is received.

EXAMPLE

G31 W100. ;

G01 X300. Z200. ;



(3) G31 is a non-modal G code.

When no skip signal is received during the execution of the block containing G31, the tool stops at the end of the block, and alarm "087" is displayed.

(4) The feedrate for blocks containing G31 are set in the following two methods, selectively specified by parameter #6019D4.

a. Designation by F in the same way as with normal programs.

b. Presetting of feedrates by parameter #6323.

(5) When a skip signal is received, the coordinate values at that moment are automatically stored as parameter data.

#6568 for storing X coordinate value

#6569 for storing Z coordinate value

These data can be used as system variables in user macros.

NOTES:

1. When parameter #6004D0 is set to 1, the program is advanced to the next block automatically even when no skip signal is received during the execution of the block of G31.
2. Before programming G31, be sure to program G40 for cancelling TOOL RADIUS COMPENSATION. Failure to do this initiates alarm "024."

2.8.15 THREAD CUTTING, CONTINUOUS THREAD CUTTING (G32)

This function is for cutting straight threads, taper threads, scrolls and for continuous threadding.

(1) G32 X(U)··· Z(W)··· F(E)··· ,

With this command, the tool cuts threads up to the point (X, Z) specified in absolute coordinates or (U, W) specified in incremental coordinate values, at a lead designated by F or E code.

(2) The range of leads to be specified by F and E codes is as follows.

2.8.15 THREAD CUTTING, COUTINUOUS THREAD CUTTING (G32) (CONT'D)

Table 2.16

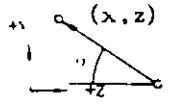
		Format	Programmable Range for Thread Cutting
Metric Output	Metric Input	F32	F0.01 - F500.00 mm
		E34	E0.0001 - E500.0000 mm
	Inch Input	F24	F0.0001 - F19.6850 in.
		E26	E0.000004 - E19.685000 in
Inch Output	Metric Input	F32	F0.01 - F1270.00 mm
		E34	E0.0003 - E1270.0000mm
	Inch Input	F24	F0.001 - F50.0000 in
		E26	E0.000010 - F50.000000 in

F code is for normal thread cutting.

E code is for precise thread cutting.

(3) The direction of lead specified by F and E codes is shown below.

Direction of Lead

Limitation of Taper Angle		Direction of Lead
	$\alpha \leq 45°$	Lead in the direction of Z-axis
	$\alpha > 45°$	Lead in the direction of X-axis

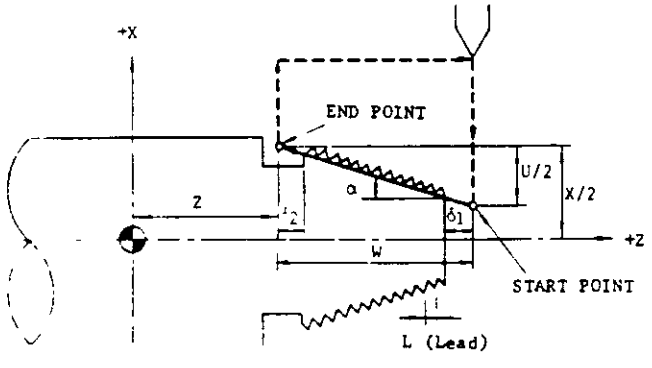


Fig. 2.19

Feedrates are limited by spindle-speed S as follows.

Metric output	F(E) × S 24,000 mm/min
Inch output	F(E) × S ≤ 2,400 in/min

The upper limit of X-axis speed component is half the above.

(4) Command format of threadcutting is shown below.

Table 2.17

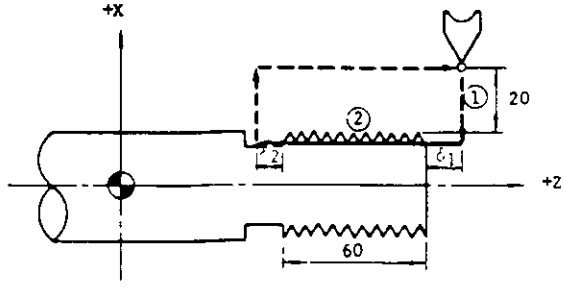
Type		Command format
Straight Thread	Normal	G32 Z(W)··· F··· ;
	Precise	G32 Z(W)··· E··· ,
Taper Thread	Normal	G32 X(U)··· Z(W)··· F··· ,
	Precise	G32 X(U)··· Z(W)··· F··· ;
Scroll Thread	Normal	G32 X(U)··· F··· ;
	Precise	G32 X(U)··· E··· ,

EXAMPLE: Straight Thread

Thread lead L = 5.0 mm
δ_1 = 5.0 mm
δ_2 = 3.0 mm
Cutting depth = 1.0 mm

① G00 U-42. ;

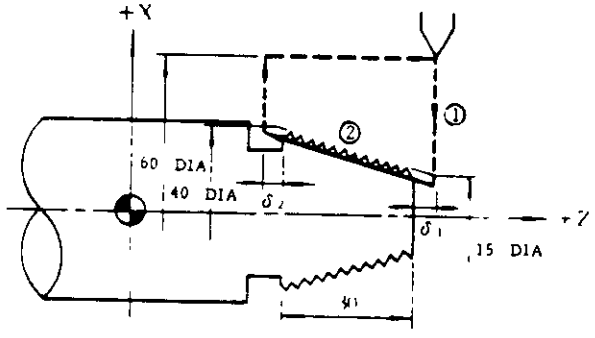
② G32 W-68. F5.0 ,
G00 U42. ;
W 68. ,
U-44. ,
G32 W-68. ;
G00 U44. ;



EXAMPLE: Taper Thread

Thread lead	L = 4.0 mm
	δ_1 = 3.0 mm
	δ_2 = 2.0 mm
Cutting depth	= 1.0 mm

```
G00 X13. ;
G32 X38. W-35. F4.0 ;
G00 X60. ;
    W35. ;
    X-11. ;
G32 X36. W-35. ;
G00 X60. ;
    .
    .
    .
```



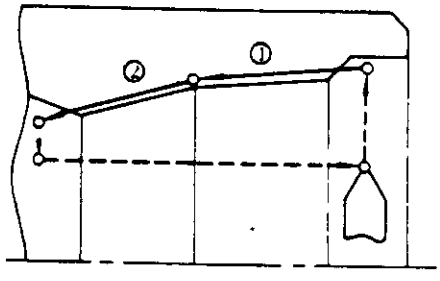
(5) Continuous thread cutting

① G32 X(U)··· Z(W)··· F(E)··· ;

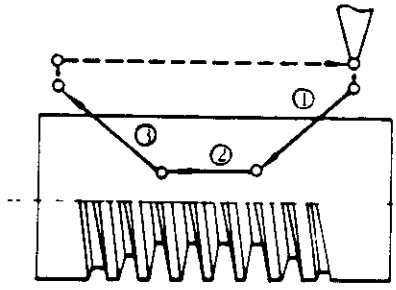
② (G32) X(U)··· Z(W)··· ;

③ (G32) X(U)··· Z(W)··· ;

This command executes thread cutting.



(a) Pipe Joint

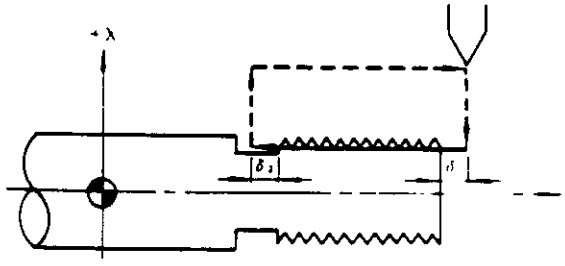


(b) Worm Screw

Since the stop time between thread cutting blocks is approximately zero, smooth, continuous thread cutting is possible If thread lead specification is changed midway, the thread becomes irregular near the boundary of blocks.

NOTES:

1. Allowances δ_1 and δ_2 are required for thread cutting because lead error occurs near the starting and end points.



2. If spindle speed is not constant during thread cutting, the leads become incorrect due to the servo lag.
3. Threading up for thread is not effective at G32. If necessary, G92 (or G76) should be commanded.
4. The following operation is disregarded during thread cutting including G32

   · Feedrate Override ··· Regarded as 100%
   · Feed Hold Operation
5. The G32 command should not be commanded in G98 mode.
6. In Dry Run mode, the tool moves at Jog feedrate.

2.8.15 THREAD CUTTING, CONTINUOUS THREAD CUTTING (G32) (CONT'D)

(6) Allowance for lead error(δ_1, δ_2)

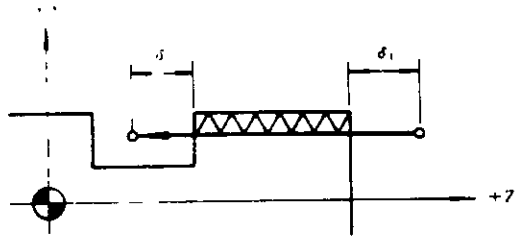


Fig. 2.20

δ_1 and δ_2 are obtained approximately from the following equation.

Table 2.18

	Equation	Meanings
δ_1	$\delta_1 > \frac{L.S}{60\ K}(\ln\frac{1}{a} - 1)$	L(mm). Lead of thread S(rpm) Spindle speed K: Constant (Normal value: 30) a (-) Accuracy of thread $= \frac{\Delta L}{L}$ Lead error ln Natural logarithm (log e)
δ_2	$\delta_2 > \frac{L\ S}{60\ K}$	

a	1/50	1/100	1/150	1/200	1/250	1/300
$(\ln\frac{1}{a} - 1)$	2 91	3.61	4.01	4 29	4.52	4.70

EXAMPLE

Lead of thread L = 3.0 mm
Spindle speed S = 500 rpm
Thread cutting a = 1/100

$$\delta_1 > \frac{L.S}{60.K}(\ln\frac{1}{a} - 1)$$

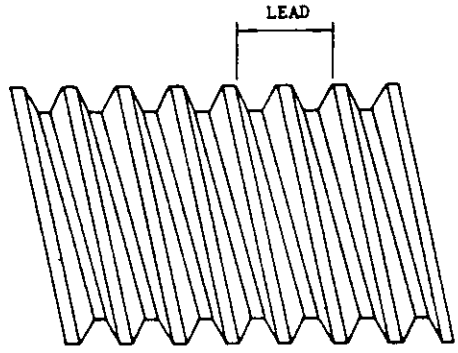
$$= \frac{3.0 \times 500}{60 \cdot K} \times 3.61 = 3.0 \text{ mm}$$

$$\delta_2 > \frac{L.S}{60\ K} = \frac{3.0 \times 500}{60.K} = 0.83 \text{ mm}$$

2.8.16 MULTI-START THREAD CUTTING (G32)†

With this function, multi-start threads containing two or more threads per lead can be machined without shifting the starting point. In thread cutting, the tool feed is started in phase with a start point pulse (1 pulse/revolution) generated by a pulse generator installed on the spindle to control the starting point of thread always at the same position around the workpiece circumference.

With the multi-start thread cutting function, after cutting a thread by controlling the starting point by the starting point pulse, another thread is cut by starting the cutting feed at an angular position of the spindle which is displaced from the starting pulse position by a preset angle.



Two-start Thread

Fig. 2.21

(1) G32 X(U)··· Z(W)·· F(E)··· B··· ,

With this command, the tool cuts a thread starting at an angular position which is displaced from the position corresponding to the starting pulse by an angle specified by B, to X(U) or Z(W) point, at a lead specified by F or E code.

(2) The data specified by address B in the multi-start thread cutting function is as follows.

Least input increment: 0.001 deg.

Programmable range: $0 \leq B \leq 360.000$

When decimal point input is used, B1.=1deg. The B code is non-modal, and is effective only in the programmed block.

(3) Number of starts and B code

In principle, the thread starting points on the workpiece circumference should divide the circumference into equal portions.

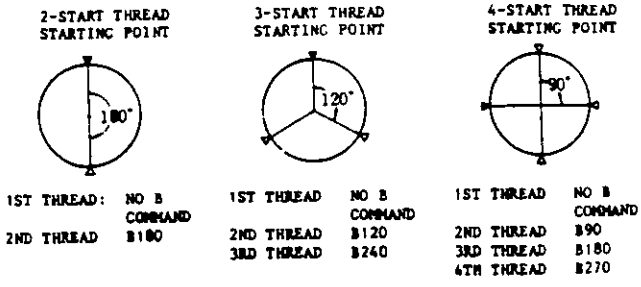


Fig. 2.22

NOTES:

1. Since the angular position detection pulses (4096 pulses/rev.) generated from the spindle pulse generator is used to define the angular position of the spindle with respect to the starting point as controlled by the B command, the least detectable increment is 360°/4096 pulses = 0.0879°/pulse. From the position commanded by B codes, an error up to ±1 pulse may occur.
2. The angular position from the starting pulse can be specified in both forward and reverse directions by B0 - B360. commands.
3. When cutting many multi-start threads in succession, if the angular position is controlled from the starting pulse by a B command in the first block, no B command is required from the 2nd block on.
4. When B command is made to specify angles outside the permissible range (0 - 360.000), alarm "065" is displayed.

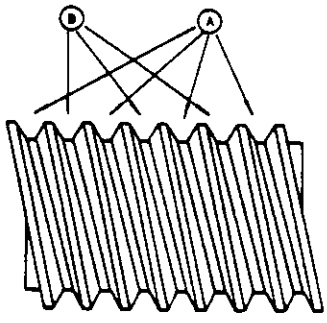
EXAMPLE: Two-start Thread

```
G00  U···  ;
G32  W···  F···  ;
G00  U···  ;
W···  ;
U···  ;
G32  W···  ;
```

} Threading of part Ⓐ

```
G00  U···  ;
G32  W···  B180.  ;
G00  U···  ,
W···  ;
U···  ;
G32  W···  B180.  ;
```

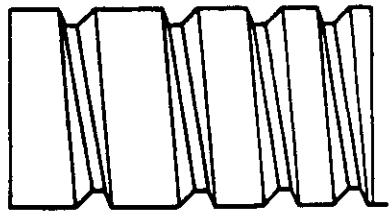
} Ⓑ



2.8.17 VARIABLE LEAD THREAD CUTTING

(1) G34 X(U)··· Z(W)··· K··· F(E)··· ;

With this command, variable lead screws are controlled with the increase or decrease of lead per revolution specified by address K.



VARIABLE LEAD SCREW

Fig. 2.23 Variable Lead Thread Cutting

(2) The range of K programmable for variable lead screws is as follows.

Least input increment:

0.0001 mm/rev. (Metric Input)	0.000001 in/rev. (Inch Input)

Programmable range:

a. The highest feedrate is within the maximum programmable feedrate range (500 mm/rev (metric) or 50 in/rev. (inch))
b. The total displacement resulting from changes in lead is within the following.

   4194.303 mm (metric output)

   419.4303 in. (inch output)

c. Feedrate change corresponding to lead variation must not exceed 5,400 mm/min (metric output) or 540 in./mm (inch output).
d. Lead value should not be minus value.

NOTES:

1. When variable lead threads are cut by continuous block programs, command pulses are interrupted at block junctions.
2. If K commands exceed the permissible range, alarm "060" will be displayed.
3. When G34 command is executed in the Dry Run mode, the tool moves only at the speed specified by the manual continuous feedrate command, if parameter SCRDRN (#6019 BIT5) is set to 1.
4. When parameter "IOIN" (#6006D5) is set to 1, least increment for K commands is 0.001 mm/rev or 0.0001 in./rev.
5. Commanding address B in G34 block causes alarm "060."

2.8.17 VARIABLE LEAD THREAD CUTTING (CONT'D)

(3) Confirmation calculation for K command of variable lead thread cutting.

a. K command is restricted in the following conditions.

(i) Feedrate at end point must not exceed programmable range.
500 mm/rev (metric output) or 50 in rev (inch output)

(ii) Feedrate at end point must not be minus value.

(iii) Accumulated value of movement due to lead variation must not exceed 4194.303 mm (metric output) or 419.4303 in (inch output).

(iv) Feedrate change corresponding to lead variation must not exceed 5,400 mm/min (metric output) or 540 in/mm (inch output

b. The control checks the restriction described above using the following equation.

F: Fixed lead command (mm/rev or in./rev)

K: Variable lead command (mm/rev or in/rev)

W: Distance between start and end points on Z-axis (mm or inch).
For facing screw, distance is specified as U on X-axis.

S. Spindle speed (rev/min)

N. Spindle speed for movement between start and end points (rev)

$$N = \frac{-(F + \frac{K}{2}) + \sqrt{(F + \frac{K}{2})2 + 2 \cdot K \cdot W}}{K}$$

(i) Equation for limit in a. (i)

$$F + \frac{K}{2} + KN \leq 500.000 \text{ mm/rev or } 50.0000 \text{ in/rev}$$

(ii) Equation for limit in a. (ii)

$$(F + \frac{K}{2})^2 + 2KW > 0$$

(iii) Equation for limit in a. (iii)

$$\frac{1}{2}KN^2 \leq 4194.303 \text{ mm or } 419,4303 \text{ in}$$

(iv) Equation for limit in a. (iv)

$$\frac{S}{60} \cdot K \cdot N \leq 5,400 \text{ mm/min or } 540 \text{ in/min}$$

4194.303 mm (metric output) or 419.4303 in (inch output)

(v) Commanding address B in G34 block causes alarm "060."

2.8.18 TOOL SET ERROR COMPENSATION (G35)

This function is for automatically rewriting the tool offset value to suit to the new tool, when the tool is replaced. For this purpose, a touch switch (contact detector) is installed externally, and when the tool comes into contact with the switch, a signal is input to the control to calculate the new tool offset value.

(1) G98 ——→ Feed/Minute

G35 X(U)··· (F···) ; ——→ X-Axis Compensation

or

G35 Z(W)··· (F···) ; ——→ Z-Axis Compensation

With the above instructions, the tool moves in the directions specified by X(U) or Z(W), a signal for tool contact with the touch switch is input to the control, the tool offset value for the tool is calculated. This value replaces the stored value. Then, the tool returns to the start point of the block, in which the position correction is incorporated, by moving in rapid traverse.

(2) The new tool offset value is as follows.

New offset =

[Absolute coordinate at which new tool touched switch (X_T or Z_T)] − [Absolute coordinat at which reference tool touched switcl (X_P or Z_P)]

Example of G35 operation for X-axis is shown below.

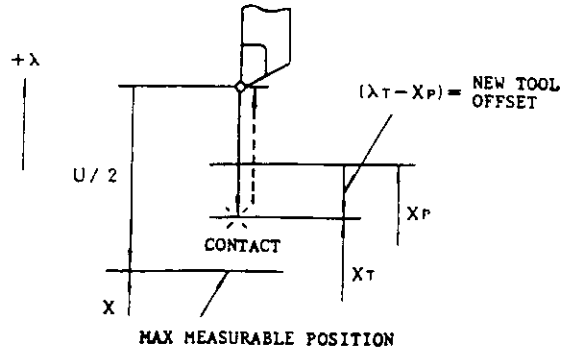


Fig. 2.24

The coordinate value (Xp or Zp) at which the reference tool makes contact with the touch switch must be set by parameters #6624 (Xp) and #6625 (Zp) in advance.

NOTE: In principle, use the same coordinate system for determining Xp or Zp, and for programming G35. The coordinate system may either be G50 system or the work coordinate system (G50T), as long as the same system is used for the respective tools. When different coordinate systems are used, the difference between the two will be added to the tool offset value described above. This may result in abnormally large or small offset values, but if the machining is commanded on the coordinate system in which G35 is programmed, the machining will be performed without trouble.

(3) For the values of X(U) and Z(W), the move direction and the limit position of this measurement should be specified. Then, if the touch switch is not tripped by the tool moving beyond this position, error "087" is displayed.

(4) The tool move speed towards the touch switch is specified in the following two methods.

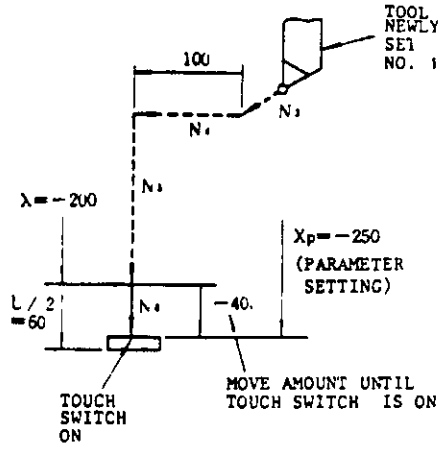
a. Programming by the F code as with ordinary programs.
(when parameter #6229 is 0)

b. By setting feedrates by parameter #6229 in advance.

When the speed is specified by F codes and by parameter #6229, the parameter has the priority.
In both a and b , program G98 (mm/min designation) in the preceding block or in the block of G35. Otherwise, error "080" will be displayed.

EXAMPLE: Program for the respective tools as follows.

```
(N1 G28 ;)
 N2 G50 X400. Z300 ;
 N3 G00 T0101 ;
 N4 W-100. ;
 N5 X-200 ;
 N6 G35 U-120. G98 F20 ;
 N7 G00 T0100 ;
 N8 G27 X400 Z300 ;
```



New tool compensation value =
(−240.) − (250.) = +10. mm

The above tool offset value is stored in the tool offset No. 01 as the X-axis tool offset value.

NOTES:

1. G35 is non-modal G code. Do not write M, S and T codes in the same block with G35.
2. For the move speed of the tool under G35 command, the Dry Run function is ineffective.
3. If T code is not programmed before G35, this constitutes an error "080."
4. When G35 is commanded in the tool radius compensation mode, an alarm code "024" is displayed.
5. The following state constitutes alarm "082," "083."

Calculated new tool offset value	<	Value of parameters #6624 or 6625

This serves as a protection against an inadvertent omission of parameter writing.

6. Do not execute the program with MACHINE LOCK ON before executing G35 command If this is done, the shift during a MACHINE LOCK ON execution will be added to the new offset value and correct compensation becomes impossible.
7. If G35 command is executed in the MACHINE LOCK ON state, no offset value is calculated; rather, the tool moves to the commanded point and then, returns to the start point.

2.8.19 STORED STROKE LIMIT (G36 - G39)

This function is for preventing the entry of the tool into the prohibited area, in both automatic operation mode and manual operation mode, to secure safer operation. Three types of prohibited areas can be set up as shown below.

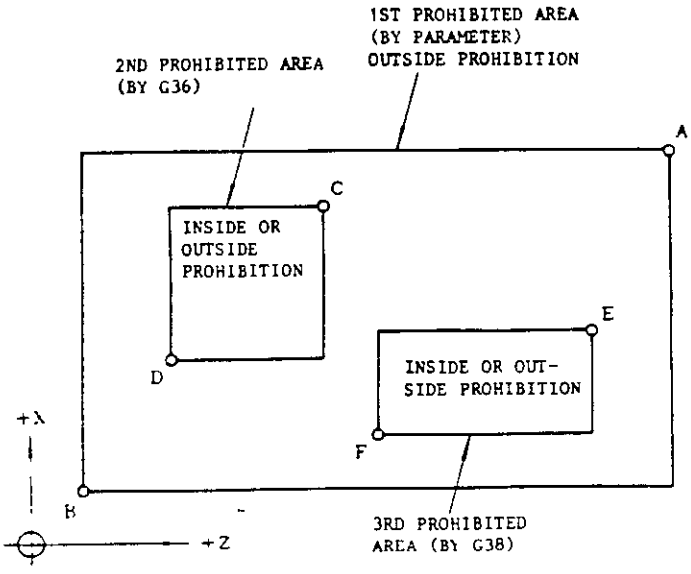


Fig. 2.25

(1) Setting 1st prohibited area

a. Set + side boundary A and - side boundary B by parameter.

b. The area outside the range between points A B is prohibited.

c. This area can not be specified by part programs.

(2) Setting 2nd prohibited area

a. Set + side boundary C and - side boundary D by the setting function.

b. Selectively designate the inside or the outside of the area between points C and E by parameter #6007D0.

#6007D0	Meaning
0	Inside prohibition
1	Outside prohibition

c. In addition to the method by the setting function, the following instruction can be used to set a prohibited area, and to turn on the area check function also.

G36 U ·· W··· I··· K··· ;

Point C coordinate — Point D coordinate

The area check function is turned off by a single code block G37 ;.

(3) Setting 3rd prohibited area

a. Set up + side boundary point E and - side boundary point F by the setting function.

b. Selectively designate the inside or the outside of the area between points E and F by parameter #6007D1.

#6007D1	Meaning
0	Inside prohibition
1	Outside prohibition

c. In addition to the method by the setting function, the following instruction can be used to set a prohibited area, and to turn on the area check function also.

G38 U··· W··· I··· K··· ,

Point E coordinate — Point F coordinate

The area check function is turned off by a single code block G39 ,.

Table 2.19 Patameters and Settings for Setting Prohibited Area

		X axis	Z axis	Division
1st prohibited area	Point A	#6600	#6601	Parameter
	Point B	#6606	#6607	
2nd prohibited area	Point C	#6500	#6501	Setting
	Point D	#6502	#6503	
3rd prohibited area	Point E	#6504	#6505	
	Point F	#6506	#6507	

(4) Coordinate system for area setting

The above Points A through G are all set in absolute values on the machine coordinate system (MACHINE). That is, the position is written as the distance from the (1st) reference point (1 = least output (move) increment). Therefore, this function will not become effective unless the manual or automatic RETURN TO REFERENCE ZERO is executed once after the energization of the control.

(5) Effective-ineffective selection of prohibited area

With the following setting, the 2nd and the 3rd prohibited areas are selectively made effective and ineffective.

Setting		Meaning
#6001D1	0	2nd prohibited area check OFF
	1	2nd prohibited area check ON
#6001D2	0	3rd prohibited area check OFF
	1	3rd prohibited area check ON

When G36 through G39 are commanded, these setting data are automatically rewritten. Therefore, the ON or OFF state ultimately specified by G code commands or setting function becomes effective. The 1st prohibited area is always in the CHECK ON mode.

(6) Starting area check

When the tool is returned to the reference point once manually or automatically after the energization of the control, the area check function is started immediately. Therefore, if the reference point is in the prohibited area, immediately, STROKE LIMIT ERROR will be caused. In this case, turn off the area check function, and change the data.

(7) Stored stroke limit error

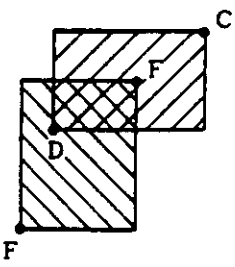
When the tool enters the prohibited area, it stops just inside the boundary line, and the control enters STORED STROKE LIMIT ERROR state. In this case, the tool can only be moved manually in the return direction.

(8) Displaying remaining distance

With this function, the distance between the current tool position and the boundary of the prohibited area in the X and Z directions are displayed on the CRT. Refer to 4.3.4.5 Stored Stroke Limit Remaining Distance Display.

NOTES:

1. The points on the boundary line in both axes are included in the prohibited area.
2. Two prohibited areas can be set with partial overlapping.
3. In the MACHINE LOCK ON mode, AREA CHECK function is not effective.



EXAMPLE

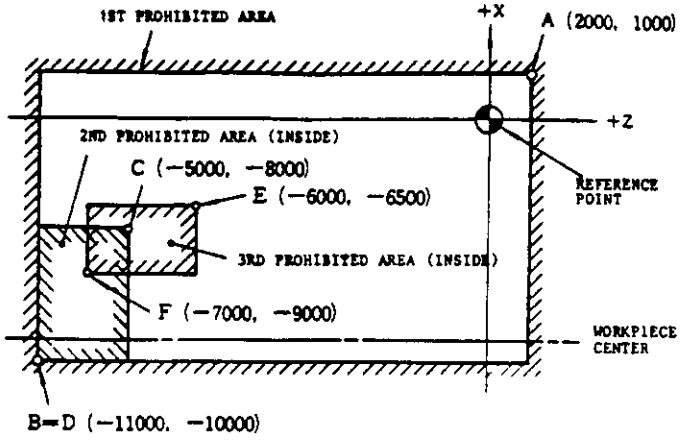


Table 2.20

	Parameter/ Setting	Contents	
Inside/Outside	#6007D0	0	
	#6007D1	0	
Second Area	#6500	-5000	C
	#6501	-8000	
	#6502	-11000	D
	#6503	-10000	
Third Area	#6504	-6000	E
	#6506	-6500	
	#6506	-7000	F
	#6507	-9000	
First Area	#6600	2000	A
	#6601	1000	
	#6606	-11000	B
	#6607	-10000	

2.8.20 TOOL NOSE RADIUS COMPENSATION (G40 THROUGH G44)†

Because of a nose radius of lathe tools, there is a deviation between the desired curve and the actual curve produced. Therefore, tool offsets are not enough for taper and circular cuttings. The tool nose radius compensation option resolves the problem of nose radius. See Fig. 2.8.20.1.

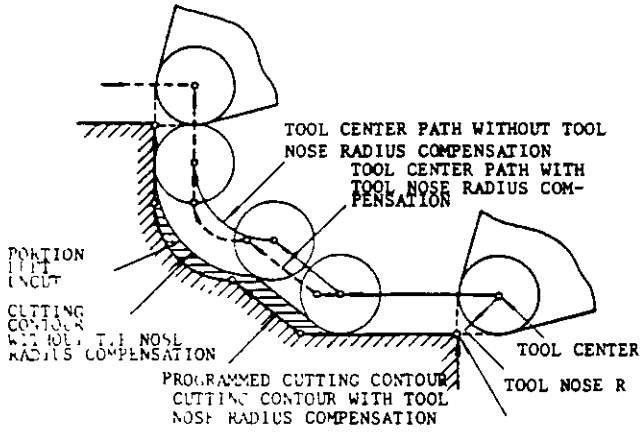


Fig. 2.26

(1) Tool nose radius values

a. Radius value storage

Tool nose radius value must be written in the storage before the tool nose radius compensation is commanded. Number of pairs that can be written in the storage depends upon the machine

T 3-digit 9 sets
T 4-digit 16 or 50 sets

Refer to 2.6.3 Tool Offset Memory†.

b. Range of tool nose radius values

Radius value can be set within the following range.

Metric	Inch
0 ±99.999	0 ±9.9999

c. Setting of tool nose radius values

Radius value of tool nose must be set without signs.

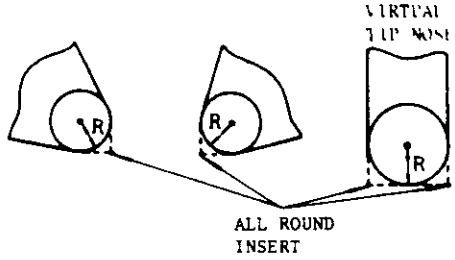


Fig. 2.27

For the writing of radius values for tool nose radius compensation, refer to Fig. 4.3.5 Displaying and Writing Tool Offset Data. The address character is R.

(2) T code designation

a. The T code for tool nose radius compensation must be programmed with sign (+ or -).

T ± □□ □□ (In the case of T 4-digit)

- Offset number for tool positio offset and tool nose radius compensation
- Tool number
- Direction of tool nose radius compensation

"+" ··· Right side viewed in the direction of tool travel

"-" ··· Left side viewed in the direction of tool travel

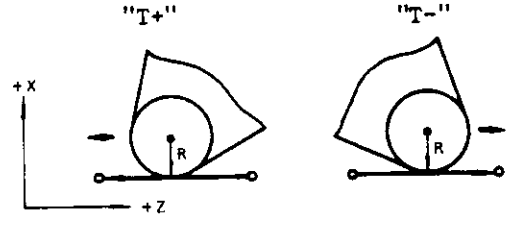


Fig. 2.28

b. When a tool is used for turning and for facing, as the direction of tool motion changes, the correct direction of compensation should be programmed with sign of T code.

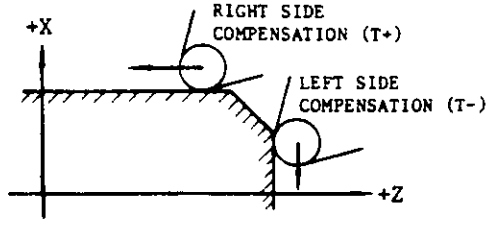


Fig. 2.29

The direction of compensation is changed from + to - or - to + during program execution. G40 or T□□ 00 command should not be necessarily programmed to cancel the tool nose radius compensation.

(3) G code designation (G40 to G44)

a. G code of tool nose radius compensation (G41 to G44)

One of G41, G42, G43 and G44 and T code should be programmed before the execution of tool nose radius compensation. These four G codes specify the relationship between the virtual tool nose and the tool center.

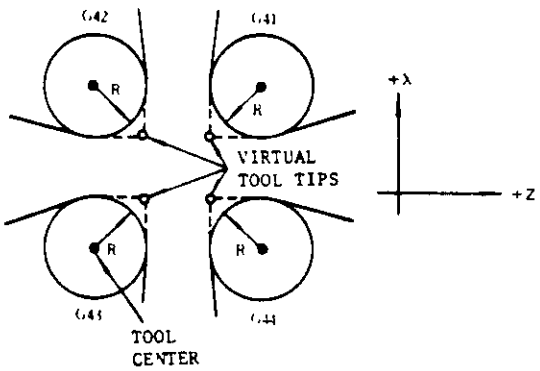


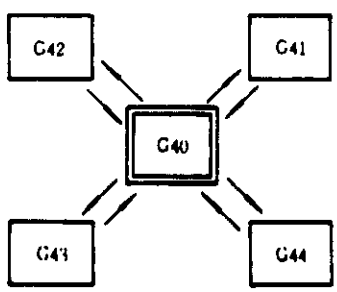
Fig. 2.30

Regardless of the mode of tool nose radius compensation, the current position of the virtual tool nose is displayed by depressing the POS pushbutton.

b. Issue G40 to cancel the tool nose radius compensation.

c. Cautions in programming G code

(i) Since G40 to G44 are modal G codes of 07 group, they are retained until the other G code is commanded. Before switching one of G41, G42, G43 and G44 to another, G40 must be intermediated to cancel the compensation.



(ii) When the power supply is turned on, G40 is in effect.

(iii) When the RESET button is depressed, G code of 06 group are cancelled and G40 becomes effective.

(4) Tool motion on the tool nose radius compensation

a. Fig. 2.31 shows the outline of the tool motion.

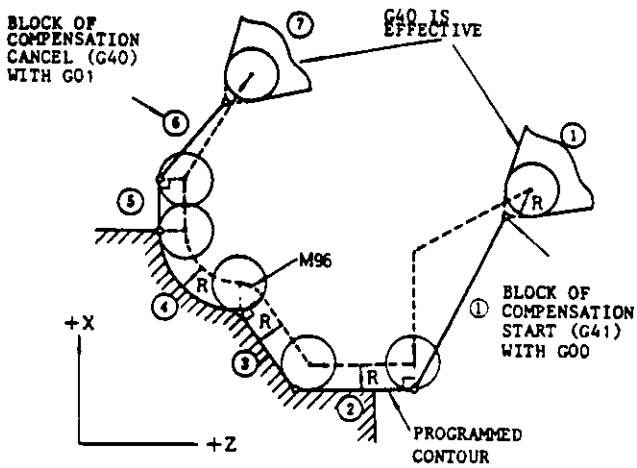


Fig. 2.31

(i) When the compensation is cancelled, the programmed contour meets with the path of virtual tool tip (⓪ and ⑦).

(ii) In compensation mode, the tool center path is deviated by radius from the programmed contour. Therefore, the path of virtual tool nose does not meet with programmed contour. But the current position displayed by depressing POS key is the position of virtual tool tip (② to ⑤).

(iii) The connection between two blocks in compensation mode is provided by the intersection of tool center paths (M97) and by the circular arc (M96). In the above diagram, blocks 3 and 4 are connected by a circular arc.

(iv) Block ① for compensation start and block ⑥ for compensation cancel perform the connection of compensation mode and compensation cancel mode. Program should be made carefully for these blocks.

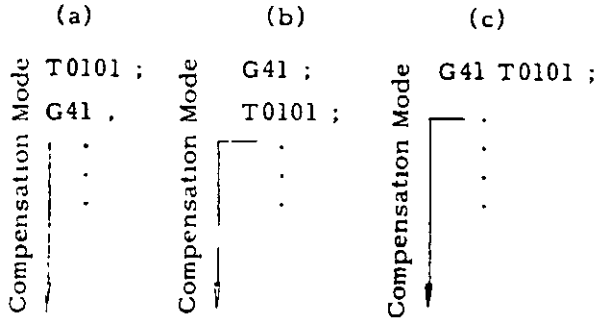
b. Relationship between tool nose radius compensation and tool position offset

Tool nose radius compensation apply to the programmed contour which has been offset by the tool position offset function.

2.8.20 TOOL NOSE RADIUS COMPENSATION (G40 THROUGH G44)

(5) How to enter compensation mode

a. Compensation mode is set when both the tool offset number by T code and G41 (or G42 through G44) are instructed. When this mode is set, tool nose radius compensation is started. More precisely, compensation mode is entered when the AND condition between T code and G code is established. Hence, the order in which these codes are specified does not affect the operation.



b. At the start of compensation, the tool center is offset onto the normal of the origin of the block G41 to G44 which entered the compensation mode first or of the block immediately after T code. The offset is made to the right of tool advancing direction when T+ is specified and to the left when T- is specified.

Sample Program (A):

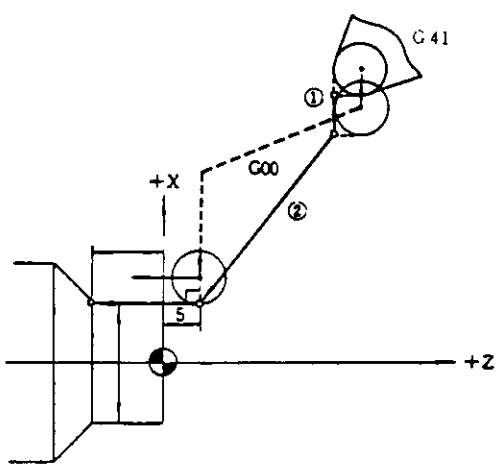
(a)

① G00 T-0101 ,

② G00 G41 X30. Z5. ; ← Movement at Start of Compensation (for G00)

③ G01 Z-18. F0.25 ;

Compensation Mode



(b)

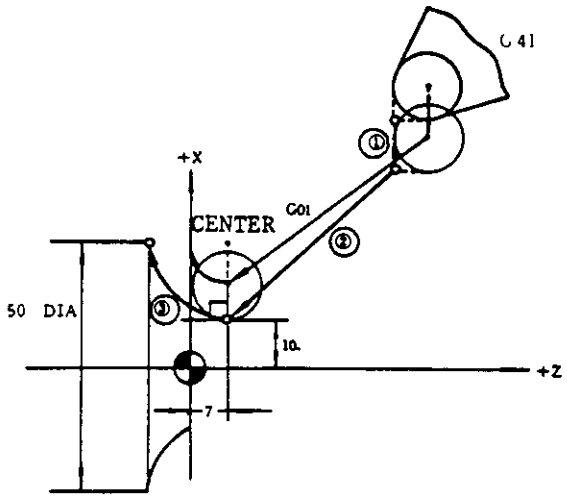
① G00 T+0202 ;

② G01 G41 X20. Z7. F600 ;

③ G02 U30. W-15. I15 F0.2 ;

Movement at Start of Compensation (for G01)

Compensation Mode



c. If the block of G41 (or G42 through G44) satisfying the compensation mode condition does not have the move command, the compensation starts and the tool center is moved on the normal.

Since G41 (or G42 through G44) involves such a movement, it is necessary to specify G00 or G01 in th last or current block for the G code of 01 group. Specification of a G code other than G00, G01, and G11 will result in alarm "026."

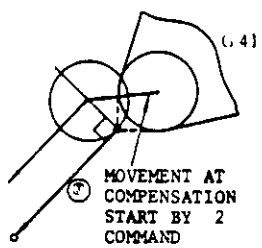
EXAMPLE (B): G41 (or G42 through G44) has no move command.

(c)

① G00 T+0303 ;

② G01 G41 F··· ;

③ G01 X··· Z··· F··· ;

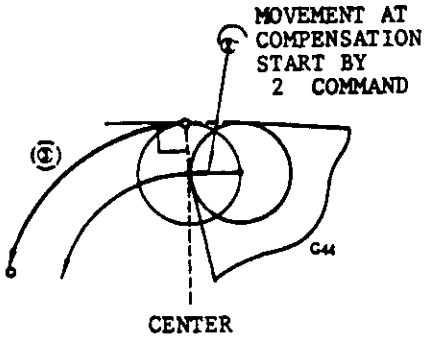


(d)

① G00 T-0404 ;

② G01 G44 F··· ;

③ G03 X··· Z··· I··· F··· ;



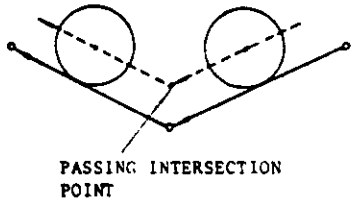
Note that the tool center is offset onto the normal to the start point of the block immediately after G41 (or G42 through G44) or T code, for each of above examples (a) through (b). If G41 (or G42 through G44) block or the block following T code has no move command, one block ahead is read and the compensation start operation is performed on that block. With no move command specified, up to two blocks may be programmed consecutively. However, if three or more blocks have no move command, an error is caused.

(6) Movement in compensation mode

When the tool nose radius compensation mode is entered by G41 (or G42 through G44) command, the tool center keeps moving along the path which has been offset by the tool nose radius by the program command, until the mode is cancelled by G40 or T□□00 command. The path is automatically calculated by the control. So, only the cut contour may be specified in the part program. However, the following should be considered for the inter-block movements and special contours:

a. Inter-block movement

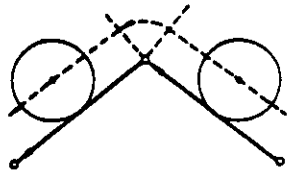
(i) For an inside corner (tangent-line angle is less than 180°), the itersection point is computed and is passed. (Intersection point computing formula.)



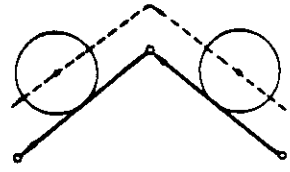
(ii) For an outside corner (tangent-line angle is more than 180°), the movement is controlled by the following M-code commands:

M96 ··· Tool radius compensation circular path ON

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



M96: Circular Path Mode



M97: Intersection Computing Mode

Movement of circular path is included in the previous block.
Normally, M96 is used for this operation However, when there is a possibility of an "overcut" in cutting special shapes with the M96, M97 should be used.

b. Movement in G00 mode

The instruction G00 positions tools independently along each axis toward the final offset position. Care should be taken on the cutter path so that tool does not contact the work.

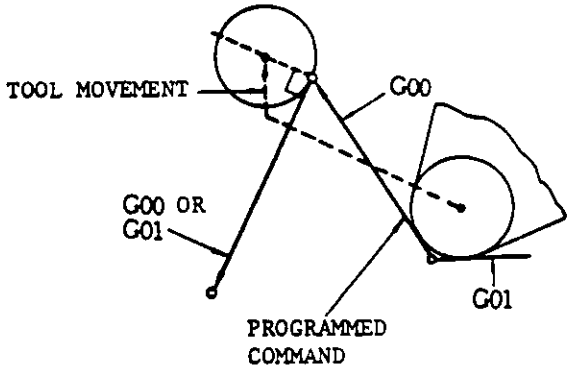


Fig. 2.32

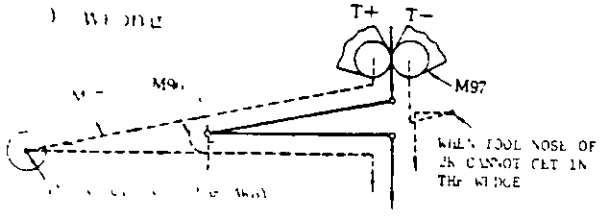
7.8.20 TOOL NOSE RADIUS COMPENSATION (G40 THROUGH G44) (CONT'D)

(6) Movement in compensation mode (Cont'd)

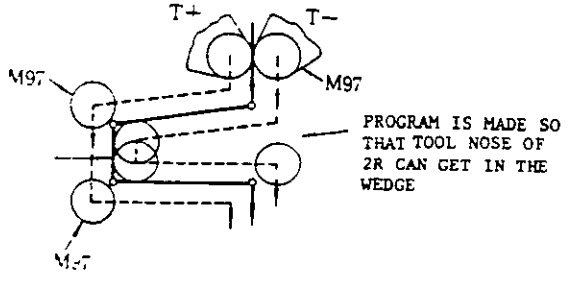
c. Programming consideration in compensation mode

(a) Be careful not to program a wedge-shaped cutting contour.

1) Wrong

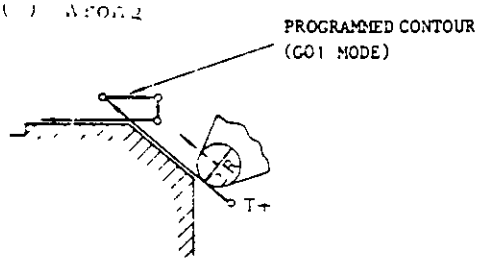


2) Correct

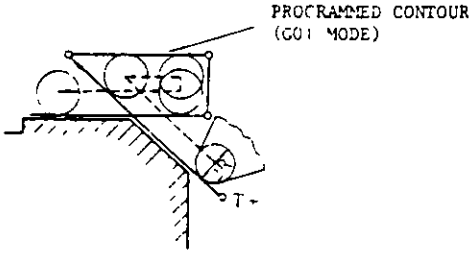


(b) Program the tool movement so that the tool nose of 2R diameter can be in the corner.

1) Wrong



2) Correct



d. Command involving no movement in compensation mode

The control normally reads two blocks ahead during tool radius compensation mode and calculates the tool path. If either of these blocks gives no coordinate instructions such as G04 (dwell), the control reads a block further ahead and makes calculations. When coordinate instructions are missing in three or more blocks, tool radius compensation becomes impossible and accurate tool path cannot be obtained. Therefore, in a program where G41 to G44 are used ensure that, after them, two or more blocks without movement command in the compensation plane will not follow.

```
G01 G41 F··· ,
  .
  .
  .
G04 U··· ,  ┐
M· · ,      ┘ Compensation is normally
              made by the two or less
              blocks without move command
  .
  .
  .
M40 ;
```

If no movement instruction is programmed in three consecutive blocks, tool center is offset on the normal line at the end point of the block immediate before them

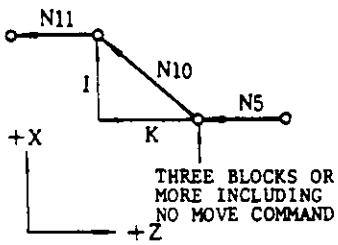
e. Use of dummy blocks

If it is impossible to specify a move command in three or more consecutive blocks and the offset on the normal line is not satisfactory, a dummy block may be inserted.
The dummy block does not cause an actual movement. This block is specifieid for the purpose of providing the data necessary for the tool nose radius computation. For the address of this dummy command, I and K are used.

I: X-axis dummy command (incremental).
K: Z-axis dummy command (incremental).

EXAMPLE:

```
N1 G01 G41 X··· Z··· F··· ;
N2 X··· Z··· ;
    .
    .
    .
N5 Z··· ;
N6 I··· K··· ;        ——— Dummy block
N7 Mxx                ]
N8 G04 U··· ;         ] Three blocks or more
N9 MΔΔ                ]
N10 X··· Z··· ;
N11 Z··· ;
```



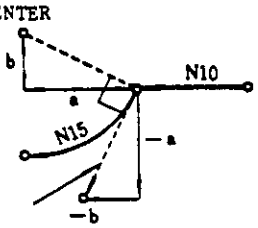
Namely, specify incremental commands I and K, which are equivalent to N10 block command, in N6 for the dummy block.

Remarks: If the purpose of the dummy block is a circular interpolation, generate the linear dummy block that specifies the direction of the tangent line at the start point of the circular command.

EXAMPLE.

```
N10 G01 Z··· F··· ;
N11 G01 I(-a) K(-b) ;
N12 Mxx ;
N13 MOO ;          Dummy block
N14 MΔΔ ;
N15 G02 X··· Z··· I··· K··· ;
N16···
```

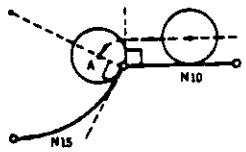
CENTER



N11 DUMMY BLOCK
(LINEAR LINE)

I and K should be signed according to the type of the circular arc.

(M96: CIRCULAR PATH MODE)



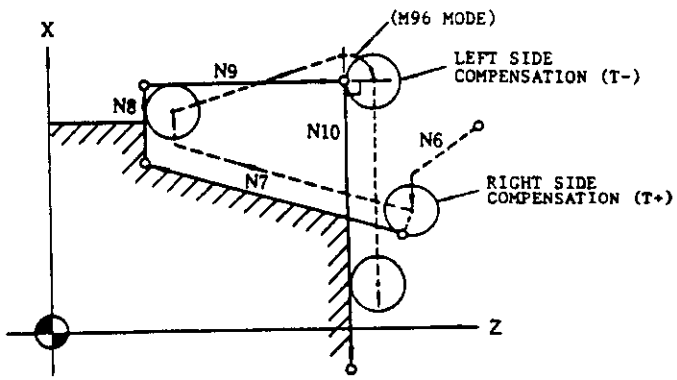
By dummy block N11, the linear command block of N10 stops at point A for the following circular movement.

f. Switching between T+ and T- in compensation mode

This compensation provides the switching between T+ and T- without cancelling the compensation by G40 or T□□00 command.

EXAMPLE:

```
N5 G00 T+0101 ;      ——— Designates right-hand side compensation facing the proceeding direction
N6 G41 X··· Z··· ;
N7 G01 X··· Z··· F··· ;
N8 X··· ;
N9 T-0101 Z··· F600 ;  ——— Changes left-hand side compensation
N10 X··· F··· ;
```



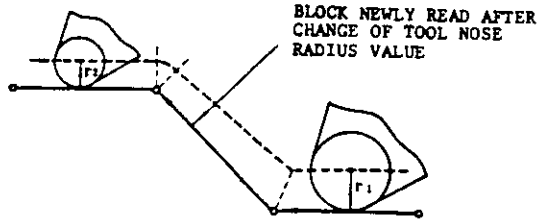
g. Modification of tool offset volume in compensation mode

It is invalid to newly specify a tool offset number by T code in compensation mode. The originally specified tool offset number remains valid until the compensation mode is cancelled by G40 or T□□00 command.

2.8.20 TOOL NOSE RADIUS COMPENSATION (G40 THROUGH G44) (CONT'D)

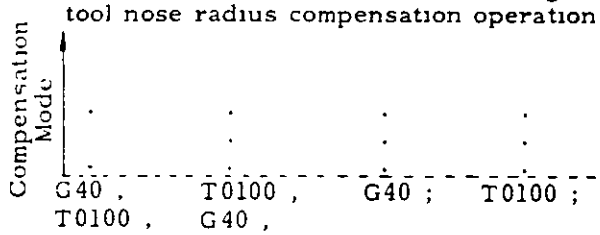
(6) Movement in compensation mode (Cont'd)

However, the tool nose radius value may be changed by varying the offset memory contents corresponding to the originally specified tool offset number, by means of MDI operation. After this modification, the new tool offset number is made valid beginning with the block newly stored in the prefetch buffer.



(7) How to cancel compensation mode

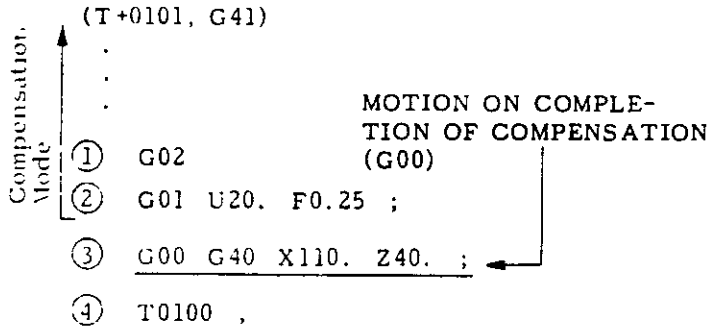
a. When G40 or T□□00 is specified, compensation mode is cancelled, terminating the tool nose radius compensation operation.

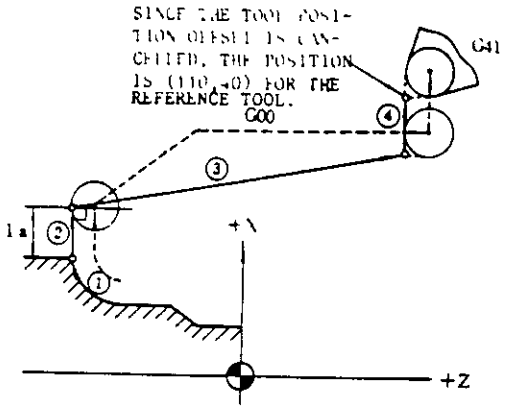


b. Upon termination of compensation, the tool center is offset onto the normal line to the end point of the final block in compensation mode, or the block immediately before that for which G40 or T□□00 has been specified. (Consequently, if a retracting which results in acute-angle contour is specified in G40 or T□□00 block, no uncut portion is produced.)
Then, the tool moves so that the virtual tool nose matches the end point of the move command specified in G40 or T□□00 block.

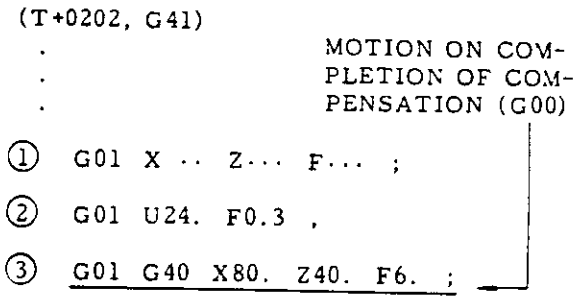
EXAMPLE A

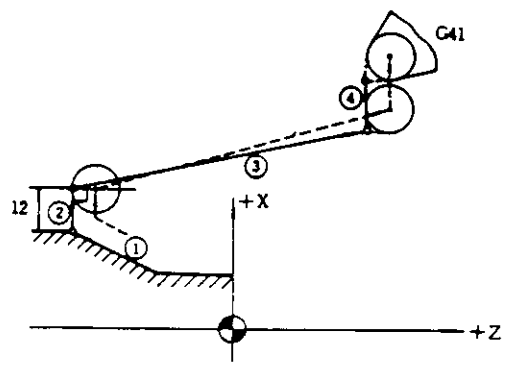
(a)





(b)





c. If the block of G40 (Nose radius compensation cancel) does not have the move command, the virtual tool nose moves to the specified end point.

EXAMPLE B: Move command is not included in G40 block for cancelling compensation.

(c)

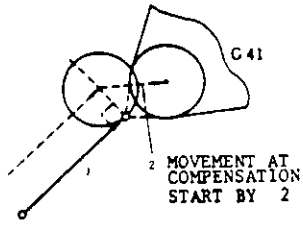
(T-0303, G41)

.
.
.

① G01 X··· Z··· F··· ;

② G01 G40 F··· ;

G00 T-0300 ;



(d)

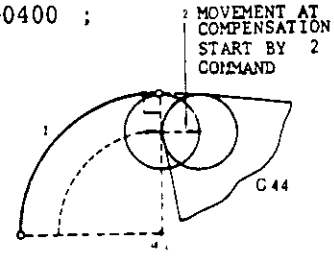
(T+0404, G44)

.
.
.

① G02 X··· Z··· K··· F··· ;

② G01 G40 F··· ;

G00 T+0400 ;



Note that, for each of above examples (a) through (b), the tool center is temporarily offset onto the normal line to the end point of the block immediately before G04 or T□□00 command.

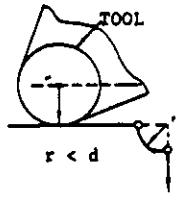
d. When tool nose radius compensation is cancelled by the use of T□□00 command, the tool position offset cancel operation is performed concurrently with the operation at tool nose radius compensation termination. The cancel operation provides the movement in which the virtual tool nose matches the last specified position for which tool position offset has been cancelled. If coexistence of these operations is not desired, cancel either of them by the use of G40 command.

NOTES:

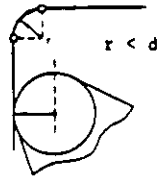
a. Programmed shapes that produce input errors

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

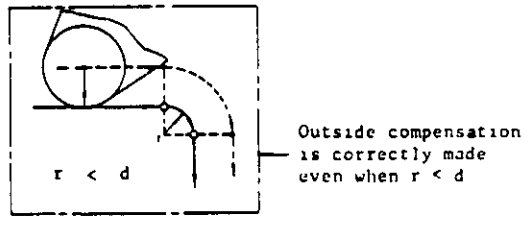
Programmed arc radius R ≦ tool radius d



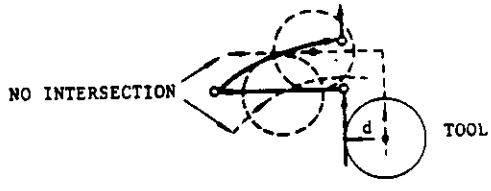
(a) Inside compensation error



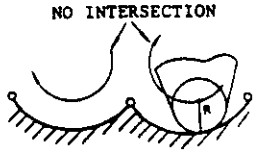
(b) Inside compensation error



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



(a)

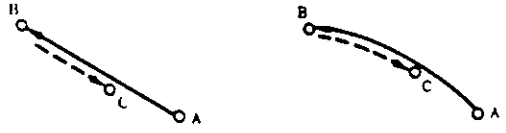


(b)

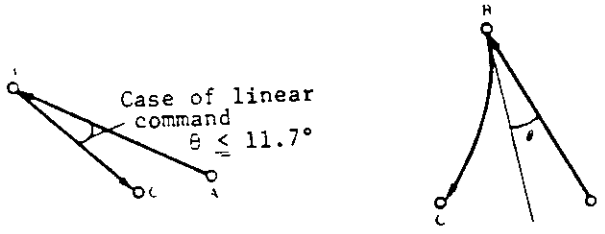
2.8.20 TOOL NOSE RADIUS COMPENSATION (G40 THROUGH G44) (CONT'D)

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

(a) Reversing command



(b) Command close to reversing



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

b. G codes usable in compensation mode

As a rule G codes other than shown below should not beused in the compensation mode.

Usable G codes	Remarks
G00, G01, G04, G06, G11 G96, G97 ··· Constant surface speed control G98, G99 · Feed function designation (G90, G91) ··· Absolute/incremental designation	
G02, G03 G12, G22, G23 ··· Command including circular arc G70, G71, G72, G73 ·· Multiple repetitive cycle G111, G112 ··· Multiple cornering (Beveling, rounding)	Inhibited in the block of compensation cancel or start

c. The subprogram (M98 or M99) may be specified in compensation mode.

d. When the tool offset number is T code command of "00," T□□00 command has the following two meanings:

(i) Tool position offset is cancelled.

(ii) Tool nose radius compensation is cancelled.

The following program can be specified

```
N2  G41  ,
N3  G00  T+0101  ;
      .                ┐
      .                │  TOOL RADIUS
      .                │  COMPENSATION
      .                │  MODE WITH TOOL
      .                │  NO. "01"
      .                │
      .                │
      .                ┘
N21 G00  T0100  ;
      .                ┐
      .                │  TOOL RADIUS
      .                │  COMPENSATION    CAN
      .                │                  CEL
      .                │  TOOL POSITION
      .                │  OFFSET
      .                │
      .                ┘
N25 G00  T-0202  ;
      .                ┐
      .                │  TOOL RADIUS
      .                │  COMPENSATION
      .                │  MODE WITH
      .                │  TOOL NO. "02"
      .                │
      .                ┘
N40 G00  T0200  ;
N41 G40  ;
```

e. Inhibition of MDI mode

(i) Operation in the MDI mode cannot be performed in the compensation mode. When RESET button is depressed, G00 (compensation cancel) becomes effective and the operation in the MDI mode becomes possible.

(ii) G40 through G44 cannot be written by the operation in MDI mode.

f. Command or operation for cancelling compensation

The following command or operation during tool radius compensation, cancels the compensation completely or temporarily.
The command or operation should not be performed:

1. Three consecutive blocks without move command
2. M00 or M01 command
3. M-code set by parameter for stopping advance reading (6 Max)

} Temporarily cancelled

4. M02, M30 commands
5. Reset operation
6. Turning off power supply

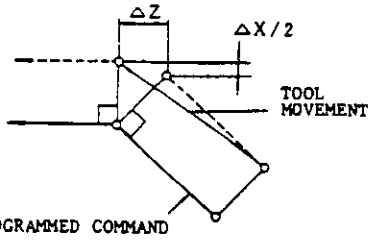
} Completely cancelled

g. Commands causing error

The following commands must not be given, for they cause errors.

1. G28, G29, G30
2. G50, G51
3. G74, G75, G76
4. G90, G92, G94
5. G31 - G35
6. G68, G69
7. G122, G123

h. Even in M96 mode (tool nose radius compensation and circular arc path are on), if circular arc distances ΔX, ΔZ are smaller than the fixed values, the tool does not follow the corner circular arc path but moves directly to point B. The fixed values are those which are set by parameter #6230.



In case of $\Delta X/2 \leq NEGNR$

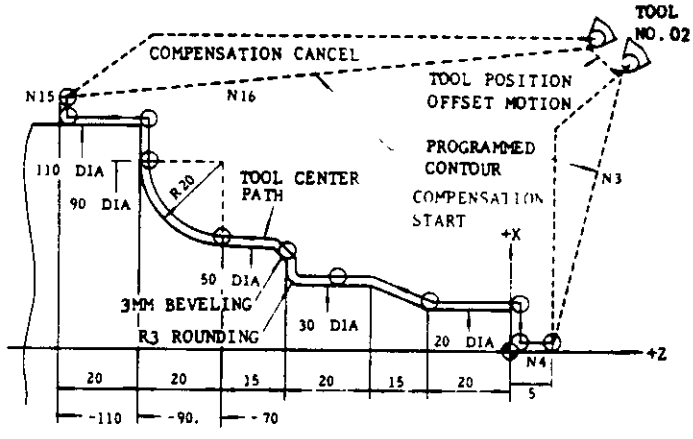
$\Delta Z \leq NEGNR$

NEGNR: Constant value for parameter setting

EXAMPLE A:

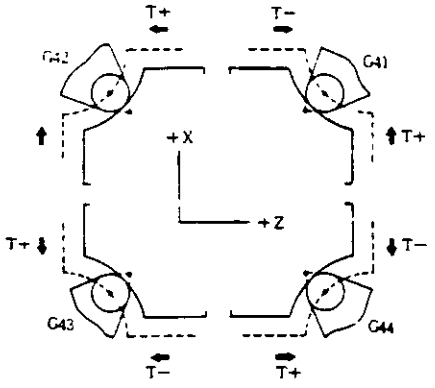
```
N1  G50 X140  Z20  .
N2  G00 S1700 M03 T+0202 .
N3  (G00) G41 X0  Z5  .   ——TOOL NOSE COMPENSATION START BLOCK
N4  G01 Z0 F0.2
N5  X20  .
N6  Z-20  .
N7  X30  W-15  S1100 .
N8  G12 W-20  I3  .   ————ROUNDING
N9  G11 X50  K-3  S700 .  —BEVELING
N10 G01 X-70  .
N11 G22 X90  Z-90  R20  S360 .  —— CIRCULAR ARC BY RADIUS
N12 G01 X110  S300 .
N13 G04 U0   ——DWELL
N14 (G01) Z-110  .
N15 X120  .
N16 G00 X140  Z30  T0200 .  —TOOL NOSE COMPENSATION COMPLETE BLOCK
N17 G40
```

TOOL NOSE COMPENSATION

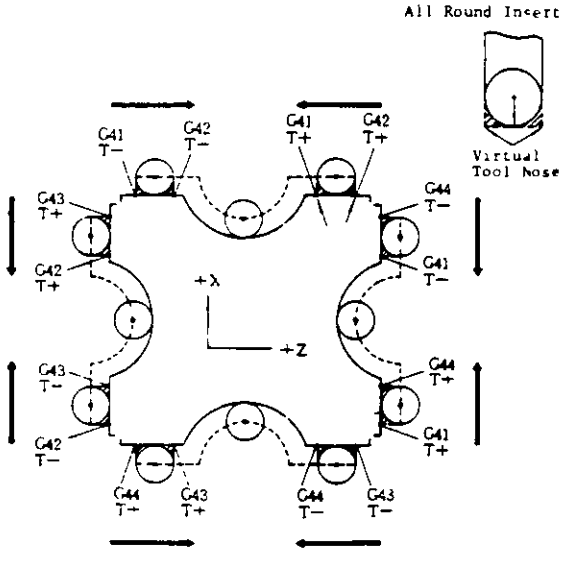


2.8.20 TOOL RADIUS COMPENSATION (G40 THROUGH G44) (CONT'D)

(7) How to cancel compensation mode (Cont'd)



(a) Normal Insert



(b) All Round Insert (G code to be used is decided by setting side of virtual tool nose.)

Fig. 2.33 Relations between G code and Sign of T Code for Tool Nose Radius Compensation

2.8.21 PROGRAMMING OF ABSOLUTE ZERO POINT (G50)

Absolute coordinate system should be set before move command. After setting up the absolute coordinate system, all motions can be commanded on the absolute coordinate system.

(1) G50 X··· Z ·· ,

This command makes the present position of tool tip the absolute coordinates (X, Z). The values with a sign following the addresses X and Z are the distances between tool tip and the absolute zero point (0, 0) to be set. Therefore, it can be said that "G50 command specifies the absolute zero point."

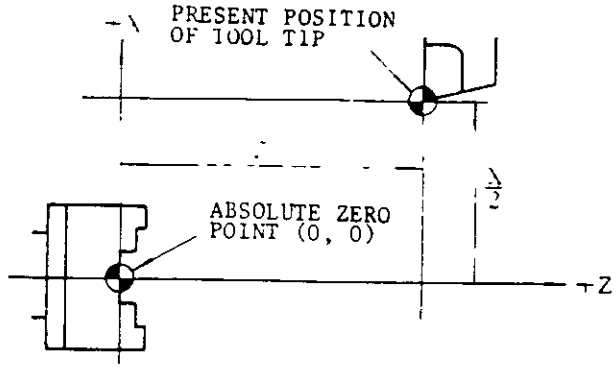


Fig. 2.34

(2) G50 U··· W··· ; (Incremental G50)

When the addresses U and W are specified instead of X and Z, the new absolute coordinate is set up by adding incremental values U (X-axis) and W (Z-axis) to the absolute coordinate previously set.

When the tools are very different in length, the incremental G50 (addresses U, W) is useful. The tools should be divided into two groups. Then, the difference between the length of the reference tool and that of the second group of tools can be set at the incremental G50 command and an absolute coordinate system can be stored.

G50 U100. W-100 ; ··· Setting of Position B

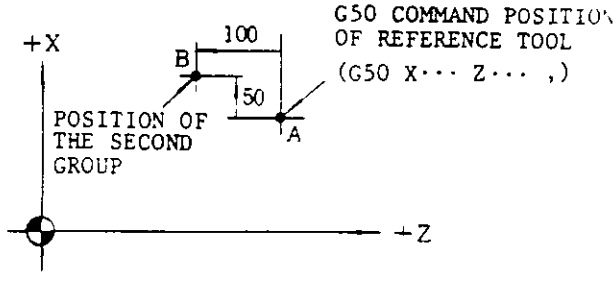


Fig. 2.35

(3) Assume that the tool No. 01 is the reference tool, and perform the setting of the following coordinate system for this tool:

```
G50 X80. Z62. ;
```

then, select the tool No. 02 which has the tool position compensation value shown in the figure below and peroform the compensation operation, and the tool No. 02 moves to point A.

```
N3 G50 X80. Z62. ;
N4 G00 T0101 ;
.
.
.
N10 G00 T0202 ;
.
.
.
```

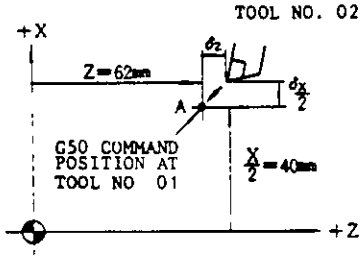


Fig. 2.36

If the coordinate system setting is performed with the reference tool and tool position compensation is applied to the other tool as shown above, the tool movement may be programmed on a single coordinate system for all tool noses.

NOTES:

1. When T, S and M commands are programmed in the block following that containing G50, G00 should be programmed in the block. This designates the traverse rate for tool offset motion.

```
G50 X··· Z··· ;
G00 S500 M03 T0101 ;
```

2. G50 is a nonmodal G code which is valid only in the specified block. Generally, the other G codes, and M, S, and T codes cannot be specified in the same block. Note that G50 S··· ; or G50 T··· ; command is a separate feature and is not for coordinate system setting.

3. G50 should be commanded after the tool offset and tool radius compensation are cancelled.

4. When the power supply is turned on, the present position of tool is set to the coordinate (0, 0). Therefore, the absolute coordinate system should be set up before operation.

5. The current position of the tool in G50 coordinate system is shown in "POSITION ABSOLUTE" of current position display.

6. The coordinate system which was set is not affected by reset operation. The coordinate system is reset by one of the following operations:

   (i) The reset operation by ORG key (see 4.3.4.2, POSITION ABSOLUTE) is performed.

   (ii) G50 X0 Z0 ; command is written in MDI mode and is executed.

   (iii) The power is turned on again.

7. When setting work coordinate system by G50, Work Coordinate System Shift in 2.6.5 will be effective.

2.8.22 MAXIMUM SPINDLE-SPEED SETTING (G50) †

This function is used for the control provided with S 4-digit designation option.

(1) G50 S··· ;

Four digits following the address S specifies the upper limit of spindle speed in rpm. If an S command exceeding the limit is issued in subsequent blocks, the spindle speed is governed at the upper limit.

(2) In G96 (Constant Surface Speed Control) mode, when spindle speed rises up too fast as the current X-coordinate of the tool is too small, the spindle speed is clipped the limit.

EXAMPLE

```
G50 S2000 ;
```

The maximum spindle speed is clipped at 2000 rpm.

NOTES:

1. Maximum spindle speed specified by G50 can be displayed on the CRT display. Refer to 4.3.2.1 Display of Command Data.

2. The specified maximum spindle speed is not cleared by reset operation.

3. In case of S 4-digit designation B, unit of address S is not shown by rpm. Refer to machine tool builder's manual. For S 2-digit designation, this function cannot be used.

2.8.23 WORK COORDINATE MULTI-SHIFT (G50T, G51)

This feature is used in combination with "6.2.3, Work Measured Value Direct Input" option. Hence it is necessary for the programmer to be familiar with paragraph 6.2.3.

The purpose of this feature is to retain a "work coordinate system" with a certain point on the work used as absolute zero point by performing G50T coordinate system setting at the replacement position of each tool. In other words, programming may be performed with a single coordinate system throughout the entire machining.

(1) Tool coordinate value memory (number)

a. It is necessary, before specifying G50T, to write the coordinate data for each tool to the tool coordinate memory. For the writing procedure, see "6.2.3, Work Measured Value Direct Input."

b. The number of available tool coordinate memory units corresponds to the number of tool offset memory combinations as shown below

	Number of tool offset memory combinations	Available tool coordinate memory number
1	0 to 9	51 to 59 (9)
2	0 to 16	51 to 66 (16)
3	0 to 50	51 to 80 (30)

c. For case of use, write the coordinate data for tool No. 01 in tool coordinate memory No. 51, etc., as shown below:

Tool Coordinate Memory	Tool No.
51	01
52	02
:	:
80	30

d. It is assumed that the tool coordinate memory contains the following coordinate data Xtn and Ztn for each tool Tn.

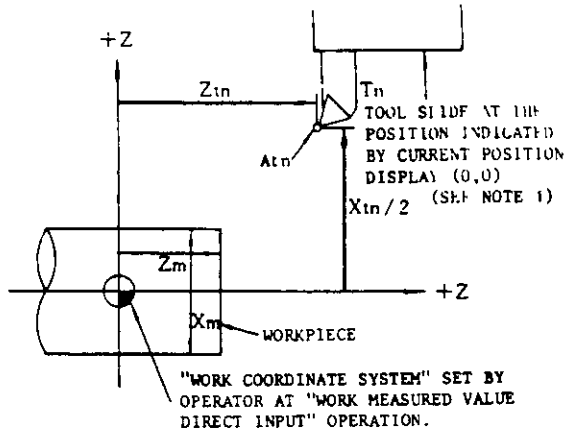


Fig. 2.37

(2) Work coordinate system setting (G50T)

a. G50 T□□ΔΔ

- Specifies tool offset number (00 to 50)
- Specifies tool coordinate memory number (51 to 80)

Using this command, set the coordinate system for each of X-axis and Z-axis with the following work coordinate system setting value:

Work coordinate system setting value = [Unit current position value (Note 1)] + [Content of programmed tool coordinate memory] + [Content of programmed tool offset memory]

Note 1: "Unit current position value" is the one which is shown in "POSITION [EXTERNAL] screen of the current position display (POS) on CRT display.

b. Usually, specify "00" in the tool offset number specification field ΔΔ.

EXAMPLE:

G50 T5100 ;

- "00" specifies the work coordinate system setting with the contents of tool offset memory being zero.

When the above program is specified with the tool slide at a given position (for example, -x, -z in the unit current position display), the work coordinate system defined by the operator is set correctly as shown below:

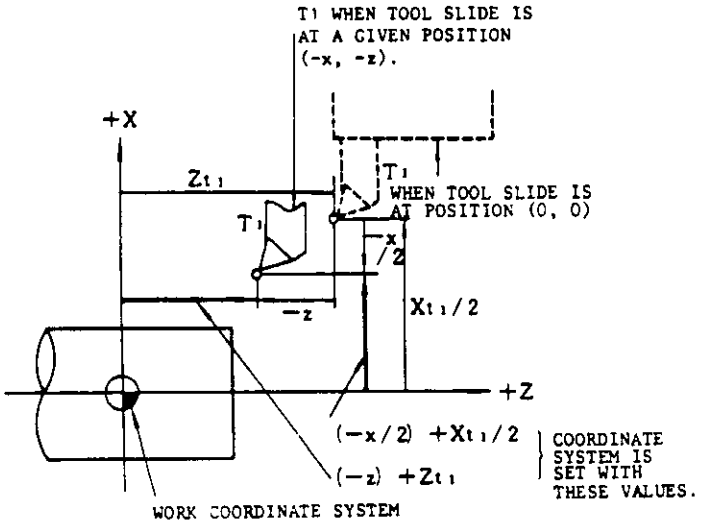


Fig. 2.38

c. G50 T0000 ,

By this command, the coordinate system is set with the unit current position value. This means that the canceling of the work coordinate system setting is performed with the content of tool coordinate memory = "0" and the content of tool offset memory = "0" by the specification of T0000.

(3) Return to current position origin (G51)

a. By G51 , command, tool is returned to the point at rapid traverse rate where the unit current position value is (0, 0), on both X-axis and Z-axis.

b. With a part program that uses work coordinate system setting, the machining start point is the current position value (0, 0) in principle. Hence, the use of G51 command facilitates the return to the start point (0, 0) after completion of machining.

c. G51 command should always be specified on a single block basis.

NOTES:

1. G50 T and G51 are nonmodal G codes which are valid only for the specified blocks.
2. When this function is used, set parameter #6005D5 to 0 (G50 preset of POS-EXTERNAL display is off).
3. G51 ; command is equivalent to the following two block commands.

```
G50 T0000 ;
G00 X0 Z0 ;
```

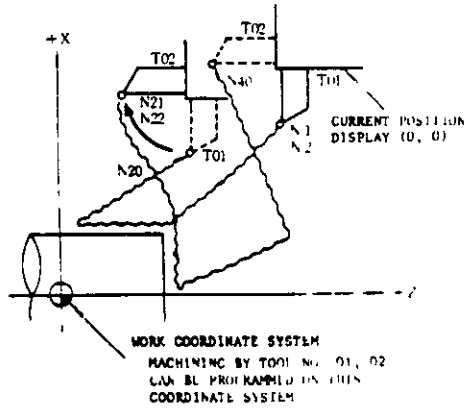
Consequently, after the execution of this command, the tool offset number is cancelled together with the work coordinate system, setting the tool offset number to "00."

4. When the G50 T work coordinate system setting is performed, "2.6.5, Work Coordinate System Shift" is made valid.
5. The current position of the tool in the set work coordinate system is shown in the current position value "POSITION ABSOLUTE." It is not shown in POSITION EXTERNAL.
6. The work coordinate system set by G50 T cannot be cancelled by a reset operation.

EXAMPLE A:
(The start point is current position display (0, 0))

```
N1 G50 T5100 ; ——— Work coordinate system
                   setting for tool No. 01.
N2 G00 T0101 M03 S100 ;
    .              Selection of tool No. 01
    .              (Note 1).
(Machining by tool No. 01)
    .
    .
    .
N20 G00 X··· Z··· ; — Positioning to a given
                       point.
N21 G50 T5200 ; ——— Work coordinate system
                   setting for tool
                   No. 02.
N22 G00 T0202 ; ——— Selection of tool
                   No. 02 (Note 1).
    .
    .
    .
(Machinig by tool No. 02)
    .
    .
    .
N40 G51 ; ——— Return to current position
              display (0, 0).
```

Note 1: The tool position offset in T0101 and T0202 commands may be used for the compensation for tool wear. When specified during machining, the tool position offset may also be used for the compensation for taper machining.



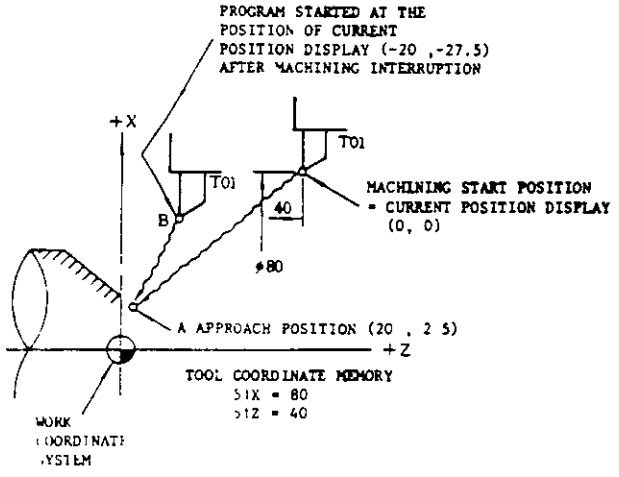
2.8.23 WORK COORDINATE MULTI-SHIFT (G50T, G51)†

NOTES.

EXAMPLE B.

If the machining which was started by the following program is interrupted and the program is restarted without returning the tool to the machining start point, the tool correctly moves to the first approach position.

```
N1  G50  T5100  ;
N2  G0101  ,
N3  G96  S150  M03  ;
Ⓐ N4  G00  X20.  Z2.5  ,
        .
        .
        .
```



This is because N1 G50 T5100 ; command at point B performs coordinate system setting with the following values to retain the work coordinate system, thus keeping approach position A unchanged:

X = (-20.) + (80.) = 60.

Z = (-27.5) + (40.) = 12.5

EXAMPLE C.

This example shows a program for which the replacement position of each tool is different from each other, and the values for work coordinate system setting.

Tool Coordinate Memory

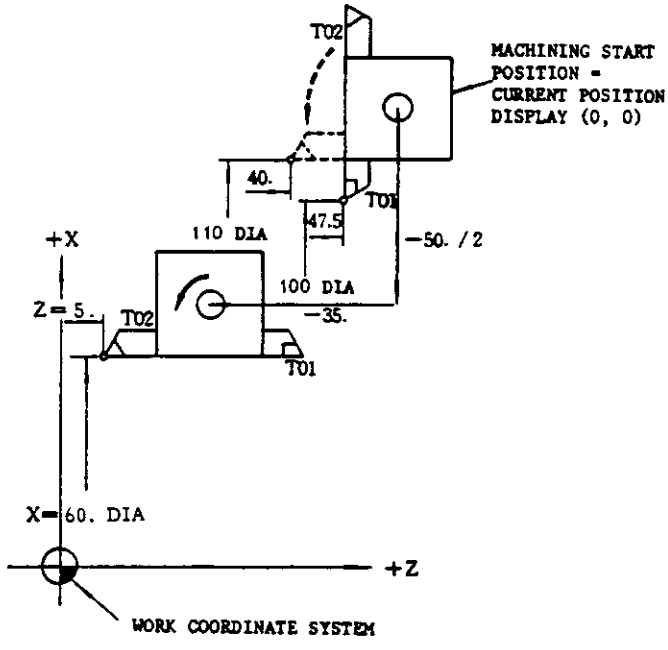
No.	X	Z
51	100.	47.5
52	110.	40.

```
N1  G50  T5100  ;
N2  G00 T0101  M03  S1000  ,
        .
  (Machined by T01)
        .
        .
        .
N25  T0000  ,
N26  G00  X-50.  Z-35.  ;
                ··· Tool replacement position
                    of T02 is current position
                    display (-50, -35).
N27  G50  T5200  ;
N28  G00  T02020  M03  S800  ;
        .
        .
    (Machining by T02)
        .
        .
        .
N48  G51  ;
```

The coordinate system setting values by this command are as follows:

X = (-50.) + 110. = 60.

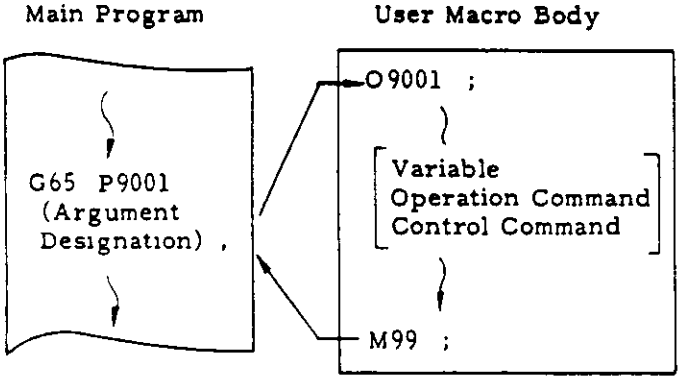
Z = (-35.) + 40. = 5.

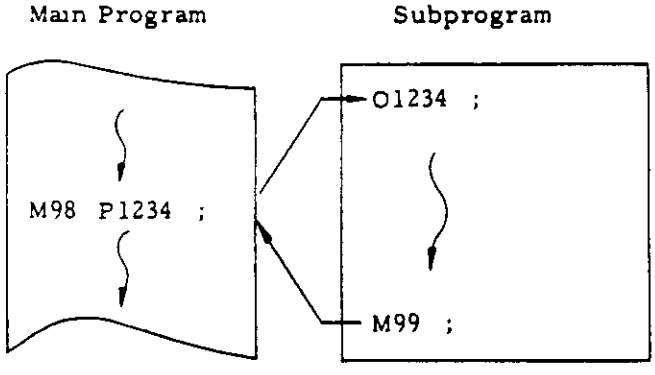


2.8.24 USER MACRO (G65 AND G66)

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

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





However, unlike a subprogram, a user macro allows.

(1) Use of variables.

(2) Computation between variables or between constants.

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

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

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

2.8.24.1 User Macro Call Commands

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

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

(1) Simple Call (G65)

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

The macro program, whose program number was specified by P, is called and is executed L times. The default value of L is 1. When the designation of an argument to the user macro is desired, specify it in (argument designation). "Argument designation" is the assignment of real numbers to the "local variables" used in the user macro. For details, see 3 3 ARGUMENT DESIGNATION.

(2) Modal Call (G66 and G67)

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

This command calls the macro subroutine specified by program number P. Each time a move command is executed, the specified macro is run L times.

G67 ;

This command cancels the modal call mode.

(3) Macro Call by Arbitary G Code

Gxx (argument designation) ;

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

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

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

\

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

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

(4) Macro Call by M Code

G· · X Z··· Mxx ;

This command may call macros. In this case, the macro is executed after the move command is completed in that block. MF and M codes are not transmitted. For Mxx, four M codes may be designated by parameter excluding M00, M01, M02, M30, M90 through M99.

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

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

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

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

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

When a macro M code is programmed in a macro subroutine that has been called by an arbitrary G code or by a macro M or T code, it will be processed like a normal M code.

(5) Macro Call by T Code

All the T code commands provide a macro call command

G ·· X··· Z··· Txxxx ;

With this command, the macro of program number O9000 is executed after the move command in the same block is completed.

Transmission of T code and TF signal is not performed. Whether a T code is to be used as a macro call command may be specified by the following parameter·

Parameter No.

#6134

0 · · T code designation is handled as a T cod

1 ··· T code designation is handled as a macro call command to call the macro of progran number O9000

When a T code is specified as a macro call command, the value designated by T "xxxx" (up to decimal 4 digits) becomes the argument of common variable #149. NOTE THAT THE ARGUMENT DESIGNATION OTHER THAN THIS IS NOT PERMITTED. When a T code is programmed in a macro subroutine that has been called by an arbitrary G code or by a macro M or T code, it wil be processed like a normal T code.

(6) Multiple Call

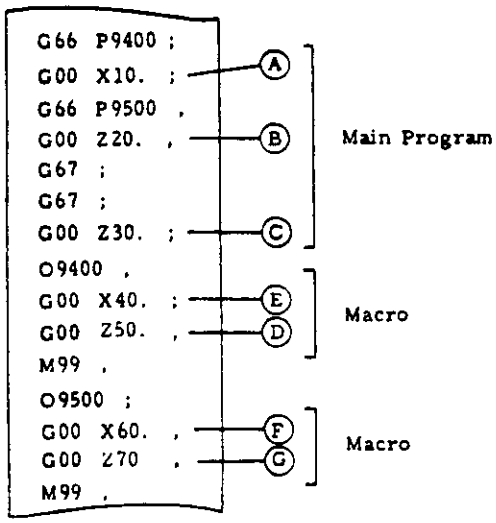
a. G65 Simple Call And G66 Modal Call

As a subprogram is called from another subprogram, a user macro may be called from another user macro. Quadruple nesting is permitted for simple call and modal call combined. Multiple call is disabled for the macro call by arbitrary G code, or M code or T code.

b. Multiple Call by G66 Modal Call

In modal call, each time a move command is executed, the designated macro is run. This is also valid for the move command in the macro called by multiple call. The macros are sequentially executed from the one designated latest.

Sample Program



The above sample program is executed in the following order:

Ⓐ → Ⓔ → Ⓓ → Ⓑ → Ⓕ → Ⓔ → Ⓓ → Ⓖ → Ⓔ → Ⓓ → Ⓒ

2.8.24.2 Argument Designation

Argument is the real value to be assigned to a variable used in the user macro body. Argument designation, therefore, is the act of assigning real values to variables. Argument designation is of type I and type II, which can be selected as required.

(1) Argument Designation I

Argument may be designated in any address except for G, L, N, O, and P. The relationships between the argument designation addresses and the variables are as shown below.

Address of Argument Designation I	Variable in User Macro Body
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

For the address in which no argument need be designated, the command may be omitted.

Sample Program

```
                #1   #2    #24     #26
                 ↑    ↑     ↑       ↑
G65 P9100 L3 A30  B60  X34.8  Z50.2 ,
                              |
                              Argument Designation Part
```

(2) Argument Designation II

A, B, and C arguments and 10 sets of I, J, and K arguments may be designated. I, J, and K must be designated in this order. The relationships between the argument designation addresses and the variables are as shown below.

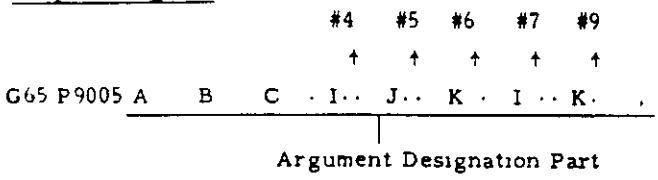
Address of Argument Designation II	Variables in User Macro Body
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11
K_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

The suffixes 1 through 10 to I, J, and K are determined by the order of the designated I, J, and K combinations.

For the address in which no argument need be designated, the command may be omitted.

2.8.24.2 Argument Designation (Cont'd)

Sample Program



(3) Position of Decimal Point in Argument

An argument may generally be designated with a sign and decimal point. For the designation without decimal point, the position of decimal point is as shown on the next page.

Address in Argument Designation	Metric Input	Inch Input
A, B	3	3
D, H	0	0
E	4	6
F (In G99 mode)	2	4
F (In G98 mode)	0	2
I, J, K, C	3 (2)	4 (3)
M, S, T	0	0
Q	0	0
R	3 (2)	4 (3)
U, V, W	3 (2)	4 (3)
X Y Z	3 (2)	4 (3)

The value shows the position of decimal point as counted from the least significant digit. The value in parentheses indicates the number of digits that follows decimal point at the time of parameter #6006-D5 = 1.

(4) Considerations in Argument Designation

a. Argument designation types I and II may be used concurrently If the same variable has been designated twice, the last one is validated.

b. For both types I and II, addresses I, J, and K should be designated in this order. The other addresses may be designated in any order.

c. In the argument designation part, negative sign and decimal point may be used regardless of the address

d. In G65 and G66 blocks, G65 and G66 should always be specified before each argument designation This holds true with the macro call by G code

2.8.24.3 Overview Of User Macro Body

A user macro body is programmed using the combination of the following commands.

(1) Variables

a. Local variable (#1 through #33)

b Common variable (#100 through #549)

c System variable (#1000 through #5104)

(2) Operation Commands

a. Arithmetical operations (+, -, *, /, ···)

b. Functional operations (SIN, COS, ROUND,···)

(3) Control Commands

a Branch command (IF ⟨qualification⟩ GO TO n)

b. Repeat command (WHILE ⟨qualification⟩ DO m)

Using these commands, a program which requires complicated operations and conditional judgements may be written in the general format. Hence, the feature of user macro is to enable the programming of the wide range of NC functions from a simple machining cycle which is rather a subprogram to a special, complicated canned cycle and the storing of these cycles in the machine. Described below are details of the commands mentioned above.

2.8.24.4 Variables

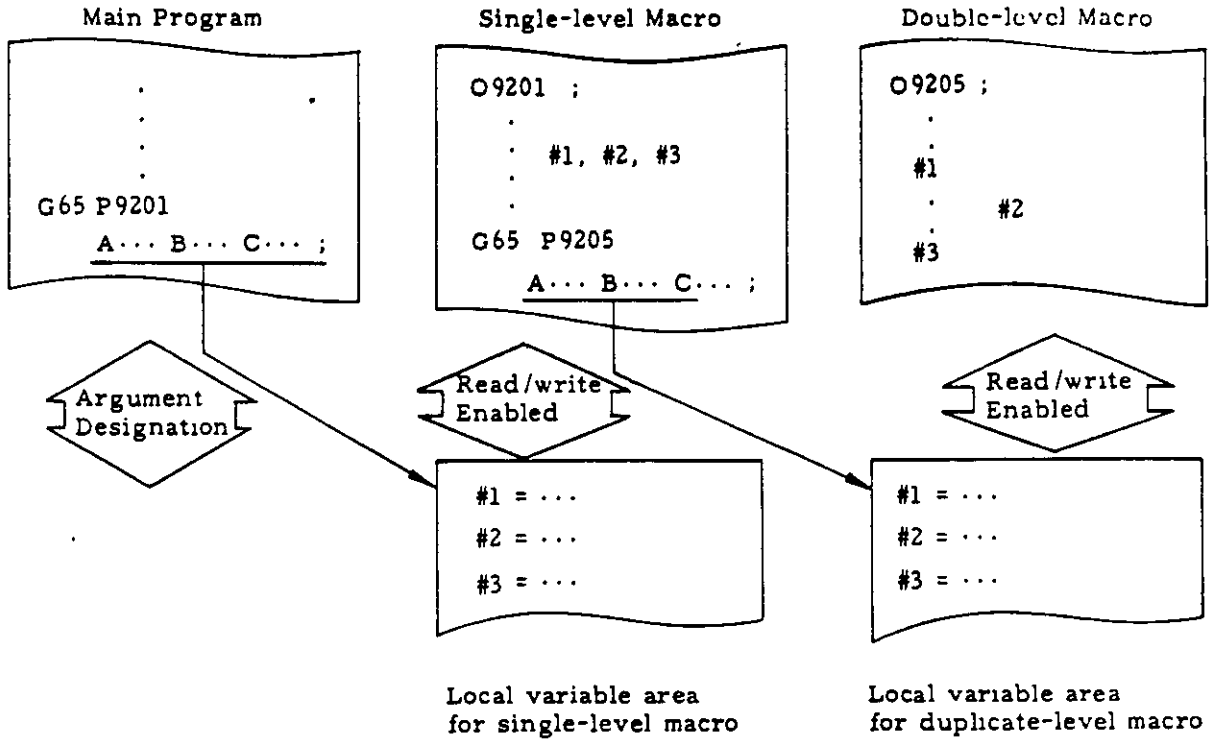
Instead of directly assigning a value to an address in a user macro body, the address may be designated by a variable. When this variable is called during execution, the corresponding value is fetched from the variable area to provide the address value.

. There are three types of variables: local variable, common variable, and system variable. Each is identified by a variable number.

To the local variables, real numbers can be assigned using the argument designation part of macro call command by G65 or G66.

(1) Local Variables (#1 through #33)

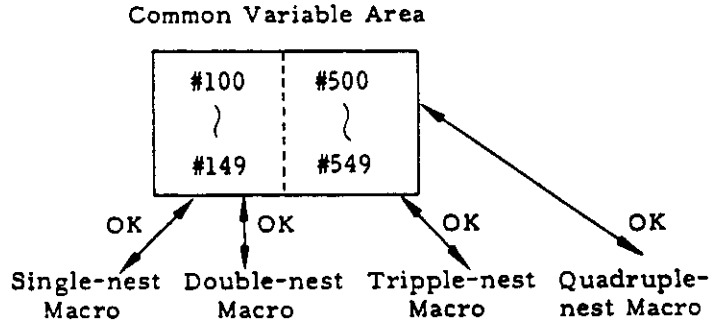
A local variable is the one that is used for each macro locally, That is, when the local variable is used, the variable area (#1 through #33) is independently allocated for each macro call, Certain values are stored by argument designation, and the results of operations in macro are retained



Hence, the variables #1, #2, #3, ··· of the same macro assume different values each time it is called. Each local variable is reset for each macro call and is registered by argument designation. The variable not designated becomes "blank." Each local variable is set to "blank" at the time of power-on and reset operations.

(2) Common Variables (#100 through #149, #500 throuth #549)

A common variable may be shared by all macros and through all macros of all nesting levels. That is, the common variable enables a macro to refer to the results obtained by another macro.

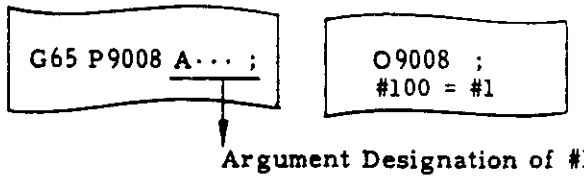


Common variables are divided into the following two types depending on clear conditions:

a. #100 through #149: These common variables are cleared at the time of power-on and reset operations and are set to "blank." In some controls, they are not cleared by reset operation if parameter #6008D1 is set at 1.

b. #500 through #549: These common variables are not cleared at the time of powe-on and reset operations.

The common variables are available to the user without restrictions. They cannot be designated by arguments. Indirectly, however, they can be designated as follows.



(3) System Variables

A system variable is the one whose use is unique to the system. There are following types of system variables:

a. Interface input signals ... #1000 through #1015, #1032†

b. Interface output signals ... #1100 through #1115, #1132†

2.8.24.4 Variables (Cont'd)

c. Tool offset amount, tool coordinate data, and tool wear amount ... #2001 through #2050, #2051 through #2080, #2081 through #2099, #2101 through #2150, #2151 through #2180, #2181 through #2199, #2201 through #2250

d. Alarm message display ... #3000

e. Clock ... #3001, #3002

f. Single-block stop and auxiliary-function completion wait control ... #3003

g. Feed-hold feedrate-override, and exact-stop control ... #3004

h. RS232C data output ... #3100 (print out feature).

i. Modal information ... #4001 through #4120

j. Positional information ... #5001 through #5102

Note: The interface input and output signals of a. and b. may not be installed. Follow the specifications of the machine tool builder.

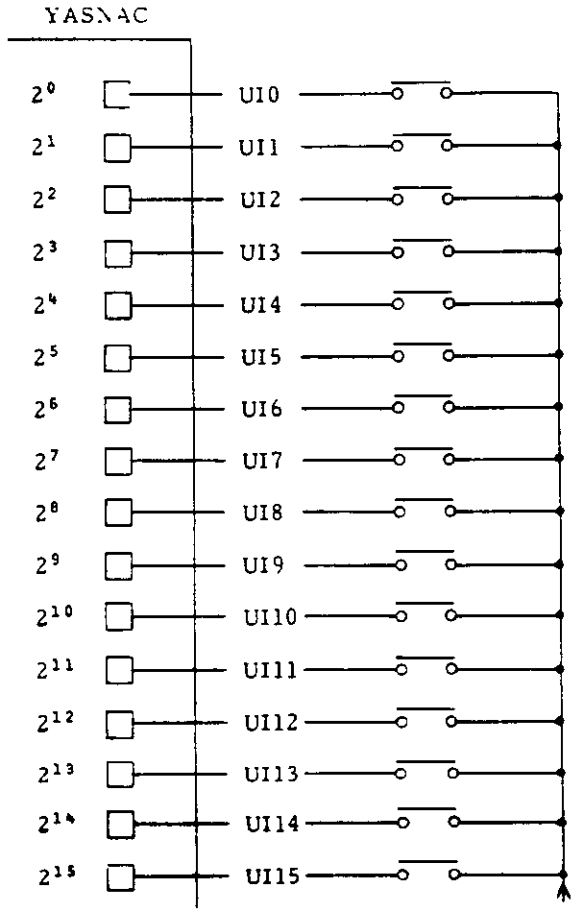
The following paragraphs describe the details of the variables mentioned above.

a. Intervace Input Signals (#1000 Through #1015, #1032)+

i. When one of the system variables, #1000 through #1015, is specified to the right-hand side of an operational expression, the on/off state of each of user-macro-dedicated 16-point input signals is read. The relationships between the input signals and the system variables are as shown below.

#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000
UI7 2^7	UI6 2^6	UI5 2^5	UI4 2^4	UI3 2^3	UI2 2^2	UI1 2^1	UI0 2^0
#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008
UI15 2^{15}	UI14 2^{14}	UI13 2^{13}	UI12 2^{12}	UI11 2^{11}	UI10 2^{10}	UI19 2^9	UI18 2^8

Variable Value	Input Signal
1	Contact Closed
0	Contact Open



Each read variable is 1, 0 or 0.0 when the associated contact is "closed" or "open" respectively, regardless of the unit system of the machine.

ii. When system variable #1032 is designated, the input signals (UI0 through UI15) that consist of 16 points (16 bits) are collectively read as a decimal positive value.

$$\#1032 = \sum_{i=0}^{15} \#\left[1000 + i\right] * 2^i$$

Sample Program

```
IF [ #1015 EQ 0 ] GO TO 100;
```

Bit 2^{15} (UI15) is read and, if it is "0," a branch is made to sequence number N100.

```
#130 = #1032 AND 255
```

Bits 2^0 through 2^7 (UI0 through UI7) are collectively read to be stored in common variable #130 as a decimal positive value.

Note: System variables #1000 through #1032 cannot be placed to the left-hand side of operational expressions.

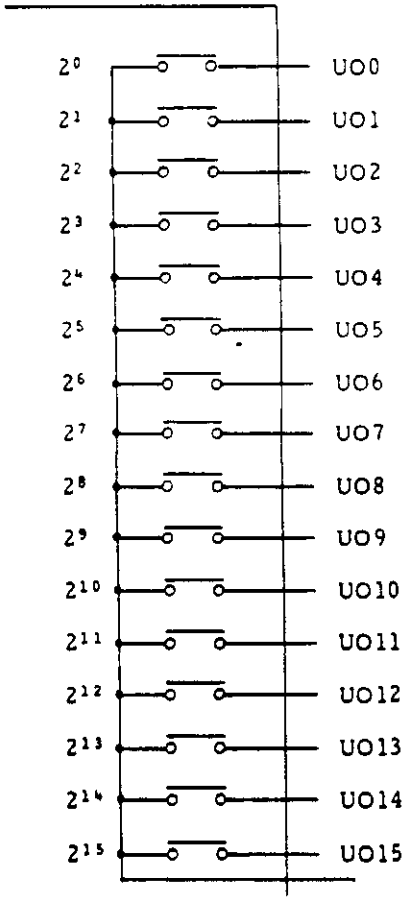
b. Interface Output Signals (#1100 Through #1115, #1132)†

i. When one of system variables #1100 through #1115 is specified to the left-hand side of an operational expression, an on or off signal can be sent to each of the user-macro-dedicated 16-point output signals. The relationships between the output signals and the system variables are as shown below:

#1107	#1106	#1105	#1104	#1103	#1102	#1101	#1100
UO7 2^7	UO6 2^6	UO5 2^5	UO4 2^4	UO3 2^3	UO2 2^2	UO1 2^1	UO0 2^0
#1115	#1114	#1113	#1112	#1111	#1110	#1109	#1108
U15 2^{15}	U14 2^{14}	U13 2^{13}	U12 2^{12}	U11 2^{11}	U10 2^{10}	U09 2^9	U08 2^8

Variable Value	Output Signal
1	Contact Closed
0	Contact Open

YASNAC



When 1.0 or 0.0 are substituted in any of #1100 through #1115, the associated output contact is output in the 'closed' or 'open' state.

ii. When system variable #1132 is specified, the output signals (UO0 through UO15) that consist of 16 points (16 bits) are collectively output. At this time, the decimal positive value substituted in #1132 is output in the form of binary 16-bit value.

$$\#1132 = \sum_{i=0}^{15} \#\left[1100 + i\right] * 2^i$$

iii. With system variables #1100 through #1132, the value sent last is retained. Hence, when one of them is written to the right-hand side of an operational expression, its value is read.

iv. Considerations

When any values other than 1.0 or 0.0 are substituted into one of the system variables, #1100 through #1115, the values are handled as follows:

"Blank" is assumed to be "0."
Values other than "blank" and 0 are assumed to be "1."

Sample Program

```
#1107 = #10 ; (#10 = 1.5)
```

The output signal of bit 2^7 (UO7) is outputted in the contact (closed) state.

```
#1132 = (#1132 AND 240) OR (#8 AND 15)
```

The output signal of bits 2^4 through 2^7 (UO4 through UO7) are outputted without change and the contents of local variable #8 are outputted to the output signals of bits 2^0 through 2^3 (UO0 through UO3).
(Decimal 240 = 11110000 Binary 15 = 00001111)

c. Tool Offset Amount And Tool Coordinate Data, Tool Wear Amount

#2001 - #2050, #2051 - #2080, #2081 - #2099,
#2101 - #2150, #2151 - #2180, #2181 - #2199,
#2201 - #2250

i. When one of the system variable #2001 through #2250 is specified to the right-hand side of an operational expression, the tool offset amount, tool coordinate data, and tool wear amount can be read.

ii. The relationships between the tool offset numbers and the system variables are as shown below:

	System Variable	Tool Offset Memory No.
X-axis	#2001 to #2050	01 to 50
Z-axis	#2101 to #2150	01 to 50
Tool Nose Radius	#2201 to #2250	01 to 50

	System Variable	Tool Coordinate Memory No.
X-axis	#2051 to #2080	51 to 80
Z-axis	#2151 to #2180	51 to 80

	System Variable	Tool Wear Amount Memory No.
X-axis	#2081 to #2099	81 to 99
Z-axis	#2181 to #2199	81 to 99

iii. When one of the above system variables is specified to the left-hand side of an operational expression, its value can be changed.

Sample Programs

```
#116 = #2016 ;
```

The contents of tool offset number 16 for X-axis are substituted for common variable #116.

```
#2081 = #24 ;
```

The tool wear amount (memory No. 81) of X-axis is erased and the contents of local variable #24 are set.

d. Alarm Message Display (#3000)

When a condition to be alarmed occurs in a user macro program, system variable #3000 may be specified to put the machine in the alarm state.

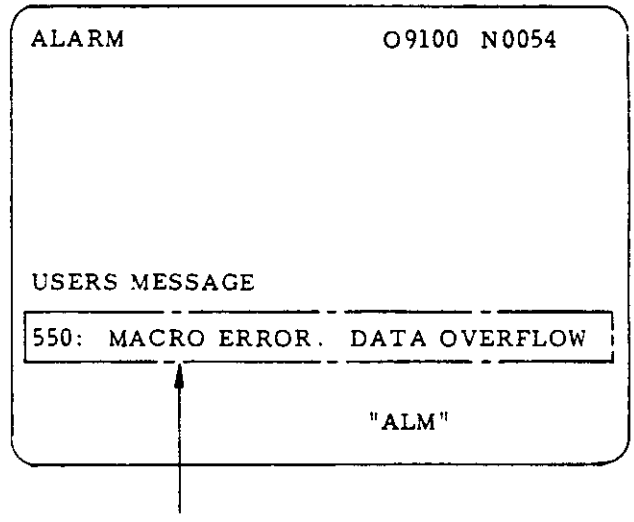
i. #3000 = n (<alarm message>);

Using this command, specify the alarm message (less than 32 characters) preceded by a 3-digit alarm number n and enclosed with control-in and control-out symbols. The alarm number should be three digits and not be one used by the machine.

ii. When this #3000 command is executed, "ALM" or "A/B" is displayed on the bottom of CRT screen regardless of the mode and function. Its message can be seen by the following operation:

Press ALM function key.

The alarm number and message and displayed on the bottom of CRT screen.



Message display area and sample display

When RESET key is pressed after removal of the cause of alarm, the message display and the alarm state can be cleared.

Sample Program

```
#3000 = 550 (MACRO ERROR: DATA OVER-
            FLOW)
```

. Clock (#3001, #3002)

. When system variable #3001 or #3002 for clock s specified, the clock can be read.

System Variable	Type	Unit	At Power-On	Count Condition
#3001	Clock 1	1 ms	Reset to "0"	Always
#3002	Clock 2	1 s	Same as power-off time	When STL signal is on

i. To preset the clock, substitute the value with his system variable put at the left-hand side of he expression.

Sample Program

#3001 = 0 , ··· The clock is preset to value "0."

ii. Restrictions

The accuracy of clock 1 is 8 ms. When 4294968000 msec has been reached, and overflow occrus, setting the clock to "0."

The accuracy of clock 2 is 8 ms. When 429496800 sec has been reached, an overflow occurs, setting the clock to "0."

Sample Program

Main Program

G65 P9351 T5 ,

Argument designation of variable #20

··· Macro to wait for 5 seconds

Macro Program

```
O9351 ;
#3002 = 0 ;
WHILE [#3002 GE #1 ] D01 ,
END 1 ,
M99 ,
```

f. Single Block Stop And Auxiliary Function Completion Wait Control (#3003)

When the value listed in the following table is substituted in system variable #3003, the single block switch can be disabled or the next block may be entered without waiting for the checking of the finish signal (FIN) of the auxiliary function (MST).

When the finish signal is not waited for, the distribution end signal (DEN) is not transmitted. In this case, the FIN is waited for in the block with the check skip cleared. Hence, when the FIN is not waited for, be careful not to specify the next auxiliary function.

#3003	Single Block Switch	FIN Signal
0	Valid	Waited
1	Invalid	Waited
2	Valid	Not waited
3	Invalid	Not waited

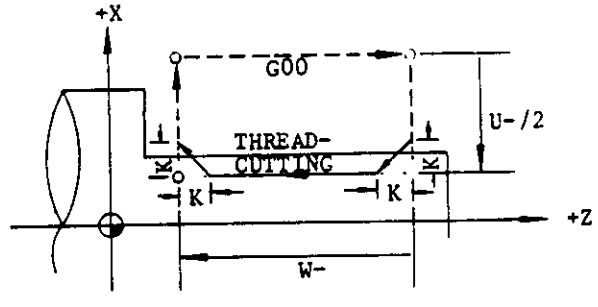
g. Feed-Hold, Feedrate-Override, And Positioning Completion Control (#3004)

When the value listed in the following table is substituted in system variable #3004, feed hold, feedrate override, and positioning completion can be made valid or invalid.

#3004	Feed Hold	Feedrate Override	Positioning Completion
0	Valid	Valid	Valid
1	Invalid	Valid	Valid
2	Valid	Invalid	Valid
3	Invalid	Invalid	Valid
4	Valid	Valid	Invalid
5	Invalid	Valid	Invalid
6	Valid	Invalid	Invalid
7	Invalid	Invalid	Invalid

Sample Program

Special Threadcutting Cycle (Incremental Command)



Macro Call

G65 P9093 U-. . W-... K... F... ;

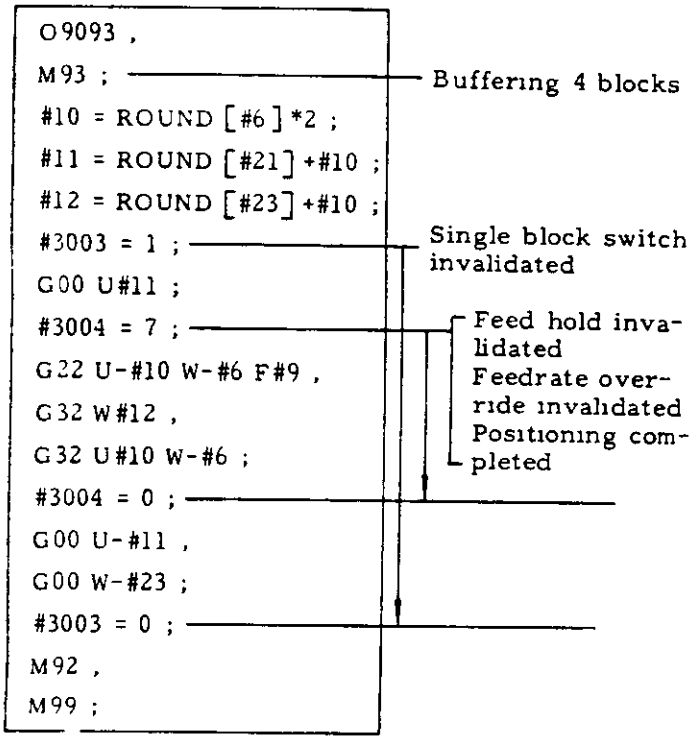
#9: Lead

#6: Without sign, radius value

#23: Negative value

#21: Negative value, diameter value

Macro Program



h. RS232C Data Output (#3100)

When system variable #3100 is specified, messages and NC internal data can be outputted to external equipment via RS232C data input/output interface. If the external equipment is a printer, the above information is printed.

i. Output of Meassages

#3100 = (<Message>)

When this command is specified, the message enclosed by control-in and control-out is outputted, via RS232C interface.

Each output message is followed by CR/LF (Carriage Return/Line Feed). Hence, when #3100 = ( ) is specified, only CR/LF is outputted, which is useful in tabulating the punched data.

ii. Output of Data

#3100 = < variable >

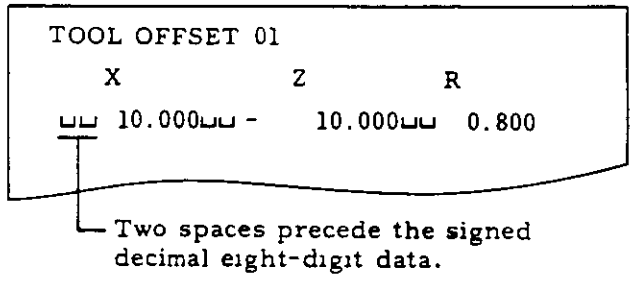
When this command is specified, the value of the local variable, common variable, or system variable at the right-hand side is outputted via RS232C interface as signed decimal 8-digit data.

iii. The above output is performed when system variable #3100 is executed in the macro program. It is required, therefore, to previously attach the external equipment such as a printer via RS232C interface and preset the parameters that use the interface.

Sample Program

```
#3100 = (  ) ; ··· Carriage return/line feed
#3100 = (TOOL OFFSET 01);
#3100 = (␣␣··· X␣␣··· Z␣␣··· R);
#3100 = #2001 ; ··· = 10.000 mm
#3100 = #2101 , ··· = -10.000 mm
#3100 = #2201 ; ··· = 0.800 mm
#3100 = (  ) ;
```

Printout Data



i. Modal Information(#4001 Through #4120)

i. When one of system variables #4001 through #4120 is specified, the modal commands that are specified up to the immediately preceding block can be known. These modal commands are sometimes called the current values of modal information commands.

System Variable	Modal Information
#4001	G code (group 01)
⟨	⟨
#4021	G code (group 21)
#4108	E code
#4109	F code
#4113	M code
#4114	Sequence number
#4115	Program number
#4119	S code
#4120	T code

ii #4001 through #4120 cannot be placed to the left-hand side of the operation expression.

Sample Program

Main Program

```
G65 P9602 <Argument Designation> ;
```

Macro Program

```
O9602 ;
#1 = #4001 ;
G00 X    Y··· ;
G01 Z··· F··· ,
G03 X··· Z··· R··· ,
G00 Z··· ;
G#1 ,
M99 ,
```

G codes (G00 through G03) of 01 group are retained.

G codes of 01 group are restored.

j. Positional Information (#5001 Through #5102)

When system variables #5001 through #5102 are specified, various positional information can be obtained.
The unit of the information is millimeters or inches

	Unit
Metric input	0.001 millimeter
Inch input	0.0001 inch

In the user macro body, the "input unit x 10" feature is invalid.

System Variable	Positional Information	Read During Move
#5001	X-axis block end position (ABSIO)	Enabled
#5002	Z-axis block end position (ABSIO)	
#5021	X-axis current position (ABSMT)	(Note) Enabled
#5022	Z-axis current position (ABSMT)	
#5041	X-axis current position (ABSOT)	(Note) Enabled
#5042	Z-axis current position (ABSOT)	
#5061	X-axis skip signal position (ABSKP)	(Note) Enabled
#5062	Z-axis skip signal position (ABSKP)	
#5081	X-axis tool offset amount	Enabled
#5082	Z-axis tool offset amount	
#5101	X-axis servo position deflection amount	(Note) Enabled
#5102	Z-axis servo position deflection amount	

Note: Reading of #5021, #5022, #5041, #5042, #5101, and #5102, when commanded during movement, will be performed after completion of the movement.

Mnemonic	ABSIO	ABSMT	ABSOT	ABSKP
Meaning	End position of block immediately before.	Command current position (same as POS. MACHINE display).	Command current position (same as POS. UNIVERSAL display).	Position at which skip signal did not go on in G31 block.
Coordinate system	Work coordinate system	Machine coordinate system	Work coordinate system	Work coordinate system
Tool position offset	Not included		Included	Included
Tool radius compensation Amount	Not included		Included	Included

Notes:

1. When the skip signal is not turned on in G31 block, the skip signal position is at the end of G31 block.

2. The "input unit x 10" feature is valid up to the macro call block (the argument designation part by G65 or G66) but is invalid in the user macro body.

3. System variables #5001 through #5102 may not be placed to the left-hand side of operational expression.

2.8.24.4 Variables (Cont'd)

Sample Program

The tool is positioned to the specified location (X, Y, Z) on machine coordinate system, performs the specified M feature, and returns to the start point.

Main Program

```
G65 P9603 X  . Z ... M... ;
```

#13
#26
#24

Macro Program

```
C 9603 ,
#1 = #5001 ,
#2 = #5002 ,

G91 ,
G00 X [#24-#5021] ;
G00 Z [#26-#5022] ;
M#13 ,
G00 Z#2 ,
G00 X#1 ,
M99 ,
```

List of Variables

Variable No	Meaning
#1 to #33	Local variables
#100 to #149	Common variables (reset to "blank" at power-off).
#500 to #549	Common variables (retained at power-off).
#1000 to #1015	Interface input signals (each signal for each bit).
#1032	Interface input signal $(\sum_{i=0}^{15} \#[1000+i] * 2^i)$.
#1100 to #1115	Interface output signals (each signal for each bit).
#1132	Interface output signal $(\sum_{i=0}^{15} \#[1100+i] * 2^i)$
#2001 to #2050 #2101 to #2150 #2201 to #2250	Tool offset amount (X-axis, Z-axis, nose radius)
#2051 to #2080 #2151 to #2180	Tool coordinate data (X-axis, Z-axis)
#2081 to #2099 #2181 to #2199	Tool wear amount (X-axis, Z-axis)
#3000	Alarm message display.
#3001	Clock 1 (in units of 1 ms)
#3002	Clock 2 (in units of 1 s).
#3003	Single block stop, auxiliary function complete wait control.
#3004	Feed-hold, feedrate-override, and exact-stop control
#3100	RS232C data output (print out feature).
#4001 to #4120	Current value of modal information command.
#5001 to #5002	End position of the immediate preceding block (for each axis).
#5021 to #5022	Current position of machine coordinate system (for each axis)
#5041 to #5042	Current position of POS. UNIVERSAL (for each axis).
#5061 to #5062	Position at which G31 skip signal is turned on (for each axis).
#5081 to #5082	Effective tool position offset amount (X-axis, Z-axis)
#5101 to #5102	Servo position deflection amount (for each axis)

(4) Variable Representation

Each variable is represented in a variable number that follows #.

a. How to designate a number directly:

#i (i = 1, 2, 3, 4, ...)

Sample
#10
#130
#2000

b. How to designate an expression as a variable number:

[<expression>]

Sample
[#100]
[#500 + 1]
[#20/2]

In the following description, variable #i may be replaced with variable #[< expression >].

(5) Variable Reference

a. The value that follows an address may be replaced by a variable.

When < address> #i or <address> -#i is specified, the value of the variable or its negative value (complement, more exactly) is made the specified value of the address.

Sample
```
#30 = 1. 0 ;
#101 = 100. ;
#103 = 300. ,
#140 = 0. 3 ;
G#30 X#101 Z-#102 F#140 ;
```

The above specification is equivalent to the specification below.

```
G01 X100. Z-300. F0.3 ;
```

Notes:

1. Address /, O, and N may not refer to variables.

Sample /#8, N#100 ... Error.

2. A variable number may not be replaced with a variable.

Sample
##20 ... Error.
#[#20] ... Correct.

3. When a variable is used as address data, the values below the least significant digit are rounded.

Sample

(i) When #1 = 45.2346
X#1 ... = X45.235 mm (for metric input)

(ii) When #2 = 0.255
F#2 ... = F0.26 (mm/rev)

(iii) When #3 = 5.37672
G04 P#3 ... = G04 P5.377 (sec)

(iv) When #4 = 2.7236
M#4 ... M03
G#4 ... G03

4. Value for each address should not exceed the maximum programmable value.

5. The value that follows an address may be replaced with <expression>.

6. The constant without decimal point enclosed in brackets [ ] is assumed to have a decimal point at its end.

(6) Undefined Variable

The value of an undefined variable is assumed to be "blank." An undefined variable occurs in the following situations:

a. The local variable for which argument designation was not performed in macro call command.

b. Common variables #100 through #149 at the time of power-on and reset operations.

c. The local variables and common variables for which the values were not written from MDI panel.

2.8.24.4 Variable (Cont'd)

Designation and function of <blank> is classified in the following two versions A and B. The control is set for either version. Switching from versions A to B and from B to A cannot be interchanged.

		Version A	Version B
1	Concept of #0.	•No conception of #0. •Commanding #0 causes alarm.	•#0 defined as variables of <blank>. •Commanding #0 at the left-hand side of the equation.
2	Variable <blank> is commanded in the replacement equation.	•Where #2 is <blank>, command #3 = #2; means #3 = 0.	•Where #2 is <blank>, command #3 = #2; means #3 = <blank>.
3	Variable <blank> is commanded in the part program.	•Where #2 is <blank>, command 600 × #2; is equivalent to command G00 × 0;	•Where #2 is <blank>, command 600 × #2; is equivalent to command G00; (Address is ignored.)
4	Variable <blank> is commanded in the condition of EQ and NE.	•Where #2 is <blank>, #3 is 0. ① Condition "IF #3 EQ #2" is established. ② Condition "IF #3 NE #2" is not established.	•Where #2 is <blank>, #3 is 0. ① Condition "IF #3 EQ #2" is established. ② Condition "IF #3 EQ #2" is established.
5	Others	#3 = # [#0 + #0] #3 = #2 * #0 ; #3 = #0 + #0 ; #3 = #0 / #0 ; #3 = 5 * #0 ; #3 = 2 - #0 ; means #3 = 2 #3 = 5 / #0 ; causes alarm. <Blank> in the replacement described above is treated as "0." •Condition IF #3 GE#2 is established when #2 and #3 are <blank>, or #2 is 0 and #3 is <blank>. •Condition IF#3 LT#2 is not established when #2 and #3 are <blank>, or #2 is <blank>, and #3 = 0.	In these commands, #3 = 0.

2.8.24.5 Operation Commands

Various operations can be performed between variables and between variables and constants. The operation expression is represented in the form of #i = <expression>, in which <expression is a general arithmetic operational expression produced by combining variables and constants with operators and functions. The available opearations and functions are as follows. Instead of #j and #k, constants may be used.

(1) Variable Difinition and Replacement

#i = #j ··· definition, replacement.

(2) Add-Type Operations

#i = #j + #k ··· Sum.
#i = #j - #k ··· Difference.
#i = #j OR #k ··· Logical sum (for each of 32 bits).
#i = #j XOR #k ··· Exclusive logical sum (for each of 32 bits).

(3) Multiply-Type Operations

#i = #j * #k ··· Product.
#i = #j / #k ··· Quotient.
#i = #j AND #k ··· Logical product (for each of 32 bits).

Note: In OR, XOR, or AND operation, the variable value (or constant) is converted into the binary 32-bit equivalent and the operation is performed on each bit.

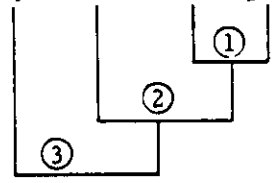
(4) Functions

#i = SIN [#j] ··· Sine (in degrees).
#i = COS [#j] ··· Cosine (in degrees).
#i = TAN [#j] ··· Tangent (in degrees).
#i = ATAN [#j/#k] ··· Arctangent (in degrees).
#i = SQRT [#j] ··· Square root.
#i = ABS [#j] ··· Absolute value.
#i = BIN [#j] ··· Convert from BCD.
#i = BCD [#j] ··· Convert into BCD.
#i = ROUND [#j] ··· Produce integer by roundin
#i = FIX [#j] ··· Truncate the fractions.
#i = FUP [#j] ··· Raise the fractions to a unit

(5) Combinations of Operations

The above operations and functions may be used in combinations. A functional operation is performed first. Then, a multiply-type operation is performed. An add-type operation is performed last.

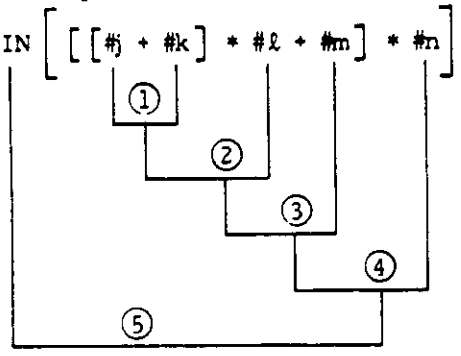
Sample #i = #j + #k * SIN [#ℓ]



(6) Change of Operational Order by [ ]

Priority may be given to an operation by enclosing it in brackets [ ]. Up to quintuple (five-hold) nesting of brackets is permitted including those of functional operations.

Sample #i = SIN [[[#j + #k] * #ℓ + #m] * #n]



(7) Considerations for Operational Commands

a. The constant without decimal point used in ⟨expression⟩ is assumed to have a decimal point at its end.

b. When used in conditional Expression IF or WHILE, function ROUND truncates the fractions.

c. When used in address data, function ROUND rounds off the part below the least significant digit.

Sample (a)

#10 = 12.3758

When the least significant digit of address X is 0.001 mm, the following command

G00 X [ROUND [#10]];
means

G00 X12.376 ;
because 8 of 12.3758 is rounded.

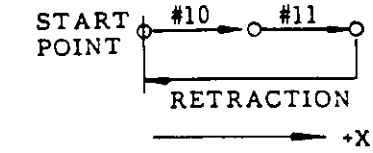
This command is also equivalent to

G00 X#10 ;

Usually, ROUND is not used as mentioned above; it is used as shown below:

Sample (b)

When ROUND is used as follows, the program returns to the start point correctly.



① #10 = 12.3758 ,
② #11 = 13.1236 ;
③ G00 U#10 ;
④ G01 U#11 F··· ;
⑤ G00 U- [ROUND [#10] + ROUND [#11]] .

This is because the data of #10 and #11 in ③ and ④ blocks are substantially rounded before being executed.

If ⑤ block is

⑤' G00 U- [#10 + #11] ;

then, the movement is made by the following amount:

U- [#10 + #11] = U - [12.3758 + 13.1236]
= U - [25.4994]
= U - [25.499]

On the other hand, block movement of ③+④ is

U#10 + U#11 = U12.376 + U13 124
= U25.500

Hence, the program of ⑤' is not correct.

(8) Operational Errors

The data format and the operational errors in the user macros are as follows:

a. Data Format

The numeric data handled in user macros are of the floating point format.

M * 2E

where, M is sign + data 52-bit binary.
E is sign + data 10-bit binary.

b. Operational Errors

Each time an operation is performed, the following error is caused and is accumulated. The number of significant digits is 15 to 16, which compensates the error sufficiently.

2.8.24.6 Control Commands

The commands which control the flow of the micro-program are of the following two types:

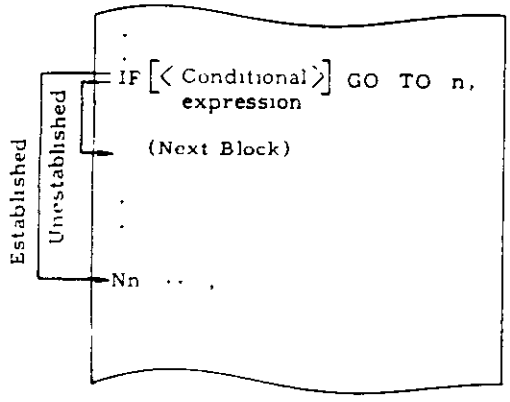
a. Branch command ... IF [< conditional expression>] GO TO n ;

b. Repeat Command ... WHILE [< conditional expression >] DO m ,

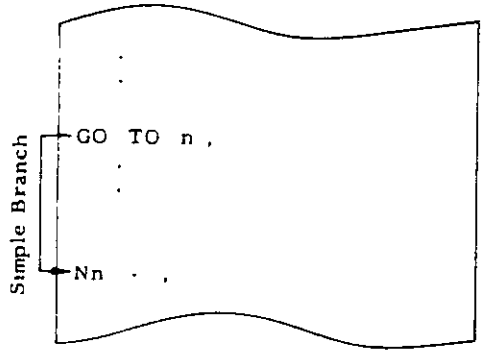
(1) Branch Command

a. IF [< conditional expression >] GO TO n ;

If < conditional expression> of this command is established, a branch is made to the block of sequence number n within the same program. When a variable or an expression is used for n, the branch destination may be changed. If the condition is not satisfied, the program proceeds to the next block.



IF [<conditional expression>] may be omitted to provide a simple branch command as shown below:



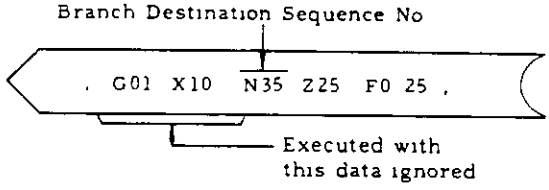
Conditional expressions are EQ, NE, GT, LT, GE, and LE. They are represented as follows:

Conditional Expression	Meaning
#i EQ #j	(#i = #j)
#i NE #j	(#i ≠ #j)
#i GT #j	(#i > #j)
#i LT #j	(#i < #j)
#i GE #j	(#i ≥ #j)
#i LE #j	(#i ≤ #j)

A constant and < expression> may be used for #i and #j. A variable and <expression > may be used for n.

Notes:

1. The sequence number must be located at the head of the block when it is called for by a branch command. Otherwise, the data prior to the sequence number is ignored as shown below:



2. The reverse branch on the program takes longer execution time than the forward branch.

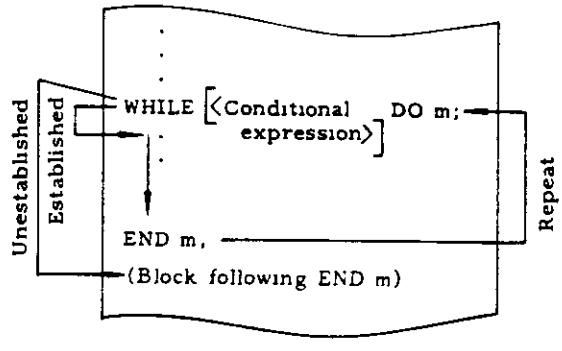
(2) Repeat Command

a. WHILE [< conditional expression >] DO m ;
(m = 1, 2 and 3)

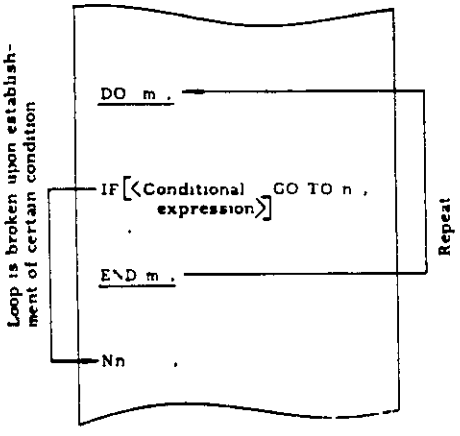
.
.
.

END m ;

While <conditional expression> is satisfied, the blocks between DO m and END m are repeated. When it is unsatisfied, the processing branches to the block following END m.



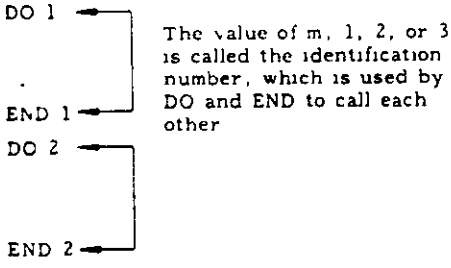
When the specification is made omitting WHILE [<conditional expression>], the blocks between DO m and END m are repeated infinitely. Generally, this is used in the format shown below.



Notes:

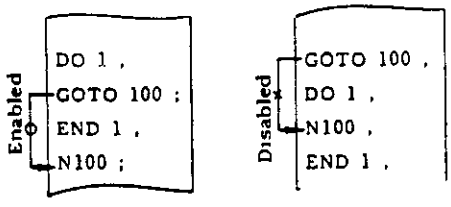
1. DO m should be specified before END m.

2. m of DO m and END m should have the same value. However, only 1, 2, or 3 may be specified in m.

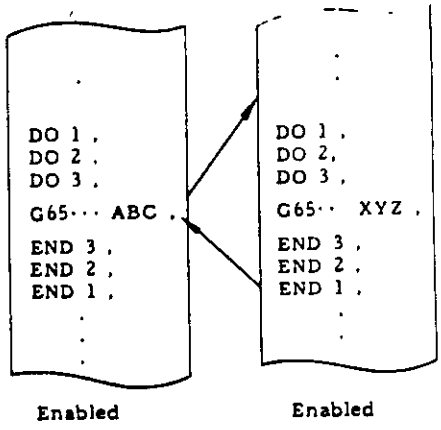


3. The same identification number may be used repeatedly except where repeat ranges overlap.

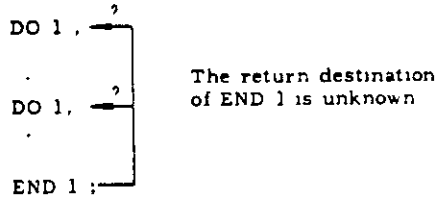
4. To get out of a DO loop, a GO TO n can be used. However, a GO TO n does not enable entrance to a DO loop as shown below:

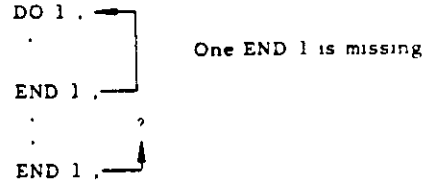


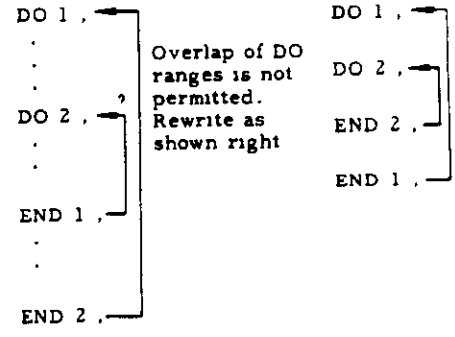
5. Triple DO-loop nesting is permitted for each micro program.



6. The codings shown below cause an error:

(i)


(ii)


(iii)


2.8.24.6 Control Commands (Cont'd)

(iv)
```
DO 1 ,
DO 2 ,
DO 3 ,
 .
 .
DO 1 ,  ──┐
 .        │  Quadruple nesting.
 .        │  Max nesting permit-
END 1 ,  ──┘  ted is triple
 .
END 3 ,
 .
END 2 ,
 .
END 1 ,
```

(v)
```
DO 1 ,
N7000  ,  ──┐
          │  DO loop may not
END 1     │  be entered from
          │  outside
IF  GO TO 7000 ,  ──┘
```

2.8.24.7 Registration Of User Macros

(1) How to Make Registration of User Macros

The registration and edit of user macro bodies (macro programs) are performed in the same manner as usual part programs and subprograms. Hence, there is no program size restriction that applies to the user macro body. Part programs, subprograms, and macro programs may be stored together in the part program memory to its full capacity.

(2) Classification of Program Numbers

The program numbers are classified into the following

Program No	Classification
O1 to O7999	These programs may be registered, erased, or edited without restrictions
O8000 to O8999	When D4 of #6004 is set to 1, the registration, erase, and edit of programs are disabled
O9000 to O9999	When D7 of #6021 is set to 1, the registration, erase, and edit of programs are disabled

2.8.24.8 Display And Write Of Local And Common Variables

Local variables (#1 through #33) and common variables (#100 through #149, #500 through #549) can be displayed and written by the following operations

(1) Display Operations

Display of Variables

a. Press SET function key. Mode select position may be provided anywhere.

b. Key-in the variable number and press [CURSOR ↓] key or [↑ CURSOR] key. However, # need not be keyed in. Ten sets of variable numbers including the specified variable number and their data are displayed. The data are displayed in the signed 8-digit integer part and the 8-digit fraction part.

```
Macro nesting level
(0  Macro not in execution)
SETTING  MACRO        O1234 N0035
LEVEL 0
#0100          -12345678  12345678
#0101                 0  00000001
#0102                 3  00000000
#0109
                                RDY
```

Sample Display of Common Variables

c. Press [PAGE ↓] key or [↑ PAGE] key, and the display may be scrolled up or down.

Remarks

a. Common variables may always be displayed for review.

b. For local variables, those of the macro currently executed are displayed. Consequently, when a macro of a nesting level is in execution, the local variables belonging to macros of the other nesting levels cannot be seen. The local variables after completion of execution are all rest to "blank."

(2) Write Operations

Writing of Values to Variables

a. Press SET function key. Mode select position may be provided anywhere.

b. Key-in the variable number to be written press [CURSOR ↓] key or [↑ CURSOR] key. However, # need not be keyed-in. The keyed-in variable number is specified and the cursor is positioned to it.

c. Key-in the value to be written. Press WR key. The keyed-in value is stored as the data of the variable number with the cursor positioned.

d. Press [CURSOR ↓] key or [↑ CURSOR] key or [↑ PAGE] key or [PAGE ↓] key to move the cursor.

e. Repeat operations in c. and d. to write the values to the desired variables.

Remarks

a. Common variables can always be changed.

b. Local variables may not be written at any time other than when a macro is in execution. Any attempt to do so is invalidated. However, rewriting of local variables during macro execution may cause an unexpected failure. Before attempting the rewriting, stop the machine operation by single stop function and check to see if it is safe to rewrite variables.

c. The written local variables and common variables #100 through #149 are reset to "blank" by the reset operation or the power-on operation.

2.8.24.9 Considerations And Remarks For User Macros

(1) Summary of Restrictions

a. Available Variables

#1 through #33 ··· Local variables.

#100 through #149 ┐ Common variables.

#500 through #549 ┘

System variables

b. Available Variable Values

Maximum value ··· $\pm 10^{+308}$

Minimum value ··· $\pm 10^{-308}$

c. Constant Values Usable in < Expression >

±(8 digits above decimal point). (7 digits below decimal points).

Sample Maximum value ±99999999.9999999

Minimum value ±0.0000001

d. Operational Accuracy

Decimal 15 digits significant.

e. Macro Call Maximum Nesting Level

Quadruple (four-hold).

f. Maximum Nesting Level of Repeat Command

Triple (three-hold) for each macro.

g. Repeat Command (DO) Identifier m

m = 1, 2, and 3.

h. Maximum Nesting Level of Brackets

Quintuple (five-hold).

(2) Difference between User Macro and Subprogram

a. User macros G65 and G66 allow argument designation but the subprogram (M98) does not.

b. The user macro directly branches to the user macro body without executing any command that was specified in G65 or G66 block and has no relationship with the macro. With the subprogram, however, a branch is performed after the execution of the command (if any) other than P and L in M98 block.

2.8.24.9 Considerations And Remarks For User Macros (Cont'd)

c. The maximum nesting level of user macro is quadruple including G65 and G66 calls. That of subprograms is also quadruple but separately.

d. If user macros are specified via MDI during automatic operation, the maximum nesting level is restricted to quardruple. With subprograms, up to four levels of nesting are permitted in tape mode or memory mode, or separately in MDI mode.

(3) Relationship with MDI Operation

a. MDI writing permits the macro call and the execution of the called macro.

b. MDI writing does not permit or execute macro body commands such as operational commands and control commands.

c. When a macro program being executed is stopped by the single block stop function, any MDI writing command not related to the macro may be specified and executed.

(4) Relationship with Address Search

The address search function is not permitted to search for the sequence numbers in the user macro body.

(5) Relationship with Single Block Switch

a. The operational command and control command blocks do not single-block stop if the single block switch is turned on. This switch is enabled for the other macro program blocks.

b. However, when setting number #6004D_1 = 1, the single block switch is enabled for the operational command and control command.

c. System variable #3003 (for the control of single block stop, see 2.8.24.5) and setting #6004 D_1 mentioned above operate as shown below:

Setting #6004	System Variable #3003	When Single Bock Switch is on
D1 = 0	= 1 or 3	None of the operational commands, control commands, and general commands stop.
D1 = 0	= 0 or 2	Operational commands and control command do not stop. General commands stop.
D1 = 1	= 1 or 3	None of the operational command, control commands, and general commands stop.
D1 = 1	= 0 or 2	All of the operational commands, control commands, and general commands stop.

(6) Relationship with Optional Block Skip

The slash "/" character used in the right-hand side of an operational expression or in brackets is assumed to be the operator for quotient. It does not mean the optional skip.

(7) Parameter Setting of Program Number Classification

(1) Disabling of Program Registration, Erase, And Edit

The following setting is permitted to protect the registered user macros and subprograms from inadvertent destruction.

Setting Number

#6004

D2 = 1 ··· The programs of program numbers #8000 through #8999 are disabled for registration, erase, and edit.

D2 = 0 ··· Registration, erase, and edit are enabled.

Parameter Number

#6021

D7 = 1 ··· The programs of program numbers #9000 through #9999 are disabled for registration, erase, and edit.

D7 = 0 ··· Registration, erase, and edit are enabled.

(8) Effects of Reset Operation

a. A reset operation resets all local variables (#1 through #33) and part of common variables (#100 through #149) to "blank."

b. A reset operation resets the user-macro multiple call state and the multiple DO loop state making the program pointer return to the program head.

(9) Special Codes Usable in User Macro Body

(1) The special codes listed below may be used in the user macro body:

	Code	Use	EIA Code 8	7	6	5	4	○	3	2	1	ISO Code 8	7	6	5	4	○	3	2	1
	SP	For comment				○		∘				○		○			∘			
*	(	For alarm message comment				○	○	∘		○				○		○	∘			
*	)			○			○	∘		○		○		○		○	∘			○
	+	Add		○	○	○		∘						○		○	∘		○	○
	-	Subtract		○				∘						○		○	∘	○		○
		For comment		○				∘	○	○				○	○	○	∘		○	
	/	Divide			○	○		∘			○			○		○	∘	○	○	○
	#	Variable	Parameter designation											○			∘		○	○
*	*	Multiply					○	∘	○	○		○		○		○	∘		○	
*	=	Equal			○	○	○	∘		○	○	○		○	○	○	∘	○		○
*	[	Bracket(open)				○	○	∘	○			○	○			○	∘		○	○
*	]	Bracket (close)			○		○	∘	○			○	○			○	∘	○		○
	$	For comment	○		○		∘	○					○			∘	○			
	@		○			○	∘	○	○	○	○	○				∘				
	?		○		○	○	∘	○	○				○	○	○	∘	○	○	○	
		Decimal point		○	○		○	∘		○	○		○		○	∘	○	○		

Notes.

1. For the hole pattern of EIA code, when the character is attached with an asterisk, the pattern shown above is standard. However, other patterns may be specified by using the following parameters:

#6110 ·· [
#6111 ···]
#6112 ··· *
#6113 ··· =
#6114 ··· (
#6115 ···)

Read the desired hole pattern in the binary value, convert it into the decimal equivalent, and set it to the parameter. For example, the hole pattern shown below is set as "152":

8	7	6	5	4	0	3	2	1
○			○	○	∘			

When the value of the parameter is "0," the hole pattern listed in the above table is provided.

2. When the codes shown below are outputted from the NC unit for punch-out or other purposes, the upper code (UC) or lower code (LC) is outputted immediately before.

a. Codes preceded by UC ··· #, +, $, ?.

b. Code preceded by LC ··· @.

c. Codes preceded by UC only at parameter designation ... (,), *, =.

2.8.24.10 Alarm Number Of User Macros

Shown below are the user-macro-associated alarms and their causes.

105 MACRO ERROR (CONSTANT)

The number of constants is in excess of the specified range.

106 MACRO ERROR

There are too many G67 cancel codes.

107 MACRO ERROR (FORMAT)

A format other than expression has an error.

108 MACRO ERROR (UNDEFIN #NO)

The value not defined as a variable number is designated.

109 MACRO ERROR (#NO NOT LEFT)

The variable of assignment statement is the one that is disabled for assignment.

110 MACRO ERROR ([ ] 5 LIMIT)

The bracket nesting level is in excess of the upper limit (5).

111 MACRO ERROR (MOVE G66 - M99)

A move command is specified in the macro end command M99 called by G66.

112 MACRO ERROR (5)

The macro call nesting level is in excess of the upper limit (4).

113 ———

2.8.24.10 Alarm Number Of User Macros (Cont'd)

114 MACRO ERROR (DO FORMAT)

DO and END are not paired.

115 MACRO ERROR ([ ]UNMATCH)

The format of <expression> has an error.

116 MACRO ERROR (DO - END NO.)

DO m is not in the range of $1 \leq m \leq 3$.

117 ———

118 MACRO ERROR (GO TO N)

GO TO n is not in the range of $0 \leq n \leq 9999$.

2.8.24.11 Exercises Of User Macro

(1) Canned Cycle by G92

T (Teacher). We have discussed many complicated rules you have to understand to write user macros. Now, let's create some user macros as exercises. Let's take straight thread-cutting cycle by G92, because it is a simple operation.

S (Student): Where shall we start?

T: An example of usual G92 command takes the following format:

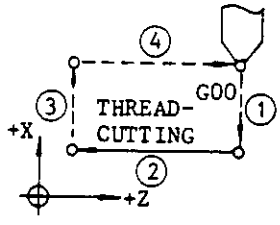
(P1)

```
G92  U-50.  W-60.  F6.0  ;
```

This command is divided into the following and executed within the NC unit. It is assumed that Rapid Pull Out of Threading is not included in this command.

(P2)

```
① G00 U-50. ,
② G32 W-60. F6.0 ,
③ G00 U50. ,
④ G00 W60. ,
```



First, these moving distances and lead threads can all be converted into variables.

S: They are local variable #1 through #33, aren't they? But which type of local variable?

T: Type I for small number of variables. This type allows the use of U, W and F and therefore makes the argument designation easier to understand.

S: OK. When type I is used, we have the following variables:

(P3)

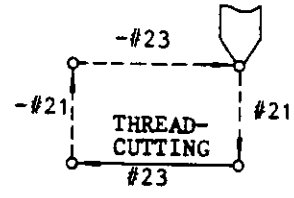
```
U-50.  W-60.  F6.0
 |      |      |
#21    #23    #9
```

T: Right. Using these variables, rewrite the former program (P2).

S: OK.

(P4)

```
① G00 U#21 ;
② G32 W#23 F#9 ;
③ G00 U-#21 ;
④ G00 W-#23 ;
```



Is this all right?

T: Yes. Just add this and we have a complete user macro body.

```
⑤ M99 ;
```

S: That's easy.

T: Then, using G65, create this macro call and the user macro body in the complete formats.

S: Let me try it.

Supposing the program No. of macro body is O9093; the macro call command will be:

(P5)

```
G65 P9093 U-50. W-60. F6.0 ;
```

The user macro body is:

(P6)

```
O9093 ;
G00 U#21 ;
G32 W#23 F#9 ;
G00 U-#21
G00 W-#23 ;
M99 ;
```

T: That looks OK.

****

S: I think something is wrong. With this program, I have to specify points W and F every time!

T: That's true. With a usual canned cycle, when points W and F have been specified once, their values are retained. Thereafter, only U is specified.

S: Do you have any trick to overcome this inconvenience?

T: I do. In such a case, common variables (#100 - #549) help. Using common variables, write the macro to designate the position of points W and F.

S: I've got it! Now, I divide the macro body into two parts as follows:

(P8)

```
O9000 ;
#100 = #23 ;
#101 = #9 ;
M99 ;
```

(P9)

```
O9093 ;
G00 U#21 ,
G32 W#100 F#101 ;
G00 U-#21 ;
G00 W-#100 ;
M99 ;
```

and I write the macro call as follows:

(P10)

```
G65 P9000 W-60. F6.0 ;
G65 P9093 U-50. ;
G65 P9093 U-51.4 ;
G65 P9093 U-52.6 ;
G65 P9093 U... ;
  .
  .
  .
```

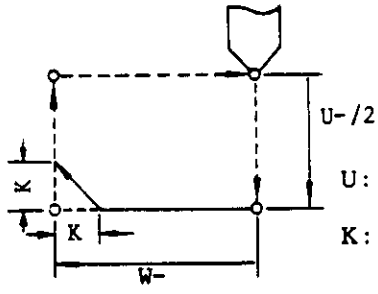
T: Very good.

****

S: I'd like to try to program Rapid threading pull-out.

T: OK. How about designating the width of rapid threading pull-out using address K?

S: All right. Let's see .....



U: DIAMETER VALUE
K: RADIUS VALUE WITHOUT SIGN

Macro call is as follows:

```
G65 P9000 W-60. K4.8 F6.0 ;
G65 P9093 U-50. ;
G65 P9093 U-51.4 ;
G65 P9093 U-... ;
  .
  .
  .
```

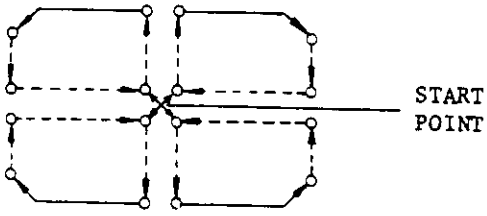
Macro body is as follows:

```
O9000 ;
#100 = #23 ;
#101 = #9 ;
#102 = ABS [ #6];
M99 ;
```

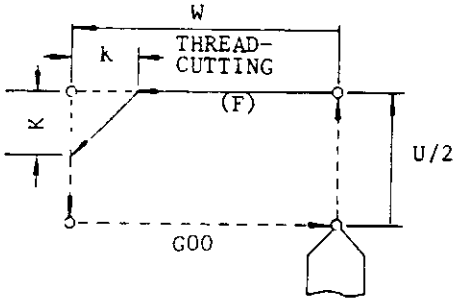
```
O9093 ;
#10 = ROUND [ #102] *2 ;
#11 = ROUND [ #21] + #10 ;
#12 = ROUND [ #100] + ROUND [ #102] ;
G000 U#21 ;
G32 W#12 F#101 ;
G32 U#10 W-#102 ;   RAPID THREADING PULL-OUT
G00 U-#11 ;
G00 W-#100 ;
M99
```

Is this OK?

T Yes. Your reasoning is right. Practically, you had better prevent a malfunction by programming #3003 invalid control of single block or #3004 invalid control of feedhold. This threadcutting can be performed in U- and W-directions only. Now we'd like to expand this function in four directions.



S. I see. Let me think.
--- How about the next program?



U, K Designation with a sign
k Designation without a sign

U = #21 (DIAMETER VALUE)
W = #23
K = #6 (RADIUS VALUE)
F = #9

The macro call command is as follows:

```
G65 P9000 W-45  K4.0 F5.0 ,
G65 P9093 U40  ,
G65 P9093 U41 4 ;
G65 P9093 U .. ,
```

The user macro body is as follows:

```
O9000 ,
#100 = #23 ,------------ W
#101 = #9 , ------------ F
#102 = ABS [#6] ,------ |K|
M99 ,
```

```
O9093 ,
#3003 = 1 ; ------ Single block invalid
M93 ; ------------ 4-block buffering
#10 = ROUND [#102] *2 ,
IF [ABS][#21] LT #10] GO TO 4 ;
IF [#21 GT 0 ] GO TO 1 ;
IF [#21 EQ 0 ] GO TO 4 ;
#11 = ROUND [#21 ] + #10 ; ------ U. Negative
#12 = #10 ;
GO TO 2 ;
N1 #11 = ROUND [#21] -#11 ; ------ U: Positive
#12 = -#10 ;
N2 #13 = ROUND [#102] ;
IF [ABS [#100] LT #13] GO TO 4 ;
IF [#100 GT 0 ]G0T0 3 ;
IF [ #100 EQ 0 ]G0T0 4;
#14 = ROUND [#100] + #13 ; ------ W: Negative
#15 = -#13 ,
GO TO 5 ;
N3 #14 = ROUND [#100] -#13 ; ------ W· Positive
#15 = #13
```

```
GO TO 5 ;
N4 #3000 = 499 (MACRO INPUT ERR.) .
                          |------ Error display
N5 G00 U#21 ,
#3004 = 7 , ------ [Feedhold
                   Feedrate override
                   Positioning
                   completion ] Invalid
G32 W#14 F#101 ;
G32 U#12 W#15 ; ------ [Rapid threading
                       pull-out
#3004 = 0 ;
G00 U-#11 ;
G00 W-#100 ;
M92 ;
#3003 = 0 ;
M99 ;
```

T: Well. If U or W = 0, and |U/2| or |W| < K, error will be displayed in your programming. That's good.

2.8.25 PROGRAM MIRROR IMAGE (G68, G69)†

Program mirror image is the feature to reverse the NC program operation in all directions around the work center line (Z-axis) by the use of G command.

(1) G68 ; ··· Program mirror image on.
The program mirror image on state is held until G69 is specified.

G69 ; ··· Program mirror image off.
The program mirror image off state is held until G68 is specified.

When program mirror image is on, the X-axis operation by the NC program is inverted with Z-axis being the center line. The manual operations (manual continuous feed and handle/step feed) are not affected by this feature.

(2) Details of program mirror image

When the X-axis mirror image feature is on, the movement by the NC program is inverted with Z-axis being the center line. The following inversion is processed in the NC unit:

a. X command for X-axis coordinate value is inverted.

b. U command for X-axis incremental coordinate value is inverted.

c. I command for X-axis coordinate value of arc center is inverted.

d. Circular motion direction inverted.

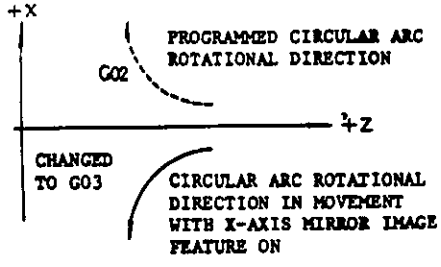


Fig. 2.39

e. I command for X-axis beveling/rounding volume and direction is inverted.

f. I command for canned cycle taper X-axis distance is inverted.

g. U and I commands for special canned cycles finishing allowance, etc. are inverted.

h. The operational direction after X-axis tool position compensation is inverted.

i. T command for tool nose radius compensation tool nose center is inverted in the sign.

j. G command for tool nose radius compensation virtual tool nose position is inverted.

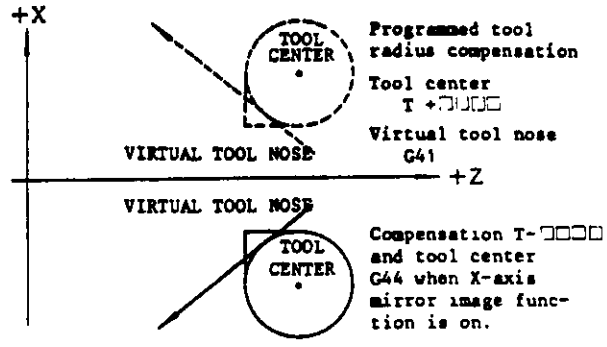


Fig. 2.40

2.8.25 PROGRAM MIRROR IMAGE (G68, G69)† (CONT'D)

(3) Cautions for G68 and G69 commands

a. G68 and G69 are modal G commands which belong to "10" group. They must be specified on a single block basis, in principle.

b. G69 (program mirror image off) is used at the time of power-on, reset operation, and program reset.

c. These commands must be specified in the tool nose radius compensation cancelled state.

d. These commands may not be specified in the finishing shape program of the special canned cycle.

e. If automatic origin return "G28 X··· Z··· ;" is specified when the X-axis mirror image feature is on, the positioning of the intermediate point specified in X and Z is affected by the mirror image, but the machine origin, which is an absolute position, is not affected by this feature.

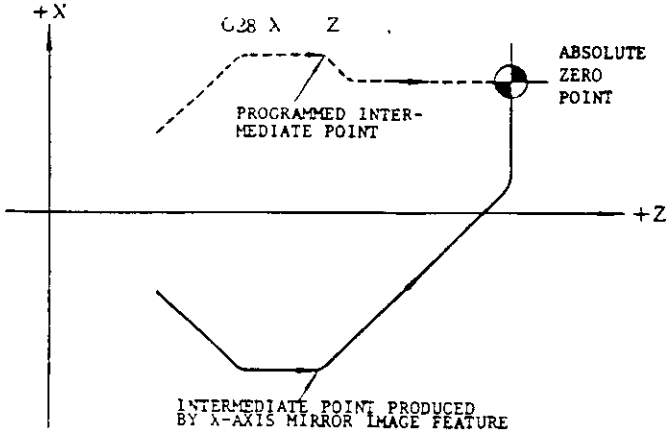


Fig. 2.41

EXAMPLE

Described below is a sample program which uses G68 and G69 for the opposed tool rest shown in the diagram on the following page, and the X-axis movements.

Power on

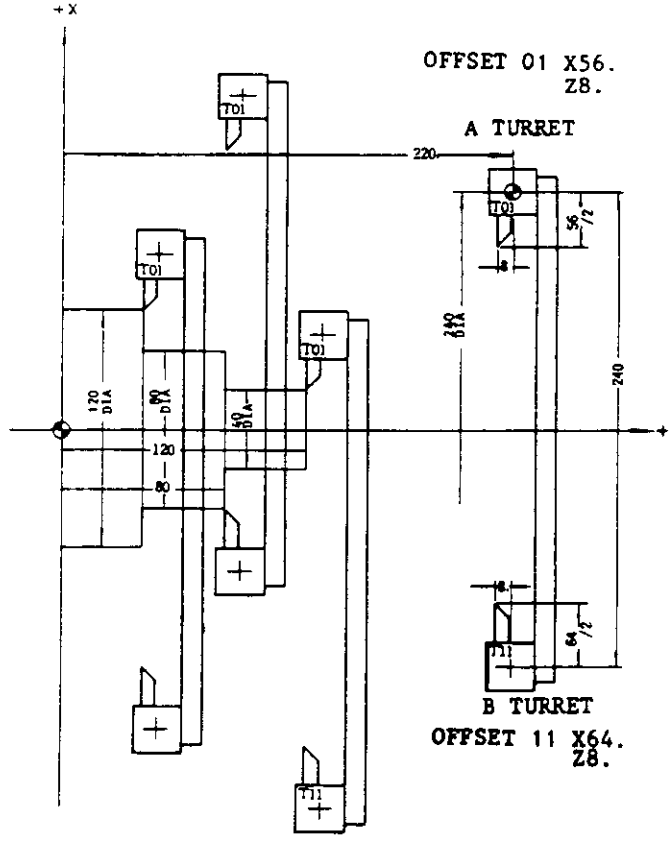
↓

Manual reference point return

↓

Program execution

	X-axis command value display	X-axis movement
N0001 G50 X240. Z220. ;	240.000	
N0002 X40. Z120. T0101 ;	96.000	U-144.000
N0003 G68 ;	-96.000	
N0004 G50 U480. ;	+384.000	
N0005 X80. Z80. T1111 ;	+144.000	U240.000
N0006 G69 ;	-144.000	
N0007 G50 U480. ;	336.000	
N0008 X120. Z40. T0101 ;	176.000	U-160.000



2.8.26 MULTIPLE REPETITIVE CYCLES (G70 TO G76) †

2.8.26.1 General

This option makes program simple and short. For instance, both stock removal and finishing are performed only by commanding the finishing work shape.

Table 2.21 Multiple Repetitive Cycles

G code	Name	Remarks	
G70	Finishing cycle		
G71	Stock removal in turning	Finishing by G70 possible	Tip nose radius compensation possible
G72	Stock removal in facing		
G73	Pattern repeating		
G74	Peck drilling in Z axis	Tip nose radius compensation impossible	
G75	Grooving in X axis		
G76	Automatic threadcutting		

(1) G70 through G76 are in * group and non-modal.

(2) The program of finishing shape specified by G71, G72 and G73 are stored in memory. The memory capacity for the finishing shape is 45 blocks.

Program of finishing shape ≦ 45 blocks

Note: When cornering (G11, G12) and multiple cornering (G111, G112) are used, each block containing them must be counted as the value listed below.

	No of blocks
One block including G11 or G12	Two blocks
One block including G111	Four blocks
One block including G112	Five blocks

(3) The internal memory for storing the finishing shape program:

To shorten the stock removal cycle computation time, the finishing shape program is binary-converted and then is stored in the memory for storing finished shape program (one pair) in the unit. This memory is called the internal memory for finishing shape program, which differs from the part program memory.

(4) In the block after the cycle of G70 through G76, the G codes of 01 group should be specified again. This is because the 01-group G codes specified before the cycle may have been changed to other G codes by the execution of this cycle.

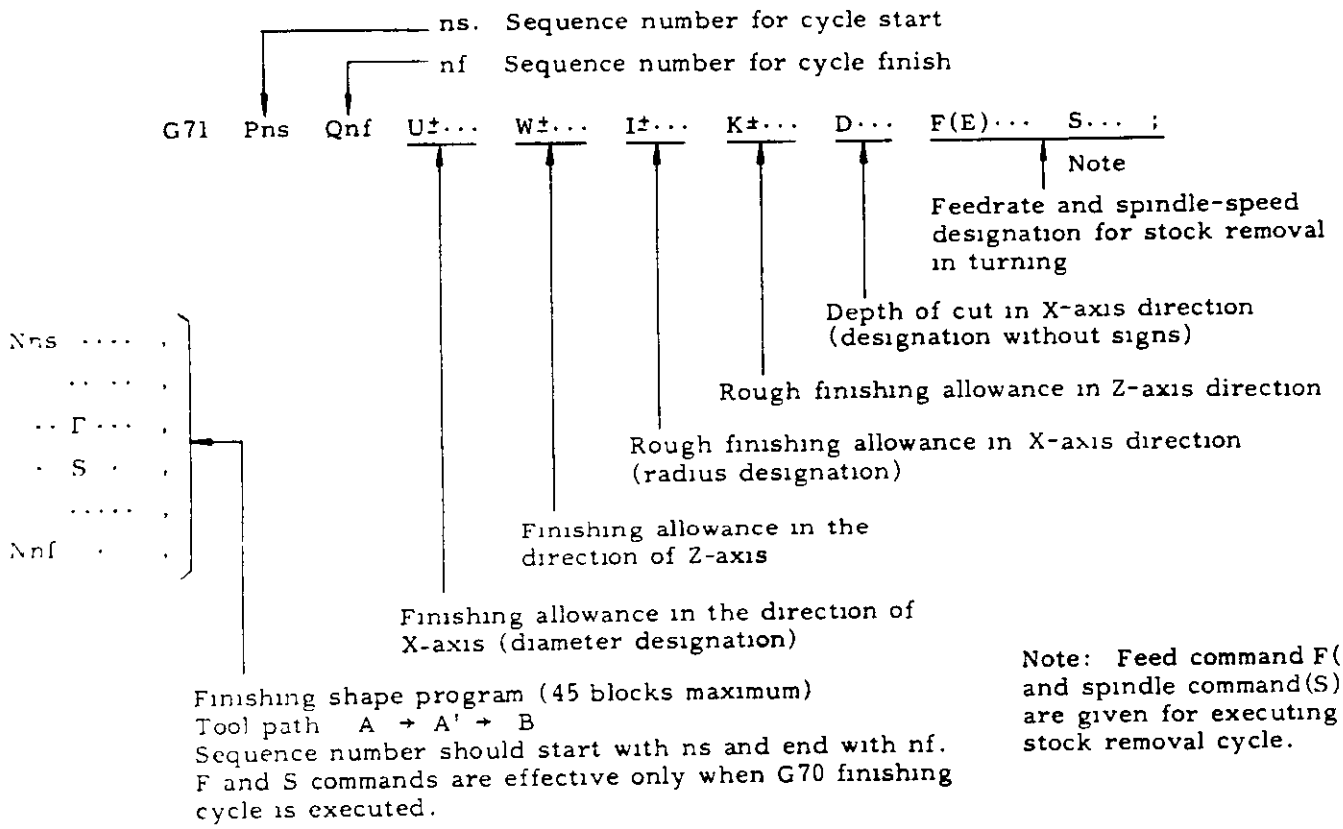
(5) It is possible to perform tool nose radius compensation on the cycle of G70 through G73.

(6) Tool nose radius compensation cannot be performed on the cycle of G74 through G76. Any attempt to do so will result in an error.

2.8.26.2 Stock Removal in Turning (G71)

Stock removal in turning with the finishing allowance remained uncut can be commanded by G71. Commands for finished contour are different between monotonous increase/decrease and concaved shaped path.

(1) Monotonous increase/decrease finishing shape



Note: Feed command F(E) and spindle command(S) are given for executing stock removal cycle.

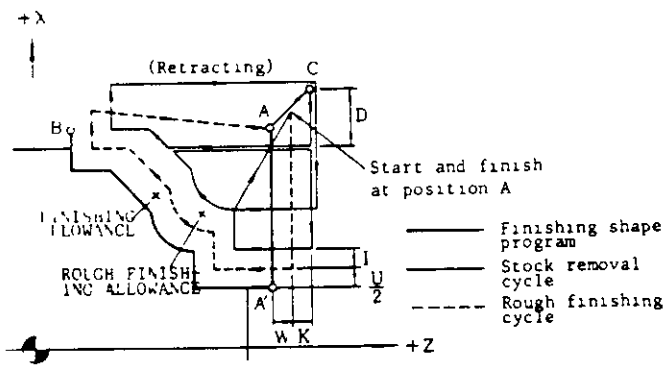


Fig. 2.42

G71 starts at point A, executes rough finish cycle (——) and the rough finishing cycle (---), and returns to point A to be terminated

In the case of I = 0 and K = 0 (or no designation), the rough finishing cycle is omitted

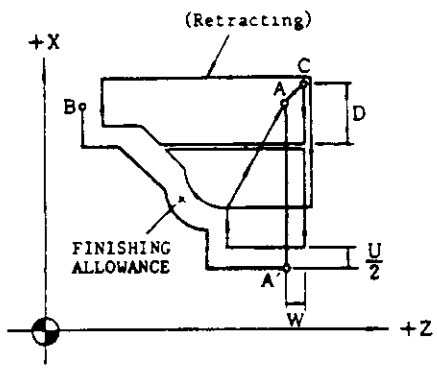


Fig. 2.43

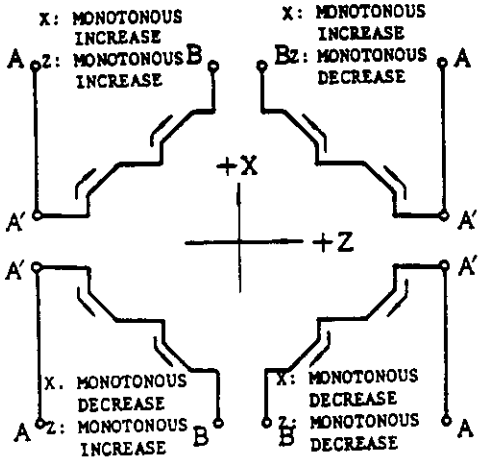
Retracting is performed by rapid traverse G00. Thrust motion depends on the speed (G00 or G01) specified by the program of AA'.

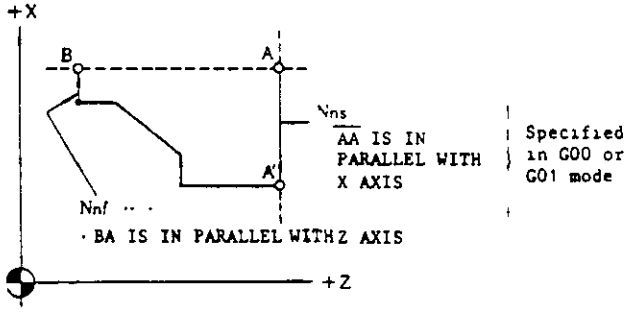
Each depth of cut D along X-axis can be overrided by 10% step within the range of 0 to 200%, by G71/G72 cut depth override selection or setting. See item (3), g.

NOTES:

1. The tool path of finishing shape should be programmed to be monotonous increase or decrease in X and Z coordinates.

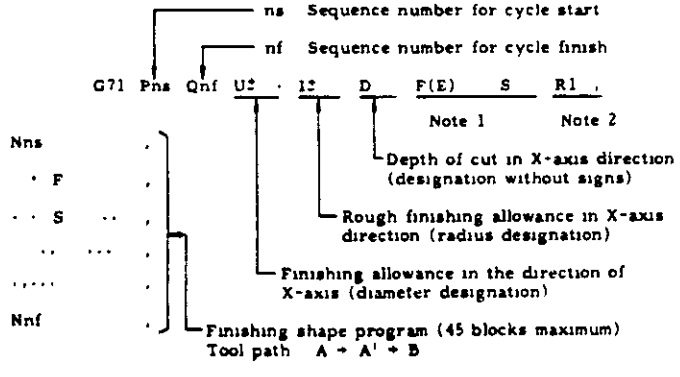
2. The following should be taken into consideration in programming the start block (Nns) and the end block (Nnf) of a finishing shape program.





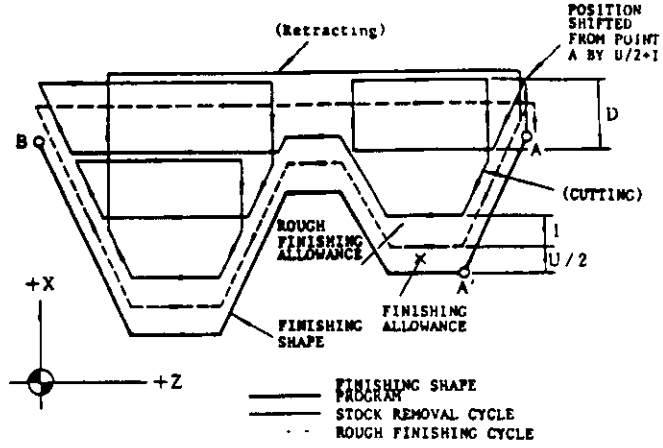
(2) Concaved finishing contour

a. Command format



Note 1: Specifies the feed command (F(E)) and spindle command (S) for the execution of stock removal cycle.

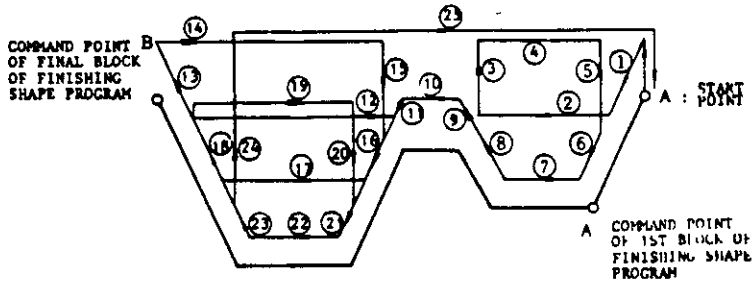
Note 2: Computes the cutting path for the concaved finishing shape program if R1 is specified.



G71 starts at point A, executes the rough cutting cycle (——) and the rough finishing cycle (---), and returns to point A to be terminated. If I is not specified, the rough finishing cycle is skipped.
Retracting is performed by rapid traverse G00. Thrust motion depends on the speed (G00 or G01) specified by the program of AA'. Each depth of cut D along X-axis can be overrided by 10% step within the range of 0 to 200%, by G71/G72 cut depth override selection or setting. See item (3), g.

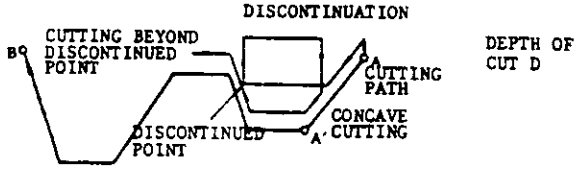
b. Cautions for concaved finishing shape program

(i) Rough cutting cycle by G71 starts from the closest concave to the start point.

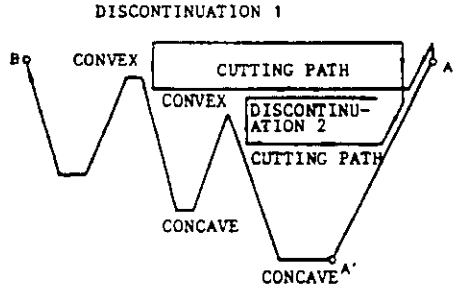


2.8.26.2 Stock Removal in Turning (G71) CONT'D)

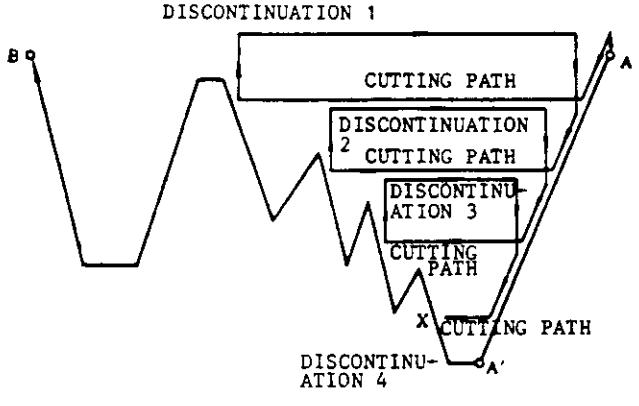
Since cutting starts with the concave nearest the start point, the cutting path is interrupted if it hits the convex beyond the concave. The concave is cut to its bottom. Then, the cutting cycle returns to the interrupted point to perform cutting beyond it.



(ii) For a simple concave, only one interrupted point is provided. However, for a complex concave containing a smaller concave and a convex as shown below, the cutting path is interrupted first at the larger convex then at the smaller convex.



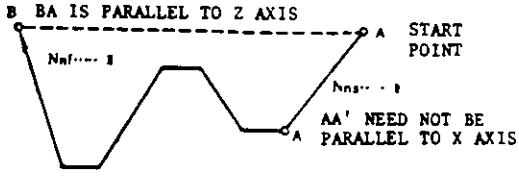
The maximum number of interrupted points that allows cutting is three, beyond which cutting is disabled, causing "097" error. Within this limitation, any number of concaves is allowed.



Sample Contour Not Allowing Cutting

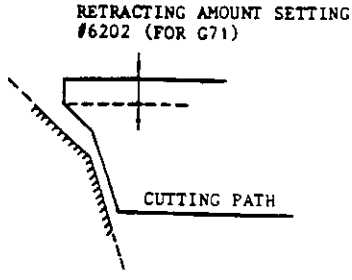
(iii) Any contour having an overhang does not allow cutting. Hence, the Z-axis specification value of the finishing shape program should be a monotonous variation.

(iv) The termination block for the finishing shape program has the following limitations:



For the G command of the termination block (Nnf... ;), specify G01 or G00, in principal.

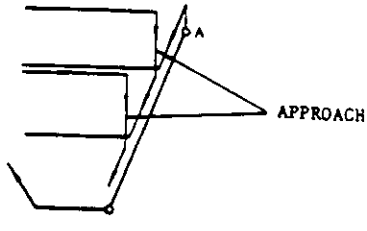
(v) The retracting amount after each approach cycle may be set by the setting.



(vi) Each block of the finishing shape program should be of monotonous increase or monotonous decrease. A circular arc which extends over two or more quadrants must be divided into two blocks before being programmed.

(vii) Generally, Z-axis finishing allowances W and K are not specified. Otherwise, a bite is caused into the wall of the corresponding side. If "R1" is not specified, the conventional monotonous increase/monotonous decrease stock removal cycle is provided.

(viii) Approach is performed at the feedrate. It is not affected by the G code of the finishing shape program. Hence, depending on the finishing shape program, positioning may be performed by rapid traverse after the approach at feedrate.



3) Rules in programming G71

a. Addresses U, W, I and K must be programmed with signs. If a wrong sign is programmed, the workpiece may be gouged. An address D for depth of cut must be programmed without signs.

b. Finishing shape program must be programmed immediately after the block containing G71. Even a block between them is ignored.

c. When F and S codes are not specified in the block containing G71, F and S codes specified in the preceding block are effective for G71 mode.
F and S codes specified in the program of finishing shape become effective only for G70 mode and are disregarded in G71 mode.

d. The following should be taken into consideration in programming the start block (Nns) and the end block (Nnf) of a finishing shape program.

Usable G code	Remarks
G00, G01, G03, G06, G22, G23	
G11, G12	A block containing these codes must be counted as two blocks.
G111	A block containing these codes must be counted as four blocks.
G112	A block containing these codes must be counted as five blocks.

e. When a program has entered the tool nose radius compensation mode before the G71 is commanded, the compensation is effective for the G71 cycle.

However, the compensation is executed not in the stock removal cycle but in the rough finishing cycle.
Thus, the compensation is ineffective for the program in which the rough finishing cycle is omitted. (I = 0, K = 0)

f. The above rules and cautions in programming G71 also apply to G72 cycle. In other words, the G72 cycle is the same as G71 except that cutting is made in parallel with X-axis.

g. Cut depth override of G71 and G72

Ten percent step override may be applied, within the range of 0 to 200%, to the depth of cut D of each time in the following two manners:

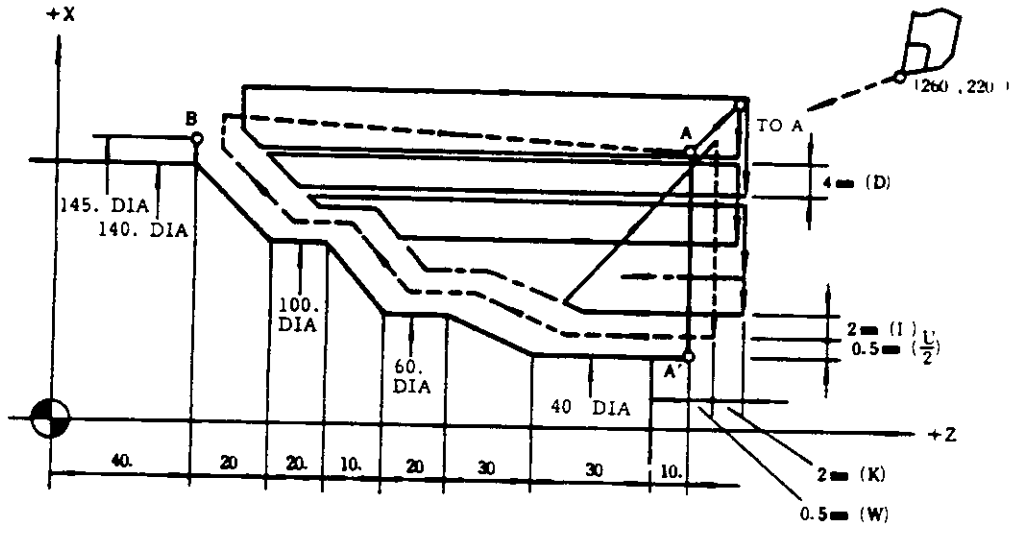
(i) By setting #6004D3 through D7. (Set with 5-bit code.)

(ii) By G71/G72 cut depth override switching. Either of the above methods shown in (i) or (ii) may be selected by parameter #6023D2.

Parameter	Function
#6023D2 = 0	Setting in (i)
#6023D2 = 0	Setting in (ii)

2.8.26.2 Stock Removal in Turning (G71) (CONT'D)

EXAMPLE A: Tool nose compensation applied to finishing shape without concaves




```
N1  G50  X260.  Z220.  ;
N2  G00  S1000  M03  T0101  ;
N3  G41  ,
N4  X145.  Z180.  ,
```

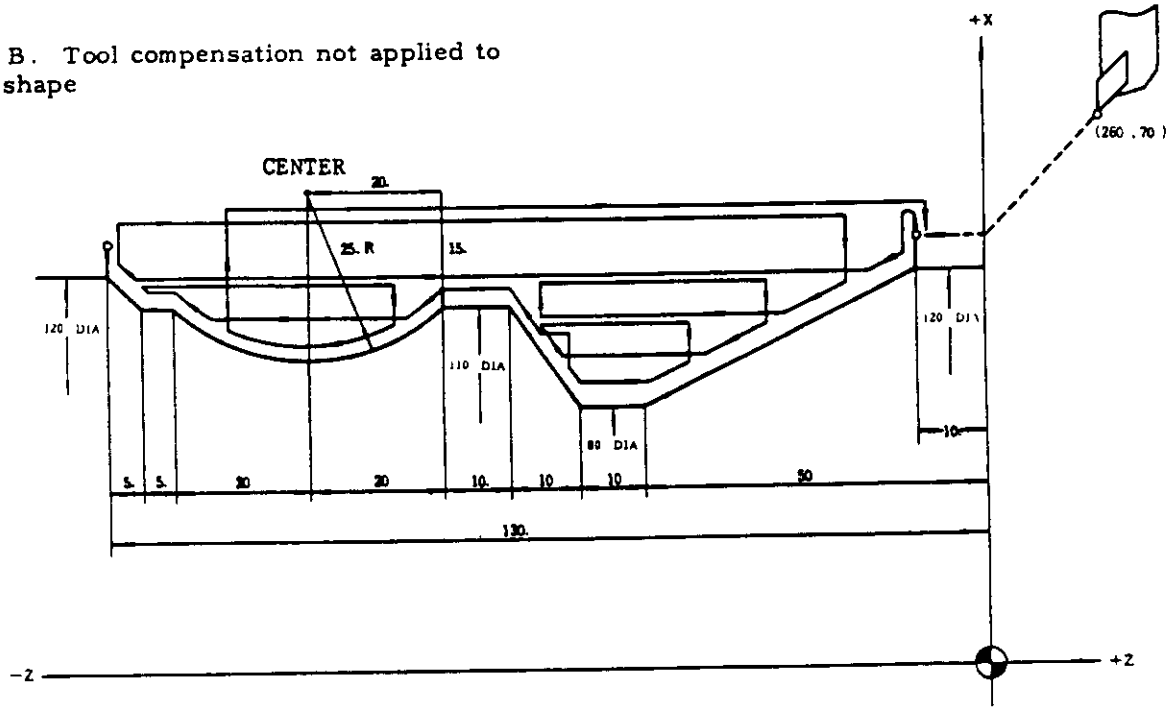
Stock removal in turning

```
N5  G71  P6  Q13  U1.  W0.5  I2.  K2.  D4.  F0.3  S800  ;
N6  G00  X40  S800  ;  ···  Rapid traverse
                           cutting
N7  G01  W-40.  F0.15  ;
N8  X60  W-30  S600  ,
N9  G12  W-20  I5  ,  ···  Equivalent to
                           2 blocks
N10  G01  X100.  W-10.  S300  ,
N11  W-20.  ,
N12  X140.  W-20.  S200  ;
N13  X145.  ,
```

Finishing shape = 9 blocks

```
N14  G41
N15  G00  X260.  Z220.  T0100  ,
```

EXAMPLE B. Tool compensation not applied to concaved shape




```
N01  G50 X260. Z70. ;
N02  G00 S500 M03 T0101 ;
N03  X124. Z-10. ;                    ← Stock removal in facing
N04  G71 P5 Q14 U2. D6. F0.2 S250 ;
N05  G01 X120. ;
N06  X80. Z-50. F0.1 S500 ;
N07  W-10. ;
N08  X-110. W-10. ;                   } Finishing shape
N09  W-10. ;
N10  G02 X95. W-20. I15. K-20. ;
N11  X110. W-20. I25. ;
N12  G01 W-5. ;
N13  X120. W-5. ;
N14  X124. ;
N15  G00 X260. Z70. T0100 ;
N16  T0202 ;
N17  G50 X255. Z70. ;
N18  X124. Z-10. ;
N19  G70 P5 Q14 ;                     ← Executes finishing cycle of G71 shown above.
```

2.8.26.3 Stock Removal in Facing (G72)

This cycle provides stock removal and rough finishing in facing with the finishing allowance remaining.

G71 is for cutting in parallel with Z axis and G72 is for cutting in parallel with X axis.

(1) Monotonous increase/decrease finishing shape

a. Command format

G72 Pns Qnf U±... W±... I±... K±... D... F(E)... S... ;

Nns ..... ;
..... ;
F ..... ,
S ..... ,
..... ,
Nnf ..... ,

Depth of cut in Z axis direction (without signs)

Significance of each address is the same as that of G71 command.

Finishing shape program (45 blocks maximum)
Tool path. A → A' → B

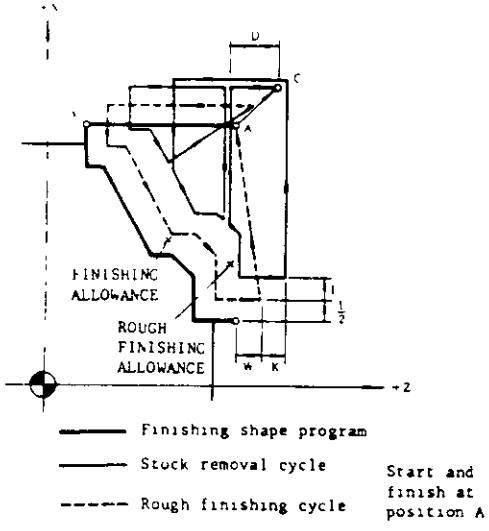


Fig. 2.44

Cycle starts at point A executes stock removal cycle and rough finishing cycle, returns to A when completed.

In case of I = 0 and K = 0 (or no designation), the rough finishing cycle is omitted

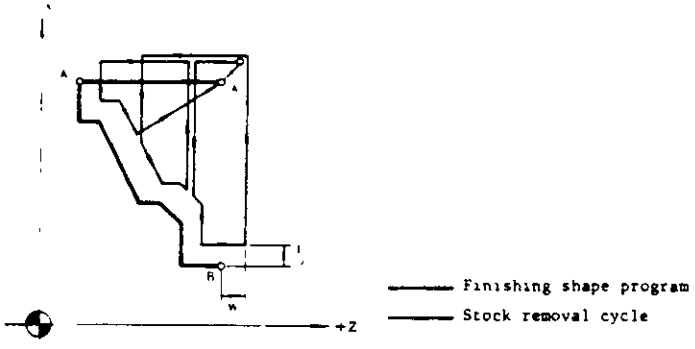
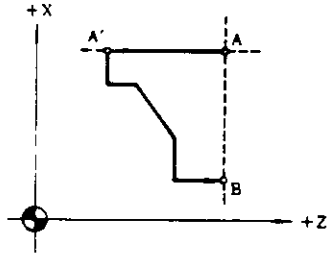


Fig. 2.45

Retracting is performed by rapid traverse G00. Thrust motion depends on the speed (G00 or G01) specified by the program of AA'. Each depth of cut D along Z-axis can be overrided by 10% step within the range of 0 to 200%, by G71/G72 cut depth override selection or setting.

b. Rules and cautions in programming finish shape

G72 is the same as G71 except that the tool cuts into the workpiece in parallel with Z axis. Refer to 2.8.26.2 Stock Removal in Turning.

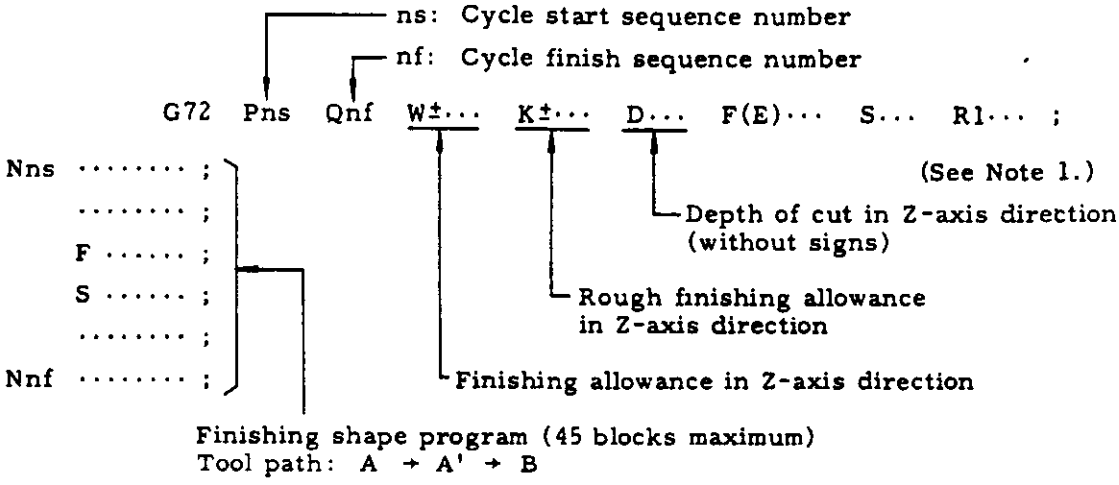


Nns ··· ;: AA' IS IN PARALLEL WITH Z AXIS.
Nnf ··· ;: BA IS IN PARALLEL X AXIS.

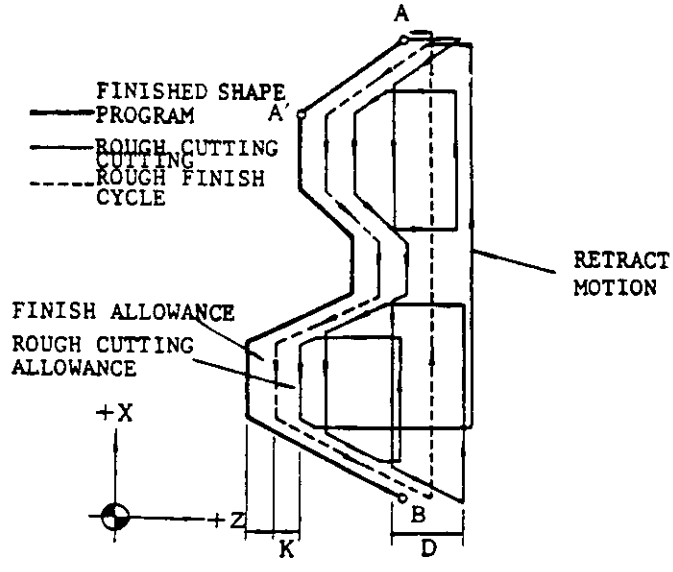
} Specified in G00 or G01 mode

Fig. 2.46

(2) Concaved finishing shape



Note 1· If "R1" is specified, cutting path is computed for the concaved finishing shape program.



G72 starts at point A, performs stock removal cycle (——) and rough finishing cycle (---), and returns to point A when completed. If K is not specified, rough finishing cycle is skipped. Retracting is performed by rapid traverse G00. Cutting rate depends on the speed (G00 or G01) specified by the program of AA'.
Each depth of cut D along X-axis can be overrided by 10% step within the range of 0 to 200%, by G71/G72 cut depth override selection or setting.

b. Cautions of concaved finishing shape program

G72 only performs the operations which are parallel to Z axis. The same cautions as with G71 are applied to G72 except that the retracting amount of G72 may be set by setting #6203.

(3) Cautions for G72 command

The same cautions as with G71 are applied to G72.

EXAMPLE: In case of I = K = 0, tool nose radius compensation is not applied.

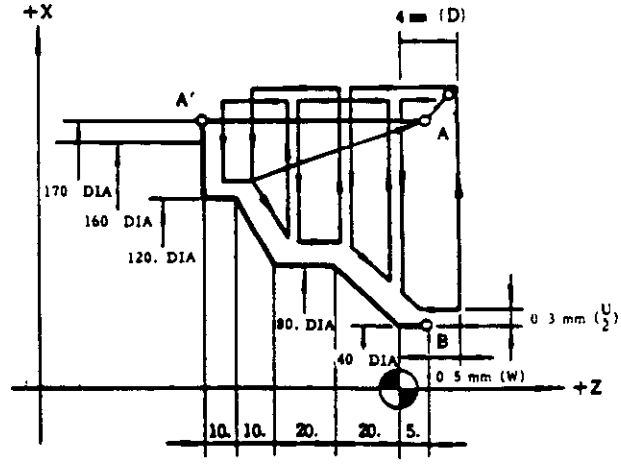


Fig. 2.8.26.19

2.8.26.3 Stock Removal in Facing (G72) (CONT'D)

```
N1  G50 X260  Z60.  ;
N2  G00 S1000 M03 T0202 ;
N3  X170. Z5.  ;                    ┌── Stock removal in facing
N4  G72 P5 O11 U0.6 W0.5 I0 K0 D4.0 F0.3 S200 ;
N5  G01 Z-60. F0.15 ;  ····· Cutting at feed
N6  X120. S250 ;
N7  Z-50.  ;
N8  X80. Z-40. S400 ;
N9  Z-20.  ;                        Finishing shape program
N10 X40. Z0 S800 ;
N11 Z5.  ;
N12 G00 X260. Z60.
N13 T0303 ;
N14 X170. Z5.  ,
N15 G70 P5 O11 ;  ····· Executes finishing cycle
```

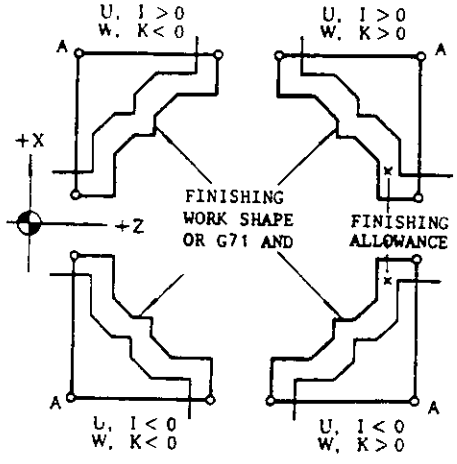
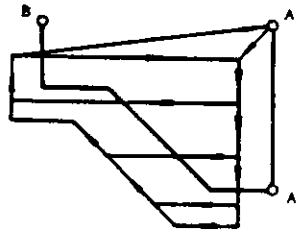


Fig. 2.47 Relation between Finishing Shape Program and Signs of Addresses U, W, I and K



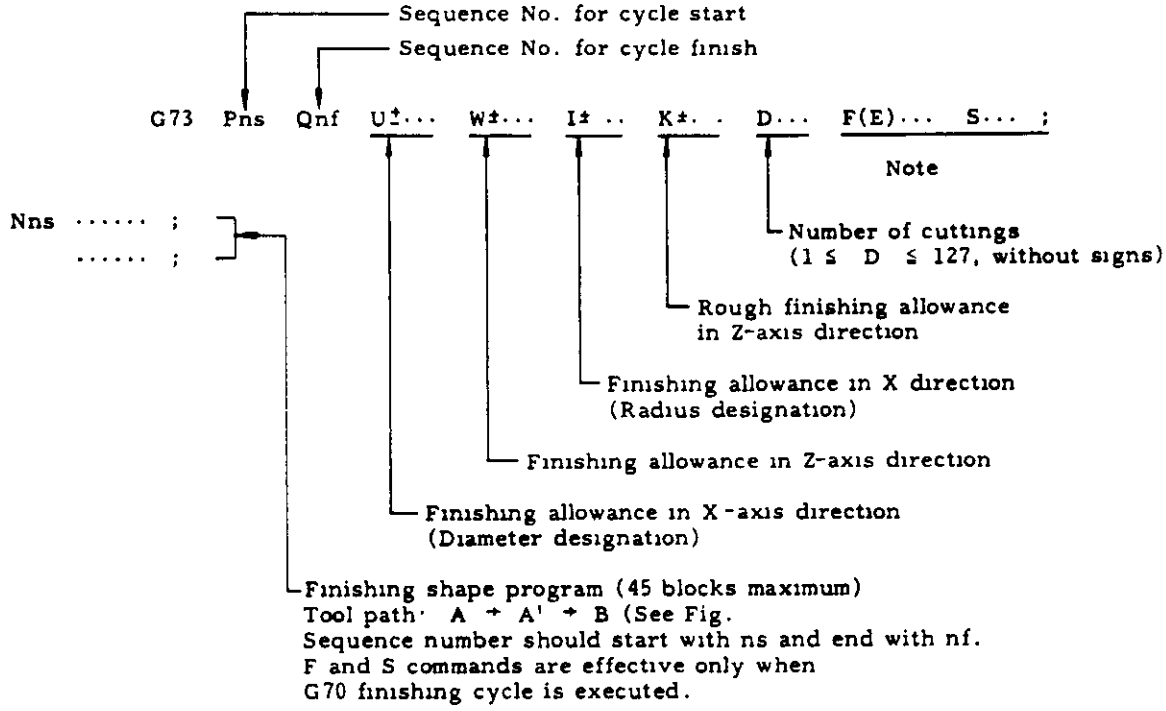
In the case that U, W, I and K < 0 are erroneously programmed.

A wrong sign will cause a grouging of the workpiece as shown below.

2.8.26.4 Pattern Repeating (G73)

This cycle is useful for cutting the workpiece such as moldings and forgings whose cutting shapes are roughly made beforehand.

(1) Command format



Note: Feed command (F(E)) and spindle speed command(S) are given for executing closed loop cutting cycle.

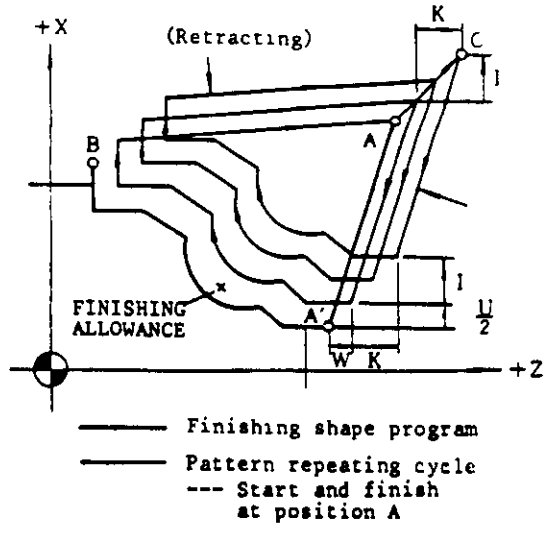


Fig. 2.48

(2) Rules in programming G73

a. Address U, W, I and K must be programmed with signs.

b. Address D for number of cuttings must be programmed without signs, obeying the following restriction.

$$1 \leq D \leq 127$$

D command out of the above range causes data error. (Alarm code "096")
When D is 1, the cutting of I and K values is completed in a single cycle remaining finishing allowance.

c. Finishing shape should be programmed immediately after the block containing G73.

d. The start (Nns) and end (Nnf) block of a finishing shape cycle must be programmed with G00 or G01.
But these 2 blocks need not be parallel with X or Z axis.

e. Finishing shape program does not need to be repetitive monotonous increase or decrease in X or Z coordinate.

NOTES.

1. When F(E) and S functions are not specified in the block containing G73, the F (E) and S functions specified in the preceding blocks are effective in the pattern repeating cycle. F(E) and S functions specified in the program of finishing work shape are effective in finishing cycle G70 and ignored in pattern repeating cycle.
2. Table below shows the G codes which can be specified in the program excluding the blocks of Nns and Nnf.

Usable G codes	Remarks
G01, G06, G02, G03, G22, G23	
G11, G12	Counted as 2 blocks
G111	Counted as 4 blocks
G112	Counted as 5 blocks

3. When I and K are 0 or not designated, it causes input error. (Alarm code "096" is displayed)
4. Δ I and Δ K (rough cutting allowance per cycle) are calculated as follows.

$$\Delta I = \frac{I}{D-1}, \quad \Delta K = \frac{K}{D-1}$$

where $D \geq 2$

Note that the control ignores the value below 0.001 millimeter. As a rule, the program should be made so that Δ I and Δ K are not smaller than 0.001 millimeter.

Processing of Δ I and Δ K

EXAMPLE 1

In case of I = 0.005 mm, K = 0.005 mm, D = 7

$$\Delta I = \frac{0.005}{6} = 0 \quad \Delta I = \frac{0.005}{6} = 0$$

} Input error occurs.

EXAMPLE 2

In case of I = 0.01 mm, K = 0.01 mm, D = 7

$$\Delta I = \frac{0.01}{6} = 0.001 \text{ mm}$$

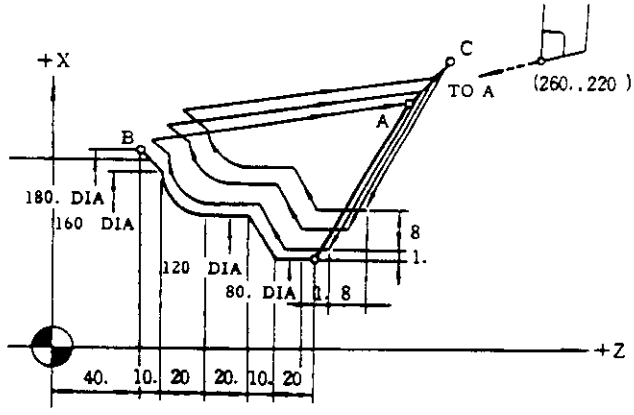
$$\Delta K = \frac{0.01}{6} = 0.001 \text{ mm}$$

Therefore, the cutting allowance of each cycle is as follows.

1st to 6th cycle ··· Δ I = Δ K = 0.001 mm
7th cycle ····· Δ I = Δ K = 0.004 mm

(6) When the program has entered the tool nose radius compensation mode before G73 is commanded, the compensation is effective for all cycles of G73.

EXAMPLE




```
N10 G50 X260. Z220 .
N11 G00 S300 M03 T0303 .
N12 X220 Z160 .                          ┌ Pattern repeating
N13 G73 P14 O19 U2. W1 I8 K8 D3 F0 3 S200 .
N14 G00 X80 W-40 S400 .
N15 G01 W-20 F0 15
N16 X120 W-10 S300 .                     Finishing shape program
N17 W-20 .
N18 G22 X160 W-20 R20. S200 .
N19 G01 X180 W-10 .
N20 G00 X260 Z220 .
```

2.8.26.5 Finishing Cycle (G70)

After rough cutting of G71, G72 and G73, the finishing cutting can be made by the commands following G70.

(1) G70 Pns Qnf ;

- nf: Sequence number for cycle finish
- ns: Sequence number for cycle start

This command permits the execution of the finishing shape program in G71, G72 or G73 which is commanded previously.

F(E) and S functions specified in the finishing shape program are effective in the finishing cycle.

F(E) and S functions for rough cutting specified in the block containing G71, G72 or G73 are ignored in the finishing cycle.

NOTE: The internal memory for storing the finishing shape program:
To shorten the stock removal cycle time, the finishing shape program is binary-converted and then is stored in the memory (one) pair) in the unit. This memory is called the internal memory for finishing shape program.

(2) G70 does not need to be commanded immediately after the block of G71, G72 or G73. Necessary information such as tool change from a rough cutting cutter to a finishing cutter can be inserted between them.

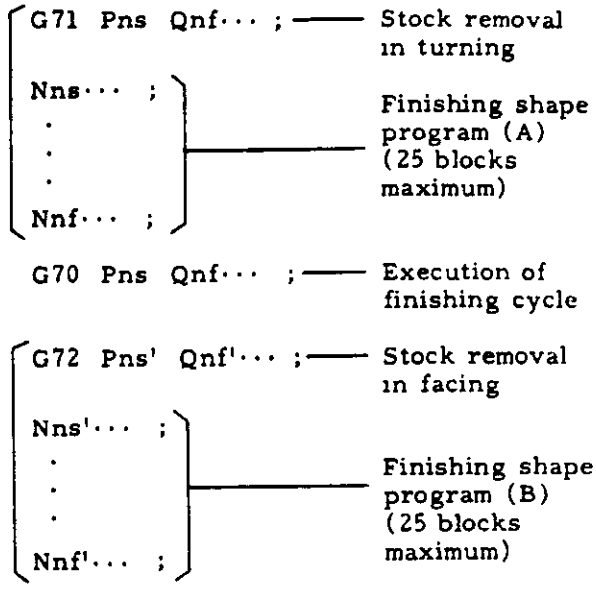
However, the following command and operation should not be programmed between them.

Inhibited command and operation	Result
M02 and M30 commands with internal reset	Finishing shape program in the memory are eliminated.
Reset operation	

(3) Storage and search of the finishing shape program

The processing of the finishing shape program is different in tape operation mode and memory operation mode.

a. Operation in Tape Mode



After executing the above program the program (A) is eliminated and the program (B) is retained in the finishing shape memory. Therefore, the finishing command with G70 in the trailing program is effective for the finishing shape program (B).
If the sequence number specified by G70 does not match the sequence number in the memory for finishing shape program, an error ("091") is caused.

b. Operation in memory (MEM) mode

When the sequence number specified by G70 matches the sequence number in the memory of finishing shape program, the finishing cycle is normally executed. If they do not match, a search for the specified finishing shape program from the part programs is performed. Then, the program is stored in the memory and is executed. This is called the finishing shape program search feature. This feature enables, only in memory mode, the programming in which the stock removal cycle (or closed loop cutting cycle) is performed two or more times as shown below and then the finishing cycle is performed for each stock removal cycle:

Stock removal cycle (A)
↓
Stock removal cycle (B)
↓
Finishing cycle (A)
↓
Finishing cycle (B)

2.8.26.5 Finishing Cycle (G70) (CONT'D)

Supplement

(i) The use of the finishing shape program feature increases the cycle time as follows:

- Example of finishing shape program search time:

  When the machining tape is 5 m (2000 characters) long and the finishing shape program is stored in the middle of the tape, the cycle time is about 100 msec.

- Example of the execution time for storing the searched finishing shape program in internal memory:

  When the size of the finishing shape program is 30 blocks, the execution time is about 1 sec. However, the storing of the searched program in internal memory is performed in the block which precedes the block of G70 command. Hence, the cycle time is increased by the amount obtained by subtracting the execution time of the preceding block from the above time (about 1 sec).

(ii) This search feature is executed only in the part program having the program number for which G70 is specified.

NOTES:

1. For the sequence number ns for cycle start and nf for cycle finish, the following case causes input error.

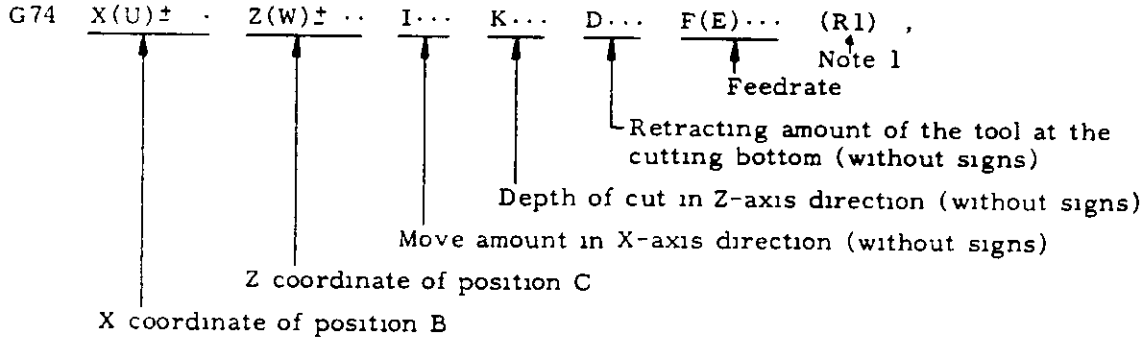
When the sequence numbers ns and nf of G70 are not commanded in the finishing shape program (Tape operation)
When the sequence number ns of G70 is commanded in advance of sequence number ns, or ns = nf.

2. When the program has entered the tool nose radius compensation mode before G70 is commanded, the compensation is effective for G70.

2.8.26.6 Peck Drilling in Z-axis (G74)

This command permits the operation of peck drilling with pecking motion in parallel with Z axis.

(1) Command format



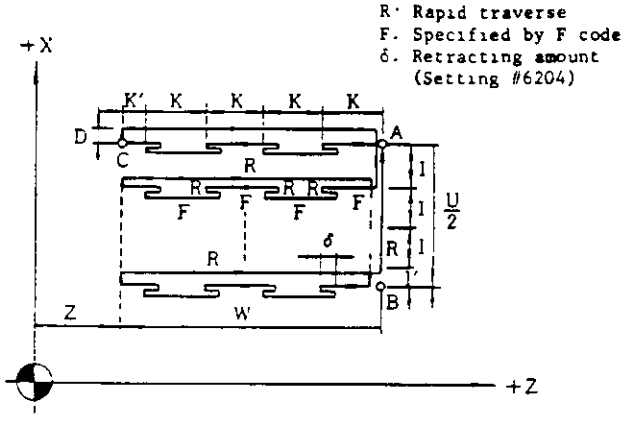


Fig. 2.49

The cycle starts and finishes at position A. The fixed amount of pecking is set by the parameter No. 74.

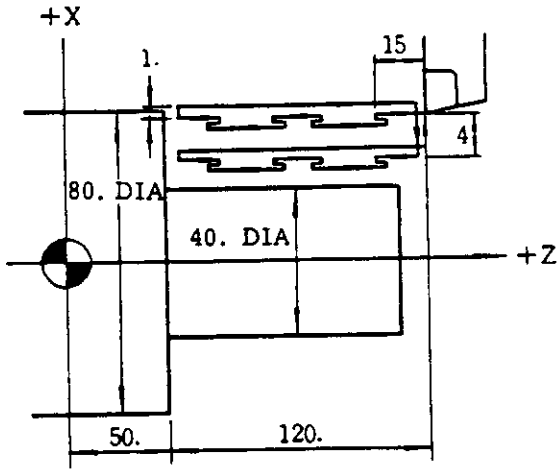
Note 1: The above diagram is of the cause where "R1" is not specified. When "R1" is specified, the retracting amount (δ) for each approach is ignored and the tool returns to the approach start point, namely point A in Z-axis.
This cycle starts at point A and ends at it.

NOTES:

1. Addresses I, K and D must be programmed without signs.
2. When the command of I $>|U/2|$ is issued, the cycle finishes after the finish of the pecking motion from position B following the pecking motion from position A.
3. When the command of K $>|W|$ is issued, the cutting is made at once to the cutting bottom without the pecking motion.
4. When D is programmed as 0 or D is not programmed, the retracting motion is not made at the cutting bottom.
5. The final cutting amount in the Z direction K' and the final move amount in the X direction I' are automatically calculated.
6. If X(U) , I or D is omitted, only one-cycle operation is made in the direction of Z-axis, which is used for drilling.
7. When the contents of setting #6204 are set to 0, the cutting is made at once to the cutting bottom without pecking motion.
8. The tool nose radius compensation is ineffective for G74 and G75.

EXAMPLE

G74 X40. Z50. I4. K15. D1. F0. 25 ;



2.8.26.7 Grooving in X-Axis (G75)

(1) This command permits the operation of peck drilling with pecking motion in parallel with X-axis.

a. Command format

G75 X(U)±... Z(W)±... I... K... D... F(E)... (R1) ;

Note 1

- F(E)...: Feedrate
- D...: Retracting amount of the tool at the cutting bottom (without signs)
- K...: Move amount in Z-axis direction (without signs)
- I...: Cutting depth in X-axis direction (without signs)
- Z(W)±...: Z coordinate of position B
- X(U)±...: X coordinate of position C

2.8.26.7 Grooving in X-Axis (G75) (CONT'D)

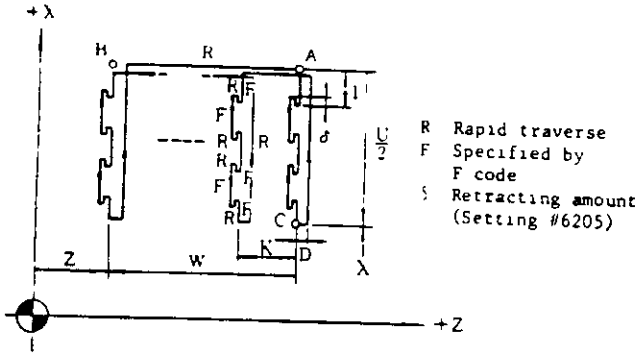


Fig. 2.50

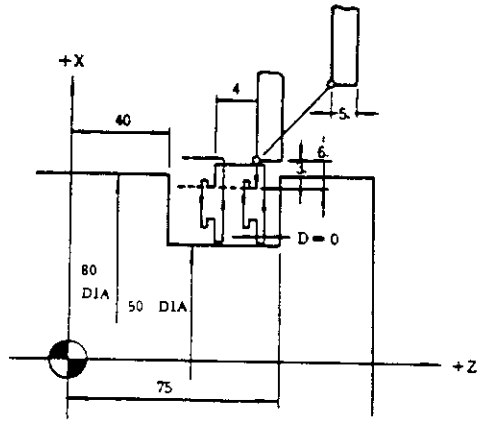
Note 1: The above diagram is of the case where "R1" is not specified. When "R1" is specified, the retractingamount (δ) for each approach is ignored and the tool returns to the approach start point, namely point A in X-axis.
This cycle starts at point A and ends at it.

NOTES G74 permits the cutting in the direction of Z-axis, and G75 in the direction of X-axis. Therefore, the cautions in programming G75 is the same as those of G74. Refer to 2.8.26.6 Peck Drilling in Z-axis (G74).

EXAMPLE.

N1 G00 X86. Z70. ,

N2 G75 X50. Z40. I6. K4. (D0) F0.2 ,



2.8.26.8 Automatic Threading Cycle (G76)

This cycle provides automatic cutting of straight and taper threadings along the angle of thread.

(1) Command format

G76 X(U)±... Z(W)±... I±... K... D... F(E)... A... ;

- X(U)±... : X coordinate of position C
- Z(W)±... : Z coordinate of position C
- I±... : Distance of tapered section (radius designation)
- K... : Height of thread (without signs)
- D... : First depth of cut (without signs)
- F(E)... : Lead of thread
- A... : Angle of thread (deg)

The sign of figure following the address I is decided by the direction of position B' viewed from position C.

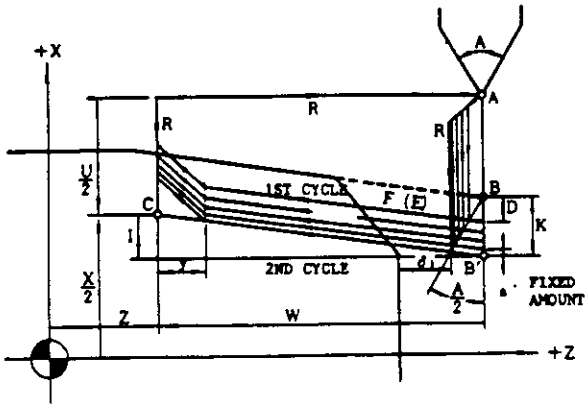


Fig. 2.51

The following shows the cutting position around point B (In case of taper thread). The G76 cycle starts and ends at point A.

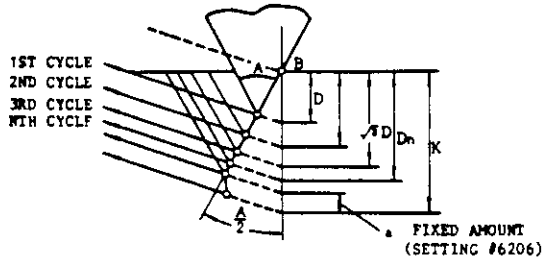


Fig. 2.52

The depth of cut in Nth cycle is:

$$Dn = \sqrt{n}\,D$$

The following six angles can be used as the command of thread angle.

$$A = 0°,\ 29°,\ 30°,\ 55°,\ 60°,\ 80°$$

Cutting in final cycle is made with the depth of fixed amount a, which is set by the setting #6206.

(2) Straight thread

When the address I is 0 or not designated, a straight thread is cut as shown below.

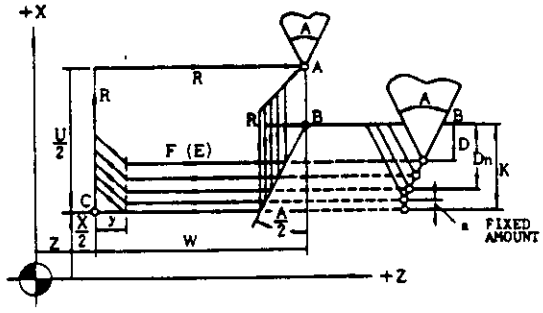


Fig. 2.53

(3) Rapid threading pull-out

If "Rapid threading pull-out input (CDZ)" is on when G76 is specified, Rapid threading pull-out is performed. (γ) may be set to parameter #6080, in the range of 0 to 25.5L and in increments of 0.1L, where L is the lead of the specified thread.

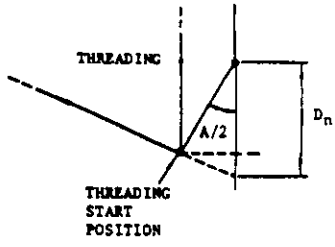
NOTES:

1. The depth of cut D in the first cycle is restricted by height of thread K as follows.

$$\frac{1}{6}K \leq D < K$$

Addresses K and D must be programmed without signs.

2. When taper threading is commanded with an effective angle, except 0, X coordinate of threading start position does not extend to the depth of cut.



3. If the thread angle other than the above listed (0°, 29°, 30°, 55°, 60°, 80°) is arbitrarily commanded, the next larger angle is selected.

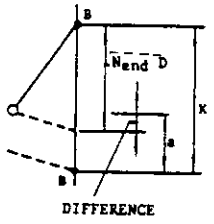
EXAMPLE

(Command) A15 ⟶ (Execution) A29

When A > 80°, A80 is executed.

2.8.26.8 Automatic Threading Cycle (G76) (Cont'd)

4. When the depth of cut in the final cycle along the thread angle ($\sqrt{n}$end D) is not met with (K - a), the difference between K and a is deducted from D. The depth of cut in first cycle never becomes larger than D.



EXAMPLE

In case of D = 5.0 mm, K = 9.8 mm, a (Fixed amount) = 0.2 mm

$\sqrt{n}$end D = $\sqrt{4}$ x 5.000 = 10.000 mm

Difference = $\sqrt{n}$end D - (K - a)
= 10.000 - (9.800 - 0.200)
= 0.400 > 0

Thus, the depth of cut in each cycle is as follows.

First cycle ····· 5.000 - 0.400
= 4.600 mm

Second cycle ··· $\sqrt{2}$ x 5.000 - 0.400
= 6.671 mm

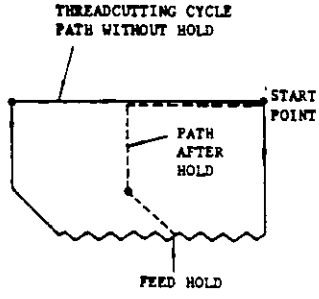
Third cycle ··· $\sqrt{3}$ x 5.000 - 0.400
= 8.260 mm

Fourth cycle · · $\sqrt{4}$ x 5.000 - 0.400
= 9 600 mm

Fifth cycle · ····· 9.600 + 0.200 (a)
= 9.800 mm

5. Thread cutting hold (option)

When this option is used, depressing the FEED HOLD during thread cutting causes the tool to perform threading up immediately and then return to start point A. When parameter #6019D7 = 1, the tool stops at position B where Rapid threading pull-out is terminated. Then, the tool returns to point A upon depression of CYCLE START.



If the thread cutting hold option is not selected, depressing of the FEED HOLD during thread cutting does not hold the operation, which stops at point C where the retracting operation is completed.

(4) Tool nose radius compensation† is not available for G76.

(5) G code of 01 group must be newly programmed in the next block of G76 cycle.

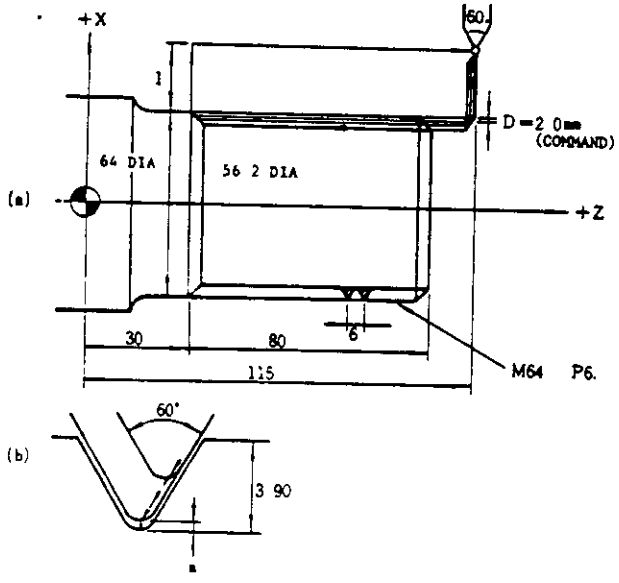
EXAMPLE

G76··· ;

G00 M30 ,

EXAMPLE:

```
      .
      .
G00  X66.  Z115.  ;
G76  X56.2  Z30.  K3.9  D2.  F6.  A60  ;
G00  ...
```



Depth of cut for each cycle when a (fixed amount) is 0.2 mm

1st cycle	——	1.700 mm
2nd cycle	——	2.528 mm
3rd cycle	——	3.164 mm
4th cycle	——	3.700 mm
5th cycle	——	3.900 mm

Though D200 (2.0 mm) is programmed, the actual depth of cut becomes 1.7 mm by the calculation of $\sqrt{n_{end}}$ D because of the difference between $\sqrt{n_{end}}$ D and (K - a).

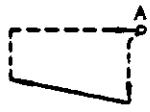
2.8.26.9 Precautions in Programming G70 through G76

Prohibition of MDI mode

- Operation in MDI mode cannot be made while multiple repetitive cycles (G70 through G76) are executed.
- Multiple repetitive cycles (G70 through G76) cannot be written-in through the operation in MDI mode.

Single block operation

- Executing G70 through G76 at SINGLE BLOCK switch ON brings the following results.

G70, G71, G72 G73, G74, G75	Program stops at every block.
G76	 Program stops at position A after each cycle is finished.

Symmetrical pattern

- The symmetrical four patterns can be commanded by each of G71 to G76.
- Signs of U, W and I should be properly specified in the finishing shape program for G71 to G73. (See Fig. 2.8.26.33)
- Command positionof (X, Z) or (U, W) with respect to position A should be properly specified for G74 to G76.

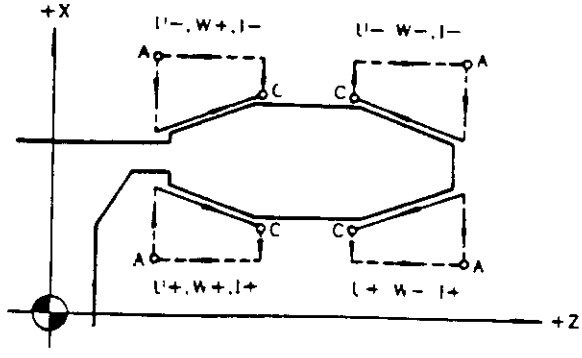
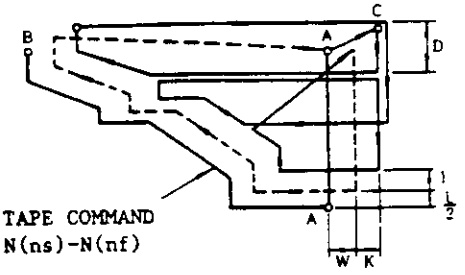
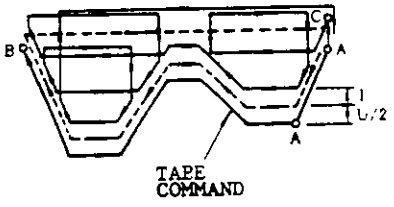
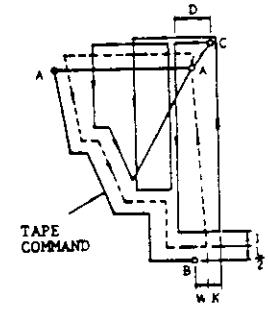
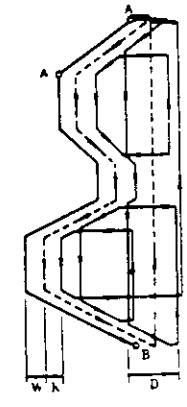


Fig. 2.54 Four Patterns of G76

Table 2.22 Multiple Repetitive Cycles

	Cutting Cycle	Command Format
G71 Stock Removal in Turning	 	(1) Monotonous increase/decrease path G71 Pns Qnf U ·· W· · I··· K··· D··· F(E)··· S··· , Nns ; · · · Nnf··· , } Finishing shape program (2) Concaved shape G71 Pns Qnf U ·· I · D · F(E)· · S··· R1 , Nns· · , · · Nnf·· , } Finishing shape program (U, W, I, and K must be programmed with signs.)
G72 Stock Removal in Facing	 	(1) Monotonous increase/decrease path G72 Pns Qnf U··· W··· I··· K ·· D· · F(E) ·· S· , Nns·· ; · · Nnf··· , } Finishing shape program (2) Concaved shape G72 Pns Qnf W··· K·· D ·· F(E)··· S··· R1 ; Nns··· , · · · Nnf··· , } Finishing shape program (U, W, I, and K must be programmed with signs.)

(3) Cautions in programming G90

a. T, S and M functions for G90 cycle must be specified beforehand in the preceding blocks. Specifying T, S and M functions requiring answer of FIN signal in the effective area of G90 causes error.

```
G90 X··· Z··· I··· F(E)··· ;
    X··· ;
    X··· ;
    T0505 M05 ;   <-- Error
G00 X··· Z··· ;
```

Effective area of G90

```
G90 X··· Z··· I··· F(E)··· ;
    X··· ;
    X··· ;
G00 T0505 M05 ;   <-- Correct
    X· · Z··· ;
```

Effective area of G90

The effective area of G90 is from the block containing G90 to the one before the block in which the other G code of 01 group is specified. This rule also applies to the G92 and G94 described later.

b. In Single Block mode, the execution of G90 cycle stops after the completion of the cycle ① to ④ .

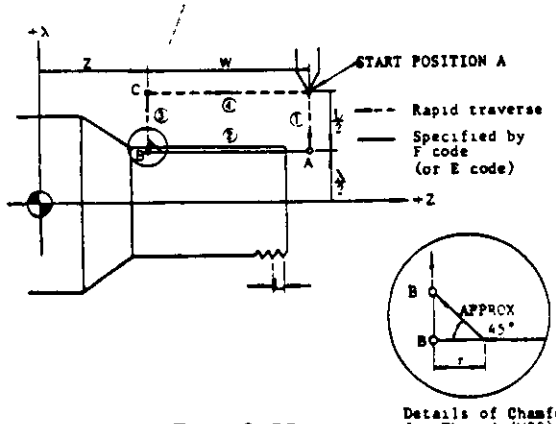
2.8.27.2 Threading Cycle (G92)

(1) Straight threading cycle

G92 X(U)··· Z(W)··· F(E)··· ,

F(E)··· : Lead designation (L)

The cycle ① to ④ shown below is executed by this command.



Details of Chamfering for Thread (M23)

Fig. 2.57

Rapid threading pull-out.

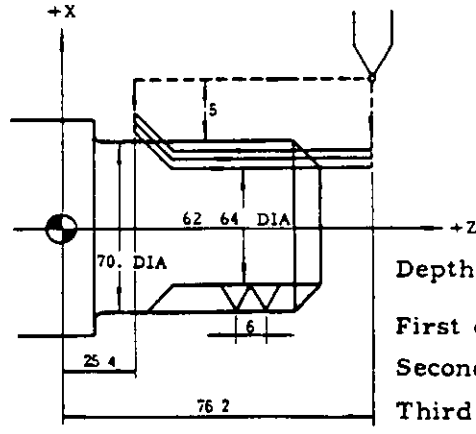
If "rapid threading pull-out input (CDZ)" is on when G92 is specified, rapid threading pull-out is performed. Rapid threading pull-out value (γ) may be set to parameter #6080, in the range of 0 to 25.5L and in increments of 0.1L, where L is the lead of the specified thread. Generally, it is convenient to create and use the sequence in which "rapid threading pull-out (CDZ)" is turned on/off through the use of a given M code.

Since G92 is modal, the cycle operation will be continued by specifying depth of cut in the direction of X-axis.

X(U)··· ;
X(U)··· ;

EXAMPLE:

```
N30 G00 X80. Z76.2 MOO ;--MOO: Rapid threading pull-out ON
N31 G92 X66.4 Z25.4 F6. ;
N32 X65. ;                  Threading cycle cutting four times
N33 X63.8 ;
N34 X62.64 ;
N35 G00 X100.Z100. MΔΔ ;--MΔΔ: Rapid threading pull-out OFF
```



Depth of cut

First cycle:	1.8 mm
Second cycle.	0.7 mm
Third cycle·	0.6 mm
Fourth cycle:	0.58 mm

(2) Straight thread cutting cycle with angle

G92 X(U)··· Z(W)··· K··· F(E)··· ;

This command permits the thread cutting along the angle of thread. The cycle ① to ④ shown below is executed.

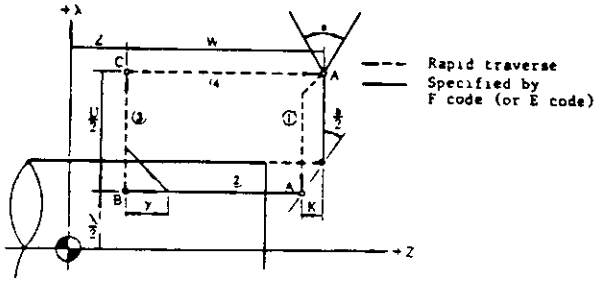


Fig. 2.58

The sign of figure following address K is decided by the direction of position A' viewed from position A.

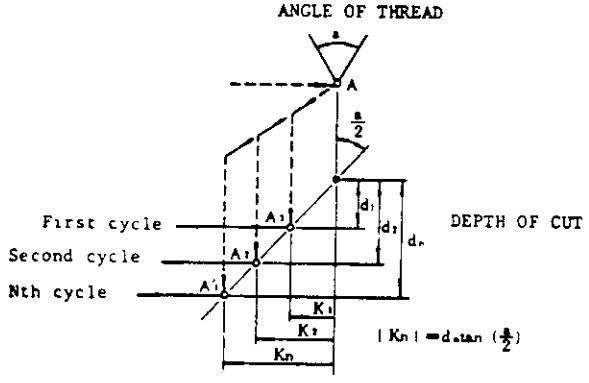
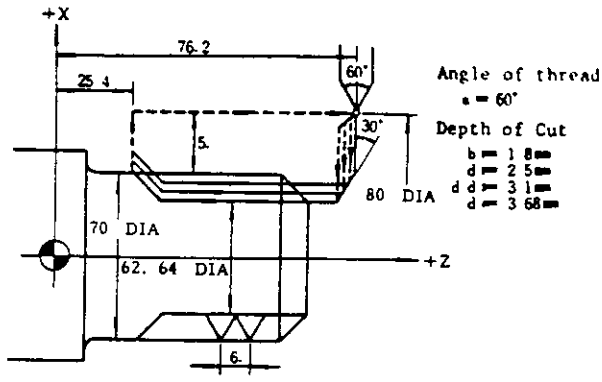


Fig. 2.59

For the threading along the angle of thread, the K for each cycle obtained from the above formula should be programmed.

a	tan (a/2)
29°	0.258618
30°	0.267949
55°	0.520567
60°	0.577350
80°	0.839100

EXAMPLE



Calculation of K··· K = d·tan(60°/2)

K_1 = -1.8 x 0.57735 = -0.866 mm

K_2 = -2.5 x 0.57735 = -1.443 mm

K_3 = -3.1 x 0.57735 = -1.790 mm

K_4 = -3.68 x 0.57735 = -2.125 mm

```
N40 G00 X80. Z76.2 M00 ;
N41 G92 X66.4 Z25.4 K-0.87 F6  ;
N42 X65. K-1.44 ;
N43 X63.8 K-1.79 ;
N44 X62.64 K-2.13 ;
N45 G00 X100. Z100. MΔΔ ;
```

(3) Taper threading cycle

G92 X(U)··· Z(W)··· I··· F(E)··· ;

The cycle ① to ④ shown below is executed by this command.

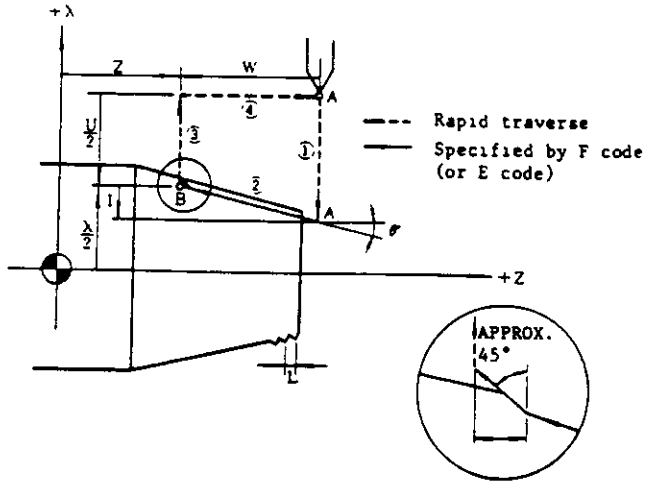


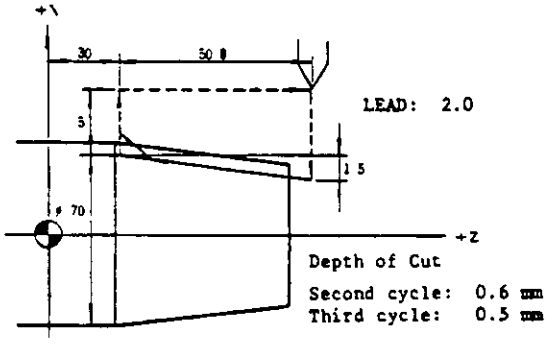
Fig. 2.60 Details of Rapid Threading Pull-out

The sign of figure following address I is decided by the direction of position A' viewed from position B. Since G92 is modal, the cycle operation is continued by specifying threading depth in the X-axis direction in the trailing blocks as follows.

```
X(U)···  ;
X(U)···  ;
```

EXAMPLE:

```
N50 G00 X80. Z80.8 M00 ;
N51 G92 X70. W-50.8 I-1.5 F2. ;
N52 X68.8 ;
N53 X67.8 ;
N54 G00 X100. Z100. MΔΔ ;
```



(4) Taper threading cycle with angle (Cutting along the angle of thread)

G92 X(U)··· Z(W)··· I··· K··· F(E)···;

This command permits the threading along the angle of thread in taper threading. The figure following the address K must be programmed with a sign.

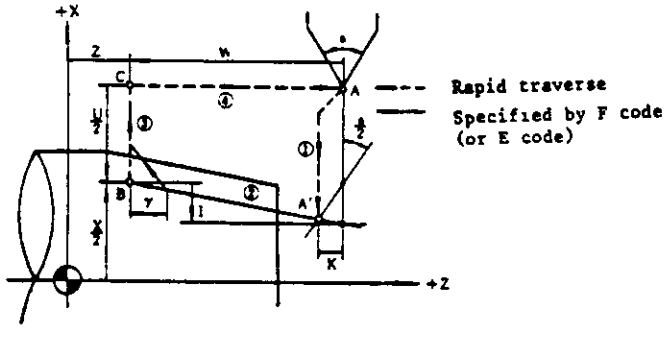


Fig. 2.61

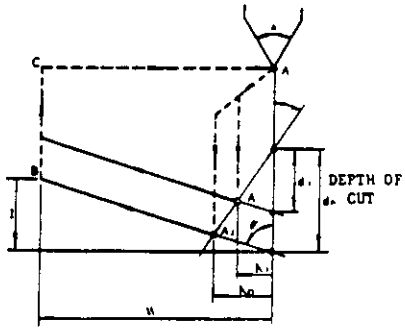


Fig. 2.62

For the threading along the angle of thread, the K for each cycle obtained from the following formula should be programmed.

$$|Kn| = \frac{dn.\tan(a/2)}{1 \pm \left|\frac{I}{W}\right| .\tan(a/2)}$$

The sign in the denominator depends on θ':

θ' < 90° → "+"
θ' > 90° → "-"

NOTE: For the control equipped with Multiple Repetitive Cycle†, the above troublesome calculation can be omitted by using G76 (Automatic Thread Cutting Cycle). The control performs the above calculation automatically by the G76 command.

(5) Cautions in programming G92

a. Specifying T, S and M codes in the effective area of G92 causes format error.

b. In Single Block mode, the execution of G92 cycle is stopped after the completion of the cycle ① to ④.

c. Thread cutting hold (option)

When this option is used, depressing the FEED HOLD during thread cutting causes the tool to perform threading up immediately and then return to start point A. When parameter #6019D7 = 1, the tool stops at position B where threading up terminated. Then, the tool returns to point A upon depression of the CYCLE START.

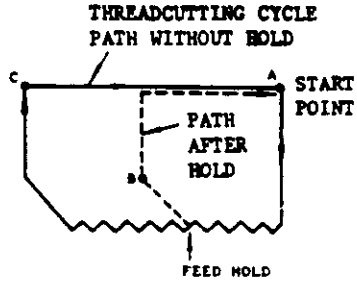


Fig. 2.63

2.8.27.2 Threading Cycle (G92) (CONT'D)

If the threadcutting hold option is not selected, depressing of the FEED HOLD during thread cutting does not hold the operation, which stops at point C where the retracting operation is completed.

d. Six angles of thread can be used in Multiple Repetitive cycle G76. In the G92 command, arbitrary angles of thread can be performed for threading.

2.8.27.3 Facing Cycle B (G94)

(1) Straight facing cycle

G94 X(U)··· Z(W)··· F(E)··· ;

The straight facing cycle ① to ④ is executed by this command.

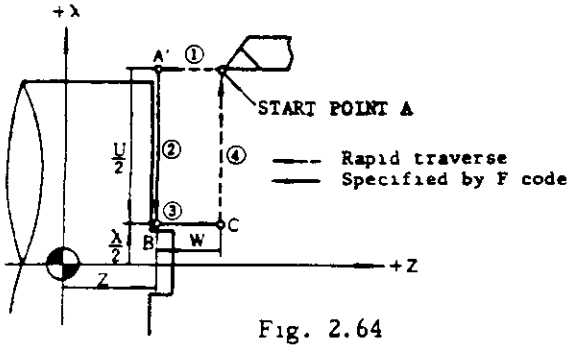


Fig. 2.64

Since G94 is modal, the cycle operation will be continued by specifying depth of cut in the Z-axis direction in the following blocks as follows.

Z(W)··· ;

Z(W)· · ,

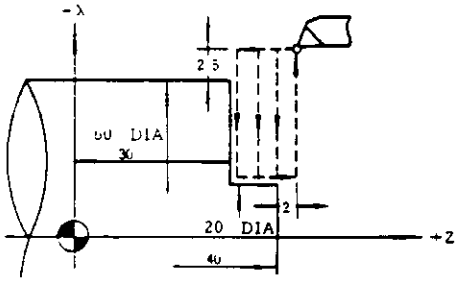
EXAMPLE

N60 G00 X65. Z42. ;

```
N61 G94 X20. Z38. F0.35 ;
N62 Z34. ;
N63 Z30. ;
```

Three cycles by G94 command.

N64 G00· · ;



(2) Taper facing cycle

G94 X(U)··· Z(W)··· K··· F(E)··· ;

The taper facing cycle ① to ④ is executed by this command.

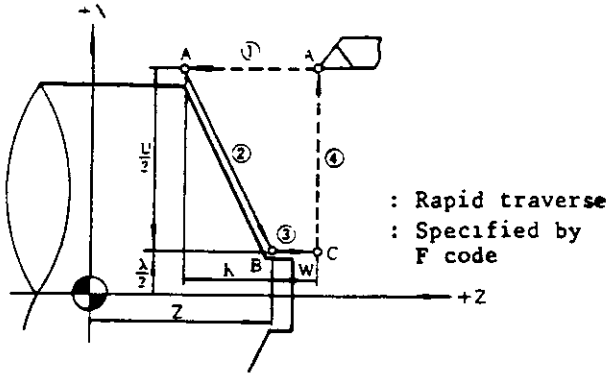


Fig. 2.65

The sign of figure following the address K is decided by the direction of position A' viewed from position B.

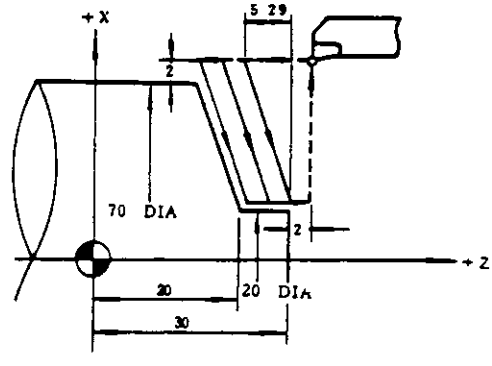
EXAMPLE:

N70 G00 X74. Z32. ;

```
N71 G94 X20. Z30. K-5.29 F0.3 ,
N72 Z25. ;
N73 Z20. ;
```

N74 G00··· ;

Three cycles by G94 command

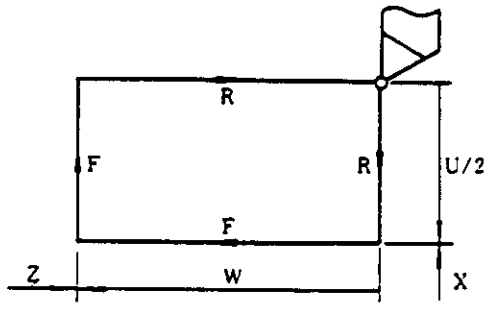
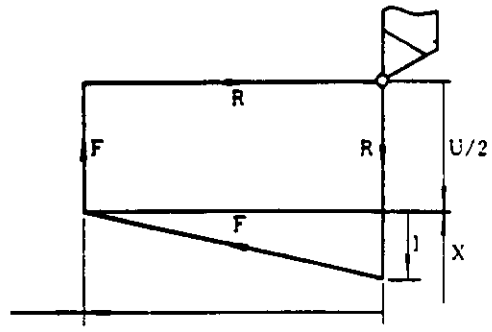
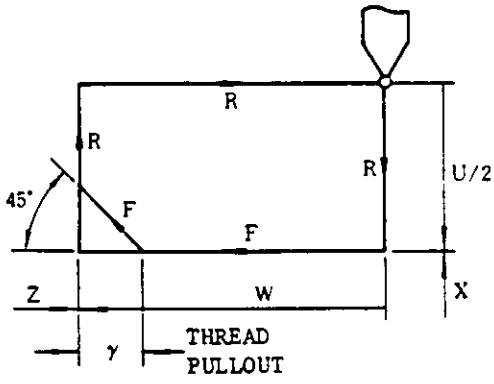
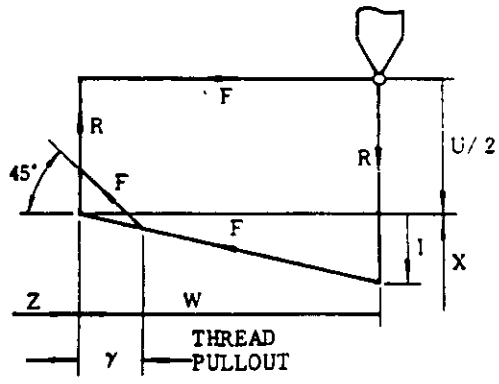
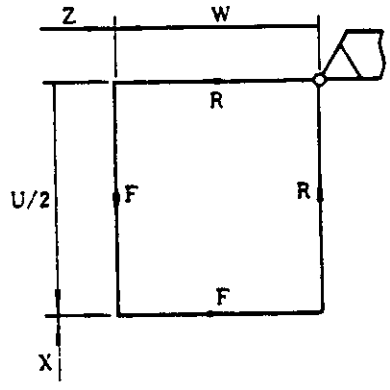
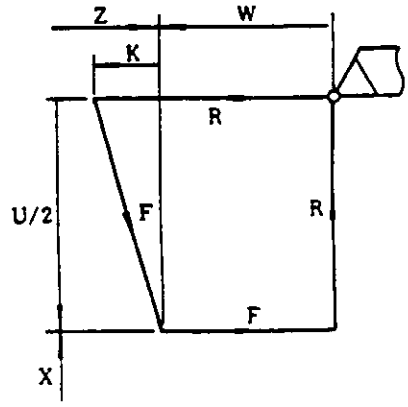


(3) Cautions in programming G94

a. M, S and T functions cannot be specified in the effective area of G94.

b. In Single Block mode, the execution of G94 cycle is stopped after the completion of ① to ④.

Table 2.23 Canned Cycles

Code	Straight Cycle	Taper Cycle
G90 Turning Cycle	G90 X(U)··· Z(W)··· F(E)··· ;	G90 X(U)··· Z(W)··· I··· F(E)··· ;
		
G92 Threading Cycle	G92 X(U)··· Z(W)··· F(E)··· ;	G92 X(U)··· Z(W)··· I··· F(E)··· ;
		
G94 Facing Cycle	G94 X(U)··· Z(W)··· F(E)··· ;	G94 X(U)··· Z(W)··· K··· F(E)··· ;
		

2.8.28 CONSTANT SURFACE SPEED CONTROL (G96, G97) T

This feature may be installed when the S4 digit specification option is selected.

The following G codes are used:

G96	Specifies the constant surface speed control.
G97	Cancels the constant surface speed control.

At the time of power-on, G97 (cancel) is provided.

These are modal G codes which belong to 02 group.

(1) Constant surface speed control (G96)

a. G96 S·· (M03) ;

Using this command, specify the surface speed of the work in a numeral of up to 4 digits after address S. The unit of the surface speed is as follows:

	Unit
Metric input	m/min
Inch input	ft/min

When the surface speed is specified, the equipment assumes the X-axis current value (Note 1) to be the diameter of the work and computes the spindle RPM every 100 msec so that the specified surface speed is obtained. The computation result is sent as the analog voltage output or the binary 12-bit signal output. In the subsequent blocks, the surface speed may be varied by S specification.

b. For a machine tool that allows spindle gear change, instruct the M code for gear change specification before the block of G96 command. For details, refer to the instruction manual for that machine tool.

EXAMPLE:

```
N8  MΔΔ  ; ———— M code for spindle
                 gear change (gear ratio
                 No. 4 is specified).
N9  G96  S100  M03  ;
    .
    .
    .
```

c. Before the block of G96 command, the spindle maximum RPM should be specified by G50. This specification prevents the spindle RPM from getting abnormally high by the computation of surface speed when the X-axis current value is smaller.

EXAMPLE:

```
N10  G50  S2000  ; ———— Specified the upper
                        limit of spindle RPM
N11  MΔΔ  ;
N12  G96  S150  M03  ,
```

(2) Constant surface speed control cancel (G97)

G97 S··· (M03) ;

Using this command, directly specify the spindle RPM in a numeral of up to 4 digits after address S. The constant surface speed control is cancelled and the usual spindle feature of S 4-digit specification is resumed.

(3) Cautions for constant surface speed control

a. How to set coordinate system (See Note)

When performing the constant surface speed control, make programming by setting G50 coordinate system or G58 work coordinate system so that X-axis coordinate value of the spindle rotational center become "0." In other words, the coordinate system should be set such that its X-axis coordinate value correctly represents the diameter of the work at its machining point.

b. How to handle tool position offset amount

(i) When performing the constant surface speed control, parameter #6020D$_1$ is generally set to "0." This enables the control to perform the computation for constant surface speed control without adding the tool position offset amount to the specified coordinate value. If a large value is used for the offset amount, the tool position offset is normally executed and, at the same time, the constant surface speed control is correctly performed.

(ii) When #6020D$_1$ is set to "1," the value (Specified coordinate value + tool position offset amount) is assumed to be the work diameter to compute the constant surface speed. Hence, care should be taken not to use a large value for the offset amount by performing coordinate system setting for each tool and using the tool position offset for tool wear compensation.

Note: #6020D1 and #6020D7 are independent from each other.

#6020D7 = "0" ··· The position obtained by adding the tool position offset amount and the tool nose radius compensation amount is shown in the current position display "POSITION ABSOLUTE."

#6020D7 = "1" ··· The position obtained by not adding the tool position offset amount is shown in the current position display "POSITION ABSOLUTE."
Thus, the switching of the computation for constant surface speed control and the switching of the computation for current position display on CRT are performed separately.

c. The spindle gear may be changed in the maximum of 4 steps. The parameters for the gear change are as shown below. For details, see the Parameter List.

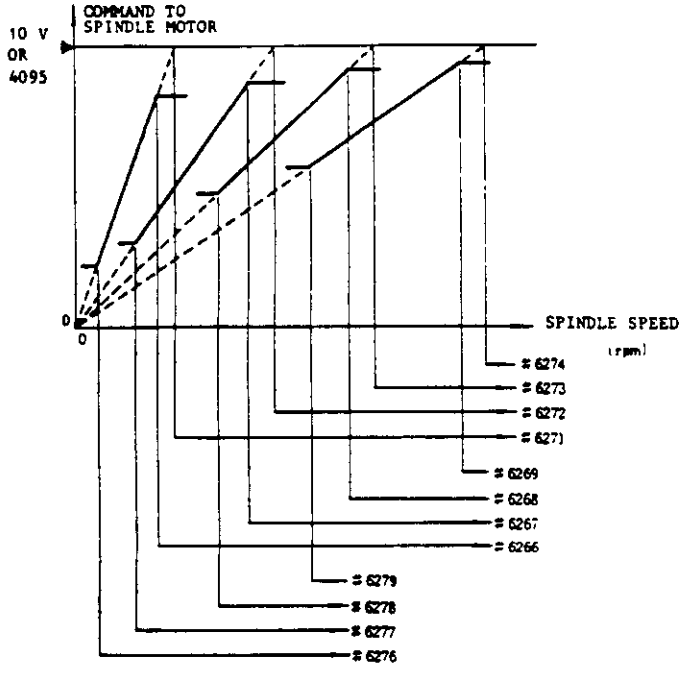


Fig. 2.66

d. When parameter #6020D0 = "1," the constant surface speed control is performed on all positioning blocks (G00, G06) also. For positioning, however, the constant surface speed is computed for the end coordinate of the positioning block. The constant surface speed is at every moment computed for only the cutting block.
When #6020D0 = "0," the constant surface speed control is performed only on the cutting block and the positioning block immediately before it. For the positioning block, the constant surface speed is computed for its end coordinate.

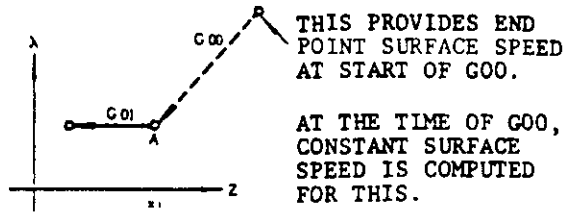
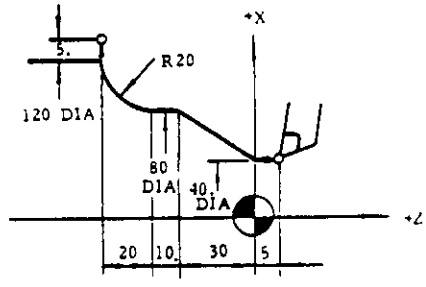


Fig. 2.67

EXAMPLE: #6020 = 0

```
N4 G50 S1500 ;      Upper limit of spindle speed clamp (in rpm)
N5 MΔΔ ;            M code for gear change
N6 G96 S150 M03 ;   Constant surface speed 150 m/min
N7 G00 X40. Z5. ;
N8 G01 Z0 F0.15 ;
N9 X80. Z-30. ;
N10 W-10. ;
N11 G22 X120. W-20. R20. ;
N12 G01 U10. ;
N13 G97 S500 ;      Cancel of constant surface speed control
N14 G50 S2000 ;
```

Constant surface speed control



2.8.29 FEED FUNCTION DESIGNATION (G98, G99)

These G codes are used to switch between the designation of feed per minute and the designation of feed per revolution, before specifying F(E) code (feed) command.

(1) G98 ;

By this command, the F(E) code specified after is executed on a feed per minute basis.

G98	Meaning
Metric input	mm/min designation
Inch input	in/min designation

2.8.29 FEED FUNCTION DESIGNATION (G98, G99) (CONT'D)

(2) G99 ;

By this command, the F(E) code specified after is executed on a feed per revolution basis.

G99	Meaning
Metric input	mm/rev designation
Inch input	inch/rev designation

NOTES:

1. G98 and G99 are modal G codes, which remain valid until the designation is changed.
2. When switching between G98 and G99 is performed, F(E) code designated before is cancelled. So, it must be designated again. Otherwise, an error is caused at the next cutting block.
3. The initial G code at power-on may be switched between G98 and G99 by the use of the following parameter

Parameter	Initial G code
#6005D1 = 0	G98
#6005D1 = 1	G99

2.8.30 TOOL LIFE CONTROL (G122, G123)†

This feature allows a long, unattended operation by replacing the tools each time the designated number of pieces or the designated use hour is reached

(1) The tool life control consists of the following three steps

a Registration of tool information

(i) Registration of tool group number. The tool numbers of the tools of the same type are registered as one group.

(ii) Registration of life of each tool group. The number of pieces that can be machined by a tool to be registered in each tool group and the usable hours of it are registered.

(iii) Registration of use offset number of each tool.

These must be registered from the part program in the memory of the equipment before starting machining.

b. T designation for tool life control

For the T designation in the part program, the special T designation for tool life control must be used.

c. Input/output signals for tool life control

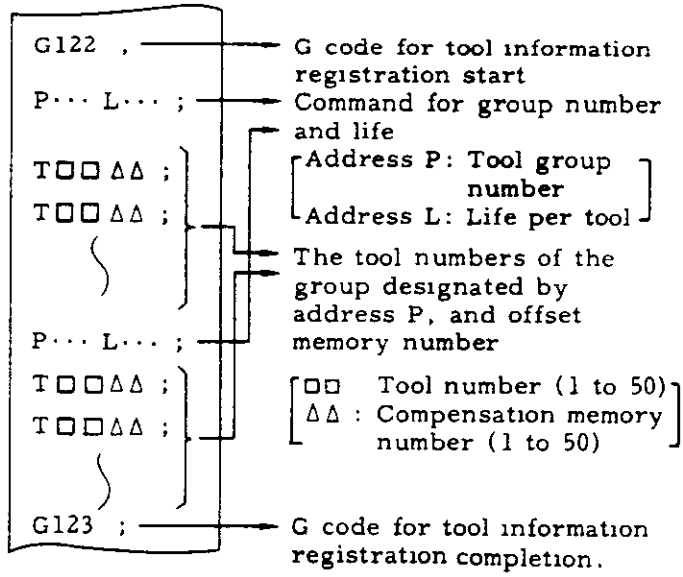
- Tool life control I/O signal.
- Tool skip input.
- Tool replacement completion input.
- Tool replacement completion group number input.

The above inputs are provided to implement the highly efficient tool life control feature.

The following describe these three steps in detail:

(2) Registration of tool information

a The part program for tool information registration



Group No.		19 groups, 1 to 19, may be used.
Life per tool	Number of pieces	For groups 1 to 9, life is designated by the number of works (1 to 9999 works).
	Use hours	For groups 10 to 19, life is designated by use hours (1 to 9999 minutes).

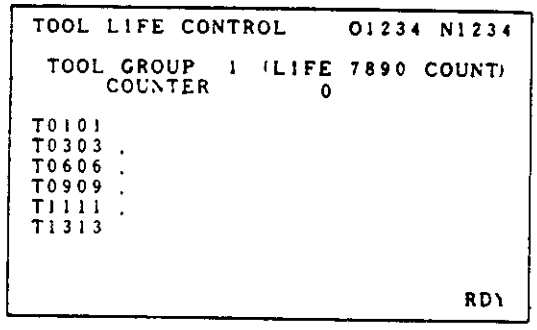


Fig. 2.68 Tool Life Data on Tool Life Control by No. of Pieces to be Machined

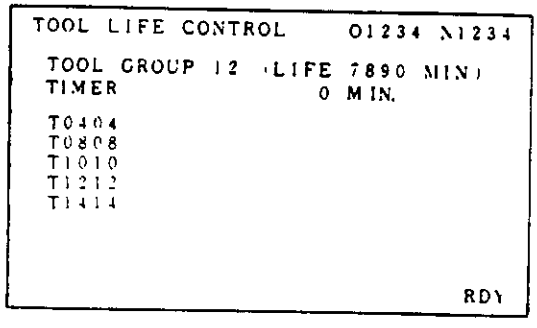


Fig. 2.69 Tool Life Data on Tool Life Control by Operating Time

b. When the part program for tool information registration is executed, the following screen is shown in the COM function display on CRT The screen enables the confirmation of the registered information. (Depress PAGE key to display the following screen.)

EXAMPLE:

The display of the information registered by executing the following program is shown.

```
O··· ,
G122 ;
P1 L7890 ;
T0101 ,
T0303 ,
T0606 ;
T0909 ,
T1111 ;
T1313 ;
P12 L7890 ;
T0404 ,
T0808 ,
T1010 ;
T1212 ;
T1414 ,
G123 ,
M30 ,
```

(3) T designation for tool life control

Using the following special T commands, create the part program for machining.

a. Use the following T commands.

T□□90	The tool of the group specified in "□□" is given as T command. However, tool position offset is cancelled.
T□□91	The tool of the group specified in "□□" is given as T command and, at the same time, the content of the registered offset number is applied as offset.

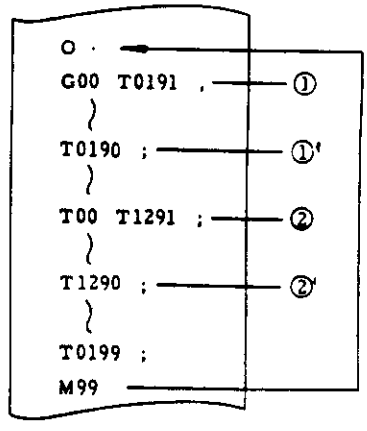
Note: When two or more compensation memories are used for a single tool, refer to the description in subparagraph (5), a.

b. When a group number registered by the number of pieces machinable per tool of group number 1 to 9 is specified, give "T□□99" (□□is the group number) for the command for counting the number of pieces, at the completion of machining.
When the group numbers 10 to 19 which designate the maximum operating time of a single tool, the control automatically counts the feed time as operating time. T-command need not be programmed for time counting on part program.

c. Display for confirmation of the contents of "T□□91" command.

EXAMPLE:

When the following part program has been executed after the tool information registration described in subparagraph (2).



2.8.30 TOOL LIFE CONTROL (G122, G123)† (CONT'D)

At the execution of the part program specified with tool life control, the tool number in the current group and the offset number being used by "T□□91" command may be shown in COM display on CRT screen.

In the above program, ① shows that "T0191" specifies the machining by "T0606" in the registered tool information.
(①' shows that "T0190" becomes "T0600.")

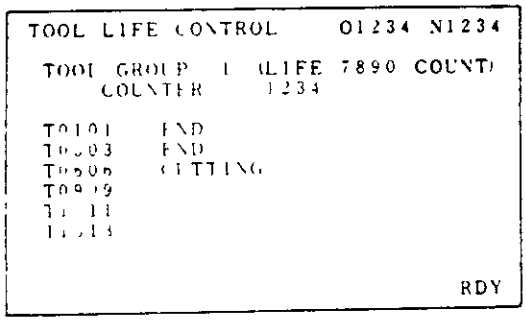
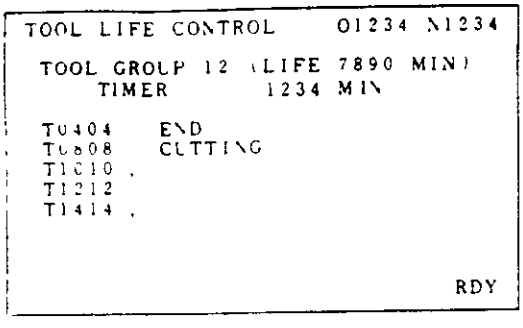


Fig. 2.70 Example A of Display of Tool No. and Offset No of Current Execution



② shows that "T1291" command specifies the machining by "T0808" in the registered tool information
(②' shows that "T1290" becomes "T0800" command.)

Fig. 2.71 Example B of Display of Tool No. and Offset No. of Current Execution

(4) Input/output signals for tool life control

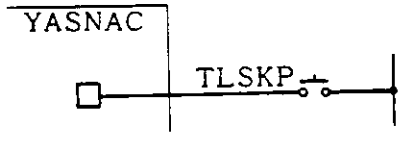
The following input/output signals are provided for the control

Table 2.24

Signal Name	Description
TLSKP	Tool skp input.
TLCH	Tool replacement request output.
TLRST	Tool replacement completion input.
TLA11 TLA12 TLA14 TLA18 TLA21	Tool replacement completion group number input. (BCD code)

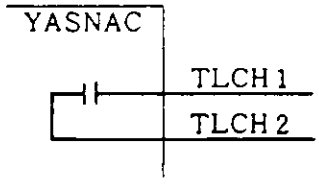
The operation for each input/output depends on the operation panel and external sequences provided by the machine builder. For details, refer to the instruction manual offered by the machine builder. The following describes the control function in response to the input/output signals to facilitate the understanding of the tool life control feature

a. TLSKP. Tool skip input



This is the input signal for replacing the tool before the registered life of it expires. When TLSKP input is "closed" during the time from the output of the tool registered by group number "1 to 19" as T command to the output of another T command, the control outputs the T command of the next tool registered in the group at the time of the next T command output.

b. Tool replacement request output



All tools in each group number are given when the registered life (the number of pieces or use hours) expires. The output is performed when T command for life completion group number was executed after the expiration.

(iii) T command for life completion group number was executed.

When the tool replacement request output signal goes on, alarm number 157 is displayed on the CRT screen.
When the tool replacement request output signal goes on, look at the CRT screen, namely, the tool life control display screen of COM function, make confirmation of the number of the group in which all tools have come to the expiration of useful life, and replace them.

EXAMPLE

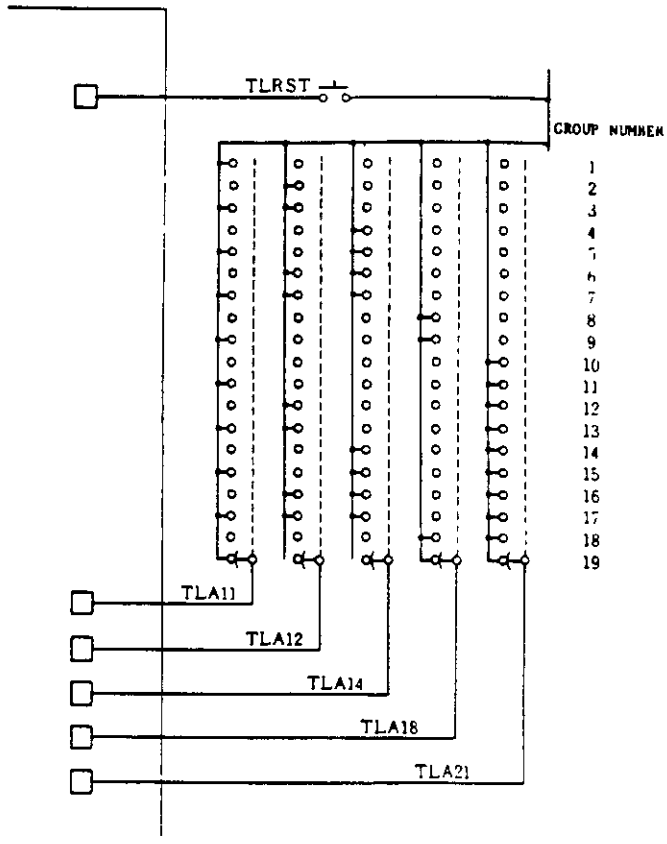
Shown below is a display example in which the life of the tools belonging to group number "1" has been completed.

```
TOOL LIFE CONTROL      01234 N1234

 TOOL GROUP   1  (LIFE 7890 COUNT)
     COUNTER     7890

 T0101    END
 T0303    END
 T0606    END
 T0909    END
 T1111    END
 T1313    END
```

c. Tool replacement completion input (TLRST) and tool replacement completion group number inputs (TLA11 through TLA21).
These signals are used to indicate the replacement completion to the control after the tool replacement request output (TLCH1, TLCH2) goes on and the tools whose life has expired are replaced with new ones.
When the tool replacement operation is completed, set the group number to "tool replacement completion group number input (TLA11, TLA12, TLA14, TLA18, TLA21) and put the tool replacement completion input (TLRST) in the "closed" state.



When the tool replacement operation has been completed with the tool replacement completion input "closed," look at the tool life control display screen of COM function on CRT and check to see if the control internal memory is accepting the completion of the replacement operation. When this operation is found accepted, perform a reset operation (depress RESET key on MDI & CRT panel or "close external reset input ERS), and the tool replacement request output goes off and the displayed alarm code disappears.

2.8.30 TOOL LIFE CONTROL (G122, G123)± (CONT'D)

EXAMPLE.

Shown below is a display example in which the life of the group "12" tools has expired.

```
TOOL LIFE CONTROL     01234 N1234

 TOOL GROP 12  (LIFE 7890 MIN)
      TIMER        7890 MIN.

 T0404    END
 T0808    END
 T1010 .  END
 T1212    END
 T1414 .  END


                                RDY
```

Shown below is a display example in which "12" is set as the tool replacement completion group number input and the tool replacement completion input is "closed" after replacement of all tools.

```
TOOL LIFE CONTROL     01234 N1234

 TOOL GROUP 12  (LIFE 7890 MIN)
      TIMER           0 MIN)

 T0404
 T0808
 T1010
 T1212 .
 T1414


                                RDY
```

(5) Supplementary explanation for tool life control

a. The tool life control in which multiple tool offset memories are used by a single tool is performed as follows:

(1) Make specification as follows by the use of the part program for tool information registration.

```
G122 ;
P··· L··· ;
T □□ □□ ,
T □□ □□ ,
T □□ □□ ,
T □□ □□ ,
T □□ □□ ,
T □□ □□ ;
T □□ □□ ;
T □□ □□ ;
T □□ □□ ,
T □□ □□ ;
P··· L··· ;
T···
  ⟩
G123 ;
```

□□ . Tool number

ΔΔ : Tool offset memory number

As shown above, consecutively specify the tool numbers to be used and the offset memory numbers in the group to be specified by address P. Up to 5 offset memories may be used for one tool.

EXAMPLE

```
G122 ;
P1 L7890 ;
T0101 ,
T0111 ;
T0121 ;
T0131 ;
T0141 ;
T0202 ;
T0212 ;
T0222 ,
T0232 ;
T0242 ;
P··· L··· ;
  ⟩
G123 ;
```

This is the display of tool life control group No. 1 of COM function on CRT screen at registration of tool information by the execution of the above program.

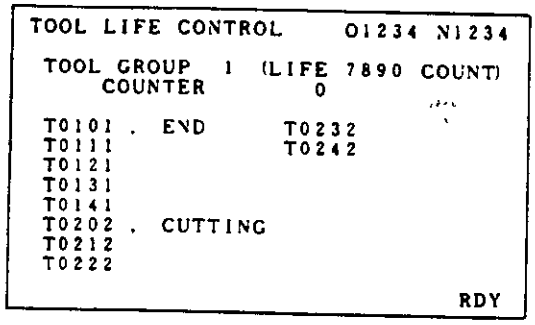


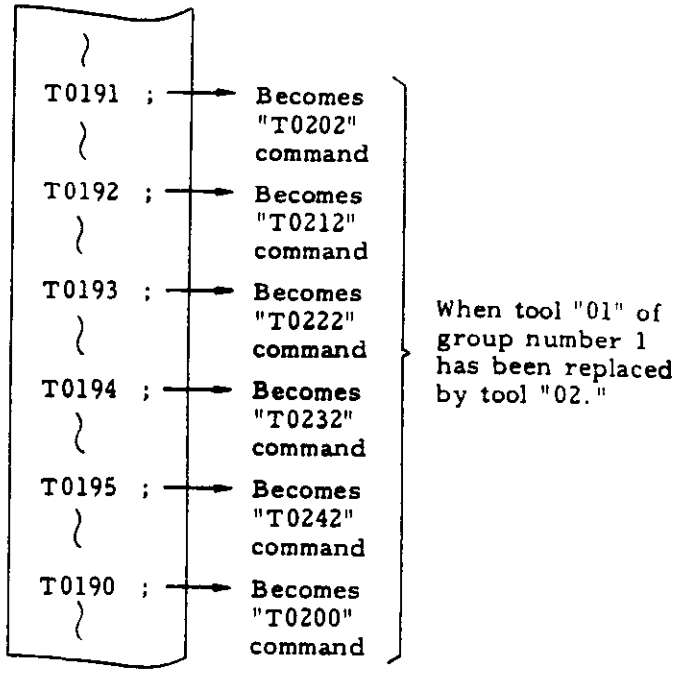
Fig. 2.72 Display of Tool Offset Memories Used for One Tool

ii) The part program
Use the following T commands:

Table 2.25

T□□90	The tool of the group specified in "□□" is given as T command.	Compensation is "00."
T□□91		Compensation is the compensation memory number registered first.
T□□92		Compensation is the compensation memory number registered second.
T□□93		Compensation is the compensation memory number registered third.
T□□94		Compensation is the compensation memory number registered fourth.
T□□95		Compensation is the compensation memory number registered fifth.

The T commands in the execution of the following program after the registration of tool information shown in EXAMPLE are as follows:



The display of tool life control group No. 1 of COM function on CRT screen at execution of the above program is as follows:

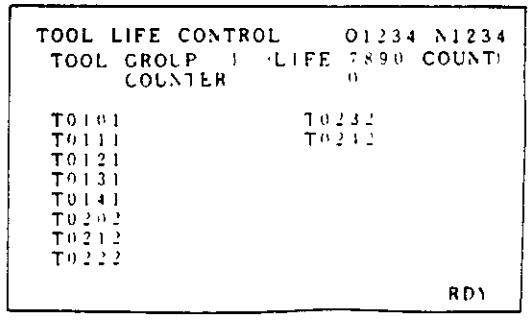


Fig. 2.73

b. Tool life control in which work coordinate system setting (G50 T···) is used.

(i) Use of work coordinate system setting does not require to modify the program for tool information registration.

(ii) The work coordinate system setting command to be used in the part program should be as follows:

G50 T□□90 ;
└ Group number (1 to 19)

2.8.30 TOOL LIFE CONTROL (G122, G123)+ (CONT'D)

There are following restrictions·

· The tool number used is 01 to 30.

· The tool coordinate memory number to be used for each tool is as follows:

Table 2.26

Tool No.	Tool Coordinate Memory No.
01	51
02	52
03	53
⟩	⟩
30	80

(iii) When the command "G50 T□□90" is executed in the following programs, the control operates as shown in Table 2.8.30.5.

Program for Tool Information Registration

```
O· ·
G122 ,
P1 L100 ,
T0101 ,
T0303 ,
T0606 ,
P  · L
  ⟩
G123 ,
M30 ,
```

Part Program

```
O·
  ⟩
T0190 ,
G50 T0190 ,
  ⟩
```

Table 2.8.30.5

Tool no. of group no. 01	Operation by "G50T0190"
01	"G50 T5100"
03	"G50 T5300"
06	"G50 T5600"

c. Tool life control and setting

The registration of tool information and the monitoring of the currently used tools are performed using the setting area. Shown below is the list of settings used for tool life control.

Note. "Registration of tool information not from the part program but by the writing of settings" and "the presetting of the number of tools by settings" are provided for special operations. However, these operations should not be performed in principle.

(6) List of Settings for Tool Life Control

a. Registration of group number for each tool

Table 2.27

Setting No.	Contents	Remarks
#8601	The group number of tool to be selected by "T01**" command.	Tool groups are "1 to 19." The tool whose life has expired has a minus value.
#8602	The group number of tool to be selected by "T02**" command.	
⟩	⟩	
#8650	The group number of tool to be selected by "T50**" command.	

** indicates compensation number.

b. Registration of life of each tool group

Table 2.28

Setting No.	Contents	Remarks
#6161	The life of group number "1." (The number of machinable pieces)	The setting range of the number of machinable pieces is 1 to 9999 (units)
⟩	⟩	
#6169	The life of group number "9." (The number of machinable pieces)	
#6170	The life of group number "10." (Machinable hours)	The setting range of the number of machinable hours is 1 to 9999 (minutes)
⟩	⟩	
#6179	The life of group number "19." (Machinable hours)	

c. Registration of tool offset memory and tool number

Table 2.29

Setting	Contents	Remarks
TOFN01 (#8651)	The tool number which uses offset memory 01 (T**01).	Tool number is "1 to 50."
TOFN02 (#8652)	The tool number which uses offset memory 02 (T**02).	
⌇	⌇	
TOFN50 (#8700)	The tool number which uses offset memory 50 (T**50).	

d. Registration when multiple tool offset memories are used by a single tool

Table 2.30

Setting	Contents	Remarks
TOFO01 (#8701)	The registration of the offset designation number in the tool life control program of offset memory 01 (T**01).	The number not used in tool life control is "0."
TOFO02 (#8702)	The registration of the offset designation number in the tool life control program of offset memory 02 (T**02).	The number used in tool life control program is 91 to 95. 91 designation is "1." 92 designation is "2." 93 designation is "3." 94 designation is "4." 95 designation is "5."
⌇	⌇	
TOFO50 (#8750)	The registration of the offset designation number in the tool life control program of offset memory 50 (T**50).	

When the part program shown in sub-paragraph (5), a, (1) is registered, any one of 1 to 5, and 0 is stored here.

e. Monitoring of the currently used tool in each group

Table 2.31

Setting	Contents	Remarks
TG1CNT (#6181)	The number of pieces being machined by the currently used tool of group number "1."	The number of machinable pieces is set in #6161 to #6179.
⌇	⌇	
TG9CNT (#6189)	The number of pieces being machined by the currently used tool of group number "9."	
TG10CNT (#6190)	The use time of the currently used tool of group number "10."	The machinable hour is set in #6180 to #6189.
⌇	⌇	
TG19CNT (#6199)	The use time of the currently used tool of group number "19."	

NOTES:

1. To use the tool life control feature, select the "T 4-digit designation" and the "offset memory addition" options.
2. When "90" through "95," or "99" is specified in the low-order 2 digits of the T command (4-digit designation) in the part program, the control executes the tool life control feature. A T-command other than above does not cause the execution of this feature and is processed as an ordinary T command. Hence, it is possible to specify the T command for tool life control and an ordinary T command in a single part program.
3. The maximum number of "tool number + offset memory number" pairs which can be registered in a single tool group is 16.
4. G122 and G123 commands should always be specified on a single block basis.
5. Between G122 command and G123 command, only the following should be specified.

   P: Group number.

   L: Life for each tool.

   T: Tool number and compensation memory number.

   Any other addresses cannot be specified.

2.8.30 TOOL LIFE CONTROL (G122, G123)^{T} (CONT'D)

The tool replacement completion input is accepted also when the tool replacement request output is not on (the usable tool is still left) if the automatic operation is in the hold state. Care should be used in the handling of this feature.

7. Tool life control is applied only to the T command in the part program. The tool operation by the manual intervention during run is not affected by this control.

8. The count time of the tool life control by use hour is held after the power-off. Precisely, however, the maximum of one minute of count time may be discarded between power-off and power-on. This is because the count time is held in units of a minute.

9. At the start of the tool information registration by G122 command, the control cancels all the registered contents before starting new registration. Hence, the registration of partial tool information is not allowed. Always register the entire tool information.

10 "G50 T□□91" to "G50 T□□95" may be specified for the work coordinate system setting command to set the work coordinate system added with the contents of offset memory. In this case, however, the deviation of machining start position or the like may be caused. This specification should not be performed, in principle.

11 For tool life control during use (group No. 10 to 19), do not use multiple tool offset memories for a single tool. (Refer to (5) Supplementary Explanation in 9.)

In this case, if T- command is given for changing tool offset values after tool life expectancy has been reached, the tool will be exchanged for a new one causing an unexpected impact on the machine.

(6) List of alarms to be given by tool life control

Table 2.32

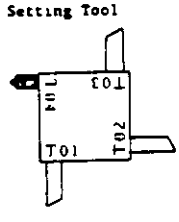
Alarm No.	Cause
150	G121 or G122 command is not specified on a single block basis.
151	The designation of group number P is not provided. Or a value other than 1 ≦ P ≦ 19 is designated.
151	The designation of life per tool L is not provided. Or a value other than 1 ≦ L ≦ 9999 is designated.
152	A value other than 1 ≦ tool number ≦ 50 is designated for the tool number.

Table 2.32 (Cont'd)

Alarm No.	Cause
152	A value other than 1 ≦ compensation memory number ≦ 50 is designated for the compensation memory number.
153	The tool information of the same group number is registered twice.
154	It was attempted to register more than 16 pairs of "tool number + compensation memory number" in a single group number.
150	An address other than P, L, and T is designated in the tool information registration program.
159	More than 6 pairs of compensation memory numbers are registered for a single tool.
155	The tool of the specified group number is not registered.
155	T□□92, T□□93, T□□94, or T□□95 is specified but the corresponding compensation memory number is not registered.
152	Zero or a value greater than 20 is designated in (group number) of T 90 through T 95, or T 99.
158	Some registered tool in the tool group designated in □□ of work coordinate system setting (G50 T□□90) has a tool number greater than "31."
156	Tool life control is designated in the control having no "T 4-digit designation" and "offset memory addition" options.
157	The tool replacement request output is on.

EXAMPLE

Setting Tool



Tool No.	Offset Memory	Type of Cutting
T01	01	Rough cutting
T02	02	
T03	03	
T04	04	Finish cutting

Programs for Tool Information Registration

```
O··· ;
G122 ,
P01 L10 ,
T0101 ,
T0202 ,
T0303 ,
G123 ,
M30 ,
```

When the above program is executed, T0101, T0202, and T0303 are registered as group number 1 and with the number of machinable pieces per tool being 10.

Part Program

```
O ··
N0001 G50 X    Z    ,
G00 T0191 ;  ——①
     ( ROUGH CUTTING PROGRAM )
G00 X··· Z··· T0190 ;  ——②
N002 G50 X··· Z· ·
G00 T0404 ;
     ( FINISH CUTTING PROGRAM )
G00 T0400 ;
T0199 ;
M99 ;  ——
```

When the above program is executed, blocks of ① and ② provide the following commands:

No. of Executions	Block shown by ①	Block shown by ②
1 - 10 times	G00 T0101	G00 T0100
11 - 20 times	G00 T0202	G00 T0200
21 - 30 times	G00 T0303	G00 T0300
31ST time	Tool change request	

The processing after the completion of tool life control:

When the machining of T01, G02, and T03 has been completed and the tool replacement request output is turned on, input the value of compensation memory for the new tool then perform the following operations.

(i) Set "1" as the tool replacement completion number input (see (4), d.)

(ii) Turn on the tool replacement completion input (see (4), d.)

(iii) Perform the RESET operation.

2.8.31 MULTIPLE CORNERING (G111, G112)†

These commands are used to perform beveling and rounding on the taper and circular arc portions of a work.

G code	Meaning
G111	Multiple cornering on taper portion.
G112	Multiple cornering on circular arc portion.

These commands enable the control to perform beveling and rounding on taper and circular arc portions without making complex computation.

2.8.31.1 Taper Multiple Cornering (G111)

The following four operations may be specified in a single block:

Taper → Beveling / Rounding → Taper → Beveling / Rounding

The typical contours for which taper multiple cornering is specified are shown in Fig. 2.8.31.1 to 2.8.31.5 on pages 102 and 103.

2.8.31.1 Taper Multiple Cornering (G111) (CONT'D)

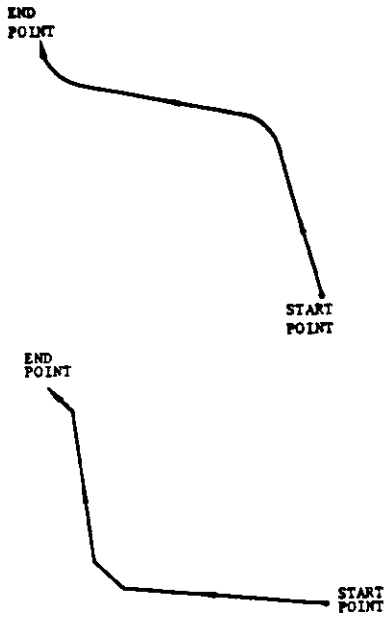
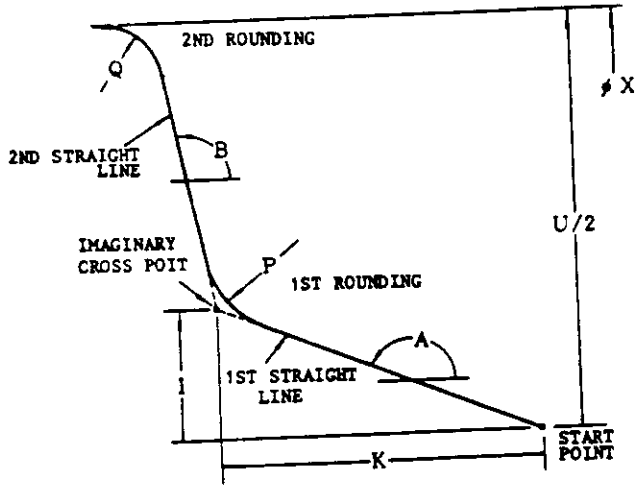


Fig. 2.74

(1) Command format for configurations for multiple cornering

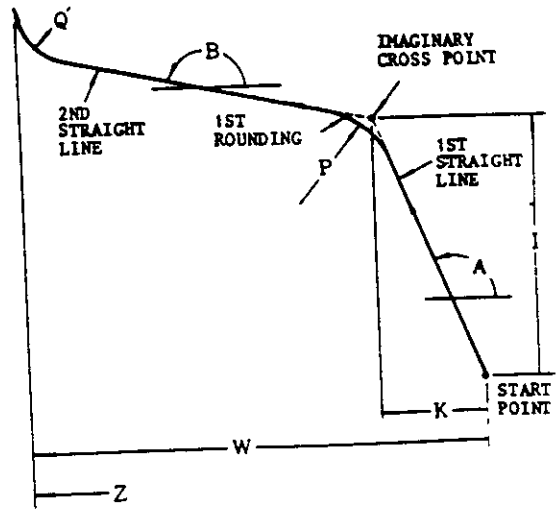
a.



G111 X(U)··· I··· A··· B··· P··· Q··· ;
G111 X(U)··· K··· A··· B··· P··· Q··· ;
G111 X(U)··· I··· K··· B··· P··· Q··· ;

Fig. 2.75

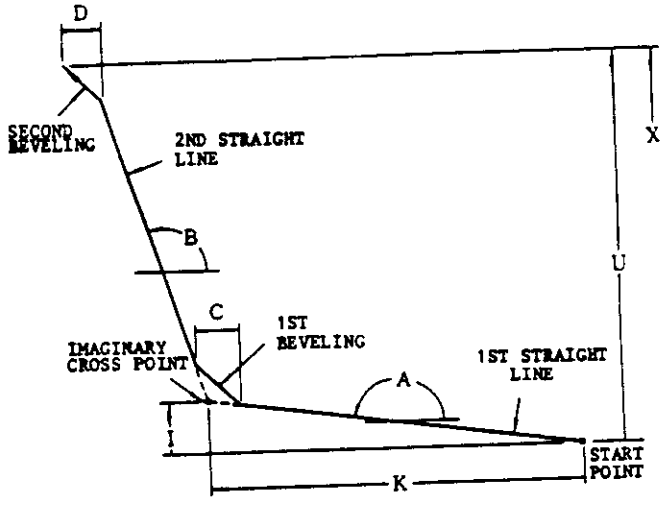
b.



G111 Z(W)··· I··· A··· B··· P··· Q··· ;
G111 Z(W)··· K··· A··· B··· P··· Q··· ;
G111 Z(W)··· I··· K··· B··· P··· Q··· ;

Fig. 2.76

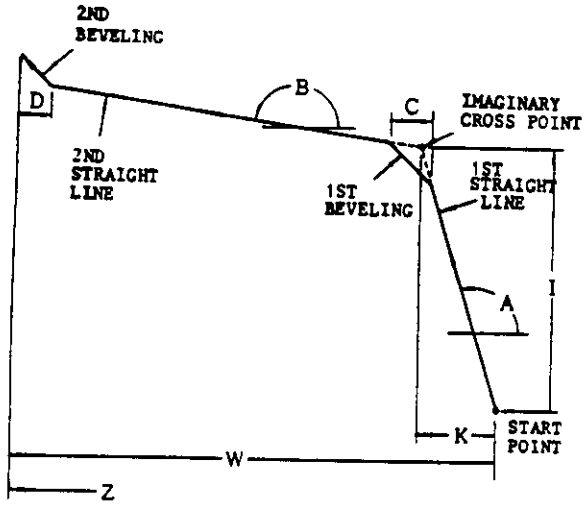
c.



G111 X(U)··· I··· A··· B··· C··· D··· ;
G111 X(U)··· K··· A··· B··· C··· D··· ;
G111 X(U)··· I··· A··· B··· C··· D··· ;

Fig. 2.77

d.



G111 Z(W)··· I··· A··· B··· C··· D··· ;

G111 Z(W)··· K··· A··· B··· C··· D··· ;

G111 Z(W)··· I··· A··· B··· C··· D··· ;

Fig. 2.78

(2) Meaning of addresses

The following address words may be specified for the taper multiple cornering command. Simply specifying the contour determining address words permits the required operation.

Table 2.33

Address Word	Contents	Unit
X(U)	X-axis end point coordinate (U Increment from start point)	1 = 0.001 mm or 1 = 0.0001 in.
Z(W)	Z-axis end point coordinate (W: Increment from start point)	
A	Move angle of the first straight line	1 = 0.001 deg
B	Move angle of the second straight line	
I	Virtual intersection between first and second straight lines. X-axis distance from start point (radius value).	1 = 0.001 mm or 1 = 0.0001 in.

Table 2.33 (Cont'd)

Address Word	Contents	Unit
K	Virtual intersection between first and second straight lines. Z-axis distance from start point.	1 = 0.001 mm or 1 = 0.0001 in
P	The first rounding radius (without sign).	
Q	The second rounding radius (without sign).	
C	The first beveling amount (without sign).	
D	The second beveling amount (without sign).	

(3) Designation of contours

a. Designate the contour as shown below.

Table 2.34

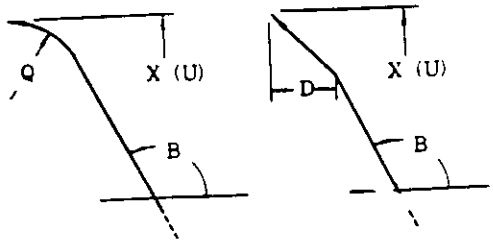
First straight line	A: First straight line move angle I. Virtual intersection X-axis distance from start point K: Virtual intersection Z-axis distance from start point	Specify two
First beveling or rounding	C: First beveling amount D: First rounding radius	Specify either
Second straight line	B: Second straight line move angle. X(U): X-axis end point coordinate [(U): Increment from X-axis start or end point)] Z(W): Z-axis end point coordinate [(W): Increment from Z-axis start or end point)]	Specify two. However, the following combinations are not permitted: X and U, Z and W
Second beveling or rounding	D. Second beveling value Q: Radius for second rounding Either D or Q should be commanded.	

2.8.31.1 Taper Multiple Cornering (G111) (CONT'D)

b. The first rounding touches the first and second straight lines.

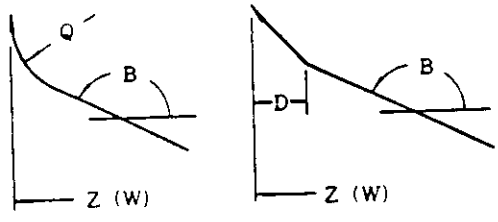
c. The second beveling and rounding depend on the designation of second straight line as shown below:

(i) The second straight line is specified with B and X(U).



ROUNDING TOUCHING THE STRAIGHT LINE PARALLEL TO Z-AXIS

(ii) The second straight line is specified with B and Z(W).



ROUNDING TOUCHING THE STRAIGHT LINE PARALLEL TO X-AXIS

d The second beveling and rounding are performed in the direction in which the second straight line advances. For details, refer to Table 2.8.31.4.

Table 2.35 Directions of Second Beveling

B command value for second straight line move angle	Beveling direction +X, +Z	Other conditions
B = 0 = -360.000 = 360.000	Beveling in X, Z positive direction	First straight line moves in X positive direction
	Beveling in X negative direction, Z positive direction	First straight line moves in X negative direction
0 < B, B < 90.000 -360.000 < B, B < -270.000	Beveling in X, Z positive direction	
B = 90.000 = -270.000	Beveling in X, Z positive direction	First straight line moves in Z positive direction
	Beveling in X positive, Z negative direction	First straight line moves in Z negative direction
90.000 < B, B < 180.000 -270.000 < B, B < -180.000	Beveling in X positive, Z negative direction	
B = 180.000, =-180.000	Beveling in X positive, Z negative direction	First straight line moves in X positive direction
	Beveling in X negative, Z negative direction	First straight line moves in X negative direction

Table 2.35 Directions of Second Beveling (Cont'd)

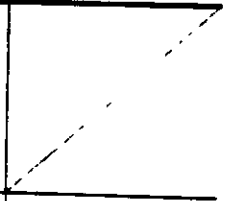
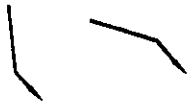
B command value for second straight line move angle	Beveling direction +x, +z	Other conditions
180.000 < B, B < 270.000 -180.000 < B, B < -90.000	Beveling in X, Z positive direction 	
B = 270.000 = -90.000	Beveling in X, Z negative direction 	First straight line moves in Z negative direction
	Beveling in X negative, Z positive direction 	First straight line moves in Z positive direction
270.000 < B, B < 360.000 -90.000 < B, B < 0	Beveling in X negative, Z positive direction 	

Table 2.36 Directions of Second Rounding

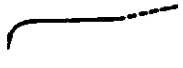
B command value for second straight line move angle	Rounding direction +x, +z	Other conditions	
B = 0 = 360.000 = -360.000	Rounding in X negative, Z positive direction 	First straight line moves in X-axis negative direction	X(U) command may not be used.
	Rounding in X, Z positive, direction 	First straight line moves in X-axis positive direction	

Table 2.36 Direction of Second Rounding (Cont'd)

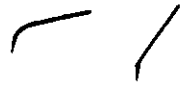
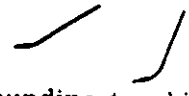
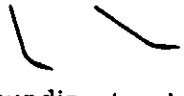
B command value for second straight line move angle	Rounding direction +x, +z	Other conditions	
0 < B, B < 90.000 -360.000 < B, B < -270.000	Rounding in X, Z positive direction  Rounding touching the straight line parallel to X-axis	Second straight line is specified with B, Z(W).	
	 Rounding touching the straight line parallel to Z axis	Second straight line is specified with B, X(U).	
B = 90.000 = -270.000	Rounding in X, Z positive direction 	First straight line moves in Z-axis positive direction	Z(W) command may not be used.
	Rounding in X positive, Z negative direction 	First straight line moves in Z-axis negative direction	
90.000 < B, B < 180.000 -270.000 < B, B < -180.000	Rounding in X positive, Z negative direction  Rounding touching the straight line parallel to Z axis	Second straight line is specified with B, X(U).	
	 Rounding touching the straight line parallel to X axis	Second straight line is specified with B, Z(W).	

Table 2.36 Direction of Second Rounding (Cont'd)

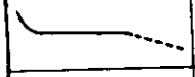
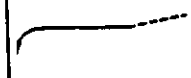
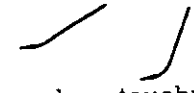
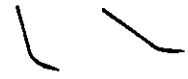
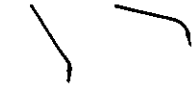
B command value for second straight line move angle	Rounding direction +X / +Z	Other conditions
B = 180.000 =-180.000	Rounding in X positive, Z negative direction 	First straight line moves in X-axis positive direction	X(U) command may not be used.
	Rounding in X, Z negative direction 	First straight line moves in X-axis negative direction	
180.000 < B, B < 270.000 -180.000 < B, B < -90.000	Rounding in X, Z negative direction  Rounding touching the straight line parallel to X axis	Second straight line is specified with B, Z(W).
	Rounding touching the straight line parallel to Z axis 	Second straight line is specified with B, X(U).
B = 270.000 = -90.000	Rounding in X, Z negative direction 	First straight line moves in Z-axis negative direction	Z(W) command may not be used.
	Rounding X negative, Z positive direction 	First straight line moves in Z-axis positive direction	

Table 2.36 Direction of Second Rounding (Cont'd)

B command value for second straight line move angle	Rounding direction +X / +Z	Other conditions
270.000 < B, B < 360.000 -90.000 < B, B < 0	Rounding in X negative, Z positive direction  Rounding touching the straight line parallel to Z axis	Second straight line is specified with B, X(U).
	Rounding touching the straight line parallel to X axis 	Second straight line is specified with B, Z(W).

e. Supplementary description

(i) When all B, X(U), and Z(W) of the second straight line are specified, the first straight line may provide one of the A, I, and K commands.

(ii) The taper multiple cornering command specifies the first and second and straight lines by selecting addresses X, Z, I, K, A, and B. Hence, unlike the other G commands, X, Z, I, K, and B do not allow the omission of "0" specification. The specification of "0" have different meaning for X, Z, I, K, and B. "0" should always be specified.

Omission of address "0" specification of taper multiple cornering command.

Table 2.37

Address	"0" specification may be omitted/not omitted
X Z I K A B	May not be omitted.
P Q C D	May be omitted. (Cornering amount is "0")

(iii) When the second straight line is designated by X(U) and Z(W) commands, the second beveling and rounding are disabled. If this designation is attempted, an error is caused.

(iv) The combination of the first beveling and the second rounding or the first rounding and the second beveling is also available.

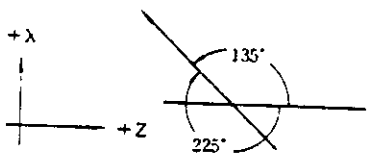
(v) When the first straight line designation addresses A, I, and K are all specified, A is ignored. The first straight line is created by I and K commands alone.

(vi) When the second straight line designation addresses B, X(U), and Z(W) are all specified and two of the first straight line designation addresses A, I, and K are further specified, B is ignored. The second straight line is created by X(U) and Z(W) commands.

(vii) How to specify straight line move angles A and B.

Specify a positive value for the counterclockwise rotational angle from Z-axis positive direction, and a negative value for the clockwise rotational angle from Z-axis positive direction.
(Specification range: $-360.000 \leq A, B \leq 360.000$)

EXAMPLE:

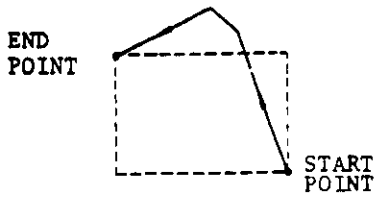


A135 or A225

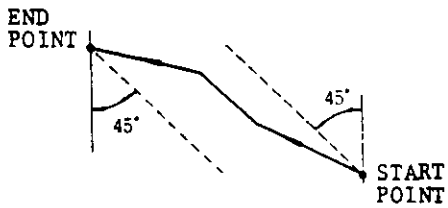
NOTES:

1. G111 is nonmodal G code and is valid only in the specified block.
2. Addresses M, S, and T cannot be specified in the block specified with G111.
3. If the first beveling portion to be specified by address C of the block specified with G111 has the contour shown below, the specification is disabled:

(i) Outside the rectangle enclosed by the start and end points.



(ii) Between the straight line going from start point to end point at angle of 45° and the straight line going from end point to start point at angle of 45°.



The end point in the above diagram is the end point of the second straight line with no second beveling and rounding.

4. When G111 block is executed on a single block basis, the movement up to the end point is performed assuming the maximum of four blocks.
5. Writing G111 block in the buffer, the control unit performs all computations for the first and second straight lines and the first and second beveling and rounding. For some contours, the computation time requires more than 500 msec. If the move time in the preceding block is shorter than the computation time, the movement stops, sometimes causing undesirable effects on the cutting surface. To prevent the stop of movement due to the computation time, it is recommended to provide the buffering state (M93 command) before the four blocks before specifying G111 block command.
6. List of alarms caused by incorrect G111 command

Table 2.38 Data Setting Range

Item		Metric output (screw)		Input output (screw)	
		Metric input	Inch input	Metric input	Inch input
Least input increment		0.001 or 0.01 mm	0.0001 or 0.001 in	0.001 or 0.01 mm	0.0001 or 0.001 in
Tool offset		0 - ±8388.607 mm	0 - ±330.2601 in	0 - ±999.999 mm	0 - ±838.8607 in
Tool radius		0 - ±99.999 mm	0 - ±9.9999 in	0 - ±99.999 mm	0 - ±9.9999 in
Minimum step/handle feed		0.001 mm	0.0001 in	0.001 mm	0.0001 in
Stored stroke limit area designation unit	Program designation	0.001 mm	0.0001 in	0.001 mm	0.0001 in
	Parameter & setting	0.001 mm		0.0001 in	
Rapid traverse rate Manual jog etc	Upper limit value	24 m/min		2400 inch/min	
2nd reference point coordinate value		0 - ±99999.999 mm		0 - 9999.9999 in	
Backlash compensation value		0 - 255 pulses (Note 1)		0 - 255 pulses	

Notes 1 1-pulse = least input increment
2 X-axis designated with diameter (except for pulse display)

6. List of alarms caused by incorrect G111 command

Table 2.39

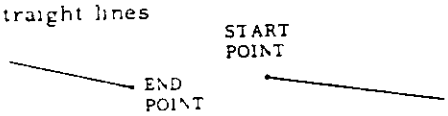
Alarm Code	Cause
140	Commanding one address of addresses B, X(U), Z(W) specifying second straight line
140	Commanding two addresses of addresses B, X(U), Z(W) specifying second straight line. In addition to this, one or no address commanded among addresses A, I, K specifying first straight line.
140	Address C specifying first beveling and address P specifying first rounding commanded
140	Address D specifying second beveling and address Q specifying second rounding commanded
141	Angle for angle programming A, B by G111 out of range $-360 \leq A, B \leq 360$
142	1st beveling part outside the rectangle composed by start and end points
142	1st beveling portion between 45° straight lines of start to end points and end to start points
143	No intersection between 1st and 2nd straight lines 

Table 2.39 (Cont'd)

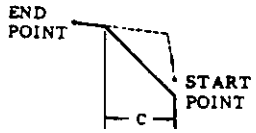
Alarm Code	Cause
143	1st straight line and second straight line on the same line.
144	M, S, T commanded in G111 block
143	Command values for addresses A, I, K specifying first straight line are determined as follows, and programmed shape cannot be formed

Command value for A	
-360 000, -180 000, 0, 180.000, 360.000	Address I commanded for specifying first straight line
-27.000, -90.000, 90.000, 270 000	Address K commanded for specifying first straight line.

Alarm Code	Cause
143	Command values for addresses B, X(U), Z(W) specifying first straight line are determined as follows, and programmed shape cannot be formed

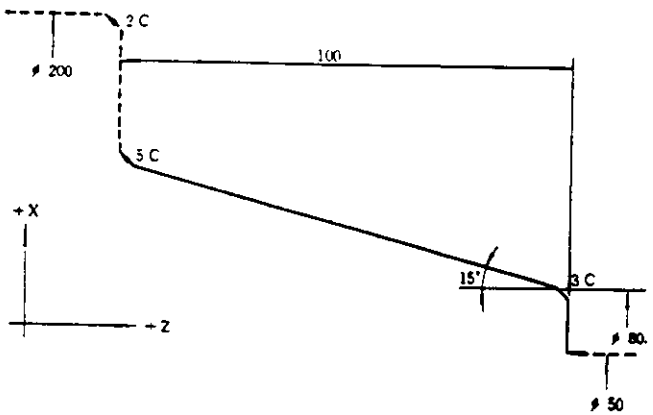
Command value for B	
-360.000, -180.000, 0, 360.000	Address X(U) commanded for second straight line
-270 000, -90.000, 90.000 270 000	Address Z(W) commanded for specifying second straight line.

Table 2.39 (Cont'd)

Alarm Code	Cause
143	Command values for addresses C and D for beveling too large for the programmed shape. Operation cannot be made according to the command. 
143	Command values for addresses P and Q specifying radius for rounding too large for the programmed shape. Operation cannot be made according to the command. 
140	Commanding addresses X and Z specifying second straight line and Q and D specifying second beveling and rounding.

EXAMPLE

a. Taper combined beveling



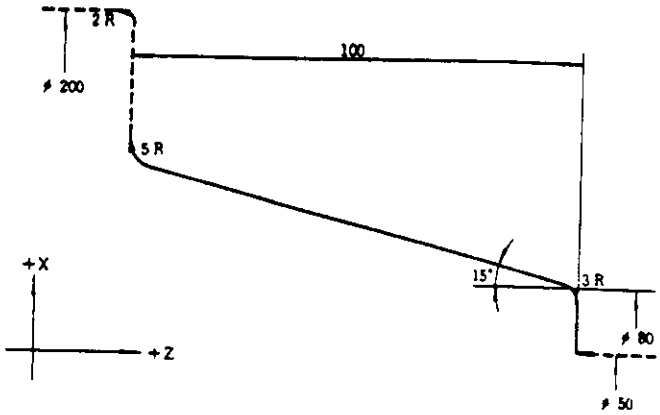
(G01 W··· ;)——— φ50 command shown by broken line

G111 W-100. I15. A90. B165. C3. D5. ;

or G111 W-100. I15. K0 B165. C3. D5. ;

Command shown by solid line

b. Taper combined rounding



(G01 W ·· ;)——— φ50 command shown by broken line

G111 W-100. I15. A90. B165. P3. Q5. ;

or G111 W-100. I15. K0 B165. P3. Q5. ;

Command shown by solid line

(G12) X200. K-2. ;)—Command shown by broken line after the command shown by solid line

2.8.31.2 Circular Arc Multiple Cornering (G112)

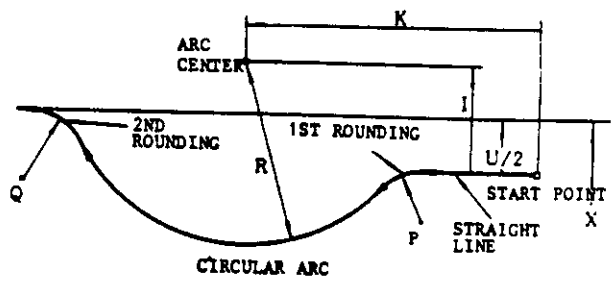
G112 be able to specify the following four operations in a single block:

Straight line → Beveling Rounding → Circular arc → Beveling Rounding

Depending on the direction of arc combined beveling/rounding or taper in turning combined beveling/rounding in facing may be executed.

(1) Cutting configurations and command format

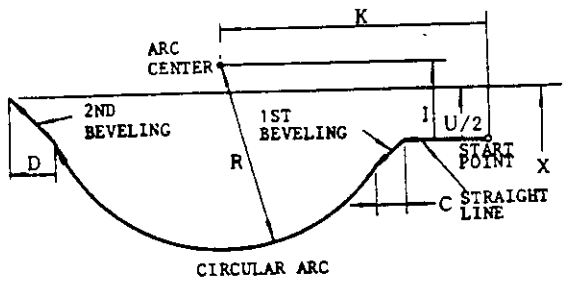
a. Arc combined rounding in turning



G112 X(U)···I ··· K··· P··· Q··· R··· ;

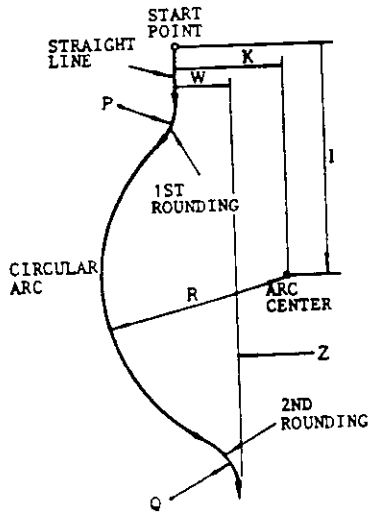
2.8.31.2 Circular Arc Multiple Cornering (G112) (CONT'D)

b. Arc combined beveling in turning



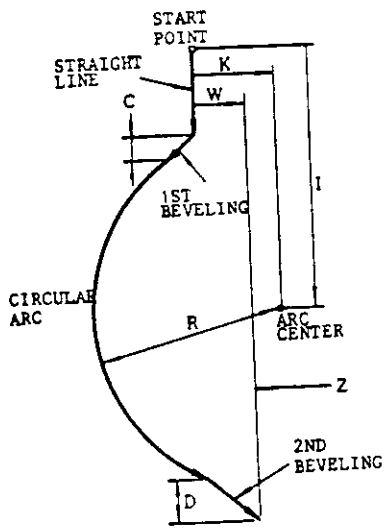
G112 X(U)··· I··· K··· C··· D··· R··· ;

c Arc combined rounding in facing



G112 Z(W)···I ··· K··· P··· Q·· R··· ;

d Arc combined beveling in facing



G112 Z(W) ·· I··· K··· C··· D··· R··· ;

(2) Meaning of addresses

The word addresses for circular arc multiple cornering command are as shown below:

Table 2.40

Address Word	Contents	Unit
X(U)	X-axis end point coordinates for arc combined rounding in turning (U. Increment from start point)	1 = 0.001 mm or 1 = 0.0001 in. (decimal point may be entered)
Z(W)	Z-axis end point coordinates for arc combined rounding in facing (W: Increment from start point)	
I	X-axis distance from arc center start point	
K	Z-axis distance from arc center start point	
R	Circular arc radius	
P	First rounding radius (without sign)	
Q	Second rounding radius (without sign)	
C	First beveling amount (without sign)	
D	Second beveling amount (without sign)	

(3) Designation of contours

a. The contours of the portions to be subjected to circular arc multiple cornering are as shown below:

Table 2.41

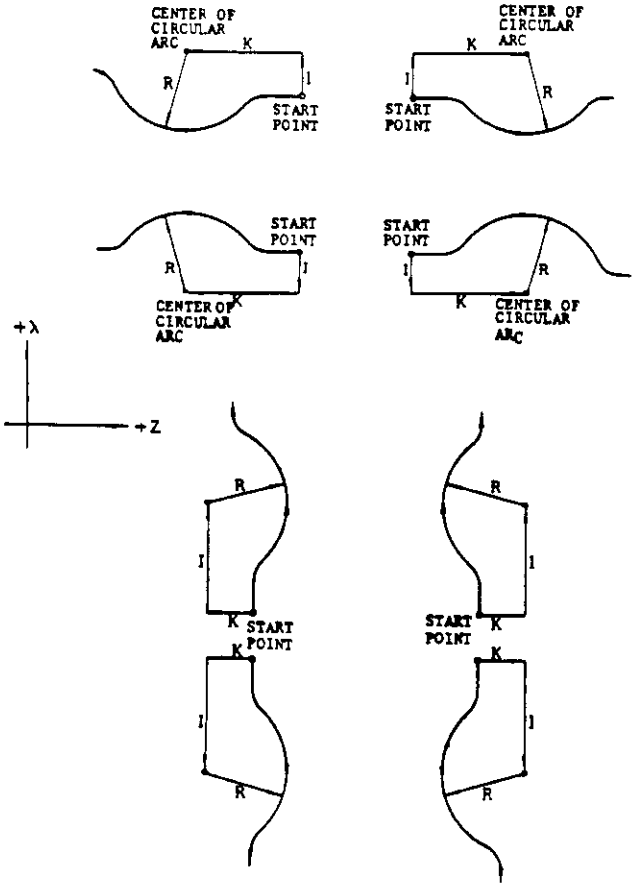
Straight line	The straight line which is parallel to Z-axis (arc in turning) or X-axis (arc in facing) from the start point.
Circular arc	The point from which the arc is circulated is designated by I and K commands from the start point.
First beveling	The beveling which is performed in the dimensions designated by C command at the intersection of straight line and circular arc

Table 2.41 (Cont'd)

First rounding	The rounding which is performed in the radius designated by P command in contact with straight line and circular arc
Second beveling	The beveling which is performed in the dimensions designated by D command at the intersection between the circular arc and the straight line parallel to Z-axis designated by X(U) command (circular arc in turning) or the straight line parallel to X-axis designated by Z(W) command (arc in facing)
Second rounding	The rounding which is performed in the radius designated by Q command in contact with the circular arc and the straight line parallel to Z-axis designated by X(U) command (circular arc in turning) or the straight line parallel to X-axis designated by Z(W) command (arc in facing)

b. Circular arc cutting directions

As shown below, the circular arc cutting rotational direction is determined so that the circular arc is reversed along the straight line from the start point.

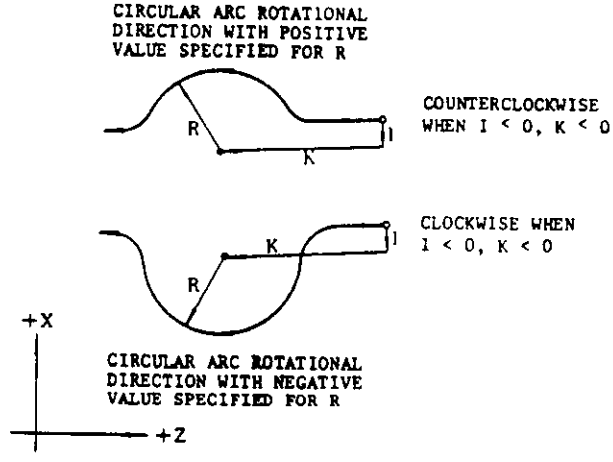


c. To supplement the above description, the following discriminants for determination of the circular arc cutting directions in the control unit are provided:

Table 2.42

Values of I and K Commands	Circular Arc Rotational Direction	
	Circular arc in turning	Circular arc in facing
I ≥ 0, K ≥ 0	Counterclockwise (CCW) (Equivalent to G03)	Clockwise (CW) (Equivalent to G02)
I ≥ 0, K < 0	Clockwise (CW) (Equivalent to G02)	Counterclockwise (CCW) (Equivalent to G03)
I < 0, K ≥ 0		
I < 0, K < 0	Counterclockwise (CCW) (Equivalent to G03)	Clockwise (CW) (Equivalent to G02)

When a negative value is specified for the circular arc R, the circular arc cutting direction mentioned above may be inverted as follows:



d. Address words X(U) and Z(W) are used to make discrimination between radius measuring circular arc and front circular arc. They cannot be omitted if the end point and the start point are on the same coordinates ("U0" or "W0" should be specified). When the address words other than X(U) and Z(W) are omitted, the following results are obtained:

2.8.31.2 Circular Arc Multiple Cornering (G112) (CONT'D)

Table 2.43

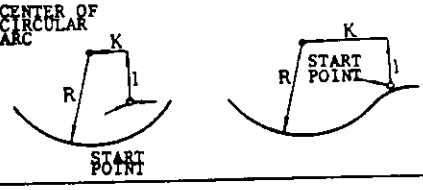
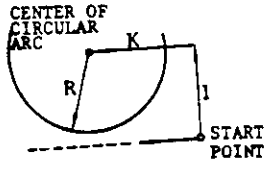
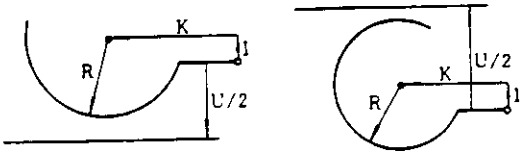
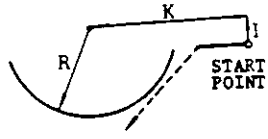
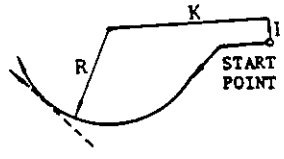
Address Word	Result
I	"I0" command is provided.
K	"K0" command is provided.
R	"R0" command and alarm are provided.
P Q C D	"0" command is provided. Beveling and rounding are not performed

NOTES.

1. G112 is a nonmodal G code and is valid only for the specified block.
2. The block specified with G112 does not allow the specification of M, S, and T.
3. When G112 block is executed on a single block basis, the movement up to the end point is performed assuming the maximum of four blocks.
4. When G112 is used for the finishing shape blocks G71 (stock removal cycle in turning), G72 (Stock removal cycle in facing), and G73 (pattern repeating) of multiple repetitive cycles the block specified with G112 is equivalent to five blocks.
5. Do not specify other codes to the G112 block; otherwise, an error will be caused.
6. After wiring G112 block to the buffer, the control unit performs all computations for the straight line, the circular arc, and the first and second beveling and rounding. For some contours, the computation time becomes more than 500 msec If the move time in the preceding block is shorter than the computation undesirable effects on the cutting surface. To prevent the stop of movement due to the computation time, provide the buffering state (M93 command) before the four blocks before specifying B112 block command.
7. List of alarms to be given by G112 command error

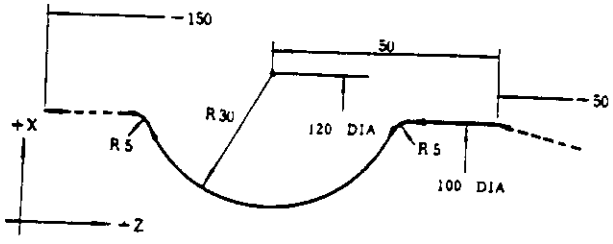
Alarm Code	Cause
145	X(U) or Z(W) not specified
145	Both X(U) and Z(W) specified
145	R not specified Or "0" specified.
145	I and K not specified Or "0 specified for both

Table 2.44

Alarm Code	Cause
145	Both P and C specified.
145	Both Q and D specified.
144	M, S, or T specified.
146	Tool moves in the direction reverse to the center of circular arc from the start point. 
146	There is no intersection between circular arc and straight line. 
146	There is no intersection between circular arc and end point command. 
146	Beveling specified by C command cannot be performed. 
146	Beveling specified by D command cannot be performed. 

EXAMPLE:

a. Arc combined rounding

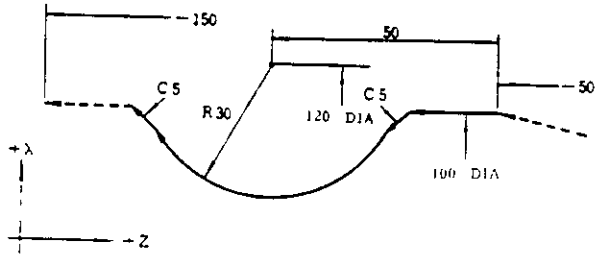


(G01 X100. Z-50. ;) ··· Shown by broken line before circular arc

G112 U0I10. K-50. P5. Q5. R30. ;

(G01 X-150. ,) ··· Shown by broken line after circular arc

b. Arc combined beveling



(G01 X100. Z-50. ;) ··· Shown by broken line before circular arc

G112 U0I10. K-50. C5. D5. R30. ,

(G01 Z-150. ;) ··· Shown by broken line after circular arc

2.8.32 ABSOLUTE/INCREMENTAL PROGRAMMING (G90, G91)

G code	Meaning
G90	Absolute designation
G91	Incremental designation

For the details of the G codes, see 2.3.5 Absolute and Incremental Inputs.

3. NC TAPE PUNCHING

3.1 TAPE CODE

3.1.1 LIST OF TAPE CODE

Both EIA code and ISO code are available for punching a paper tape.

EIA code (EIA RS-244-A).
ISO code (ISO R840).

Punching patterns according to these codings are shown in Table 3.1.

Before programming, select the code to be used. EIA and ISO codes cannot be punched mixedly through a tape.

3.1.2 EIA/ISO AUTO RECOGNITION

The control performs automatic recognition of EIA/ISO code. It recognizes the code punched on tape by reading the first EOB code in Label Skip state, and automatically adjusts the follow on data to read by the recognized code. RESET operation[1] activates Label Skip state and cancels this function.

NOTE: The setting #6000D7 can specify the code when NC internal data is outputted (punched out), and does not affect the tape reading operation

#6000D7	Meaning
= "0"	Code output
= "1	Code output

3.2 PROGRAMMING

3.2.1 PROCESS SHEET

The programming is performed with the process sheet. It is recommended that the process sheet to match final specifications should be made by users, considering the readily perceived form and convenience for rewriting. Fig. 3.2.1 shows an example of the process sheet.

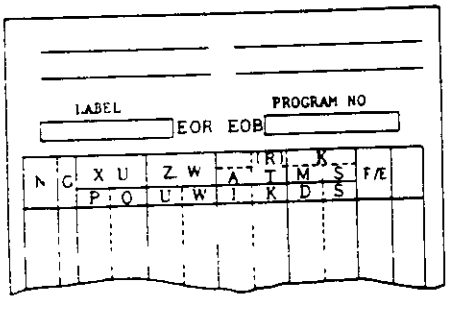
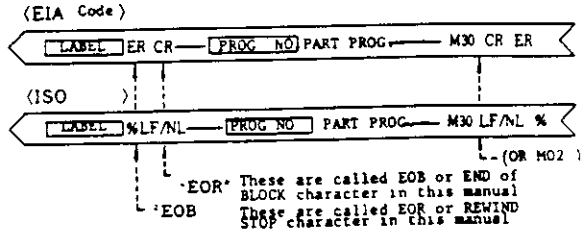


Fig. An Example of the Process Sheet

3.2.2 GENERAL PROGRAM FORM

(1) A part program will be generally made in the following form.



(2) Any LABEL can be written at the beginning of tape to classify easily the tapes. In label skip function the control ignores the data from LABEL to the first EOB code. Therefore, the undesignated address or function characters can be used as LABEL. In addition, the modified code which disregards parity is also available.

(3) EOR code next to LABEL means the stop point of tape rewinding.

(4) Where storing NC tape data into memory, with the label skipped, the memory stores the data between the first EOB code and the next EOR code. Therefore, EOR code at the end of tape must not be omitted.

PRECAUTION IN PROGRAMMING

(1) A block ends with EOB (End-of-Block) character. EOB character is represented by CR in EIA code and LF/NL in ISO code. In this manual, mark , is substituted for them to read easily this manual.

(2) A part program ends with the block including M02 (End-of-Program), M30 (End-of-Tape) or M99 (End-of-Program).

(3) When M02 or M30 is commanded, automatic operation[2] is stopped. In most cases, the control is reset, or rewinds the tape (or memory) automatically. As the details are determined by the machine, refer to the machine tool builder's manual.

1 RESET operation means resetting the control by depressing the RESET key on the operator's control station or remotely.
2 "Automatic operation" means operation in TAPE, MDI, or MEM mode.

Table 3.1 Tape Code

EIA CODE								CHARACTERS	ISO CODE								
8	7	6	5	4	0	3	2	1		8	7	6	5	4	0	3	2	1
		○			●				0			○	○		●			
					●			○	1	○		○	○		●			○
					●		○		2	○		○	○		●		○	
			○		●		○	○	3			○	○		●		○	○
					●	○			4	○		○	○		●	○		
			○		●	○		○	5			○	○		●	○		○
			○		●	○	○		6			○	○		●	○	○	
					●	○	○	○	7	○		○	○		●	○	○	○
				○	●				8	○		○	○	○	●			
			○	○	●			○	9			○	○	○	●			○
	○	○			●			○	a	A		○			●			○
	○	○			●		○		b	B		○			●		○	
	○	○	○		●		○	○	c	C	○	○			●		○	○
	○	○			●	○			d	D		○			●	○	○	
	○	○	○		●	○		○	e	E	○	○			●	○	○	○
	○	○	○		●	○	○		f	F	○	○			●	○	○	
	○	○			●	○	○	○	g	G		○			●	○	○	○
	○	○		○	●				h	H		○		○	●			
	○	○	○	○	●			○	i	I	○	○		○	●			○
	○		○		●			○	j	J	○	○		○	●		○	
	○		○		●		○		k	K		○		○	●		○	○
	○				●		○	○	l	L	○	○		○	●	○		
	○		○		●	○			m	M		○		○	●	○		○
	○				●	○		○	n	N		○		○	●	○	○	
	○				●	○	○		o	O	○	○		○	●	○	○	○
	○		○		●	○	○	○	p	P		○		●		●		
	○		○	○	●				q	Q	○	○		●		●		○
	○			○	●			○	r	R	○	○		●		●	○	
		○	○		●		○		s	S		○		●		●	○	○
		○			●		○	○	t	T	○	○		●		●		
		○	○		●	○			u	U		○		●		●		○
		○			●	○		○	v	V		○		●		●	○	
		○			●	○	○		w	W	○	○		●		●	○	○
		○	○		●	○	○	○	x	X	○	○		●	○	●		
		○	○	○	●				y	Y		○		●	○	●		○
		○		○	●			○	z	Z		○		●	○	●	○	
					●				Blank	NUL						●		
		○		○	●		○		BS		○				○	●		
		○	○	○	●	○	○		Tab	HT					○	●		○
○					●				CR	LF/NL					○	●	○	
			○		●				—	CR	○				○	●	○	○
				○	●				SP		○		○			●		
				○	●		○	○	ER	%	○		○			●	○	○
	○	○	○	○	●	○			UC	—								
	○	○	○	○	●			○	LC	—								
*									—	(			○		○	●		
*									—	)	○		○		○	●		○
	○	○	○		●					+			○		○	●	○	○
	○				●					−			○		○	●		○
	○				●	○	○	○	●	:			○	○	○	●		
		○	○		●			○		/	○		○		○	●	○	○
	○	○	○	○	●	○	○	○	Del	DEL	○	○	○	○	○	●	○	○
○	○	○	○	○	●	○	○	○	All Mark		○	○	○	○	○	●	○	○
See Note 2.									#		○		○			●		
*									*		○		○					
*									=									
*																		
*																		
									$									
									@									
									?									
									.									

Notes:

1. For the hole pattern of EIA code of the characters with an asterisk, the pattern shown in the table is standard. However, other patterns may be specified by parameters.
2. EIA code of character # can be designated by the parameter #6017.

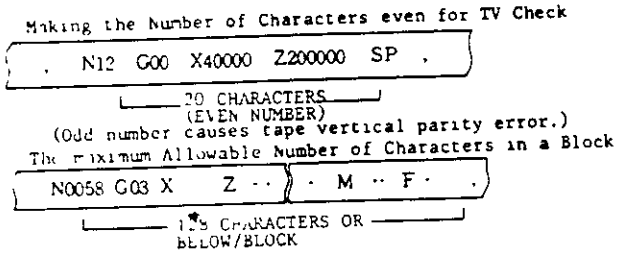
3.2.2 GENERAL PROGRAM FORM (CONT'D)

(4) The character specified on 2.1.2 Address and Function Characters should be used for programming, but others should not.

(5) Where the tape vertical parity check (TV check) is made, number of characters in a block must be even. If odd, it should be made even by using "SP" character.

(6) The disregarded characters such as "BS, Tab, SP, UC, LC and Del" should be avoided from the significant data area, if unnecessary.

The maximum allowable number of characters in a block is 128. The disregarded characters such as "Del, BS and Tab" are not included in them.



(7) SP (Space) character

SP character is usually disregarded when tape data is read in. However, in the following cases, SP is read and its function for providing space on the CRT screen is effective

a. "SP" used in parentheses.

b. "SP" programmed after two or more characters like "DO," "GOTO" in user macro body.

EXAMPLE

GO TO SP 100 ,

"SP"

3.2.3 TV CHECK (TAPE VERTICAL PARITY CHECK)

When the tapes are to be checked for vertical parity, programs must be so made that each block (including EOB) contains even number of characters. Normally, SP codes are used to make the number of characters even

The TV check function is turned on and off by the setting function. While the TV check function is on, all blocks containing odd number of characters are regarded as errors.

Setting No. · #6006D_6 = 0 ·· TV check off

#6000D_6 = 1 ··· TV check on

3.3 NC TAPE

3.3.1 PAPER TAPE

Eight-channel paper tape for computers complying with JIS[1]-6243 is used as standard. The dimensions are 25.4 ±0.08 mm (1 inch) width and 0.108 mm (0.0042 inch) thickness.

It is recommended that the color of the tape is black or gray, but not that of high transparency. If the tape with high transparency is used, the tape reader may misread it.

3.3.2 NC TAPE PUNCH

NC tape must be punched out with the tape punch er for EIA code or ISO code according to contents of process sheet.

When punching the tape, at the beginning and the end of the tape, provide the feed holes part needed for the tape feeding. Where the punched tape is wound on the reel† of tape reader, the feed holes part will be 70 cm in length.

3.3.3 NC TAPE CHECK

NC tape can be checked by using the following function.

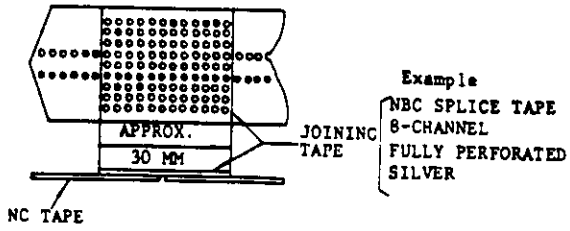
- Machine lock
- M function lock†
- Dry run
- Single block operation

3.4 NC TAPE HANDLING

3.4.1 SPLICING NC TAPES

To splice NC tapes, stick a joining tape (0.08 mm thickness) with sprocket holes, or fully perforated joining tape on the one side of the spliced NC tape. Before using the spliced NC tape, make sure that the sprocket holes are in position. The joining part of tapes should not be extremely thick, and do not use the rigid adhesive agent without flexibility.

[1] Japanese Industrial Standard



Splicing of NC Tape Tape

3.4.2 KEEPING OF NC TAPE

For life expectancy of NC tape, the following handling is recommended.

- When keeping NC tape, avoid moisture and oil.
- Do not handle the tape with oil-stained gloves.

Properly kept tapes will permit 300 times of reading and rewinding.

4. STANDARD NC OPERATOR'S STATION WITH CRT CHARACTER DISPLAY

4.1 PUSHBUTTONS, KEYS, AND LAMPS

Fig. 4.1 shows an overall view of NC operator's station with CRT display. The names and functions of operator devices are as follows.

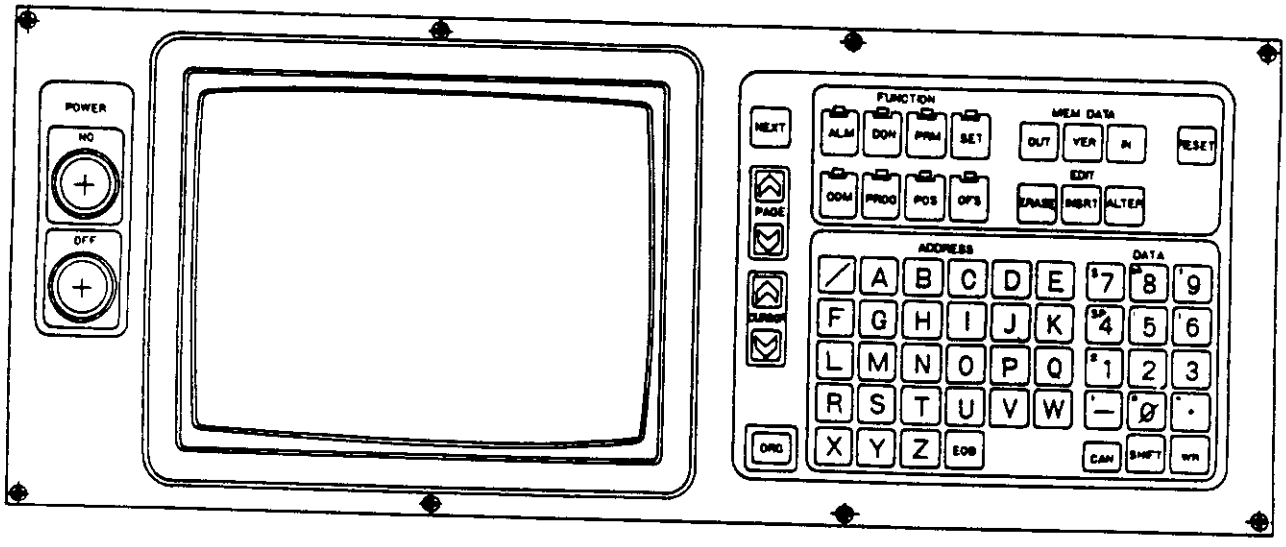


Fig. 4.1 Standard NC Operator's Station with CRT Character Display

4.1.1 POWER ON/OFF PUSHBUTTONS

- POWER ON pushbutton

To turn on the power for the control: Depress the pushbutton first to turn on the control power and depress it again to turn on the servo power. Push this button to recover the servo power after an emergency stop.

- POWER OFF pushbutton

To turn off the power for the control: Depress it to turn off both the servo and control powers.

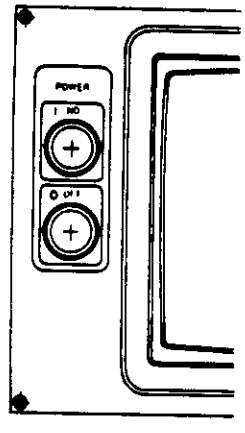


Fig. 4.2

4.1.2 CRT CHARACTER DISPLAY

According to each operation, this display indicates the alpha-numerical data in a regular size, double-size and quadruple-size of the regular size.

Braun tube size 9 inches

Maximum number of characters:
32 characters x 16 lines =
512 characters (at regular size)

Indicating characters.

Numerals - [0] through [9], [-], [.], [ ]

Alphabetic characters - [A] through [Z]

Special code - [;] (EOB), [/] (slash), etc.

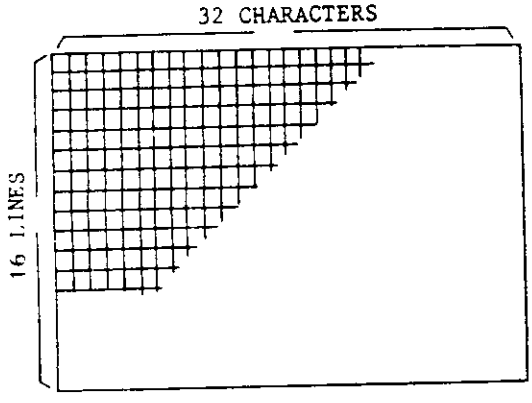


Fig. 4.3 Braun Tube

4.1.3 FUNCTION KEYS

The key selects one of eight functions for the operation of the display and MDI. Pushing a key makes it effective and light up.

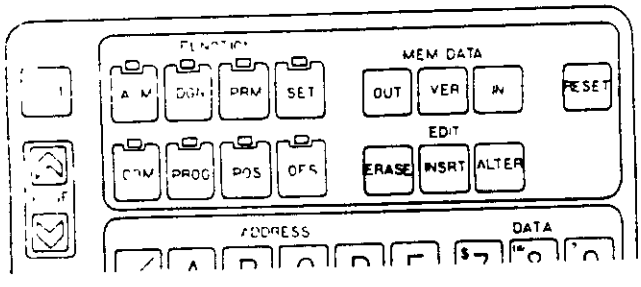


Fig. 4.4

- ALM (Alarm) key

Select this key for display of alarm and status codes. The function becomes effective when the power is turned on or an alarm occurs.

- DGN (Diagnosis) key:

Select this key for display of input/output signal status.

- PRM (Parameter) key:

Select this key for display or writing-in of parameters.

- SET (Setting) key:

Select this key for display or writing-in of setting data.

- COM (Command) key.

Select this key for display or writing-in (MDI) of the command data for automatic operation.

- PROG (Program) key:

Select this key for display or writing-in of a part program.

- POS (Position) key:

Select this key for display of various current positions.

- OFS (Offset) key.

Select this key for display or writing-in of tool offset values.

4.1.4 ADDRESS KEYS

These keys are to designate an address character when writing in various data.

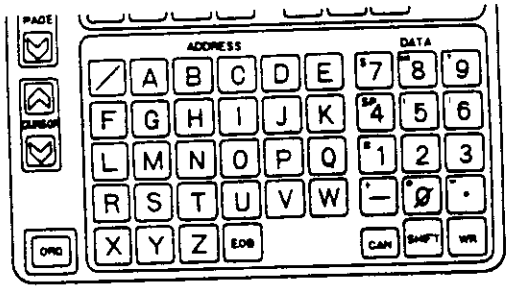


Fig. 4.5

Note: Special characters

[/] (Slash) key: For an optional block skip command.

[EOB] (EOB) key: For the block end command. On the CRT display, ";" is displayed instead of "EOB."

4.1.5 DATA KEYS

These keys consist of 15 keys in total, such as [0] through [9], [-] (minus) [.] [CAN] [SHIFT] [WR], and can be used for writing-in of such all numeral values as tool offset value setting data, parameter data, and so on, in addition to command value.

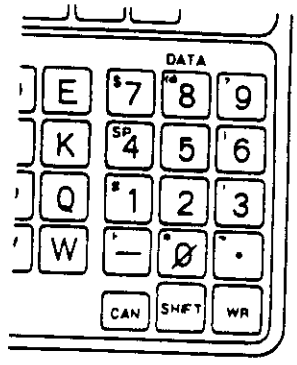


Fig. 4.6

Notes

[*0] to [?9] key
[+-] (minus) key : For input of numerical data

[=.] (decimal point) key : For input of decimal point

[CAN] (Cancellation) key:
For cancellation of the numeric value or address data erroneously keyed.

[WR] (write) key:
For storing address data by address keys and data keys into buffer storage.

[SHIFT] (shift) key.
Depressing [SHIFT] key after depressing [*0] to [?9], [-] or [.] key makes the display turn into [*] to [?], [+], [=] which are written on the upper left corner of the keys. These special characters are used in user macro.

4.1.6 NEXT KEY

The NEXT key is used for special purpose and expanding function in display or writing data.

- Writing of additional tape in EDIT mode.
- Display of specified number in DGN function.
- For other special purpose and expanding function.

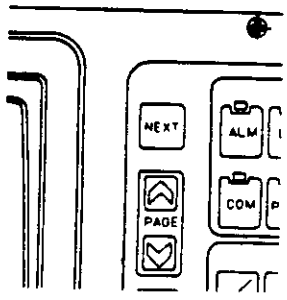


Fig. 4.7

4.1.7 PAGE KEYS

The PAGE key is used to display the next page or the previous page when CRT display is regarded as page.
For example, when a bundle of tool offset values are displayed by OFS key, this key is pushed to display the next bundle of tool offset values, which just looks like opening the pages of a book.

(1) Depressing [PAGE ↓] key displays the next page.

(2) Depressing [↑ PAGE] key displays the previous page.

(3) Keeping the PAGE key depressed makes the page step automatically forward or backward.

4.1.8 CURSOR KEYS

The CURSOR control key is used to move the cursor. For example, when a page of parameter data are displayed by PRM key.

(1) Depressing [CURSOR ↓] key moves the cursor backward.

(2) Depressing [↑ CURSOR] key moves the cursor forward.

(3) Keeping the cursor control key depressed makes the cursor move automatically forward or backward.

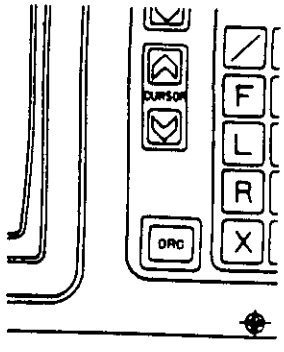


Fig. 4.8

4.1.9 ORG (ORIGIN) KEYS

The ORG key is used to set the current position of the machine tool as the zero point of coordinate system.
The origin setting can be made for each axis.
The reference coordinate system means the coordinate system which is set by G92 command or the automatic coordinate system setting.

ORG key is used for the following operation.

- Reset of current position (Position External/ Absolute)
- Reset of operation time

4.1.10 EDIT KEYS

These keys are for editing a stored part program.

ERASE key. Used for erasure of data in storage.

INSRT key: Used for insertion of data in memory.

ALTER key Used for alteration of data in memory.

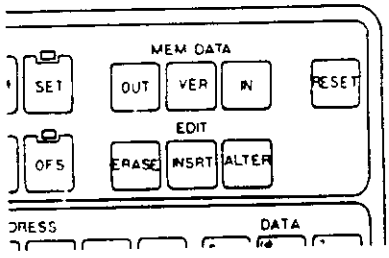


Fig. 4.9

4.1.11 MEM DATA (MEMORY DATA) KEYS

TAPE KEYs are to start the tape operation except in the automatic operation mode. They are effective only in the EDT mode.

(1) OUT key

This key is to start outputting various data in memory through data I/O interface.

(2) IN key

This key is to start storing various data into memory through tape reader or data I/O interface.

(3) VER key

This key is to start verifying between memory data and punched tape data.

4.1.12 RESET KEY

This key resets the control.

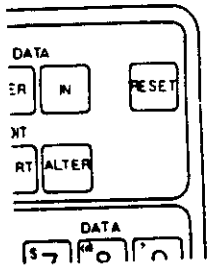


Fig. 4.10

Operations to be executed by this RESET key are.

- Move command cancel
- Buffer register clear
- Alarm code release if the cause is eliminated
- Tool offset cancel
- Auxiliary function cancel
- Label skip function ON
- Memory pointer rewind
- Sequence number reset
- RST signal transmission
- Resetting G codes

Refer to 2.8 1 List of G Codes and Groups.

The following will not be affected by operating the RESET key.

- Current position values of each axis.
- F commands
- S, T and B commands
- Tool offset values, setting data, parameter data

NOTE Depressing the RESET key or the remote reset pushbutton is defined as "Reset operation" in this manual.

4.1.13 TAPE FEED AND SYSTEM NO. SWITCHES

These switches are mounted above the tape reader.

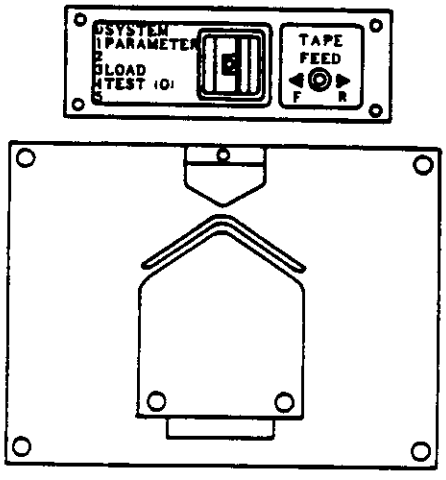


Fig. 4.11

(1) TAPE FEED switch

This is a switch to wind and rewind the tape manually. Setting the switch to F (forward) causes the tape to feed. To rewind the tape set the switch to R (reverse). This switch is effective, either manually or automatically.

(2) SYSTEM NO. switch

Set the switch at "0" during the usual operation. Functions of its each setting are as follows.

"0": SYSTEM

For usual operation. Writing parameters is prevented.

"1": PARAMETER

To write parameters. At this position, the Cycle Start is prevented.

"3": LOAD

To store the maintenance tape into the control.

"4": TEST (0)

The usual operation is similar to case of "0" SYSTEM. Self-diagnostics of the memory contents and checking of reference zero return point are omitted.

4.2 POWER ON/ OFF OPERATION

4.2.1 TURNING ON POWER

Check the machine before turning on power, referring to the machine tool builder's manual for details. Operations after completion of inspections are as follows.

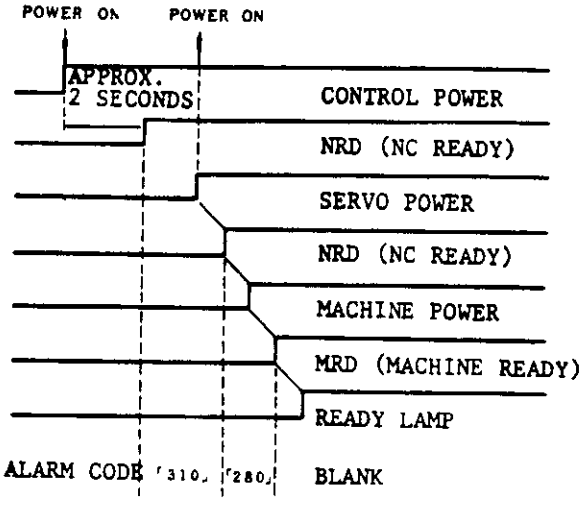


Fig. 4.12 Sequence of Turning on Operation

(1) Depressing the POWER ON pushbutton to turn on the control power. The internal timer will be read in about two seconds. Then the servo power is ready for turning on, which is shown by alarm code "310."

(2) Depress the POWER ON pushbutton again to turn on the servo power The NRD (NC READY) signal is sent out when the NC power is normally supplied.

(3) When the NRD signal turns on the machine power, and the MRD (MACHINE READY) signal returns back to the control, the READY lamp will be lit.

4.2.2 TURNING OFF POWER

Depressing the POWER OFF pushbutton causes both the servo and control powers to be turned off simultaneously. However, for more stable operation, use the following procedure.

(1) First depress the EMERGENCY STOP pushbutton to cut off the servo power. The NRD (NC READY) signal is interrupted, which usually results in turning the machine power, too.

(2) Depress the POWER OFF pushbutton to cut off the control power.

4.2.2 TURNING OFF POWER (CONT'D)

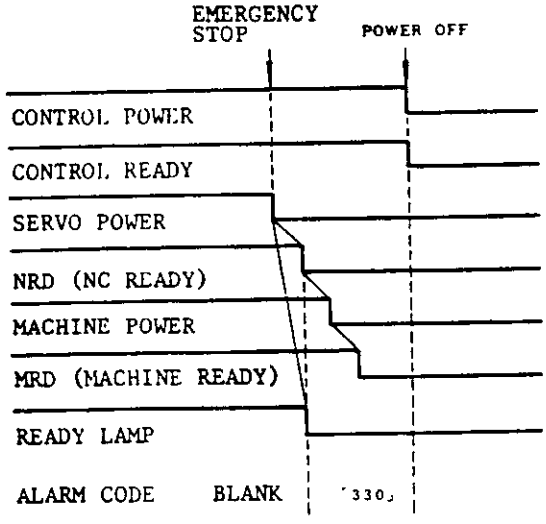


Fig. 4.13 Sequence of Turning off Operation

4.2.3 REMOTE POWER ON/OFF PUSHBUTTONS

Connect the power ON/OFF pushbuttons to EON, EOF and COM terminals on the control panel as shown below. Then the remote turning ON/OFF operation can be made in exactly the same way as with the POWER ON/OFF pushsuttons.

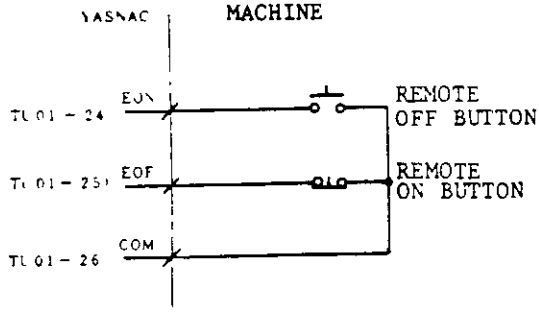


Fig. 4.14 Connections of Remote ON/OFF Pushbuttons

4.3 DISPLAY AND WRITING OPERATION

4.3.1 CONSTANT DISPLAY

The following display is made on both the top and bottom on the displayed picture of CRT, irrespective of the FUNCTION key currently selected

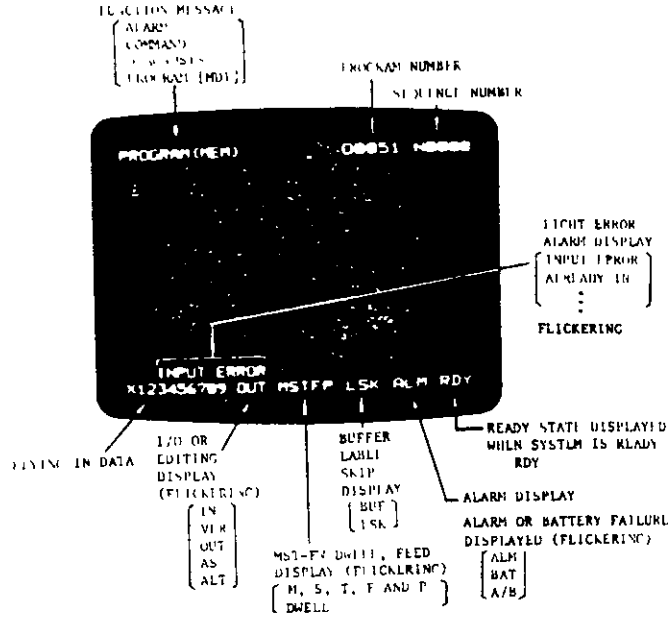


Fig. 4.15

(1) Function message

Anyone of the following eight function messages corresponding to the function key is displayed at the top of CRT display.

ALARM	COMMAND
DIAGNOSIS	PROGRAM
PARAMETER	POSITION
SETTING	OFFSET

(2) Program No.

O and 4 digits of program No. under execution is constantly displayed at the top of CRT display irrespectively of function key.

(3) Sequence No.

N and 4 digits of program No. under execution is constantly displayed at the top of CRT display irrespectively of function key.

(4) Display of keying data

Up to 32 characters of keyed in data can be displayed at one time. The data is processed by using ERASE key, INSRT key, ALTER key, etc.

(5) Display of I/O and editing (flickering)

The following messages are flickerlingly displayed during loading of punched tape, address search or edition.

"IN" ··· loading tape

"VER" ··· verifying tape

"OUT" ··· punching tape out

"AS" ··· searching address

"ALT" ··· altering data in EDIT mode

"INS" ··· inserting data in EDIT mode

"ERS" ··· erasing data in EDIT mode

6) Display of MST-FIN signal waiting, dwelling and feeding

"M" ··· waiting for FIN signal of M command

"S" ··· waiting for FIN signal of S command

"T" ··· waiting for FIN signal of T command

"F" ··· feeding
"R" is displayed at rapid traverse

"P" ··· loading tape

"DWELL"
··· dwelling

M, S, T, F and P are displayed independently each other.

7) Display of the state of buffer full and label skip

"BUF" ··· displayed at completion of advance reading

"LSK" ··· displayed at label skip on

8) Display of alarm (flickering)

Alarm continues to be displayed flickeringly until the cause is removed and RESET operation is made.

"ALM" ··· indicates alarm state occuring

"BAT" ··· indicates battery alarm occuring

"A/B" ··· indicates both of alarm and battery alarm occuring

9) Display of ready state

"RDY" ··· indicates the system is normal and the control is operable.

10) Display of light errors (flickering)

The messages shown below indicate light errors which occur in keying or searching operation. Differing from the alarm codes, these error messages are cleared by depressing some key (Generally CAN key)

"INPUT ERROR!" ···Format error of keyed-in data

"ALREADY IN!" ···The same number of part program has been stored already.

"EDIT LOCK ON!" ···Editing operation is made with Edit Lock on

"MEMORY OVER!" ···Part program to be stored is beyond memory capacity

"PROGRAM OVER!"
···Registered number of part program is beyond 99 (basic) or 199 (option)

"NOT FOUND!" ···Desired data has not been located.

"BREAK POINT!" ···Break point occurs.

4.3.2 COMMAND DATA DISPLAY

(1) Depress COM key.

Anyone of the following three digits appears.

a. Command data (COMMAND)

b. Repetition number of subprogram (SUB PROG. NESTING)

c. State of tool life control (TOOL LIFE CONTROL)

(2) The above display steps forward or backward by depressing PAGE ↓ or ↑ PAGE one by one.

4.3.2.1 Command Data Display

The display shows the block data under execution or just prior to execution in which compensation calculations have been completed. The conditions of the data to be displayed is as follows.

(1) The data shows the contents of the active register during an automatic operation or a feed hold.

(2) While the control is stopped at a block end, the contents of the buffer regiter are displayed. If the buffer register blank (BUF is not displayed), the contents of the just executed block are displayed.

4.3.2.1 Command Data Display (Cont'd)

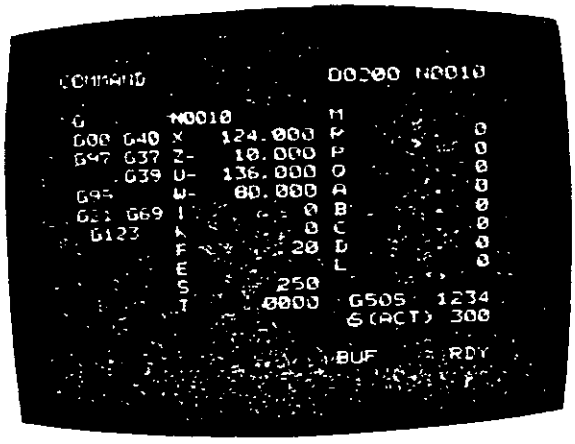


Fig. 4.16 Command Display

4.3.2.2 Display of Subprogram Run Status (SUB PROG. NESTING)

When the program being executed is in the subprogram called by M98 (subprogram call command), the following information is displayed:

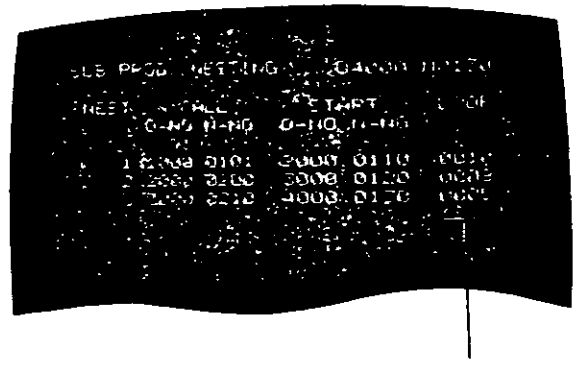
CALL The program number (O to NO) and sequence number (N to NO) specified with M98 (subprogram call command)

START. The program number (O to NO) and number (N to NO) of the subprogram called by M98

LOOP: The remaining number of repetitions of the subprogram by L (subprogram repeat command, indicates the number of repetitions)

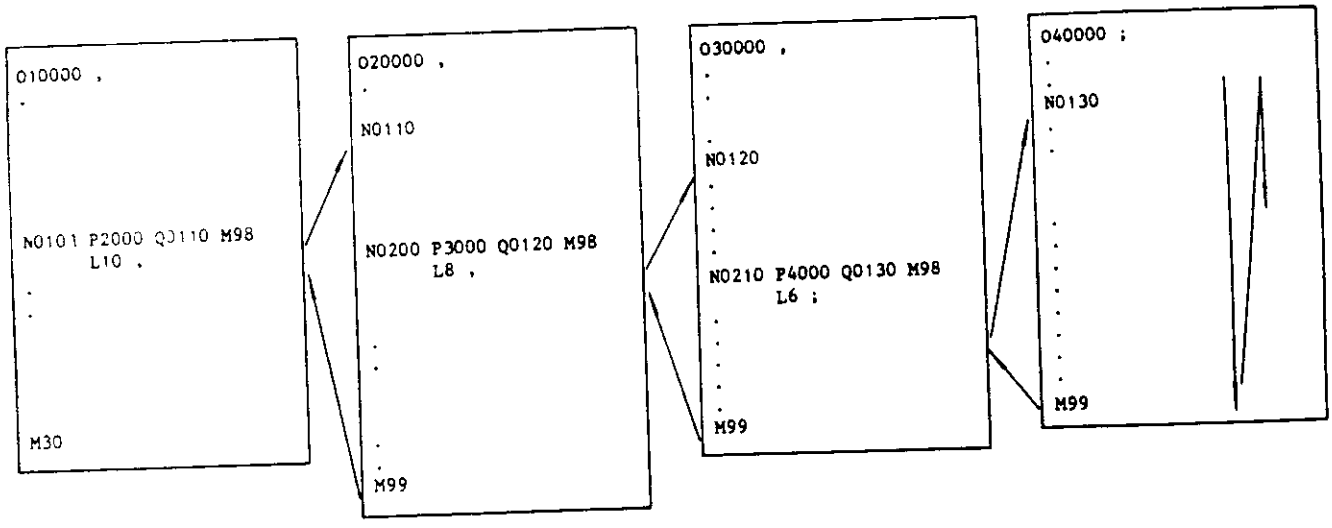
NEST. The order inwhich subprogram multiple call commands are called

Example of Subprogram Run Status Display



Subprogram Remaining Number of Repetitions

Fig. 4.17



CRT screen displays that the subprogram has executed the 3rd level one time and entered into the execution of 3rd time of the 3rd level

4.3.2.3 Display of Tool Life Control Use Status (TOOL LIFE CONTROL)

The following information is displayed for the status (e.g. condition, use) of each tool in each tool group under the tool life control feature:

LIFE: The life of the tool group displayed on the screen

Groups 1 to 9: The number of machining operations.

Groups 10 to 19: Machining time.

COUNTER/TIMER: The number of machining operations/machine time of the currently used tool.

T codes corresponding to the tools registered in the group displayed on screen are all shown.

T□□**; END } The tool whose life has expired.
T□□**; END }

T□□**; CUTTING ← The tool currently used

T□□**; } The tool to be used.
.
.
.
T□□**;

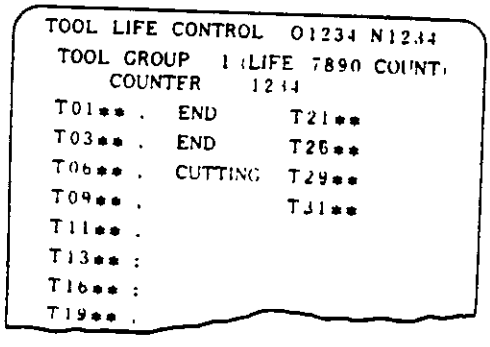


Fig. 4.18 Example A of Tool Life Control Use Status

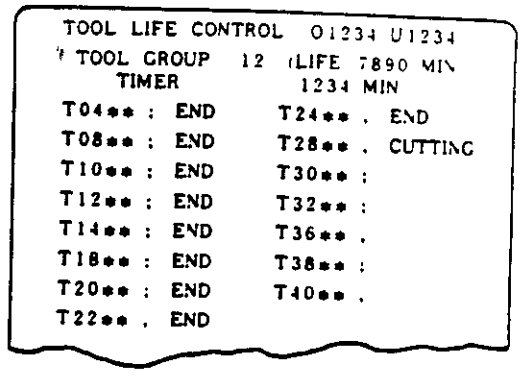


Fig. 4.19 Example B of Tool Life Control Status

4.3.3 WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI

4.3.3.1 Multi-block Writing and Operation in MDI Mode

(1) Multi-block writing in MDI mode

By the following operations, a maximum of 10 lines of data may be written in MDI mode.

a. Select MDI mode.

b. Depress the PROG function key.
··· PROGRAM (MDI) is displayed on the screen.

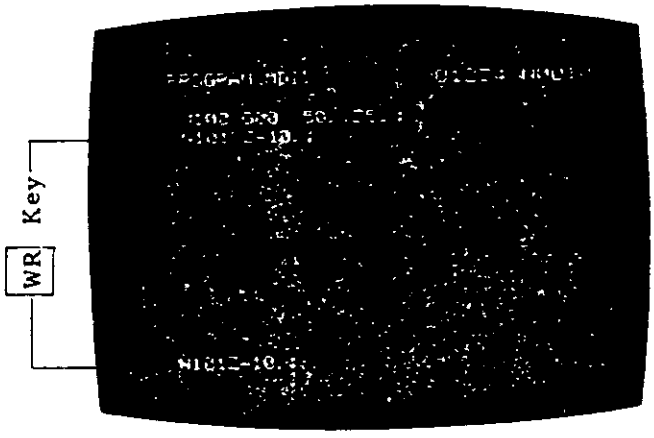
c. Depress the RESET key.
The buffer for MDI is emptied.

d. The part program is written by the use of the address key and data key. As shown below, the keyed data is written to the bottom line on CRT screen from left to right. The maximum number of characters that can be written at a time is 32. If the data is comprised of 32 characters or less, it may be keyed in over multiple words or blocks. However, when the 10th character is keyed in, the normal display at the right of this line is blanked.

e. Depress the WR key.
The keyed data is stored in the MDI buffer. The blanked display is restored to normal.

f. Up to 10 lines of the part program for MDI operation may be written by repeating the operations in d. and e. above.

4.3.3.1 Multi-block Writing and Operation in MDI Mode (Cont'd)



Refered to as "the data which has just been entered" (32 characters maximum).
Enter [N], [1], [0], [1], [2], [-], [1], [0], [.], [EOB] in this order.

Note: The depression of the EOB key displays ";"

Fig. 4.20

(2) Editing MDI data

The [CURSOR ↓], [↑ CURSOR], ERASE, INSRT, and ALTER keys permit editing multi-block data written in. Address (word) pointed to by the cursor will be edited.
The [CURSOR ↓] and [↑ CURSOR] keys move the cursor forward and backward.

a. ERASE key: When this key has been depressed, the whole word designated is erased.

b. INSRT key: This key inserts the data which has just been entered next to the word the cursor points to.

c. ALTER key: This key replaces the word which the cursor points to with the data which has just been entered.

d. WR key: This key appends the data which has just been entered at the end of the program displayed.

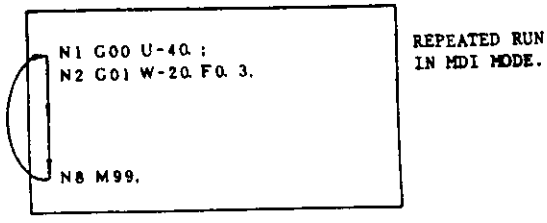
In MDI mode, only one screen currently displayed may be edited. Unlike EDIT mode and MEM mode, the display and edit of multiple screens cannot be performed. When the RESET key is depressed, the stored programs are all erased.

(3) Operation in MDI mode

a. Depressing the CYCLE START button in MDI mode can automatically execute the part programs stored in the MDI buffer. When the PROG function is active, the cursor is displayed at the head of the block currently executed. When the execution of all part programs is completed, the part programs and the CRT displays are erased.

b. If "M99" is written at the end of a part program, this program is executed repeatedly. The repetition may be stopped by depressing the FEED HOLD then RESET.

c. While a program is being run, the PROG function need not be active. Depressing POS can display the current values on the CRT.



4.3.3.2 Display in Memory Run Mode (PROGRAM [MEM])

The part program being executed in memory run mode (MEM mode) may be displayed by the following operations:

(1) Select the MEM mode

(2) Depress the PROG function key.
On the CRT screen, the cursor is positioned at the head of the block currently executed. The cursor moves to the next block when its execution is started.

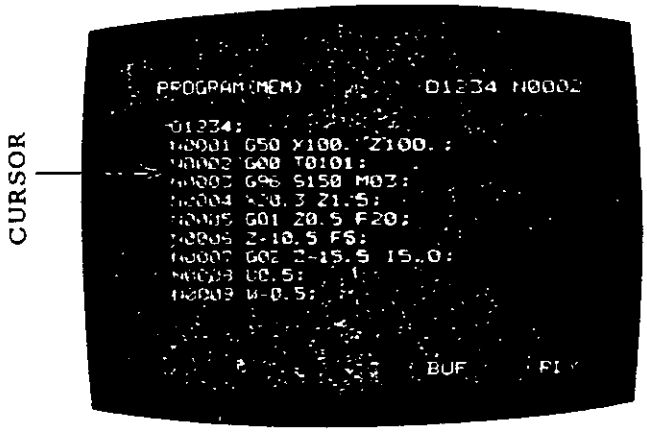


Fig. 4.21 Display of Part Program in Memory Operation

Up to 10 lines may be displayed at a time. When execution of the ninth has been completed, the next page appears with the tenth line of the last screen appearing at the top.

4.3.3.3 Display in EDIT Mode

See 4.6 EDIT.

4.3.3.4 Address Search

Search continues until a data (character string) held on tape or in the memory which coincides with the data (character string) entered through the NC operator's station. The contents of tape will be searched in TAPE mode and those of the part program memory in MEM or EDIT mode.

(1) Operation (MEM, EDIT mode)

a. Select MEM, or EDIT mode.

b. Depress the PROG function key.

c. Depress the RESET key. "LSK" appears and the pointer returns to the top of the program number in MEM mode.

d. Search is performed in one of the following three methods:

(i) Key-in the word (an address data) to be searched. If the leading zero of the data is omitted, the search operation is still possible.

(ii) Key-in only a single character without data. This permits searching the character read first.

(iii) Depress the NEXT key then key in any data (less than 32 characters) to be searched. In this case, the search operation is performed exactly according to the keyed data (character string or numeral string), thus disabling the omission of leading zero.

e. Depress the CURSOR ↓ key. Search starts. "AS" blinks during search.

f. When the NEXT key is depressed, depress it again to cancel the pattern search function.

(2) Operation (TAPE mode)

a. Select the TAPE mode.

b. Depress the PROG function key.

c. Depress the RESET key. "LSK" appears and the pointer returns to the top of the program number in MEM mode.

d. Search is performed in one of the following two methods.

(i) Key-in a single character only without data. This enables the control to search the character which was read first.

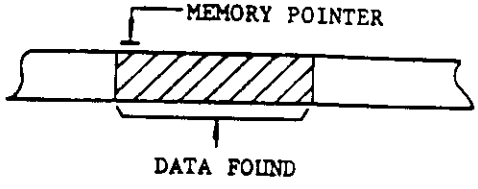
(ii) Key in the arbitrary data (32 characters max). Since search is performed according to the keyed-in data, the leading zero cannot be omitted.

e. Depress the CURSOR ↓ key. Search starts. "AS" blinks during search.

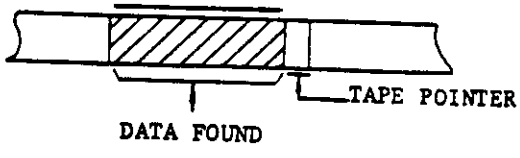
(3) Completion of search

a. When the search is completed, "AS" will disappear.

(i) In MEM or EDIT mode, the pointer of the part program memory points to the top of the data of block found (indicated by the cursor). In all cases, only search will be performed but neither BUF display nor advance reading will be performed.



(ii) In TAPE mode, the tape pointer points to the character that immediately follows the data found and the tape stops.



4.3.3.4 Address Search (Cont'd)

b "AS" disappears and "NOT FOUND!" appears on the CRT if the desired data is not found. This message will disappear when you depress a key (CAN normally) of the control station.

(4) Remarks

a. Commands encountered during search will be ignored even if they are modal commands.

b. On Cycle Start after search, the data of a block which the pointer points to will be read in and executed.

(5) Search of program number

The address search function also permits to search a part program out of those stored in the memory.

a. Select MEM or EDIT mode.

b. Depress the PROG function key.

c. Depress the RESET key.

d. Enter the program number "O □ □ □ □." Leading zero can be omitted.

e. Depress the [CURSOR ↓] key.

The designated program number will be searched. The result of search is as described in (2). In MEM mode, you may depress the CYCLE START button immediately after completion of search to start automatic operation from the beginning of the program.

.3.4 CURRENT POSITION DISPLAY

The current position of X-, or Z-axis can be displayed at any time in all modes. Operating procedure is as follows.

(1) Depress POS key.

One of the following will be displayed on the CRT screen

a POSITION EXTERNAL

b. POSITION ABSOLUTE

c. POSITION INCREMENT

d POSITION

e. PROGRAM RETURN

f DISTANCE TO LIMIT

g. PULSE COUNTER

h. ERROR PULSE

i. COMMAND PULSE

(2) Depress [PAGE ↓], [↑ PAGE] keys to select the page including any of the above.

NOTE: Page including error pulse or command pulse will be displayed when SYSTEM No. switch is set at 4.

4.3.4.1 POSITION [EXTERNAL]

(1) The current value to be displayed in EXTERNAL is the accumulated value of the tool movement from the position reset to "0" by the ORG key.

(2) How to reset POSITION [EXTERNAL]

Display the POSITION [EXTERNAL] screen on the CRT, select the axis by the address key, and depress the ORG key. The display of the selected axis becomes "0." This display reset operation is always valid even during time when the movement is being made by the automatic run of part program.

NOTES:

1. When parameter POSEXT (#6005D5) is set to "1," the value displayed in POSITION EXTERNAL becomes the same value as displayed in POSITION ABSOLUTE .

2. Regardless of the state of parameter POSEXT, the value displayed in the external current value display (option) is the same as displayed in POSITION [EXTERNAL]. Hence, resetting the external current value display automatically causes the reset of POSITION EXTERNAL .

3. The "value in the current value display of the equipment" described in 2.8.23, Work Coordinate Multi-Shift G50T and 6.2.3, Work Measuring Value Direct Input refers to this value in POSITION EXTERNAL .

4. The display lock feature is valid for POSITION [EXTERNAL].

Fig. 4.22 Display of
POSITION [EXTERNAL]

Fig. 4.23 Display of
POSITION [ABSOLUTE]

4.3.4.2 POSITION [ABSOLUTE]

(1) The current value displayed in POSITION ABSOLUTE indicates the tool position on the coordinate system provided by coordinate system settings. Coordinate system setting is performed in the following cases:

a. The execution of G50 coordinate system setting.

b. The operation of automatic coordinate system setting (option).

c. The current value reset operation by the ORG key (see (2) below).

d. The execution of G50T work coordinate system setting.

(2) How to reset POSITION [ABSOLUTE]

The reset operation by the ORG key described above is as follows:

Display the POSITION ABSOLUTE screen on CRT, select the axis by the address key, and depress the ORG key. The display of the selected axis becomes "0." However, this display reset operation is valid only in the manual operation modes (RAPID, JOG, and STEP (or HANDLE)). Depressing of the ORG key is invalid during operation or in the buffer full state.

NOTE: If the display lock is on, the display of POSITION [ABSOLUTE] is not locked.

4.3.4.3 POSITION [INCREMENT]

Displayed in this mode are:

- In automatic mode, distance to the end point of the block at every moment.
- In manual mode, distance to the position where manual operation is to start.
- The increment display in manual mode will be cancelled in automatic operation mode.

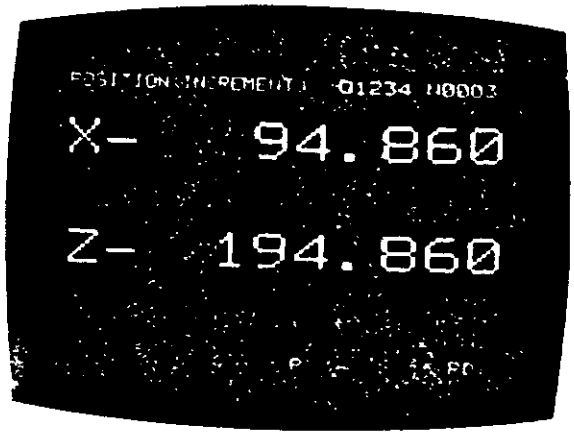


Fig. 4.24 Display of
POSITION INCREMENT

4.3.4.4 POSITION

(1) In POSITION, all positions are collectively displayed.

(2) POSITION MACHINE displays the tool current position on the coordinate system with the reference point returned by the reference point return feature being "0." The data on the following features are defined on this coordinate system

4.3.4.4 POSITION (Cont'd)

a. Stored stroke limit.

b. Stored stroke limit as arranged by tool.

c. Leadscrew error compensation.

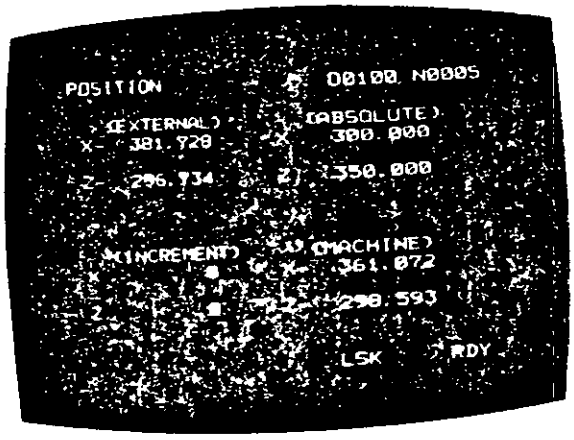


Fig. 4.25 Display of POSITION

4.3.4.5 PROGRAM RETURN

PROGRAM RETURN displays the information necessary for program restart. For details, see 6.2.6 Program Restart.

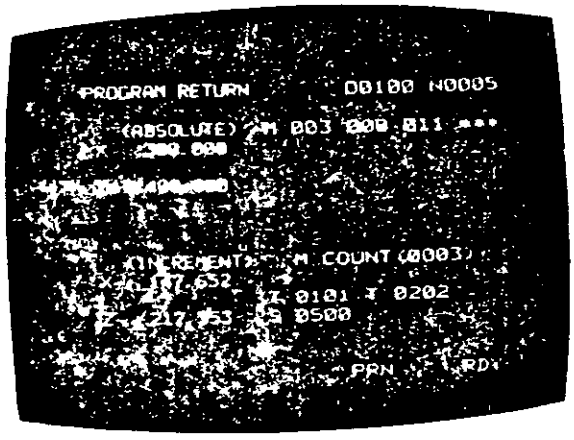


Fig. 4.26 Display of PROGRAM RETURN

4.3.4.6 STORED STROKE LIMIT†

(1) STORED STROKE LIMIT displays the remaining number of pulses in the four directions of X-axis plus/minus and Z-axis plus/minus from the tool current position to each boundary of the first, second, and third limit areas

The setting of STORED STROKE LIMIT and the display of remaining number of pulses:

a STORED STROKE LIMIT
X-axis plus direction boundary value

b STORED STROKE LIMIT
X-axis minus direction boundary value

c. STORED STROKE LIMIT
Z-axis plus direction boundary value

d. STORED STROKE LIMIT
Z-axis minus direction boundary value

} Set by parameter

① The value of X-axis plus remaining number of pulses

② The value of X-axis minus remaining number of pulses

③ The value of Z-axis plus remaining number of pulses

④ The value of Z-axis minus remaining number of pulses

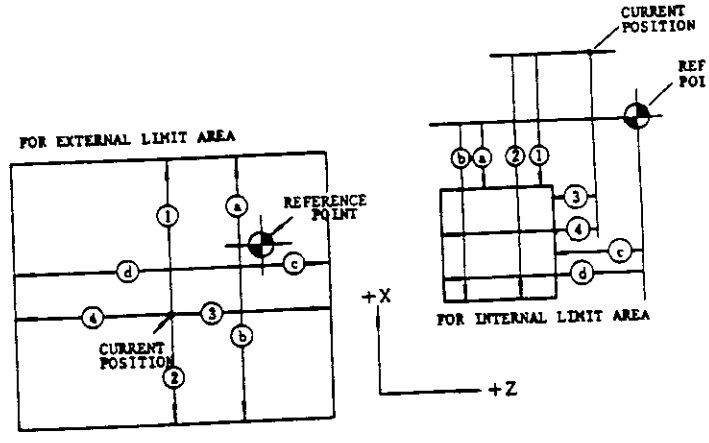


Fig. 4.27

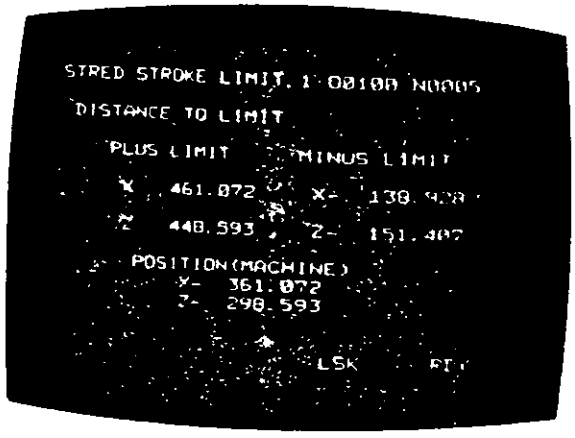


Fig. 4.28 Display of Remaining No. of Pulses in First Stored Stroke Limit

(2) The display shown above will correspon to 1st, 2nd, and 3rd prohibited area.

4.3.4.7 SPINDLE COUNTER

SPINDLE COUNTER displays the number of spindle PG pulses from the "spindle indexing origin" during the execution of the spindle indexing feature. The display is performed on a "1" = 1 pulse basis.

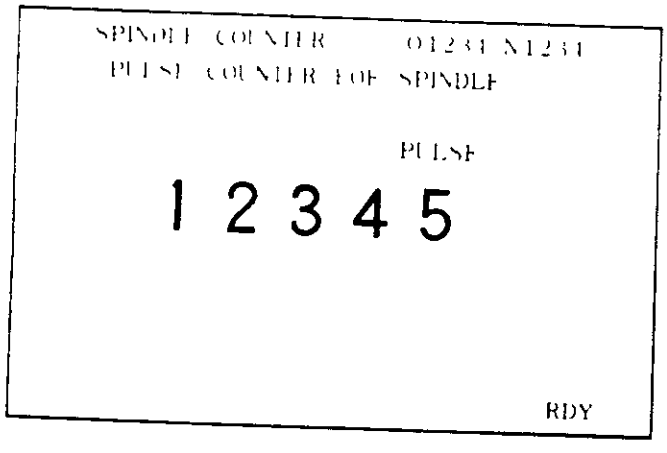


Fig. 4.29 Display of No. of Spindle PG Pulses

4.3.4.8 No. Of Servo Lag Pulses Display (ERROR PULSE)

The ERROR PULSE screen is displayed only when the system number switch is "4." Generally, this screen is used for maintenance purposes.

ERROR PULSE displays the difference between the momentarily changing command position and the tool current position. The display is performed on a "1" = 1 pulse basis (the minimum move unit).

Fig. 4.30 Display of No. of Servo Lag Pulses

4.3.4.9 Command Pulse Accumulation Register Display (COMMAND PULSE)

COMMAND PULSE displays the contents of the command pulse integration register (SMC register) in the control unit. This screen is displayed only when the system number switch is "4."

The SMC register is set to "0" when the power is turned on and keeps adding command pulses until the power is turned off.

NOTE: When a value is set to parameter XSMCB (#6658: for X-axis) or ZSMCB (#6659: for Z-axis), the value obtained by subtracting the value set above from the content of the command pulse integrating register is displayed.

Fig. 4.31

4.3.5 DIAPLAYING AND WRITING TOOL OFFSET DATA†

The tool offset amount is stored in the offset memory in the unit. Regardless of modes, the tool offset amount may be displayed and written any time including the automatic operation time.

(1) Display of tool offset amount

The display of tool offset amount and other offset memory contents is performed by the following operations:

a. Select the OFS function key.

b. Key in a two-digit tool offset number like 0, 1, and then depress [CURSOR ↓] or [↑ CURSOR].
The tool offset amount and the tool nose radius are displayed in five pairs including the tool offset number of the keyed value. The cursor is positioned to the designated tool offset number.

c. The preceding tool offset number may be designated by depressing [CURSOR ↓] or [↑ CURSOR] If the operation is performed outside the range of the tool offset numbers displayed on one screen, the following or preceding five pairs of tool offset amounts are displayed.

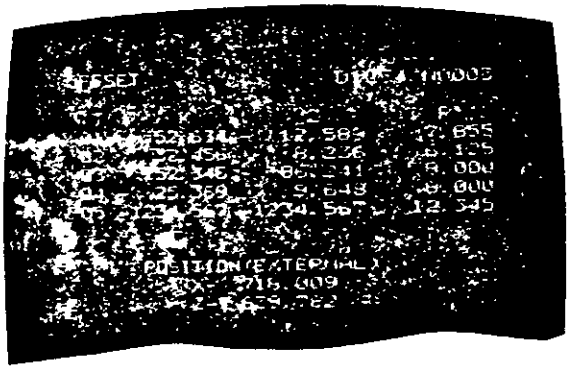


Fig. 4.32 Display of Tool Offset Amount

d. The screen of the following or preceding five pairs of tool offset amounts may be displayed by depressing [↑ PAGE] or [PAGE ↓] key. In this case, the cursor is positioned to the first of the displayed tool offset numbers.

e. When the options shown below are selected, designating the corresponding offset memory number by depressing [↑ PAGE] or [PAGE ↓] key or by two-digit value key-in operation can display each data.

(i) Work coordinate system shift option:
Work coordinate system shift amount.

(ii) Work measuring value direct input option
Tool coordinate data.

(iii) Tool wear compensation option:
Tool wear amount.

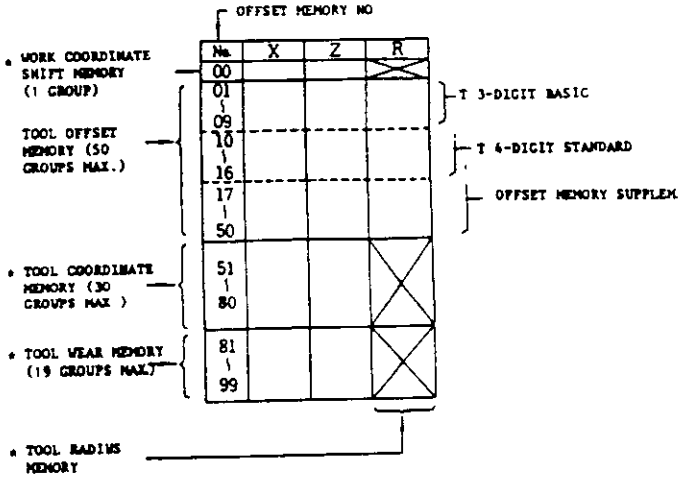


Fig. 4.33

(2) Writing of tool offset amount

The writing of the offset amount and other offset memory contents is performed by the following operations:

a. Depress the OFS function key.

b. By the operation of PAGE key and CURSOR key or keying in two digits designate the tool offset number to be written.

c. Using the address key and the data key, key in the address data to be written. In this case, the meaning depends on the address as shown below:

Table 4.1

Address	Axis Designation	Meaning
X	Write to X axis	Writing of an absolute value. Namely, the value is written to memory without change.
Z	Write to Z axis	
U	Write to X axis	Writing of an increment. Namely, the current value is added to the preceding value and the result is written to memory.
W	Write to Z axis	
R	Tool nose radius	Writing of an absolute value.

d. Depress the WR key.

The address data keyed in is written to the tool offset memory according to the meaning of the data.

e. Repeat the operations of c. and d. and b. through d. to write all necessary tool offset amounts.

f. When the options shown below are selected, if the corresponding offset memory number is designated at the operation of d., the data may be written to memory. However, address R has no significance and therefore should not be designated.

(i) Work coordinate system shift option: Work coordinate system shift amount.

(ii) Work measuring value direct input option: Tool coordinate data.

(iii) Tool wear compensation option: Tool wear amount.

NOTES:

1. The contents of offset memory are retained after the power is turned off.
2. Regardless of modes, the writing by the above operations is always possible including automatic run time.
3. The tool offset amount rewritten in automatic run is made valid from reading of the command of the next block. For the tool offset amount of the currently executed block or the block stored in the prefetch buffer, the value before change is used.
4. When the data shown below is rewritten, the timing in which the data value is made is as follows:

(i) Work coordinate system shift amount: The time when G50, G50T, or other coordinate system setting is performed next.

(ii) Tool coordinate data. The time when G50T is specified next.

(iii) Tool wear compensation amount: The time when T99ΔΔ is specified next.

5. The writing operation of the above tool coordinate data is performed when the PST INPUT button is not depressed. If this button is depressed, the data to be written is completely different from the above data. For details, see 6.2.3 Work Measuring Value Direct Input.

6. The contents of offset memory are all erased by the following operations:

(i) Depress the OFS function key.

(ii) Key in 0 - 9 9 9 9 , and depress the ORG key. The contents in offset memory are all erased.

4.3.6 DIAPLAYING AND WRITING SETTING DATA (SETTING)

With this unit, the setting data is stored in the internal memory. According to the contents of this memory, a particular function is turned on/off. These contents are also used for the control constants of functions. For details, refer to Appendix 1, List of Setting Numbers. The display and write of setting data are always enabled including the time when automatic run is being performed.

The setting data is of the following two types:

(1) Setting data of bit display format.

(2) Setting data of decimal display format.

4.3.6.1 Setting Data of Bit Display Format

Setting numbers "#6000 through #6004" have the setting data of bit display format. Each number has 8-bit information of "D7 through D0," each bit displaying on/off of the corresponding function.

(1) Display of setting data (bit display format)

Setting data is displayed by the following operations:

a. Select the SET function key.

4.3.6.1 Setting Data of Bit Display Format (Cont'd)

b. Key in 4-digit numeric setting number and depress [CURSOR ↓] or [↑ CURSOR] key. "#" need not be keyed in. Up to four pairs of setting data including the keyed setting number are displayed. To the right of bit display, the decimal value indicating the sum of the data on that line is shown. The cursor is positioned to the designated setting number.

c. The setting number designation may be updated by [CURSOR ↓] or [↑ CURSOR] key, and the screen may be updated by [PAGE ↓] or [↑ PAGE] key.

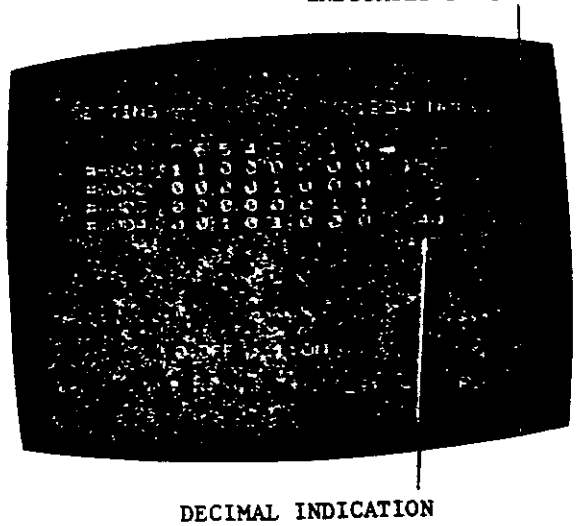


Fig. 4.34 Display of Setting Data Shown in Bit Display

Note When #6000 is designated, the screen dedicated to the display of on/off state of the "internal toggle switch" is provided. For details, see 4.3.6.2.

(2) Writing of setting data (bit display format)

The writing of the setting data of bit display format is performed by the following operations.

a. Depress the SET function key.

b. By the operation of PAGE key and CURSOR key or keying in 4 digits, designate the setting number to be written.

c. Depress the INSRT key. The cursor moves to the bit data from a setting number. Designate the data of D7.

d. Depress the [CURSOR ↓] key. Each time the key is depressed, the cursor moves by one bit toward D0. Locate the cursor at a desired bit position.

e. Depress the WR key. The designated bit data reverses (0 to 1 or 1 to 0). If you depress the WR key again, the bit data will reverse again. Normally, "1" designates on state and "0" off state.

f. To write data in decimal mode, locate the cursor at the right most column (decimal data).

EXAMPLE: Writing in decimal mode

Entered data		7	6	5	4	3	2	1	0	
0	WR ⟶	0	0	0	0	0	0	0	0	0
2 5 5	WR ⟶	1	1	1	1	1	1	1	1	255

g. Repeat steps b, through f. to write requirec data. If you keep the [CURSOR ↓] or [↑ CURSOR] key depressed, the cursor will move column by column in the screen automatically.

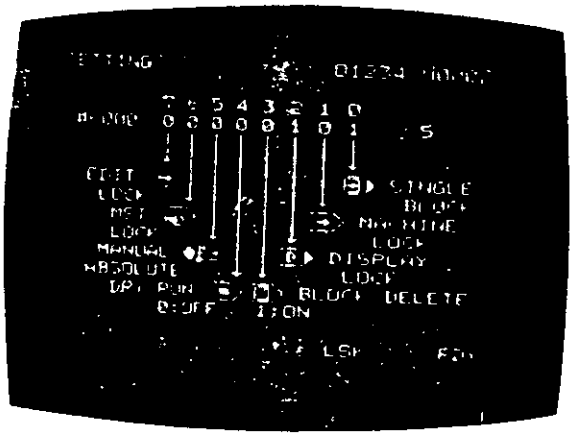
h. When data has been written, depress the INSRT key. Cursor returns to the position of setting number normally, this sequence of operations begins and ends both with the depression of the INSRT key.

4.3.6.2 Internal Toggle Switches†

(1) When the eight basic function switches showr below are omitted from the machine operator's station, each function may easily be turned on/off by the setting operation from the NC operator panel.

(2) The setting numbers are #6000D7 through D0 of bit display format. By the operation of the writing of bit display format setting described above, turn on/off each function.

When "1" is set, the function is turned on. When "0" is set, the function is turned off.



Display shows that Single Block and Display Lock are on.

Fig. 4.35 Display of Internal Toggle Switch Status

NOTES:

1. The internal toggle switch is an optional function. Hence, which internal toggle switch is available depends on the controlled machine. The display for unused toggle switches is blank. For details, refer to the instruction manual of the machine in question.
2. When the machine control station is provided with the switches that turn on and off the above functions, the state of the switch on the machine control station is ORed with that of the operator's panel to determine the final ON/OFF state.

Setting Data	Switches on Machine Control Station	Resultant ON/OFF
"0" = OFF	OFF	OFF
"0" = OFF	ON	ON
"1" = ON	OFF	ON
"1" = ON	ON	ON

4.3.6.3 Setting Data of Decimal Display Format

The following setting numbers have the setting data of decimal number display format:

#6160 through #6219

#6500 through #6579

#8600 through #8750

These setting numbers are used for the control constants of tool life control and multiple repetitive cycles.

(1) Display of setting data (decimal display format)

The setting data is displayed by the following operations:

a. Depress the SET function key.

b. Key in the 4-digit setting number and depress [CURSOR ↓] or [↑ CURSOR] key. The maximum of 10 lines of setting data including the keyed setting number are displayed. The cursor is positioned to the designated setting number.

c. The setting number designation may be updated by [CURSOR ↓] or [↑ CURSOR] key, and the screen may be updated by [PAGE ↓] or [↑ PAGE] key.

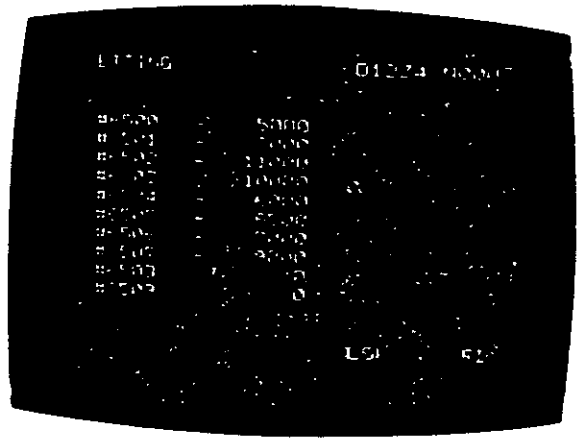


Fig. 4.36 Display of Setting Data in Decimal Mode

(2) Writing of setting data (decimal display format)

The writing of the setting data of decimal display format is performed by the following operations:

a. Depress the SET function key.

b. By the operation of PAGE key and CURSOR key or keying in 4-digit value, designate the setting number to be written.

c. Key in the value by the data key and depress the WR key. The keyed value is written as the data of the setting number designated by the cursor,

d. Repeat operations of b. and c. to write the necessary setting data.

4.3.7 DISPLAYING AND WRITING PARAMETERS

In this system, varying parameters are stored in the memory and they determine operating conditions such as tape code and feedrate. For details, see Appendix 2, LIST OF PARAMETER NUMBERS. The parameters may be displayed at any time even during automatic operation.

The parameters are of the following two types:

(1) The parameters of bit display format.

(2) The parameters of decimal display format.

4.3.7.1 Parameters of Bit Display Format

Parameter numbers #6005 through #6049 indicate the parameters of bit display format. Each number has 8-bit information of D7 through D0.

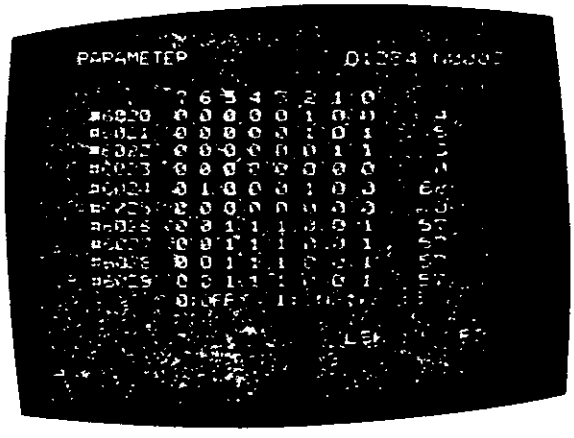


Fig. 4.37 Display of Parameters in Binary Mode

4.3.7.2 Parameters of Decimal Display Format

The following parameter numbers indicate the parameters of decimal display format:

#6050 through #6149

#6220 through #6349

#6600 through #6659

#8000 through #8225

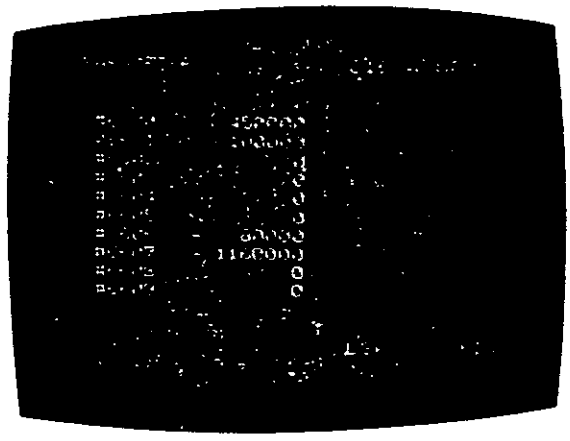


Fig. 4.38 Display of Parameters in Decimal Mode

4.3.7.3 Displaying And Writing Parameters

(1) Operations for parameter display

The operations for parameter display are the same as those for setting display except that the PRM function key is depressed instead of the SET function key. For details see 4.3.6.

(2) Operations for parameter writing

a. The parameter values are preset according to the performance of the machine and applications. Therefore, you should consult the machine tool builder if you want to change parameter settings.

b. The parameters are protected with a system No. switch provided on the tape reader so that they should not be destroyed by wrong operation. Normally system No. 0 is selected and, at this time, the parameters cannot be rewritten by any operation.

c. The operations for parameter writing are the same as those for setting display except for the following:

(i) First, set the system No. switch to "1."

(ii) Depress the PRM function key instead of the SET function key. Then, the parameter data may be written by the same operations as those for the writing of setting data of bit display format (4.3.6.1, (2)) or for the writing of setting data of decimal display format (4.3.6.3, (2)).

(iii) After the completion of the writing operation, set the system No. switch to "0."

d. If the following parameters have been changed, be sure to turn off power then turn it on again. Otherwise the system might fail to operate properly.

#6009, #6010, #6023, #6024, #6031, #6032, #6033, #6035, #6036, #6039, #6040 - #6049, #6050 - #6075 #6092 - #6095, #6286 - #6317 #6322 - #6335 After reading in parameter tape

When a parameter other than above has been rewritten, also depress the RESET key to reset the unit.

4.3.8 DISPLAYING STATUS INPUT/OUTPUT SIGNALS

Depress the DGN function key, and the state of every input/output signal will be displayed on the CRT. This is possible at any time even during automatic operation.

For more detail of this operation, see 8.6.3 Diagnostics of INPUT/OUTPUT Signals.

The state of the input/output signal is also given in the hexadecimal notation at the rightmost column for the ease of maintenance work.

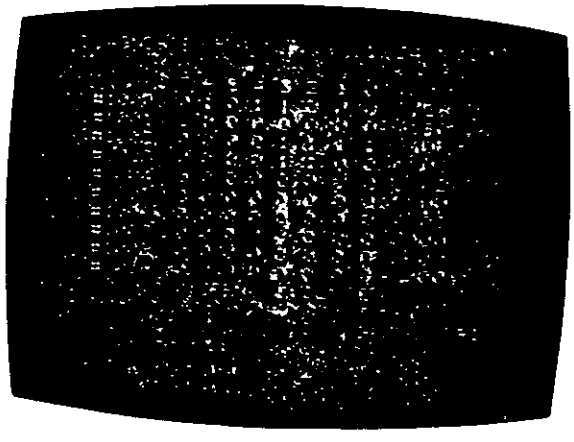


Fig. 4.39

4.3.9 ALARM CODE (ALM) DISPLAY

Alarm codes and other data are displayed by the following operations:

(1) Depress the ALM function key. One of the following screens is displayed:

a. Display of the alarm number and the message. (ALARM)

b. Display of operating time. (TIMER)

c. Display of registered program number. (PROGRAM NO. TABLE)

d. Display of maintenance history. (MAINTENANCE)

(2) The above screens may be selected by depressing PAGE ↓ or ↑ PAGE key.

4.3.9.1 Alarm Code Display

If an alarm status has happened, "ALM" or "A/B" (on battery alarm) blinks on the bottom line of the screen regardless of working mode and function. If this happens, the detailed information of the alarm status may be displayed by the following operation.

(1) Depress the ALM key.

Then up to four pairs of alarm code and message will be displayed, with more serious one on a higher line.

NOTE: The alarm screen will appear during an alarm state and, therefore, it is not needed to operate the PAGE ↓ key.

Fig. 4.40 Display of Alarms

(2) To reset the alarm status and screen, remove the cause of alarm then depress the RESET key.

For the detail of alarm code, refer to Appendix 5 List of Alarm Codes.

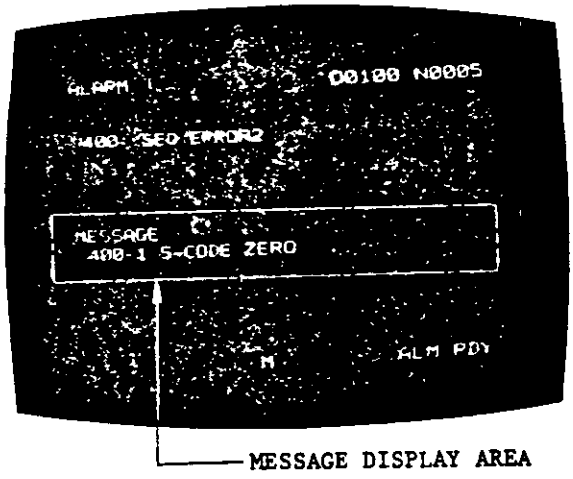
4.3.9.2 Message Display (ALARM)†

This feature can display messages on the CRT screen by the instruction of PC when the machine sequence control option (PC system) is built in the unit. Normally, this feature is used to display the cause of the alarm detected by the PC.

When the message display instruction (macro instruction) is executed in the PC, "ALM" or "A/B" flashes at the bottom of the CRT screen regardless of mode and function.

In this case, the message may be displayed by the following operation:

(1) Depress the ALM function key. The message is displayed at the bottom of the screen along with the sequence error code. The message to be displayed depends on the machine. For details, refer to the instruction manual of the machine in question.



MESSAGE DISPLAY AREA

Fig. 4.41 Example of Message Display

(2) When the RESET key is depressed after removing the cause of the alarm which displayed the message, the message display and alarm state may be cleared.

Note. In some cases, only the message display is provided without displaying "400. SEQ ERROR."

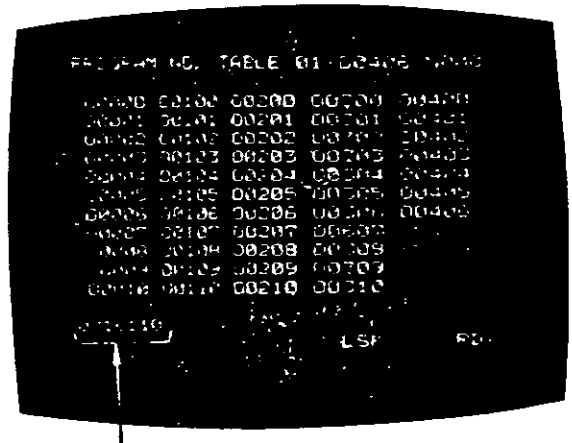
4.3.9.3 Display of Registered Program Number (PROGRAM NO. TABLE□□)†

This screen displays all registered program numbers and the number of remaining characters in the part program memory.

(1) The number of program numbers that can be registered depends on options.

No.	Max. Number of Programs	Type	Program No. Table
1	99	Basic	01 to 02
2	199	Option 1	01 to 04
3	999	Option 2	01 to 19

(2) All program numbers already registered are displayed. By depressing PAGE ↓ or ↑ PAGE key, the page shown below may be obtained.



REMAINING NUMBER OF CHARACTERS IN PART PROGRAM MEMORY

The remaining number of characters in part program memory is displayed in the lower left corner of the screen.

Fig. 4.42

Note: This screen displays only the registered program numbers. A program number is registered by depressing the PROG function key in EDIT mode.

4.3.9.4 Operation Time Display

The system counts the duration of automatic operation and it may be displayed. This function permits the display of the time it has taken for a single piece of work or the total operational time of the system.

(1) Three kinds of operation time will be displayed in hours, minutes, and seconds.

a. POWER ON: Total operating time after POWER ON

b. CYCLE START: Total operating time of CYCLE START

c. FEED: Total operating time of FEED

d. EXTERNAL INPUT†: Total operating time while external input signal is ON (optional)

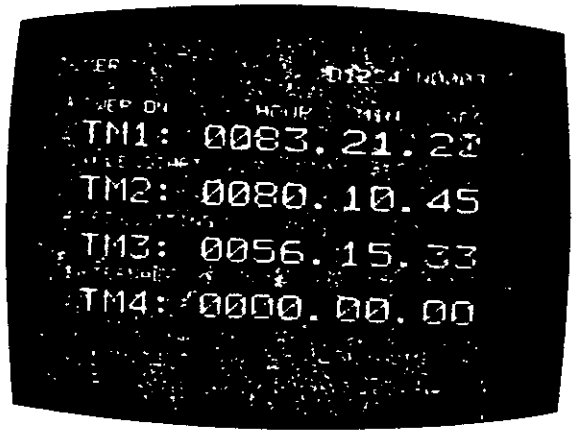


Fig. 43 Operation Time Display

(2) Reset of operating time display

The above operating time displays may be reset separately by the following operations. In the state where the operating time is displayed;

a. Depress "1" then ORG keys.
The time of "POWER ON" is reset.

b. Depress "2" then ORG keys.
The time of "CYCLE START" is reset.

c. Depress "3" then ORG keys.
The time of "FEED CUTTING" is reset.

d. Depress "4" then ORG keys.
The time of "EXTERNAL INPUT" is reset.

Unless this display reset operation is performed, then operating time display is retained when the power is turned off.

e. When "1," "2," "3," "4," "ORG" keys are depressed, all the times shown above are reset at a time.

4.3.9.5 Maintenance History Display (MAINTENANCE)

(1) This screen displays the maintenance information on the control unit. This display is independent of the control unit functions.

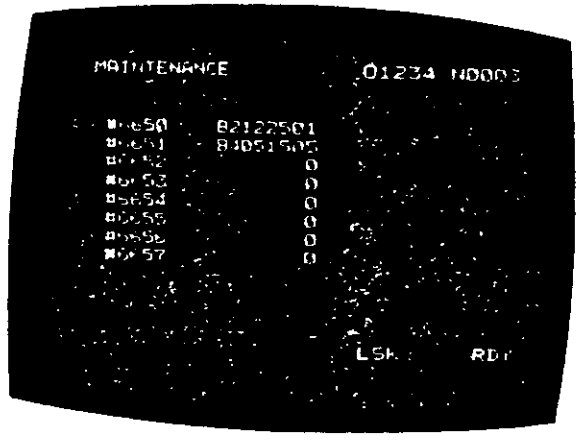


Fig. 4.44

4.4 LOADING PART PROGRAMS AND NC DATA INTO MEMORY (IN)

This paragraph describes the operations for storing the following data into the corresponding internal memory of the NC:

- Part program
- Tool offset amount
- Setting data and parameters

If these data are to be stored in the form of a punched tape, enter them through the tape reader or the data input/output interface (option). For the methods of setting the input/output equipment (setting #6003) and the baud rate (parameters #6026 and #6027), refer to 4.9 Data Input/Output Interface. The following description is made assuming that this option is installed.

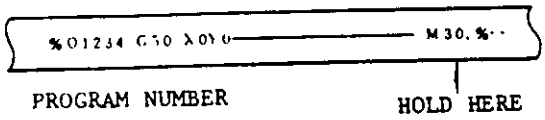
4.4.1 LOADING PART PROGRAM TAPE INTO MEMORY

(1) Loading a part program which has a program number.

a. Select EDIT mode.

b. Depress the PROG key.

c. Load the NC tape to the tape reader or an equivalent external device.



d Depress the RESET key.

e. Depress the IN key.

Then the system starts to read the tape and enlists the program number punched on the tape as the first record. The system checks for duplicator of program number as in 1. Operation ends with error if the designated program number is not found on the tape.

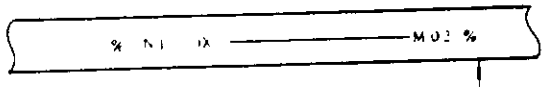
When the tape reader has read "M02 ;," "M03 ;," or "M99 ;," it stops and "IN" disappears from the CRT. Now the part program has been stored in memory.

(2) Loading a part program which has no program number.

a. Select EDIT mode.

b. Depress the PROG keys.

c Load the NC tape to the tape reader or an equivalent external device.



The tape stops at this location when loading is completed.

d Depress the RESET key.

e Depress the address O key then enter the program number.

f. Depress the IN key.
The system starts to read the tape. If the keyed-in program number coincides with the registered program number, "ALREADY IN" blinks on the CRT screen. If this happens, delete the program number, then repeat steps a. through f. while the tape is being read, "IN" blinks on the CRT.

g. When the tape reader has read "M02 ;," "M03 ;," or "M99 ;," it stops and "IN" disappears from the CRT. Now the part program has been stored in memory.

NOTES:

1. Program number "O0000" is always in the registered state, so it cannot be erased. This program number should not be used in general.

2. The tape which has no program number may be stored as described before. However, write a program number to the head of the tape, in principle. The operation of "Oxxxx IN" described before causes only program number registration. It does not cause the storing of information of "O0000" into the part program memory. Only the program number on tape is stored into the memory. Assume that a tape having no program number is stored and then all part programs are punched out by depressing "0," "-," "9," "9," "9," "9," and "OUT" keys. Since this tape contains programs with no program number, the correct restoring of all part programs may not be performed by depressing "0," "-," "9," "9," "9," "9," and "IN" keys.

3. Consequently, when a tape having no program number has been stored, write the program number to the head of part program by the EDIT operation.

EXAMPLE:

N1 G50 X0 Z0 ,

When this is in the first block, position the cursor to N and key in as follows (in EDIT and PROG modes):

Oxxxx ; N1 ALTER

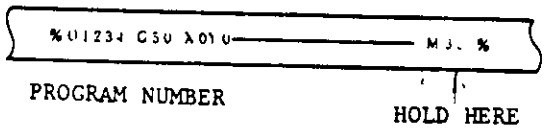
(3) Storing a program with program numbers changed !

To register a program with a program number different from the one punched on tape, perform the following operations:

a. Select the EDIT mode.

b. Depress the PROG key.

c. Set the NC tape to the tape reader or the external equivalent equipment.



d. Depress the RESET key.

e. Key in "O" and PROGRAM NUMBER.

f. Depress the IN key.

The program number entered from the key is registered in preference to the program number punched on the tape. At this time, the program number on the tape is written to the part program memory simply as a label.
M02 ;, M30 ; or M99 ; is read and the storing operation is completed.

NOTE:

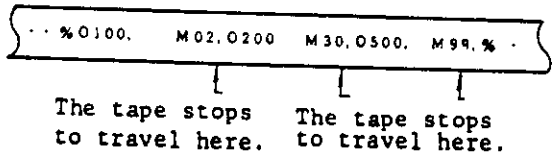
1. If a program is stored with a changed program number as described above, the program number punched on the tape is stored in the part program memory without change. Consequently, to avoid the confusion in the later handling, replace the program number in the part program memory with changed program number by the EDIT operation.

(4) Loading part programs from a tape

a. Select EDIT mode.

b. Depress the PROG key.

c. Load the NC tape to the tape reader or an equivalent external device.



d. Depress the RESET key.

e. Depress the IN key.

Then the system starts to read the tape and enlists the program number punched on the tape as the first record. The system checks for duplication of program number as described in (2).

The tape reader stops each time it has read "M02 ;," "M30 ;," or "M99 ;."

f. Depress the IN key again.

The tape reader resumes to read the tape. Repeat this operation until all programs are loaded.

(5) Storing a program with program numbers changed II

When "O" key is depressed and program number is keyed in before depressing IN key as described in (4) above, the keyed in program number is registered in preference to the program number punched on the NC tape.

(6) Loading programs continuously

Programs existing on a tape as shown in (4) may be loaded continuously without interruption. For this purpose, depress "O," "-," "9," "9," "9," and "9" before the first depression of the IN key. The tape reader stops at the position of "%."

4.4.2 MAKING ADDITION TO A PART PROGRAM

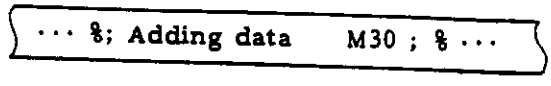
Perform the following operation to add data to a part program which is already loaded.

a. Select EDIT mode.

b. Depress the PROG key

c. Depress the O key then enter the part program number and depress the CURSOR ↓ key.
The system searches the designated program.

d. Load the tape of adding data to the tape reader.



e. Depress the RESET key.

f. Depress the NEXT and IN keys in this order.

The data will be read from the tape into the memory.

NOTE: You cannot add data to a program from the middle of it. If necessary, delete the last part of the program by editing operation and perform this adding.

4.4.3 LOADING PART PROGRAMS BY MDI

Part programs may be loaded not through the tape reader but by MDI operation. Perform the following.

a. Select EDIT mode.

b. Depress the PROG key

c. Depress the RESET key.

d. Depress the O key then enter the part program number and depress the WR key.

   The designated program number will be registered. If this number already exists, "ALREADY IN" blinks and, in this case, it is required to delete the registered program number.

e. Write the part program by operating the address key and the data key. As shown in the figure below, the keyed in data is displayed on the bottom line from left to right sequentially. The maximum number of characters that can be written at a time is 32. Within this limit, data may be keyed in over multiple words or blocks. However, when the 10th character is keyed in, the normal display shown to the right of the line is blanked.

f. Depress the INSRT key.

   The keyed in data is stored in the part program memory.

g. Repeat the operations of e. and f. above to write the part program. The program edit operation is enabled by the use of ERASE, INSRT, and ALTER keys during this program storing operation.

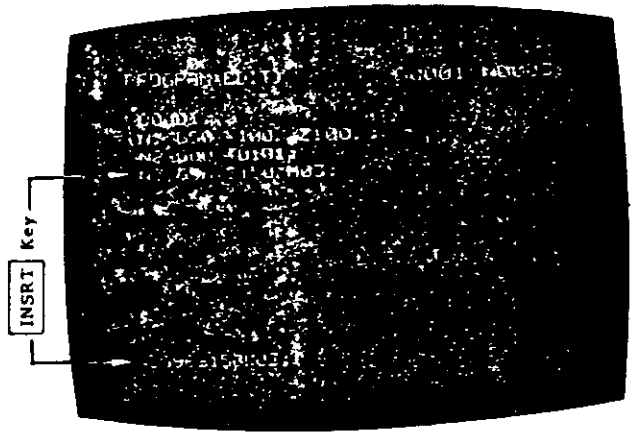


Fig. 4.45

h. Key in M02 ;, M30 ;, or M99 ; and depress INSRT key. This completes the storing of the part program.

4.4.4 INPUTTING TOOL OFFSET DATA INTO MEMORY

Normally, the tool offset data is written to the tool offset memory by MDI operation. This data may also be entered in the form of a punched tape.

(1) The format of the tool offset data tape is as shown below:

```
LABEL  % ;
T01 X····· Z····· R····· ;
T02 X····· Z····· R····· ;
T03 X····· Z····· R····· ;

 T99 X····· Z····· ;
 %
```

(2) The storing of the data by the above tape is performed by the following operations·

a. Select EDIT mode.

b. Depress OFS key.

c. Set the tool offset data tape onto the tape reader.

d. Depress the RESET key.

e. Depress the IN key.

   The tape reader starts to read the tape. "IN" blinks on the CRT while the data are read.

f. The tape reader stops when it has read "%" (or "ER"). "IN" disappears from the CRT. Now the tool offset data has been read into memory.

(3) Remarks

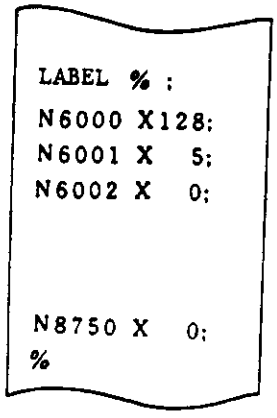
G10:

In the case of the tool offset data tape of the format by tool offset data designation, performing the cycle start in TAPE mode causes to store the data into the tool offset memory.

4.4.5 INPUTTING SETTING DATA AND PARAMETER DATA

Though setting data and parameter data are inputted in their memories by MDI operation normally, they may also be entered by means of paper tape. Setting data and parameter data may be inputted from a single tape.

(1) The tape format is as follows.



(2) The input operation is as follows.

a. Set the SYSTEM No. switch at 1.

b. Select EDIT mode.

c. Depress the PRM key.

d. Depress the RESET key.

e. Depress IN key.

The tape reader starts to read the tape. "IN" blinks on the CRT while the data are being read.

The tape reader stops when it has read "%" (or "ER"). "IN" disappears from the CRT. Now the setting/parameter data have been read into memory.

f. Return the system No. switch to 0.

g. Turn the power off then on.

The NC operation by the newly stored setting data and parameter may be performed.

Note: At the end of operation e., the operations on the NC operator's panel are all invalidated. Hence, the power must be turned on again.

4.5 TAPE VERIFYING

The punched tape of the data shown below may be compared to the contents of the NC internal memory to check if they match.

- Part program
- Tool offset data
- Setting data and parameter

The punched tape is entered through the NC tape reader or the data input/output interface (option). For the methods of setting the input/output equipment (setting #6003) and the baud rate (parameters #6026 through #6029), refer to 4.9 DATA INPUT/OUTPUT INTERFACE. The following description is made assuming that this option is installed.

4.5.1 VERIFYING PART PROGRAM TAPE

(1) Verifying a part program tape having program number

a. Select the EDIT mode.

b. Depress the PROG function key.

c. Set the part program tape to the tape reader.

d. Depress the RESET key.

e. Depress the VER key.

The tape is started to compare the contents of the part program memory to the contents of the part program tape. During this operation, "VER" keeps flashing. If a mismatch is found, "INPUT ERROR" is displayed flashing.
When a match is found and this operation is completed, the tape reader stops, upon which "VER" display is erased

Note: By the operation of "RESET, VER," the verifying feature verifies the data from the tape head to % code.

(2) Verifying a part program tape having no program number

a. Select the EDIT mode.

b. Depress the PROG function key.

c. Set the part program to the tape reader.

d. Depress the RESET key.

e. Depress the "O" key and key in program number

4.5.1 VERIFYING PART PROGRAM TAPE (CONT'D)

f. Depress VER key.

The tape starts to compare the contents of the part program memory to the contents of the part program tape. During this operation, "VER" keeps flashing. If a mismatch is found, "INPUT ERROR" is displayed flashing. When a match is found and this operation is completed, the tape reader stops, upon which "VER" display is erased. If the keyed in program number is not found in the memory, "NOT FOUND!" is displayed flashing. In this case, depress the CAN key and start with the operation of d.

NOTES

1 The operations for verification with a program number different from the program number punched on the tape are the same as those of (2) above. The keyed-in program number is processed in preference to the punched program number.

2 Verification by the operation of "Oxxxx VER" regards the punched information as the information on the keyed-in program number. Hence, when verifying a tape containing program numbers, no program number should be keyed in.

(3) Verifying a tape containing multiple part programs

Multiple part programs punched in a single tape are continuously verified by the following operations.

a. Select the EDIT mode.

b. Depress the PROG function key.

c. Set the part program tape to the tape reader.

d. Depress the RESET key.

e. Depress the VER key.

When M02, M30, or M99 is read, the tape reader does not stop but all the part programs are continuously verified up to % code. When the verification is completed, the tape reader stops at the position of % code.

4.5.2 VERIFYING TOOL OFFSET VALUE TAPE

The contents of the tool offset value tape are compared to the contents of the offset memory by the following operations:

(1) Select the EDIT mode.

(2) Depress the OFS function key.

(3) Set the tool offset value tape to the tape reader.

(4) Depress the RESET key.

(5) Depress the VER key.

The tape starts to compare the contents of the tape to the contents of the tool offset memory. During this operation, "VER keeps flashing. If a mismatch is found, "INPUT ERROR" is displayed flashing. When a match is found and this operation is completed, the tape reader stops, upon which "VER" display is erased.

4.5.3 VERIFYING SETTING AND PARAMETER TAPES

The setting data tape or the parameter tape is compared to the respective contents of the memory. It is possible to punch the setting data and parameter on a single tape and store them at a time by the following operations:

(1) Select the EDIT mode.

(2) Depress the PRM function key.'

For the tape punched only with setting data, depressing the SET key causes the same effect.

(3) Set the setting data and/or parameter tape to the tape reader.

(4) Depress the RESET key.

(5) Depress the VER key.

The tape starts to compare the contents of the tape of the contents of setting or parameter. During this operation, "VER" keeps flashing. If a mismatch is found, "INPUT ERROR" is displayed flashing. When a match is found and this operations is completed, the tape reader stops, upon which "VER" display is erased.

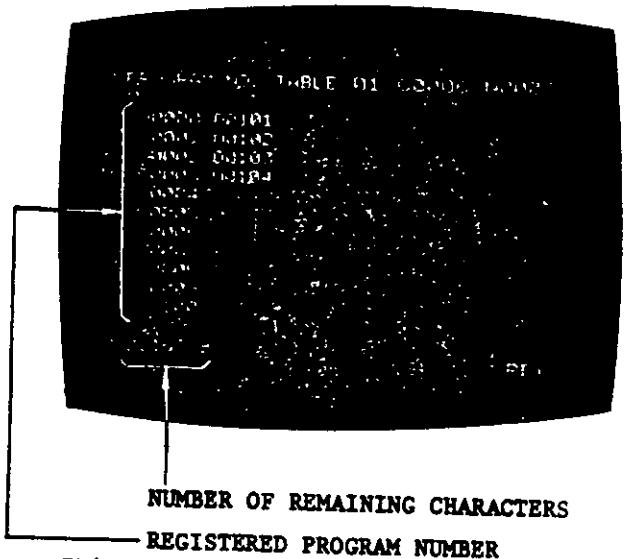
4.6 EDIT

4.6.1 CHECKING REGISTERED PART PROGRAM NUMBER

Before editing part programs, make confirmation of the registered program numbers and the remaining number of characters in the part program memory by the following operations·

(1) Depress the ALM function key.

(2) Depress the [↑ CURSOR] or [CURSOR ↓] key to select the screen (PROGRAM NO. TABLE□□) of registered program number display.



This screen only displays information and therefore cannot be used to register program numbers.

Fig. 4.46

4.6.2 DISPLAYING AND CHECKING STORED PART PROGRAMS

Stored part programs may be displayed on the CRT screen to check their contents by the following operations:

(1) Select the EDIT mode.

(2) Depress the PROG function key.

(3) Depress the RESET key.

(4) Key in O PROGRAM NUMBER.

(5) Depress the [CURSOR ↓] key.

The designated program number is searched. One screen of data (for 10 lines) from the head of the searched program is displayed on the CRT. If the program number has not been found, "NOT FOUND" is displayed flashing. This display may be reset by depressing CAN key in general.

(6) The preceding or following screen may be displayed on the CRT by depressing the [↑ PAGE] or [PAGE ↓] key.

(7) When the [↑ CURSOR] or [CURSOR ↓] key is depressed, the cursor is moved to the preceding or following word on a word basis.

The above operations make the word (address and data) designated by the cursor ready for such edit operations as modification, insertion, and deletion.

NOTE: The search for a program number may be performed also in the MEM mode. But the cursor movement by the [PAGE ↓] or [↑ PAGE] key is disabled.

4.6.3 MODIFYING PART PROGRAM BLOCKS

Modification of part programs is all performed in the EDIT mode and the PROG function by the operations which follow the operations described in the preceding paragraph 4.6.2 Displaying And Checking Stored Part Programs.

(1) Depress the page key and the cursor key to designate the word to be modified.

(2) Depress the address key and the data key to enter the word to be modified. As shown in the figure below, the keyed-in data is displayed in the bottom line on the CRT screen from left to right sequentially. The maximum number of characters that may be written at a time is 32. Within this limit, data may be keyed-in over multiple words or blocks.

(3) Depress the ALTER key.

The word designated by the cursor is deleted and the newly keyed-in data is displayed in that place. After modification, the altered word is in the designated state.

4.6.3 MODIFYING PART PROGRAM BLOCKS (CONT'D)

(4) Repeat the operations of (1) through (3) to modify any number of words.

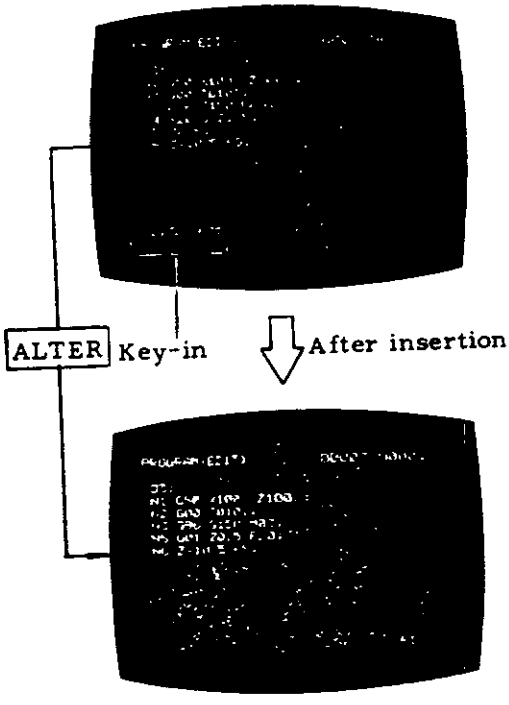


Fig. 4.47

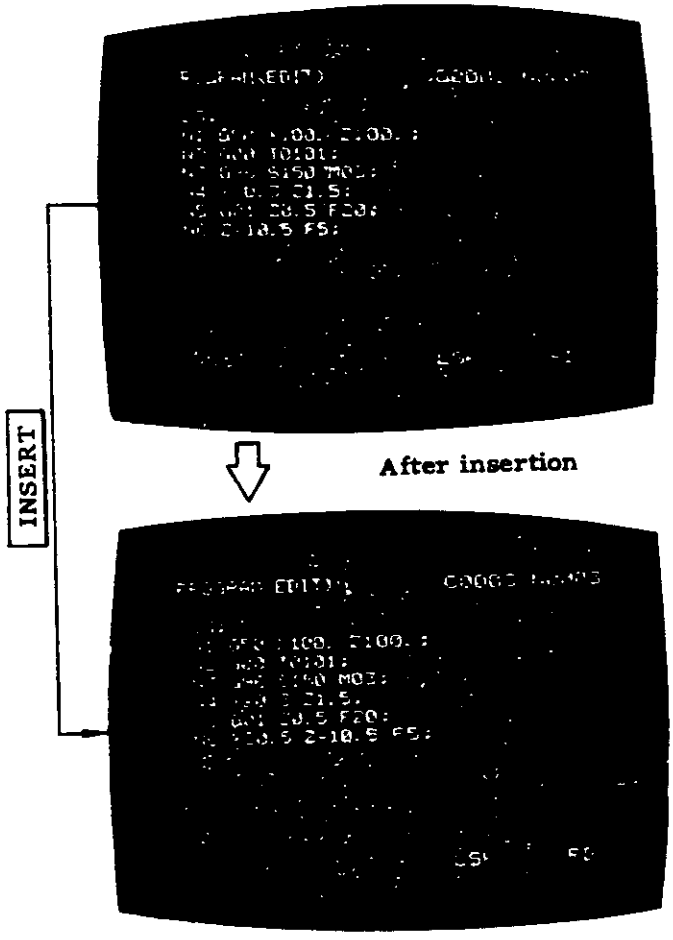


Fig. 4.48

4.6.4 ADDING PART PROGRAMS

A part program is added in the EDIT mode and the PROG function by the operations shown below which follow the operations of 4.6.2 Displaying And Checking Stored Part Programs.

(1) Depress the page key and the cursor key to designate the word immediately before the portion to be added.

(2) Depress the address key and data key to key in the word to be added. Within the limit of 32 characters, the data may extend over multiple words or blocks.

(3) Depress the INSRT key.

The keyed-in data is added to the portion immediately after the word designated by the cursor. After the addition, the added word is in the designated state.

(4) Repeat the operations of (1) through (3) to add any number of words.

4.6.5 DELETING PART PROGRAM BLOCKS

Part program blocks are deleted in the EDIT mode and the PROG function.

(1) Deleting words

Perform the operations below which follow the operations of 4.6.2 Displaying And Checking Stored Part Programs:

a. Depress the page key and the cursor key to designate the word to be deleted.

b. Depress the ERASE key.

The word designated by the cursor is deleted. After the deletion, the word subsequent to the deleted word is in the designated state.

c. Repeat the operations of a. and b. to delete any number of words.

(2) Deleting blocks

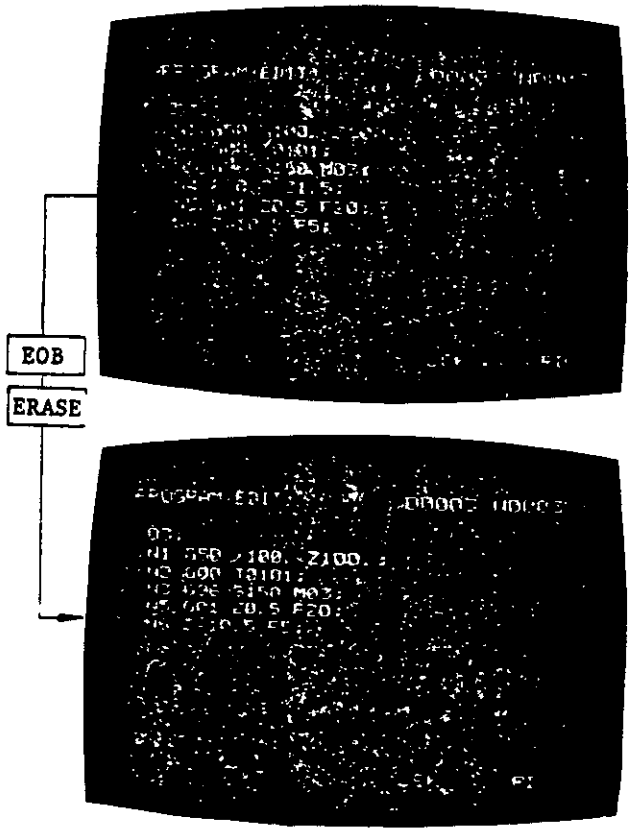
Data of one block may be deleted by the following operations:

a. Depress the page key and the cursor key to position the cursor to the head of the block to be deleted.

b. Depress EOB then ERASE.
All data of the block designated by the cursor are deleted.

NOTE: If the cursor is positioned to the middle of the block to be deleted and EOB key and ERASE key are depressed, the data from the cursor position to ";" code are deleted. Since ";" is also deleted, the words left undeleted are included in the next block.

(Before deleting)



(After deleting)
Fig. 4.49

(3) Deleting program number

A program number is deleted by the following operations:

a. Select the EDIT mode.

b. Depress the PROG function key.

c. Depress "O" key and key-in the program number.

d. Depress the ERASE key.

The keyed-in program number and the corresponding part program are deleted.

(4) Deleting all program numbers

All program numbers are deleted by the following operations:

a. Select the EDIT mode.

b. Depress the PROG function key.

c. Key in 0 - 9 9 9 9 .

d. Depress the ERASE key.

All registered program numbers and part programs are deleted. However, only program number "0" is newly registered with EOB not deleted.

4.7 PART PROGRAM AND NC DATA OUTPUT OPERATIONS

The following data, which are stored in the NC internal memory, may be sent to the external equipment through the data input/output iterface (option).

- Part program
- Tool offset value
- Setting data and parameter

When the external equipment has the tape punch feature, the data may be punched out; when it has the print feature, the data may be printed out. For the methods of setting the input/output equipment (setting #6003) and the baud rate (parameters #6026 through #6029) through this data input/output interface, refer to 4.9 DATA INPUT/OUTPUT INTERFACE.

4.7.1 OUTPUTTING PART PROGRAM TO PAPER TAPE

(1) The part program of the designated program number is punched out by the following operations:

a. Connect the external equipment such as the tape puncher to the NC via the data input/output interface.

b. Make the external equipment relay for operation.

c. Power on the NC.

4.7.1 OUTPUTTING A PART PROGRAM TO PAPER TAPE (CONT'D)

d. Select the EDIT mode.

e. Depress the PROG function key.

f. Check to see if the external equipment is ready.

g. Depress the RESET key.

h. Depress "O" key and key in program number.

i. Depress the OUT key.

The part program of the keyed-in program number is outputted to the external equipment. If it is a tape puncher, tape punch is performed. When the output of the part program is completed, the tape puncher stops automatically. During the output of data, "OUT" is flashing.

j. To discontinue the punch out operation, depress the RESET key. However, the discontinued operation cannot be resumed. Go back to f. and repeat the operations all over again.

NOTE When RESET, OUT are operated without keying in the program number, the part program of the currently displayed program number is outputted.

(2) Punch out of all part programs

All registered part programs may be outputted to the external equipment by the above operations except that 0 - 9 9 9 9 must be keyed in the operation of h.
All part programs stored in the memory are outputted (punched out, etc.) consecutively.

NOTE The contents of program number O0000 are outputted only when #6231D3 = 1.

4.7.2 OUTPUTTING TOOL OFFSETS TO PAPER TAPE

Tool offsets may be outputted to paper tape by the following operations.

(1) Connect the external equipment such as the tape puncher to the NC via the data input/output interface.

(2) Make the external equipment ready for operation.

(3) Power on the NC.

(4) Select the EDIT mode.

(5) Depress the OFS function key.

(6) Check to see if the external equipment is ready.

(7) Depress the RESET key.

(8) Depress the OUT key.

All contents of the offset memory such as tool offsets and tool coordinate data are outputted to the external equipment. If the external equipment is a tape puncher, tape punch out is performed. When the output of the data is completed, the external equipment stops automatically.

(9) To discontinue the punch out operation, depress the RESET key. However, the discontinued operation cannot be resued. Go back to (6) and repeat the operations all over again. The format of the punched out tape is the same as that of the tape describe in 4.4.4, (1).

4.7.3 OUTPUTTING SETTING/PARAMETER DAT TO PAPER TAPE

Setting/parameter data may be punched out by the following operations.

(1) Connect the external equipment such as the tape puncher to the NC via the data input/output interface.

(2) Make the external equipment ready for operation.

(3) Power on the NC.

(4) Select the EDIT mode.

(5) Depress the PRM function key if the output of both setting data and parameter is desired. If the output of only setting data is desired, depress the SET function key.

(6) Check to see if the external equipment is ready.

(7) Depress the RESET key.

(8) Depress the OUT key.

All setting data and parameters are outputted to the external equipment. If it is a tape puncher, tape punch out operation is performed. When the output of the data is completed, the external equipment stops automatically.

(9) To discontinue the punch out operation, depress the RESET key. However, the discontinued operation cannot be resumed. Go back to (6) and repeat the operations all over again. The format of the punched out tape is the same as that of the tape described in 4.4.5, (1).

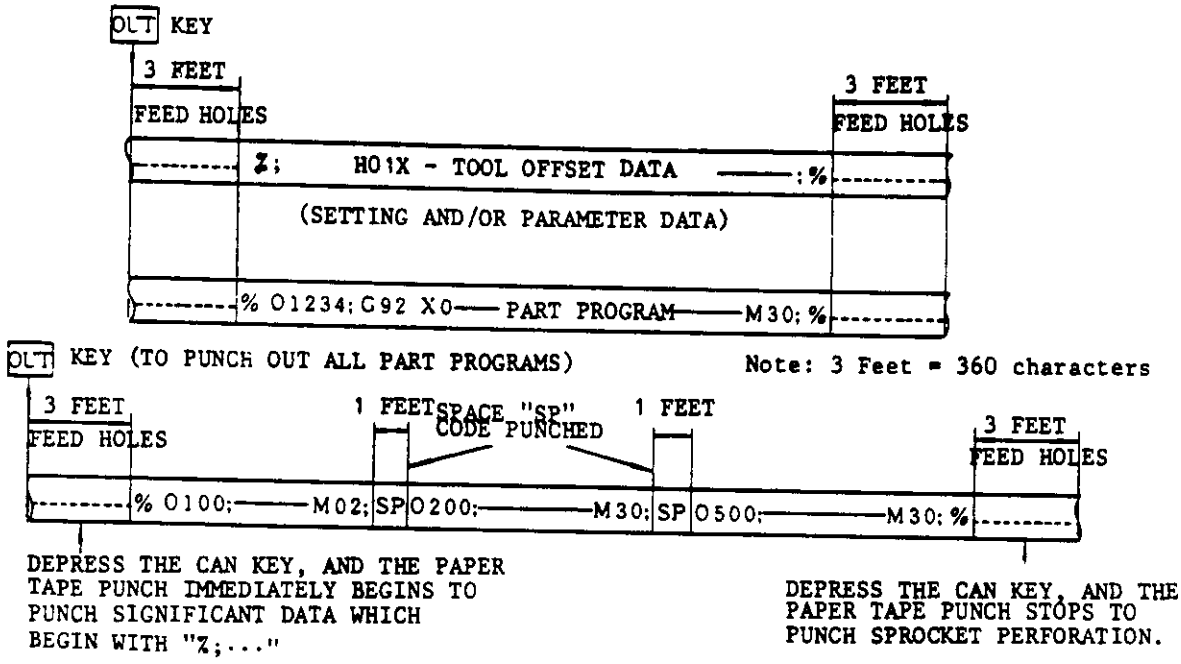


Fig. 4.50 Data and Program Formats on Paper Tape

4.8 SUMMARY OF STORING AND EDITING OPERATIONS

	Operation			Edit Lock	System No Switch	Mode	Function	Procedure
Parameter	Storing from NC operator's panel keyboard				1		PRM	Parameter number → CURSOR Data → WR
	Storing from tape (Note 4) (Note 6)				1	EDIT		RESET → IN
	Punch out (Note 3)							RESET → OUT
	Matching with tape (Note 4)							RESET → VER
Setting	Storing from NC operator's panel keyboard						SET	Setting number → CURSOR Data → WR
	Storing from tape				1	EDIT		RESET → IN
	Punch out							RESET → OUT
	Matching with tape							RESET → VER
Offset	Storing from NC operator's panel keyboard						OFS	Offset number → CURSOR Data → WR
	Storing from tape					EDIT		RESET → IN
	Punch out							RESET → OUT
	Matching with tape							RESET → VER
	Clear of all offsets							O → -9999 → ORG
Part program	Storing from NC operator's panel keyboard			OFF		EDIT	PROG	O → Program number → WR Repeat of edit opeation "addition of address data"
	Storing from tape	One part program	Tape with number O	OFF				RESET → IN
			Tape without number O	OFF				RESET → O → Program number → IN
		All part programs on tape		OFF				RESET → O → -9999 → IN
		Addition to registered part program		OFF				RESET → NEXT → IN
	Punch out	Designated part program						RESET → O → Program number → OUT
		All part programs						RESET → O → -9999 → OUT
	Matching with tape	One part program	Tape with number O					RESET → VER
			Tape without number O (Note 1)					RESET → O → Program number → VER
		All part programs on tape						RESET → VER
	Edit	Modify of address data (Note 2)		OFF				CURSOR (Set to address data to be modified) → Address data → ALTER
		Add of address data (Note 2)		OFF				CURSOR (Set to address data just before addition) → Address data → INSRT
		Delete of one address data		OFF				CURSOR (Set to address data to be deleted) → ERASE
		Delete of one block (Note 5)		OFF				CURSOR (Set to address data at head of block to be deleted) → EOB → ERASE

	Operation		Edit Lock	System No. Switch	Mode	Function	Procedure
Part program	Address search				TAPE MEM EDIT	PROG	Address data to be searched → CURSOR
	Clear	Designated part program	OFF		EDIT		O → Program number to be searched → ERASE
		All part programs on tape	OFF				O → -9999 → ERASE

Notes:

1. Storing of a part program having a program number different from program number 0 on tape is performed by the same operation as for "tape without program number 0."
2. Within the limit of 32 characters, addition of multiple address data and the change to one address data are permitted.
3. Setting is punched out at the same time.
4. If the tape contains setting information, it is also stored and matched at the same time.
5. When the cursor to the address data in the middle of a block and EOB and ERASE keys are depressed, the data following the cursor position is deleted.
6. When data has been stored from a parameter tape, turn the power on and off.

4.9 DATA INPUT/OUTPUT INTERFACE

The input/output of the following NC information may be performed by connecting the external equipment having the designated input/output interface to the NC:

(1) Part program.
(2) Tool offsets, tool coordinate data, and tool wear amount.
(3) Setting data and parameters.

4.9.1 TYPES OF INTERFACES AND FUNCTIONS

No.	1	2	3	4
Name of interface	FACIT4070 interface	Current loop (20 mA) interface	RS232C interface	RS422 interface
Type of interface	Parallel voltage interface	Serial current interface	Serial voltage interface	Serial balanced interface
Data transfer rate	70 char/s	Parameter setting	input: #6020 output: #6028	Input. #6027 Output: #6029
Coupling connector (Note 1)	MR-20MR		DB-25S	DB-37S
Max. cable length	5 m	50 m	15 m	100 m
Subject external equipment	FACIT4070 or equivalent equipment with I/F	ASR-33 or equipment with current loop (20 mA) I/F	RS232C or equipment with I/F	RS422 or equipment with I/F
Function	(1) Above NC information is outputted to external equipment (chiefly for tape punch out)		(1) Above NC information is stored into NC memory or matched. (2) Above NC information is outputted to external equipment (punch out). (3) Automatic run in TAPE mode (in place of tape reader).	

Note: This is the format of the connector on the NC side. For mating connector, use the following: MR-20F, DB-25P, and DB-37P

4.9.1 TYPES OF INTERFACES AND FUNCTIONS (CONT'D)

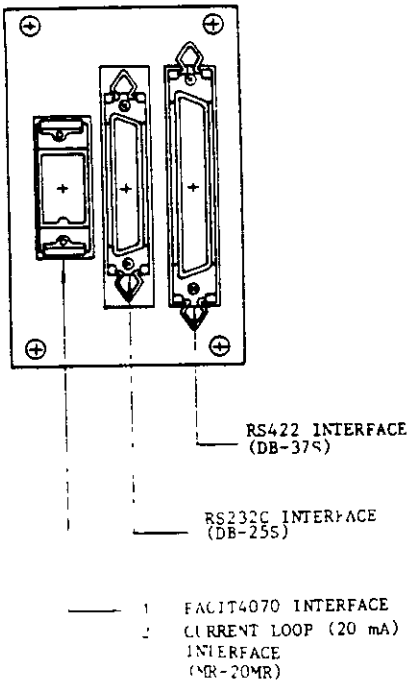


Fig. 4.51 Data Input/Output Interface

4.9.2 SETTING INPUT/OUTPUT INTERFACE TO BE USED

To use data input/output interface, it is necessary to set its type as follows.

(1) Setting of data input interface

IDVCE1 (#6003, D1)	IDVCE0 (#6003, D0)	Data Input Interface to be Used
0	0	PTR interface (Note 2)
0	1	RS232C interface
1	0	RS422 interface

PTR interface is dedicated to the tape reader unit (option). For details, refer to 5.1.2 Tape Reader Unit.

(2) Setting of data output interface

ODVCD1 (#6003, D5)	ODVCE0 (#6003, D4)	Name of Interface
0	0	FACIT4070 interface
0	1	Current loop interface RS232C interface
1	0	RS422 interface

4.9.3 BAUD RATE OF SERIAL INTERFACES AND SETTING

To use the serial interface (current loop, RS232C, and RS422), it is necessary to set the baud rate, stop bit length, and control code transmission designation to the parameter.

(1) Current loop and RS232C interface

According to the specifications of the equipment to be connected, input setting and output setting may be performed separately as follows:

a. Setting of baud rate

	For input	#6026 D3	#6026 D2	#6026 D1	#6026 D0
	For output	#6028 D3	#6028 D2	#6028 D1	#6028 D0
Baud rate value	50	0	0	0	0
	100	0	0	0	1
	110	0	0	1	0
	150	0	0	1	1
	200	0	1	0	0
	300	0	1	0	1
	600	0	1	1	0
	1200	0	1	1	1
	2400	1	0	0	0
	4800	1	0	0	1
	9600	1	0	1	0

b. Setting of stop bit length

For input: #6026 D_4 = 1 : 2 stop bits

For output: #6028 D_4 = 0 : 1 stop bit

c. Setting of cotrol code transmission designation

For input: #6026 D_5 = 1: Control code is not transmitted.

For output: #6028 D_5 = 0: Control code is transmitted.

(2) RS422 interface

According to the specifications of the equipment to be connected, input setting and output setting may be performed separately as follows:

a. Setting of baud rate

Baud rate value	For input	#6027 D3	#6027 D2	#6027 D1	#6027 D0
	For output	#6029 D3	#6029 D2	#6029 D1	#6029 D0
	50	0	0	0	0
	100	0	0	0	1
	110	0	0	1	0
	150	0	0	1	1
	200	0	1	0	0
	300	0	1	0	1
	600	0	1	1	0
	1200	0	1	1	1
	2400	1	0	0	0
	4800	1	0	0	1
	9600	1	0	1	0

b. Setting of stop bit length

For input: #6027D_4 = 1 : 2 stop bits

For output: #6029D_4 = 0 : 1 stop bit

c. Setting of control code transmission designation

For input: #6027D_5 = 1: Control code is not transmitted

For output: #6029D_5 = 0: Control code is transmitted

4.9.4 CONNECTING SPECIFICATIONS OF CABLE CONNECTORS

The connecting specifications of the cable connectors for each data input/output interface are as shown in Figs. 9.4.2 through 9.4.6. These specifications are simply for reference in this publication because they depend on the external equipment to be connected. Refer to the instruction manual of the external equipment.

4.9.5 OPERATION WITH DATA INPUT/OUTPUT INTERFACE USED

Storing of data from the memory to the punch out memory matching of the data, and TAPE mode automatic run are performed in the same operations regardless of the type of the data input/output interface used. For details, refer to the associated paragraphs as follows:

(1) Storing data into memory

4.4 LOADING PART PROGRAMS AND NC DATA INTO MEMORY.

(2) Matching with memory

4.5 TAPE VERIFYING.

(3) Punch out from memory

4.7 PART PROGRAM AND NC DATA OUTPUT OPERATION.

(4) TAPE mode automatic run

Each TAPE mode automatic run operation described in this publication.

4.9.5 OPERATION WITH DATA INPUT/OUTPUT INTERFACE USED (CONT'D)

Table 4.2

NC (MR-20F)			Connections	Interface Connecting Cable (DB-25P)	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
PR	PUNCH READY	1		12	PR
TL	TAPE LOW	2		21	TL
ERR1	ERROR	3		20	ERR1
	Not Used	4			
+6 V	FACIT/ ASR Auto-selection	5		24	+6 V
	Not Used	6			
	Not Used	7			
0 V	GROUND	8			
0 V	GROUND	9		10	SD
0 V	GROUND	10		25	0 V
CH1	PUNCH DATA 1	11		1	CH1
CH2	PUNCH DATA 2	12		2	CH2
CH3	PUNCH DATA 3	13		3	CH3
CH4	PUNCH DATA 4	14		4	CH4
CH5	PUNCH DATA 5	15		5	CH5
CH6	PUNCH DATA 6	16		6	CH6
CH7	PUNCH DATA 7	17		7	CH7
CH8	PUNCH DATA 8	18		8	CH8
CH9	FEED HOLD	19		9	CH9
PI	PUNCH INSTRUC-TION	20		11	PI

Note

Note: The pin numbers at the time the external equipment is FACIT 4070 and its mating connector is DB-25P

Table 4.3

NC (MR-20F)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
		1			
	Not Used				
		4			
+6 V	FACIT/ ASR. Auto-selection	5			
TTY2	Current loop (-)	6			
TTY1	Current loop (+)	7			
0 V	GROUND	8			
		9			
	Not Used				
		20			

Note

Note Number of connector and pin is different with external equipment.

Table 4.4

NC (DB-25P)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
FG		1			FG
SD	Sending data	2			SD
RD	Sending data	3			RD
RS	Receiving data	4			RS
CS	Capable of sending	5			CS
	Not used	6			DR
SG	Signal grounding	7			SG
		8			IO BUS
	Not used				ER (OR IO ALARM)
		25			

Note: When the external equipment does not control the CS (Capable of Sending) signal giv from NC, short-circuit pins RS and CS on both ends of the cable.

Table 4.5

NC (DB-25P			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
FG		1			FG
SD	Sending data	2			SD
RD	Receiving data	3			RD
RS	Request sending	4			RS
CS	Capable of sending	5			CS
	Not used	6			DR
SG	Signal grounding	7			SG
		8			
	Not used				ER (OR IO ALARM)
		25			

Note: When the external equipment does not control the CS (Capable of Sending) signal given from NC, short-circuit pins RS and CS on both ends of the cable.

Table 4.6

NC (DB-37P)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
SHIELD	Shield	1			
	Not used	2			
	Not used	3			
SD	Sending data	4			SD
	Not used	5			
RD	Receiving data	6			RD
RS	Request sending	7			RS
	Not used	8			
CS	Cable of sending	9			CS
	Not used	10			
	Not used	11			
ER	NC ready	12			ER
DR	I/O device ready	13			DR
		14			
	Not used				
		18			
SG	Signal grounding	19			
	Not used	20			
	Not used	21			
*SD	Sending data	22			*SD
	Not used	23			
*RD	Receiving data	24			*RD
*RS	Request sending	25			*RS
	Not used	26			
*CS	Capable of sending	27			*CS
	Not used	28			
	Not used	29			
*ER	NC ready	30			*ER
*DR	I/O device ready	31			*DR
	Not used	32			
		37			

5. TAPE READER COMPARTMENT

5.1 TAPE READER

5.1.1 SYSTEM NO. AND TAPE FEED SWITCHES

These switches provided above tape reader are exposed by opening door for tape reader compartment.

(1) SYSTEM NO. switch

This switch has been fixed at "0" for normal operation and does not need operation. Parameter writing is made with the switch set at '1.' For details on its setting, see 4.1.13 SYSTEM NO. and TAPE FEED switches.

(2) TAPE FEED switch†

This switch is effective when the control is provided with a tape reader unit (option). This is a spring return switch for feeding and rewinding the tape manually. When once the switch lever is pushed toward the F (forward) direction, the tape keeps on feeding forward (from right to left), even when the lever is released. To stop the tape feed, flip the switch lever into the R (reverse) position. To return the tape, push the switch lever to the R position while the tape is standing still.
The switch cannot be activated during automatic and manual operation or with tape bail pushed up.

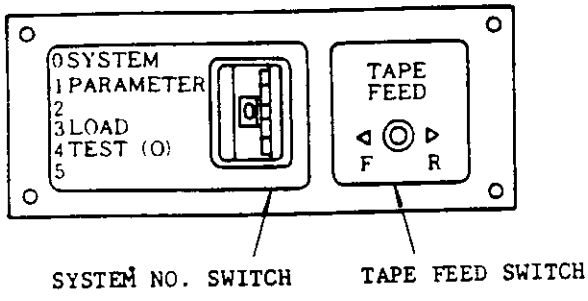


Fig. 5.1

5.1.2 TAPE READER UNIT†

When the control is provided with a tape reader unit (option), the following operation and actuation are possible.

(1) Automatic operation by part program tapes (TAPE mode).

(2) Storing of the contents of part program and NC data tape in the memory in the NC and their collation (EDIT mode).

However, to use the tape reader unit, set the setting function to PTR INTERFACE, by referring to the table below.

Table 5.1

Data Input Interface to be used	IDVCE 1 (#6003, D1)	IDVCE 0 (#6003, D0)
PTR Interface	0	0
RS232C Interface	0	1
RS422 Interface	1	0

The PTR interface is an exclusive interface for the tape reader option.

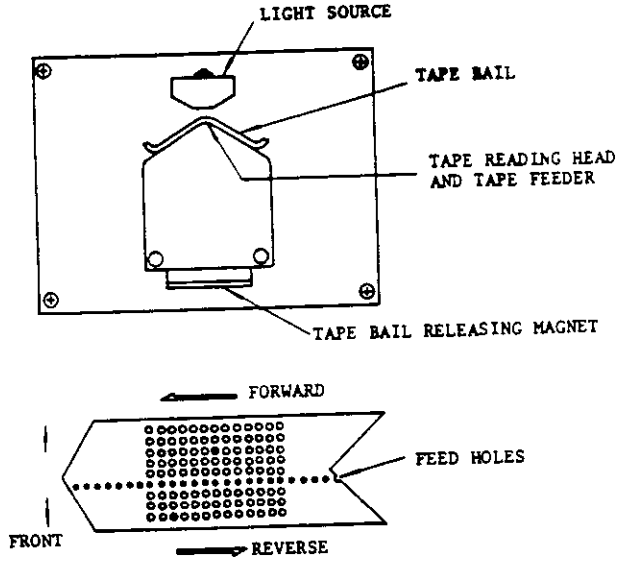


Fig. 5.2

Table 5.2 Specifications of Tape Reader Unit

Read speed & rewind speed	200 char./ sec	Speed not changed according to supply frequency.
Reading system	LED-photo-electric	

(1) Light source

LED is used for light source. It does not need maintenance operation except for removal of dust.

(2) Tape reading head and tape feeding part

Phototransistor is imbedded in the tape reading head and covered with glass. Scratch or dust on the glass causes misreading of tape reader. Make it clean periodically. See 8.1 ROUTINE INSPECTION. Feed holes on the tape should be set to the sprocket of the tape feeder.

(3) Tape bail

Push up the tape bail magnet to release tape bail, mount the tape, and push down the tape bail slowly. The tape reader will not operate until the tape bail is pushed down.

5.1.3 TUMBLE BOX

Tumble box is provided below the tape reader to accomodate NC tape. Tumble box capacity is different with type of control cabinet. See Table 5.6.3.1. The NC tape is easily taken out by pulling a braided nylon tape mounted inside the box as shown in Fig. 5.1.3. When the NC tape cannot be taken out, remove screws of tape outlet cover mounted on the lower part of the box. Clean the inside of the tumble box periodically referring to 8.1 Routine Inspection.

Type of Cabinet	Free-Standing	Built-in 1	Built-in 2	Unbundled
Tape Tumble Box Capacity	40 m	40 m	10 m	-

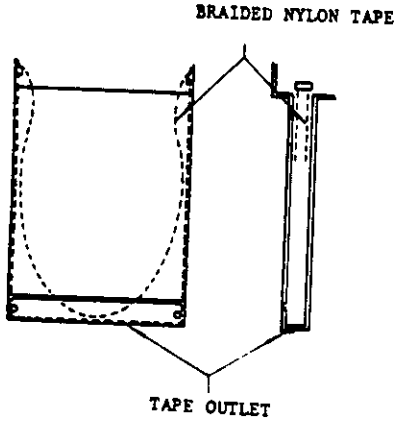


Fig. 5.3 Tumble box

5.2 TAPE REEL UNIT†

A free-standing type cabinet may be equipped with a tape reel unit (option) described below.

	Reel dia.	Tape length capacity
6" reel	150 mm	Approx. 80 m of NC tape (thickness: 0.108 mm)

This tape reel unit takes up slack tape, so that the tape winding speed and unwinding speed depend on the tape speed of the tape reader unit.

When the reels are not used, place the tension arms on the arm rests as shown in Fig. 5.2.1.

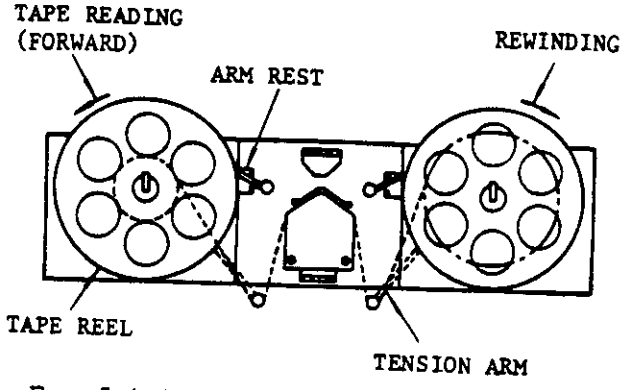


Fig. 5.4 Tape Reader with Tape Reels

When the tape is not in use, and when mounting and dismounting the tape, be sure to arrest the tension arms by the tension arm rest. While the tension arms are in the arrested positions, the reel motor is switched off, and the reels are free.

Mount the tape as follows.

(1) Pull the reel lock pawl on the right reel spindle to the horizontal position, and dismount the reel from the spindle.

(2) Insert the trailing end of the punched part program tape into the slit in the hub of the right reel, and wind the tape on the reel.

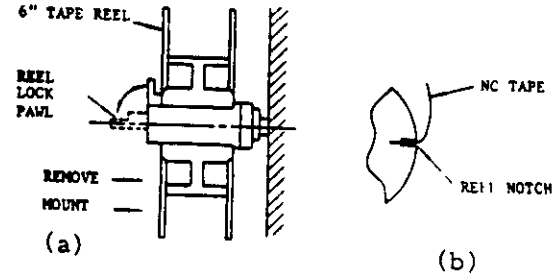


Fig. 5.5

(3) Mount a reel with a tape wound on it on the right reel shaft, align the reel lock slot with the reel lock pawl, and push the lock pawl to the vertical position to lock the reel to the shaft.

(4) Pull the tape approximately 1 m out of the reel, and pass it through the tape reader.

(5) Wind the free end of the tape on the left reel in the same way as (1) through (3), and mount the reel on the left reel shaft. At this time, wind the tape at least 3 turns to eliminate any possibility of slipping loose.

(6) Holding the reel with hand to prevent slackening of the tape, free the tension arm from the tension arm rest, and lower it gently. The broken lines in Fig. 5.2.1 show the correctly set tape

Now, the tape can be moved and wound smoothly in either directions in the automatic operation mode or by the manipulation of the TAPE FEED switch.

NOTES:

1. When dismounting the tape reel, observe the following two points.
   a. Hold the reel to be removed by hand, and let the tension arm be arrested by the tension arm rest.
   b. Push up the tape guide of the tape reading head beforehand.
2. Start operation only after making sure that the reel lock pawl is engaged in the lock slot in the reel

5.3 PORTABLE TAPE READER UNIT

When the control is equipped with an RS232C interface selected from the data I/O interface options, this portable tape reader unit can be connected to it for the following operations.

(1) Automatic operation by part program tapes (TAPE mode)

(2) Storing of part program tape contents and NC internal data tape contents into the memory in the NC, and their collation (EDIT mode)

However, to use the portable tape reader unit, set the setting data function to RS232C interface, by referring to the following table.

Table 5.3

Data Input Interface to be used	IDVCE 1 (#6003, D1)	IDVCE 2 (#6003, D2)
PTP Interface	0	0
➡ RS232C Interface	0	1
RS422 Interface	1	0

Table 5.4 Specifications of Portable Tape Reader Unit

Read and rewind speed	200 char./sec
Reading system	LED
Power supply	100 VAC, 50/60 Hz
Power lead	Approx. 2.5 m
Data I/O interface	RS232C Interface
Cable connector for data I/O interface	Approx. 3 m (DB-25P provided at both ends.) Note
Weight	Approx. 14 kg
Dimensions	Refer to Fig. 5.3.1

Note: Not always provided as an accessory.

(1) Make the following preparatory work with tape reader.

a. Unlatch the two draw latches, and pull up the top lid by the handle until the lock clicks.
The top lid is locked in the position permitting tape reader manipulation. See Fig. 5.3.1.

b. Unwind the power lead from the tape rear, and plug its plug into a 100 VAC commercial power outlet.

c. Connect the portable tape reader to the NC with the RS232C cable connector delivered with the tape reader (or prepared by the user). When the delivered cable connector is used, plug the connector marked with a "1RO" nameplate to the 1RO connector on the NC (Fig. 5.3.2).

d. Flip the POWER toggle switch on the left side of the tape reader to ON.
The portable tape reader is now energized. Make sure that the POWER LED (red) lights.

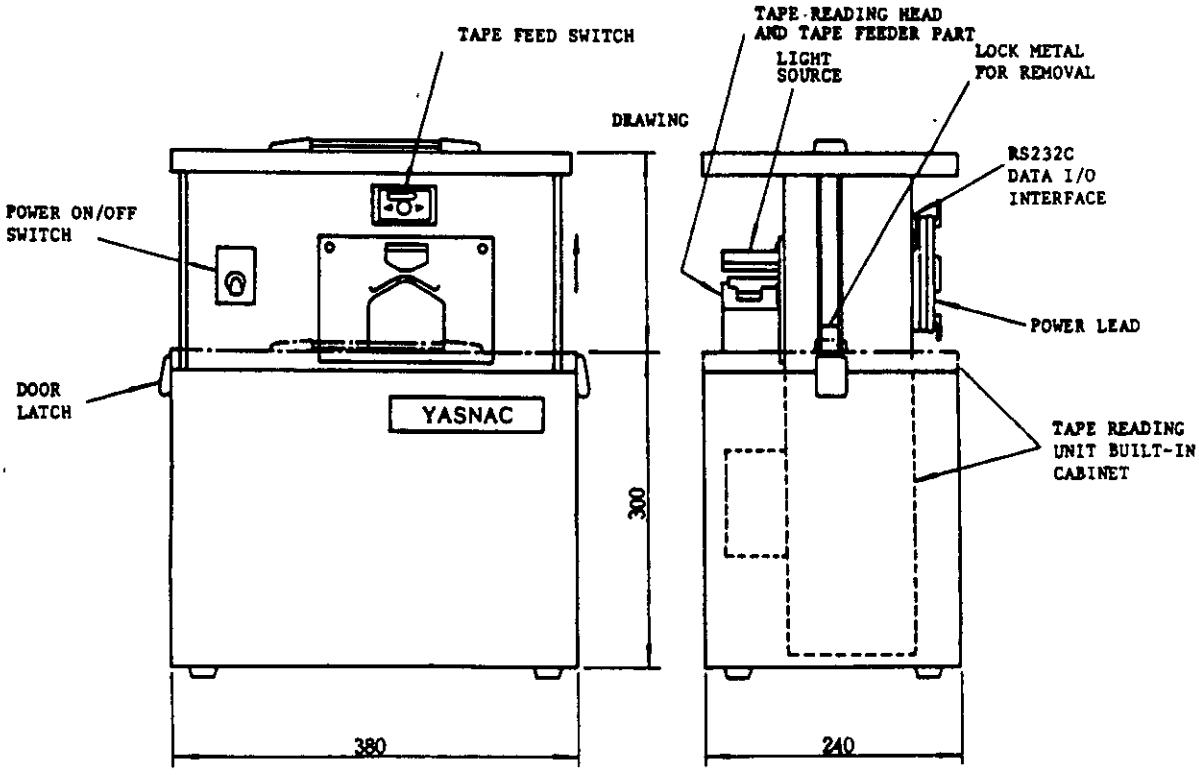


Fig. 5.6 Dimensions of Portable Tape Reader

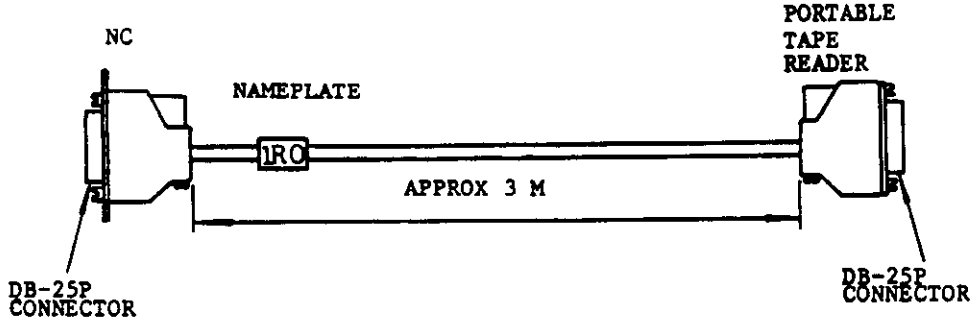


Fig. 5.7 Cable Connector for RS232C Interface

5.3 PORTABLE TAPE READER UNIT (CONT'D)

Now, the portable tape reader is ready for use. It can be used in the normal TAPE mode for tape operation (Paragraph 7.6), and for storing and collating contents of part program tapes and NC internal data tapes (Paragraphs 4.4 and 4.5).

(2) After using the tape reader, put it in the box and store it as follows.

a. Flip the POWER toggle switch to OFF.

b. Unplug the RS232C cable connector at both ends, and store it separately in a vinyl bag or the like.

c. Unplug the power lead from the outlet socket, and wind the lead on the lead wind fixture.

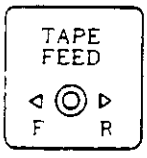
d. Holding the top lid handle, push the lock fixtures on both sides inward to unlock them, and carefully let the tape reader enter into the box.

e. Lock the top lid with the two draw latches firmly. Store the tape reader where the air is free from oil mist and humidity.

(3) The operation of the tape reader members is as follows.

a. TAPE FEED switch

This is a spring return switch for feeding the tape forward and backward manually. Pushing the switch lever to the F (forward) position feeds the tape forward (from right to left), and vice versa. The tape is fed only while the switch lever is kept tilted.



b Light Source

As the light source, an LED is used. It requires no maintenance except for daily cleaning.

c. Tape Reading Head and Tape Feeder

In the tape reading head, phototransistors are built in under a glass window. Since dust and scratches on the glass window are liable to cause reading errors, it should be cleaned periodically, and handled with care. Refer to 8.1 Routine Inspection Schedule. Located in the center is the tape feed sprocket, for engaging the feed holes of the part program tape.

d. Tape Guide

For feeding a tape through the tape reader, push up the tape guide release magnet under the reading head to raise the tape guide, pass the tape over the reading head, and then, push down the tape guide gently. With the tape guide lifted, cycle start in the TAPE mode cannot be made, and the TAPE FEED switch is ineffective. The sprocket is freed from the feed holes so that the tape can be pulled freely in both directions.

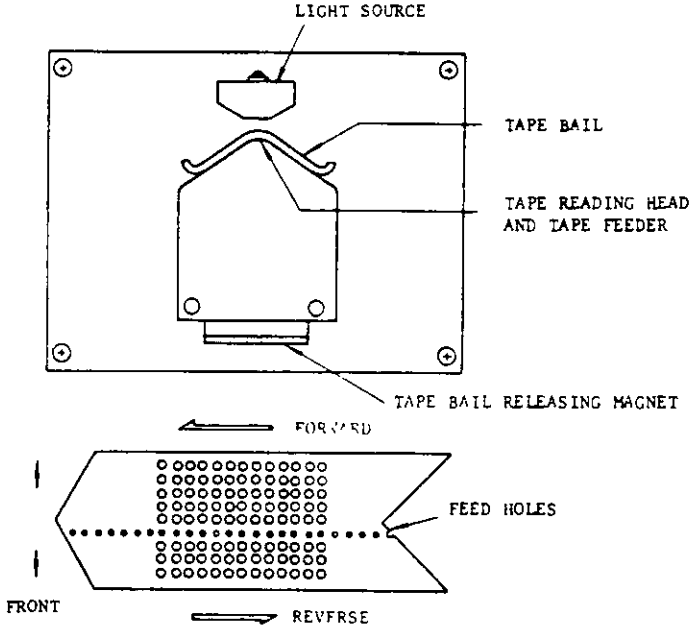


Fig. 5.8

6. MACHINE CONTROL STATION

6.1 SWITCHING UNITS ON THE CONTROL STATION

Fig. 6.1 shows a typical layout of switching units on the machine control station. For details, refer to the machine tool builder's manual.

6.1.1 MODE SELECT SWITCH

This switch selects operation mode of the NC system and consists of 6 positions (JOG, HANDLE, TAPE, MDI, MEM, EDT).

(1) JOG: To feed the tool continuously by manual opearation. Feedrate is set by FEEDRATE OVERRIDE switch.

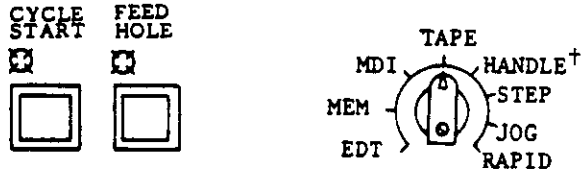
(2) HANDLE/STEP: To feed the tool by operating the manual pulse generator†. Where the control is not provided with a manual pulse generator, the tool is fed by step manually operating the JOG PUSHBUTTON(S).

(3) TAPE: To automatically control the NC system with NC tape.

(4) MDI: To enter the block of data through the DATA keyboard and control the system automatically with the data.

(5) MEM: To automatically control the system with the stored part program.

(6) EDT: To store the part program into memory and edit the part program.



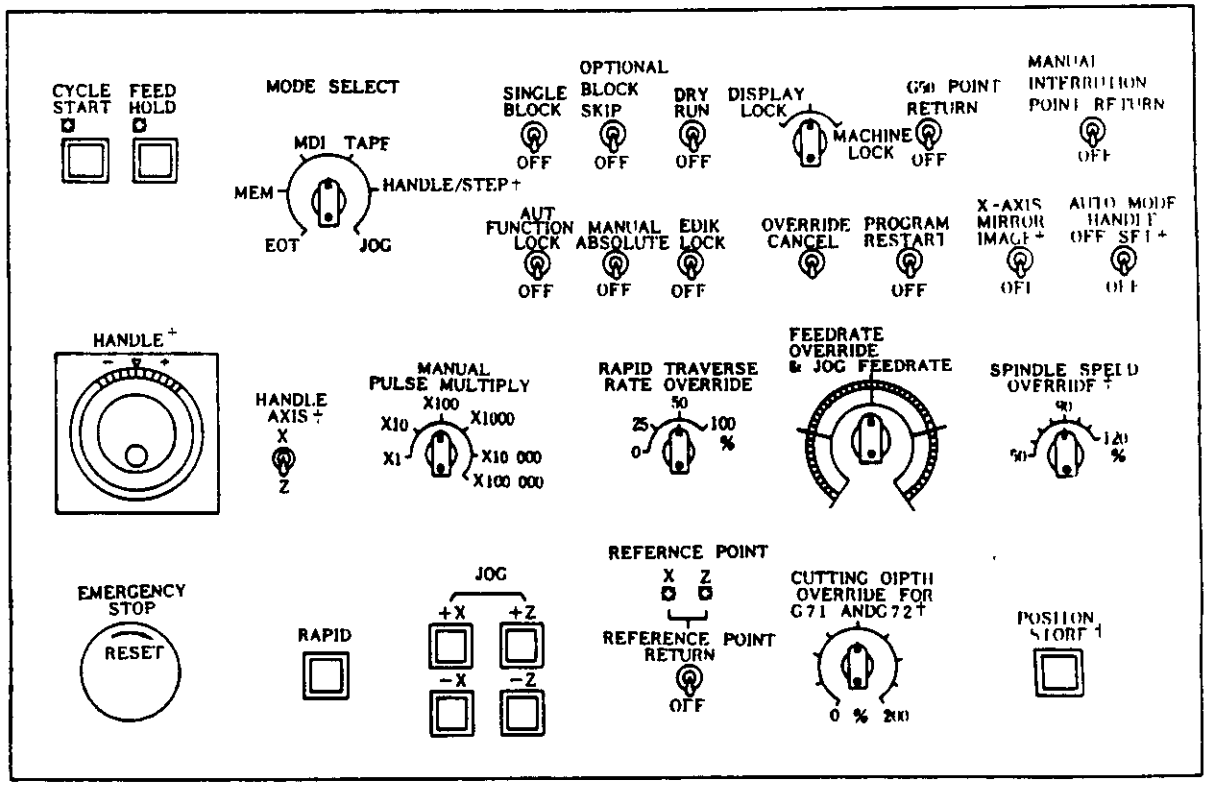


Fig. 6.1

6.1.2 CYCLE START PUSHBUTTON AND LAMP

Depress this pushbutton to start the system in the automatic operation mode (TAPE, MDI and MEM). The CYCLE START indicating lamp lights when automatic operation starts. Depress it again to start the operation after temporary stop by operating FEED HOLD pushbutton or MODE SELECT switch.

6.1.3 FEED HOLD PUSHBUTTON AND LAMP

Depress this pushbutton to temporarily stop automatic operation. The CYCLE START lamp goes off and the FEED HOLD lamp remains illuminated during temporary stop.

When the FEED HOLD pushbutton is depressed during feed operation, the feedrate is decreased immediately and the motion is stopped. Feedhold is not active during threadcutting by G32, G92, or G76 or dwell by G04. Rapid thread pull-up and retracting motion at G92 and G76 commands cannot be interrupted.
However, when the NC is provided with a "THREADING FEED HOLD" (option), the tool feed can be stopped temporarily even during thread cutting by the G92 or G76 command.

If it is depressed while M-, S-, or T-function without move command is being executed, the FEED HOLD lamp will light, but these functions will be executed continuously. On completion of the function, the lamp goes off and machine operation is stopped.

Depress the CYCLE START pushbutton to restart the operation after temporary stop by operating FEED HOLD pushbutton.

6.1.4 EMERGENCY STOP PUSHBUTTON

Depress this pushbutton to emergency-stop the machine. The servo power is turned off and the machine is stopped immediatly by dynamic brake. The RDY indication will disappear NC ALARM lamp lights and alarm code "330" is displayed

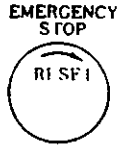
To restart the system after emergency stop, take the following procedure.

(1) Turn the EMERGENCY STOP pushbutton clockwise to release the locking.

(2) Depress the RESET key. Alarm code "310" replaces "330."

(3) Turn on the servo power again by depressing POWER ON pushbutton. NC ALARM LAMP is extinguished and READY lamp lights up.

The operation is effective in the reverse order of steps (2) and (3). Use this switch also for turning off the system.



6.1.5 HANDLE DIAL† (MANUAL PULSE GENERATOR)

The dial used as a manual pulse generator to feed the tool manually with the MODE SELECT switch set to the HANDLE. HANDLE operation is effective for an axis. Procedure of HANDLE operation is as follows:

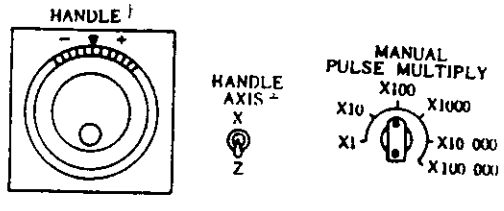
(1) Set the MODE SELECT switch to the HANDLE.

(2) Select the axis to be operated with HANDLE AXIS select switch.

(3) Set the move amount per graduation of the dial by setting MANUAL PULSE MULTIPLY switch. (See Table 6.1.7.1.)

(4) Rotate the dial to move the selected axis.

Turning it clockwise causes the axis to move in the plus direction. The axis moves in the minus direction by turning it counterclockwise.



6.1.6 HANDLE AXIS SELECT SWITCH †

This switch is used to select an axis to be operated. For operation of the switch, see 6.1.5 Handle Dial†.

6.1.7 MANUAL PULSE MULTIPLY SELECT SWITCH†

This switch is used to:

(1) Select the value from Table 6.1.7.1 corresponding to a single graduation of the HANDLE dial in the HANDLE mode.

(2) Select the move amount (1 step) from Table 6.2 corresponding to each depression of JOG pushbutton in the STEP mode.

Table 6.1 Selection of Move Amount in the HANDLE Mode†

	Metric	Inch
x 1	0.001 mm/ graduation	0.0001 in/ graduation
x 10	0.01 mm/ graduation	0.001 in/ graduation
x 100 x 1000 x 10000 x 100000	0.1 mm/ graduation	0.01 in/ graduation

Table 6.2 Selection of Move Amount in the STEP Mode

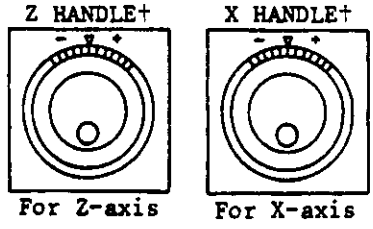
	Metric	Inch
x 1	0.001 mm/step	0.0001 in/step
x 10	0.01 mm/step	0.001 in/step
x 100	0.1 mm/step	0.01 in/step
x 1000	1.0 mm/step	0.1 in/step
x 10000	10.0 mm/step	1.0 in/step
x 100000	100.0 mm/step	10.0 in/step

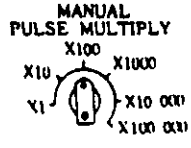
6.1.8 HANDLE DIALS FOR SIMULTANEOUS CONTROL OF UP TO TWO AXES†

When a manual pulse generator is connected for each axis, the tool can be manually moved along selected two axes simultaneously.

(1) The tool move distance per graduation of the HANDLE dial for the manual pulse generator is determined by the MANUAL PULSE MULTIPLY switch (Table 6.1.7.1). This switch is effective on all the three axes.

(2) Set the mode select switch to HANDLE, and turn the HANDLE dials for the desired axes in the positive or negative direction.





6.1.9 JOG PUSHBUTTONS AND RAPID PUSHBUTTON

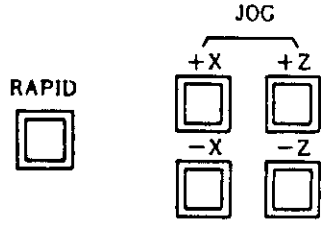
This pushbutton is used to feed the tool manually.

(1) With any of pushbuttons +X, -X, +Z, or -Z with RAPID button depressed, the axis can be moved rapidly until the button is released.

(2) These pushbuttons move the tool at the speed set by JOG FEEDRATE switch in the JOG mode.

(3) Each time the pushbutton is depressed in the STEP mode, the tool is moved by the value per step set by MANUAL PULSE MULTIPLY select switch. Maximum feedrate per step is determined by parameter # "6222."

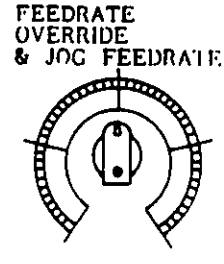
NOTE: JOG pushbuttons work on two axes simultaneously.



6.1.10 JOG FEEDRATE SWITCH AND FEEDRATE OVERRIDE SWITCH

In the automatic operation mode (TAPE, MEM, MDI), this switch is used to adjust the feedrate by 10% from 0 to 200% of the programmed feedrate specified with an F function at whatever position the switch may be set. Feed during tapping by G32, G92, and G76 follows F command. Where OVERRIDE CANCEL switch is set on, the tool will be moved at the programmed feedrate by F code regardless of switch setting.

The JOG FEEDRATE switch is used to select the jog feedrate in the JOG mode. Up to 32 steps of feedrate can be specified. Jog feedrate depends on the machine tool. For definite values, refer to the machine tool builder's manual. See Table 6.1.10. The JOG feedrate can be preset by parameters #6233 to 6264.



6.1.10 JOG FEEDRATE SWITCH AND FEEDRATE OVERRIDE SWITCH (CONT'D)

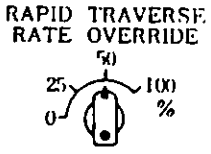
Table 6.3 Jog Speed and Feedrate Override

Step	Feedrate Override	Jog Feedrate	
		Parameter No.	mm/min
0	0%	#6233	0
1	10%	#6234	1
2	20%	#6235	2
3	30%	#6236	4
4	40%	#6237	6
5	50%	#6238	8
6	60%	#6239	10
7	70%	#6240	12
8	80%	#6241	15
9	90%	#6242	20
10	100%	#6243	25
11	110%	#6244	30
12	120%	#6245	40
13	130%	#6246	50
14	140%	#6247	60
15	150%	#6248	80
16	160%	#6249	100
17	170%	#6250	120
18	180%	#6251	150
19	190%	#6252	200
20	200%	#6253	250
21	0%	#6254	300
22		#6255	400
23		#6256	500
24		#6257	600
25		#6258	800
26		#6259	1000
27		#6260	1200
28		#6261	1500
29		#6262	2000
30		#6263	2500
31		#6264	3000

Note: Jog feedrate depends on the machine tool. For definite values, refer to the machine tool builder's manual.

6.1.11 RAPID TRAVERSE RATE OVERRIDE SWITCH

This switch is used to adjust the traverse rate by F0, 25, 50 and 100%. 100% Rate is the rapid traverse rate set by parameter #6280 and #6281. The switch is effective both in automatic operation including G00 command and in manual operation (RAPID mode). F0 is set by parameter #6231.

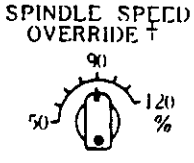


6.1.12 FEEDRATE OVERRIDE CANCEL SWITCH

Turning on the FEEDRATE OVERRIDE CANCEL switch prevents the function of FEEDRATE OVERRIDE switch.
Feedrate is fixed at 100%.

6.1.13 SPINDLE SPEED OVERRIDE SWITCH†

With this switch, the current spindle speed can be changed to an override speed which is set at 10% intervals between 50 and 120% of the current spindle speed.



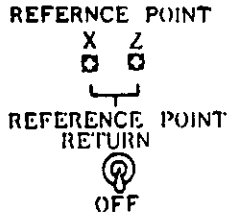
6.1.14 MANUAL REFERENCE POINT RETURN SWITCH†

This switch is for bringing the tool back to the reference point manually.

For its operation method, refer to 6.2.1 Manual Return To Reference Point.

6.1.15 REFERENCE POINT LAMPS

These lamps indicate that the tool is positioned on the reference point. They light when the tool is brought to the reference point through the manual or automatic return to reference point (G28), or by the reference point return check (G27), and goes out as the tool moves away from the reference point by a subsequent operation.



6.1.16 SINGLE BLOCK SWITCH

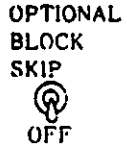
With this switch turned on, individual block-by-block operation is obtained. A block of data is executed each time the CYCLE START pushbutton is activated. In the automatic operation mode, the machine stops by turning on this switch after finishing the current block.

6.1.17 OPTIONAL BLOCK SKIP SWITCH

This switch selects whether the data in blocks including a "/" is disregarded or not.

(1) While the switch is on, all the commands in a block programmed after a "/" are neglected. However, block data appearing before the "/" remains effective.

(2) While this switch is off, blocks including a "/" are executed along with other blocks. This switch is ineffective on the block under execution and blocks stored in the advance-reading buffer. When this switch is turned on during an automatic operation cycle, it works on the block read after the switching on has occured.



NOTES:

1. The two commands "/" and "/1" are equivalent.
2. With the control provided with the optional block skip B function†, 8 independent blocks can be skipped with the switching of the switches corresponding to "/2" through "/9."

6.1.18 DRY RUN SWITCH

Turning on the DRY RUN switch in the TAPE, MDI or MEM mode causes the tool to move at the speed selected by the JOG FEEDRATE switch, ignoring all programmed F-functions. F commands can be displayed as they are programmed. This switch may be used to check the program.

Rapid traverse (G00) rate for dry run operation can be set by setting parameter # "6006D2."

Parameter # "6006D2"	Rapid Traverse at Dry Run Operation
"0"	Rapid traverse rate
"1"	Jog feedrate

NOTES:

1. Switching the DRY RUN switch during automatic operation becomes effective on the current block. Switching it in mm/rev mode becomes effective on the next block.
2. Rapid traverse rate override is kept effective during dry run operation.

6.1.19 DISPLAY LOCK/MACHINE LOCK SWITCH

This switch functions to stop updating the universal display, or to stop move command pulses to the servos. Stop the machine to operate the switch.

"OFF"

Usual operation is made at "OFF" position in both manual and automatic operation. The machine and universal display operate according to the command by CYCLE START operation or manual operation.

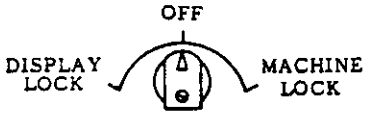
"DISPLAY LOCK"

This position is used to exclude the tool movement value from the display. Universal display is not updated, though the machine moves.

6.1.19 DISPLAY LOCK/MACHINE LOCK SWITCH (CONT'D)

"MACHINE LOCK"

With the switch at MACHINE LOCK, axis movement including Zero Return is inhibited. The position display is updated. M-, S-, and T-functions are executed. This position is selected to preset the display or to check the tape data.



6.1.20 M-FUNCTION LOCK SWITCH (AUXILIARY FUNCTION LOCK)

(1) When the M-FUNCTION LOCK switch is on, it ignores the M, S, and T commands. To check the tape data, the operation by the switch is used in combination with MACHINE LOCK function.

(2) The following M codes are executed even if the switch is set on.

a. M00, M01, M02, M30

Both its decoded signals and its BCD codes are sent out to the machine.

b M90 to M109 (Internal processing M code)

BCD code is not sent out.

(3) Turning on the M-FUNCTION LOCK switch during automatic operation becomes effective on the block after the next block of the current block.

(4) This switch does not affect S 4-digit programming (option).

6.1.21 MANUAL ABSOLUTE SWITCH

(1) When MANUAL ABSOLUTE switch is on.

When automatic operation is restarted after interrupted by manual operation, the tool performs the rest of the command in the interrupted block from the end point of manual operation. The tool moves in parallel with the path specified by the program.

When the command of the next block is G00 or G01, the tool moves automatically to the coordinate specified by the program.

Then the operation is performed according to block of data.

When the command of the next block is G02 or G03 (circular interpolation), the interpolation is performed in parallel with program command. The tool automatically returns to the target coordinate when G00 or G01 is commanded after the interpolation.

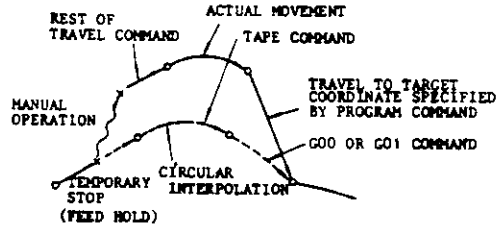


Fig. 6.2 Tool Movement with MANUAL ABSOLUTE Switch On

(2) When MANUAL ABSOLUTE switch is off

After the automatic operation is interrupted by manual operation, the coordinate system is shifted. Therefore the tool performs the reset of the travel command and continues operation in parallel with program command.

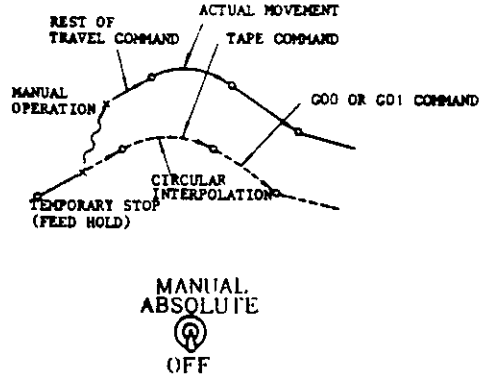


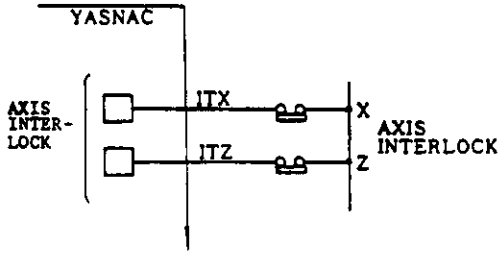
Fig. 6.3 Tool Movement with MANUAL ABSOLUTE Switch Off

6.1.22 EDIT LOCK SWITCH

Turning on the EDIT LOCK switch prevents the function of ERS, INS, and ALT keys, and storing from NC tape. When editing is made with EDIT LOCK switch turned on, "EDIT LOCK" flickers on the CRT display.

6.1.23 INTERLOCK INPUT (INTERLOCK)

INTERLOCK INPUT is used for stopping the axis movement during automatic operation. Turning on the interlock (closed) during axis movement in automatic operation mode stops the axis with automatic operation activation lamp (STL) lighting. Turning off the interlock resumes the axis motion. The interlock input does not affect the movement by manual operation.



6.1.24 G50 POINT RETURN SWITCH†

This switch is for returning the tool to the coordinate system setup point (where G50 has been programmed) manually. For its usage, refer to 6.2.5 Setup Point Return.

6.1.25 MANUAL INTERRUPTION POINT RETURN SWITCH

This switch is for manually returning the tool to where the NC was switched over from the AUTO mode to the MANUAL mode in order to make intervention with manual control. For the usage, refer to 2.2.6 Manual Interruption Point Return.

6.1.26 PROGRAM RESTART SWITCH†

This switch is for restarting the part program from any desired sequence No. For the usage, refer to 6.2.7 Program Start.

6.1.27 X-AXIS MIRROR IMAGE SWITCH†

This switch is for turning on the MIRROR IMAGE function with respect to the Z axis. When this switch is turned on, the sign of all the X-coordinate command values is reversed. The mirror image function is effective on all the X-coordinate values including those by G50. This switch is effective only while the offset function is cancelled on X-coordinate values.

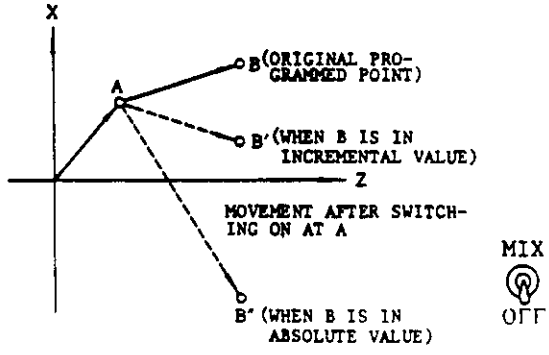


Fig. 6.4 Motion with the Switch Turned on at Point A

6.1.28 AUTO MODE HANDLE OFFSET SWITCH†

This is a switch for enabling tool motion through the use of manual pulse generator. For the usage, refer to Paragraph, 6.2.8 Auto Mode Handle Offset.

6.1.29 CUTTING DEPTH OVERRIDE SWITCH† FOR G71 AND G72

This switch is for effective cutting depth override on the specified depth of cut command D in outside rough turning cycle (G71) and face rough turning cycle (G72) between 10 and 200% at 10% increments. However, to make this switch effective, Parameter #6023D2 should be set to 1 in advance.

6.1.30 POSITION STORE PUSHBUTTON†

This button switch is for directly inputting measured workpiece values.

When this button is pushed, the current tool values (position or external display values) are stored temporarily in the register, the MEASURED WORKPIECE VALUE DIRECT INPUT mode is turned on, and the LED of the OFS function key flickers. At the same time, the CRT displays the tool offset value diagram.

For detailed procedure, refer to 6.2.3 Measured Workpiece Value Direct Input†.

6.2 OPERATION PROCEDURE

6.2.1 MANUAL RETURN TO REFERENCE POINT

With this function, the tool is returned to the reference point manually. The procedure is as follows.

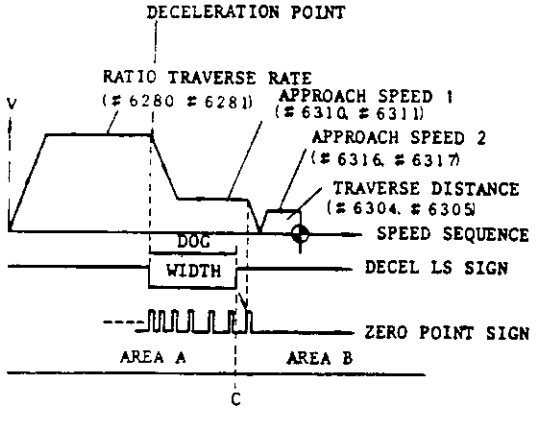
(1) Set the mode select switch to RAPID or JOG.

(2) Manually move the tool to a position some distance away from the reference point. When the tool is within the range A shown below, it can be brought back to the reference point in the normal way, as described below.

(3) Turn on the REFERENCE POINT RETURN switch.

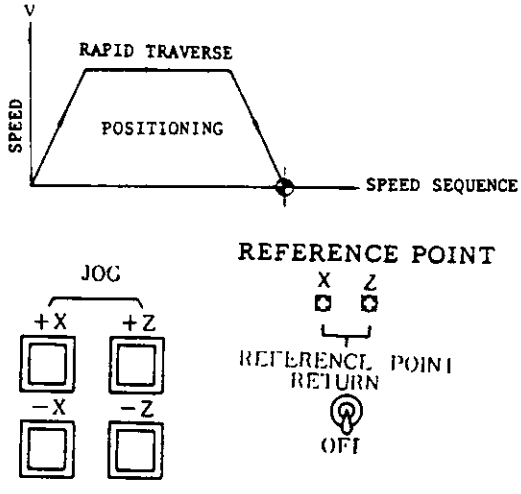
(4) Keep the JOG button for the return direction depressed. The tool starts to move as in the normal manual control, but the speed is decelerated at the deceleration point, and the motion stops automatically at the reference point

(4) Then, the REFERENCE POINT lamp for the relevant axis lights.



NOTES:

1. As long as the power supply is turned on, either the manual or the automatic return to reference point can be initiated, regardless of the tool position, but the tool will not return to the reference point accurately if the tool is started from a point in the area B. Be sure to bring the tool into the area A before initiating a manual or automatic return motion.
2. Once the tool is returned to the reference point, the point C is stored, and if the reference return motion is initiated from a point in the area B, this is regarded as an error. Start the reference return motion from a position in the area A.
3. Once the tool is returned to the reference point, it can not be further moved in the same direction unless the REFERENCE POINT RETURN switch is turned off.
4. While the MACHINE LOCK switch is on, the reference point return function is ineffective.
5. Do not return the tool to the reference point by the manual reference point return function, while the buffer is loaded with blocks read in advance of execution, because the stored motion data will be erased by the reference point return motion.



6.2.2 AUTOMATIC COORDINATE SYSTEM SETTING†

With this function, a new coordinate system is set up automatically upon the return of the tool to the reference point by the manual reference point return function. The coordinates of the new origin are preset with the following parameters. The coordinate system set up by this function is equivalent to the ones set up by G50.

(1) Parameters for metric system

Parameter	Meaning
#6636	X-axis coordinate
#6637	Z-axis coordinate

(2) Parameters for inch system

Parameter	Meaning
#6630	X-axis coordinate
#6631	Z-axis coordinate

(3) Axis can be selected by parameter #6015 for both metric and inch systems.

6.2.3 MEASURED WORDPIECE VALUE DIRECT INPUT†

(1) Writing into tool coordinate memory

With this function, coordinate data for the respective tools required for setting G50T work coordinate systems can be written into the TOOL COORDINATE MEMORY by simple processes. The required processes are as follows.

a. Bring the tool slide to the START point.

b. Set the current coordinate displayed values as (0, 0).
The "current coordinate displayed values" as meant here are values displayed as POSITION EXTERNAL on the CRT. To reset these values, push the address key for the desired axes, and then, push the ORG key, while the data are being displayed on the CRT. With this keying, the coordinate values become (0, 0).

c. Push the OFS function key.

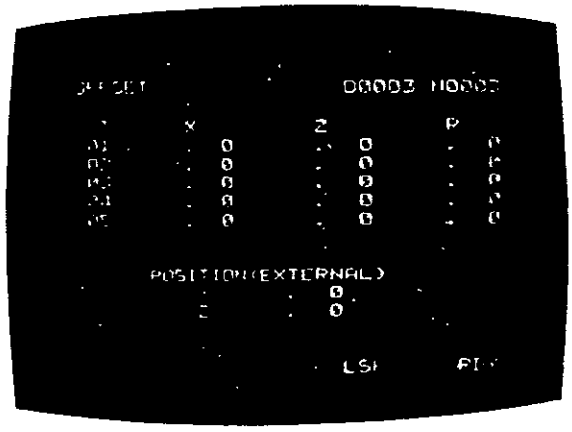


Fig. 6.5

The tool offset values will be displayed as shown above. However, the POSITION EXTERNAL values in the lower part of the CRT will have been changed to 0 for both axes, by the keying under the operation shown in b.

d. Select the HANDLE/STEP JOG mode.

e. Select a tool for which tool coordinate data are to be written in.

f. Start the spindle, and test-turn the workpiece circumference (surface A) with that tool in the MANUAL mode.

g. Push the CURRENT VALUE STORE button (PST input) on the machine control station first, and then, retract the tool and stop the spindle.
--- When the CURRENT VALUE STORE button is pushed, the displayed POSITION EXTERNAL values are temporarily stored in the register, and at the same time, the LED for the OFS function key starts to flicker, and the DIRECT INPUT mode is turned on.

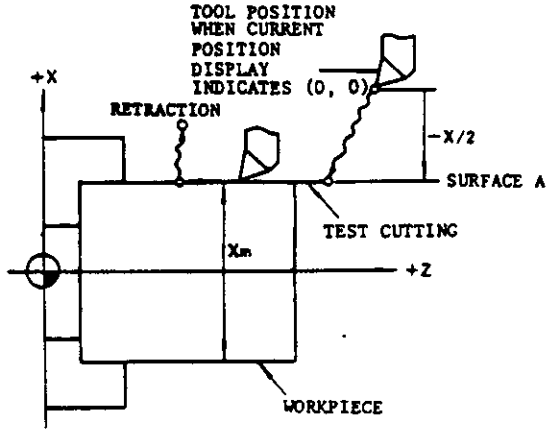


Fig. 6.6

In the example shown above, the -x value is temporarily stored in the register.

h. Measure the outer diamter of the turned workpiece and read the value "xm."

i. Select the desired tool coordinate memory No. (one among offset Nos. 51 through 80). For example, if the tool coordinate memory No. 51 is desired, key in 5 , 1 and depress the CURSOR key.

j. Then, proceed to key in x measured value (xm) and WR .
--- The NC make the calculation to obtain the tool coordinate data expressed by the following equation, and stores them into the specified tool coordinate memory X-axis. The writing of the required X-axis data has been completed.

$$\text{Tool coordinate data} = \text{Keyed in measured value} - \text{Current values temporarily stored in register}$$

--- Equation A

k. Repeat the same processes on the Z axis, by test turning the end face (surface B) of the workpiece with the tool.

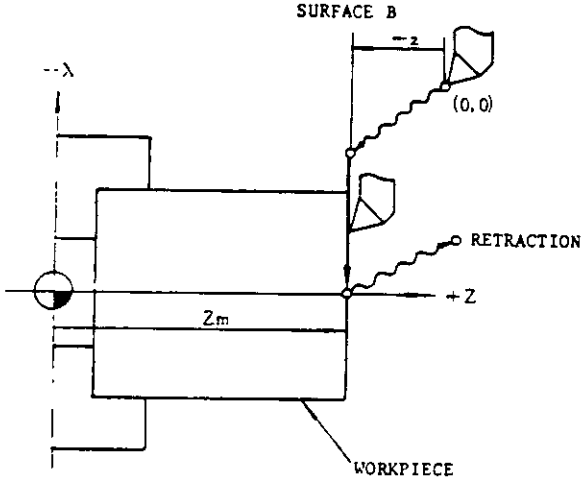


Fig. 6.7

l. Push the CURRENT VALUE STORE button again, and then, retract the tool and stop the spindle.
--- The current values at the time of the button pushing are stored again in the register temporarily.

m. Measure the distance "Zm" between the desired work coordinate origin to the test cut surface (surface B).

n. Key in Z , measured value (Zm) , and WR in this sequence.
--- The NC performs the same calculation as before with respect to Z-axis, and stores the results in the Z-axis of the tool coordinate memory.

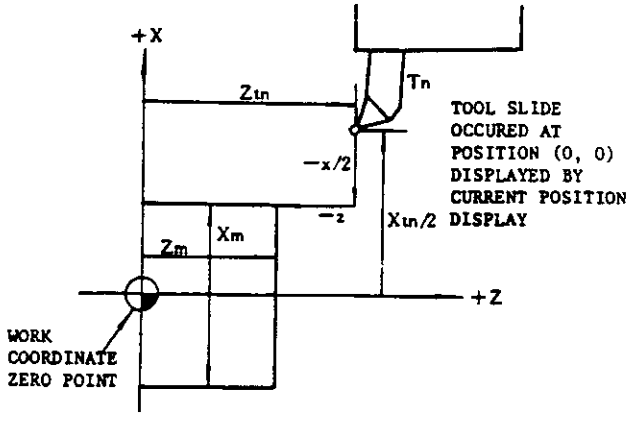


Fig. 6.8

With the above processes, all the data for one tool has been written into the tool coordinate memory.

o. Repeat the processes e. through o. for all the tools to write their data into the tool coordinate memory.

p. After writing all the tool data, push the RESET key to cancel the DIRECT INPUT mode.

With the above processes, Xtn and Ztn value for all the tools as shown below are stored in the tool coordinate memory.

The reason for this is that the following calculations have been performed.

$$Xtn = Xm - (-x) = Xm + x$$

$$Ztn = Zm - (-z) - Zm + z$$

NOTES:

1. The above example is based on test cutting. The method using a gauge or a workpiece of known dimensions is also convenient.
2. When tool positions for test cutting or contact are on the minus coordinate area, key in the measured values in negative.

3. When Parameter #6020D2 is "0," tool offset memory (offset Nos. 01 through 50) can be written in the same way as above. When parameter #6020D2 is "1," the following calculation equation is used only for tool offset memory (offset Nos. through 50).

$$\begin{bmatrix}\text{Stored}\\ \text{data}\end{bmatrix} = \begin{bmatrix}\text{Current value temporarily}\\ \text{stored in the register}\end{bmatrix} - \begin{bmatrix}\text{Keyed in}\\ \text{value}\end{bmatrix}$$

--- Equation B

As can be seen, in Equation B, sign is reversed from that in Equation A given before.

4. When parameters are not used, and the CURRENT VALUE STORE button is pushed, the data are always calculated by Equation A for writing into the TOOL COORDINATE MEMORY (51 - 81) and the WORK COORDINATE SYSTEM SHIFT MEMORY (00).

5. When the CURRENT VALUE STORE button is pushed, the OFS function is automatically turned on, its LED flickers, and the tool offset value is displayed.

(2) Automatic writing into tool offset memory

When Parameter #6020D2 is set to "1," tool offset values are automatically written into the tool offset memory by the following processes. The tool offset values written in this case are distance between the position of the reference tool and the tool under consideration.

a. Bring the reference tool to any desired reference point by manual control.

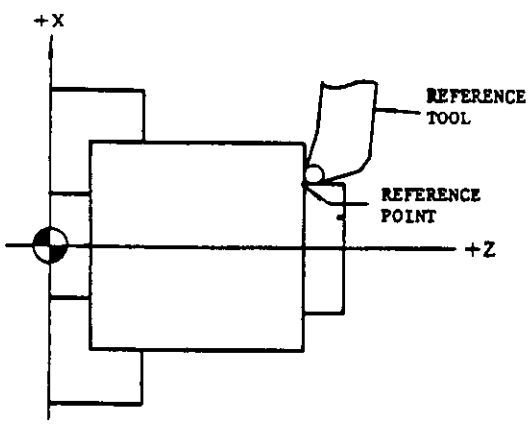


Fig. 6.9

b. Reset the current coordinate values (the displayed POSITION EXTERNAL values) to (0, 0).

c. Push the OFS function key.

--- The displayed POSITION EXTERNAL values in the lower area of the CRT have been changed to (0, 0).

d. Retract the reference tool, and bring the tool by manual control to where it can be replaced conveniently.

e. Select a tool with which offset value should be written in.

f. Bring the selected tool manually to the reference point, push the CURRENT VALUE STORE button, and retract the tool.

--- The current values at the time of button pushing are temporarily stored in the register.

g. Select a "tool offset No." for writing (one among 01 through 50).
For example, to write into tool offset No. "02," key in 0 , 2 , and CURSOR .

h. Then, go on keying as follows.

X , 0 , and WR ··· (for X axis calculation and storing)
Z , 0 , and WR ··· (for Z axis calculation and storing)

--- Now, the control performs the following calculation separately for X and Z axes, and stores the results to the specified tool offset memory.

$$\begin{bmatrix}\text{Stored}\\ \text{data}\end{bmatrix} = \begin{bmatrix}\text{Current values temporarily}\\ \text{stored in register}\end{bmatrix} - \begin{bmatrix}\text{Keyed in}\\ \text{value}\end{bmatrix}$$

However, since no value is keyed in the above process,the "current values temporarily stored in register" only are stored as tool offset values.

i. By repeating the processes 1. through g. with all the tools, their tool offset values can be automatically written. After completing the processes, push the RESET key. The values written by the above processes are all differential distances between the reference tool and the actual tools in consideration.

NOTES: For tools with which tool nose directions are different from that of the reference tool, prepare a 2nd reference point as shown below, and bring the tool to that point manually.

6.2.3 MEASURED WORKPIECE VALUE DIRECT INPUT (CONT'D)

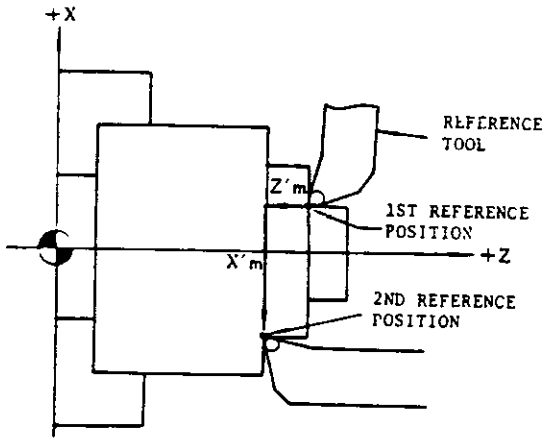


Fig. 6.10

Push the CURRENT VALUE STORE button at that position, and then, retract the tool.

Then, make the following keying, in place of the ones given in g. above.

[X] , [measured value (Xm')] , [WR]
[Z] , [measured value (Zm')] , [WR]

where Xm' and Zm' are distances with signs from the 1st reference point to the 2nd reference point. In the example shown above, these values have minus signs. With the above processes, the distances between the reference tool and a tool having different point direction are stored in the specified tool offset memory.

6.2.4 G50 POINT RETURN

With this function, the tool can be brought back to the start point of the program (where coordinate system was set up), from any position along the automatic operation tool locus. This is convenient when an automatic machining cycle is interrupted due to tool breakage, etc. and, after replacing the tool, etc., the automatic machining cycle is to be started from the beginning.

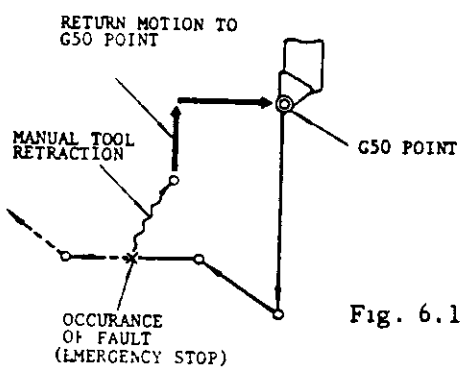


Fig. 6.11

The operation procedure for this return is as follows.

(1) Interrupt the automatic operation cycle by pushing the emergency stop button.

(2) Turn on the MANUAL mode.

(3) Retract the tool by the manual tool motion control.

(4) Eliminate the cause of the machining process interruption, and make the machine and workpiece ready for operation.

(5) Turn the G50 POINT RETURN switch on.

(6) Turn on the MANUAL JOG mode.

(7) Push one or two JOG keys that correspond to the required return motion direction among the four (+X, -X, +Z, -Z). The tool moves at a preset speed towards the G50 point, and stops at the G50) point.

(8) Turn off the G50 point RETURN switch.

(9) Turn on the AUTO mode, and push the RESET key on the MDI & CRT panel. (In the TAPE mode, reset the tape to the leading end.)

(10) Push the CYCLE START key to restart the automatic machining cycle.

NOTES:

1. While the G50 POINT RETURN switch is on, JOG motion is effective only towards the G50 point.
2. When the tool is at the G50 point, JOG motion control is not effective unless the G50 POINT RETURN switch is turned off.
3. The point where a coordinate system setup process has been executed latest is regarded as the G50 point. Therefore, in the following cases, the tool will not return to the start point of the machining cycle by this function.

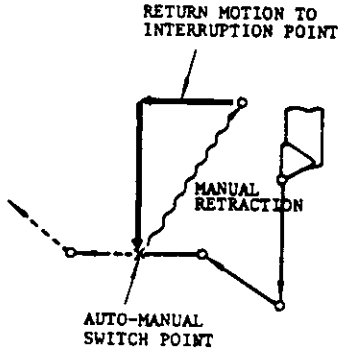
a. The respective tools are set for different coordinate systems.

b. When the ORG key is pushed after the interruption of the automatic cycles.

c. When a coordinate system has been set up automatically.

6.2.5 MANUAL INTERRUPTION POINT RETURN†

With this function, automatic machining cycles can be interrupted and the tools can be retracted for workpiece measurement, chip removal, etc., and then, the tools can be brought back to the interruption point.



The procedure for this function is as follows.

(1) Interrupt the automatic machining cycle by any of the following methods.

(i) Turning on the SINGLE BLOCK switch

(ii) Pushing the FEED HOLD key

(2) Turn on the MANUAL mode.

(3) Retract the tool by manual control.

(4) Measure the workpiece, remove chips, or perform any other required operations.

Note: When the mode is switched from AUTO to MANUAL, the spindle conditions, etc. may change. For these details, refer to the operation manual of the machine tool prepared by the machine builder.

(5) Turn on the MANUAL INTERRUPTION POINT RETURN switch.

(6) Turn on the MANUAL JOG mode.

(7) Push the relevant one or two of the JOG keys (+X, -X, +Z, -Z). The tool moves towards the interruption point at the preset speed, and stops upon arrival at the interruption point.

(8) Turn off the MANUAL INTERRUPTION POINT RETURN switch.

(9) Turn the AUTO mode, and push the CYCLE START key to restart the automatic machining cycle.

NOTES:

1. While the MANUAL INTERRUPTION POINT RETURN switch is on, JOG motion is effective only towards the interruption point.
2. When the tool is at the interruption point, the JOG keys are ineffective, unless the MANUAL INTERRUPTION POINT RETURN switch is turned off.
3. When the RESET key on the MDI & CRT panel is pushed or an external reset input is received after the switching from the AUTO to MANUAL mode, the MANUAL INTERRUPTION POINT RETURN function becomes ineffective thereafter.
4. When the tool has been manually retracted after a switching from the AUTO to MANUAL mode, then, the switch is returned to AUTO again, and thereafter, the switch is set again to MANUAL for manual tool motion, the point where the mode switch has been switched from AUTO to MANUAL last is regarded as the INTERRUPTION point.

6.2.6 PROGRAM RESTART†

With this function, when automatic machining cycles are interrupted due to tool breakage, chip entanglement, etc., by the EMERGENCY STOP button, the automatic machining cycles can be restarted, not from the beginning of the program, but from the beginning of the interrupted block.

(1) OPERATION PROCEDURE

a. Interrupt the automatic machining cycle by one of the following two functions.

(i) Emergency stop

(ii) Resetting

When the machine has been stopped by the emergency stop function, turn on the servo power supply and reset the ALARM code to make further machine motion possible.

b. Turn on the MANUAL mode and retract the tool.

c. Replace the broken tool, remove chips, etc. to prepare the machine for further operation.

When the tool is replaced, the offset values may have to be corrected.

d. Move the tool by manual control to return it to the start point of the part program. However, with a program in which G50T □□□□ is used, the tool need not be returned to the start point, and may be left at the position where chip removal, etc. has been made.

6.2.6 PROGRAM RESTART† (CONT'D)

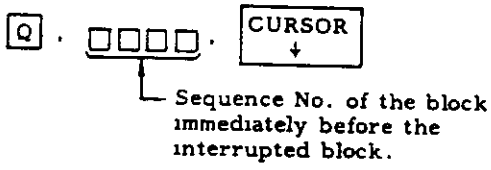
e. Turn on the AUTOMATIC mode.

f. Turn on the PROGRAM RESTART switch.

g. Push the PROG function key on the CRT & MDI panel.

h. Where the automatic operation is under the control of a tape, set the leading end of the tape to the tape reader.

i. Make the following keying on the CRT & MDI panel.



The NC starts the preparation from the starting end of the program to the block specified by Q. Upon the completion of the preparation, the CRT display changes to the "PROGRAM RETURN" page of the "POS" function to display the program restart data.

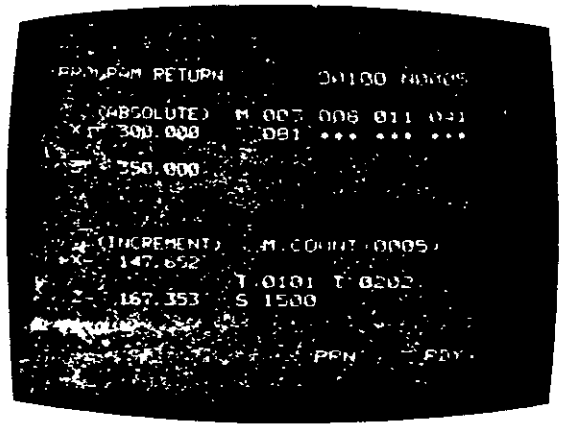


Fig. 6.12

Display of program restart data

(i) (ABSOLUTE): Display of current tool position

(ii) (INCREMENT): Display of the distance from the current tool position to the end point of the block specified by Q □□□□

(iii) Display of all the M codes and the number of M code commands programmed between the leading end of the restart program to the block specified by Q □□□□ However, if there are more than 28 M codes, those 28 M codes immediately next to the block specified by Q □□□□ in the program are displayed.

(iv) Display of the last programmed T command and the one before preceding the block specified by Q □□□□

(v) Display of the last programmed S command up to the block specified by Q □□□□

Note: M and T commands are displayed in the programmed sequence. Therefore, the last displayed one is the one programmed immediately before the block specified by Q □□□□

j. Turn off the PROGRAM RESTART switch.

k. Command required M, S, and T obtained them from the displayed program restart data as follows.

(i) Turn on the MDI mode.

(ii) Push the PROG function key on the CRT & MDI panel, and input the required M, S and T commands.

(iii) Push the CYCLE START key, and execute the M, S and T commands.

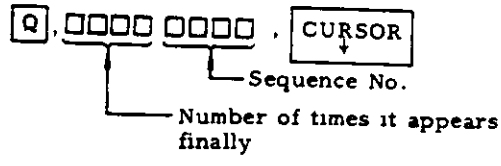
(iv) Push the POS function key on the CRT & MDI panel to revert the CRT to the program restart data display.
Check the conditions of the machine again.

l. Turn on the AUTOMATIC mode again.

m. Push the CYCLE START key. The tool moves to the PROGRAM RESTART position displayed in the (INCREMENT) column, by moving at JOG speed first along the X-axis, and then along the Z-axis, and then, restart the machining cycle from the block immediately following the block specified by Q □□□□.

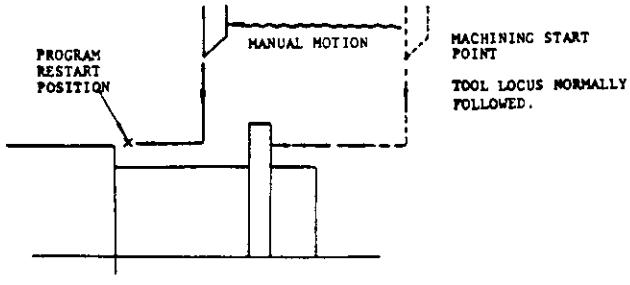
(2) Supplementary explanation

If the same sequence No. as for the block to be input by Q□□□□ is used several times in the program, input the command in the following form.



NOTES:

1. Before letting the tool move to the PROGRAM RESTART position by motion along the X and Z axes in succession, be sure to check the tool for freedom from interference with the workpiece.
   If there is a possibility of a tool interference with the workpiece, first, move the tool in the MANUAL mode to a position from where it can move without interference, before bringing the tool to the PROGRAM RESTART position in the AUTOMATIC MODE.



   For the manual tool motion, turn on the MANUAL ABSOLUTE switch. If this switch is not turned on, the PROGRAM RESTART position will be shifted by the distance covered by the manual motion. For the manual motion required to avoid tool collision on the workpiece as described above, the following procedure is recommended.

   (i) Turn on the SINGLE BLOCK switch.

   (ii) Push the CYCLE START key to move the tool along the X-axis.

   (iii) Check the position.

   (iv) Push the CYCLE START key to move the tool along the Y-axis.

   (v) Check the tool position for the PROGRAM RESTART position.

   (vi) Turn off the SINGLE BLOCK switch, and push the CYCLE START key to restart the machining cycle.

2. If a reset process is executed after the display of the PROGRAM RESTART data by the keying of Q □□□□□, CURSOR the data is cancelled. When this happens, make the PROGRAM RESTART data display keying again.

3. While the NC is making the preparation for PROGRAM RESTART after the keying of Q □□□□□, CURSOR , do not push the FEED HOLD key, turn the MODE switch, or make other manipulations. If these manipulations are made, make the PROGRAM RESTART data display keying again.

4. When the tool offset values are changed to compensate for the new tool, etc., the PROGRAM RESTART position is, needless to say, shifted accordingly.

5. If the PROGRAM RESTART switch is on, the CYCLE START key is ineffective.

6. In principle, do not move the machine with the MACHINE LOCK switch turned on, before and after pushing the PROGRAM RESTART switch.

7. If the block specified by Q □□□□□,CURSOR is not found, alarm code "120" is displayed.

8. Only those M and T commands which are output to the outside are displayed as PROGRAM RESTART data.
   Those M commands (M90 - M109) and T commands which are internally processed are not displayed.

9. Although not common with lathe operations, somethimes the power supply is turned off after interrupting the automatic cycle, and is turned on again before restarting the automatic cycle. In this case, be sure to return the tool to the reference point once, before starting the PROGRAM RESTART process.
   In this case, be sure to start the PROGRAM RESTART process even with those programs in which work coordinate system is used.

10. During the time after the display of the PROGRAM RESTART data till the start of the return motion to the PROGRAM RESTART position, the machine can not be moved by the MDI mode. When this is attempted, alarm code "124" will be displayed.

11. Blocks in complex canned cycle programs for finish shapes can not be specified by Q □□□□□, CURSOR .
    In this case, restart from the block before making tool radius compensation.

6.2.7 AUTO MODE HANDLE OFFSET

With this function, the handwheel for the manual pulse generator can be turned during an automatic operation cycle under the control of a tape, MDI or memory, to superimpose certain feed distances to the programmed feed distances. With this function, workpiece mounting errors, etc. can be compensated.

For this function, the required manual operations are as follows.

(1) Turn on the AUTO MODE HANDLE OFFSET switch.

(2) Select the axis along which motion is desired by the HANDLE AXIS SELECT switch. --- If the control is provided with the SIMULTANEOUS 2 AXES CONTROL MANUAL PULSE GENERATOR, the manual motion can be made along the two axes simultaneously.

(3) Select the movement distance per graduation of the handwheel with the MANUAL PULSE MULTIPLY switch. With this switch, the move distance per graduation can be selected among 1, 10 and 100 pulses.

(4) When the handwheel is turned, the tool motion along the axis selected in process (2) is superimposed on the programmed feed distance.

Turning CW: in plus direction
Turning CCW: in minus direction

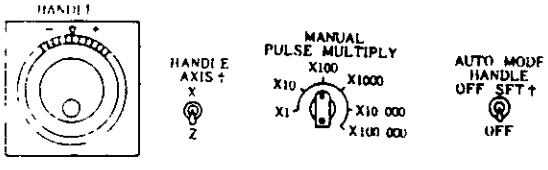


Fig. 6.13

NOTES

1. During the time the tool is moving in rapid traverse, the AUTO MODE HANDLE OFFSET motion is ineffective. It is effective only during interpolation motion.
2. In the alarm state, the AUTO MODE HANDLE OFFSET motion is ineffective.
3. While an interrupt input (STLK) is on, the AUTO MODE HANDLE OFFSET motion is ineffective.
4. The move distance by the AUTO MODE HANDLE OFFSET function is superimposed on the display of POSITION EXTERNAL and POSITION ABSOLUTE.
5. The AUTO MODE HANDLE OFFSET motion along the respective axes can be made ineffective by parameter settings.

Axis	Parameter		
	No.	Setting	
X-axis	#6022D0 (HOFSX)	1	Effective
		0	Ineffective
Z-axis	#6022D1 (HOFSZ)	1	Effective
		0	Ineffective

6. When parameter #6022D7 (HOFSMV) is set to 1, the motion by the AUTO MODE HANDLE OFFSET function is limited to the interpolation motion in automatic operation.

7. OPERATION PROCEDURE

7.1 INSPECTION BEFORE TURNING ON POWER

Make sure that the front and rear doors of the control are firmly closed. The control employs a totally-enclosed, dustproof enclosure to shut out surrounding air. If the door is open, lock it by turning two door locks. In addition, inspect the machine referring to the machine tool builder's manual.

7.2 TURNING ON POWER

(1) Check to see that the main power is supplied for the control.

(2) Depress the POWER ON pushbutton on the operator's panel, and the control power is supplied and then the cooling fans will start running. Make sure that air blows out from the exhaust ports of the upper side of the control.

(3) Depress the POWER ON pushbutton again to turn on the servo power supply. When the machine is ready to operate, READY lamp lights.

(4) If READY lamp does not light, detect and eliminate the cause according to the alarm code displayed. Refer to 4.3.9 DISPLAYING ALARM CODE.

7.3 MANUAL OPERATION[1]

When the MODE SELECT switch on the machine control station is set to RAPID, JOG, STEP or HANDLE position, the machine can be operated manually.

Operation in RAPID Mode

(1) Set MODE SELECT switch to RAPID.

(2) Select the speed using RAPID TRAVERSE RATE OVERRIDE switch.

Speed setting range: 100% - 50% - 25% - F0

(3) Push JOG button to select the axis and direction of movement. The machine moves at the specified speed while the JOG button

[1] Manual operation is defined as the operation in RAPID, JOG, STEP, or HANDLE.

Operations in JOG Mode

(1) Set MODE SELECT switch to JOG.

(2) Adjust the feedrate to the desired setting with JOG FEEDRATE switch (Up to 32 steps).

(3) Push JOG button to select the axis and direction of movement. The machine moves at the specified speed while the JOG button depressed.

Operation in STEP Mode

(1) Set MODE SELECT switch to STEP.

(2) Select the move amount per step using MANUAL PULSE MULTIPLY switch.

(Move amount setting range)

Metric: 0.001 - 0.01 - 0.1 - 1.0 - 10.0 - 100.0 mm/step

Inch: 0.0001 - 0.001 - 0.01 - 0.1 - 1.0 - 10.0 in/step

(3) Depress JOG button to select the axis and direction of movement. The machine moves by the move amount per step each time the button is depressed.

Operation in HANDLE Mode †

The control with HANDLE dial† can permit the operation described below.

(1) Set MODE SELECT switch to HANDLE.

(2) Select the axis with HANDLE AXIS switch.

(3) Select the move amount of the machine corresponding to one scale of HANDLE dial using MANUAL PULSE MULTIPLY switch.

Metric: 0.001 - 0.01 - 0.1 (mm per graduation)

Inch: 0.0001 - 0.001 - 0.01 (inch per graduation)

NOTE: "X1000" or "X10000" is regarded as "X100."

(4) Rotate HANDLE dial

Turning the dial clockwise:
The machine moves in the positive direction.

Turning the dial counterclockwise:
The machine moves in the negative direction.

7.4 PREPATION FOR STORED LEAD-SCREW ERROR COMPENSATION AND STORED STROKE LIMIT†

(1) Return to Reference Point

With an NC equipped with the stored lead-screw error compensation or the stored stroke limit functions, either of the following two reference point return motions must be performed after switching on the power supply and before starting automatic operation.

a. Manual return to reference point (See 6.2.1)

b. Execute G28 U0 W0 ; in the MDI mode.

This procedure is to teach the reference point to the control, since doing so is necessary because both leadscrew error compensation and stored stroke check are performed with reference to the reference point.

Checking Parameter #6006D_1, D_0

When the control is equipped with leadscrew error compensation function or the stored stroke limit function, set this parameter to "1 " With the parameter #6006D_1, D_0 set to "1," a return to the reference point is required before starting cycles, alarm codes (001 - 002 "reference point return incomplete") are displayed, if the CYCLE START key is pushed without making a reference point return immediatly after turning on the power supply. Be sure to perform the operation for return to reference point.

7.5 PREPARATIONS FOR AUTOMATIC OPERATION

To start to operate the machine in the automatic mode, the machine must be brought to the start point, after the application of the power supply. The panel operation required for this varies with programs as shown below. For details, refer to the operation manual of the machine tool builder.

(1) When G28 (AUTOMATIC REFERENCE POINT RETURN) is used

Where G28 is written in the beginning of the program, move the tool manually to a point a short distance away from the reference point.
If the start point is on the side of the reference point from the traverse speed reducing point, the NC enters an error state when the CYCLE START key is pushed.

EXAMPLE.

```
EOR ;
N001 G28 ;
N002 G50 X··· Z··· ;
```

(2) When MANUAL RETURN TO REFERENCE POINT function is used

Where G28 is not programmed, and the coordinate set up point is the reference point, bring the tool manually to the reference point before starting the automatic cycle operation. Refer to 6.2.1 Manual Return to Reference Point.

EXAMPLE:

```
EOR ;
N001 G50 X··· Z··· ;
  ·
  ·
  ·
  ·
```

(3) When automatic and manual return to reference point functions are not used

To set up the programmed work coordinate system with workpiece as a basis without using the reference point, proceed as follows:

a. Select the reference tool and set the test workpiece.

b. Position the Z-axis at the workpiece face (reference surface) by manual operation.

c. Reset the current position display of Z-axis. Then Z-axis is determined as Z-axis coordinate point.

d. Position the X-axis at the outer surface of the workpiece by manual operation. Then the control executes cutting outer surface if necessary.

e. Turn on MACHINE LOCK and set the dimensions of the work outer surface as the current position display of the X-axis. Then the center is determined as X-axis coordinate point.

f. Turn off MACHINE LOCK.

g. Move the tool to the setup point for each axis, checking against the current position display.

When the operations mentioned above are proceeded correctly, tool position offset amount for the tool will be zero.

7.6 OPERATION IN TAPE AND MEMORY MODE

(1) Make sure that NC ALARM lamp is not illuminated. If illuminated, detect and eliminate the cause by the indication of alarm code. Refer to 4.3.9 DISPLAYING ALARM CODE.

(2) Check and correct the stored offset values, and then put the machine in the correct start point.

(3) Set the switches on the control station of machine to the proper positions.

- MODE SELECT switch
- SINGLE BLOCK toggle switch
- RAPID TRAVERSE RATE OVERRIDE switch
- MANUAL ABSOLUTE toggle switch
- OPTIONAL BLOCK SKIP toggle switch
- OPTIONAL STOP (M01) toggle switch
- DRY RUN toggle switch
- FEEDRATE OVERRIDE & JOG FEEDRATE switch

(4) Set the punched tape onto the tape reader. In MEM mode, this operation is not required.

(5) Depress RESET key on the control station. Then LSK will be illuminated and the memory will be rewound.

(6) Depress CYCLE START button to give a Cycle Start to the system.

(7) When the Feed Hold is required for the machine during the system operation, depress FEED HOLD button.

(8) If the unexpected event occurs in the system, immediately depress EMERGENCY STOP pushbutton.

7.7 MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION

(1) Stop the automatic operation temporarily by depressing FEED HOLD pushbutton or by setting SINGLE BLOCK switch to ON position.

(2) Record the current positions of each axis on a paper using the current position display operation.

(3) Set MODE SELECT switch to manual operation mode (HANDLE, JOG or RAPID), and the machine can be manually operated.

(4) Return the machine manually to the recorded positions.

(5) Set MODE SELECT switch to the interrupted automatic-mode (TAPE, MDI or MEM).

(6) Depress CYCLE START pushbutton, and the machine will resume the automatic operation.

NOTES:

1. Where MODE SELECT switch is changed without depressing FEED HOLD pushbutton.
   a. When the automatic-mode (TAPE, MDI or MEM) is changed to the manual-mode (HANDLE, JOG or RAPID), the machine rapidly slows down and stops.
   b. When the automatic-modes are changed the machine is stopped at the block end.
2. Where the machine is restarted by depressing CYCLE START button, the tool path shifted due to manual operation will be changed by ON-OFF operation of MANUAL ABSOLUTE switch. Refer to 6.1.21 MANUAL ABSOLUTE SWITCH. In manual operation mode, when the CYCLE START button is depressed after writing F, M, S, T or $B^{\dagger}$ code by use of the same procedure as that of MDI operation, the command becomes effective and is executed as soon as written. This procedure is used to add new data to an active buffer. However, M00, M01, M02, M30 and M90 to M99 cannot be written.

7.8 AUTOMATIC OPERATION IN MDI MODE

(1) Set MODE SELECT switch to MDI operation.

(2) Write up to 10 blocks of data by MDI operation, and execute by pressing CYCLE START. Refer to 4.3.3.1.

(3) Depress CYCLE START button, and automatic operation can be executed in MDI mode.

7.9 MDI OPERATION INTERRUPTING AUTOMATIC OPERATION

To modify the block data after interrupting operation in TAPE or MEM mode, the following operation should be done after interrupting the operation.

(1) Turn on SINGLE BLOCK switch, and the operation is interrupted after the completion of the block being executed. At the same time, the next blocks of data may be read in advance.

(2) Display the data on CRT DISPLAY according to 4.3.2 DISPLAY OF COMMAND DATA, and check it.

(3) Set MODE SELECT switch to MDI operation.

(4) Write the data referring to 4.3.3 Writing in Blocks and Displaying Contents by MDI. Execute the data by depressing the CYCLE START button.

(5) Set back MODE SELECT switch to the interrupted automatic mode (TAPE or MEM).

(6) Return SINGLE BLOCK switch to OFF position.

(7) Depress CYCLE START button, and TAPE or MEM operation can be continued.

NOTES

1. Writing data by MDI cannot be executed in tool radius compensation modes (G41 - G44) because two-three blocks are read ahead.
2. Writing data by MDI cannot be performed in canned cycle modes (G70 - G76). The machine may not operate properly.
3 Excepting in tool radius compensation and canned cycle modes, MDI opeation is possible.

7.10 PREPARATION FOR TURNING OFF POWER

(1) Make sure that the machine is at standby and CYCLE START lamp is extinguished.

(2) Check to see that NC ALARM is not indicated on CRT. If alarm is displayed, detect the causes of displayed alarm code and eliminate them. Refer to 4.3.12 DISPLAYING ALARM CODE.

(3) Inspect the machine referring to the machine tool builder's manual.

7.11 TURNING OFF POWER

(1) Depress EMERGENCY STOP pushbutton to turn off the servo power supply.

(2) Depress POWER OFF pushbutton on the operator's panel to turn off the control power supply.

(3) Cut off the main power supply from the control.

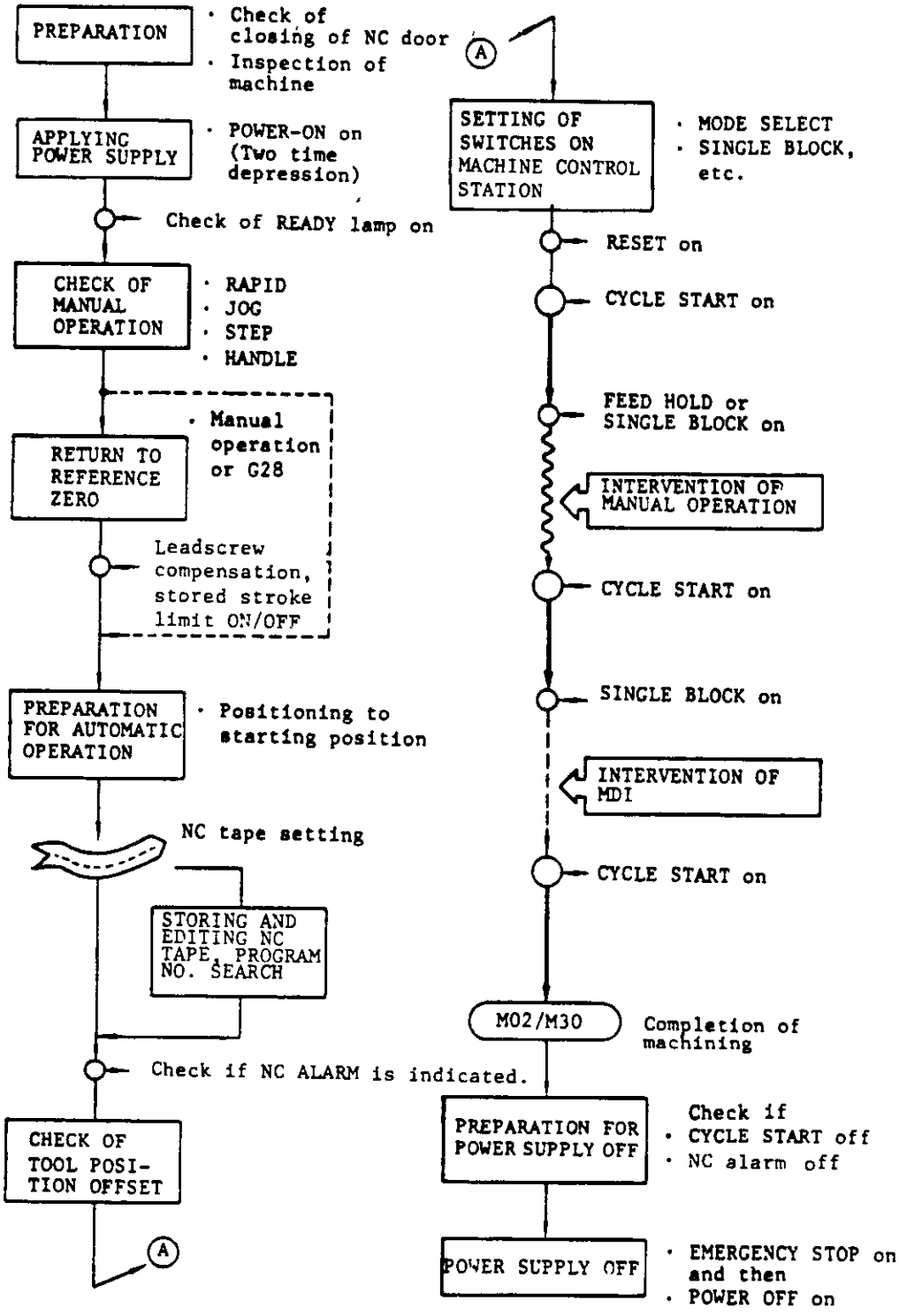


Fig. 7.1 Operating Procedure

8. MAINTENANCE

8.1 ROUTINE INSPECTION SCHEDULE

The following table shows the minimum requirements to be observed for maintenance according to time in order to keep the equipment in optimum condition for extended period.

Table 8 Inspection Schedule

Items		Frequency	With the system-off	With the system-on	Remarks
Tape reader	Cleaning of reading head	Daily	○		Including light source part.
	Cleaning of tape tumble box	Weekly	○		
	Lubricating of tension arm shaft end	As required	○		
Control panel	Tight closing the doors	Daily	○		
	Checking for loose fit and gaps of side plates and worn door gaskets	Monthly	○		
Servomotor and DC motor for spindle	Vibration and noise	Daily		○	Feel by hand, and do the audible inspection.
	Motor contamination and breakage	Daily or as required	○	○	Inspect visually.
	Clearance of ventilation openings		○	○	Inspect mainly spindle DC motor.
	Burned spots, cracks, wear, and pressure of brushes	Every three months	○		Check the length of brushes.
	Roughened commutator surface		○		Check dark bar, threading and grooving of commutator.
	Dirt in interior of motor		○		Clean with compressed air.
Battery		Daily	○	○	See if alarm for BATTERY is displayed on CRT screen.

8.1.1 CAUTIONS ON INPUT POWER SUPPLY

Except for those checks which can be made with the NC in the energized state, such as checks for external cleanliness for vibration and for noise, be sure to turn off the power supply to the NC before starting to undertake routine maintainance service.

For this, turning off the power supply by pushing the POWER OFF button on the NC operator's panel is not sufficient, because after this button is pushed, still several areas in the housing are energized, and are potentially dangerous.

8.1.2 TAPE READER UNIT

(1) Cleaning head of tape reader

a. Remove tape rubbish and dust on the glass with a blower brush. If the glass is stained with oil or oily dust, wipe it using a gauze or soft cloth with absolute alcohol. Also clean the tape guide and the tape retainer.

b. Remove the dust, if any, on LED (light source) with a blower brush.

Be sure to turn off the MCBs on (or in) the power switchboard near the machine to turn off the supply of power to the NC.

When the power supply to the NC is turned off, all the fans in the NC housing stop. This should be taken as the indication of the total stoppage of power supply to the NC.

(2) Cleaning of tape tumble box

a. Clean the polyester leading tape with a clean, soft cloth.

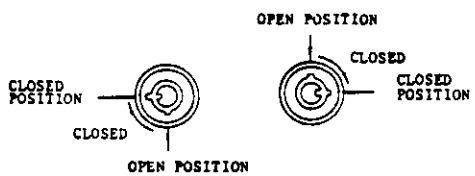
b. Remove the tape outlet cover (See Fig. 5.1.3) by loosening two mounting screws and clean the bottom of the tape tumble box with cloth or brush.

8.1.3 CONTROL PANEL

(1) The control panel is a dustproof, sheet-steel enclosure with gasketed doors.

a. Front and rear doors of the control should be shut tightly, even if the control is not operating.

b. When inspecting the control with the door open, upon completion of inspection, lock door by turning two door locks with the key attached to the control panel. Turning reaction of door locks is as follows.



Note: If the optional door interlocking switch is provided, opening the door shuts off the main power supply and stops all operations.

c. Check gaskets on the rims of front and rear doors.

d. See if the inside of enclosure is dusty. Clean it, if necessary.

e. Check for any opening in the door base with the doors shut tightly.

8.1.4 SERVOMOTOR AND DC SPINDLE MOTOR

(1) Vibration and noise.

Vibration can be checked by resting the hand on the motors, and for noise, using a listening stick is recommended. If any abnormality is found, contact maintenance personnel immediately.

(2) Motor contamination and impairment.

Check the motor exterior visually. If dirt or damage should be observed, inspect the motor by removing the machine cover. Refer to the machine tool builder's manual.

(3) Clearance of ventilation window blockage

Check the ventilation window of DC spindle motor. If it is clogged with dust or dirt, inspect DC spindle motor removing the machine cover. Refer to the machine tool builder's manual.

Inspection of commutators and brushes is essential for maintaining the excellent performance of the control. Inspection work to be executed is described in the following three items.

Quarterly Inspection of Commutators and Brushes

The carbon dust from brushes, accumulated around the commutator, inside the motor, may cause motor troubles such as the layer short of armature and the flashover of commutator. In the worst case, it may lead to fatal damage. To avoid this, be sure to have an inspection on the commutators and brushes at least every three months.

Be sure to turn off the power supply to the NC before starting to check the brushes and motor interior.

For this, turning the circuit breaker on the power supply unit (DCP UNIT) for the servo control unit (CPCR-MR-K) off is not sufficient. To prevent electric shocks and shorting, be sure to shut off the supply of power to the NC.

(4) Carbon brushes

a. Under normal operating conditions, brush wears by 2 to 4 mm per 1000 operating hours. If wear is excessive, check to see if oil has contaminated commutator surface, or if abnormal overcurrent flow through motor circuit.

8.1.4 SERVOMOTOR AND DC SPINDLE MOTOR (CONT'D)

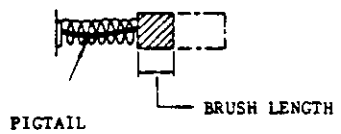
b. When brush length becomes shorter than those shown below, replace the brush with a new one.

c. If either of brush, or pigtail is broken, brush assembly must be replaced as a whole unit.

Minertia motor junior series: 6 mm or below

Spindle DC motor· 17 mm or below

Minertia motor J series: 7 mm or below



NOTE· When replacing the brush assembly, consult the company.

(5) Commutator surface

a. Visually check surface roughness of the commutator through inspection window.

After 100 to 200 operating hours, the commutator should take on a polished light brown or chocolate color. The motor has developed an ideal commutator film and needs no attention other than to be kept clean.

b. See if a blackened bar, threading (or grooving) is on the commutator. If any of the above is observed, investigate the cause of trouble.

Tnreading or grooving on the commutator surface may be due to too small motor load. Blackened bar is a result of carbon dust in commutator slots, or accidentally produced sparkings. If the carbon dust is a cause of blackened bar, wipe the commutator with a clean dry cloth to smooth the surface. If sparking occurs, contact the maintenance representative.

(6) Motor inside (dirty)

a. Visually check the motor interior through inspection window.

The dried carbon dust will not affect motor running, but it is recommended that the inner parts such as commutator, brush-holders and brushes be cleaned with a dry compressed air (air pressure: 2-4 kg/cm^2).

b. Where oily carbon dust exists inside the motor due to poor oil seal or defective enclosure, contact Yaskawa.

(7) Servomotor with oil seal

As the life expectancy of oil seal and brush is 5000 hours (about five years), the inspection and maintenance by the company should be done every 5 years. If possible, yearly inspection taking less than 8 hours is recommended.

8.1.5 BATTERY

Make sure that "BAT" or "A/B" on the right-low position of CRT screen is not displayed. If it is displayed, inform maintenance personnel. The battery must be replaced with a new one within a month.

The control with a bubble memory board (optiona does not require a battery.

8.2 BATTERY REPLACEMENT

The battery is used as power source for memory in order to prevent programming data stored in memory, such as parameter, tool offset and part program from erasing.

When the battery is discharged after a long period of use, "BAT" or "A/B" is blinked on CRT screen to give warning for replacement. On such occasions, the battery must be replaced within 30 days. When replacing, never remove the old battery with power off, otherwise the data stored in memory may be cleared.

Replacing Procedure

(1) Depress POWER OFF pushbutton to shut off the power supply to the operator's station.

(2) Open the front door of the control. The battery of the memory (printed circuit) board can be seen on the CPU module which is mounted on rear of the front door.

(3) Where the control is equipped with a door interlock switch, pull it out by hand. The power can be turned on, with the door open

(4) Depress POWER ON pushbutton.

(5) Check to see if LLED on memory board is illuminated. Fig. 8.2.1 shows the arrangement of LED and the battery. If illuminated replace the battery "660S" with new one.

660S ··· Type: JZNC-GBA02

(6) With the control power turned on, connect the receptacles of the new battery in to the plugs (1CN or 2CN) on memory circuit board, and LED will be turned off. See Fig. 8.2.1. If LED is still illuminated, it is due to the improper insertion of battery connectors, or defective battery.

IMPORTANT: Two plug stations, 1CN and 2CN (or 2CN and 3CN) are connected together with common leads. When an old battery is replaced with a new one, connect the new battery first to the plug station not occupied, then remove the receptacles of the old battery.

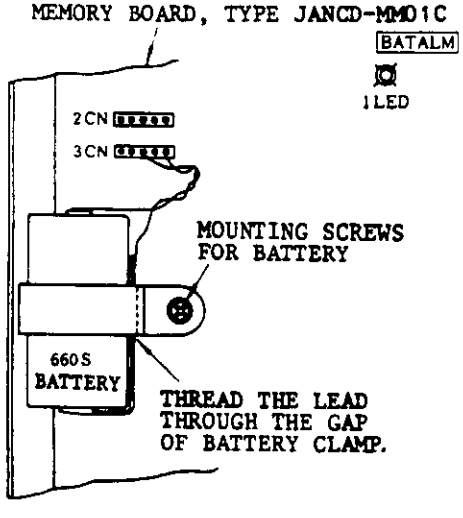
(7) Depress POWER OFF pushbutton to shut off the power supply to the operator's station.

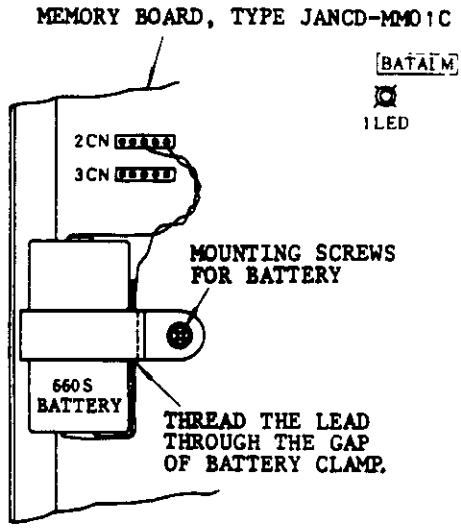
(8) Remove the mounting screw of old battery, and then replace the battery with new one. In this case, pass the battery lead through gaps between the battery and the battery clamp, and use care not to contact the lead with memory circuit board.
Where the control is provided with a door interlock switch, push it back in place since power cannot be turned on with the door open.

(9) Tightly close the front door.

(10) Depress POWER ON pushbutton.

(11) Make sure that "BAT" or "A/B" on CRT screen goes off.





NOTES:

1. While battery is being replaced, exercise utmost care to prevent contaminants from entering the control, and accomplish the work as quickly as possible.
2. Use special care to prevent water, oil, or dust, to adhere to the devices (printed circuit board, connectors, cables, etc.) inside the control.
3. Never leave any screws or washers in the control.
4. According to the type of the NC cabinet, mounting of CPU rack is turned upside down. The figures in Fig. 8.2.1 and 8.2.2 will be turned upside down also in that case.

8.3 FUSE AND CIRCUIT BREAKER

The NC is provided with the following fuses and breakers.

Table 8.1

Name of unit	Fuse or breaker	Rating
Integrated power supply unit for control (CPS-20N)	Glass tube fuse x 2	10 A, 10 A
Turning on unit (TU11)	Glass tube fuse x 2	1 A, 1 A
Servo-control power supply unit (DCP UNIT)	Glass tube fuse x 1 Circuit breaker x 1 (for 2-phase)	1 A or 2 A 10 A - 60 A

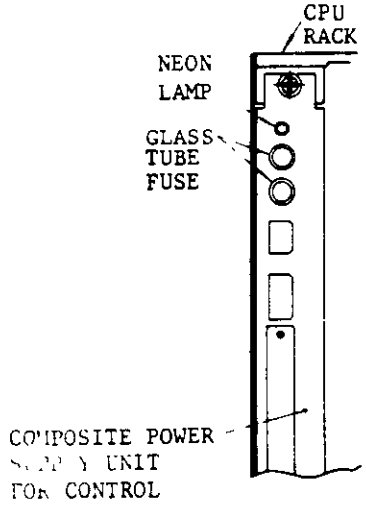
8.3 FUSE AND CIRCUIT BREAKER (CONT'D)

When any of these fuses or the circuit breaker is blown or tripped, report to the maintenance personnel immediately for the elimination of the cause.

Below, these fuses and breaker are outlined.

8.3.1 FUSE FOR INTEGRATED POWER SUPPLY UNIT FOR CONTROL

This unit is installed in the CPU rack. While correct input power is supplied to the NC, and both fuses are not blown, the orange neon lamp above the fuses are on. When the neon lamp is off, while the power supply is in order, the fuses are suspected to have been blown. These fuses will be blown when the integrated power supply unit itself develops faults. They will not be blown by causes on the output side of the unit such as overloading.



When the neon lamp is off, take the following measures.

(1) Turn off the power supply to the NC.

The input power supply is directly connected to this integrated power supply. Never touch it before shutting off the power supply

(2) Find the cause and eliminate it.

(3) Replace the blown fuse with a fuse of the same rating among the spare fuses.

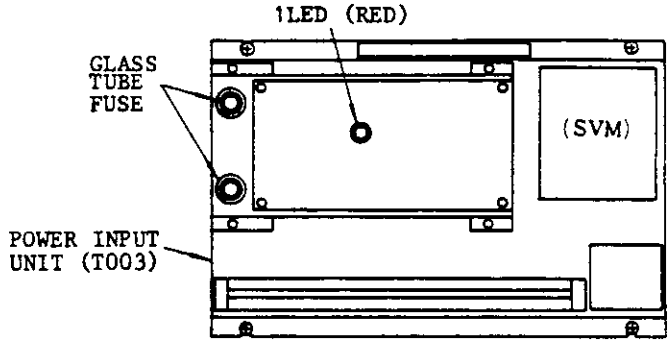
(4) Turn on the power supply to the NC, and make sure that the neon lamp is on. Then, the POWER ON button, to make the integrated power supply unit ready for operation.

(5) If the fuse is blown again, notify our service department.

8.3.2 FUSES OF POWER INPUT UNIT

When correct power is supplied to the NC, and both the fuses of the turning on unit (TUC3) are not blown, 1LED (red) on the printed circuit board is on.

When 1LED is off, take the following measure.



(1) Turn off the supply of power to the NC. Since this turning on unit is directly connected to the input power supply lines, never touch it before stopping the supply of power to it.

(2) Find the cause and eliminate it.

(3) Replace the blown fuse with a fuse of the same rating among the spare fuses.

(4) Turn on the power supply to the NC, and make sure that 1LED is on. Then, push the POWER ON button to make the NC ready for operation.

(5) If the fuse is blown again, notify our service department.

8.3.3 CIRCUIT BREAKER FOR SERVO CONTROL

When the DCP unit is overloaded through the shorting of the output circuit, faults of the DCP unit itself, etc., this circuit breaker trips to disconnect the main circuit.

In this case, an alarm "350:OL" is displayed on the CRT. (350:OL may be displayed also by other causes.) When this circuit breaker is tripped to the OFF state, take the following measure.

(1) Push the POWER OFF button to turn off the power supply, and then, stop the supply of power to NC.

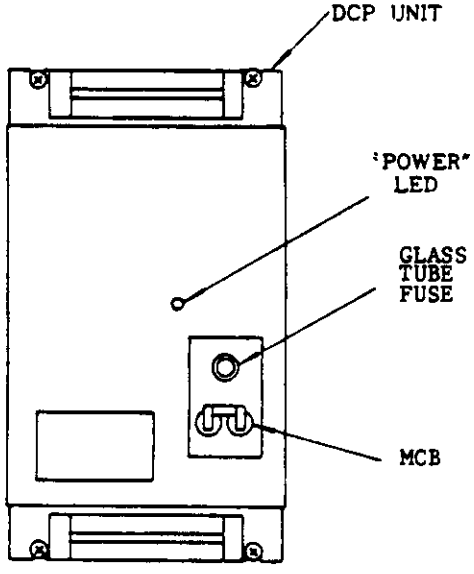
(2) Find the cause of the tripping of the circuit breaker, and eliminate it.

(3) Push up the breaker lever to the ON position.

(4) Supply power to NC, and then, push the POWER ON button twice.
The servo circuit is energized, and is ready for operation.

(5) If the circuit breaker is tripped again, notify our service department.

NOTE: When the circuit breaker is manually turned off under power, not turning off automatically by overloading, the alarm "310:SERVO OFF" is displayed on the CRT. The DCP unit can be brought to the operating conditions by the same measures as above, also in this case.



8.4 COUNTERACTING ALARM STATUS FOR SERVO CONTROL UNIT

The servo control unit has the function of detecting the following alarm states.

· Fuse (main circuit) blowing	(alarm No.)	331(X), 332(Z)
· Overloading	(alarm No.)	351(X), 352(Z)
· TG error	(alarm No.)	391(X), 392(Z)

When the above alarm states occur, the maintenance personnel must be immediately notified.

8.4.1 FUSE BLOWING (Alarm No. 331, 332)

When the main circuit of the servo control unit is shorted or when the servo control unit itself becomes faulty, the fuse in the unit will be blown, and the following alarm Nos. will be displayed on the CRT.

331: FUSE(X) ··· for X axis

332: FUSE(Z) ··· for Z-axis

When a fuse is blown, and the alarm No. 331 or 332 is displayed, do not attempt to take measures, but the user should immediately notify our service department.

8.4.2 OVERLOAD (ALARM Nos. 351, 352)

The servo control unit is provided with electronic thermal relays respectively and independently for the X and Z axes, and they trip under the following conditions.

· Programs involving excessively heavy cuts are executed.

· Programs involving excessively frequent speed changes are executed.

· Frictions in the machine system become abnormally large.

When the electronic thermal relay trips, the servo power supply is turned off, and the following alarm Nos. are displayed on the CRT.

351: OL(X) ··· X-axis overload

352: OL(Z) ··· Z-axis overload

When this is the case, take the following measures.

(1) Push the POWER OFF button to turn off the power supply, and then, stop the supply of power to NC.

(2) Find the cause of the overloading. For example, the cause may be eliminated through modifications of the part program, or by the elimination of abnormally large load on the machine.

(3) Supply power to the NC, and push the POWER ON button to turn on the power supply and make the system ready for operation. However, since the servo motor requires approximately 30 minutes to cool down after being overloaded to the extent of tripping the electronic thermal relay, wait at least 30 minutes before starting to operate NC.

(4) If the electronic thermal relay trips, notify our service department.

8.4.3 TG ERROR (ALARM NOS. 391, 392)

The servo control unit can detect the following alarm states.

- Wire breaking in the tachometer generator (TG) or overspeeding
- Main circuit overcurrent
- Main circuit overvoltage

When any of these faults occurs, the following alarm Nos. will be displayed on the CRT.

391. TG ERROR (X)

392. TG ERROR (Z)

When this is the case, take the following measures.

(1) Push the POWER OFF button to turn off the power supply, and then,stop the supply of power to NC.

(2) Find the cause of the alarms, and eliminate it. In this case, if the fault conditions are reported to our service department, we will be able to give advice on troubleshooting.

(3) Resupply power to NC, and then, push the POWER ON button to make the unit ready for operation.

(4) If TG ERROR is displayed again, notify our service department.

8.5 MOLDED-CASE CIRCUIT BREAKER (MCB)

Those special housing type controls, with which all the power sequence control circuits are converted to the NC area, are sometimes provided with MCBs which can be turned on and off externally.

Generally, when these MCBs are turned off, the power supply to the NC is stopped. For details, refer to the manual of the machine tool builders.

8.6 TROUBLE CAUSES AND REMEDIES

8.6.1 ON-LINE DIAGNOSTICS

On-line diagnostics are implemented to locate a trouble quickly and protect the machine against malfunctions. Shown below are the displaying functions executed by the control being on-line and machining.

- Display of three-digit alarm code including a code showing an axis in error.
- Display of four-digit status code including a function code showing M, S, T, V, DWL.
- Input/output signal display.

These displays can be made at any time, while the machine is in automatic operation, or at stand by.

8.6.2 ALARM CODES AND REMEDIES

Where "ALM" or "A/B" on CRT screen is blinking and the machine stops, depress the ALM key. Then alarm code and message will be displayed on CRT screen. Alarm codes "800," "810," "820," "830" and "840" are displayed as soon as the corresponding error occurs.

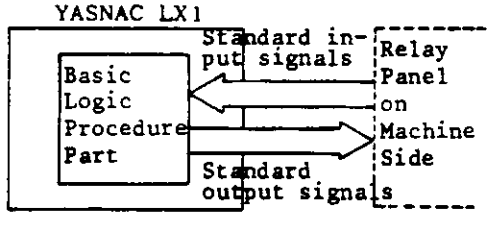
For the remedies for trouble causes represented by alarm codes, see APPENDIX 5 LIST OF ALARM CODES on the last part of this manual.

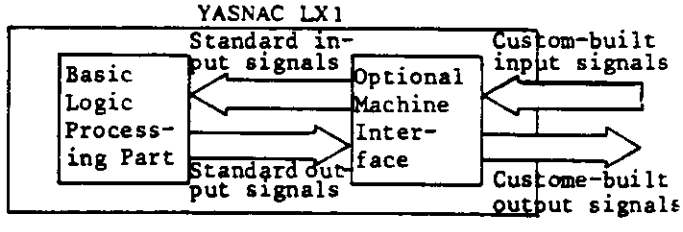
8.6.3 INPUT/OUTPUT SIGNALS

To clear up the causes indicated by alarm codes, check the input/output signals on the CRT screen

Input/output signals are divided into standard and custom-built ones, and displayed by specifying the corresponding diagnostic number with keys on the operator's station.

Standard signals are included in every type of YASNAC LX1. Custom-built signals are provided for optional machine interface equipped with some type of YASNAC controls.





To display input/output signals, proceed as follows.

1) Depress the DGN key.

A page containing the diagnostic number specified previously occurs on the CRT screen. The input/output signals are shown in "1," "0" and hexadecimal digit.

"1": contact close

"0": contact open

2) Key-in the diagnostic number to be displayed.

3) Depress the cursor ↓ or ↑ key to page the keyed-in diagnostic number on the screen.

a. By depressing the cursor [CURSOR ↓] key, a cursor on the screen moves to the next diagnostic number (line). When down to the last lower line, the next page is displayed on the screen.

b. By depressing the cursor [↑ CURSOR] key, the cursor moves to the previous line. When up to the most upper line, the previous page is displayed.

c. By depressing the page [PAGE ↓] key, the next page appars on the screen.

d. By depressing the page [↑ PAGE] key, the previous page appears.

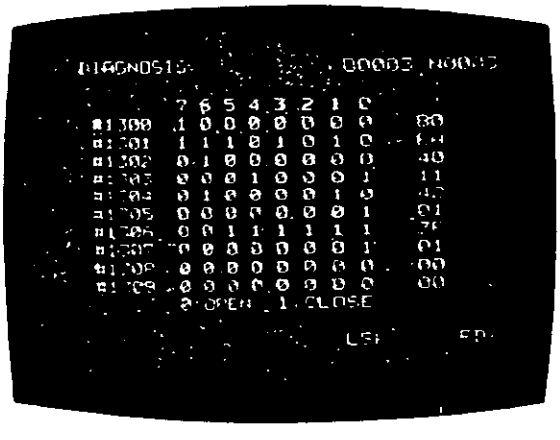


Fig. 8.1 Example of Input/Output Signal Display

8.6.4 BEFORE MAINTENANCE CALL

If the cause of trouble cannot be found by using alarm codes or I/O signals (described in 8.6.1 to 8.6.3), or correct action for the trouble cannot be taken, record the following items, and notify the company as soon as possible.

- Alarm codes and the accompanying data with them.
- The types and characteristics of the troubles.
- The operational procedures just before the trouble occured and number of applied tape.
- Whether the trouble recurs each time, the operation is repeated after depressing the RESET key.
- Data and time when the trouble occured.
- Name of the discoverer of the trouble and the operator.

If trouble occurs, keep the control in the same condition until it can be checked by your Yaskawa representative. If the situation permits, avoid turning off control power, or depressing POWER OFF button.

APPENDIX 1 LIST OF SETTING NUMBERS

#6000

D_7	D_6	D_5	D_4	D_3	D_2	D_1	D_0

INHEDTT D_7

1 Turns on Edit Lock function.
0. Turns off Edit Lock function.

AFLT D_6

1 Turns on Auxiliary Function Lock.
0 Turns off Auxiliary Function Lock.

ABST D_5

1 Turns on Manual Absolute function.
0 Turns off Manual Absolute function.

DRNT D_4

1 Turns on Dry Run function
0 Turns off Dry Run function.

BDTT D_3

1 Turns on Block Delete function.
0 Turns off Block Delete function.

DLKT D_2

1. Turns on Display Lock function.
0. Turns off Display Lock function.

MLKT D_1

1 Turns on Machine Lock function
0. Turns off Machine Lock function.

SBKT D_0

1 Turns on Single Block function.
0 Turns off Single Block function.

Notes.

1 These settings are for setting internal toggle switches.

2 When each switch is provided with machine control station, the logical sum of these settings and toggle switch setting determines function on/off state.

Internal toggle switch	OFF	ON	OFF	ON
Toggle switch on machine	OFF	OFF	ON	ON
Resultant ON/OFF state	OFF	ON	ON	ON

#6001

D_7					D_2	D_1	D_0

BUZON D_7

1 Turns on touch buzzer (key switch on operator's panel).
0. Turns off touch buzzer.

SLT 3: D_2

1: Effective on the third Stored Stroke Limit.
0. Ineffective on the third Stored Stroke Limit.

The value of limit automatically changes by G38 or G39 command in part program.

SLT 2: D_1

1 Effective on the second Stored Stroke Limit.
0 Ineffective on the second Stored Stroke Limit.

The value of limit automatically changes by G36 or G37 command in part program.

INCHMM D_0

1: Selects inch input increment.
0: Selects metric input increment.

#6002

D_7	D_6						

ISOEIA D_7

1. Punches out tape code with ISO code.
0 Punches out tape code with EIA code.

TVCHK D_6

1. Executes TV check.
0. Does not execute TV check.

#6003

		D_5	D_4			D_1	D_0

D_5, D_4
ODVCE 1. Selects the output device of data
ODVCE 0. I/O interface.
D_1, D_0
IDVCE 1. Selects the input device of data
IDVCE 0. I/O interface

Setting code		I/O device No.	Input device	Output device	Parameter No requiring baud rete setting
0	0	0	Tape reader	FACIT PUNCHER	
0	1	1	RS232C	RS232C ASR33/43	#6026 #6028
1	0	2	RS422	RS422	#6027 #6029

#6004	D_7	D_6	D_5	D_4	D_3	D_2	D_1	D_0

COV16I(D_7), COV8I(D_6), COV4I(D_5), COV2I(D_4), COV1I(D_3):

Sets the override of cut depth for Stock Removal in Turning (G71) and Stock Removal in Facing (G72) cycles.

COV16I	COV8I	COV4I	COV2I	COV1I	Cut depth override
0	0	0	0	0	0%
0	0	0	0	1	10%
0	0	0	1	0	20%
0	0	0	1	1	30%
0	0	1	0	0	40%
0	0	1	0	1	50%
0	0	1	1	0	60%
0	0	1	1	1	70%
0	1	0	0	0	80%
0	1	0	0	1	90%
0	1	0	1	0	100%
0	1	0	1	1	110%
0	1	1	0	0	120%
0	1	1	0	1	130%
0	1	1	1	0	140%
0	1	1	1	1	150%
1	0	0	0	0	160%
1	0	0	0	1	170%
1	0	0	1	0	180%
1	0	0	1	1	190%
1	0	1	0	0	200%

Note: These settings are effective when parameter #6023D_2 (COVP) is "0."

UMO8000 D_2

1: Inhibits editing and punchout operations of the part program of program No. 8000 to 8999.
0: Permits editing and punchout operations.

UMSBK D_1

1: Makes Single Block Stop effective for the programs in user macro when single block input is on.
0: Does not permit Single Block Stop for the user macro blocks commanding operation and control.

SKIPIN D_0

1: Executes the next block when the skip signal is not given before completion of movement of block including Skip Function (G31).
0: Alarm 087 is displayed.

#6161	TG1LF
#6162	TG2LF
#6163	TG3LF
#6164	TG4LF
#6165	TG5LF
#6166	TG6LF
#6167	TG7LF
#6168	TG8LF
#6169	TG9LF

TG1LF to TG9LF:

Individual life expectancy for tools in groups 1 to 9 is set by part program.

Setting range: 0 - 9999 (Tool life control)
Setting: "1" = 1

#6170	TG10LF
#6171	TG11LF
#6172	TG12LF
#6173	TG13LF
#6174	TG14LF
#6175	TG15LF
#6176	TG16LF
#6177	TG17LF
#6178	TG18LF
#6179	TG19LF

TG10LF to TG19LF:

Individual life expectancy for tools in groups 10 to 19 is set by part program.

Setting range: 0 - 9999 (Tool life control)
Setting: "1" = 1 minute

#6181	TG1CNT
#6182	TG2CNT
⋮	⋮
#6198	TG18CNT
#6199	TG19CNT

TG1CNT to TG19CNT:

No. of times used and operating times are indicated individually for tools in groups 1 to 19.

Note. Writing is not permitted in this setting.

#6202	G71OFL

G71OFL

Sets retraction value after completion of each cutting cycle in Stock Removal in Turning (G71).

Setting range 0 - 65536
Setting Least input increment

#6203	G72OFL

G72OFL

Sets retraction value after completion of each cutting cycle in Stock Removal in Facing (G72).

Setting range 0 - 65536
Setting Least input increment

#6204	G74OFL

G74OFL

Sets retraction value (δ) in Peck Drilling in Z-axis (G74)

Setting range. 0 - 65536
Setting Least input increment

#6205	G75OFL

G75OFL.

Sets retraction value (δ) in Grooving in X-axis (G75).

Setting range 0 - 65536
Setting Least input increment

#6206	G76OFL

G76OFL.

Sets cut depth (in X-axis) "a" in Automatic Threadcutting (G76).

Setting range. 0 - 65536
Setting. Least input increment

#6207	TINON

When the tape without program no. is stored, program no. is set for the tape.

#6500	XSL2P
#6501	ZSL2P

XSL2P, ZSL2P:

Sets the boundary area in positive direction of Stored Stroke Limit second prohibit area on X-axis and Z-axis, respectively.

Setting range: 0 to ±99999999
Setting. Least output increment

#6502	XSL2M
#6503	ZSL2M

XSL2M, ZSL2M:
Sets the boundary area in minus direction of Stored Stroke Limit second prohibit area on X-axis and Z-axis, respectively.

Setting range: 0 to ±99999999
Setting: Least output increment

#6504	XSL3P
#6505	ZSL3P

XSL3P, ZSL3P.

Sets the boundary area in positive direction of Stored Stroke Limit third prohibit area on X-axis and Z-axis, respectively.

Setting range· 0 to ±99999999
Setting: Least output increment

#6506	XSL3M
#6507	ZSL3M

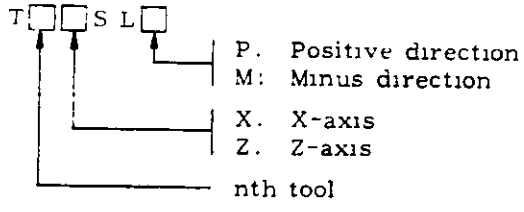
XSL3M, ZSL3M:

Sets the boundary area in minus direction of Stroed Stroke Limit third prohibit area on X-axis and Z-axis, respectively.

Setting range: 0 to ±99999999
Setting: Least output increment

#6508	T1XSLP
#6509	T1ZSLP
#6510	T1XSLM
#6511	T1ZSLM
#6512	T2XSLP
#6513	T2ZSLP
#6514	T2XSLM
#6515	T2ZSLM
#6516	T3XSLP
#6517	T3ZSLP
#6518	T3XSLM
#6519	T3ZSLM
#6520	T4XSLP
#6521	T4ZSLP
#6522	T4XSLM
#6523	T4SLM
#6524	T5XSLP
#6525	T5ZSLP
#6526	T5XSLM
#6527	T5ZSLM
#6528	T6XSLP
#6529	T6ZSLP
#6530	T6XSLM
#6531	T6ZSLM
#6532	T7XSLP
#6533	T7ZSLP
#6534	T7XSLM
#6535	T7ZSLM
#6536	T8XSLP
#6537	T8ZSLP
#6538	T8XSLM
#6539	T8ZSLM
#6540	T9XSLP
#6541	T9ZSLP
#6542	T9XSLM
#6543	T9ZSLM
#6544	T10XSLP
#6545	T10ZSLP
#6546	T10XSLM
#6547	T10ZSLM
#6548	T11XSLP
#6549	T11ZSLP
#6550	T11XSLM
#6551	T11ZSLM

#6552	T12XSLP
#6553	T12ZSLP
#6554	T12XSLM
#6555	T12ZSLM
#6556	T13XSLP
#6557	T13ZSLP
#6558	T13XSLM
#6559	T13ZSLM
#6560	T14XSLP
#6561	T14ZSLP
#6562	T14XSLM
#6563	T14ZSLM
#6564	T15XSLP
#6565	T15ZSLP
#6566	T15XSLM
#6567	T15ZSLM



Sets the distance of Stored Stroke Limit from reference point

Setting range 0 to ±99999999
Setting: Least output increment

#6568	XSKIP

Indicates X-axis coordinate value when the skip signal is detected.

#6569	ZSKIP

Indicates Z-axis coordinate value when the skip signal is detected.

#8601	TGPN01
#8602	TGPN02
⟆	⟆
#8649	TGPN49
#8650	TGPN50

TGPN01 to TGPN50:

Part program determines the number of groups including tools (number 01 to 50).

Setting range: 0 to 20
(Tool life control)

#8651	TOFN01
#8652	TOGN02
⟆	⟆
#8669	TOFN49
#8670	TOFN50

TOFN01 to TOFN50:

Part program sets tool number using offset value of offset memory numbers 01 to 50 orderly.

Setting range: 0 to 50
(Tool life control)

#8701	TOFO01
#8702	TOFO02
⟆	⟆
#8749	TOFO49
#8750	TOFO59

TOFO01 to TOFO05:

Part program sets the order of using offset values in offset memories "01" to "50," sequentially.

Setting range: 0 to 5
(Tool life control)

#6005	D7	D6	D5			D2	D1	D0

GCDSP D7

1: Uses special G code I as G coce.
0: Uses standard G code as G code.

RSTG01 D6

1: Determines G code of 01 group as G01 when resetting.
0: Determines G code of 01 group as G00 when resetting.

POSEXT D5

1: Presets position external display by setting coordinate system.
0: Does not preset position external display by setting coordinate system.

PONG01 D2

1 Sets the G code in the 01 group to G01 when power is applied.
0: Sets the G code in the 01 group to G00 when power is applied.

PONG05 D1

1: Sets the G code in the 05 group to G99 when power is applied.
0: Sets the G code in the 05 group to G98 when power is applied.

PONG03 D0

1: Sets the G code in the 03 group to G91 when power is applied.
0: Sets the G code in the 03 group to G90 when power is applied.

Note: Where the control is provided with special G code II option, determination of setting is changed as follows.

1: Uses special G code II.
0: Uses standard G code.

#6006	D7	D6	D5	D4	D3	D2	D1	D0

SDASGN2, SDASGN1: D7 , D6

Setting of S4-digit (analog output) output.

SDASGN2	SDASGN1	At M03 output	At M04 output
0	0	Plus	Plus
0	1	Minus	Minus
1	0	Plus	Minus
1	1	Minus	Plus

1OIN D5

1: Sets ten times the least input increment.
0: Sets the least input increment

SAGRCH D4

1: Checks to see if the spindle speed match signal (SAGR) is off upon transition from a rapid traverse block to a cutting feed block.
0: Provides no check on the spindle speed match signal (SAGR).

XRAD D3

1: Radius designation.
0: Diameter designation.

RPDDRN D2

1: Enables Dry Run in response to the rapid traverse command.
0: Disables Dry Run in response to the rapid traverse command.

ZZRNILK D1

1: Causes an alarm ("002") upon Cycle Start when Reference Point Return on Z-axis is not made manually after power is applied.
0: Causes no alarm.

XZRNILK D0

1: Causes an alarm ("001") upon Cycle Start when Reference Point Return on X-axis is not made manually after power is applied.
0: Causes no alarm.

NOTE: Set "1" when Stored Lead Screw Error Compensation or Stored Stroke Limit is provided, set ZZRNILK at 1, XZRNILK at 1.

#6007	D7	D6		D4	D3	D2	D1	D0

EDTSTLK D7

1: Does not cause an alarm upon Cycle Start without reset operation after part program edit operation.
0: Causes an alarm 005.

STUD D6

1: Effective on Cycle Start when cycle start signal "1" changes to "0."
0: Effective on Cycle Start when cycle start signal "0" changes to "1."

#6007	D_7	D_6		D_4	D_3	D_2	D_1	D_0

(Cont'd)

RWDOUT D_4

1: Provides Rewinding Activate Signal when NC program is rewound by RESET & REWIND signal.
0 Provides no Rewinding Activate Signal when NC program is rewound by RESET & REWIND signal.

OUTPUT D_3

1 Sets the least output increment at 0.0001 inch.
0. Sets the least output increment at 0.001 mm.

SCRSOV D_2

1: Makes the Spindle Override 100% during tapping
0 Does not make the Spindle Override 100% during tapping.

SLT3IO D_1

1 Establishes the prohibited area of the Stored Stroke Limit 3 outside the boundary.
0 Establishes the prohibited area of the Stored Stroke Limit 3 inside the boundary.

SLT2IO D_0

1 Establishes the prohibited area of the Stored Stroke Limit 2 outside the boundary.
0. Establishes the prohibited area of the Stored Stroke Limit 2 inside the boundary.

#6008			D_5				D_1	D_0

PONM97 D_5

1: M97 command (calculation of intersection) is selected at power-on.
M96 command (circular arc) is selected at power-on.

CVSAVE D_1

1: Does not clear user macro comman variable #100 thru #149 by reset.
0: Clears user macro common variables #100 thru #140 by reset operation.

ZRNOFS D_0

1 Cancels the commanded block when the second reference point by G30 is commanded during Tool Position Offset or Tool Nose Radius Compensation.
0 Cancels the blocks following the commanded block.

#6009	D_7	D_6					D_1	D_0

ZMOVILK D_7

1: After turning on power, if move command except by G28 is executed without returning Z-axis to reference point manually or automatically, alarm "001" will be caused.
0: Does not cause alarm in the same condition shown above.

XMOVILD D_6

1: After turning on power, if move command except by G28 is executed without returning X-axis to reference point manually or automatically, alarm "001" will be caused.
0: Does not cause alarm.

BLZDR, BLXDR D_1, D_0

Specify the start direction of backlash compensation on Z-, and X-axis, respectively.

1: Minus direction
0. Positive direction

#6010			D_5	D_4			D_1	D_0

AZRNHS D_5

1: Executes the first reference point return (deceleration limit switch) and the subsequen automatic reference point returns in the same way when power is applied.

0. Executes high-speed reference point return (position at reference point).

MZRNHS D_4

1 Executes the first reference point return and the subsequent automatic reference point returns in the same way when power is applied.

0: Executes high-speed reference point return.

ZRNDRZ, ZRNDRX D_1, D_0

Specify the start direction of Backlash Compensation on Z-, and X-axis, respectively.

1: Minus direction
0: Plus direction

#6015							D_1	D_0

ATSUPZ, ATSUPX D_1, D_0

Specify whether or not the Automatic Coordinate System Setting is effective on the Z- and X-axis, respectively.

1. Effective
0: Ineffective

NOTE: The Automatic Coordinate System is established with the following parameters:

Inch system: #6631, #6630
Metric system: #6637, #6636

#6017	D7	D6	D5	D4	D3	D2	D1	D0

EIA#B7-B0 D7 - D0

Specify whether or not a hole is to be made on channels 8-1, respectively, in a code corresponding to symbol "#" (used with user macro) in the EIA code.

1: Hole
0: No hole

Example: EIA#B7-B0 = 01001001

The code with holes on channels 7, 4, and 1 is considered equivalent to symbol "#" in the EIA code. No code for use by the unit can be set.

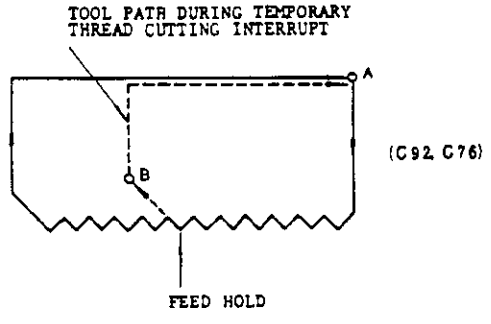
NOTE: The specification of EIA#B7-B0 = 00000000 assumes that symbol "#" is not used in the EIA code.

#6019	D7	D6	D5	D4				

G92FHP D7

Specifies the position of temporary stop of threadcutting.

1: Stops at the position B where Threading-up is completed.
0: Returns to start point A and stops after Threading-up is completed.



INTG D6

1: Sets the least input increment at 1 mm (or 1 inch).
0: Sets the least input increment using parameter #6006D5 (10IN).

SCRDRN D5

1: Enables Dry Run at threadcutting.
0: Disables Dry Run at threadcutting.

SKPFED D4

1: Employs the feedrate set in parameter #6232 (G31F) for the Skip Function command (G31).
0: Employs the F code command as the feedrate for the Skip Function command (G31).

#6020	D7		D5	D4		D2	D1	D0

OFSDSP D7

1: Displays programmed position in current position display (POSITION ABSOLUTE).
0: Displays programmed position modified with tool position offset in current position display (POSITION ABSOLUTE).

FOVAB D5

1: Effective with feedrate override signal "0."
0: Effective with feedrate override signal "1."

SSTPAB D4

1: Analog output zero with spindle S command zero input signal SSTP "0."
0: Analog output zero with spindle S command zero input signal SSTP "1."

PSTSGN D2

Shown in the calculation formula of storing data during MDI of measured work point into tool offset memories 00 to 50.

1: (Data of tool coordinate memory) = (Current value temporarily stored in the register) - (Written measurement value)

0: (Data of tool coordinate memory) = (Written measurement value) - (Current value temporarily stored in the register)

OFSG96 D1

1: Specifies the surface speed calculated by the X-axis coordinate value modified by tool position offset value in Constant Surface Speed Control.
0: Specifies the surface speed calculated by the programmed X-axis coordinate value in Constant Surface Speed Control.

#6020	D7		D5	D4		D2	D1	D0

POSG96 D0

1: Surface Speed Control functions on the block including Rapid Traverse (G00).
0: Surface Speed Control functions on the block including Rapid Traverse (G00), if programmed before the Cutting Feed block.

#6021	D7	D6	D5	D4	D3	D2	D1	D0

UMO9000 D7

1 Inhibts editing and punchout operations of the part program of program No. 9000 to 9999.
0 Permits editing and punchout operations

MERSIN D6

1 Replaces the stored program with a new one when part program is already stored.
0 Displays ALREADY ALARM.

PSONOF D5

1 Sets on and off RS (RS232C signal) by "%" character.
0. Keeps RS signal on until reading-in is finished

CHKDR D4

1 Recognizes DR
0 Does not recognize DR.

O - 9999O D3

1 Punches O0 when tape is punched with O, -, 9, 9, 9, 9 keyed in and OUT key depressed.
0 Does not punch O0 when tape is punched with O, -, 9, 9, 9, and 9 keyed in and OUT key depressed

PONON D2

1 Does not clear program No. on power application
(Program number is stored at power supply shut off)
0 Clears program No. on power application.

PRGNO D1

1 Employs the value following address O or N as the program number (specifiable in one block).
0 Employs the value following address O as the program number.

M02M99 D0

1 Considers M02, M30 and M99 as the program end when part program is stored into memory.
0. Does not consider M02, M30 and M99 as the program end when part program is stored into memory.

#6022	D7	D6					D1	D0

HOFSMV D7

1: Enables the movement of automatic mode handle offset during cutting feed by interpolation
0: Enables the movement of automatic mode handle offset except during execution of rapid traverse.

ISOPO D3

1: Does not output parity bit (8th bit)when outputting ISO codes from NC by operating OUT key (in the EDIT mode).
0: Outputs parity bit.

ISOPI D3

1: Ignores parity bit (8th bit) when outputting ISO codes by operating IN key (in the EDIT mode) and when reading-in ISO tape data in the TAPE mode.
0: Performs parity check.

HOFSZ, HOFSX D1, D0

Specifies whether automatic mode handle offset movement is effective or ineffective.

1. Effective automatic mode handle offset movement.
0. Ineffective automatic mode handle offset move ment.

#6023	D7			D4		D2		

PERIAB D7

1: Incremental setting of offset value for Stored Leadscrew Error Compensation.
0: Absolute setting of offset value for Stored Leadscrew Error Compensation.

PERST D6

1: Regards"%" code as M30, if "%" is commanded before M02 or M30 in TAPE or MEM mode operation.
0: Ignores "%" code if commanded before M02 or M30 in TAPE or MEM mode operation.

WOPMCT D4

1: Ignores the second input when tool wear compensation input WOP (or WOM) is inputted continuously two times.
0: Adds or subtracts the offset value when tool wear compensation input WOP (or WOM) is inputted continuously two times.

CONP D2

1: Sets cut depth value override with cut depth override input in Stroke Removal in Turning (G71) and stock removal in facing (G72).
0: Sets cut depth value override with setting #6004.

#6024		ORGZ	NZZ	CNZZ		ORGX	NZX	CNZX

Sets the method of reference point return on Z- and X-axis.

Return to reference point system (X-axis / Z-axis)	ORGX / ORGZ	NZX / NZZ	CNZX / CNZZ
Grid system (Reference pulse)	1	0	0
Near zero system (Signal "1")	0	1	0
Near zero system (Signal "0")	0	1	1

Input for Current Loop and RS232C

#6026			D5	D4	D3	D2	D1	D0

SIF1CI D5

Determines whether the input control code for current loop and RS232C interface is given or not.

1: Does not send control code.
0: Sends control code.

SIF1SI D4

Determines the input stop bit for current loop and RS232C interface as two bits or one bit.

1: Determines stop bit as two bits.
0: Determines stop bit as one bit.

SIF1BID - SIF1BIA D3 - D0

Sets input baud rate for current loop and RS232C interface.

Baud rate	SIF1BID	SIF1BIC	SIF1BIB	SIF1BIA
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

Input for RS-422 Interface

#6027			D5	D4	D3	D2	D1	D0

SIF2CI D5

Determines whether the input control code for RS422 interface is sent or not.

1: Does not send control code.
0: Sends control code.

SIF2SI D4

Determines the input stop bit for RS422 interface as two bits or one bit.

1: Determines stop bit as two bits.
0: Determines stop bit as one bit.

SIF2BID-SIF2BIA D3 - D0

Sets input baud rate for RS422 interface.

Baud rate	SIF2BID	SIF2BIC	SIF2BIB	SIF2BIA
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

Output for Current Loop and RS232C

#6028			D5	D4	D3	D2	D1	D0

SIF1CO D5

Determines whether output control code for current loop and RS232C interface is sent or not.

1 Does not send control code.
0. Sends control code.

SIF1SO D4

Determines output stop bit for current loop and RS232C interface as two bits or one bit.

1 Determines stop bit as two bits.
0 Determines stop bit as one bit.

SIF1BOD-SIF1BOA D3 - D0

Sets output baud rate for current loop and RS232C interface.

Baud rate	SIF1BOD	SIF1BOC	SIF1BOB	SIF1BOA
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

Output for RS422 Interface

#6029			D5	D4	D3	D2	D1	D0

SIF2CO D5

Determines whether output control code for RS422 interface is sent or not.

Does not send control code.
Sends control code.

SIF2SO D4

Determines output stop bit for RS422 interface as two bits or one bit.

1 Determines stop bit as two bits.
0 Determines stop bit as one bit.

SIF2BOD-SIF2BOA D3 - D0

Sets output baud rate for RS422 interface.

Baud rate	SIF2BOD	SIF2BOC	SIF2BOB	SIF2BOA
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

#6050	XBLP
#6051	ZBLP

XBLP, ZBLP:

Sets backlash compensation value for X- and Z-axis.

Setting range: 0 - 255
Setting Least output increment

#6056	XPSET
#6057	ZPSET

XPSET, ZPSET

Sets position error range for X- and Z-axis.

Setting range: 0 - 255
Setting Least output increment

#6068	XPERML
#6069	ZPERML

XPERML, ZPERML:

Sets leadscrew error compensation multiplication factor for X- and Z-axis.
Outputs the result of the preset compensation value multiplied by the multiplication factor as the error compensation value.

Setting range 0 - 3
(Setting 0 will not execute compensation.)

#6074	XSVER
#6075	ZSVER

XSVER, ZSVER:

Sets servo error limit for X- and Z-axis. Position deviation exceeding the preset value causes an alarm "34 Δ."

Setting range: 0 - 255
Standard setting: 16
Setting: 1/16 x (D/A saturation value)

#6080	CUPRD

Rapid threading pull-out width during thread-cutting

Setting range: 0 - 255
Setting: 0.1 lead

#6081	SIDRG

Spindle indexing completion output allowable range

Setting range: 0 - 255
Setting: 1 = 1 pulse (= 360/4096 deg)

#6083	SIDGAN 1

Sets spindle indexing command voltage gain No.1.

Setting range: 0 - 255
Setting: 1 = 0.31 mV/pulse

#6084	SIDGAN2

Sets spindle indexing command voltage gain No. 2.

Setting range: 0 - 255
Setting: 1 = 0.31 mV/pulse

#6085	SIDSER

Sets the percentage of the spindle speed for starting spindle indexing.

Setting range: 0 - 10
Setting: 1 = (Spindle indexing speed command) $\times \frac{1}{100}$

#6092	CUTACC

CUTACC:

Time constant at Exponential Acceleration/ Deceleration during feed.

Setting range: 0 - 255
Setting: "n" = $\frac{t}{4} - 1$ t: constant (ms), set by 4 ms.

#6093	CUTBAS

CUTBAS:

Sets bias speed at Exponential Acceleration/ Deceleration during feed.

Setting range: 0 - 255
Setting: 120 mm/min (metric output)
120 in/min (inch output)

#6094	SCRACC

SCRACC

Time constant at Exponential Acceleration/ Deceleration during threadcutting.

Setting range: 0 - 255
Setting: "n" = $\frac{t}{4} - 1$ t: constant (ms), set by 4 ms.

#6095	SCRBAS

SCRBAS:

Sets bias speed at Exponential Acceleration/ Deceleration during threadcutting.

Setting range: 0 - 255
Setting: 2Kpps

#6096	WOIMUL

WOIMUL:

Sets the multiplication factor of changed compensation value from external input during external tool compensation function (M94, M95). The final changed value is the result of the changed compensation value by external input multiplied by this multiplication factor.

Setting range: 1 - 10
Setting: 0.1

#6108	UMEIA[
#6109	UMEIA]

#6110	UMEIA*
#6111	UMEIA=
#6112	UMEIA(
#6113	UMEIA)

UMEIAs:

Specify the punching pattern in EIA for special characters employed in user macro; [,] , *, =, (,), used in turn, beginning with #6108.

Setting range: 0 - 255
Setting Sets the punching pattern using the decimal value converted from the binary value which defines the pattern.

Note: When "0" is set for each character, punching pattern will be as listed below.

Special Character	8	7	6	5	4	∘	3	2	1
[				○	○	∘	○		
]		○			○	∘	○		
*					○	∘	○	○	
=			○	○	○	∘		○	○
(				○	○	∘		○	
)		○			○	∘		○	

#6114	NBUFM1
#6115	NBUFM2
#6116	NBUFM3
#6117	NBUFM4
#6118	NBUFM5
#6119	NBUFM6

NBUFM1, 2, 3, 4, 5, 6:

Sets up to 6 M codes for stopping advance reading function (buffering).

Setting range. 0 - 255

#6120	UMG1
#6121	UMG2
#6122	UMG3
#6123	UMG4
#6124	UMG5
#6125	UMG6
#6126	UMG7
#6127	UMG8
#6128	UMG9
#6129	UMG10

UMG1 - 10:

Sets G codes for calling user macro of program No. O9010 to O9019.

Setting range: 0 - 255

#6130	UMM1
#6131	UMM2
#6132	UMM3
#6133	UMM4

UMM1, UMM2, UMM3, UMM4:

Sets M codes for calling user macro of program No. O9001 to O9004.

Setting range: 0 - 255

#6134	UMT

UMT

1: Regards T-code command as macro call command calling the macro of program No. O9000.
0: Regards T-code command as basic T-code.

Note: This selection is effective only for the user macro option.

#6220	MSTF

MSTF:

Sets the interval from the time M, S, and T codes are transmitted until the time MF, SF, and TF are transmitted.

Setting range: 0 - 65536 msec

#6222	HPMAX

HPMAX:

Specifies the maximum handle feedrate, which is common to the all axes.

Setting: "1" = 125 pulses/sec

#6224	SAGRT

SAGRT:

Specifies the delay time for checking the spindle speed reaching signal (SAGR).

Setting range. 0 - 65536 msec

#6228	G98MAX

G98MAX:

Specifies the maximum feedrate at G98 command (feed per minute) common to all axes.

Setting range:

Setting: "1" = 1000 pulses/min

#6229	G35F

G35F:

Specified the feedrate at Tool Set Error Compensation (G35).

Setting range:

Setting: "1" = 1000 pulses/min

When the parameter is set at "0," feedrate follows F command.

#6230	NEGNR

NEGNR:

When a circular path is drawn in Tool Radius Compensation outside a corner approaching 180°, the movement follows describing a very small circular arc. This parameter is used to set the critical arc value, if this arc movement is considered to affect the workpiece surface machining.

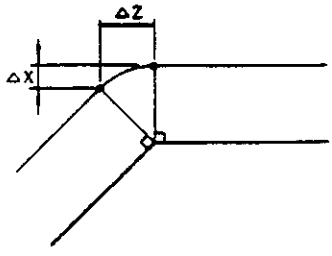
Setting range: 0 - 65536
Setting: Least input increment

The corner arc setting is ignored when:

$\Delta X \leq$ NEGNR

$\Delta Y \leq$ NEGNR

Standard setting = 5



#6231	ROVFO

ROVFO:

Specifies the FO speed for Rapid Traverse Override.

Setting range:
Setting: "1" = 125 pulses/sec

#6232	G31F

G31F:

Specifies the feedrate in the skip function (G31).

Setting range:

Setting: "1" = 1000 pulses/min

This setting is effective when parameter #6019D4 (SKPFED) = 1.

#6233	JOG0
⟨	⟨
#6264	JOG31

JOG0~JOG31:

Specify the feedrates for the respective positions on the jog feedrate select switch.

Setting range:

Setting: "1" = 0.5 mm/min (metric output)
"1" = 0.05 in./min (inch output)

Switch Position	Feedrate Override %	Parameter Number	Parameter Setting	Continuous Manual Feedrate mm/min	
0	0	#6233	0	0	
1	10	#6234	1	1	
2	20	#6235	2	2	
3	30	#6236	4	4	
4	40	#6237	6	6	
5	50	#6238	8	8	
6	60	#6239	10	10	
7	70	#6240	12	12	
8	80	#6241	15	15	
9	90	#6242	20	20	
10	100	#6243	25	25	
11	110	#6244	30	30	
12	120	#6245	40	40	
13	130	#6246	50	50	
14	140	#6247	60	60	
15	150	#6248	80	80	
16	160	#6249	100	100	
17	170	#6250	120	120	
18	180	#6251	150	150	
19	190	#6252	200	200	
20	200	#6253	250	250	
21	0	#6254	300	300	
22	0	#6255	400	400	
23	0	#6256	500	500	
24	0	#6257	600	600	
25	0	#6258	800	800	
26	0	#6259	1000	1000	
27	0	#6260	1200	1200	
28	0	#6261	1500	1500	
29	0	#6262	2000	2000	
30	0	#6263	2500	2500	
31	0	#6264	3000	3000	

#6266	MACGR1
#6267	MACGR2
#6268	MACGR3
#6269	MACGR4

MACGR1-MACGR4:

Sets spindle speed upper limit for gear 1, 2, 3, and 4 orderly.

Setting range 0 - 6000 (rpm)

When the setting is at 0, the speed is not clamped.

#6270	GRSREV

GRSREV:

Sets the speed command output to spindle motor when gear shift input (GRS) is given.

Setting value. $\frac{\text{Gear shift spindle motor speed}}{\text{Spindle motor max speed (Command = 10 V)}}$ x 2047 --- 12-bit output

$\frac{\text{Gear shift spindle motor speed}}{\text{Spindle motor max speed (Command = 10 V)}}$ x 32512 --- Analog output

Setting range: 0 - 6000

#6271	GR1REV
#6272	GR2REV
#6273	GR3REV
#6274	GR4REV

GR1REV-GR4REV:

Specify the maximum speed of the spindle, respectively, for gears 1, 2, 3 and 4 each selected by an input signal. Set the spindle speed applicable when the speed command voltage is 10 V.

Setting range. 0 - 6000 (rpm)

#6275	GSCREV

GSCREV:

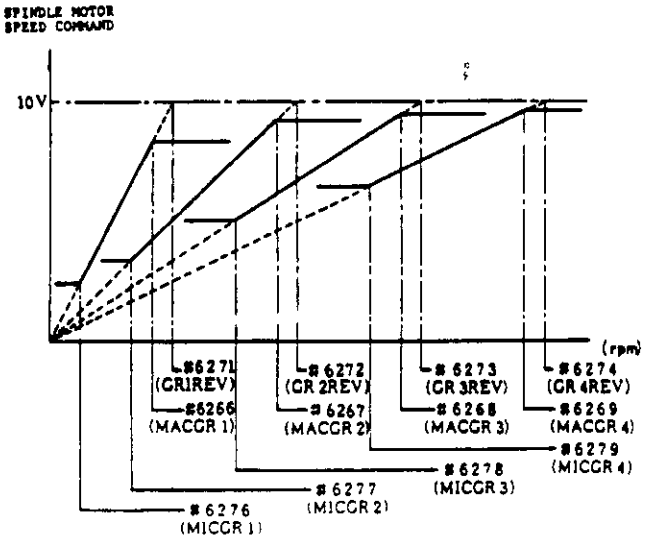
Specifies the spindle motor speed in effect when a spindle operation (GSC) input is entered.

Setting range: 0 - 6000 (rpm)

#6276	MICGR1
#6277	MICGR2
#6278	MICGR3
#6279	MICGR4

Specify the minimum speed of the spindle, respectively for gears 1, 2, 3 and 4 each selected by an input signal.

Setting range. 0 - 6000 (rpm)



#6280	RPDX
#6281	RPDZ

RPDX, RPDZ:

Specify the rapid traverse rate for X- and Z-axis, respectively.

Setting range: 0 - 3200
Setting: "1" = 125 pulses/sec

#6286	ACCX1
#6287	ACCZ1

ACCX1, ACCZ1:

Set the time constant for Linear Accel/Decel for X- and Z-axis, respectively.

Setting range:

Setting: "1" = $125/8 \times 10^3$ P/sec^2
(P: least output increment)

#6304	XREFP
#6305	ZREFP

XREFP, ZREFP:

Sets the traverse distance for Reference Point Return, respectively, on the X- and Z-axis.

Setting range: 0 - 65535

Setting: "1" = 1 pulse

#6310	XREFV1
#6311	ZREFV1

XREFV1, ZREFV1:

Specify the approach speed 1 for Reference Point Return, respectively, on the X- and Z-axes.

Setting range: 0 - 200

Setting: "1" = 125 pulses/sec

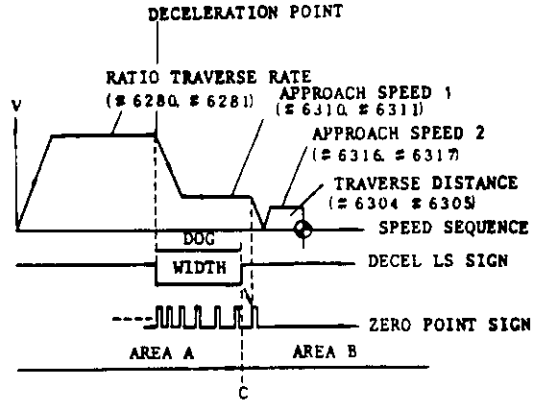
#6316	XREFV2
#6317	ZREFV2

XREFV2, ZREFV2:

Specify the approach speed 2 for Reference Point Return, respectively, on the X- and Z-axes.

Setting range: 0 - 200

Setting: "1" = 125 pulses/sec



· Reference point return direction.
#6010 (ZRNDRX, ZRNDRZ)

#6322	XPERED
#6323	ZPERED

XPERED, ZPERED:

Specify the number of the end point for Leadscrew Error Compnesation, respectively, on the X- and Z-axes.

Setting range: 0 - 255

#6328	XPERST
#6329	ZPERST

XPERED, ZPERED

Specify the number of the start point for Leadscrew Error Compensation, respectively, on X- and Z-axes.

Setting range: 0 - 255

#6334	XPEROR
#6335	ZPEROR

XPEROR, ZPEROR.

Specify the reference point for Leadscrew Error compensation, respectively, on the X- and Z-axes.

Setting range 0 - 255

#6342	SIFREF

SIDREF

Sets the reference point for spindle indexing.

Setting range 0 - 4095

Setting: "1" = 1 pulse (= 360/4096 deg.)

#6343	SIDRV1

SIDRV1

Sets the spindle speed for spindle indexing.

Setting range. 0 - 32512

Setting. 1 = 0.31 mV

#6344	SIDCRP

SIDCRP

Sets the spindle indexing creep speed.

Setting range 0 - 31512

Setting: "1" = 0.31 mV

#6345	SIDCRS

SIDCRS

Sets the spindle indexing creep start position.

Setting range: 0 - 4095

Setting: 1 = 1 pulse (= 360/4096 deg)

#6346	SIDGEP

SIDGEP

Sets the spindle indexing command voltage gain No. 2 start position.

Setting range: 0 - 4095

Setting: "1" = 1 pulse (= 360/4096 deg)

#6600	XSL1P
#6601	ZSL1P

XSL1P, ZSL1P:

Specify the plus direction boundary value for Stored Stroke Limit 1, respectively, on the X-, and Z-axes.

Setting range. 0 - 99999999

Setting: "1" = 1 pulse

#6606	XSL1M
#6607	ZSL1M

XSL1M, ZSL1M

Specify the minus direction boundary value for Stored Stroke Limit 1, respectively, on the X-, Z-axes.

Setting range: 0 - 99999999

Setting: "1" = 1 pulse

#6612	XZP2L
#6613	ZZP2L

XZP2L, ZZP2L:

Specify the distance between the first and the second reference point, respectively, on the X-, Z-axes.

Setting range: -99999999 - 99999999

Setting: "1" = 1 pulse

#6624	XBPTS
#6625	ZBPTS

XBPTS, ZBPTS:

Sets the absolute coordinate values of X- and Z-axis where the reference tool turns on touch switch during tool set error compensation of X- and Z-axis.

#6630	XSETI
#6631	ZSETI

XSETI, ZSETI:

Specify the value for Automatic Coordinate System Setting at the time of inch input, respectively, on the X-, and Z-axes. A desired value should be set in inches for the distance between the first reference point and the reference point of the coordinate system to be established.

Setting range: -99999999 - 99999999

Setting. "1" = 0.0001 in.

#6636	XSETM
#6637	ZSETM

XSETM, ZSETM:

Specify the value for Automatic Coordinate System Setting at the time of metric input, respectively, on the X-, and Z-axes. A desired value should be set in millimeters for the distance between the first reference point and the reference point of the coordinate system to be established.

Setting range: -99999999 - 99999999

Setting: "1" = 0.001 mm

#6642	XPEINT
#6643	ZPEINT

XPEINT, ZPEINT:

Specify the compensation inverval in Leadscrew Error Compensation, respectively, on the X- and Z-axes.

Setting range: -99999999 - 99999999

Setting: "1" = 1 pulse

#8000	PEMN0
⟆	⟆
#8255	PEMN255

PEMN0-PEMN255:

Specify the respective values of Leadscrew Error Compensation.

Setting range: 0 - ±15 (Incremental designation)
0 - ±128 (Absolute designation)

Setting: "1" = Output increment

Incremental/absolute designation is selected by parameter #6023D7 (PERIAB).

Axis for compensation is specified by parameters #6322, 6323, 6328, and 6329.

APPENDIX 3 STORED LEADSCREW ERROR COMPENSATION

This function automatically compensate for leadscrew error on each axis according to the compensation data set by parameter and is effective after completion of reference point return. The compensation data are made on the distances between the reference point on each axis and specified points.

Compensation axes: X, Z axes

No. of correction points: 256 Max.

Compensation base point: Reference point

Compensation interval: 6000 Pulses or more

Data setting system: Absolute/incremental
(Set by Parameter #6023D7 PERIAB)

Compensation value.

Minimum compensation unit: 1 pulse (least output increment)

Compensation multiplication factor: 3X max.

One-time-compensation value: 15 pulses max. (Compensation multiplication)

Notes:

1. Regardless of absolute/incremental setting, the difference between neighboring compensation values should be (15 pulses x compensation multiplication) and below.
2. Maximum set value in case of absolute setting is ±127 pulses. Compensation multiplication is taken on this value.
3. No. of correction points on each axis can be arbitrary as far as the total compensation points are within 256.

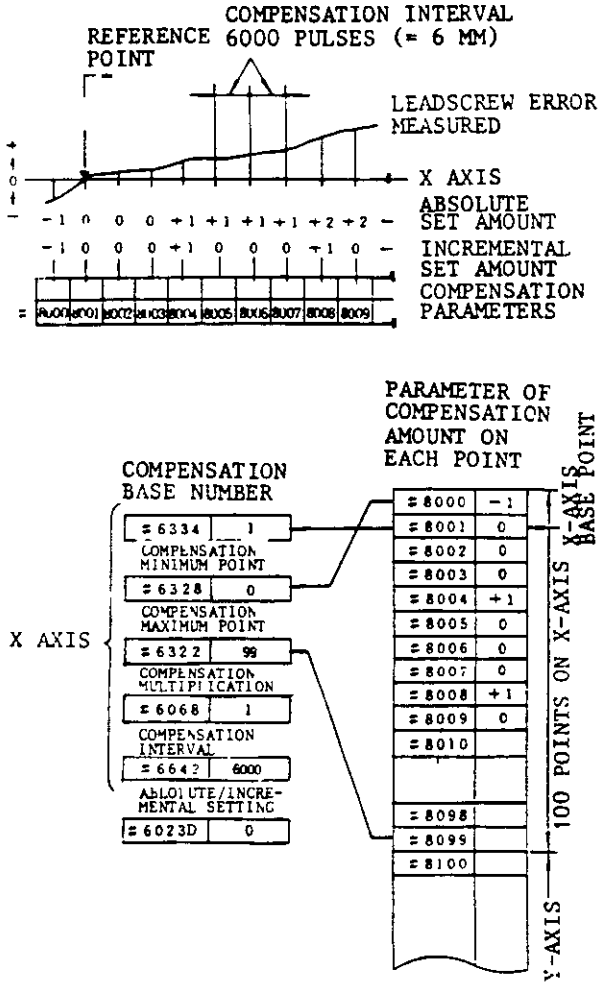


Table 3.1

	Axis	Parameter #	Functions
Compensation interval	X	#6642 (XPEIN1)	0000 OR MORE "1" = 1 pulse
	Z	#6643 (ZPEINT)	
Absolute/incremental settig switchable		#6023D7 (PERIAB)	"0" = Incremental setting "1" = Absolute setting
Compensation reference no	X	#6334 (XPEROR)	Value of parameter # of compensation on each point minus 8000 will be written
	Z	#6335 (ZPEROR)	
Compensation max point	X	#6322 (XPERED)	
	Z	#6323 (ZPERED)	
Compensation min point	X	#6328 (XPERST)	
	Z	#6329 (ZPERST)	
Compensation value on each point	X	#8000 - #8255	0 to ±7 (Incremental setting) 0 to ±127 (Absolute setting) "1" = 1 pulse
	Z		
Compensation multiplication factor	X	#6068 (XPERML)	0 to 3 "1" = 1X
	Z	#6069 (ZPERML)	

APPENDIX 4 LIST OF STANDARD INPUT/OUTPUT SIGNALS

Table 4-1 shows the list of diagnostic numbers and signal names of standard input/output signals and monitor signals.

Diagnostic number	Display
#1000-#1096	Input signals from machine
#1100-#1157	Output signals to machine
#1200-#1223	Output signals to machine interface (PC)
#1300-#1331	Input signals from machine interface (PC)
#1280-#1295	Monitor signals

Notes:

1. Monitor signals are used to check the internal condition of the control.
2. The functions of signals #1000 - #1096, #1100 - #1157

Refer to machine tool builder's manual.

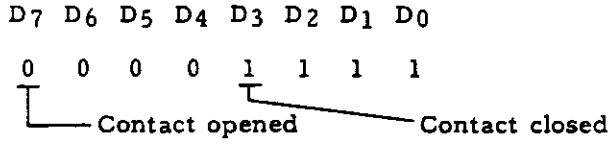
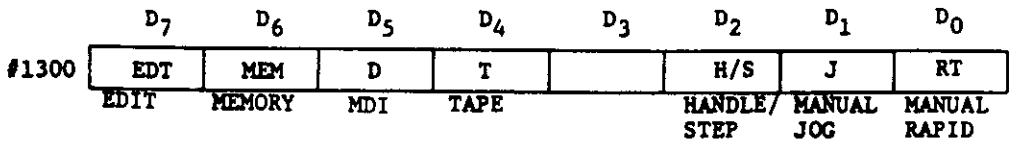
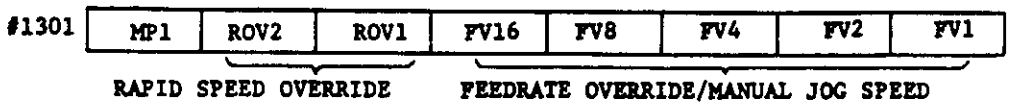


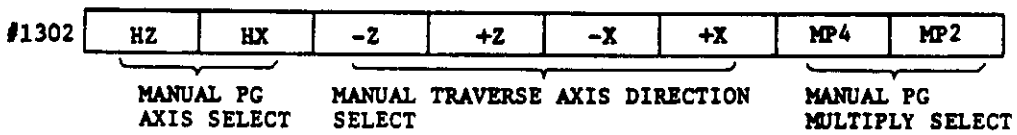
Fig. 4.1 Status Display of Input/Output Signals

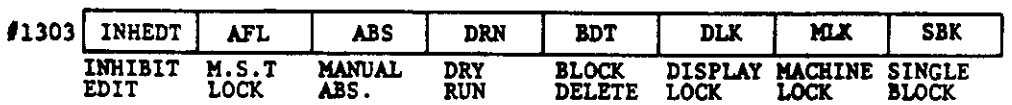
Table 4-1 List of Standard Input/Output Signals

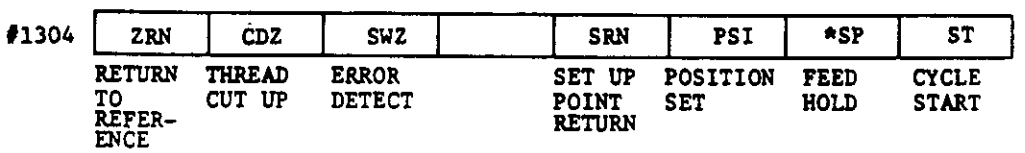
Input Signals

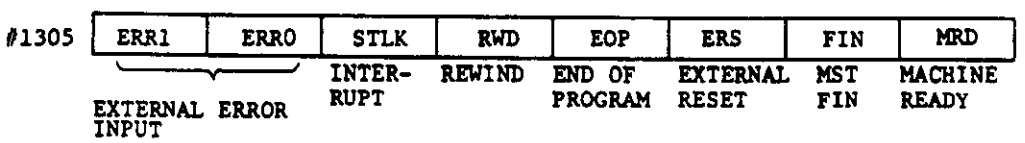


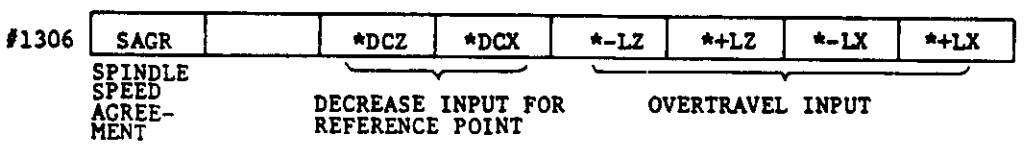












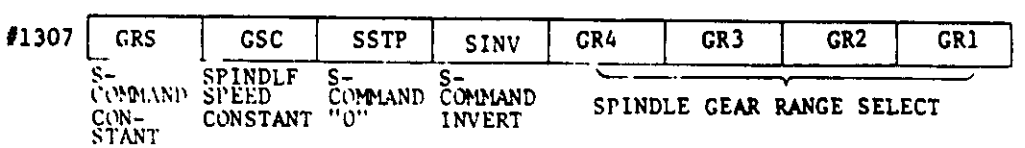


Table 4-1 List of Standard Input/Output Signals (Cont'd)

Input Signals

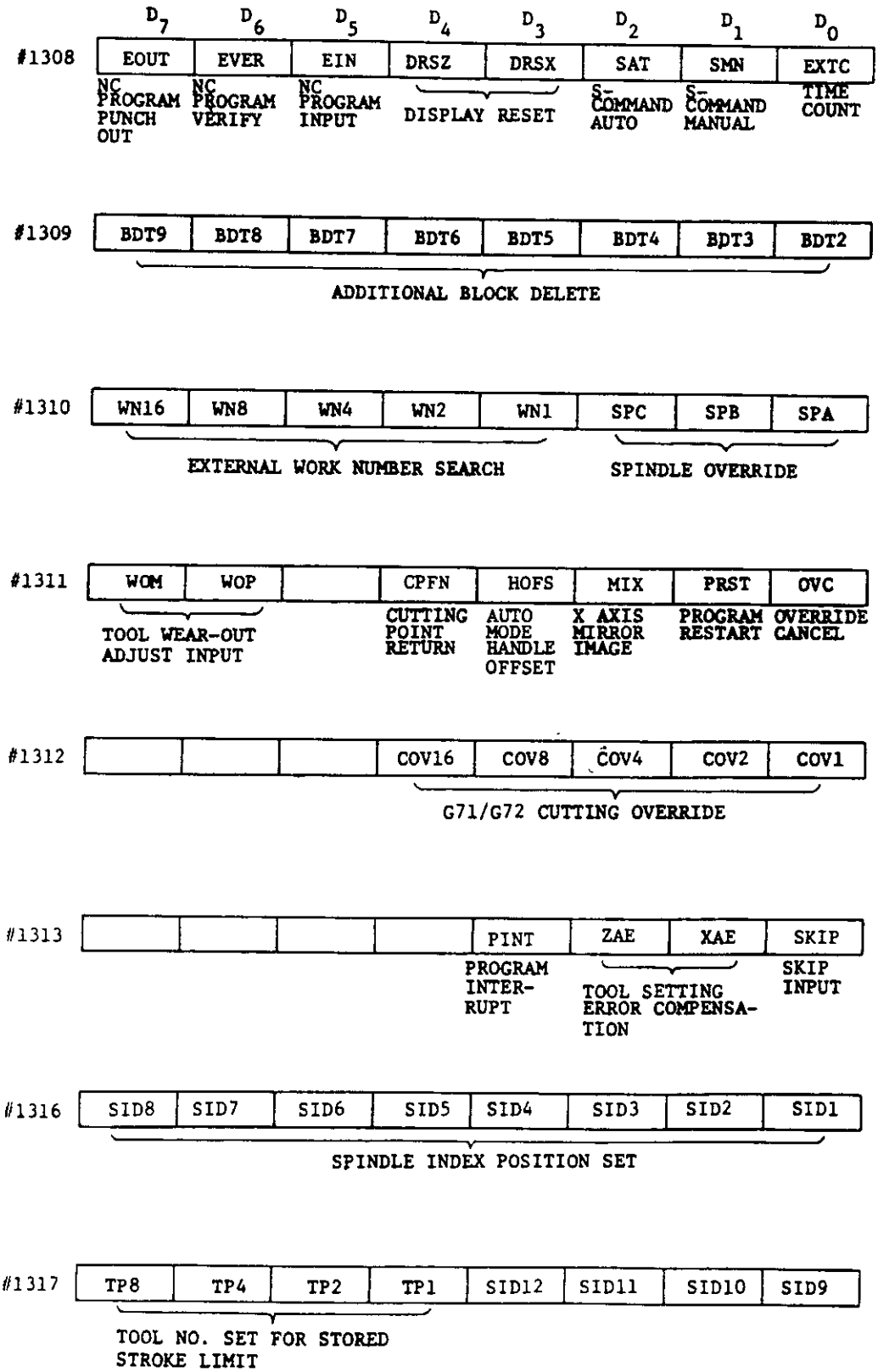


Table 4-1 List of Standard Input/Output Signals (Cont'd)

Input Signals

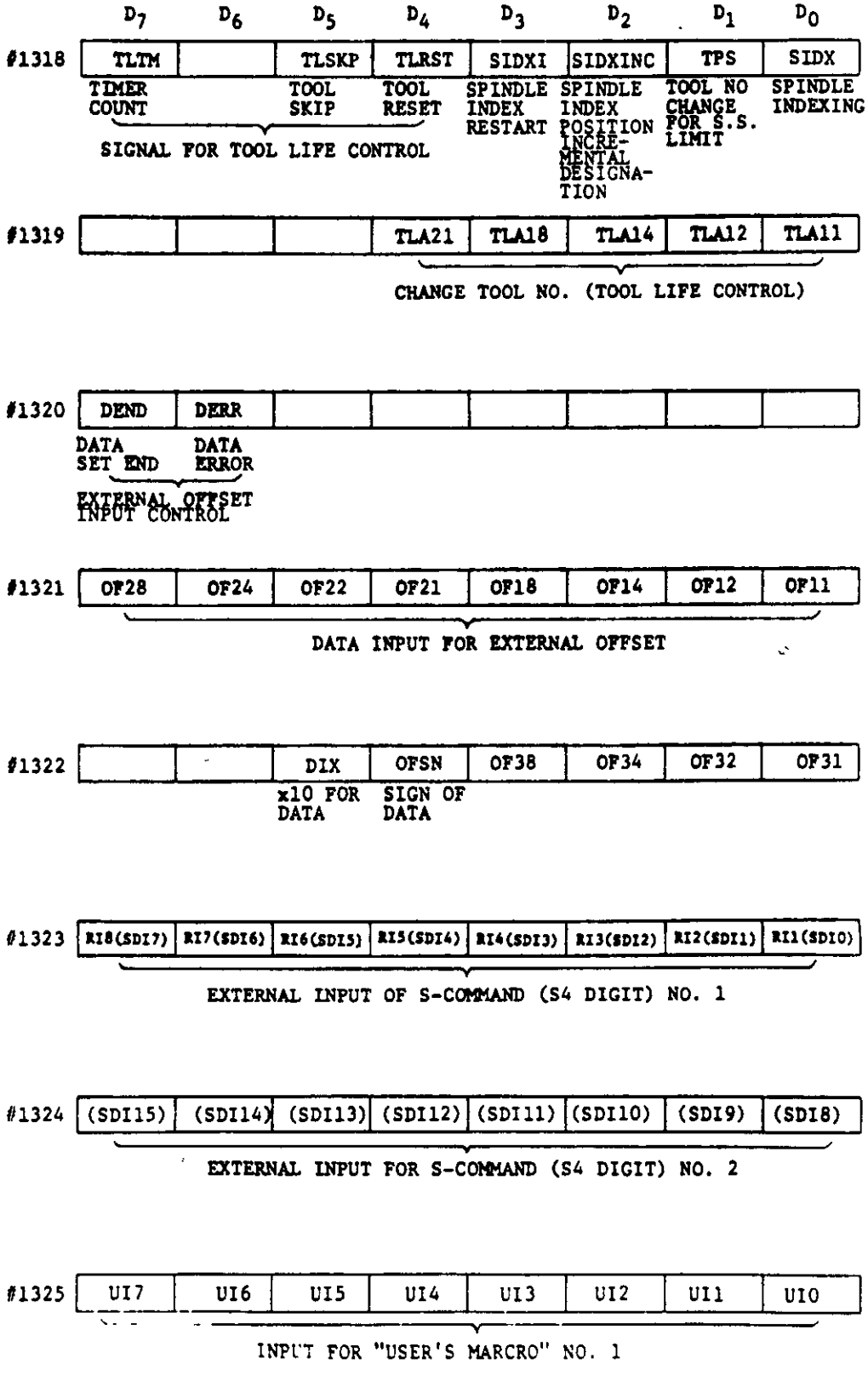
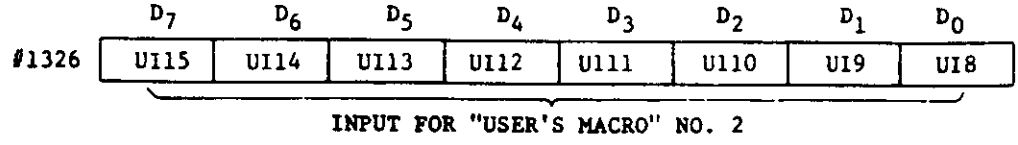
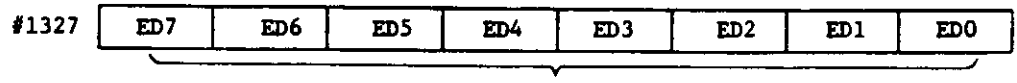
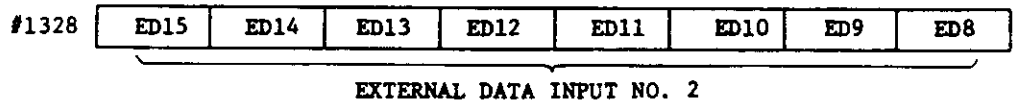


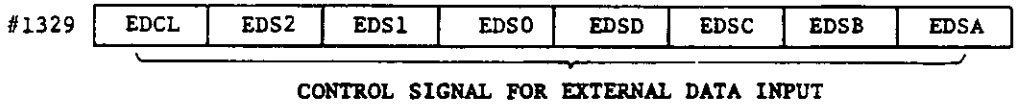
Table 4-1 List of Standard Input/Output Signals (Cont'd)

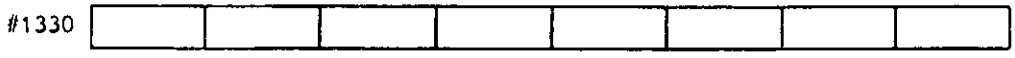
Input Signals

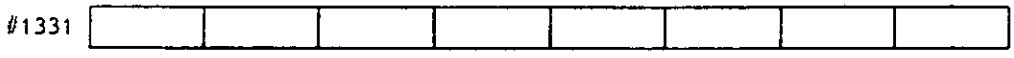


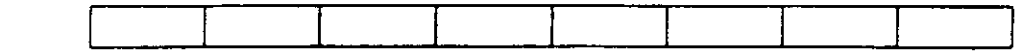












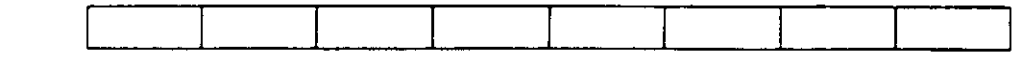


Table 4-1 List of Standard Input/Output Signals (Cont'd)

Output Signals

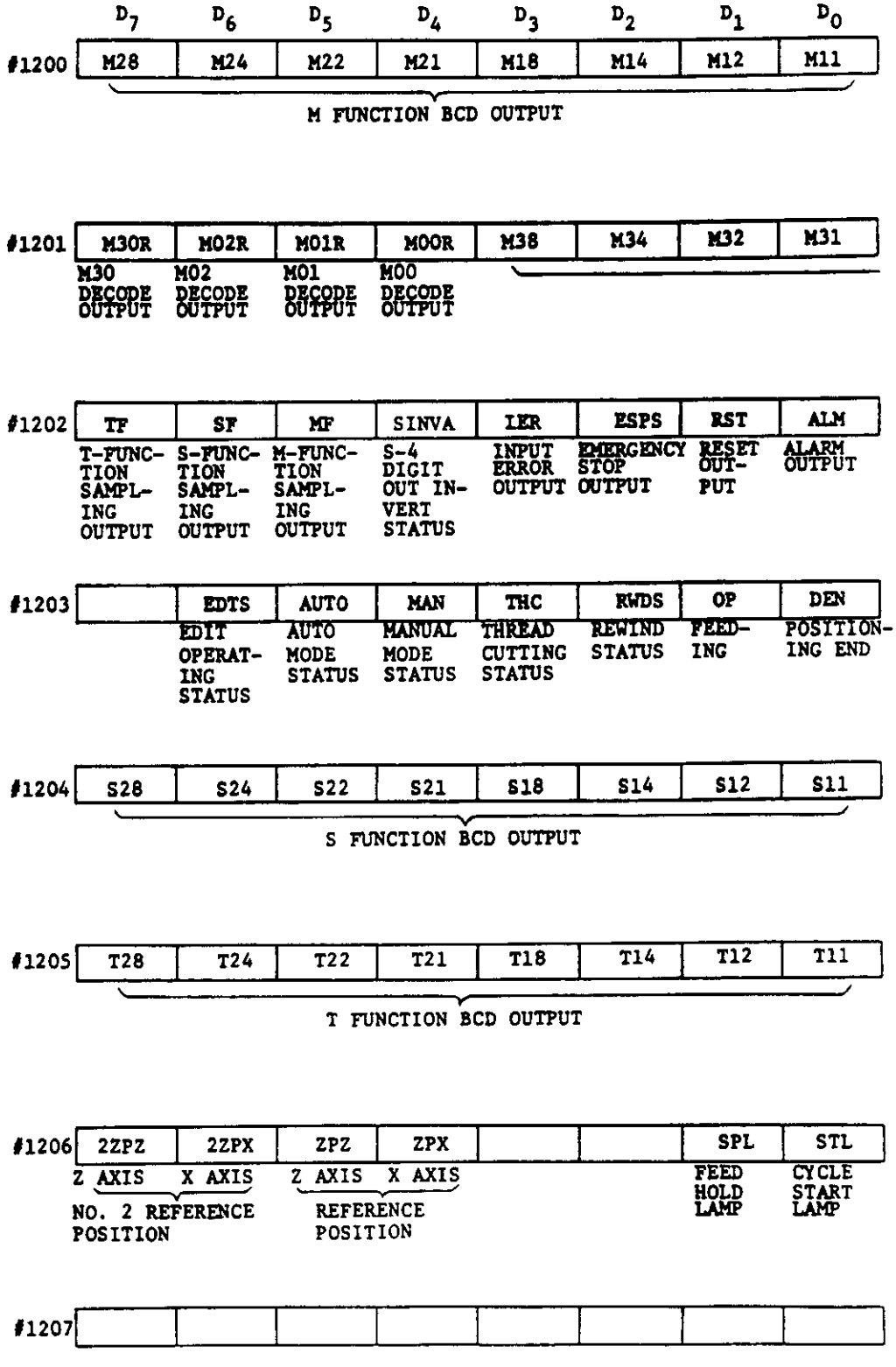
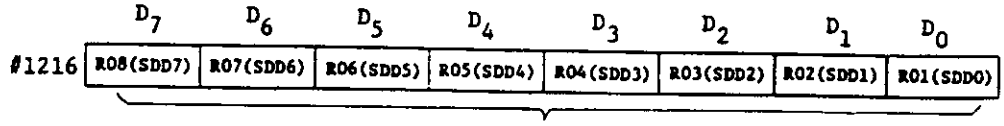
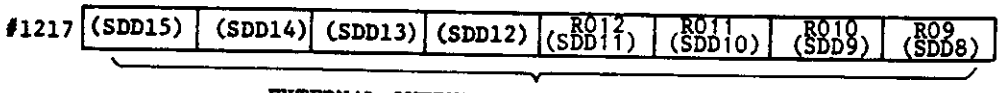


Table 4-1 List of Standard Input/Output Signals (Cont'd)

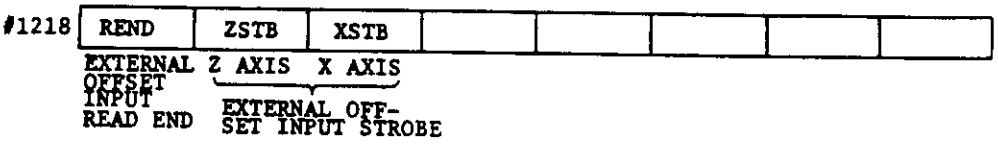
Output Signals

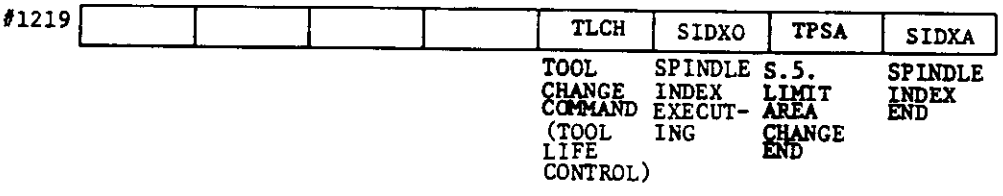


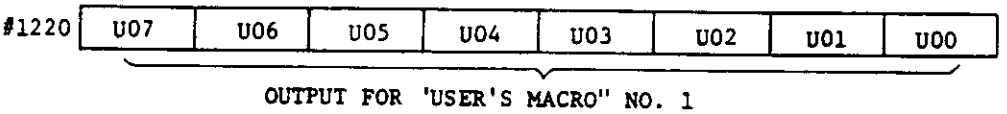
EXTERNAL OUTPUT FOR S-COMMAND (S4 DIGIT) NO. 1



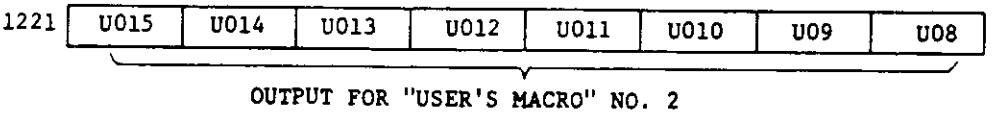
EXTERNAL OUTPUT FOR S-COMMAND (S4 DIGIT) NO. 2







OUTPUT FOR 'USER'S MACRO" NO. 1



OUTPUT FOR "USER'S MACRO" NO. 2

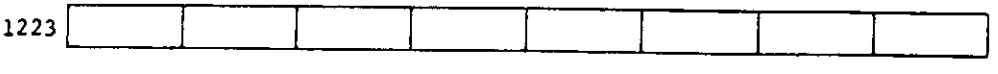


Table 4-1 List of Standard Input/Output Signals (Cont'd)

Output Signals

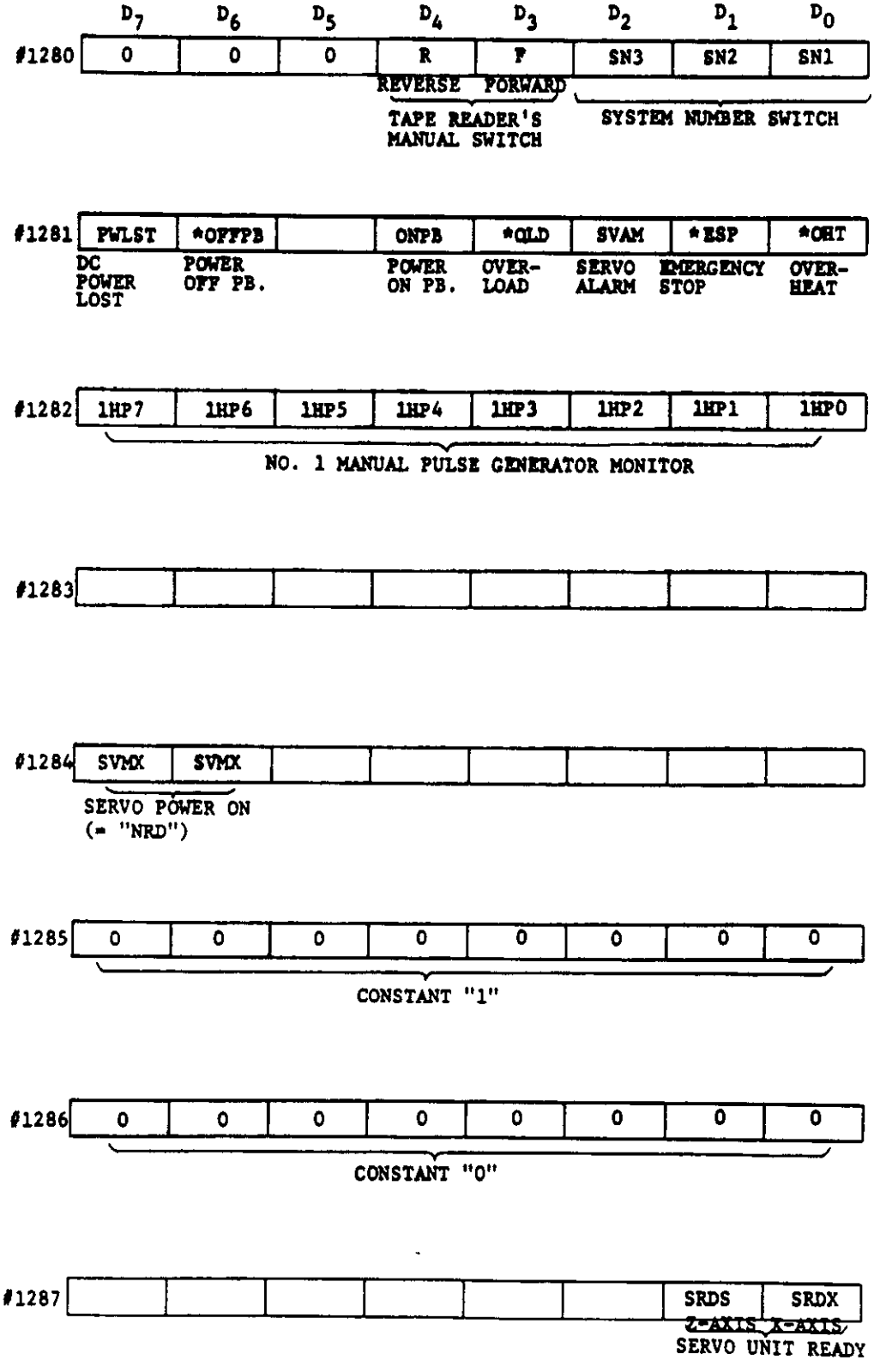
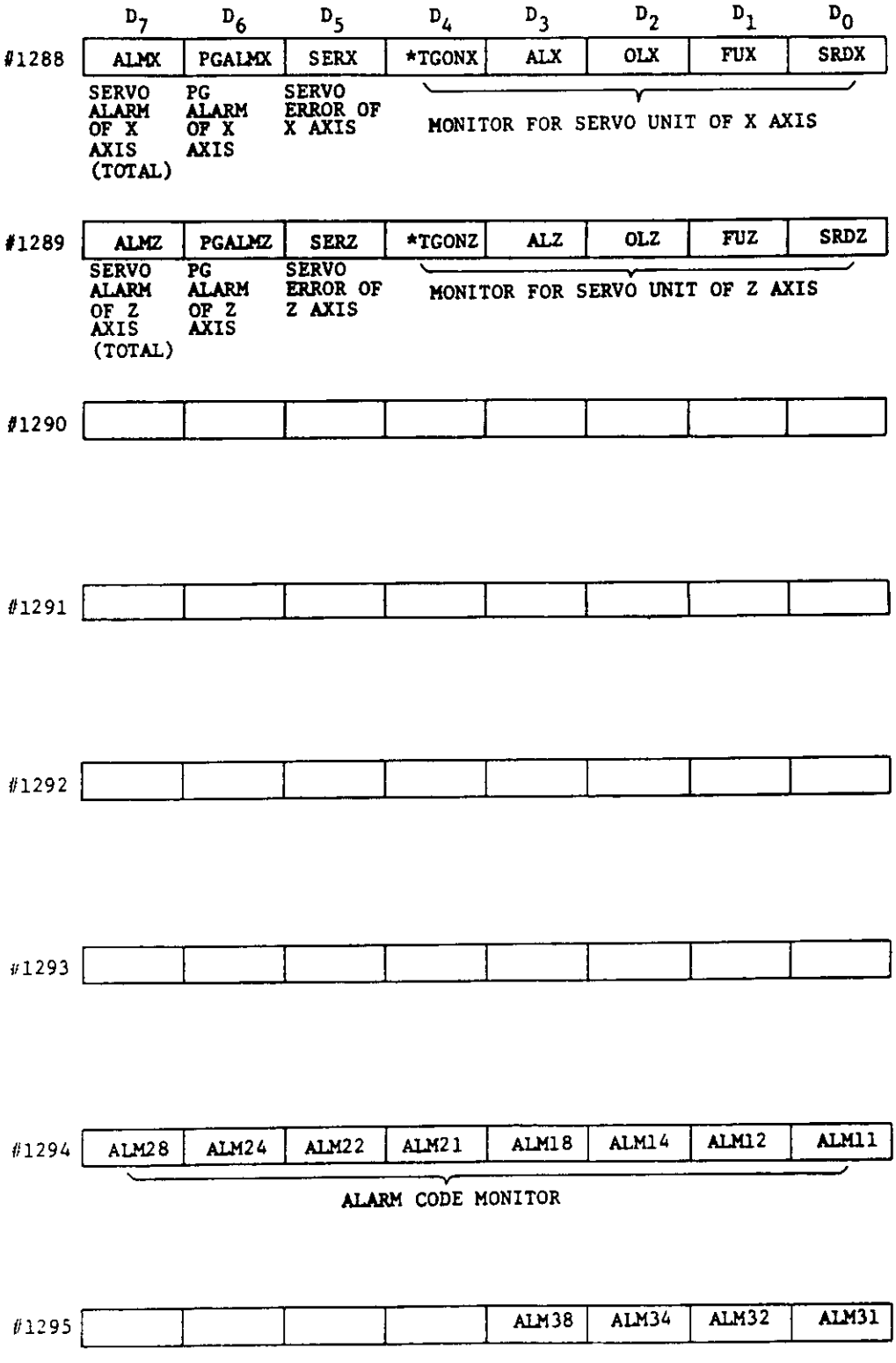


Table 4-1 List of Standard Input/Output Signals (Cont'd)

Output Signals



APPENDIX 5 LIST OF ALARM CODES

Code	Causes	Code	Causes
000		012	OVERFLOW (128CH) BUFFER CAPACITY OVERFLOW IN A BLOCK (128 CHARACTERS).
001	ZR UNREADY (X) REFERENCE POINT RETURN NOT COMPLETED X.	013	PROG ERROR (NO ADDRESS) ADDRESS PLUS NO DATA AND NEXT ADDRESS COMMAND. OR NO ADDRESS PLUS DATA.
002	ZR UNREADY (Z) REFERENCE POINT RETURN NOT COMPLETED Z.	014	PROG ERROR ("-," ".") SIGN "-," AND "," NOT CORRECTLY USED
003		015	PROG ERROR (UNUSABLE CH) UNUSABLE CHARACTER PROGRAMMED IN INSIGNIFICANT DATA AREA
004		016	
005	RESET UNREADY (AFTER EDITING) CYCLE START WITHOUT DEPRESSING RESET BUTTON AFTER EDITING.	017	PROG ERROR (8 DIGITS) INPUT DATA OVERFLOW (MORE THAN 8 CHARACTERS).
006		018	
007		019	
008		020	PROG ERROR (G) UNUSABLE G CODE OR G CODE NOT INCLUDED IN OPTIONS PROGRAMMED.
009		021	PROG ERROR (G) G CODES IN 1, AND * GROUPS PROGRAMMED SIMULTANEOUSLY IN A BLOCK.
010	TH ERROR TAPE HORIZONTAL PARITY ERROR.	022	
011	TV ERROR TAPE VERTICAL PARITY ERROR	023	

Code	Causes	Code	Causes
024	PROG ERROR (G, G41 - 44) UNUSABLE G CODE COMMANDED DURING NOSE RADIUS COMPENSATION	036	PROG ERROR (P-G10) TOO LARGE P (NUMBER DESIGNATION) WHEN OFFSET IS PROGRAM-INPUT
025	RISE ERROR IN NOSE RADIUS COMPENSATION (COMMAND WHICH CANNOT BE ACCOMODATED CORRECTLY IN THE COMPENSATION)	037	PROG ERROR (G10) TOO LARGE R WHEN WORK COORDINATE SYSTEM IS PROGRAM-INPUT.
026	PROG ERROR (G41 - 44) RISE ERROR IN NOSE RADIUS COMPENSATION	038	
027	PROG ERROR (G41 - 44) ERROR DURING NOSE RADIUS COMPENSATION (ERROR IN CIRCULAR INTERPOLATION MODE).	039	
028		040	PROG ERROR (M98, G65/66) P NOT PROGRAMMED IN G65/66 BLOCK. P OR Q NOT PROGRAMMED IN M98 BLOCK.
029		041	NO PROG PROGRAM NO. (SEQUENCE NO.) NOT FOUND WHEN PROGRAM IS CALLED BY M98, M99, G65, G66, G, M, AND T.
030	PROG ERROR (F/E) NO F OR E COMMAND IN FEED COMMAND	042	PROG ERROR (M98, G65/66 NEST) SUBPROGRAM (M98) OR MACRO CALL (G65/G66) FIVE-NESTED.
031	PROG ERROR (R = 0) CIRCLE WITH RADIUS 0 COMMANDED IN CIRCULAR ARC COMMAND	043	
032		044	
033		045	
034	PROG ERROR (G02/03) CIRCULAR ARC R DESIGNATION ERROR	046	
035	PROG ERROR (T OFS) TOO LARGE NO. OF T OFS CODE FOR TOOL RADIUS COMPENSATION AND TOOL LENGTH COMPENSATION	047	

Code	Causes
048	PROG ERROR (G41 - 44) INTERSECTION POINT NOT OBTAINED BY INTERSECTION COMPUTATION.
049	PROG ERROR (G41 - 44) REVERSE OR ALMOST REVERSE COMMANDED IN M97 MODE.
050	PROG ERROR (G11/12) I, K, R NOT CORRECTLY COMMANDED FOR BEVELING AND ROUNDING. VALUES OF I, K, R TOO LARGE.
051	PROG ERROR (G11/12) TAPERING COMMAND IN BLOCKS FOR BEVELING AND ROUNDING.
052	PROG ERROR (G01) ANGLE PROGRAMMING NOT CORRECT DURING ANGLE PROGRAMMING LINEAR INTERPOLATION BY G01.
053	PROG ERROR (G50T/G92T) VALUES OF TOOL COORDINATE MEMORY OUT OF THE RANGE BETWEEN 51 TO 80 IN WORK COORDINATE SYSTEM SETTING BY G50T.
054	
055	PROG ERROR (M, S, T) M, S, T COMMANDS IN THE BLOCK IN WHICH M, S, T CODE CANNOT BE COMMANDED.
056	PROG ERROR (AXIS) AXIS COMMAND IN G20, G21 BLOCKS. AXIS NOT CORRECTLY COMMANDED IN G04, G36-G38.
057	
058	
059	ZR UNREADY G28 NOT COMPLETED ON THE AXIS WHICH HAS G29 COMMAND OR REFERENCE POINT RETURN NOT COMPLETED ON THE AXIS WHICH HAS G30 COMMAND

Code	Causes
060	PROG ERROR (G34) LEAD INCREASE/DECREASE VALUE EXCEEDING MAXIMUM PROGRAMMABLE VALUE DURING VARIABLE LEAD THREAD CUTTING. MINUS VALUE OF LEAD COMMANDED.
061	PROG ERROR (G11/G12 IN THREAD) ROUNDING, BEVELING COMMANDED IN THREAD CUTTING BLOCK.
062	PROG ERROR (G32/G33) THREAD CUTTING COMMANDED IN G98 MODE.
063	PROG ERROR (G92/G78/G21) RAPID THREAD PULL-UP VALUE IN X-AXIS DIRECTION IN THREAD CUTTING WITH BEVELING SMALLER THAN BEVELING VALUE SET BY PARAMETER.
064	PROG ERROR (G92/G78/G21) RAPID THREAD PULL-UP VALUE IN Z-AXIS DIRECTION IN THREAD CUTTING WITH BEVELING VALUE SET BY PARAMETER
065	
066	
067	POWER OFF WHILE EDITING POWER TURNED OFF DURING WRITING MEMORY.
068	EDITING INHIBIT AREA EDITING BEING EXECUTED IN THE EDIT INHIBIT AREA.
069	
070	PROG ERROR (M02/M30/M99) MEMORY OPERATION COMPLETION COMMAND NOT GIVEN.
071	
072	

Code	Causes
073	
074	
075	RS232C ERROR (BAUD RATE) DISAGREEMENT OF NO. OF BITS AND NO OF BAUD RATES FOR RS232C INTERFACE
076	RS232C ERROR (SIGNAL LEVEL) DATA TRANSMISSION FAILURE THROUGH RS232C INTERFACE.
077	RS232C ERROR (OVER-RUN) 10 CHARACTERS MORE HAVE BEEN READ IN AFTER STOP CODE HAS BEEN TRANSMITTED THROUGH RS232C INTERFACE
078	
079	
080	TOOL SET CMP ERROR T CODE COMMANDED BEFORE G35 BLOCK. G05 COMMANDED IN OR BEFORE G35 BLOCK
081	TOOL SET CMP ERROR ERROR OF PARAMETER SETTING FOR TOOL SET ERROR COMPENSATION (X)
082	TOOL SET CMP ERROR. ERROR OF PARAMETER SETTING FOR TOOL SET ERROR COMPENSATION (Z)
083	TOOL WEAR CMP ERROR COMPENSATION NO. EXCEPT 01 TO 19 DESIGNATED AT TOOL WEAR COMPENSATION.
084	TOOL WEAR CMP ERROR TOOL WEAR COMPENSATION INPUTS WOM, WOP GIVEN SIMULTANEOUSLY.
085	EXTERNAL CMP ERROR MULTIPLICATION FACTOR SET BY PARAMETER EXCEEDING 11 FOR EXTERNAL TOOL COMPENSATION.
086	EXTERNAL CMP ERROR ERROR INPUT TURNED ON DURING EXTERNAL TOOL COMPENSATION.
087	PROG ERROR (G31/G35) TOUCH SWITCH NOT ON WHEN MOTION REACHES END POINT BY SKIP OR TOOL SET ERROR COMPENSATION COMMANDS.
088	
089	
090	PROG ERROR (G70-76/G72-78) P, Q NOT COMMANDED IN G70, 71, 72, 73 BLOCKS.
091	PROG ERROR (G70-76/G72-78) BLOCK OF SEQUENCE NO. SPECIFIED BY P, Q IN G70 NOT FOUND PROG NO. INCLUDING IN G70 BLOCK.
092	PROG ERROR (G70-76/G72-78) NO. OF BLOCKS INCLUDING FINISHED SHAPE PROGRAM SPECIFIED BY P, Q IN G70, G71, G72, AND G73, OVER 46
093	PROG ERROR (G70-76/G72-78) UNABLE G- AND M-CODE IN FINISHED SHAPE PROGRAM SPECIFIED BY P, Q IN G70, G71, G72, AND G73.
094	PROG ERROR (G70-76/G72-78) BEVELING AND ROUNDING COMMANDS AS LAST MOVE COMMAND FOR FINISHED SHAPE PROGRAM SPECIFIED BY P, Q IN G70, G71, G72, AND G73
095	PROG ERROR (G70-76/G72-78) FAULTS IN FINISHED SHAPE PROGRAM SPECIFIED BY P, Q IN G71, G72
096	PROG ERROR (G70-76/G72-78) D (CUTTING FREQUENCY) SPECIFIED BY G73 ZERO OR 128 OR MORE. I, K (ROUGH CUTTING) SPECIFIED BY G73 BOTH ZERO. D, K OF G76 EXCEEDING PROGRAMMABLE RANGE.

Code	Causes	Code	Causes
097	PROG ERROR (G70-76/G72-78) FOUR OR MORE PROCESSING INTERRUPTIONS BY FINISHED SHAPE PROGRAM IN STOCK REMOVAL CYCLE BY G71 R1. OR G72 R1.	109	MACRO ERROR (# NO NOT LEFT) PROHIBITED VARIABLE DESIGNATED AS SUBSTITUTION
098	PROG ERROR (G70-76/G72-78) DATA SPECIFIED BY G70 P, Q NOT REGISTERED IN INTERNAL KEEP MEMORY.	110	MACRO ERROR ([ ] 5 LIMIT) MULTIPLE LAYERS OF PARENTHESES EXCEEDING THE UPPER LIMITS
099	PROG ERROR (G70-76/G72-78)	111	MACRO ERROR (MOVE G66-M99) MOVE COMMAND IN M99 FINISHING COMMAND OF MACRO CALLED BY G66
100	CAL ERROR (FIXED POINT) MAGNITUDE OF FIXED POINT DATA EXCEEDING UPPER LIMIT	112	MACRO ERROR MULTIPLE LEVELS OF MACRO CALL EXCEEDING THE UPPER LIMIT 4
101	CAL ERROR (FLOATING) EXPONENT OF FLOATING POINT DATA EXCEEDING ALLOWABLE RANGE	113	
102	CAL ERROR (DIVISION) CALCULATION DIVISOR ZERO OR OVERFLOW ERROR	114	MACRO ERROR (DO-FORMAT)
103	CAL ERROR (SQUARE ROOT) ROOT VALUE IS A NEGATIVE $\sqrt{(-)}$	115	MACRO ERROR ([ ] UNMATCH)
104	PROG ERROR (DOUBLE ADR) CHARACTER WHICH CANNOT BE REPEATED IN A BLOCK COMMAND IN REPETITION	116	MACRO ERROR (DO END NO.)
105	MACRO ERROR (CONSTANT) CONSTANTS EXCEEDING THE LIMIT.	117	
106	MACRO ERROR TOO MANY CODES FOR CANCELLING G67.	118	MACRO ERROR (GO TO N)
107	MACRO ERROR (FORMAT) ERROR IN THE FORMAT EXCEPT FOR EQUATION	119	
108	MACRO ERROR (UNDEFIN #NO.) UNDEFINED VARIABLE NO. DESIGNATED	120	PRTN ERROR (NOT FOUND) SEQUENCE NO. SEARCHED NOT FOUND IN PART PROGRAM

Code	Causes	Code	Causes
121	PRTN ERROR (G50/G92) G31 COMMANDED DURING PROGRAM RESTART.	133	EXT MESSAGE NO CORRESPONDING ALARM NO. WHEN EXTERNAL ALARM MESSAGE IS CLEARED.
122	PRTN ERROR	134	NO PROG (EXT) NOT FOUND PROGRAM NO. SPECIFIED BY EXTERNAL NO. SEARCH.
123	PRTN ERROR (ORG)	135	EXT DATA ERROR IN DATA GIVEN BY EXTERNAL DATA INPUT.
124	PRTN ERROR (MDI MOVE) AXIS OPERATED BY MDI AFTER PROGRAM RESTART PREPARATION	136	
125		137	
126		138	
127		139	
128		140	PROG ERROR (G111/G112) ERROR IN ADDRESS WORD COMMANDING OF G111 BLOCK.
129		141	PROG ERROR (G111/G112) ANGLE FOR ANGLE PROGRAMMING A, B BY G111 OUT OF RANGE $-360 \leq A, B \leq 360$.
130	EXT DATA DATA ERROR IN A GROUP DATA.	142	PROG ERROR (G111/G112) 1ST BEVELING PORTION OUTSIDE RECTANGLE COMPOSED BY START AND END POINTS OR BETWEEN 45° STRAIGHT LINES OF START TO END POINTS AND END TO START POINTS.
131	EXT MESSAGE NO ALARM NUMBER CORRESPONDING TO EXTERNAL ALARM MESSAGE TO BE CLEARED	143	PROG ERROR (G111/G112) ERROR IN G111 COMMAND BLOCK
132	EXT MESSAGE NO CORRESPONDING ALARM NO. WHEN EXTERNAL ALARM MESSAGE IS CLEARED	144	PROG ERROR (G111/G112) M, S, T COMMANDED IN G111, G112 BLOCK.

Code	Causes	Code	Causes
145	PROG ERROR (G111/G112) ERROR IN COMMANDING ADDRESS WORD FOR G112 BLOCK.	157	
146	PROG ERROR (G111/G112) ERROR IN COMMANDING PROGRAMMED SHAPE FORMED BY G112 BLOCK.	158	
147		159	
148		170	MEM ERROR (OFS) TOOL OFFSET TOTAL CHECK ERROR.
149		171	
150		172	MEM ERROR (SET) SETTING AREA TOTAL CHECK ERROR
151		173	MEM ERROR (PRM) PARAMETER AREA TOTAL CHECK ERROR.
152		174	
153		175	
154		176	
155		177	
156		178	

APPENDIX 5 LIST OF ALARM CODES (CONT'D)

Code	Causes	Code	Causes
179	OVER TEMP PANEL INSIDE TEMPERATURE TOO HIGH.	201	OT (X) OVERTRAVEL X.
180	SEQ ERROR SEQUENCE ERROR (1)	202	OT (Z) OVERTRAVEL Z.
181		203	
182		204	
183		205	
184		206	
185		207	
186		208	
187		209	
188		210	
189		211	S-OT1 (X) STORED STROKE LIMIT FIRST AREA X.
200		212	S-OT1 (Z) STORED STROKE LIMIT FIRST AREA Z.

213	225 S-OT3 (Z) STORED STROKE LIMIT THIRD AREA (OUTSIDE INHIBIT) Z.
214	226
215	227
216	228
217	229
218	230
219	231 ZR ERROR-AREA (X) REFERENCE POINT RETURN AREA ERROR X.
220 S-OT2 (INSIDE) STORED STROKE LIMIT SECOND AREA (INSIDE INHIBIT).	232 ZR ERROR-AREA (Z) REFERENCE POINT RETURN AREA ERROR Z.
221 S-OT2 (X) STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) X.	233
222 S-OT2 (Z) STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) Z.	234
223 S-OT3 (INSIDE) STORED STROKE LIMIT THIRD AREA (OUTSIDE INHIBIT).	235
224 S-OT3 (X) STORED STROKE LIMIT THIRD AREA (OUTSIDE INHIBIT) X.	236

APPENDIX 5 LIST OF ALARM CODES (CONT'D)

Code	Causes	Code	Causes
237		249	
238		270	
239		271	P-SET ERROR (X) P SET ERROR X
240		272	P-SET ERROR (Z) P SET ERROR Z.
241	ZR ERROR-POS (X) REFERENCE POINT RETURN POSITION ERROR X.	273	
242	ZR ERROR-POS (Z) REFERENCE POINT RETURN POSITION ERROR Z.	274	
243		275	
244		276	
245		277	
246		278	
247		279	
248		280	MACH UNREADY MACH RDY OFF.

Code	Causes	Code	Causes
281		313	
282		314	
283		315	
284		316	
285		317	
286		318	
287		319	
288		320	NC UNREADY NC UNREADY P SET UNREADY.
289		321	
310	SERVO OFF SERVO POWER NOT SUPPLIED.	322	
311		323	
312		324	

APPENDIX 5 LIST OF ALARM CODES (CONT'D)

Code	Causes	Code	Causes
325		337	
326		338	
327		339	
328		340	
329		341	SERVO ERROR (X) SERVO ERROR X.
330	EMERGENCY STOP EMERGENCY STOP.	342	SERVO ERROR (Z) SERVO ERROR Z.
331	FUSE (X) FUSE BLOWN X.	343	
332	FUSE (Z) FUSE BLOWN Z.	344	
333		345	
334		346	
335		347	
336		348	

Code	Causes	Code	Causes
349		361	PG ERROR (X) PG ERROR X.
350		362	PG ERROR (Z) PG ERROR Z.
351	OL (X) OVERLOAD (1) X.	363	
352	OL (Z) OVERLOAD (1) Z.	364	
353		365	
354		366	
355		367	
356		368	
357	OL (OTHER) OVERLOAD (2).	369	
358		370	
359		371	FG ERROR (1) FG ERROR 1.
360		372	FG ERROR (2) FG ERROR 2.

APPENDIX 5 LIST OF ALARM CODES (CONT'D)

Code	Causes	Code	Causes
373		385	
374		386	
375		387	
376		388	
377		389	
378		390	
379		391	TG ERROR (X) TG LEAD DISCONNECTION.
380		392	TG ERROR (Z) TG LEAD DISCONNECTION.
381	PRG ERROR PRG ERROR.	393	
382		394	
383		395	
384		396	

Code	Causes	Code	Causes
397		409	
398		810	RAM ERROR RAM CHECK ERROR.
399		811	
400	SEQ ERROR SEQUENCE ERROR (2).	812	
401		813	
402		814	
403		815	
404		816	
405		817	
406		818	
407		819	
408		820	ROM ERROR ROM CHECK ERROR.

APPENDIX 5 LIST OF ALARM CODES (CONT'D)

Code	Causes	Code	Causes
821		833	
822		834	
823		835	
824		836	
825		837	
826		838	
827		839	
828		840	CPU ERROR CPU ERROR (2).
829		841	
830	CPU ERROR CPU ERROR (1).	842	
831		843	
832		844	

Code	Causes
845	
846	
847	
848	
849	
910	TAPE-MEM ERROR MEMORY VERIFYING ERROR (OFF-LINE).
920	TAPE ERROR TAPE READING-IN ERROR (OFF-LINE).

ALARM "095"

X-coordinates differnt between G71 command cutting start point and last block for finished shape program.

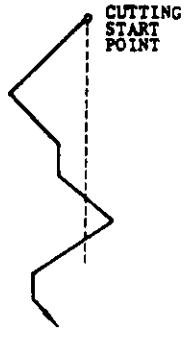
Z-coordinates different between G72 command cutting start point and last block for finished shape program.

Z-coordinate for cutting start point by G71 command different from Z-coordinate for the first block of the finished shape program. (Command G71 ... R1 is excepted.)

X-coordinate for cutting start point by G72 command different from X-coordinate for the first block of the finished shape program. (Command G72 ... R1 is excepted.)

X-coordinate for finished shape program by G71 ... R1 Command exceeding cutting start point.

Z-coordinate for finished shape program by G72 ··· R1 command exceeding cutting start point.



ALARM "140"

- Commanding one or no address of addresses B, X(U), Z(W) specifying second straight line.
- Commanding two addresses of addresses B, X(U), Z(W) specifying second straight line. In addition to this, one or no address commanded among addresses A,I, K,specifying first straight line.
- Address C specifying first beveling and address P specifying first rounding commanded.
- Address D specifying second beveling and address Q specifying second rounding commanded.
- Commanding addresses X and Z specifying second straight line and Q and D specifying second beveling and rounding.

ALARM "143"

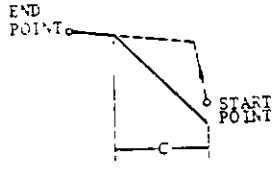
Command values for addresses A, I, K specifying first straight line are determined as follows, and programmed shape cannot be formed.

Command value for A	
-360.000, -180.000 0, 180.000, 360.000	Address I commanded for specifying first straight line.
-270.000, -90.000 90.000, 270.000	Address K commanded for specifying first straight line.

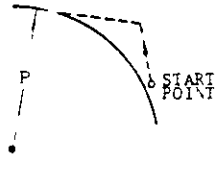
Command values for addresses B, X(U), Z(W) specifying second straight line are determined as follows, and programmed shape cannot be formed.

Command value for B	
-360 000, -180 000 0, 180.000, 360.000	Address X (U) commanded for specifying second straight line.
-270.000, -90 000 90.000, 270 000	Address Z (W) commanded for specifying second straight line.

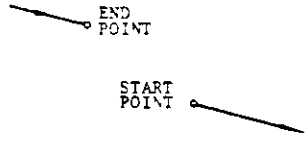
Command values for addresses C and D for beveling too large for the programmed shape. Operation cannot be made according to the command



Command values for address P and Q specifying radius for rounding too large for the programmed shape. Operation cannot be made according to the command



No intersecting point for first straight line and second straight line



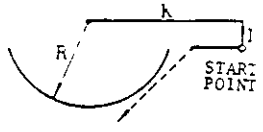
First straight line and second straight line on the same line

ALARM "145"

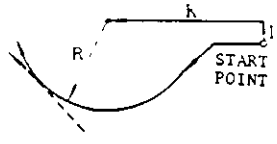
- X(U) or Z(W) not commanded.
  X(U) and Z(W) both commanded.
- R not commanded. Or "0" commanded for R.
- I and K not commanded.
  "0" commanded for I and K.
  P and C both commanded.
- Q and D both commanded

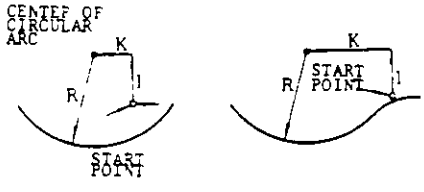
ALARM "146"

Beveling for command C cannot be made

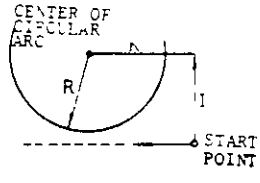


Beveling for command D cannot be made.

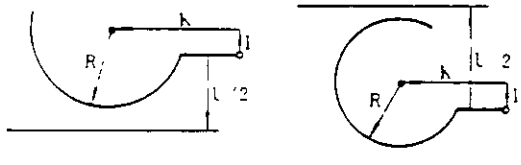




No intersecting point between circular arc and straight line



No intersecing point between circular arc and end point.



APPENDIX 6 LIST OF DATA

Table 6.1 Address Characters

Address	Meaning	B. Basic O Optional
A	Angle designation for G01 and G111, included angle for G76	O
B	Spindle shift angle 01 multiple thread, angle designation for multiple cornering	O
C	User macro character	O
D	Depth of cut and number of cutting cycles for G71 to G76	O
E	Specifications for precise feed and precise lead for cutting	B
F	Specifications for normal feed and normal lead for cutting	B
G	Preparatory function (G-function)	B
H	User macro character	O
I	X-component of arc center, canned cycle parameter, beveling value (radius value)	B, O
J	User macro character	O
K	Z-component of arc center, canned cycle parameter, beveling value	B, O
	Incremental value of variable lead thread	O
L	Number of subprogram repetition, G13 to G16 angle and coordinate	B, O
M	Miscellaneous function (M-function)	B
N	Sequence number	B
O	Program number	B
P	Dwell, canned cycle starting sequence number, program number, user macro number	B, O
Q	Subprogram starting sequence number, canned cycle ending sequence number	B, O
R	Radius of arc, rounding value, tool radius value	B, O
S	Spindle function (S-function), maximum spindle revolution	B
T	Tool function (T-function), tool coordinate memory number	B, O
U	X-axis incremental command value, dwell, canned cycle parameter	B, O
V	User macro character	O
W	Z-axis incremental command value, canned cycle parameter	B, O
X	X-axis coordinate value	B
Y	User macro character	O
Z	Z-axis coordinate value	B

Table 6.2 Function Characters

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

Notes

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

Table 6.3 Tape Code

EIA CODE 8	7	6	5	4	0	3	2	1	CHARACTERS	ISO CODE 8	7	6	5	4	0	3	2	1
		○			●				0			○	○		●			
					●			○	1	○		○	○		●			○
					●		○		2	○		○	○		●		○	
			○		●		○	○	3			○	○		●		○	○
					●	○			4	○		○	○		●	○		
			○		●	○		○	5			○	○		●	○		○
			○		●	○	○		6			○	○		●	○	○	
					●	○	○	○	7	○		○	○		●	○	○	○
				○	●				8	○		○	○	○	●			
			○	○	●			○	9			○	○	○	●			○
		○	○		●			○	a A		○				●			○
		○	○		●		○		b B		○				●		○	
		○	○	○	●		○	○	c C	○	○				●		○	○
		○	○		●	○			d D		○				●	○		
		○	○	○	●	○		○	e E	○	○				●	○		○
		○	○	○	●	○	○		f F	○	○				●	○	○	
		○	○		●	○	○	○	g G		○				●	○	○	○
		○	○		○	●			h H		○			○	●			
		○	○	○	○	●		○	i I	○	○			○	●			○
		○		○	●			○	j J	○	○			○	●		○	
		○		○	●		○		k K		○			○	●		○	○
		○			●		○	○	l L	○	○			○	●	○		
		○		○	●	○			m M		○			○	●	○		○
		○			●	○		○	n N		○			○	●	○	○	
		○			●	○	○		o O	○	○			○	●	○	○	○
		○		○	●	○	○	○	p P		○		○		●			
		○		○	○	●			q Q	○	○		○		●			○
		○			○	●		○	r R	○	○		○		●		○	
			○	○	●		○		s S		○		○		●		○	○
			○		●		○	○	t T	○	○		○		●	○		
			○	○	●	○			u U		○		○		●	○		○
			○		●	○		○	v V		○		○		●	○	○	
			○		●	○	○		w W	○	○		○		●	○	○	○
			○	○	●	○	○	○	x X	○	○		○	○	●			
			○	○	○	●			y Y		○		○	○	●			○
			○		○	●		○	z Z		○		○	○	●		○	
						●			Blank NUL						●			
			○		○	●		○	BS	○				○	●			
			○	○	○	●	○	○	Tab HT					○	●			○
○					●				CR LF/NL					○	●		○	
									— CR	○				○	●	○		○
			○		●				SP	○		○			●			
				○	●		○	○	ER %	○		○			●	○		○
		○	○	○	●	○			UC —									
		○	○	○	●		○		LC —									
*									— (				○		●	○		
*									—)	○			○		●	○		○
*		○	○	○		●			+				○		●		○	○
		○				●			—				○		●	○		○
		○				●	○	○					○	○	●	○		○
			○	○		●		○	/				○	○	●		○	
		○	○	○	○	●	○	○	Del DEL	○	○	○	○	○	●	○	○	○
○	○	○	○	○	○	●	○	○	All Mark	○	○	○	○	○	●	○	○	○
See Note 2									#	○		○			●		○	○
* ○			○	○	○				*	○		○		○	●		○	
* ○				○	●	○			=	○		○	○	○	●	○		○
* ○		○	○		●					○	○		○	○	●		○	○
* ○		○			●		○			○	○		○	○	●	○		
○			○		●	○			$			○			●	○		
○				○	●		○	○	@	○	○				●			
○			○	○	●	○	○		?			○	○	○	●	○	○	○
	○	○		○	●			○	.			○		○	●	○	○	

Notes:

1. For the hole pattern of EIA code of the characters with an asterisk, the pattern shown in the table is standard. However, other patterns may be specified by parameters.
2. EIA code of character # can be designated by the parameter #6017.

Table 6.4 Tape Format

No.	Address	Metric output		Inch output		B: Basic O: Option
		Metric Input	Inch input	Metric Input	Inch input	
1	Program No.	O4		O4		B
2	Sequence No.	N4		N4		B
3	G-Function	G3		G3		B
4	Coordinate Word‡ a: X, Z, I, K, U, W, R	a + 43 (a + 53)	a + 34 (a + 44)	a + 53 (a + 53)	a + 34 (a + 44)	B
5	Feed/min	F50	F32	F50	F42	B
6	Feed/rev and Thread Lead	F32	F24	F42	F24	B
		E34	E26	E44	E26	B
7	S-Function	S2		S2		B
		S4		S4		O
8	T-Function	T(2 + 1)		T(2 + 1)		B
		T(2 + 2)		T(2 + 2)		O
9	M-Function	M3		M3		B
10	Dwell	U(P)53		U(P)53		B
11	Program No. Designation	P4		P4		B
12	Sequence No. Designation	Q(P)4		Q(P)4		B, O
13	No. of Repetitions	L8		L8		B
14	Angle Designation for Straight Line	A(B)33		A(B)33		O
15	Angle Designation for Multiple Thread	B3		B3		O

‡ Parenthesized data indicates maximum cumulative value.

Notes.

1. Inch/Metric output is set by setting parameter #6007D_3.
2. Inch/Metric input is set by setting (#6001D_0).
3. F codes for feedrate/min or feedrate/rev can be switched by G98, G99.

Table 6.5 List of Program Commands

Address		Metric output		Inch output	
		Metric input	Inch input	Metric input	Inch input
Program No. O		1 - 9999		1 - 9999	
Sequence No. N		1 - 9999		1 - 9999	
G function G		0 - 199		0 - 199	
Coordinate Address[1] X, Z, I, K, U, W, R		±8388.607 mm (±99999.999 mm)	±330.2601 in. (±9999.9999 in.)	±21307.061 mm (±99999.999 mm)	±838.8607 in. (±9999.9999 mm)
Feed/min	F	1 - 24000 mm/min	0.01 - 944.88 in/min	1 - 60960 mm/min	0.01 - 2400.00 in/min
Feed/rev and leadscrew	F	0.01 - 500.00 mm/rev	0.0001-19.6850 in/rev	0.01 - 1270.00 mm/rev	0.0001 - 50.0000 in/rev
	E	0.0001 - 500.0000 mm/rev	0.000004 - 19.685000 in/rev	0.0003 - 1270.0000 mm/rev	0.000010 - 50.000000 in/rev
S-function	S2	0 - 99		0 - 99	
	S4	0 - 9999		0 - 9999	
T-function	T3	0 - 99		0 - 99	
	T4	0 - 9999		0 - 9999	
M-function		0 - 999		0 - 999	
Dwell U, P		0.001 - 99999.999 sec		0.001 - 99999.999 sec	
Program No. Designation		1 - 9999		1 - 9999	
Sequence No Designation		1 - 9999		1 - 9999	
No of Repetitions		0 - 99999999		0 - 99999999	
Angle Designation for Straight Line[2]		0 - ±360.000°		0 - ±360.000°	
Angle Designation for Multiple Thread		0 - 360°		0 - 360°	

[1] Parenthesized data indicates maximum cumulative value.

[2] For angle designation of included angle for G76, see 2.8.26.8 Automatic Threading Cycle (G76).

Table 6.6 Data Setting Range

Item			Metric output (screw)		Input output (screw)	
			Metric input	Inch input	Metric input	Inch input
Least input increment			0.001 or 0.01 mm	0.0001 or 0.001 in.	0.001 or 0.01 mm	0.0001 or 0.001 in.
Tool offset			0 - ±8388.607 mm	0 - ±330.2601 in	0 - ±999.999 mm	0 - ±838.8607 in.
Tool radius			0 - ±99.999 mm	0 - ±9.9999 in.	0 - ±99.999 mm	0 - ±9.9999 in
Minimum step/handle feed			0.001 mm	0.0001 in.	0.001 mm	0.0001 in.
Stored stroke limit area designation unit	Program designation		0.001 mm	0.0001 in.	0.001 mm	0.0001 in.
	Parameter & setting		0.001 mm		0.0001 in.	
Rapid traverse rate		Upper limit value	24 m/min		2400 inch/min	
Manual jog						
F0						
2nd reference point coordinate value			0 - ±99999 999 mm		0 - 9999.9999 in.	
Backlash compensation value			0 - 255 pulses Note 1)		0 - 255 pulses	

Notes:

1. 1-pulse = least input increment
2. X-axis designated with diamter (except for pulse display)

Table 6.7-1 List of G Codes

B Basic
O Optional

G Code	Special G Code I	Special G Code II	Group	Function		Section
G00	G00	G00	01	Positioning (rapid traverse feed)		B
G01	G01	G01		Linear interpolation, angle programming for linear interpolation		B, O
G02	G02	G02		Circular interpolation CW,(radius R designation)		B, O
G03	G03	G03		Circular interpolation CCW, (radius R designation)		B, O
G04	G04	G04	*	Dwell		B
G06	G06	G06		ERROR DETECT OFF positioning		B
G10	G10	G10		Tool offset value setup		O
G11	G11	G11	01	Beveling	Cornering	O
G12	G12	G12		Rounding		O
G20	G20	G70	05	Inch input specification		O
G21	G21	G71		Metric input specification		O
G22	G22	G22	01	Radius programming for circular interpolation CW		O
G23	G23	G23		Radius programming for circular interpolation CCW		O
G27	G27	G27	*	Reference point return check		B
G28	G28	G28		Automatic return to reference point		B
G29	G29	G29		Return from reference point		B
G30	G30	G30		Return to 2nd reference point		O
G31	G31	G31		Skip function		O
G32	G33	G33	01	Threadcutting, continuous threadcutting, multi-start threadcutting		B, O
G34	G34	G34		Variable lead threadcutting		O
G35	G35	G35	*	Tool set error compensation		O
G36	G36	G36	07	Stored stroke limit 2nd area ON		O
G37	G37	G37		Stored stroke limit 2nd area OFF		O
G38	G38	G38	08	Stored stroke limit 3rd area ON		O
G39	G39	G39		Stored stroke limit 3rd area OFF		O
G40	G40	G40	06	Tool radius compensation cancel		O
G41	G41	G41		Tool radius compensation No. 1		O
G42	G42	G42		Tool radius compensation No. 2		O
G43	G43	G43		Tool radius compensation No 3		O
G44	G44	G44		Tool radius compensation No 4		O

Notes:

1. G codes in groups from 01 through 11 are modal. When the control is energized with the power switch or reset, the G codes marked with ◤ are automatically selected.

   For G00/G01, G98/G99, and G90/G91, either one is selected as initial state by setting parameters.

2. G codes of * group are non-modal. They should not be commanded together with the other G codes in one block.

3. The modal G codes can be commanded mixedly in a block.

4. G codes in section B are basic.

5. Standard G codes can be converted to special G codes I by parameters. (basic feature)

6. Special G code II can be selected as optional function. When selected, the standard G codes and special G code II cannot be used.

7. The initial states of G codes of 05, 07, 08 group when the control is powered correspond to their respective setting data.

Table 6.7-2 List of G Codes (Cont'd)

B Basic
O Optional

G Code	Special G Code I	Special G Code II	Group	Function		Section
G50	G92	G92	*	Coordinate system setup		B
				Maximum spindle revolution setup, work coordinate system setup		O
G51	G51	G51	*	Return of current display value to origin		O
G65	G65	G65		User macro simple call		O
G66	G66	G66	09	User macro modal call		O
G67	G67	G67		User macro modal call cancel		O
G68	G68	G68	10	Mirror image by programming ON		O
G69	G69	G69		Mirror image by programming OFF		O
G70	G70	G72	*	Finishing cycle	Multiple repetitive cycles	O
G71	G71	G73		Stock removal in turning		O
G72	G72	G74		Stock removal in facing		O
G73	G73	G75		Pattern repeating		O
G74	G74	G76		Peck drilling in Z-axis		O
G75	G75	G77		Grooving in X-axis		O
G76	G76	G78		Automatic threadcutting cycle		O
G90	G77	G20	01	Turning cycle A		B
G92	G78	G21		Threading cycle		B
G94	G79	G24		Facing cycle B		B
G96	G96	G96	02	Constant surface speed control		O
G97	G97	G97		Constant surface speed control cancel		O
G98	G94	G94	04	Feed per minute (mm/min)		B
G99	G95	G95		Feed per revolution (mm/rev)		B
	G90	G90	03	Absolute command		B
	G91	G91		Incremental command		B
G122	G122	G122	11	Tool registration start	Tool life control	O
G123	G123	G123		Tool registration end		O
G111	G111	G111	*	Taper multiple beveling/rounding		O
G112	G112	G112		Arc multiple beveling/rounding		O

1. G codes in groups from 01 through 11 are modal. When the control is energized with the power switch or reset, the G codes marked with ◤ are automatically selected.

   For G00/G01, G98/G99, and G90/G91, either one is selected as initial state by setting parameters.

2. G codes of * group are non-modal. They should not be commanded together with the other G codes in one block.

3. The modal G codes can be commanded mixedly in a block.

4. G codes in section B are basic.

5. Standard G codes can be converted to special G codes I by parameters. (basic feature)

6. Special G code II can be selected as optional function. When selected, the standard G codes and special G code II cannot be used.

7. The initial states of G codes of 05, 07, 08 group when the control is powered correspond to their respective setting data.

YASNAC LXI OPERATOR'S MANUAL

A Better Tomorrow for Industry through Automation

YASKAWA Electric Mfg. Co., Ltd.

TOKYO OFFICE Ohtemachi Bldg Chiyoda-ku Tokyo 100 Japan
Phone (03) 284 9111 Telex 222 2273 YASKWA J Fax (03) 284 9034

YASKAWA ELECTRIC AMERICA, INC SUBSIDIARY
Chicago Office (YASNAC America)
305 Era Drive Northbrook Illinois 60062 U S A
Phone (312) 564 0770 Telex (230) 270197 YSKW YSNC NBRK Fax (312) 564 3276

Los Angeles Office
14811 Myford Road Tustin California 92680 U S A
Phone (714) 731 6841 Telex (230) 678396 YASKAWAUS TSTN Fax (714) 730 8294

YASKAWA ELECTRIC EUROPE GmbH SUBSIDIARY
Monschauerstrasse 1 4000 Dusseldorf 11 West Germany
Phone (0211) 501127 Telex (41) 8588673 YASD D Fax (0211) 507737

Printed in Japan June 1983 83-2 0.8TA ②