TOE-C843-5 2
June, 1981

YASNAC™ 2000G

OPERATOR'S MANUAL

YASKAWA Electric Mfg. Co., Ltd.

PREFACE

This manual is primarily intended to give operators the descriptions for programming, operation and maintenance for YASNAC 2000G.

The descriptions in this manual are given as to the basic functions and the optional functions of YASNAC 2000G. The optional functions are marked with †. For the specifications for your YASNAC 2000G, refer to the manual provided by the machine-tool builder.

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

- Code: EIA code is employed.
- Least Command Increment: Specification A
- Least Input Increment: 0.01 mm
- Feed Function Selection: G99 (mm/rev)
- Absolute Zero Point:
- Fixed Original Point (Zero Return Position):
- Dimensions: in MM

TABLE OF CONTENTS

1. INTRODUCTION

YASNAC 2000G is a microprocessor-based CNC specifically designed for turning applications. It incorporates the latest microelectronics technology in every design feature, which remarkably upgrades the basic functions and widens the scope of optional features of YASNAC 2000G.

The improved transistorized PWM servos combined with the optimum pulse distribution control offer quick response and higher accuracy of machining.

The totally-enclosed, dustproof enclosure protects all components from the attack by rugged industrial environment. This also assures greater control reliability.

On-line diagnostics of YASNAC 2000G speeds the detection of the source of trouble and drastically reduces the downtime.

2. PROGRAMMING

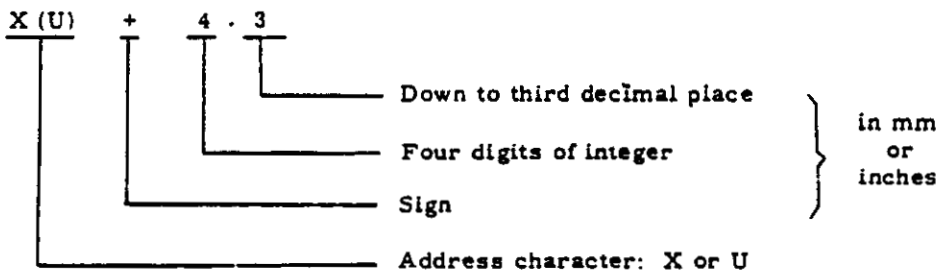
2.1 TAPE FORMAT

2.1.1 TAPE FORMAT

A variable block format conforming to JIS# B 6313 is used for YASNAC 2000G.

Table 2.1.1 shows the tape format. Numerals following the address characters in Table 2.1.1 indicate the programmable number of digits.

EXAMPLE



Note: A decimal point must be omitted in actual 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 an asterisk (*). In actual programming, CR (EIA code) or LF/NL (ISO code) should be used instead of the asterisk (*).

Japanese Industrial Standard

Table 2.1.1 Tape Format

Items \ Least input increment		Metric		Inch†	
		0.01 mm	0.001 mm	0.001 inch	0.0001 inch
Sequence Number		N4		N4	
Preparatory Function		G2		G2	
Move Command	X	X(U)+4.2	X(U)+4.3	X(U)+3.3	X(U)+3.4
	Z	Z(W)+4.2	Z(W)+4.3	Z(W)+3.3	Z(W)+3.4
Arc Radius Designation	X	I + 4.2	I + 4.3	I + 3.3	I + 3.4
	Z	K + 4.2	K + 4.3	K + 3.3	K + 3.4
Feedrate	mm/rev inch/rev	F2.2 F2.3¯		F1.4	
	mm/min inch/min	F4		F3.2	
Threadcutting	Normal	F2.2 F2.3¯		F1.4	
	Precise	E2.4		E1.6	
Tool Function		T2 T4¯		T2 T4¯	
Spindle-Speed Function		S2 S4¯		S2 S4¯	
Miscellaneous Function		M2		M2	
Dwell		U3.3	U3.3	U3.3	U3.3
Radius Designation for G22, G23¯		R + 4.2	R + 4.3	R + 3.3	R + 3.4
Multiple Repetitive Cycles+	Sequence Number	P4		P4	
		O4		O4	
	Depth of Cut, Retracting Value	D + 4.2	D + 4.3	D + 3.3	D + 3.4
	Number of Cutting Cycles	D3		D3	
	Angle of Thread	A2		A2	
Repetition of Subprogram¯		L4		L4	
End of Block		*		*	

Notes:

1. Functions with ¯ are optional.
2. Only the numbers of digits are shown in the above table. Some of them cannot exceed the maximum programmable value. Refer to the description of each item.
3. Least input increment (0.01/0.001 mm) is switched by parameter setting.
4. Feedrate is designated by either mm (inch) per minute or mm (inch) per revolution, which can be switched by G code. (G98, G99)
5. When F2.3¯ is used for F code, E2.4 cannot be used.
6. Inch input is available for the control equipped with an optional Inch/Metric selection. Inch/Metric is switched by parameter setting.

2.1.2 ADDRESS AND FUNCTION CHARACTERS

Address characters and the meanings are shown in Table 2.1.2.1.

Function characters and the meanings are shown in Table 2.1.2.2.

Table 2.1.2.1 Address Characters

B: Basic
O: Optional

Address Characters	Meanings	Section
A	Angle of thread for multiple repetitive cycle G76	O
B	Unused	-
C	Unused	-
D	Depth of cutting for multiple repetitive cycles G70 to G76	O
E	Precise lead designation for threadcutting	B
F	Feed function and lead designation for threadcutting	B
G	Preparatory function	B
H	Unused	-
I	X-coordinate of arc center (radius value), Parameters of multi repetitive cycles, Amount of rounding	B O
J	Unused	-
K	Z-coordinate of arc center, Parameters of multiple repetitive cycles, Amount of rounding	B O
L	Number of repetitions of subprogram	O
M	Miscellaneous functions	B
N	Sequence number	B
O	Unused	-
P	Sequence number for start of multiple repetitive cycle, Sequence number designation for jump	O
Q	Sequence number for finish of multiple repetitive cycle	O
R	Radius designation of a circular arc for G22 and G23	O
S	Spindle-speed function	B
T	Tool function	B
U	Incremental X-coordinate, Dwell time	B
V	Unused	-
W	Incremental Z-coordinate	B
X	Absolute X-coordinate	B
Y	Unused	-
Z	Absolute Z-coordinate	B

Note: Any address characters not defined above cause errors in significant data area if they are programmed.

Table 2.1.2.2 Function Characters

EIA	ISO	Meanings	Remarks
Blank	NuL	Error in significant data area of EIA Disregarded in ISO	
BS	BS	Disregarded	
Tab	HT	Disregarded	
CR	LF/NL	End of block	
/	CR	Disregarded	
SP	SP	Disregarded	
ER	%	Rewind stop	
UC	/	Disregarded	
LC	/	Disregarded	
/	(	Control out	
/	)	Control in	
+	+	Disregarded	
-	-	Negative sign	
0 to 9	0 to 9	Numerals	
a to z	A to Z	Address characters	
0	:	Regarded as N	Usually not used
/	/	Optional block skip	
Del	DEL	Disregarded (Including All Mark)	

Notes:

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

2.1.3 BUFFER REGISTER

While the control operates with the data in active register, the next block of data is read out from the tape and stored into buffer register. By this means, the tape reading operation does not cause the loss time in the machining operation.

Buffer capacity is 64 characters including EOB. A command of block exceeding 64 characters causes errors and the alarm code "14" is displayed. The following characters are not read into buffer register and are not restricted by buffer capacity.

- Disregarded codes (SPACE, TAB, ALL MARK, etc.)
- Insignificant data from the start to the first EOB at the Label Skip state.
- Control Out, Control In and codes between them.

NOTE: During tip nose radius compensation, two or three blocks are read ahead. In this case, each buffer capacity is the same as the above.

2.1.4 TV CHECK (TAPE VERTICAL PARITY CHECK)

Number of characters in one block must be even in TV check. SP code is used for making the number even when programming.

ON/OFF switching of TV check is set with parameter. A block containing odd number of characters causes error and the alarm code "13" is displayed. (Refer to the 4.3.15 TV Check.)

2.1.5 OPTIONAL BLOCK SKIP ("/" CODE)

A block with "/" before the address N for sequence number is skipped at OPTIONAL BLOCK SKIP switch on. "/" after the address N is read into buffer register but is ignored.

2.2 SEQUENCE NUMBER

2.2.1 4-DIGIT SEQUENCE NUMBER

Sequence number is a reference number for the block and does not affect the machining operation and order. Therefore, sequential numbers, discontinuous numbers, overlapped numbers and no number are acceptable. But it is recommendable to use sequential numbers.

Sequence number is represented by four digit-integers from 0001 to 9999 with the proceeding N. The leading zeros can be suppressed.

EXAMPLE

N1, N01, N001, N0001 ---- All correct

NOTE: When a number of 5 digits or over is given as a sequence number, the latter 4 digits are effective. When address search is made for overlapped sequence number, address search stops after reading out the block first searched. The block without sequence number can be searched by searching the address data in the block.

2.2.2 3-DIGIT SEQUENCE NUMBER (SPECIAL SPECIFICATION)

When Workpiece Number Designation option is provided, three digits are used for sequence number. The forth digit is used for designation of workpiece.

2.3 MOVE COMMAND

2.3.1 LEAST COMMAND INCREMENT

The least command increment is the minimum unit of movement in which the machine can move and is represented in millimeters per pulse. Two types of increment system are available. The system to be used is specified according to the machine. The control is specified by one of them which depends upon the machine. Refer to the machine tool builder's manual.

Table 2.3.1 Least Command Increment

	Specification A	Specification B
X axis	0.001 mm/pulse	0.0005 mm/pulse
Z axis	0.002 mm/pulse	0.001 mm/pulse

Notes:

1. X axis is shown by radius value.
2. Inch/Metric selection is effective for the above metric based machine.

2.3.2 LEAST INPUT INCREMENT

The least input increment is the minimum unit that can be programmed and is represented in millimeters or in inches†. Values for X axis must be designated by diameter.

Table 2.3.2 Least Input Increment

	Metric		Inch‡	
X and Z axes	0.01 mm	0.001 mm	0.001 inch	0.0001 inch

Switching of 0.01 mm/0.001 mm (or 0.001 inch/0.0001 inch‡) is made by parameter setting (Parameter No. 88). When the contents of parameter No. 88 is "0," programming can be made in the unit of 0.001 mm (or 0.0001 inch‾) and when "1," in the unit of 0.01 mm (or 0.001 inch‾). However, the tool offset value can be given in 0.001 mm (or 0.0001 inch‾) regardless of the content of parameter No. 88.

For example, 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:

- Note that if NC tape programmed in 0.001 mm is stored in the control set to 0.01 mm incremental system, the machine will move by the ten times of intended dimensions.
- Note that if the increment system is switched under the state that the contents of NC tape is stored in memory‾, the machine will move by the ten times or one tenth of the commanded dimensions.
- When the stored program is punched out on the tape‾, the stored figures are punched out regardless of the switching of increment system.
- If 0.001 mm increment is commanded to Z axis whose least command increment is 0.002 mm per pulse, the machine will not move. However, since the command value is retained in the control, the machine will move at the next command of 0.001 mm.

2.3.3 ABSOLUTE AND INCREMENTAL INPUTS

Both of 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··· *

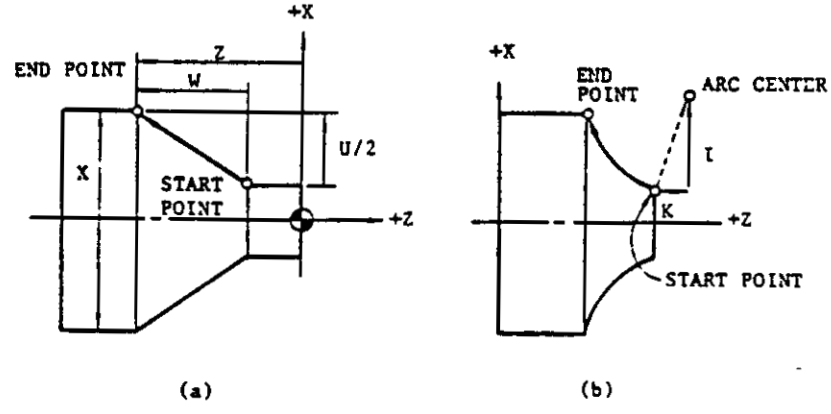
NOTE: When addresses X and U or addresses Z and W are used in one block, the latter is effective.

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

Table 2.3.3

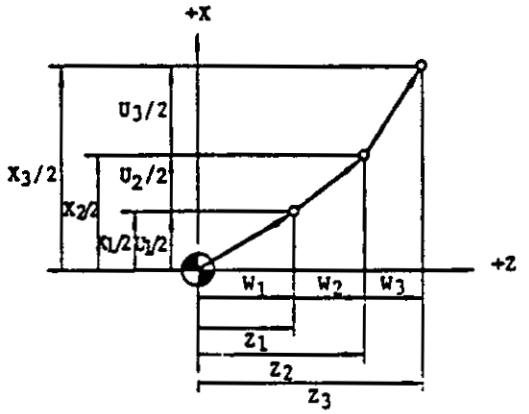
Address	Increment System	Designation	Meaning
X	Absolute Input	Diameter	Position in X axis direction (Note)
Z		-	Position in Z axis direction
U	Incremental Input	Diameter	Move amount in X axis direction (Note)
W		-	Move amount in Z axis direction
I	Incremental Input	Radius	Distance in X axis direction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

Note: Designation of addresses X and U depends upon the option. Refer to 2.3.4 "DIAMETER/RADIUS Switching."



(a) (b)

Fig. 2.3.3.1



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. 2.3.3.2 Absolute Coordinate Values and Incremental Coordinate Values

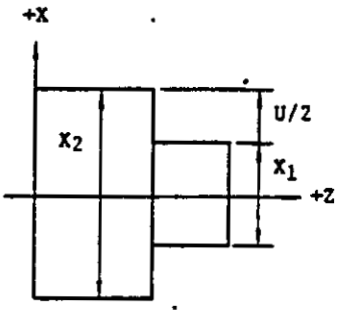
2.3.4 DIAMETER/RADIUS SWITCHING

When the control is equipped with DIAMETER/RADIUS switching option, the addresses X and U can be used for designation of both diameter and radius. The switching is made by the setting of parameter No. 70.

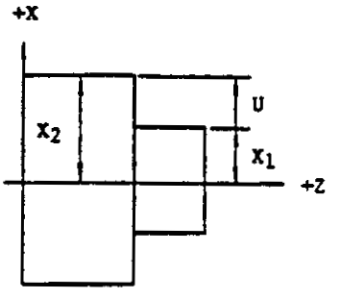
Contents of Parameter No. 70

0: Diameter designation } Effective only for the addresses X and U

1: Radius designation }



(a) In the case of Diameter Designation



(b) In the case of Radius Designation

Fig. 2.3.4

Addresses Z, W, I and K are not affected by the switching function.

2.3.5 MAXIMUM PROGRAMMABLE VALUE

Maximum programmable value of move command is shown below.

Table 2.3.5 Maximum Programmable Values (Addresses X, Z, U, W, I and K)

Least Command Increment / Least Input Increment		X axis: 0.001 mm/pulse Z axis: 0.002 mm/pulse (Specification A)	X axis: 0.0005 mm/pulse Z axis: 0.001 mm/pulse (Specification B)
Metric	0.01 mm	±8388.60 mm	±4194.30 mm
	0.001 mm	±8388.607 mm	±4194.303 mm
Inch	0.001 inch	±330.260 inch	±165.130 inch
	0.0001 inch	±330.2601 inch	±165.1300 inch

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

In absolute programming, input values and move amount of each axis specified by the inputs must not exceed the maximum programmable value.

The machine may not properly operate if the move command over the maximum programming value is given.

2.4 RAPID TRAVERSE RATE

2.4.1 RAPID TRAVERSE RATE

Each axis moves at the rapid traverse rate when G00 (positioning) is commanded or RAPID mode (manual rapid traverse) is selected.

Two rates of rapid traverse Lo and Hi are available, and are set with parameter No. 93 and 94. An operator can switch Hi/Lo by means of setting functions (Parameter No. 01).

The contents of parameter No. 01:

0: Rapid traverse rate Hi

1: Rapid traverse rate Lo

Rapid traverse rate Lo is used to prevent an accidental collision when the Single Block operation or the dry run is performed for the test of newly punched tape.

2.4.2 RANGE OF RAPID TRAVERSE RATE

Rapid traverse rates Hi and Lo are set for each axis independently in the following step.

Step of rapid traverse rate (Specification A of increment system)	7.5 mm/min

Note: Half of the above value is available for the control with the specification B.

The maximum traverse rate is 100 kPPS. However, since the maximum traverse rate depends upon the servo motor and the machine, it is set suitably according to the machine. Refer to the machine tool builder's manual.

2.5 CUTTING FEED

2.5.1 FEED FUNCTION A (F-FUNCTION A)

G code of D group must be designated before F function is commanded.

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

Note: For the details, refer to 2.10.17 Feed Function Designation.

Since F code is modal, the code is effective until the next F code is commanded. However, when G98/G99 is switched over, F code must be designated again.

- Feedrate per revolution (G99)

  After the designation of G99, the feedrate of tool per spindle revolution can be commanded by 4 digits following F. Table 2.5.1.1 shows the programmable range of the F code.

Table 2.5.1.1 F code (G99)

	Format	Range of Feedrate	Meaning
Metric	F2.2	F1 to F9999	0.01 to 99.99 mm/rev.
Inch†	F1.4	F1 to F39366	0.0001 to 3.9366 inch/rev.

However, the programming of feedrate is restricted by the spindle speed as shown below

$$F\ (\text{mm/rev}) \times S\ (\text{rpm}) \leqq 4800\ (\text{mm/min})$$

Note: In the case of specification B of incremental system, the restriction is the half of the above value.

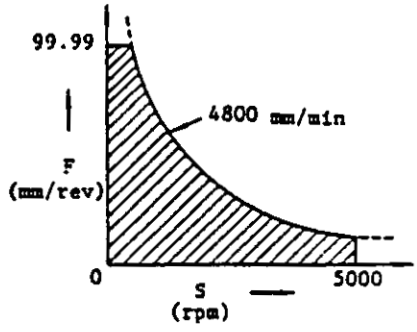


Fig. 2.5.1.1 Restriction of Feedrate and Spindle Speed

· Feedrate per minute (G98)

The feedrate of tool per minute can be commanded by 4 digits following the address F after the designation of G98. Table 2.5.1.2 shows the programmable range of F code.

Table 2.5.1.2 F code (G98)

	Format	Range of Feedrate	Meanings
Metric	F4	F1 to F5400	1 to 5400 mm/min
Inch†	F3.2	F1 to F21259	0.01 to 212.59 inch/min

Notes:

1. In the case of specification B of incremental system, the upper limit of feedrate is the half of the above values.
2. The upper limit of feedrate may be restricted by the servo and mechanical systems. Refer to the machine tool builder's manual.

NOTES:

· A command "F0" causes data errors and the alarm code "15" will be displayed.

· Any minus value should not be specified for F commands. If specified, the machine will not operate properly.

EXAMPLE

F-250 * . . . . Wrong

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

EXAMPLE 1

G99 S350 (rpm) *

G01 U10000 F200 *

In the above case, the feedrate is:

F x S = 2.0 mm/rev. x 350 rpm
= 700 mm/min

EXAMPLE 2

G98

G01 X20000 F700 *

In the above case, the feedrate is:

F = 700 mm/min

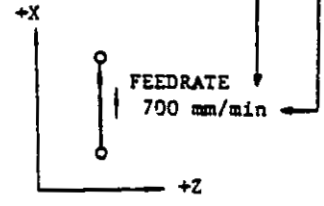


Fig. 2.5.1.2

· 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 the above case, the feedrate is:

F = 500 (mm/min)

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

Z axis component

X axis component

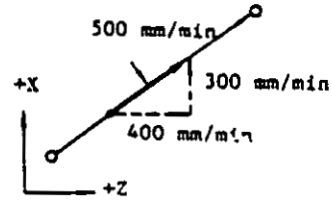


Fig. 2.5.1.3 (a)

EXAMPLE 2

G99 S1000 (rpm) *

G03 U . . . W. . . I. . . F20 *

In the above case, the feedrate is:

$$F \times S = \underset{(mm/rev)}{0.2} \times \underset{(rpm)}{1000}$$

$$= 200 \text{ mm/min}$$

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

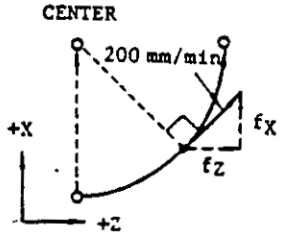


Fig. 2.5.1.3 (b)

2.5.2 FEED FUNCTION B (F-FUNCTION)

Feedrate per revolution in metric system is commanded by the format of F2.3. The other functions are the same as those of F-function A.

Table 2.5.2 F code (G99 mode)

	Format	Range of Feedrate	Meaning
Metric	F2.3	F1 to F99999	0.001 to 99.999 mm/rev.

Note: No change in inch

E code cannot be used for the control with F-function B.

2.6 AUTOMATIC ACCELERATION AND DECELERATION

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

2.6.1 ACCELERATION AND DECELERATION OF RAPID TRAVERSE AND MANUAL FEED

In the following operation, the pattern of automatic acceleration and deceleration is linear. (See Fig. 2.6.1.)

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

"Automatic operation" means operation in TAPE, MDI or MEM mode in this manual.

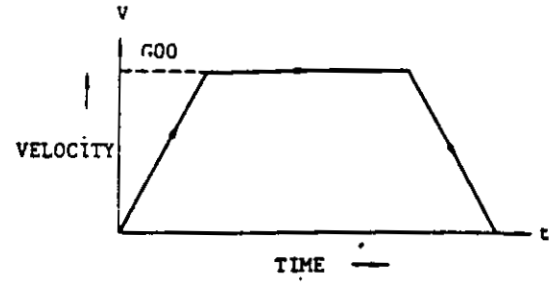


Fig. 2.6.1

2.6.2 ACCELERATION AND DECELERATION OF CUTTING FEED

In the following operation, the pattern of automatic acceleration and deceleration is of expotential curve. (See Fig. 2.6.2.)

- Cutting feed (G01 to G03)
- Threadcutting (G32)
- Handle (HANDLE mode)

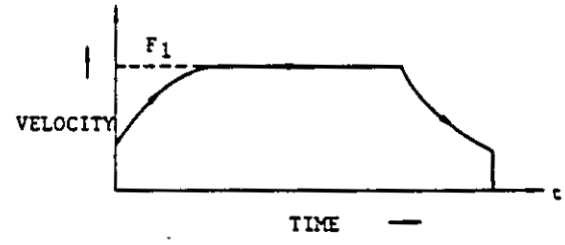


Fig. 2.6.2

2.7 MISCELLANEOUS FUNCTIONS (M-FUNCTION)

The miscellaneous function is specified with the address M and two 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 (Program Stop)

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

- M01 (Optional Stop)

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

· M02 (End-of-Program)

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

· M30 (End-of-Tape)

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

NOTES:

· When M00, M01, M02 or M30 is commanded, it prevents the control from reading ahead the next block of information. The single decode signal is fed in addition to the 2-digit BCD code output for M codes. For the timing of output, refer to the Appendix 1.

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

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

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

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

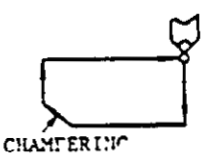
Refer to the machine tool builder's manual and Appendix 1.

2.7.2 M CODES FOR ON/OFF OF FUNCTIONS

· M23/M24 (Chamfering with threadcutting ON/OFF)

M23 . . . Chamfering ON

M24 . . . Chamfering OFF



When M23 is commanded before the command of threadcutting cycle G92 (or G76†), chamfering of thread is performed. If M24 is commanded, chamfering with threadcutting is not performed.

When power supply is turned on, M23 is in effect. For the details, refer to 2.10.15 Canned Cycles.

· M51/M52 (Error Detection ON/OFF‡)

These codes are available for the control provided with the Error Detection ON/OFF function‡. The control without the option operates as follows.

(a) At the end of the execution of G00 (Positioning) command, the control proceeds to the next block after the lag pulse of servo coming within the allowable range.

(b) In the end of the execution of G01, G02 or G03 (Linear and circular interpolations) is effective, the control proceeds to the next block smoothly regardless of the lag pulse of servo.

The control with Error Detection ON/OFF operates as follows.

M51: Error Detection OFF

M52. Error Detection ON

M51 permits the machine to move smoothly between blocks regardless of the lag pulse of servo. When M51 is in effect, the control proceeds to the next block after the completion of the pulse distribution to servo for linear and circular interpolation.

When M52 is in effect, the control proceeds to the next block after the lag pulse of servo comes within the allowable range. M52 is used to avoid rounding of a corner.

G00 (positioning) is always executed at the state of Error Detection ON regardless of the command of M51 or M52.

M51 and M52 are modal. When the power supply is turned on, M51 is in effect.

When M51 or M52 is commanded in a block together with the move command of cutting feed (G01 to G03), it becomes effective at the end of the block containing them.

"Automatic operation" means operation in TAPE, MDI or MEM mode in this manual.

G00 X . . . Z . . . *

① G01 W-3000 F30 M51 *

② (G01) U4000 M52 *

Z . . . *

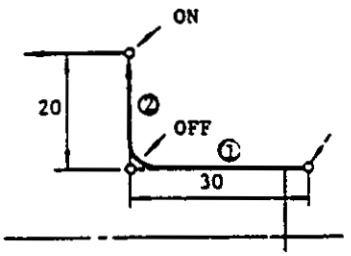


Fig. 2.7.2.1

· M94/M95 (Traverse Rate Clamp ON/OFF)†

These codes can be used when Traverse Rate Clamp option is supplied.

M94: Traverse Rate Clamp ON

M95: Traverse Rate Clamp OFF

After the M94 is commanded, the following feedrates are decelerated to the values set by the parameter while the limit switch for traverse rate clamp is ON.

(a) Rapid traverse (G00)

(b) Cutting feedrate specified by G98 command (Feedrate per minute)

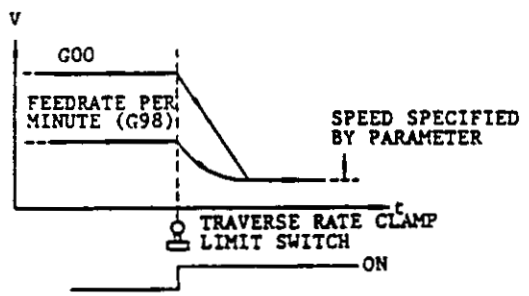


Fig. 2.7.2.2

M94 and M95 are modal. When the power is turned on, M94 is in effect.

When M94 or M95 is commanded together with the move command in a block, it becomes effective at the block containing them.

NOTE: Note that the Traverse Rate Clamp function is not effective in the G99 mode (Feedrate per revolution: mm/rev).

· M98/M99 (Subprogram function)†

These codes are used for jump instruction. For the details, refer to 2.7.4 Subprogram Function.

NOTE: Above M codes (M23/M24, M51/M52, M94/M95, M98/M99) are effective only for the control and give no signals (2-digit BCD output) to the machine.

2.7.3 OTHER M CODES

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

Table 2.7.1 Typical Example of M codes for Machine

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

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

2.7.4 SUBPROGRAM FUNCTION (M98, M99)†

The sequence of operation can be altered by this option which is available for the control supplied with part program storage. The M98 and M99 commands can specify the sequence number of the block to be performed next. (The previous or the following block.)

Jump Instruction: P . . . M99 *

This command jumps to the block of the sequence number specified by the address P. When the P command is omitted, the execution returns to the first address of the memory and the operation will be repeated.

- Jump Instruction to Subprogram:
  P . . . L . . . M98 *

  M98 command is used when the program is required to jump from the executing program (main program) to the other program (subprogram) and must return to the main program after the completion of the subprogram.

  Address P specifies the sequence number of the first block of subprogram.

  Address L specifies the number of repetitions of subprogram. The maximum programmable value of the address L is 9999. After the completion of specified times of execution, the command returns to the main program. The L command must be programmed in each execution of this instruction.

- Return Block Instruction from Subprogram:
  P . . . M99 *

  When an address P is commanded with M99 at the end of subprogram, return block can be designated by the address P. The program returns to the block of main program whose sequence number is designated by the address P.

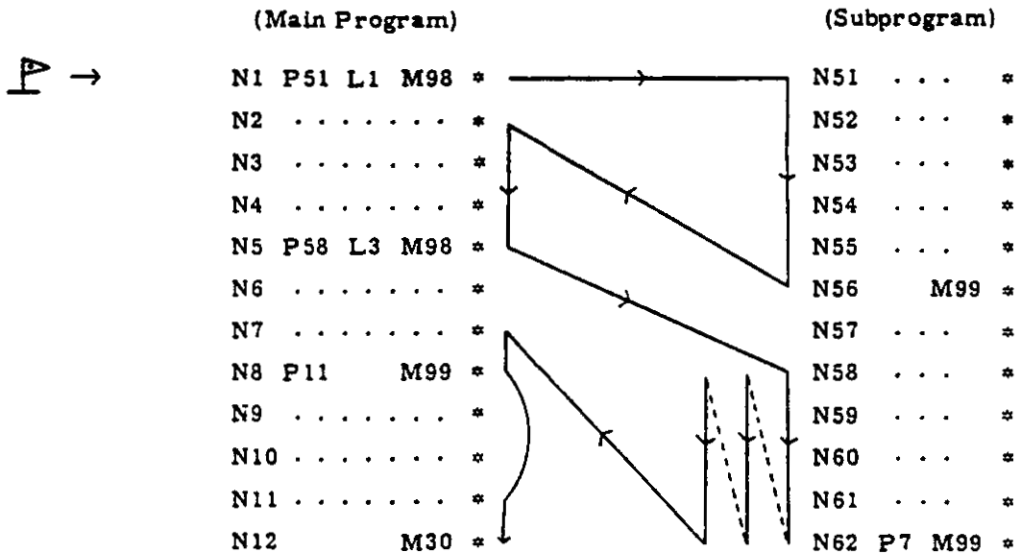


Fig. 2.7.4.1

NOTE:

- "Subprogram" simply means the program specified by M98 command. Therefore, any special consideration is not required for storing and editing of subprogram.
- M98 cannot be used in a subprogram.
- Signals of M98 and M99 are not fed to the machine.
- The sequence number designated by the address P is searched from the first address of memory and the execution of program starts from the first block of the sequence number searched. Therefore, the same sequence number should not be used.
- When the sequence number designated by the address P is not found, it causes error.
- Addresses P and N designate sequence numbers. Up to four-digit number can be used for sequence numbers and leading zeros can be suppressed.
- The sum of the characters of main program and subprogram should not exceed the memory capacity of the control.
- Address L cannot be used together with M99 command.
- The remaining number of repetitions of subprogram is displayed when POS key is depressed and the address L is set on the address indicator during the operation of repeating command P . . . L . . . M98 *.
- Subprogram cannot be executed by writing M98 in MDI mode.
- Endless program can be made by using M99 command in a main program. When the RESET button is depressed, the program returns to the first address of memory. Refer to Fig. 2.7.4.2 (a).

· Some part programs which are frequently used can be stored in advance and executed required times when in need. Refer to Fig. 2.7.4.2 (b). When M codes causing rewind (M02 or M30) is used, the program returns to the initial address. When M codes causing internal reset are commanded, the following program is executed as a main program.

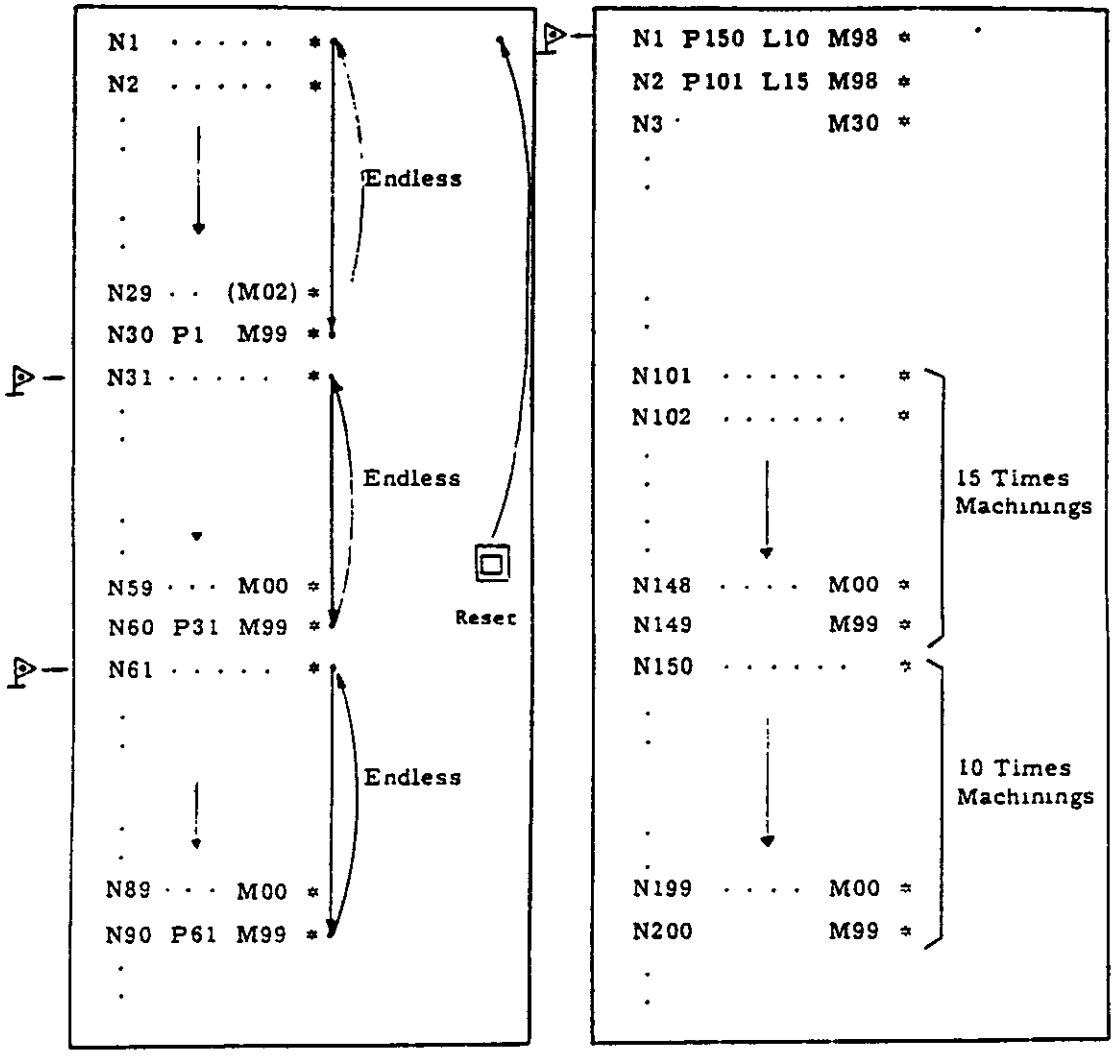


Fig. 2.7.4.2

2.8 SPINDLE-SPEED FUNCTION (S-FUNCTION)

2.8.1 S 2-DIGIT COMMAND

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

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

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

The S command is modal. Although the spindle is actually stopped by the M05 (spindle stop) command, the S command remains effective.

EXAMPLE

```
G50 X20000 Z30000 *
G00 S11 M03 T11 *      . . . S command
    X . . . Z . . . *        Spindle CW
G01 Z . . . F . . . *                      S11: Effective
    .
    .
G00 X20000 Z30000 M05 *    . . . Spindle stop
    T22 M03 *
    X . . . Z . . . *                      S11: Effective
G01 Z . . . F . . . *
    S22 *      . . . Change of S
    U . . . F . . . *                      S22: Effective
    .
    .
    .
```

NOTE: The two-digit BCD output is sent to the machine when S and two-digit command is issued. For the timing of output and the finish-answer-back signal (FIN), refer to the APPENDIX 1.

EXAMPLE

```
S1000 M03 *
```

2.8.2 S 4-DIGIT+ COMMAND

Four digits following S are used to specify the spindle speed in rpm.

The S command becomes effective instantaneously when issued. When the S command and the move command are issued in a block, the move command and the spindle speed command are fed at a time.

When S command is issued in a block together with the M03 (Spindle forward running) or the M04 (reverse running), the spindle speed signal is fed instantaneously, but the control never proceeds to the next block before the completion of the M command. "The completion of the M command" means generally the time when the spindle speed reaches the speed commanded by the S code. Refer to the machine tool builder's manual.

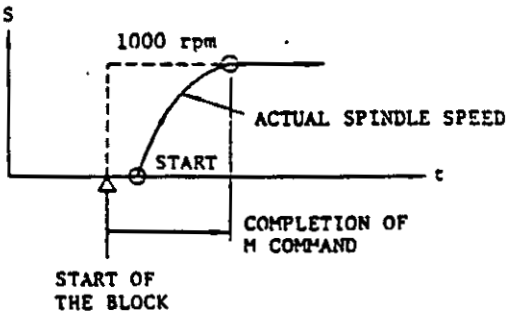


Fig. 2.8.2

S commands are modal. Although the spindle is stopped actually at the M05 command, the S command is retained. Therefore, when the M03 (or M04) is commanded, the spindle runs again according to the S command.

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

NOTES:

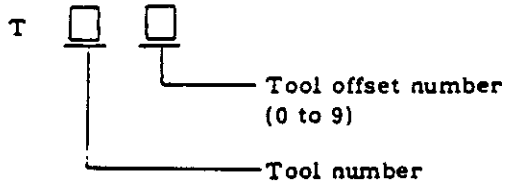
- The lower limit of the spindle speed depends on the spindle drive. If the low speed out of the limit is commanded, the machine will stop. Refer to the machine tool builder's manual for the limit.
- When S and 4 digits are used to command the spindle speed, the output for the spindle drive is given by the either of the following two ways.
  - 12 Bits binary non-contact output
  - Analog D/A converter output (±10 V Max.)
- The S and 4 digits must be commanded in Constant Surface Speed Control. For the details, refer to 2.10.16 Constant Surface Speed Control.

2.9 TOOL FUNCTION (T-FUNCTION)

This function performs tool selection and tool offset.

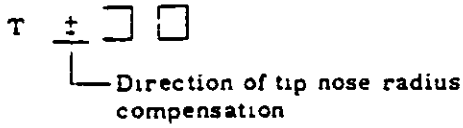
2.9.1 T 2-DIGIT

Two digits following the address T specifies the tool number and the tool offset number.



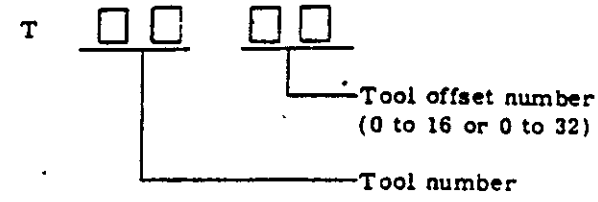
The figures used for the designation of tool number are determined by the machine. Refer to the machine tool builder's manual.

When the Tip Nose Radius Compensation option is provided, the T code must be programmed with sign (+ or -). For the details, refer to 2.10.11. Tip Nose Radius Compensation.



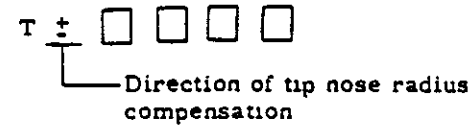
2.9.2 T 4-DIGIT†

Four digits following the address T specifies the tool number and the tool offset number.



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

† When the Tip Nose Radius Compensation option is provided, the T code must be programmed with sign (+ or -). For the details, refer to 2.10.11.



NOTE:

- 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
- Tool offset number 0 or 00 cancels the tool offset.

2.9.3 STORAGE OF TOOL OFFSET VALUES

There are two kinds of tool offsets· tool position offsets and tip nose radius compensation. Each offset value must be stored in memory beforehand.

Table 2.9.3

	Offset Method	G code	Storage of Offset Values	T code
Tool Length Offsets	Offset Value Commanded	Unnecessary	X Z .. No. 1 .. No. 2 .. No. 16	T □ □ 0 2
Tip Nose Radius Compensation†	Full Automatic	G40 G41 G42 G43 G44	R ... No. 1 ... No. 2 ... No. 16	

NOTE: Number of offset values usable are determined by the machine.

T 2-digit: 9 sets

T 4-digit: 16 or 32 sets

2.9.4 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 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.

· Range of tool position offset value

The programmable range of tool offset value is shown below.

Table 2.9.4

Metric	Inch†
0 to ±999.999 mm	0 to ±40.0000 inch

Note: Diameter must be set for the X axis.

· Signs of tool position offset values

The tool position offset value is a difference between the coordinate value of the programmed tool tip and the actual tool tip, and the sign of the offset value specifies the direction of offset motion.

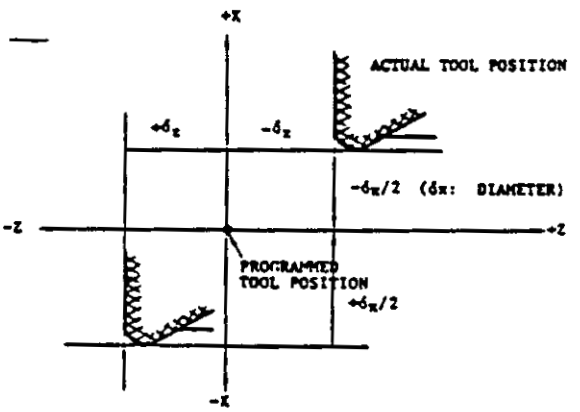


Fig. 2.9.4.1

For the writing of tool offset values, refer to 4.3.6 Writing of Tool Offset Value.

· 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 the amount of the difference between them.

OFFSET VALUE

T101
↓
$(+\delta X_1, +\delta Z_1)$

T115
↓
$(+\delta X_2, +\delta Z_2)$

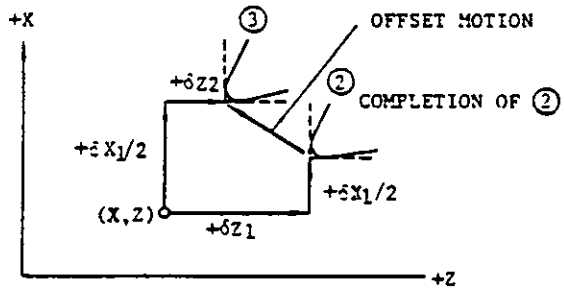


Fig. 2.9.4.2

EXAMPLE

T101 ✲ ..... ①

.
.
.

G01 X · · · Z · · · F · · · ✲ .... ②

T115 ✲ ..... ③

(Block of the offset motion)

· 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 ✲ ←

X · · · Z · · · ✲

.
.
.

Offset motion is made at the rapid traverse rate.

· Instruction of tool position offset

G code is not required to instruct tool position offset but the tool offset number corresponding to the actual tool must be designated.

1. 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 2 is selected. Tool offset motion is made according to the contents of the tool offset number.

2. When the tool offset value is required to change, 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 16 is replaced with 02. Tool offset motion is made at the cutting feedrate.

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

3. 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 taper cutting feedrate command.

EXAMPLE

```
1 G00 T0202 *
  G01 X . . . Z . . . F . . . *
        .
        .
        .
2 G01 U+. . . W-. . . F . . . T0216 *
```

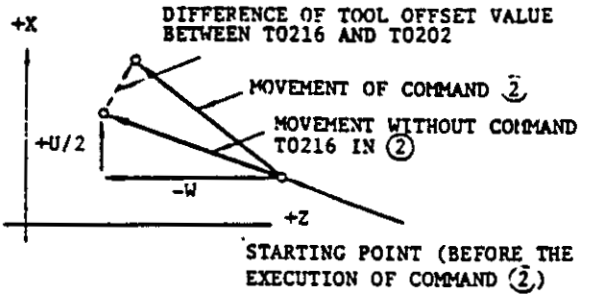


Fig. 2.9.4.3

When the T command and the move command are issued in the same block, the tip 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.

- 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 *
      .
      .
      .
③G00 X . . . Z . . . T0200 * ← The offset motion is cancelled.
```

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

```
G00 X . . . Z . . . *
T0200 ** * ·— Only cancel motion is made at rapid traverse rate.
```

NOTE:

- Tool position offset is cancelled by depressing RESET pushbutton.
- The tool offset must be cancelled before M02 or M30 is commanded.
- The tool offset should be cancelled also before Automatic Zero Return (G28) is commanded.
- When the control is reset by M02 or M30 command or by depressing RESET pushbutton, the tool offset number becomes 0 (or 00).
- When the Zero Return (auto or manual) is executed, the tool offset is cancelled automatically.
- The tool offset must be also cancelled before Zero Return Check (G27) is commanded. If the G27 is commanded at the state that 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.

2.10 PREPARATORY FUNCTIONS (G-FUNCTION)

2.10.1 LIST OF G CODES

The preparatory functions are programmed with the address G and two digits. Table 2.10.1.2 shows the list of G codes. Leading zeros can be omitted. G codes are grouped into the five classes as shown in Table 2.10.1.1. Initial states at the power on are also shown below.

Table 2.10.1.1 Five Groups of G Codes

Group	Modal or Non-modal	Initial State	Remark
A	Modal	G00	For basic function
B	Modal	G40	For tip nose radius compensation
C	Modal	G97	For constant surface speed control
D	Modal	G99 (Note)	For feedrate per minute or per revolution
*	Non-modal	/	One shot command

Note: G99 is the initial state when the contents of parameter No. 71 are "0" and when "1," the initial state is G98.

- G codes of group A to group D are modal. The modal G code, once commanded, is effective till another G code of the same group is commanded.
- The modal G codes can be commanded mixedly in a block. When G codes of the same group are commanded in a block, the last one is effective.
- G codes with an asterisk (*) in the Table 2.10.1.1 are non-modal. They are effective only for the block containing them. They should not be commanded together with the other G codes in one block.
- Depressing the RESET button does not affect the G codes of A, C and D groups, but changes the G codes of B group to G40.
- After the completion of the multiple repetitive cycles (G70 to G76), the G code commanded i the finishing shape program is in effect. Therefore, the G code of each group should be commanded again in the program following the block.
- The effective G codes including the G codes commanded in the previous blocks can be displayed in each group on the universal display. Refer to 4.3.1 Display of Command Data.

Table 2.10.1.2 List of G Codes

G Code	Group	Function	B: Basic O: Optional
G00	A#	Positioning	B
G01	A	Linear interpolation	B
G02	A	Circular interpolation, CW	B
G03	A	Circular interpolation, CCW	B
G04	*	Dwell	B
G11	A	Beveling	O
G12	A	Rounding	O
G22	A	Circular interpolation by radius, CW	O
G23	A	Circular interpolation by radius, CCW	O
G27	*	Zero return check	B
G28	*	Automatic zero return	B
G32	A	Threadcutting	B
G40	B#	Tip nose radius compensation cancel	O
G41	B	Tip nose radius compensation No. 1	O
G42	B	Tip nose radius compensation No. 2	O
G43	B	Tip nose radius compensation No. 3	O
G44	B	Tip nose radius compensation No. 4	O
G50	*	Programming of absolute zero point, Programming of maximum spindle-speed	B O
G70	*	Finishing cycle	O
G71	*	Stock removal in turning	O
G72	*	Stock removal in facing	O
G73	*	Pattern repeating	O
G74	*	Peck drilling in Z axis	O
G75	*	Grooving in X axis	O
G76	*	Automatic threadcutting	O
G90	A	Turning cycle	B
G92	A	Threading cycle	B
G94	A	Facing cycle	B
G96	C	Constant surface speed control	O
G97	C#	Constant surface speed control cancel	O
G98	D	Feed amount per minute	B
G99	D	Feed amount per revolution	B

Note: When power is turned on, the code with # is effective in each group.

2.10.2 POSITIONING (G00)

· 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.

· G00 X · · · W · · · ✱ or
G00 U · · · Z · · · ✱

As shown above, absolute coordinate values and incremental coordinate values can be mixedly commanded in a block.

· For the rapid traverse rate, as it depends upon the machine, refer to the machine tool builder's manual. Since a tool is moved on each axis independently, the tool path is generally nonlinear.

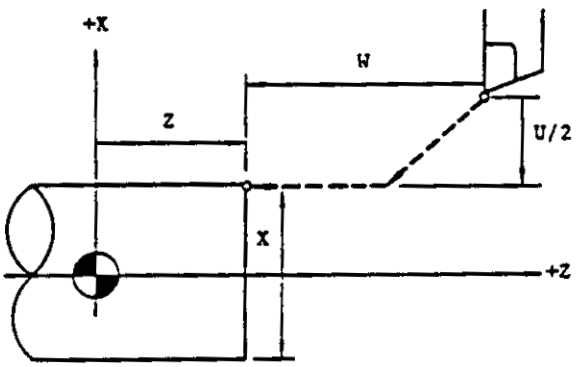


Fig. 2.10.2.1

Note: When T, S or M code is commanded, the G00 should be put in the block. The G code is required for the designation of tool traverse rate for tool offset motion.

EXAMPLE

G50 X6000 Z5000 ✱

G00 T0101 S500 M03 ✱ ··· G00 for designation of traverse rate for tool offset

(G00) X3000 Z500 ✱ ··· G00 can be omitted in positioning.

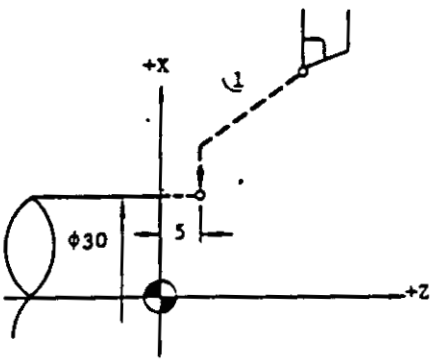


Fig.2.10.2.2

2.10.3 LINEAR INTERPOLATION (G01)

· G01 X(U) · · · Z(W) · · · F · · · ✱

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

· F 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 code is the tangential feed rate.

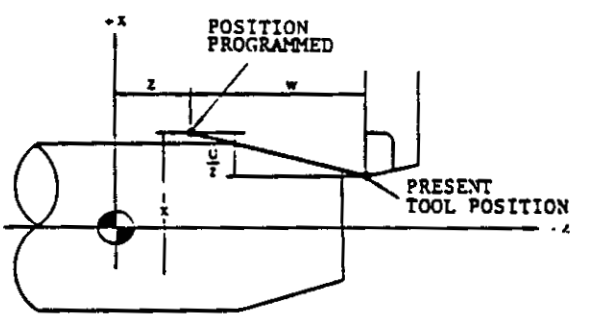


Fig. 2.10.3.1

G50 X10000 Z6000 ✱

G00 T0202 S600 M03 ✱

X3500 Z500 ✱

① G01 Z0 F100 ✱

② X6000 F20 ✱

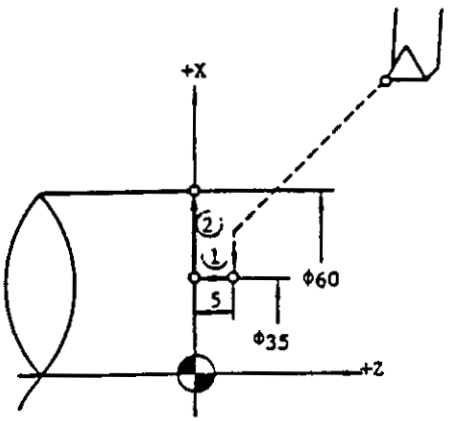


Fig. 2.10.3.2

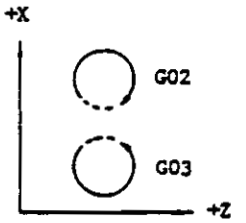
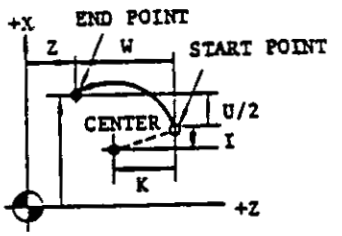
2.10.4 CIRCULAR INTERPOLATION (G02, G03)

· G02(G03) X(U)··· Z(W)··· I··· K··· F··· *

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 an circular arc at the traverse rate specified by the F code.

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

Table 2.10.4

	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	

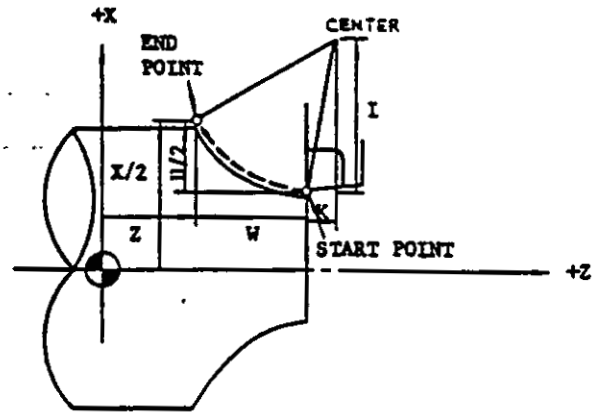


Fig. 2.10.4.1

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

EXAMPLE

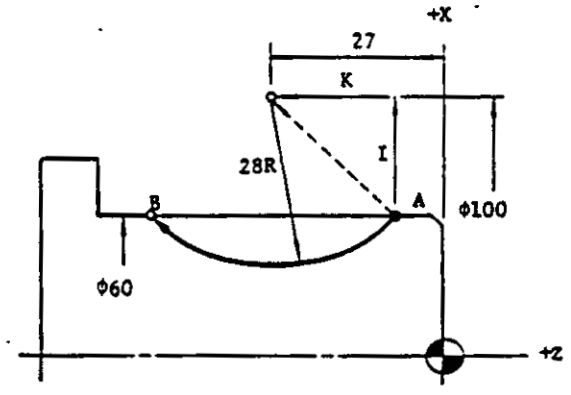


Fig. 2.10.4.2

Arc center coordinates	(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 X6000 Z-4660 I2000 K-1960 F · · · ✲

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

- 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-handed coordinate system, the tool moves clockwise from the beginning point of the arc."
Therefore, the direction of rotation in the plane (-X.Z plane) of Fig. 2.10.4.3 is represented inversely.

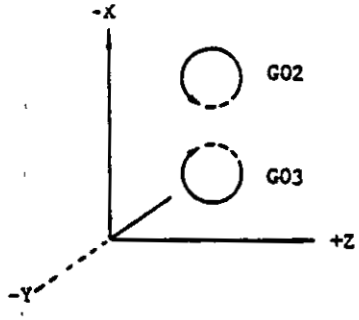


Fig. 2.10.4.3

- 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 ○ indicates the end point of arc.

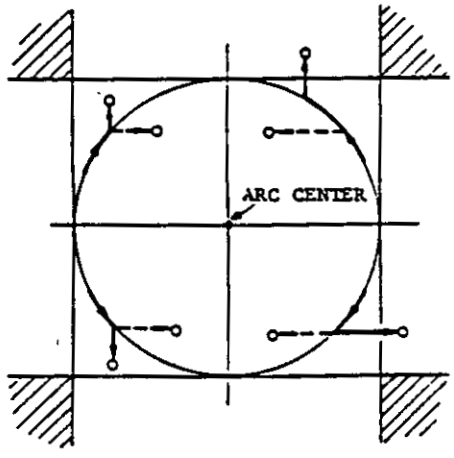


Fig. 2.10.4.4

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 tip 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.

· Although the circular interpolation on multi-quadrant can be done, a closed circle cannot be designated for interpolation.

EXAMPLE

G00 X(a) Z(b) *

G02 X(a) Z(b) I··· K··· F··· * (Designation of closed circle)

In the above, the control will regard the designation as the completion of circular interpolation without tool motion. The program can be executed if there is a longer distance than least input increment, between the end point and the start point. See Fig. 2.10.4.5.

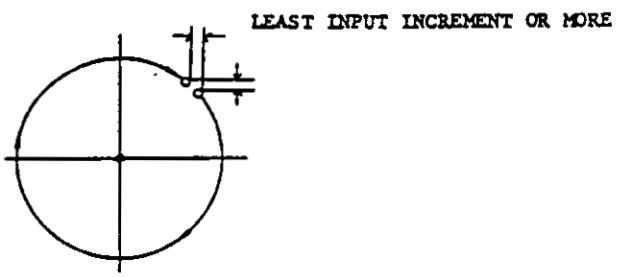


Fig. 2.10.4.5 Designation of Circle

This rule also applies to the circular arc compensated by the operation of tip nose radius compensation.

2.10.5 DWELL (G04)

· G04 U · · · *

This command stops feed for the duration specified by the address U. Dwell should be solely programmed in a block without the other instructions.

· The maximum duration time specified by the address U is shown below.

Table 2.10.5 Maximum Programmable Value of Dwell Time

Metric		Inch†	
0.01 mm	0.001 mm	0.001 inch	0.0001 inch
999.999 sec		999.999 sec	

➡ Least command increment

Since block motions are smoothly connected in cutting feed at Error Detect Off, the corner of workpiece is rounded a little bit. The dwell function prevents or controls the rounding.

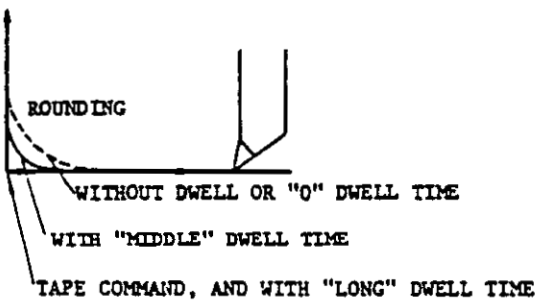


Fig. 2.10.5

NOTES:

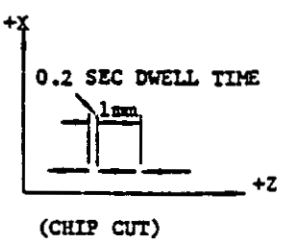
· Since G04 (dwell) is non-modal, it is effective only for the block containing it.

· No command should be put in the block containing the G04.

G01 W-100 F25 * ··· Feed: 1 mm

G04 U200 * ····· Dwell time: 0.2 sec

G01 W-100 * ····· Feed: 1 mm



(CHIP CUT)

2.10.6 CORNERING (G11, G12)†

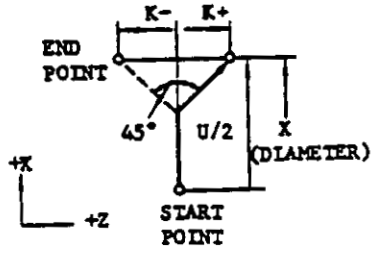
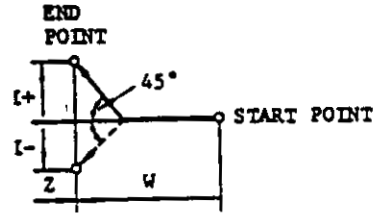
This command removes the sharp corner of workpiece. Addresses X and Z cannot be specified simultaneously in a block.

Meaning of each address is shown below.

1. Beveling (G11)

$$G11 \begin{Bmatrix} X(U) \cdots K \cdots \\ Z(W) \cdots I \cdots \end{Bmatrix} F \cdots *$$

Table 2.10.6.1

Beveling for X axis	Beveling for Z axis
G11 X(U) · · · K · · · F · · · *	G11 Z(W) · · · I · · · F · · · *
	
K ± · · · · · Beveling Value Beveling Direction	K ± · · · · · 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.

When G11 command starts to be executed, the display for G code is changed to G01. After the completion of the execution, G11 is displayed again.

```
G00 X3000 Z0 *
① G11 Z-2000 I800 F30 *
② (G11) X8000 K-700 *
```

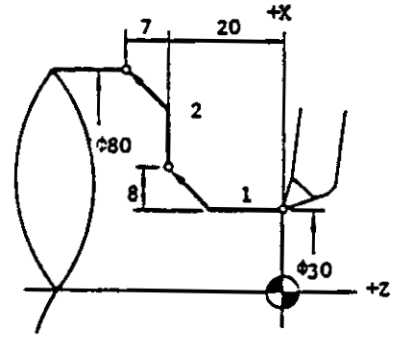


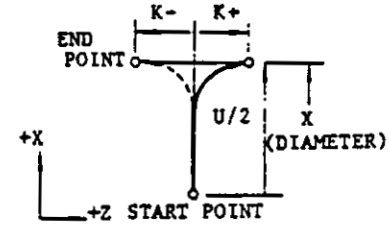
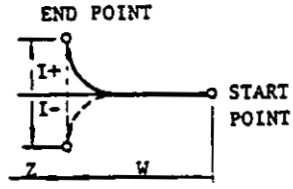
Fig. 2.10.6.1

2. Rounding (G12)

$$G12 \begin{Bmatrix} X(U) \cdots K \cdots \\ Z(W) \cdots I \cdots \end{Bmatrix} F \cdots *$$

This command performs the rounding of the corner. Addresses X and Z cannot be specified simultaneously in a block. The corner is rounded to a quadrant. Meaning of each address is shown below.

Table 2.10.6.2

Rounding for X axis	Rounding for Z axis
G12 X(U) · · · K · · · F · · · *	G12 Z(W) · · · I · · · F · · · *
	
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.

When G12 command is executed, G01 is displayed while the tool moving on the linear line, and G02 or G03 is displayed depending on the rounding direction while along the circular arc. After the completion of the execution, G12 is displayed again.

G00 X2000 Z0 *

① G12 Z-2500 I900 F30 *
② (G12) X7000 K-600 F20 *

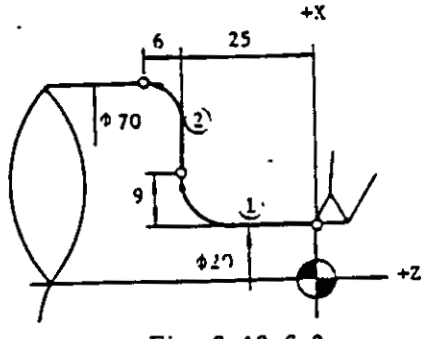


Fig. 2.10.6.2

NOTES

- Since G11 and G12 are modal G codes of group A, they are retained until the other G codes of group A are specified.
- X and Z axes cannot be specified simultaneously in G11 or G12 command. The block containing addresses of both axes causes format error.

EXAMPLE

G12 X··· W··· K··· * ←——Error

- The block without I and K or the block in which I and K are zero, should not be issued. The tool may not move properly.
- ·· Tip nose radius compensation is effective for the blocks including G11 or G12 command.
- * The block including G11 or G12 command can be specified in the finishing shape commands of multiple repetitive cycles (G70 through G73).

2.10.7 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.

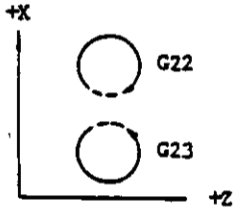
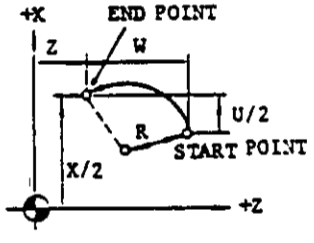
In programming of G22 or G23, the control automatically calculates the arc center coordinates (I, K) from the radius value designated by the address R and performs circular interpolation.

· G22 / G23 } X(U)··· Z(W)··· R··· F··· ;

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

· The meanings of G22, G23 and each address are shown below.

Table 2.10.7

	Meanings	
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 no sign)	

· Designation of radius value R

R is an incremental value and commanded with no sign. Maximum programmable value of R is in accordance with 2.3.5 Maximum Programmable Value. When the result of an operation exceeds the maximum programmable value, it causes error.

· Angle restriction of circular interpolation

θ must be less than 180° at G22 and G23 commands. Therefore, a circular arc over four quadrants cannot be programmed.

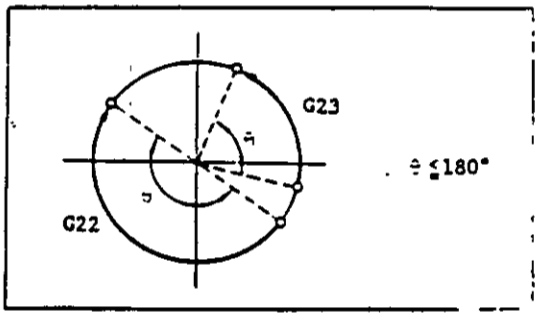


Fig. 2.10.7.1

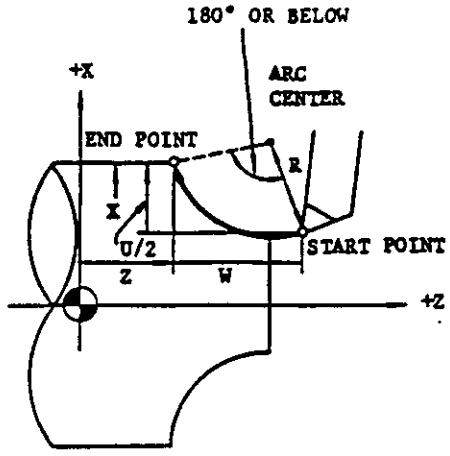


Fig. 2.10.7.2

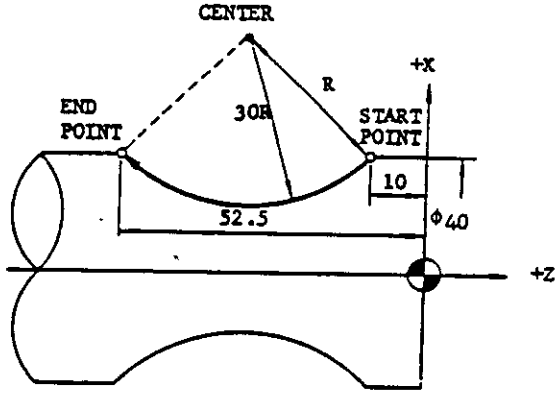


Fig. 2.10.7.3

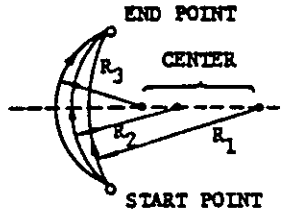
G01 X4000 Z-1000 F20 *

① G22 (X4000) Z-5250 R3000 (F20) *

Note: Parenthesized commands can be omitted in this case.

NOTES:

- G22 and G23 are modal. They are retained until the other G code of group A is programmed.
- In the G22 or G23 mode, the block in which R is not contained or R is designated to zero should not be commanded.
- When R is varied at 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.

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

- † Tip nose radius compensation is effective for the block containing G22 or G23.
- † The block containing G22 or G23 can be designated in finishing shape commands of multiple repetitive cycles (G70 to G73).

2.10.8 ZERO RETURN CHECK (G27)

This function checks that the program is made so that the machine starts from and returns to the fixed original point.

- G27 X(U) . . . Z(W) . . . *

  After positioning at the absolute coordinates (X, Z) or the incremental coordinates (U, W), the position is checked if it is the fixed original point or not.

- When the position meets with the fixed original point ZERO RETURN lamp is lit. When the position meets with the fixed original point as for both axes, the automatic operation# will be continued. When either of the axes does not meet with the fixed original point, the alarm code "24" (the zero return position error) is displayed and the automatic operation is stopped. (The CYCLE START lamp is turned off.)
- When G27 is commanded in tool offset mode, the position will not meet with the fixed original point. Tool offset mode should be cancelled as follows when G27 is commanded.

"Automatic operation" means operation in TAPE, MDI or MEM mode in this manual.

To cancel tool offset mode prior to the block containing G27,

T□□00 *

G27 U . . . W . . . *

To cancel tool offset mode in the block containing G27,

G27 U . . . W . . . T□□00 *

NOTE: The fixed original point means the definite position on the machine where the tool is returned by the manual Zero Return operation or Automatic Zero Return command (G28). It is represented by ⊕ in this manual.

2.10.9 AUTOMATIC ZERO RETURN (G28)

· G28 X(U) . . . Z(W) . . . *

After positioning two axes simultaneously at the absolute coordinates (X, Z) or the incremental coordinates (U, W), the Zero Return motion is performed for two axes simultaneously.

· After the completion of Zero Return, ZERO POSITION lamp of the axis is lit. After the completion of Zero Return for both axes, the automatic operation is continued.

· For Zero Return operation, the tool should be in the area from which the tool can return to the fixed original point, e.g. the area where Zero Return limit switch is not operated. If not, the tool can be moved to the area by positioning command.

· Tool offset mode should be cancelled as follows when G28 is commanded.

To cancel tool offset mode prior to the block containing G28,

T□□00 *

G28 X . . . Z . . . *

To cancel tool offset mode in the block containing G28,

G28 X . . . Z . . . T□□00 *

NOTES:

· The Zero Return motion means the same motion as that by operating the Manual Zero Return. Refer to 6.2.1 Manual Zero Return.

· When G28 is commanded in tool offset mode, the tool offset is finally cancelled without cancelling motion.

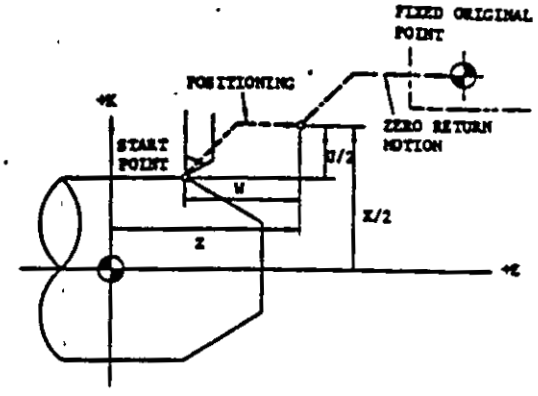
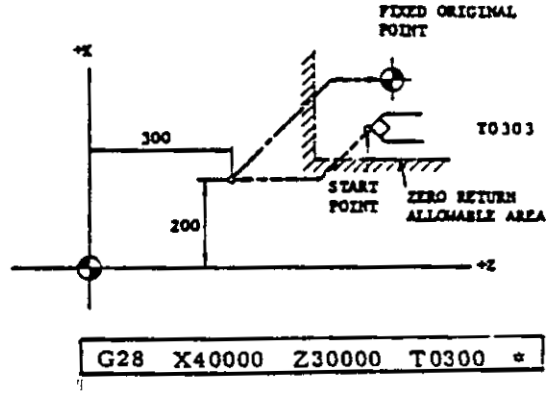


Fig. 2.10.9.1



G28 X40000 Z30000 T0300 *

Fig. 2.10.9.2

2.10.10 THREADCUTTING (G32)

Straight thread, taper thread and scroll thread are cut by G32 command.

· G32 X(U) . . . Z(W) . . . $\begin{Bmatrix} F \cdot \cdot \cdot \\ E \cdot \cdot \cdot \end{Bmatrix}$ *

Threadcutting is performed in the lead specified by the R or E code. The end point is absolute coordinates (X, Z) or incremental coordinates (U, W).

Programmable lead ranges of F and E codes are shown below.

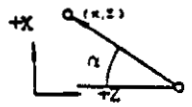
Table 2.10.10.1 Programmable Lead Range

	Format	Programmable Range	Meanings
Metric	F2.2	F1 to F9999	0.01 to 99.99 mm
	E2.4	E1 to E999999	0.0001 to 99.9999 mm
Inch†	F1.4	F1 to F39366	0.0001 to 3.9366 inch
	E1.6	E1 to E3936600	0.000001 to 3.936600 inch

F code is for normal threadcutting.
E code is for precise threadcutting.

· The direction of lead specified by F and E codes is shown below.

Table 2.10.10.2 Direction of Lead

Limitation of Taper Angle		Direction of Lead
	α ≤ 45°	Lead in the direction of Z axis
	α > 45°	Lead in the direction of X axis

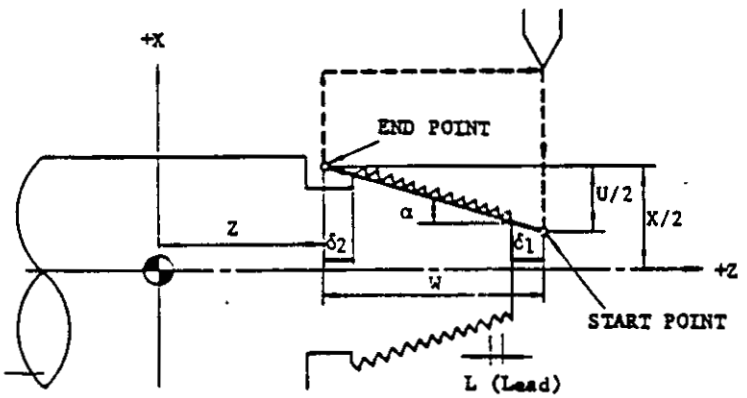


Fig. 2.10.10.1

· Command format of threadcutting is shown below.

Table 2.10.10.3

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) · · · E · · · *
Scroll Thread	Normal	G32 X(U) · · · F · · · *
	Precise	G32 X(U) · · · E · · · *

NOTES:

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

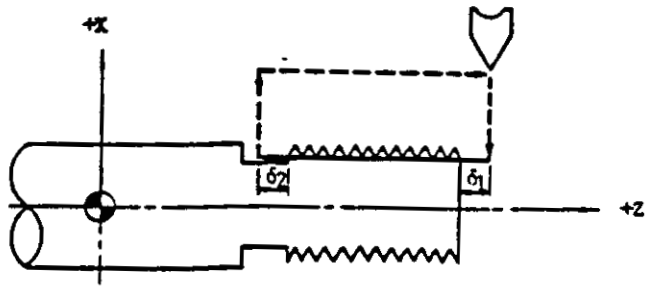


Fig. 2.10.10.2

- If spindle speed is not constant during thread-cutting, the leads become incorrect due to the servo lag.
- Chamfering for thread is not effective at G32. If necessary, G12 (or G76†) should be commanded.
- The following operation is disregarded during threadcutting including G32, G92 and G76†.
  - Feedrate Override . . . Regarded as 100%
  - Feed Hold operation
- The G32 command should not be commanded in G98 mode. If commanded, the alarm "14" will be displayed.
- In Dry Run mode, the tool moves at Jog feed-rate.

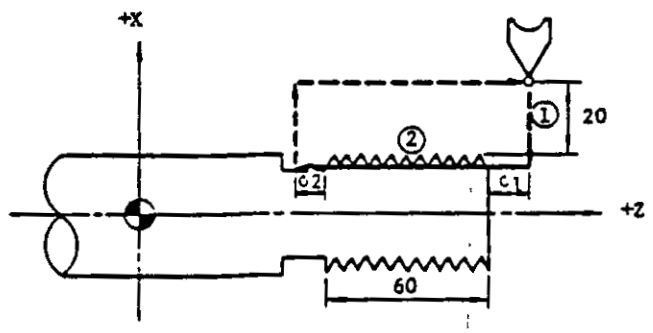


Fig. 2.10.10.3

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

① G00 U-4200 *

② G32 W-6800 F500 *

G00 U4200 *
W6800 *
U-4400 *

G32 W-6800 *

G00 U4400 *

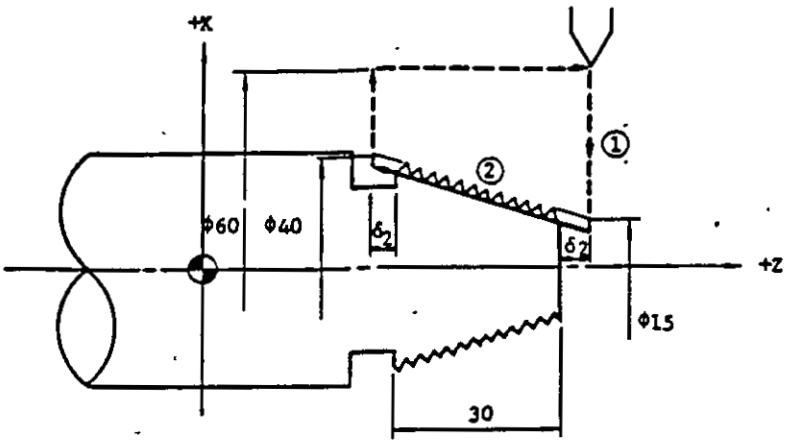


Fig. 2.10.10.4

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

```
① G00 X1300 *
② G32 X3800 W-3500 F400 *
  G00 X6000 *
      W3500 *
      X1100 *
  G32 X3600 W-3500 *
  G00 X6000 *
```

· Allowance for lead error (δ_1, δ_2)

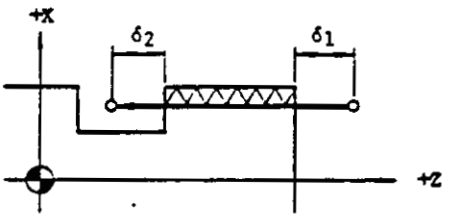


Fig. 2.10.10.5

δ_1 and δ_2 are obtained approximately from the following equation.

Table 2.10.10.4

	Equation	Meanings
δ_1	$\delta_1 > \frac{L \cdot S}{60 \cdot K}(\ln\frac{1}{a} - 1)$	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	$\delta_2 > \frac{L \cdot S}{60 \cdot K}$	

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 accuracy a = 1/100

$$\delta_1 > \frac{L \cdot S}{60 \cdot K}(\ln\frac{1}{a} - 1) = \frac{3.0 \times 500}{60 \cdot K} \times 3.61 = 3.0 \text{ mm}$$

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

2.10.11 TIP 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 tip nose radius compensation option resolves the problem of nose radius. See Fig. 2.10.11.1.

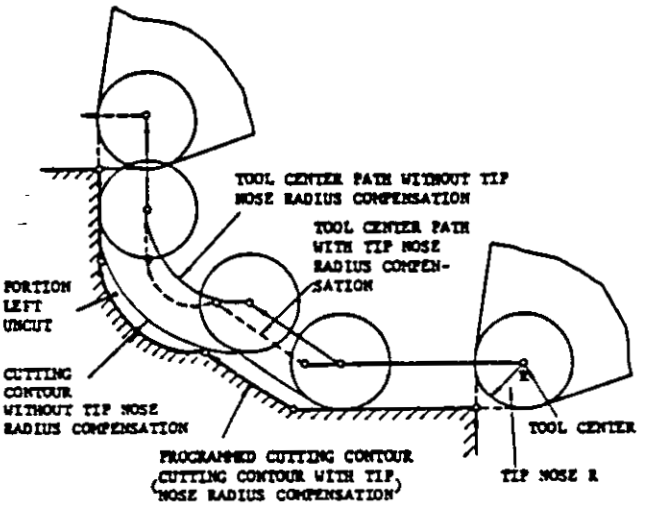


Fig. 2.10.11.1

1. Tip nose radius values

A. Radius value storage

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

T 2-digit: 9 sets
T 4-digit: 16 or 32 sets

Refer to Fig. 2.9.3 Storage of Tool Offset Value.

B. Range of tip nose radius values

Radius value can be set within the following range.

Table 2.10.11.1

Metric	Inch†
0 to 24.999 mm	0 to 0.9999 inch

C. Setting of tip nose radius values

Radius value of tip nose must be set without signs.

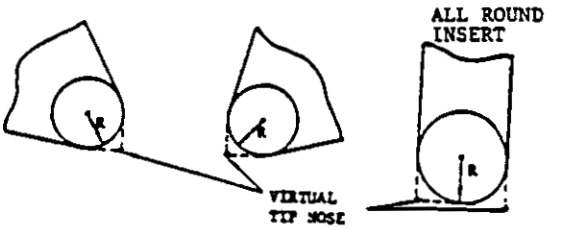
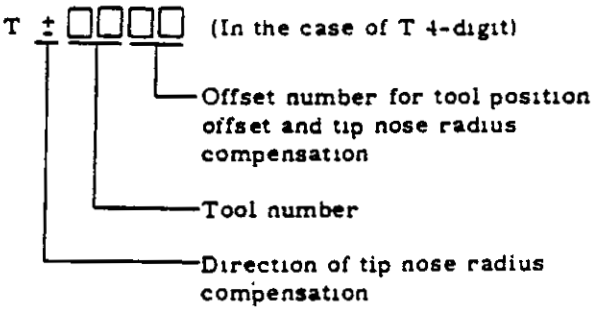


Fig. 2.10.11.2

For the writing of radius values for tip nose radius compensation, refer to Fig. 4.3.6 Writing of Tool Offset Value. The address character is R.

2. T code designation

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



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

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

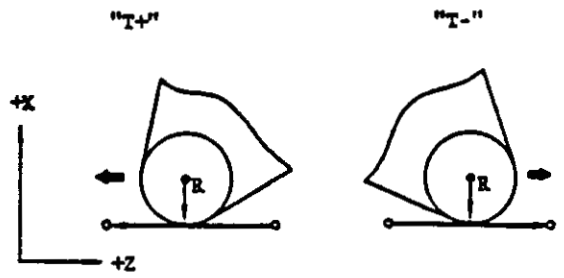


Fig. 2.10.11.3

B. Before the direction of compensation is changed from + to - or - to +, T□□00 and G40 should be programmed to cancel the tip nose radius compensation. Even if the sign is changed without cancelling, the direction of the compensation will not be changed. For instance, 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.

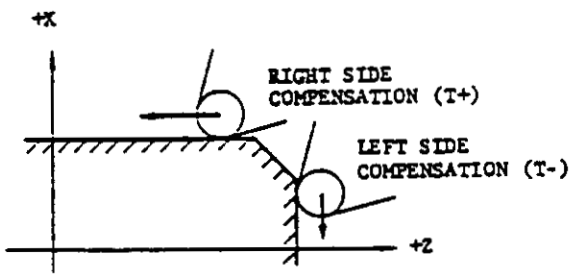


Fig. 2.10.11.4

3. G code designation (G40 to G44)

A. G code of tip nose radius compensation (G41 to G44)

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

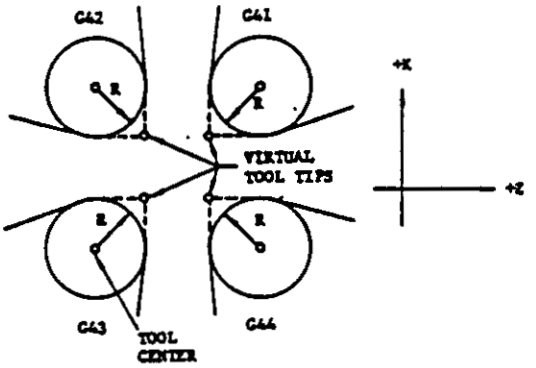


Fig. 2.10.11.5

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

B. Issue G40 to cancel the tip nose radius compensation

C. Cautions in programming G code

(1) Since G40 to G44 are modal G codes of B 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.

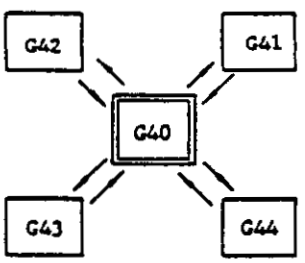


Fig. 2.10.11.6

(2) G40 to G44 must be programmed solely without the other command in the same block.

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

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

4. Tool motion on the tip nose radius compensation

A. Fig. 2.10.11.7 shows the outline of the tool motion.

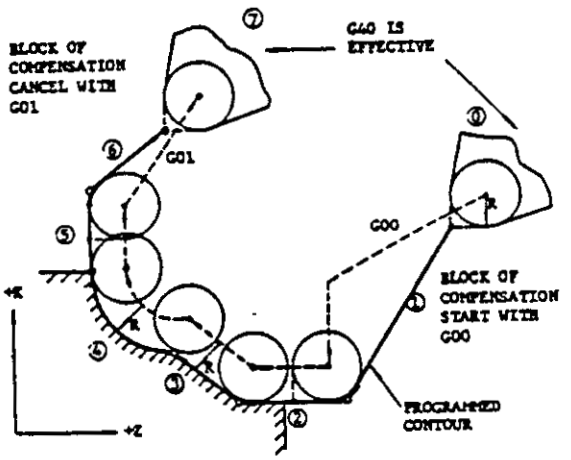


Fig. 2.10.11.7

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

(2) In compensation mode, the tool center path is deviated by radius from the programmed contour. Therefore, the path of virtual tip 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).

(3) In compensation mode, an intersection of the tool center paths of two blocks must be calculated so that the tool center passes the intersection to perform the next block. Therefore, in compensation mode, the control is required to read ahead 2 blocks of information for movement.

(4) 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. Reading ahead of two or three blocks

The control reads ahead two blocks of data to calculate the next position in compensation mode.

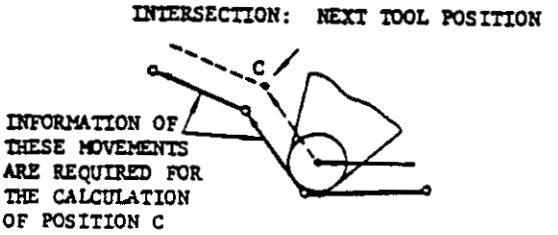


Fig. 2.10.11.8

However, when the second block does not include move command, the third block is read into calculate the next tool position. If the third block does not include a move command, the compensation is temporarily cancelled. Therefore in compensation mode, two blocks without move command should not be programmed in series.

```
G41 *
 .
 .
 .
G04 U··· *  ]  Compensation is temporarily
M··· *      ]  cancelled due to the series
 .             of two blocks without move
 .             command.
 .
G40 *
```

C. Tip nose radius compensation and tool position

In compensation mode, tool position offset motion is regarded as the same as the motion by move command. Therefore, the tip nose radius compensation is also effective for tool position offset motion. In other words, the path of tool position offset motion is compensated.

5. Programming procedure of tip nose radius compensation

The tip nose radius compensation is executed when tool offset number (T code) and G code for the compensation are specified.

A. Compensation start

Use the following format when the G40 mode is switched to G41, G42, G43 or G44 mode.

```
.
.        } G40 mode
.
```

```
G00··· T±□□△△*          ----Designation of tool offset number
G41 *                   ----Or G42 to G44
G01 X(U)··· Z(W)··· F···*----Block of compensation start with G01 or G00
.
.
.
```

(1) Specify tool number (T code) before G code of compensation mode is programmed.

(2) Program G code solely without the other commands in the same block.

(3) G01 or G00 must be programmed for the start of compensation. G01 is for linear interpolation and G00 is for positioning. Circular interpolations (G02, G03) cannot be used. If used, the alarm code "15" is displayed.

NOTE.

The following program should be avoided.

```
┌G41 *
│G00 T±□□△△* ···  Block of compensation start
└G01 ···
```

This program enters the tip nose radius compensation simultaneously with the tool position offset, which makes a programmer hard to understand the actual motions.

① G41 *

② G01 T-0606 F··· *

③ G01 X··· Z··· *

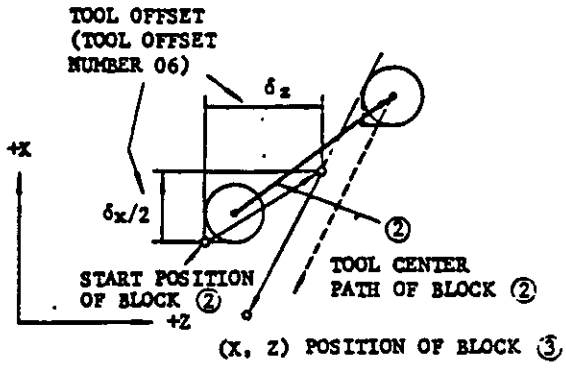


Fig. 2.10.11.9

EXAMPLE 1:

```
G50 X11000 Z4000 *
G00 S600 M03 T0202 *
G41 *
① (G00) X3000 Z500 *  ··· Compensation start
② G01 Z-1800 F25 *
.
.
.
```

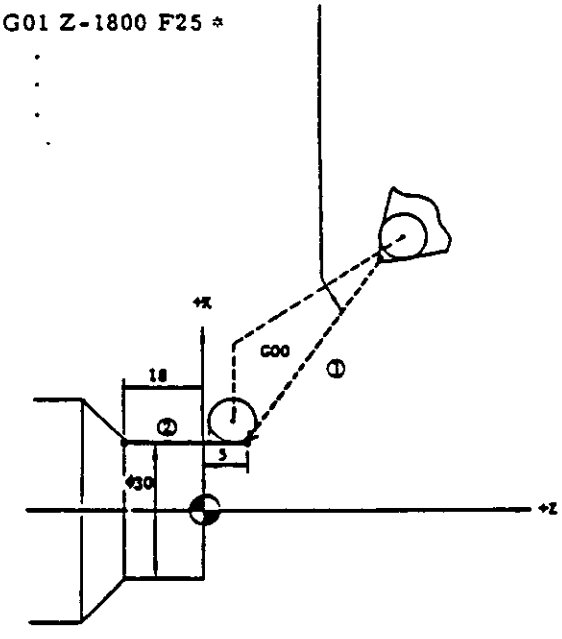


Fig. 2.10.11.10

EXAMPLE 2:

```
G50 X8000 Z4000 *
G00 S400 M03 T0303 *
G41 *
① G01 X2000 Z700 F600 * ... Compensation start
②     X5000 Z-1500 F30 *
```

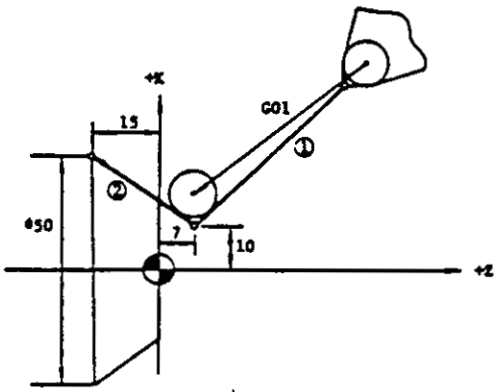


Fig. 2.10.11.11

B. Compensation cancel

When G41, G42, G43 or G44 is switched to G40, use the following format.

```
(G41 to G44)
      .
      .
G01 X(U)··· Z(W)··· F···
    T±□□ 0 0 * —        > Compensation cancel with G01 or G00
G40 *
      .
      .
```

(1) Program the tool offset number 00 (T code) before the block of G40 to cancel the compensation.

(2) T00 should be programmed in a block together with G01 (linear interpolation) or G00 (positioning). G02 and G03 (circular interpolation) cannot be used. If used, the alarm code "15" will be displayed.

(3) G40 (compensation cancel) must be programmed solely without the other commands in the same block.

NOTE:

The following program should be avoided.

```
┌G00 X··· Z···, *
│  T±□□00 * ··· Cancel of compensation
└G40 *
```

This program cancels the tip nose radius compensation simultaneously with the tool position offset cancel, which makes a programmer hard to understand the actual motions.

```
(G41, T-0505)
      .
      .
① G01 Z··· F··· *
②     T-0500 *
G40 *
```

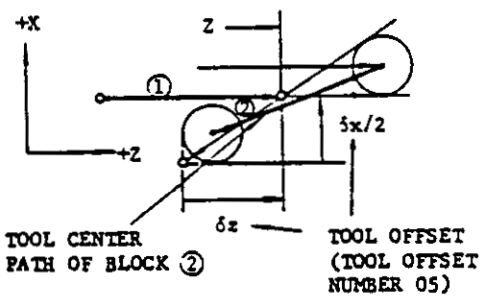


Fig. 2.10.11.12

EXAMPLE 1

```
(G41, T+0202)
      .
      .
G02 .....
G01 U2000 F25 *
G00 X11000 Z4000 T0200 * ... Compensation cancel
G40 *
```

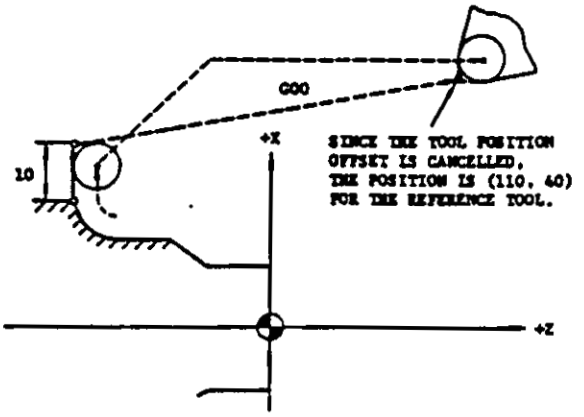


Fig. 2.10.11.13

EXAMPLE 2

```
(G41, T+0303)
   .
   .
   .
G01  ..
G01  U2400  F30  *
G01 X8000 Z4000 F600 T0300 *
                          ... Compensation
                              cancel
G40  *
```

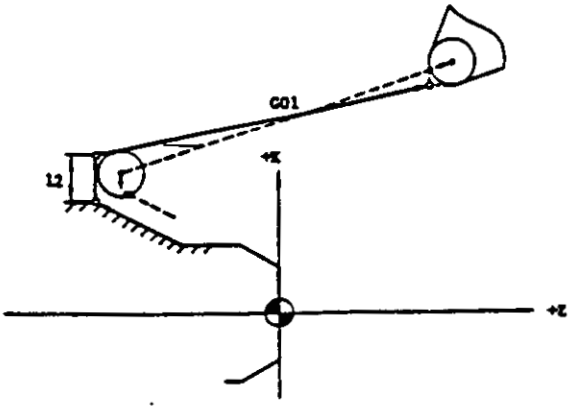
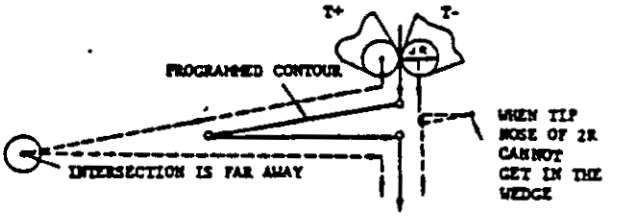


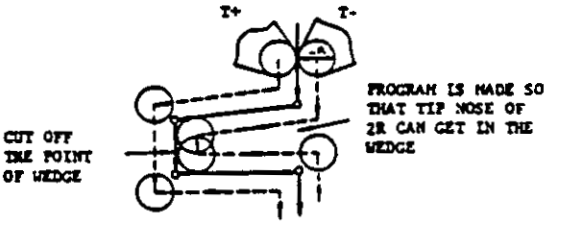
Fig. 2.10.11.14

C. Programming consideration in compensation mode

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



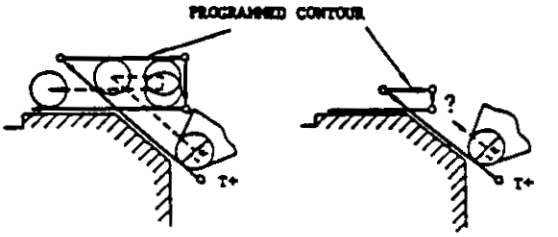
(a) Wrong



(b) Correct

Fig. 2.10.11.15

(2) Program the tool movement so that the tip nose of 2R diameter can get in the contour.



(a) Correct (b) Wrong

Fig. 2.10.11.16

NOTE:

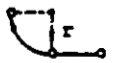
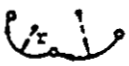
When a program becomes error or causes wrong operation in tip nose radius compensation, usually it is caused by the above two cases. Check the program.

6. Precautions on tip nose radius compensation

A. Max programmable value

Max programmable value of tip nose radius compensation shown below.

Table 2.10.11.2

Item		Restriction
Length of Linear Interpolation		$\ell \leq \pm 5931.64\ (0)$ mm
Joint of Linear Line and Circular Arc		$r \leq \pm 8388.60\ (7)$ mm $(r = \sqrt{I^2 + K^2})$
Joint between Circular Arcs		$r \leq 2965.81\ (8)$ mm I and K $\leq 2097.15\ (0)$ mm

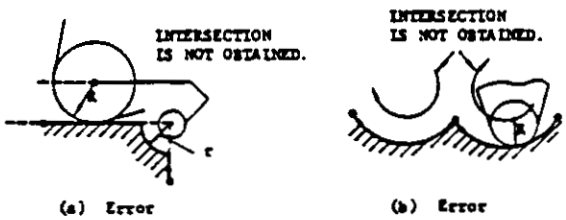
Note: Above figures are for the control of specification A.

B. Program which causes data error

Data error (the alarm code "15") occurs in the following cases.

(1) A command other than G00 and G01 is issued in the block of compensation start or compensation cancel. (When G02, G03, G12, G22 or G23 is issued, the ALARM lamp is lit. When G11, not.)

(2) The contour whose intersection between blocks cannot be obtained on the tool center path.

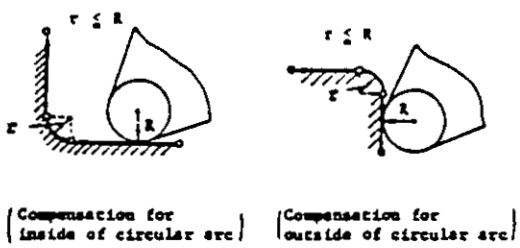


(a) Error (b) Error

Fig. 2.10.11.17

(3) The following circular arc is commanded for compensation for the inside of circular arc.

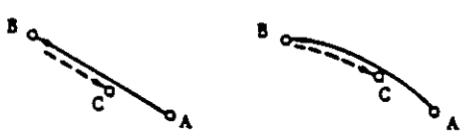
Radius of commanded circular arc (r) ≦ Radius of tip nose (R)



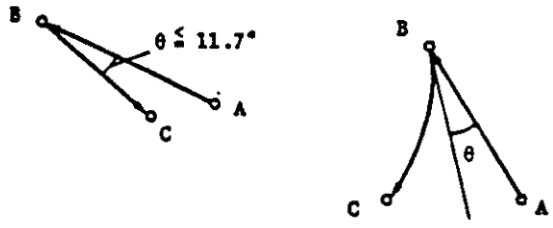
(a) Error (b) Correct

Fig. 2.10.11.18

(4) Inverse or nearly inverse cutting is programmed.



(a) Inverse cutting



Note: In the case of circular arc, tangential angle θ is not enough to judge whether the program causes error or not.

(b) Nearly Inverse Cutting

Fig. 2.10.11.19

C G codes usable in compensation mode

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

Table 2.10.11.3

Usable G codes	Remarks
G00, G01, G04, G11 G96, G97 ··· Constant surface speed control G98, G99 ··· Feed function designation	
G12, G22, G23 ··· Command including circular arc G70, G71, G72, G73 ··· Multiple repetitive cycle	Inhibited in the block of compensation cancel or start

D. Change of tip nose radius value

As a rule, tip nose radius values should not be changed in compensation mode. If changed directly for example T±□□02 → T±□□03, the old tip nose radius value (the contents of tool offset number "02") remains in effect. However, when the new tip nose radius value (the contents of tool offset number "03") is 0, the compensation is temporarily cancelled.

Before the tip nose radius value is changed, the compensation mode should be cancelled. Refer to page 38.

NOTE: When T±□□00 is commanded, both functions of tip nose radius compensation and tool position offset are cancelled. Though the compensation can be cancelled by the following program, note that a wedge-shaped contour may occur due to tool position offset motion. It is recommendable to change the radius value after taking the procedure on page 38 for compensation cancel.

```
(G41, T+0202)
      .
      .
      .
T+0200 * ·· Cancel of compensation
T+0203 * ··· Tool offset number 03
      .        becomes effective.
      .
```

E. Inhibited commands and operations

When the commands or operations listed in Table 2.10.11.4 are performed in the compensation mode, the compensation is cancelled or temporarily cancelled.

Table 2.10.11.4

No.	Commands or Operations	Compensation
1	Command of two blocks without move command in series	Temporary cancel
2	Command of M01 or M02	
3	Command of tool offset number whose contents are 0	
4	T± □ □ 00 (Note)	
5	Commands of the following G codes: G27, G50, G74, G75, G76, G90, G92, G94	
6	Command of M02 or M30	Cancel
7	Command of G28	
8	Reset operation	
9	Power off operation	

Note: When the compensation is cancelled by G40 * in the next block, this command can be programmed.

F. Inhibition of MDI mode

(1) Operation in the MDI mode cannot be performed in the compensation mode (G41 through G44). When RESET button is depressed, G40 (compensation cancel) becomes effective and the operation in the MDI mode becomes possible.

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

G. Tip nose radius compensation OFF for G00

Tip nose radius compensation is effective also on positioning (G00) motions. But when the contents of parameter No. 89 are set to "0," the compensation becomes ineffective. Normally it is set to "1" and the tip nose radius compensation is effective for G00.

EXAMPLE

```
    (G99)
①  G50  X14000  Z3000  *
②  G00  S1700  M03  T(+)0202  * ...  [· Tool position offset is executed.
                                     [· Compensation No. is designated.
③  G41  * . . . Compensation mode is specified.
④  (G00)  X0  Z500  * . . . Tip nose radius compensation start
   G01  Z0  F20  *
        X2000  *
        Z-2000  *
        X3000  W-1500  S1100  *
   G12  W-2000  I300  * . . . (Rounding)
   G11  X5000  K-300  S700  * . . . (Beveling)
   G01  Z-7000  *
   G22  X9000  Z-9000  R2000  S360  * ... (Radius programming
                                           for circular arc)
   G01  X11000  S300  *
   G04  U200  * . . . (Dwell)
   (G01)  Z-11000  *
          X12000  *
⑰  G00  X14000  Z3000  T(+)0200  * ... Tip nose radius compensation
                                       cancel
⑱  G40  * . . . Compensation cancel mode is specified.
```

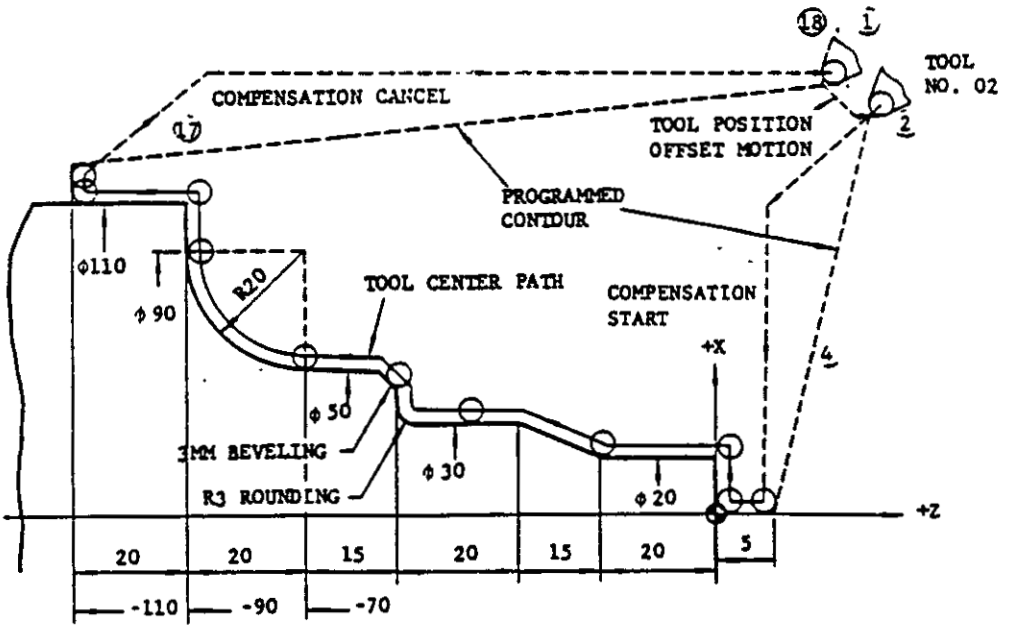
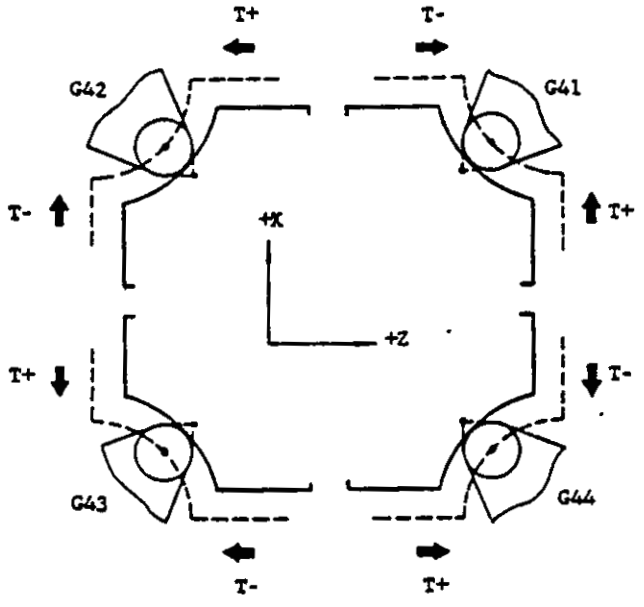
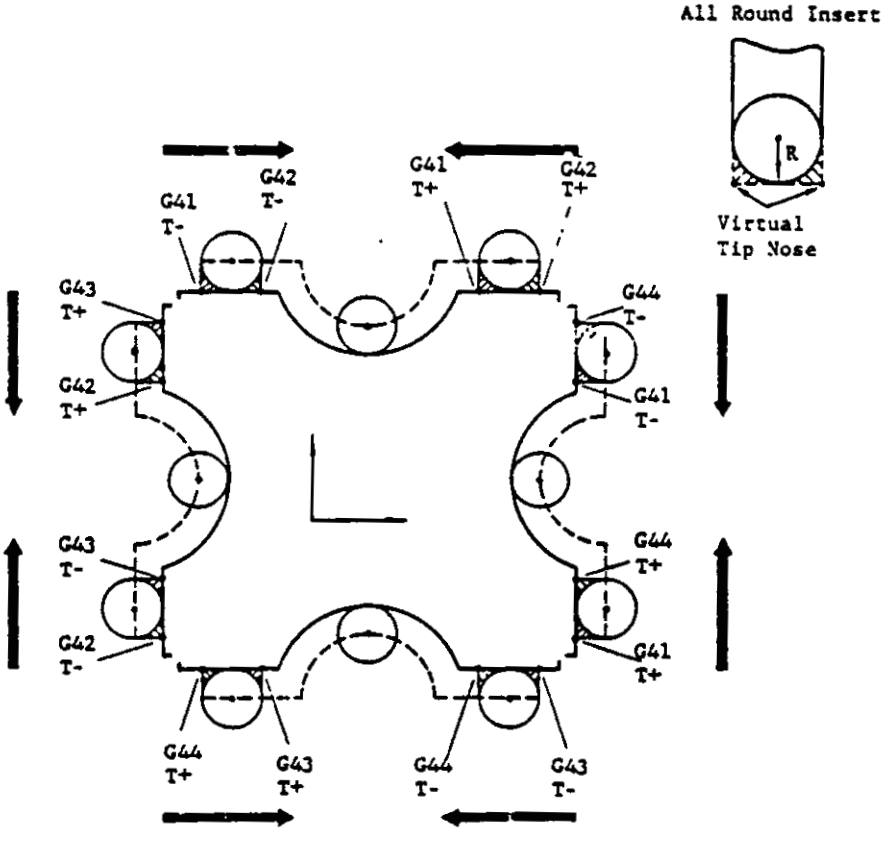


Fig. 2.10.11.20



(a) Normal Insert



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

Fig. 2.10.11.21 Relations between G Code and Sign of T Code for Tip Nose Radius Compensation

2.10.12 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.

· 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."

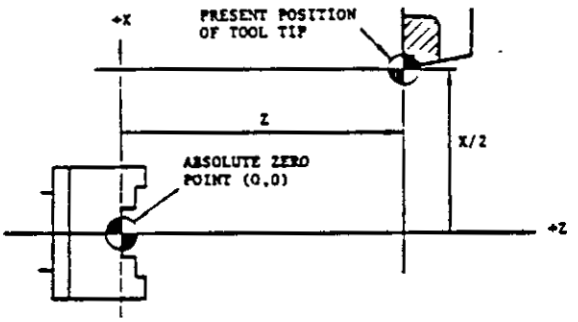


Fig 2.10.12.1

Fig. 2.10.12.1 shows the programming of the absolute zero point at the fixed original point. The G50 can be commanded at any position.

All motions of tool can be programmed on a single coordinate system by setting up G50 for the reference tool and applying tool offset to the other tools set up for the reference tool (for example, Tool No. 1) by the following command.

G50 X8000 Z6200 *

When tool offset for the Tool No. 2 is performed, the Tool No. 2 moves to the position A in Fig. 2.10.12.2.

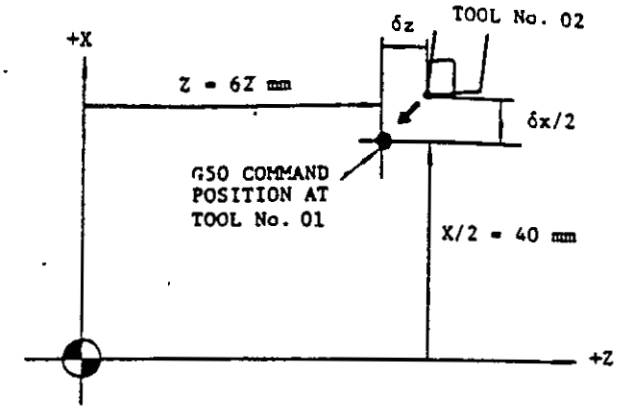


Fig. 2.10.12.2

· 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 by the addresses U and W at the incremental G50 command.

G50 U10000 W-10000 * ··· Setting of Position B

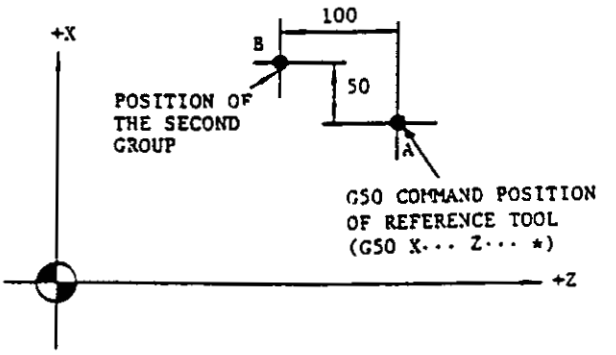


Fig. 2.10.12.3

NOTES:

- When T, S and M commands are programmed in the next block containing G50, G00 should be programmed in the block. This is for the designation of traverse rate for tool offset motion.

  G50 X· · · Z· · · *

  G00 S500 M03 T0101 *

- G50 is in effect only for the block containing it. T, S, M and the other G commands cannot be programmed in the block containing G50. G50 command is performed without movement of tool.

- G50 should be commanded after the tool offset is cancelled.

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

- The absolute coordinate system set up is not cleared by reset operation. It is cleared by taking the following procedure.

  1. Write G50 X0 Z0 * in MDI mode and depress the START button.
  2. Depress the POWER OFF button.

2.10.13 MAXIMUM SPINDLE-SPEED SETTING (G50)T

The upper limit of spindle speed can be specified by the G50 command.

- G50 S· · · *

  Four digits following the address S directly specifies the upper limit of spindle speed in rpm. If the S command exceeding the limit is issued in later blocks, the spindle speed is clipped at the upper limit.

  In G96 (Constant Surface Speed Control) mode, when spindle speed rises up too fast as the current X-coordinate of the tool is tool small, the spindle speed is clipped to the limit. Refer to 2.10.16 Constant Surface Speed Control.

EXAMPLE

G50 S2000 *

The maximum spindle speed is clipped at 2000 rpm.

NOTES:

- Maximum spindle speed specified by G50 can be displayed on the universal display. Refer to 4.3.1 Display of Command Data.

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

- The G50 function is effective for S and four digits designation but not for S and two digits designation.

2.10.14 MULTIPLE REPETITIVE CYCLES (G70 THROUGH G76)+

1. General

This option makes program simple and short. for instance, both stock removal and finishir are performed only by commanding the finist. ing work shape. .

Table 2.10.14.1 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		

A G70 through G76 are in * group and non-modal.

B. The program of finishing shape specified by G71, G72 and G73 are stored in memory. The memory capacity for the finishing shape is 25 blocks.

Program of finishing shape ≤ 25 blocks

Note: When G11 and G12 are used, each block containing G11 or G12 must be counted as two blocks.

C.

```
G71  Pns  Qnf . . .  *  ———————— Stock removal in turning
Nns . . .  *
  .
  .                    ———————— Finishing shape program (A)
  .                             (25 blocks maximum)
Nnf . . .  *
G70  Pns  Qnf . . .  *  ———————— Execution of finishing cycle
G72  Pns' Qnf' . . .  *  ———————— Stock removal in facing
Nns' . . .  *
  .
  .                    ———————— Finishing shape program (B)
  .                             (25 blocks maximum)
Nnf' . . .  *
```

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 (G).

D. G code of A group should be commanded again in the block after the performance of the cycles G70 through G76.

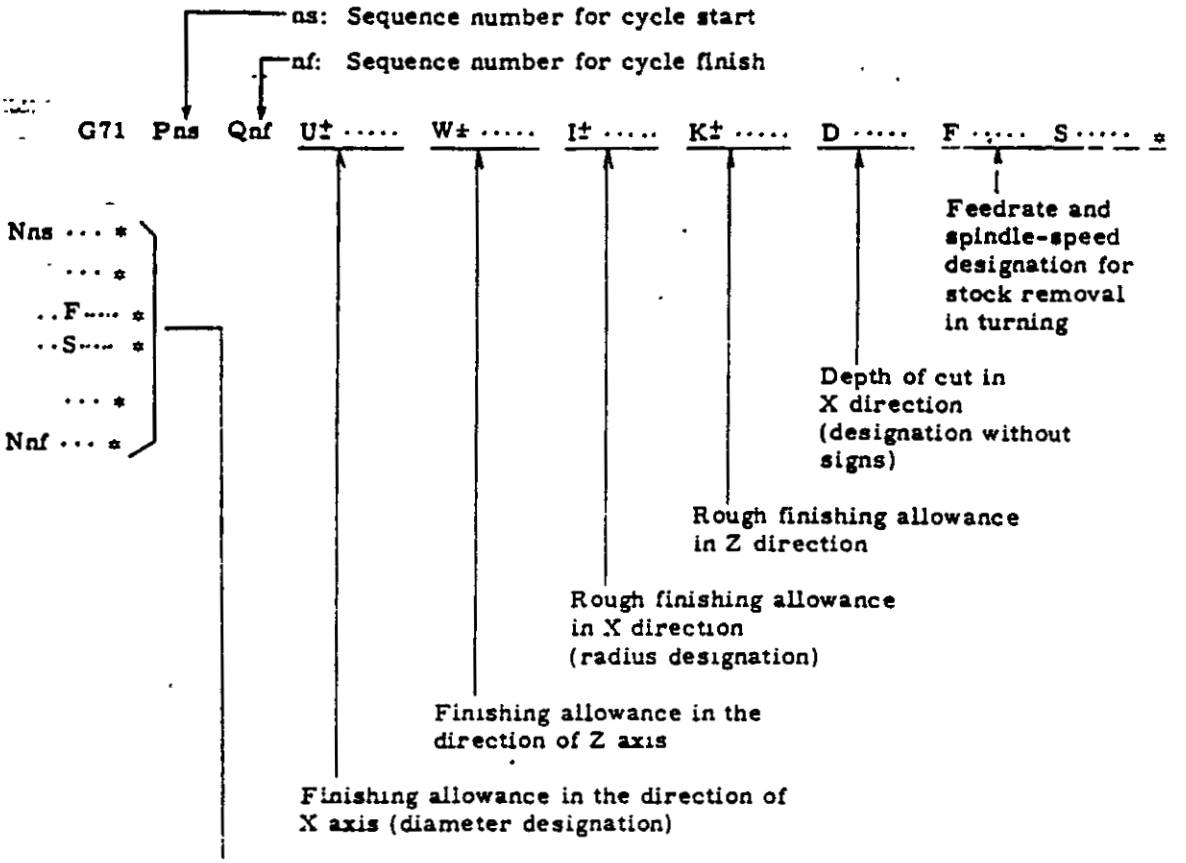
E. Tip nose radius compensation can be effective for the cycles G70 through G73.

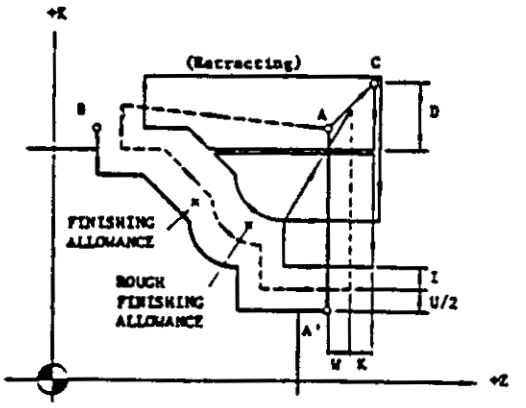
F. Tip nose radius compensation is ineffective for the cycles G74 through G76 The compensation is ignored

2. Stock removal in turning (G71)

Stock removal in turning with the finishing allowance remained uncut can be commanded by G71.

A. Command format





Finishing shape program
Stock removal cycle } Start and finish at position A
Rough finishing cycle }

Fig. 2.10.14.1

In the case of I = 0 and K = 0 (or no designation), the rough finishing cycle is omitted.

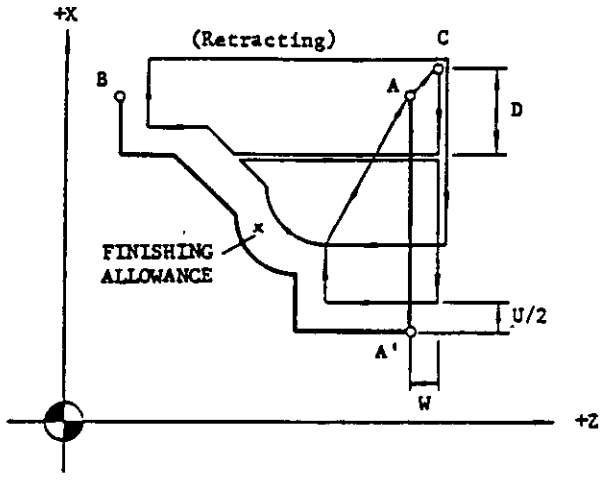


Fig. 2.10.14.2

The retracting motion is performed by G00 command. Traverse rate for cutting of D is determined according to the program of $\overrightarrow{AA'}$ (G00 or G01)

B Rules in programming G71

(1) 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.

(2) Finishing shape program must be programmed immediately after the block containing G71. Even a block containing only EOB between them causes an input error.

(3) The following should be taken into consideration in programming the start block (Nns) and the end block (Nnf) of a finishing shape program.

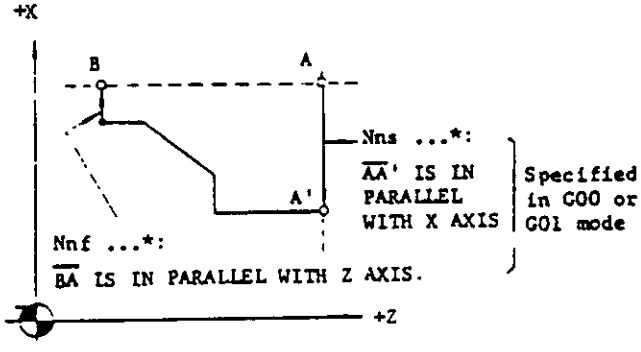


Fig. 2.10.14.3

(4) The tool path of finishing shape should be programmed to be monotonous increase or decrease in X and Z coordinates.

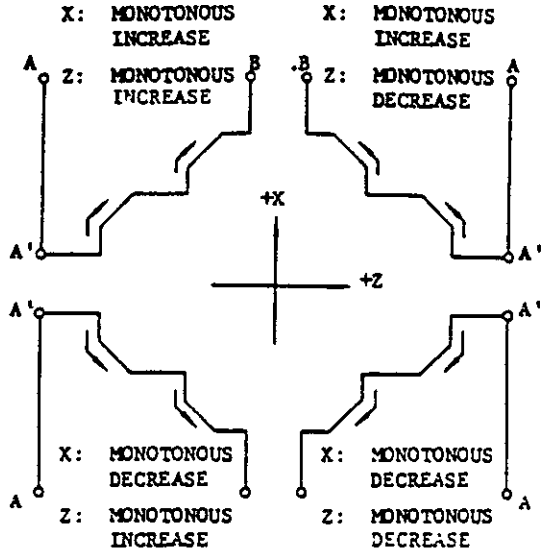


Fig. 2.10.14.4

Thus, the following tool path cannot be programmed.

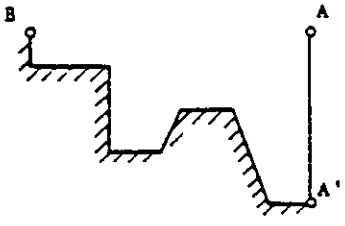


Fig. 2.10.14.5

NOTES

(1) 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.

(2) Table 2.10.14.2 shows the G codes that can be specified in the program of finishing shape excluding the blocks of Nns and Nnf.

Table 2.10.14.2

Usable G code	Remarks
G01, G02, G03, G22, G23	M52 should be used for cornering.
G11, G12	A block containing these codes must be counted as two blocks.

(3) † When a program has entered the tip 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)

(4) 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.

EXAMPLE (The case with tip nose radius compensation)

```
    (G99)
N01 G50 X26000 Z22000 *
N02 G00 S200 M03 T0101 *
N03 G41 *                 . . . . . Compensation mode
N04 (G00) X14500 Z18000 *   . . . . . To A position
N05 G71 P6 Q13 U100 W50 I200 K200 D400 F30 S250 *
N06 G00 X4000 S500 *    . . . . . Cutting feed at rapid traverse
N07 G01 W-4000 F15 *
N08     X6000 W-3000 S350 *
N09 G12 W-2000 I500 *       . . . . . G12 (two blocks equivalent)
N10 G01 X10000 W-1000 S250 *
N11     W-2000 *
N12     X14000 W-2000 S200 *
N13     X14500 *
N14 G00 X26000 Z22000 T0100 *
N15 G40 *
```

Finishing shape program = 9 blocks

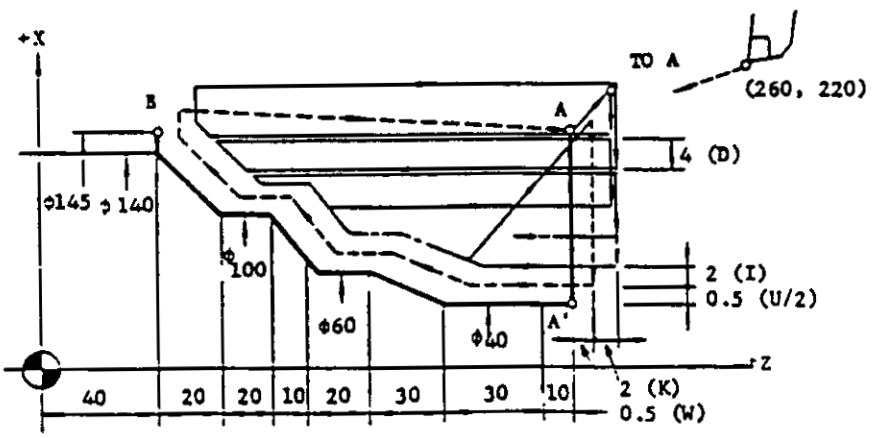


Fig. 2.10.14.6

3. Stock removal in facing (G72)

This cycle provides stock removal and rough finishing in facing with the finishing allowance remained.

G71 is for cutting in parallel with Z axis and G72 is for cutting in parallel with X axis.

A. Command format

G72 Pns Qnf U± ..... W± ..... I± ..... K± ..... D ..... F ..... S ..... *

Nns ..... *
..... *
F ..... *
S ..... *
..... *
Nnf ..... *

Significance of each address is the same as that of G71 command.

Depth of cut in Z direction (without signs)

Finishing shape program (25 blocks maximum)
Tool path: A → A' → B

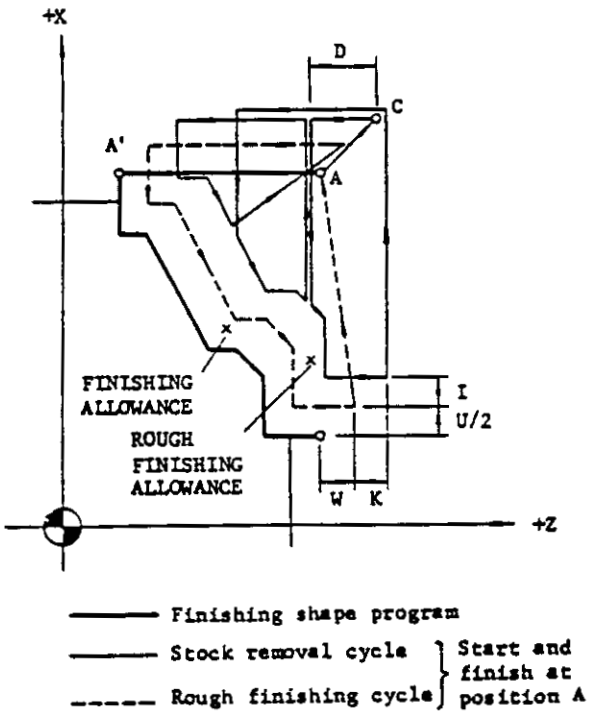


Fig. 2.10.14.7

In case of I = 0 and K = 0 (or no designation), the rough finishing cycle is omitted.

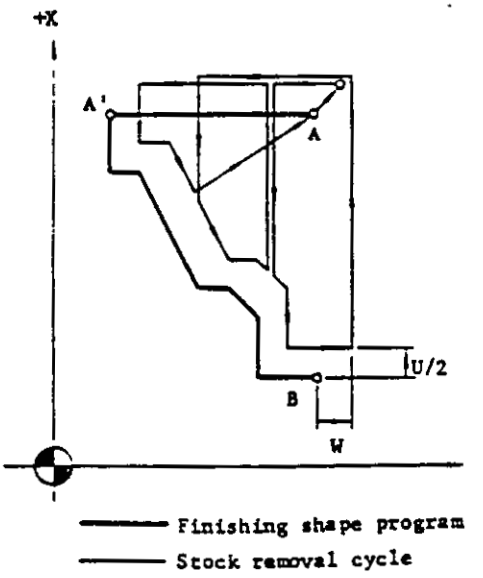


Fig 2.10.14.8

The retracting motion is performed by G00 command. Traverse rate for cutting of D is determined according to the program of $\overline{AA'}$ (G00 or G01).

B. Rules and cautions in programming G72

G72 is the same as G71 except that the tool cuts into the workpiece in parallel with Z axis. Refer to the description of G71.

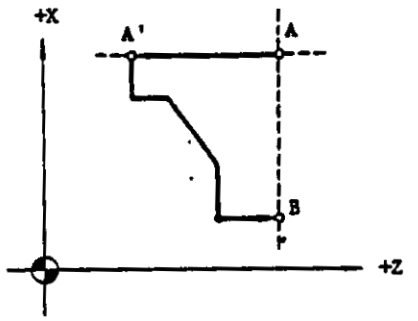


Fig. 2.10.14.9

EXAMPLE (I = 0, K = 0 without tip nose radius compensation)

```
N01 G50 X26000 Z6000 *
N02 G00 S350 M03 T0202 *
N03     X17000 Z500 *                    Stock removal in facing
N04 G72 P05 Q11 U60 W50 I0 K0 D400 F30 S400 *
N05 G01 Z-6000 F15 *          . . . . . Cutting feed
N06     X12000 *
N07     Z-5000 S450 *
N08     X8000 Z-4000 *
N09     Z-2000 S600 *
N10     X4000 Z0 *
N11     Z500 *
N12 G00 X26000 Z6000 *
N13     T0303 *               . . . . . Tool change for finishing
N14     X17000 Z500 *
N15 G70 P05 Q11 *             . . . . . Finishing cycle
```

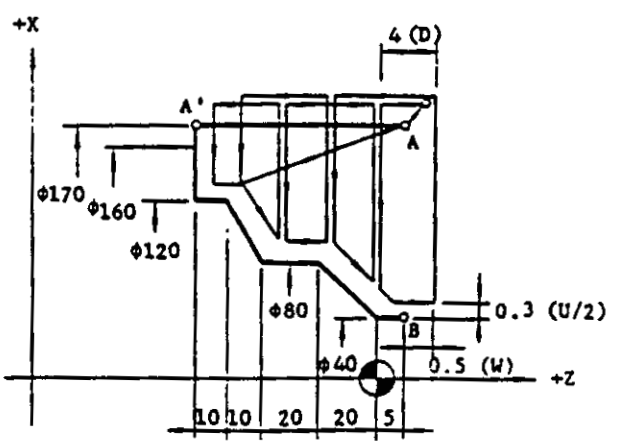


Fig. 2.10.14.10

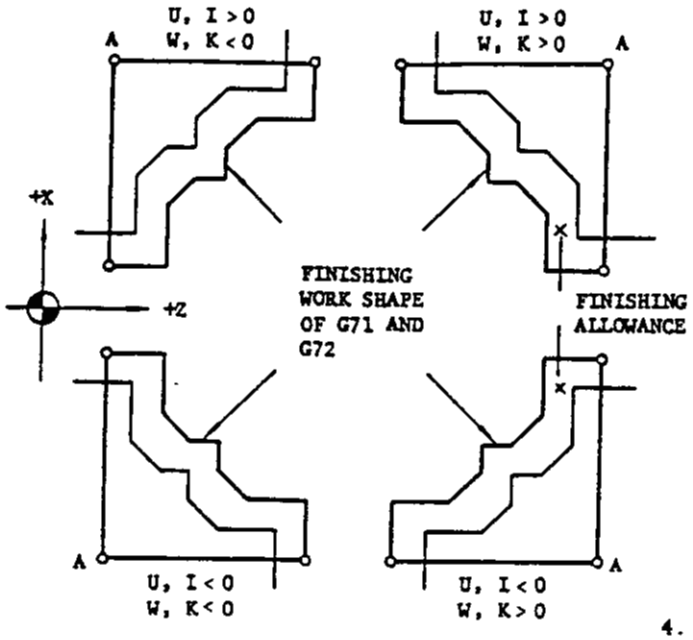
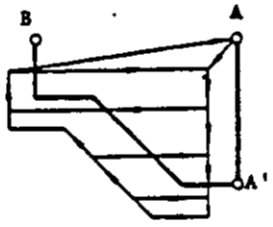


Fig. 2.10.14.11 Relation between Finishing Shape Program and Signs of Addresses U, W, I and K

A wrong sign will cause a gouging of the workpiece as shown below.



In the case that U, W, I and K < 0 are erroneously programmed.

Fig. 2.10.14.12

4. Pattern repeating (G73)

This cycle is useful for cutting the workpiece such as moldings and forgings whose cutting shapes are roughly made beforehand.

A. Command format

G73 Pns Qnf U± ..... W± ..... I± ..... K± ..... D ..... F ..... S ..... *

Nns ..... *
..... *
... F ... *
... S ... *
..... *
Nnf ..... *

Feedrate and spindle-speed designation

Number of cuttings
($1 \leq D \leq 127$, without signs)

Rough finishing allowance in Z direction

Finishing allowance in X direction (Radius designation)

Finishing allowance in Z direction

Finishing allowance in X direction (Diameter designation)

Finishing shape program (25 blocks maximum)
Tool path: A → A' → B (See Fig. 2.10.14.13.)
Sequence number should start with ns and end with nf.
F and S commands are effective only when G70 finishing cycle is executed.

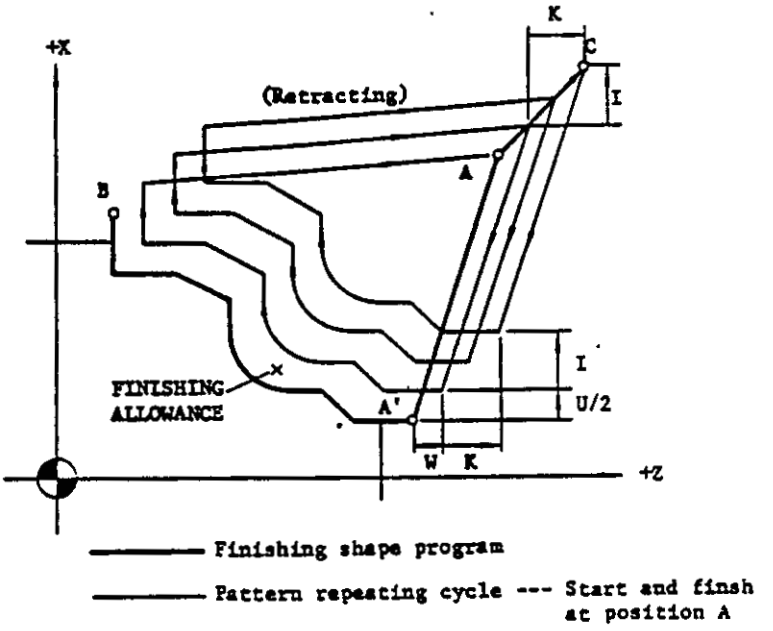


Fig. 2.10.14.13

B. Rules in programming G73

(1) Addresses U, W, I and K must be programmed with signs.

(2) 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 "15")
When D is 1, the cutting of I and K values is completed in a single cycle remaining finishing allowance.

(3) Finishing shape should be programmed immediately after the block containing G73.

(4) 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.

(5) Finishing shape program does not need to be monotonous increase or decrease in X or Z coordinate.

NOTES:

(1) When F and S functions are not specified in the block containing G73, the F and S functions specified in the preceding blocks are effective in the pattern repeating cycle.
F 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 2.10.14.3 shows the G codes which can be specified in the program excluding the blocks of Nns and Nnf.

Table 2.10.14.3

Usable G codes	Remarks
G01, G02, G03, G22, G23	Use M52 for cornering.
G11, G12	Counted as 2 blocks

(3) When I and K are 0 or not designated, it causes input error. (Alarm code "15" is displayed.)

(4) ΔI and ΔK (rough cutting allowance per cycle) are calculated as follows.

$$\Delta I = \frac{I}{D-1}, \quad \Delta K = \frac{K}{D-1}$$
$$\text{where} \quad D \geq 2$$

Note that the control ignores the value below 0.001 millimeter. As a rule, the program should be made so that ΔI and ΔK are not smaller than 0.001 millimeter.

(5) Processing of ΔI and ΔK

EXAMPLE 1

In case of I = 0.005 mm, K = 0.005 mm, D = 7

$$\Delta I = \frac{0.005}{7} = 0$$
$$\Delta K = \frac{0\ 005}{7} = 0$$

} Input error occurs.

EXAMPLE 2

In case of I = 0.01 mm, K = 0.01 mm, D = 7

$$\Delta I = \frac{0.01}{7} = 0.001\ \text{mm}$$

$$\Delta K = \frac{0.01}{7} = 0.001\ \text{mm}$$

Therefore, the cutting allowance of each cycle is as follows.

- 1st to 6th cycle ... ΔI = ΔK = 0.001 mm
  7th cycle ..... ΔI = ΔK = 0.004 mm

(6) *When the program has entered the tip nose radius compensation mode before G73 is commanded, the compensation is effective for all cycles of G73.

EXAMPLE 2

```
N10 G50 X26000 Z22000 *
N11 G00 S350 M03 T0303 *
N12     X22000 Z16000 *
N13 G73 P14 Q19 U200 W100 I800 K800 D3 F30 S400 *
N14 G00 X8000 W-4000 *
N15 G01 W-2000 F15 S600 *
N16     X12000 W-1000 S500 *
N17     W-2000 *
N18 G22 X16000 W-2000 R2000 S300 *
N19 G01 X18000 W-1000 *
N20 G00 X26000 Z22000 *
```

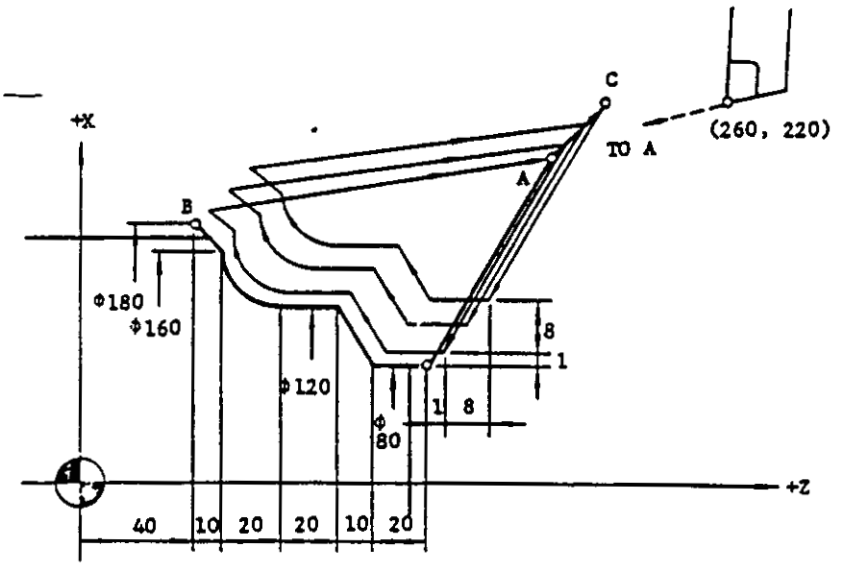


Fig. 2.10.14.14

5. Finishing cycle (G70)

After rough cutting of G71, G72 and G73, the finishing cutting can be made by the commands following G70.

A. 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. When some finishing shape program has been commanded, the last one which is retained in the memory is effective. The finishing cycle is completed to return to position A by G00 command.

B F and S functions specified in the finishing shape program are effective in the finishing cycle.
F and S functions for rough cutting specified in the block containing G71, G72 or G73 are ignored in the finishing cycle.

C. 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.

Table 2.10.14.4

Inhibited command and operation	Result
M02 and M30 commands with internal reset	Finishing shape program in the memory are eliminated.
Reset operation	

NOTES:

(1) For the sequence number ns for cycle start and nf for cycle finish, the following case causes input error (Alarm code "15" is displayed.)

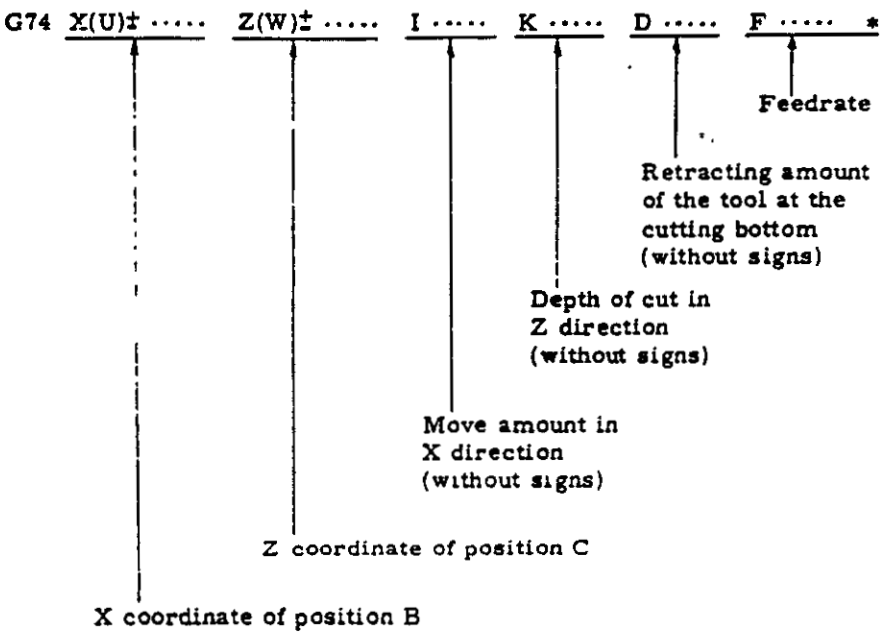
When the sequence numbers ns and nf of G70 are not commanded in the finishing shape program
When the sequence number ns of G70 is commanded in advance of sequence number ns, or ns = nf

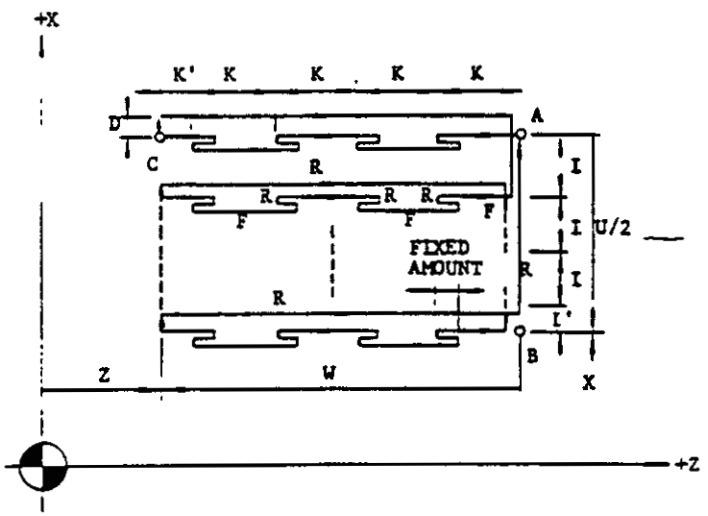
(2) When the program has entered the tip nose radius compensation mode before G70 is commanded, the compensation is effective for G70.

6. Peck drilling in Z axis (G74)

This command permits the operation of peck drilling with pecking motion in parallel with Z axis.

A. Command format





R: Rapid traverse

F: Specified by F code

Fig. 2.10.14.15

The cycle starts and finishes at position A.
The fixed amount of pecking is set by the parameter No. 74.

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 = 0 is programmed 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.

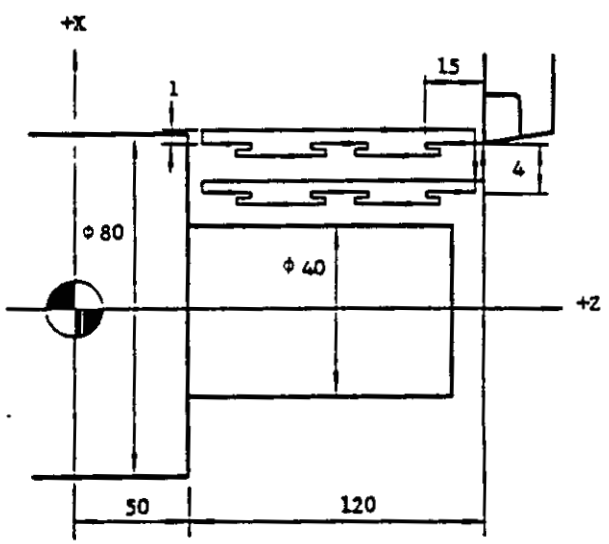
(6) If X(U), I or D is omitted, only one-cycle operation is made in the direction of Z axis, which is used for drilling.

(7) When the contents of parameter No. 74 are set to 0, the cutting is made at once to the cutting bottom without pecking motion.

(8) The tip nose radius compensation is ineffective for G74 and G75.

EXAMPLE

```
G74 X4000 Z5000 I400 K1500 D100 F25 *
```



R: Rapid traverse

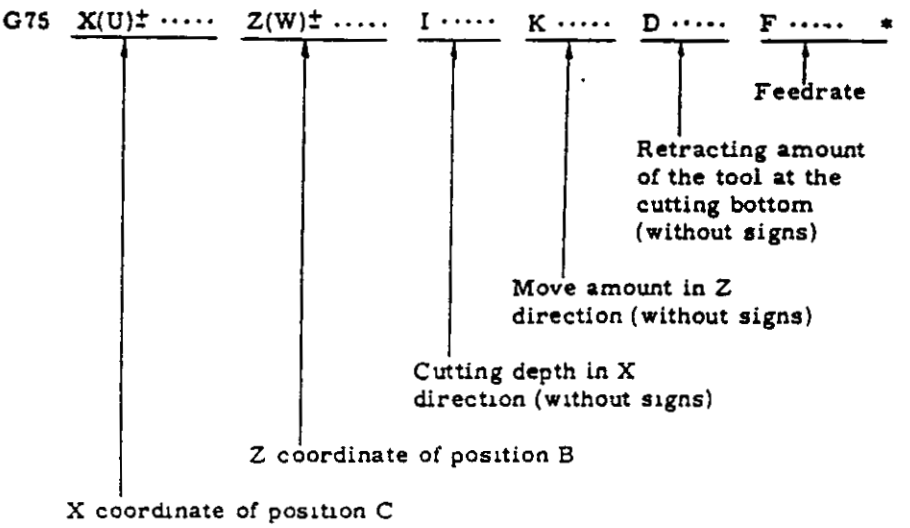
F: Specified by F code

Fig. 2.10.14.16

7. Grooving in X aixs (G75)

This command permits the operation of peck drilling with pecking motion in parallel with X axis.

A. Command format



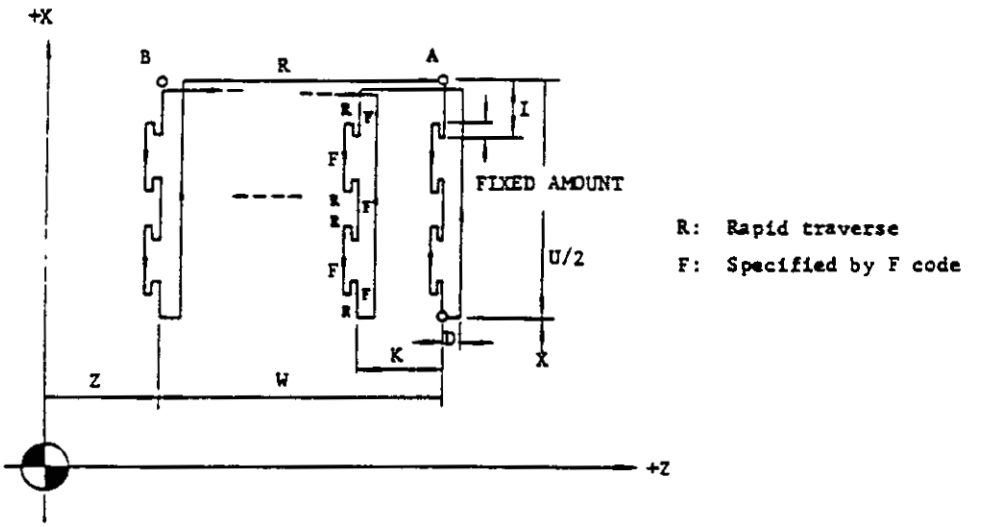


Fig. 2.10.14.17

The cycle starts and finishes at position A.
The fixed amount of pecking is set by the parameter No. 75.

NOTE: 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 the description of G74.

EXAMPLE

G00 X8600 Z7000 *

G75 X5000 Z4000 I600 K400 (D0) F20 *

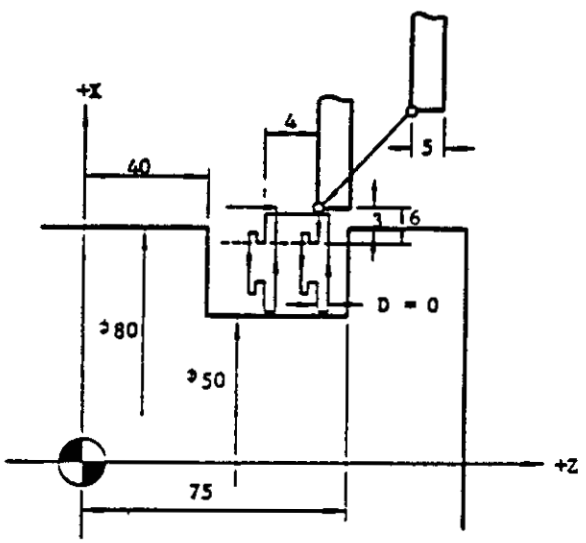


Fig. 2.10.14.18

8. Automatic threadcutting cycle (G76)

This cycle provides automatic cutting of straight and taper threadings along the angle of thread.

A. Command format

G76 X(U)± ····· Z(W)± ····· I± ····· K ····· D ····· {F ····· / E ·····} A ····· *

- A: Angle of thread (deg)
- F / E: Lead of thread
- D: First depth of cut (without signs)
- K: Height of thread (without signs)
- I±: Distance of tapered section (radius designation)
- Z(W)±: Z coordinate of position C
- X(U)±: X coordinate of position C

The sign of figure following the address I is decided by the direction of position B' viewed from position C.

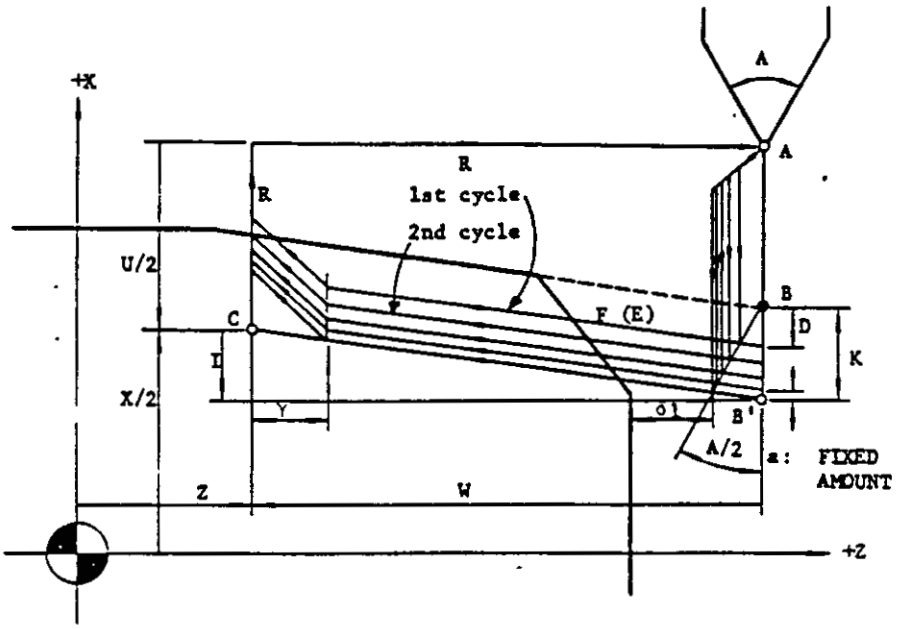


Fig. 2.10.14.19

The following shows the cutting position around point B (In case of taper thread).

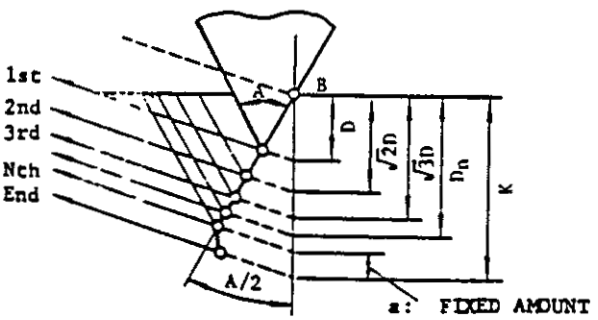


Fig. 2.10.14.20

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 parameter No. 76.

<Straight thread>

When the address I is 0 or not designated, a straight thread is cut as shown below.

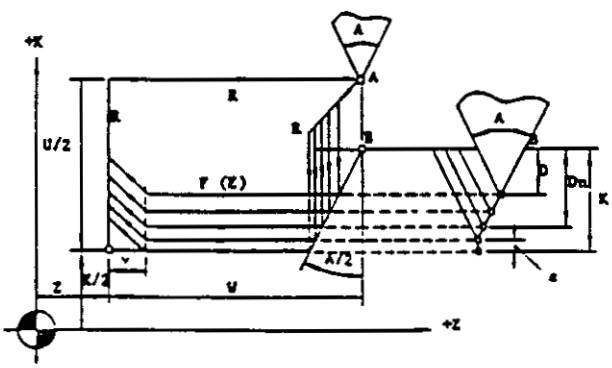


Fig. 2.10.14.21

B. Rules in programming G76

(1) If M23 is commanded before G76, the thread is chamfered. If M24 is commanded, the thread is not chamfered.

(2) Each cycle in G76 mode is similar to threading cycle G92. Refer to precautions on threading cycle G92.

(3) The depth of cut D in the first cycle is restricted by height of thread K as follows.

$$\frac{1}{6}K \leqq D \leqq K$$

Addresses K and D must be programmed without signs.

NOTES:

(1) When taper threading is commanded with effective angle but 0, X coordinate of threading start position does not meet with the depth of cut.

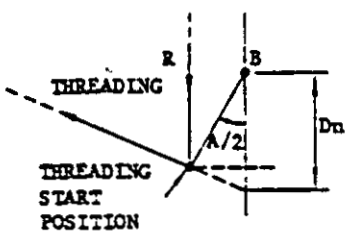


Fig. 2.10.14.22

(2) If the thread angle other than the above listed (0°, 29°, 30°, 55°, 60°, 80°) is arbitrarily commanded, the bigger and nearest angle is selected.

EXAMPLE

(Command) A15 ⟶ (Execution) A29

When A > 80°, A80 is executed.

(3) 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 them is deducted from D. The depth of cut in first cycle never becomes larger than D.

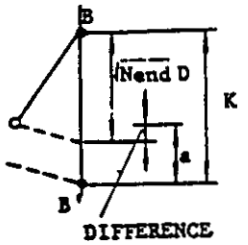


Fig. 2.10.14.23

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

(4) Tip nose radius compensation is not available for G76.

(5) G code of A group must be newly programmed in the next block of G76 cycle.

EXAMPLE

G76 ..... *

G00 M30 *

EXAMPLE

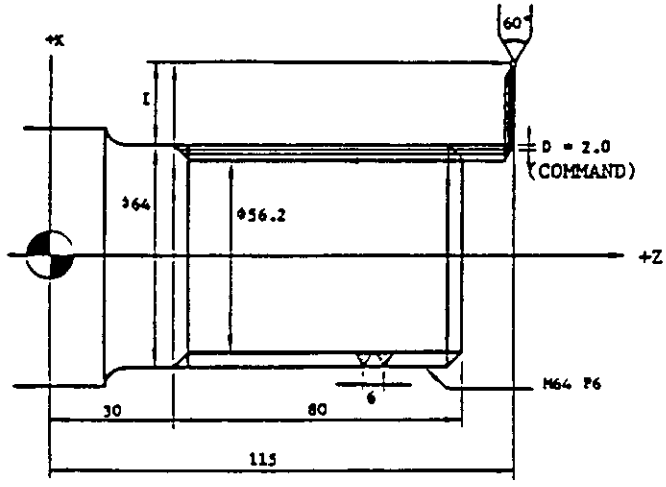
.
.
.

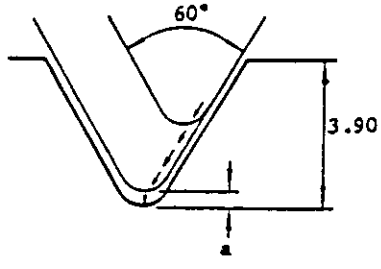
G00 X6600 Z11500 M23 ;

G76 X5620 Z3000 K390 D200 F600 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

Fig. 2.10.14.24

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 of $\sqrt{n}$end D and (K - a)

9. Precautions in programming G70 through G76

A. 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 G7 cannot be written in through the operation in MDI mode.

B. 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.

C 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. 2.10.14.11)

Command position of (X, Z) or (U, W) with respect to position A should be properly specified for G74 to G76

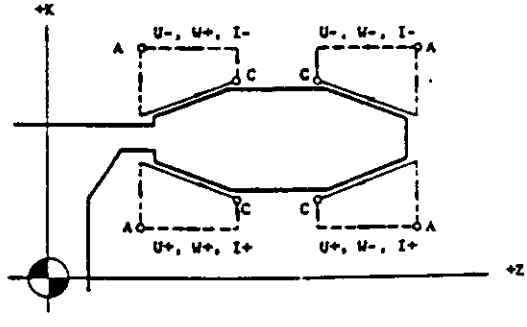


Fig. 2.10.14.25 Four patterns of G76

Table 2.10.14.5 Multiple Repetitive Cycles (1)

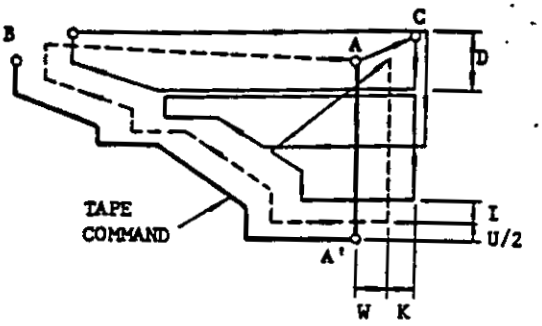
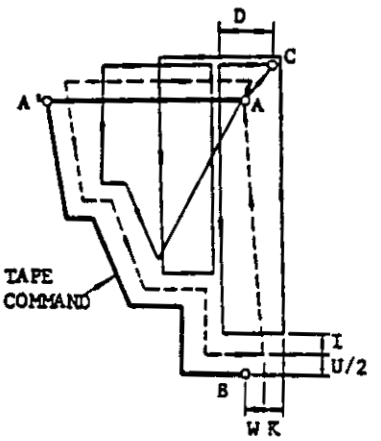
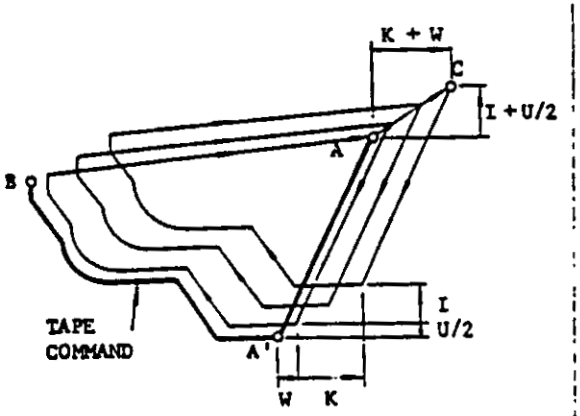
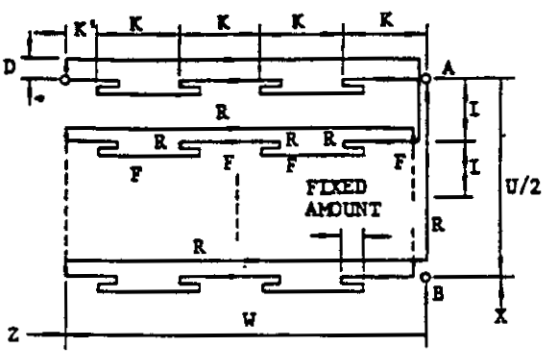
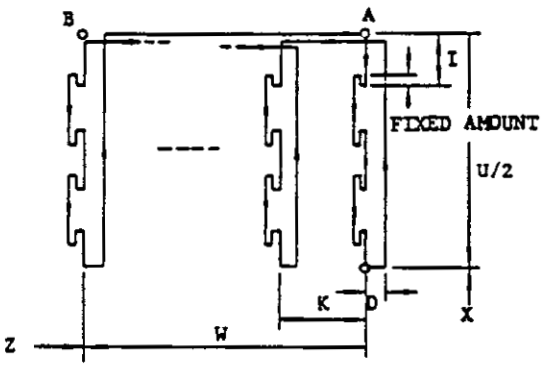
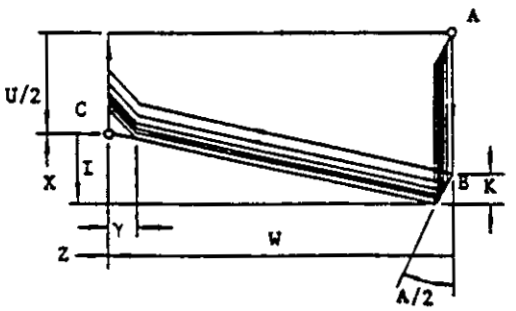
Code	Cutting Cycle	Command Format
G71 Stock removal in turning		G71 G72 G73 (ns) , (nf) P ··· Q ··· U ··· W ··· I ··· K ··· D ··· F ··· S ··· *
G72 Stock removal in facing		N (ns) ··· * F ··· * S ··· * M ··· * N (nf) ··· * Finishing shape program (A→ A'→B)
G73 Pattern repeating		P, Q ··· Sequence number for cycle start and finish U, W ··· Finishing allowance in X and Z directions (U· Diameter designation) I, K ··· Rough finishing allowance in X and Z directions for G71 and G72. Cutting allowance in X and Z direction for G73 D ··· Depth of cut at rough finishing for G71 and G72. Number of cuttings for G73 1 ≦ D ≦ 127 (D must be programmed without signs.) (U, W, I, K must be programmed with signs.)
G70	Finishing of N(ns) to N(nf)	G70 P ·· Q ··· *

Table 2.10.14.5 Multiple Repetitive Cycles (2)

Code	Cutting Cycle	Command Format
G74 Peck drilling in Z axis		G74 G75 } X(U)··· Z(W)··· I··· K··· D··· F··· * G74: X(U) ··· X coordinate of position B Z(W) ··· Z coordinate of position C I ····· Move amount in X direction K ····· Depth of cut in Z direction D ····· Retracting amount at cutting bottom
G75 Grooving in X axis		G75: I ····· Move amount in Z direction K ····· Depth of cutting in X direction The other addresses are the same with those of G74. I, K and D must be programmed without signs. Fixed amount is set by parameter.
G76 Automatic threadcutting		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 D ····· Depth of cut in first cycle A ····· Angle of thread (deg) (K and D must be programmed without signs. $\frac{1}{6}K \leqq D \leqq K$)

2.10.15 CANNED CYCLES (G90, G92, G94)

A series of basic lathe operations specified usually in four blocks, can be commanded in one block.

There are the following three canned cycles.

1. Turning cycle A (G90)

A. Straight turning cycle

G90 X(U) · · · Z(W) · · · F · · · *

The cycle ①to④shown below is executed by this command.

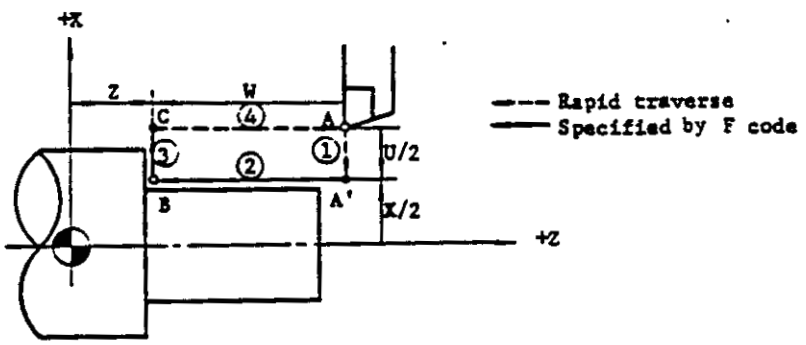
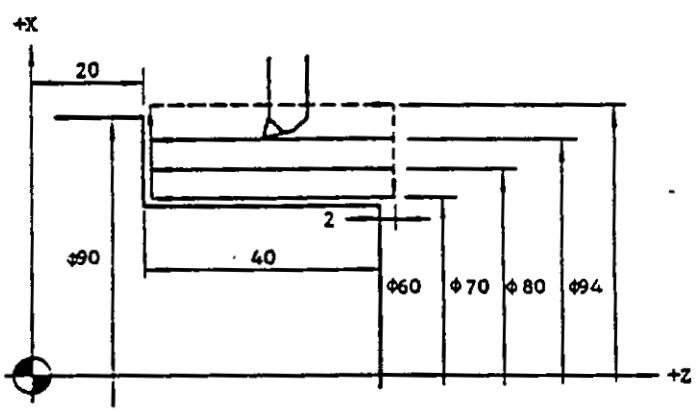


Fig. 2.10.15.1

Since G90 is modal, the cycle operation will be continued by specifying the depth of cut in the X direction in the following blocks as follows.

X(U) · · · *

X(U) · · · *




```
G00  X9400  Z6200  *

G90  X8000  W-4200  F30  *      ——  Start cycle
     X7000  *                   ]   Depth of cut is changed.
     X6000  *

G00 · · ·                       ——  Cancel cycle
```

Fig. 2.10.15.2

B. Taper turning cycle

G90 X(U)· · · Z(W)· · · I· · · F· · · *

The cycle ① to ④ shown below is executed by this command.

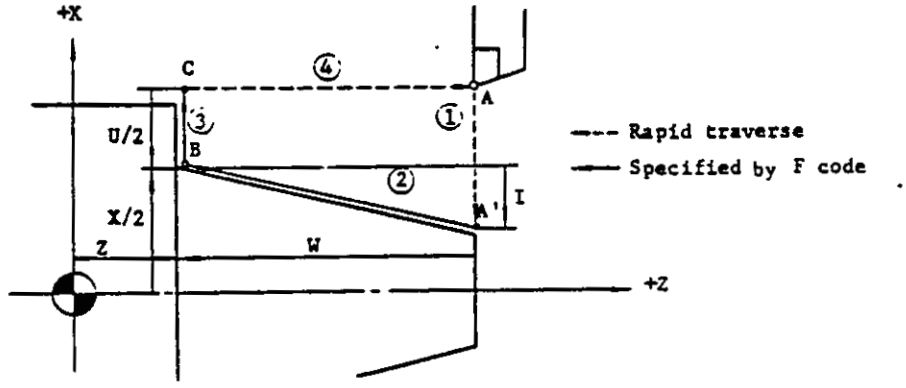


Fig. 2.10.15.3

The sign of figure following the address I is decided by the direction of position A' viewed from position B.

```
G00  X8700  X7200  *
G90  X8500  W-4200  I-1050  F25  *
     X8000  *
     X7500  *
     X7000  *
G00 · · ·
```

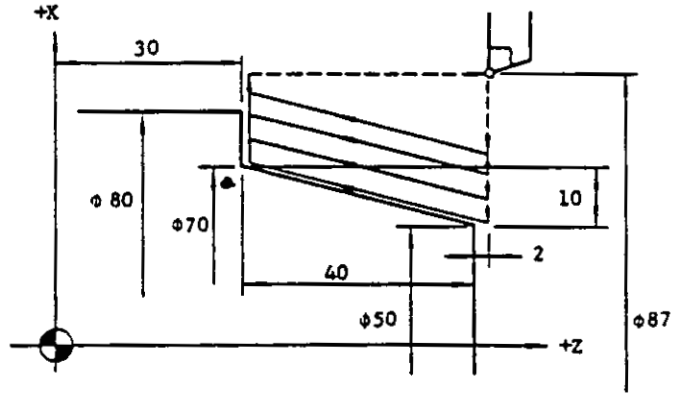


Fig. 2.10.15.4

C. Cautions in programming G90

T, S and M functions for G90 cycle must be specified beforehand in the preceding blocks.

```
G90 X· · · Z· · · I· · · F· · · *
    X· · · *
    X· · · *
    T0505 M05 * ← Error
G00 X· · · Z· · · *
```

Effective area of G90

```
G90 X· · · Z· · · I· · · F· · · *
    X· · · *
    X· · · *
G00 T0505 M05 * ← Correct
    X· · · Z· · · *
```

Effective area of G90

Specifying T, S and M functions requiring answer of FIN signal in the effective area of G90 causes error.

The effective area of G90 is from the block containing G90 to the one before the block in which the other G code of A group is specified. This rule also applies to the G92 and G94 described later.

In Single Block mode, the execution of G90 cycle stops after the completion of the cycle ① to ④.

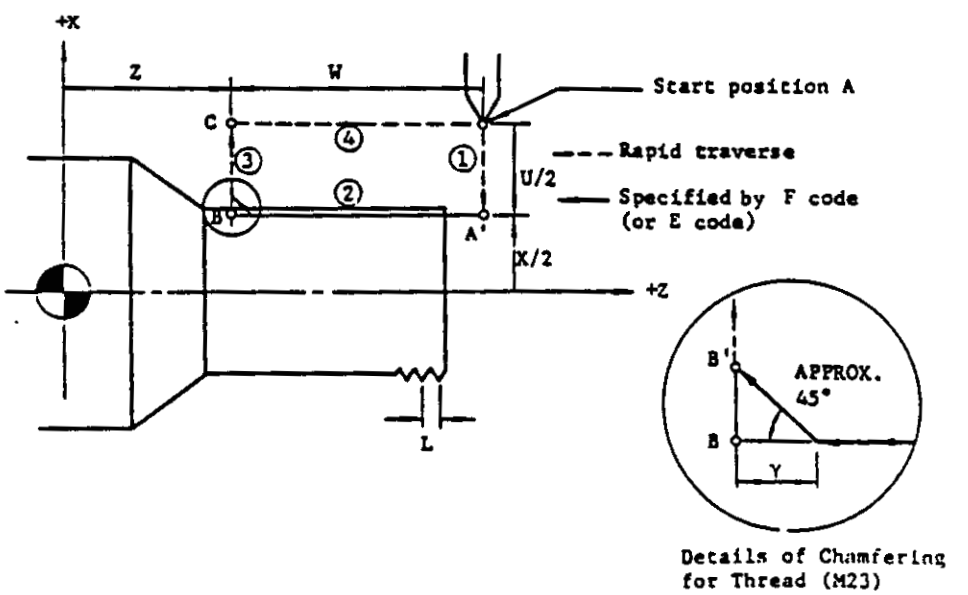
2. Threading cycle (G92)

A. Straight threading cycle

G92 X(U)· · · Z(W)· · · {F· · ·} *
{E· · ·}

Lead designation (L)

The cycle ① to ④ shown below is executed by this command.



Details of Chamfering for Thread (M23)

Fig. 2.10.15.5

(1) Chamfering

M23 (chamfering on) and M24 (chamfering off) are modal codes. When M23 is commanded in the previous block of G92, chamfering is performed. For the detail, refer to Fig. 2.10.15.5.

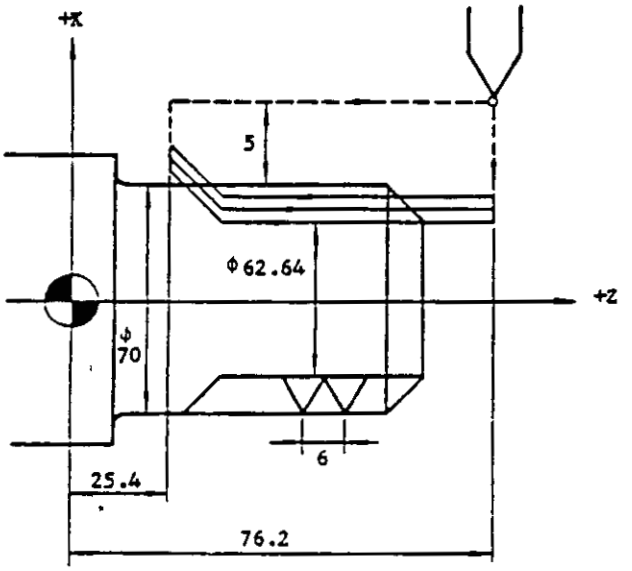
Chamfering width (γ) is set by the parameter No. 77. " γ " can be set in the range of 0.1L to 3.1L (L is specified lead in F or E). Normally, it is set to "0.8L." For the detail, refer to the machine builder's manual. When M24 is commanded in the previous block of G92, chamfering for thread is not performed.

(2) Restrictions on the threading cycle are the same with those of G32.

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) . . . *



Depth of cut

First cycle: 1.8 mm
Second cycle: 0.7 mm
Third cycle: 0.6 mm
Fourth cycle: 0.58 mm

```
G00  X8000  Z7620  M23  *  . . .  Chamfering on

G92  X6640  Z2540  F600  *
     X6500  *                     Four cycles of chamfering
     X6380  *
     X6264  *

G00  X10000  Z10000  M24  * . . .  Chamfering off
```

Fig. 2.10.15.6

B. Straight threadcutting cycle with angle

G92 X(U) . . . Z(W) . . . $\left\{\begin{matrix} F \cdot \cdot \cdot \\ E \cdot \cdot \cdot \end{matrix}\right\}$ *

This command permits the threadcutting along the angle of thread. The cycle ① to ④ shown below is executed.

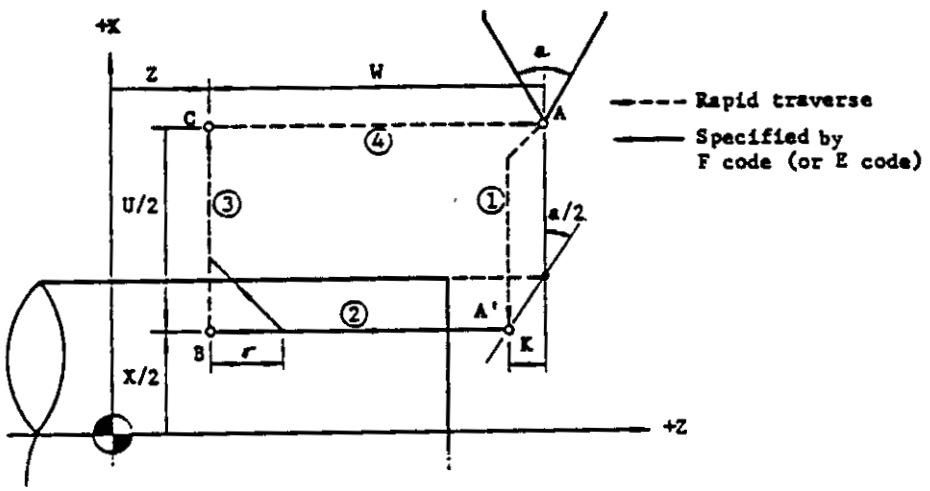


Fig. 2.10.15.7

The sign of figure following address K is decided by the direction of position A' viewed from position A.

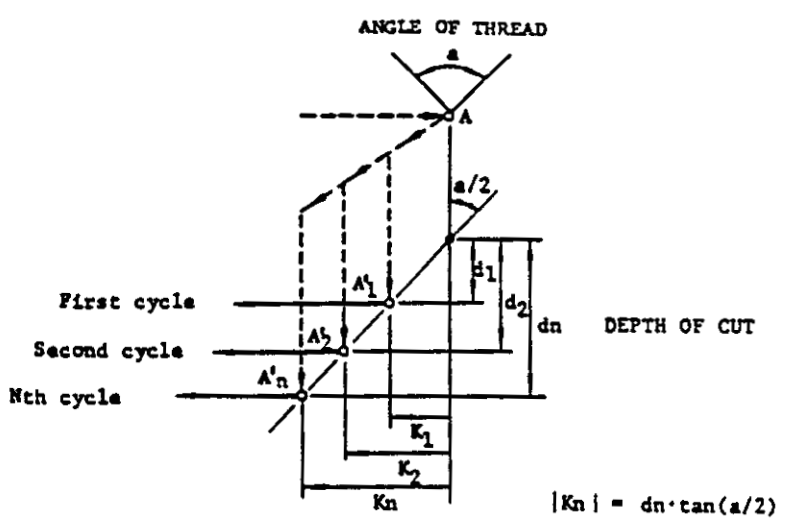
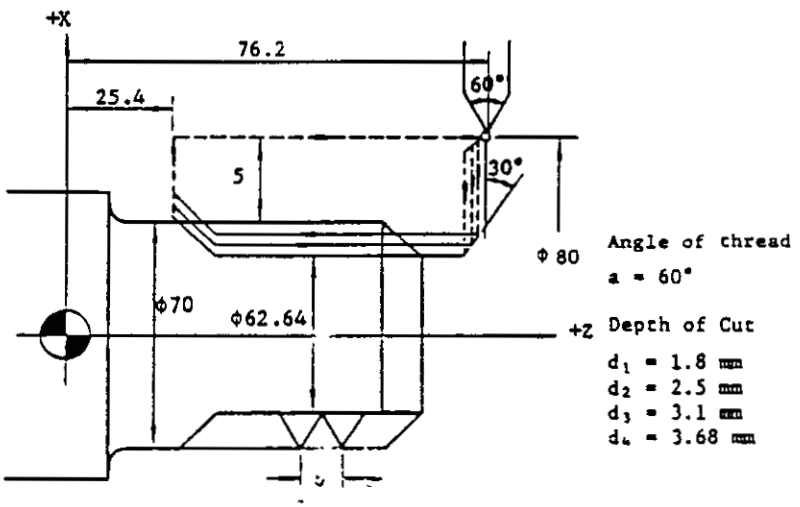


Fig. 2.10.15.8

For the threading along the angle of thead, the K for each cycle obtained from the above formula should be programmed.

a	tan (a/2)	a	tan (a/2)
29°	0.258618	60°	0.577350
30°	0.267949	80°	0.839100
55°	0.520567		



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

```
G00  X8000  Z7620  M23  *
```

```
G92  X6640  Z2540  K-87   F600  *
     X6500          K-144  *
     X6380          K-179  *
     X6264          K-213  *
```

```
G00  X10000  Z10000  M24  *
```

Fig. 2.10.15.9

C. Taper threading cycle

G92 X(U)··· Z(W)··· I··· $\left\{\begin{matrix} F\cdots \\ E\cdots \end{matrix}\right\}$ *

The cycle ① to ④ shown below is executed by this command.

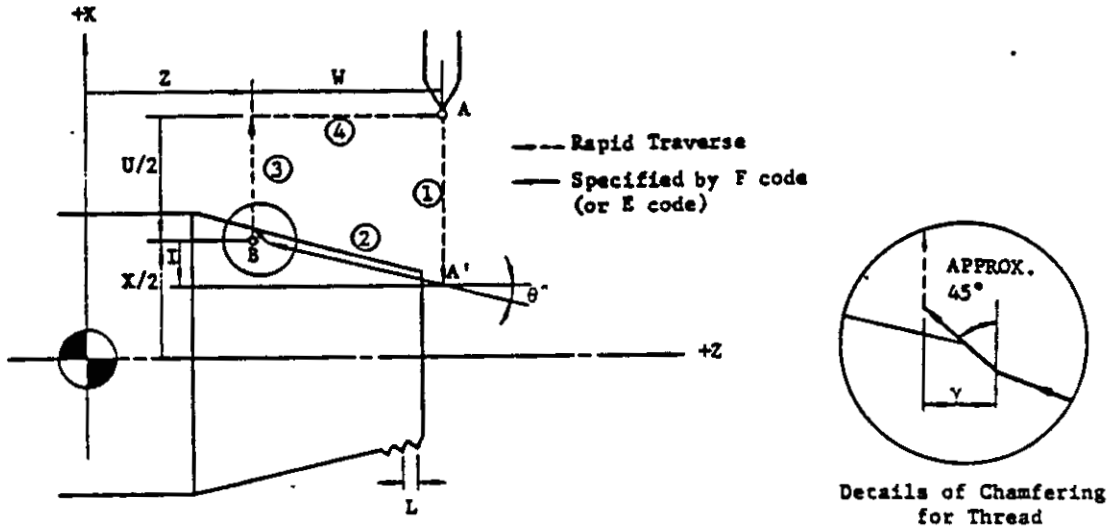
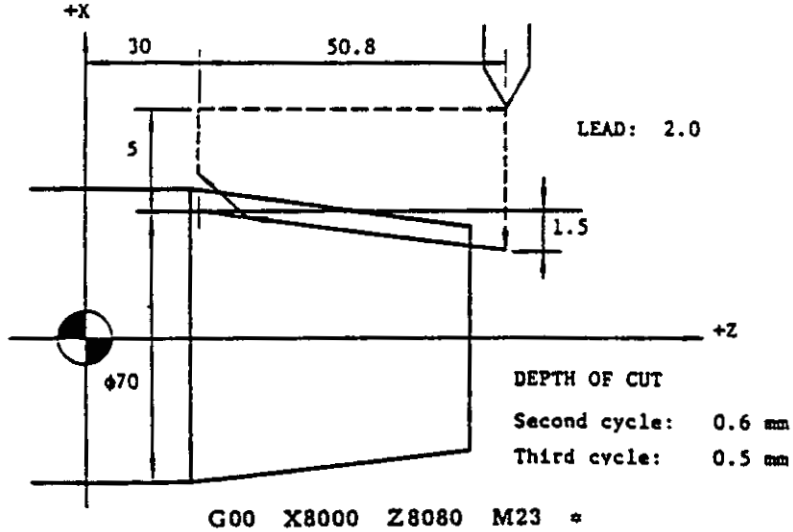


Fig. 2.10.15.10

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 direction in the trailing blocks as follows.

X(U) · · · *
X(U) · · · *




```
G00  X8000  Z8080  M23  *
G92  X7000  W-5080  I-150  F200  *
     X6880  *
     X6780  *
G00  X10000  Z10000  M24  *
```

Fig. 2.10.15.11

D. Taper threading cycle with angle (Cutting along the angle of thread)

G92 X(U)··· Z(W)··· I··· K···$\left\{\begin{matrix}F\cdots\\E\cdots\end{matrix}\right\}$*

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.

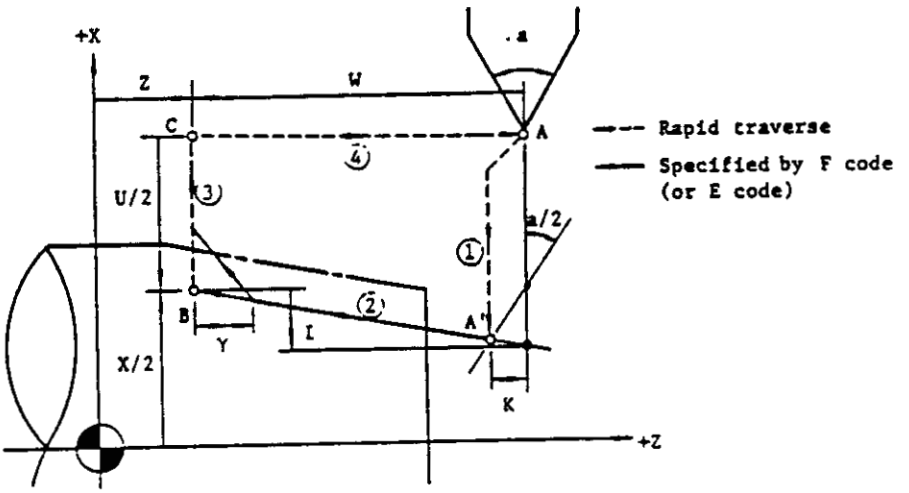


Fig. 2.10.15.12

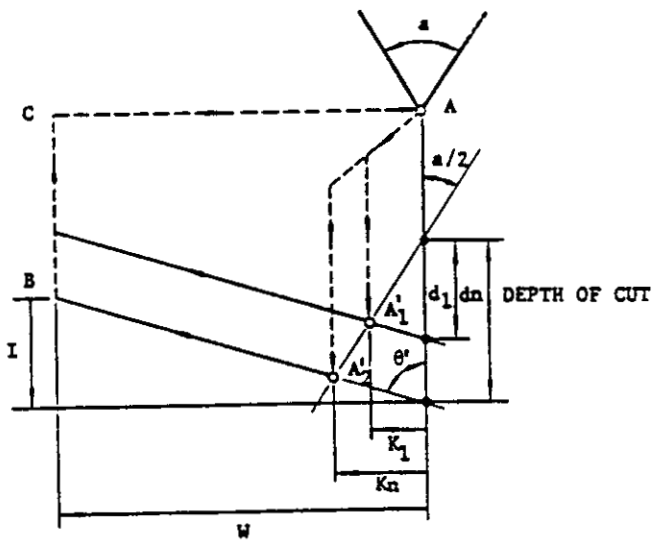


Fig. 2.10.15.13

For the threading along the angle of thread, the K for each cycle obtained from the following formula should be programmed.

$$|Kn| = \frac{dn \cdot \tan(a/2)}{1 \pm \left|\frac{I}{W}\right| \cdot \tan(a/2)}$$

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 Threadcutting Cycle). The control performs the above calculation automatically by the G76 command.

E. Cautions in programming G92

(1) Specifying T, S and M codes in the effective area of G92 causes format error.

(2) When the FEED HOLD button is depressed in G92 mode, the feeding is halted after the completion of threadcutting and retracting motion.

(3) In Single Block mode, the execution of G92 cycle is stopped after the completion of the cycle.

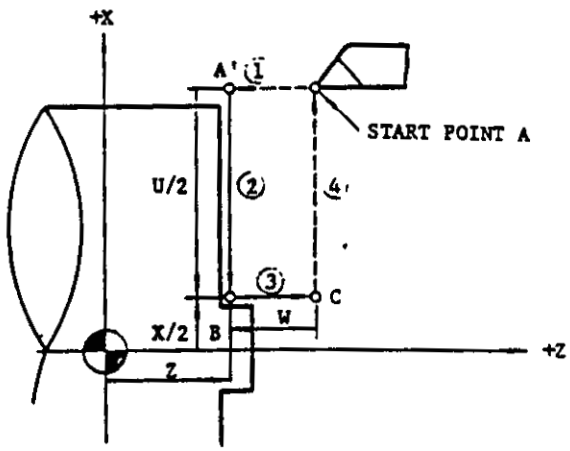
(4) Six angles of thread can be used in Multiple Repetitive cycle G76. In the G92 command, arbitrary angles of thread can be performed for threading.

3. Facing cycle B (G94)

A. Straight facing cycle

G94 X(U)··· Z(W)··· F··· ;

The straight facing cycle ① to ④ is executed by this command.



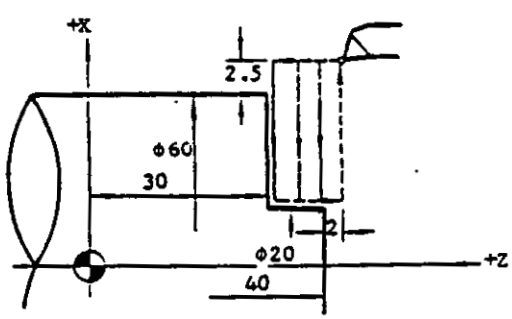
—— Rapid traverse
—— Specified by F code

Fig. 2.10.15.14

Since G94 is modal, the cycle operation will be continued by specifying depth of cut in the Z direction in the following blocks as follows.

Z(W) · · · ;

Z(W) · · · ;



G00 X6500 Z4200 ;

```
G94 X2000 Z3800 F35 ;
    Z3400 ;
    Z3000 ;
```

Three cycles by G94 command

G00 · · ·

Fig. 2.10.15.15

B. Taper facing cycle

G94 X(U)··· Z(W)··· K··· F··· ✲

The taper facing cycle ① to ④ is executed by this command.

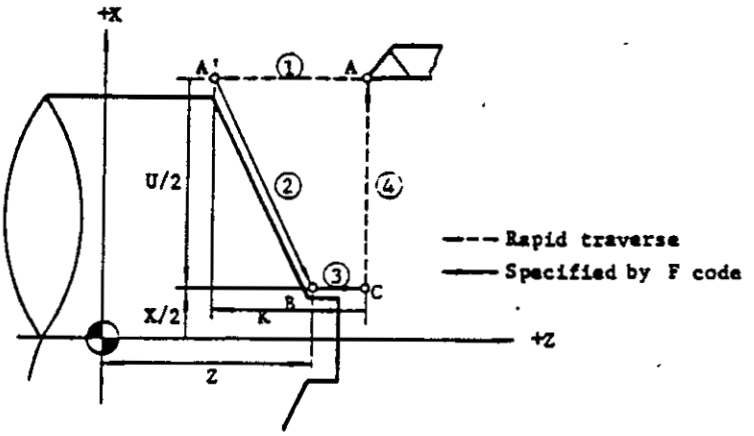
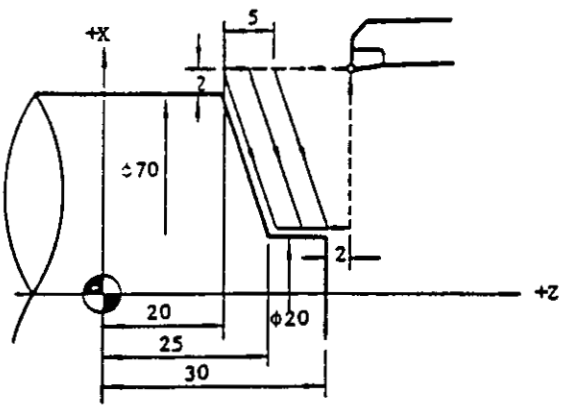


Fig. 2.10.15.16

The sign of figure following the address K is decided by the direction of position A' viewed from position B




```
G00   X7400   Z3200   ✲
G94   X2000   Z3000   K-529   F30   ✲
      Z2500   ✲
      Z2000   ✲
G00 · · ·
```

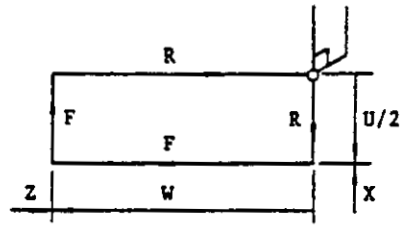
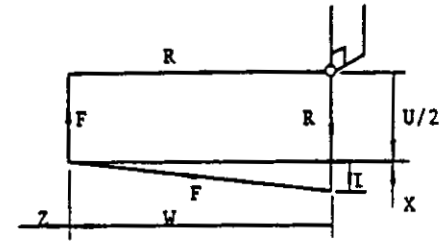
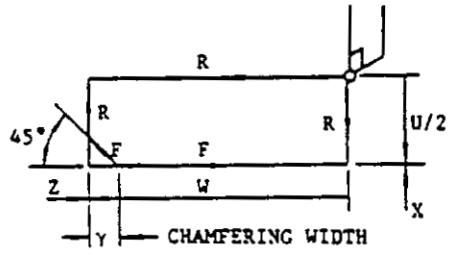
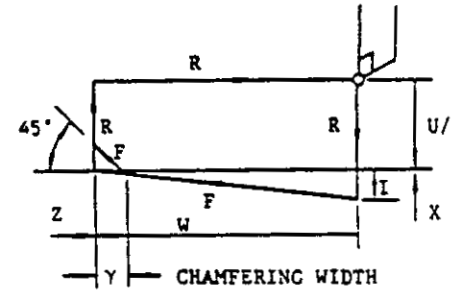
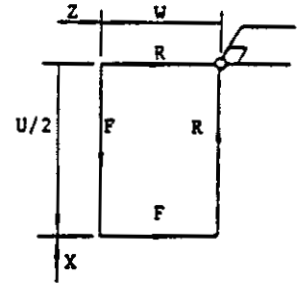
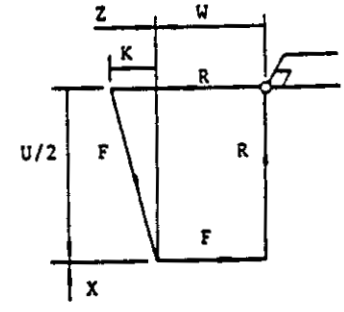
Fig. 2.10.15.17

C Cautions in programming G94

M, S and T functions cannot be specified in the effective area of G94.

In Single Block mode, the execution of G94 cycle is stopped after the completion of ① to ④.

Table 2.10.15 Canned Cycles

Code	Straight Cycle	Taper Cycle
G90	G90 X(U)··· Z(W)··· F··· * 	G90 X(U)··· Z(W)··· I··· F··· * 
G92	G92 X(U)··· Z(W)··· F··· * (E) 	G92 X(U)··· Z(W)··· I··· F··· * (E) 
G94	G94 X(U)··· Z(W)··· F··· * 	G94 X(U)··· Z(W)··· K··· F··· * 

2.10.16 CONSTANT SURFACE SPEED CONTROL (G96, G97)*

1. By using G96 with S and four digits command, the spindle speed is controlled to make surface speed constant. By using G97 with S and four digits command, the Constant Surface Speed Control is cancelled and the spindle runs normally at the speed specified by the S command.

   G96 and G97 are modal G codes of C group. When power supply is turned on, G97 (Constant Surface Speed Control Cancel) is in effect.

2. Constant Surface Speed Control (G96)

   A. G96 S ··· (M03) *

   This command specifies the surface speed (m/min) directly by S and four digits. When inch input is selected by Inch/Metric Selection option, the surface speed is specified in Ft/min.

   By the command, the control calculates the spindle speed to keep the surface speed at the specified speed corresponding to the move of current position of tool on X axis (diameter of workpiece) and outputs the signal in each 100 msec. In the following blocks, the surface speed can be changed by S command.

   B For the machine whose gear ratio of the spindle can be changed, M code for changing gear ratio should be specified in the previous blocks of G96. For the details, refer to the machine tool builder's manual.

   EXAMPLE

   ```
   M41 * ... Example of M code
             for low gear
   G96 S100 M03 *
   ```

   C. G50 S ····· *

   This command directly specifies the maximum spindle speed (rpm) by S and four digits. The specified maximum spindle speed and surface speed can be displayed. Refer to 4.3.1 Display of Command Data. If the calculated value of spindle speed exceeds the maximum spindle speed specified, it is clipped at the specified value. This occurs by commanding too small X coordinates in G96 mode.

D. Cautions in programming G96

(1) Setting of the coordinate system

In G96 mode, the coordinate system should be set by G50 so that the spindle center line meets with the zero in X axis. By this setting, the X coordinate comes to be equal to the diameter of workpiece at the current cutting point.

Tool position offset in G96 mode

The calculation for constant surface speed is normally made with the current value of X coordinate shown below.

X-coordinate value (POS# value in X axis)
* Commanded value + Tool position offset value

Therefore, too large value of tool position offset causes incorrect surface speed control. The following procedure should be taken for correct control.

- The coordinate system should be set for each tool by the method described above.
- Tool position offset function should be used for offset of tool wear.

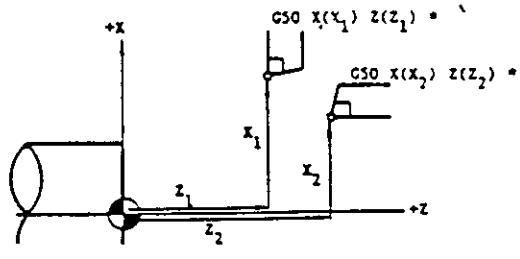


Fig. 2.10.16.1

NOTE. When the content of the parameter No. 05 are "1," the value obtained by the following formula is used for the calculation of spindle speed.

POS value is displayed by depressing POS key

X-coordinate value (= POS value# in X axis)
- Tool position offset value

(The above value is not displayed by depressing POS key.)

In this case, the tool position offset function can be used. But, it is recommendable to start, cancel and change the tool offset command in G97 (cancel) mode.

(3) The maximum spindle speed for each gear ratio is set with parameters.

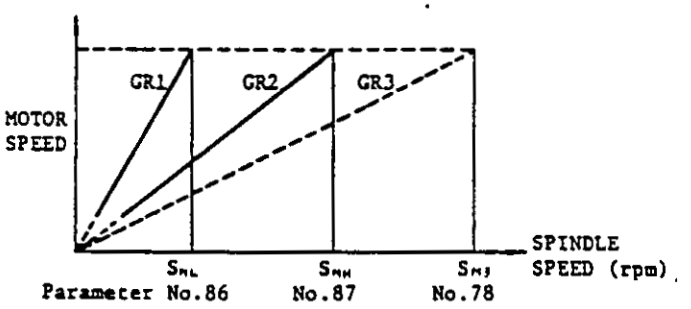


Fig 2.10.16.2

For the details of gear ratio, refer to the machine tool builder's manual.

(4) For the interface such as output signals and gear change signals, refer to the APPENDIX 2.

EXAMPLE

```
G00  X4000  Z500  *
     M42                          ---------- Example of gear Hi command
G50  S1000  *                     ---------- Spindle speed is clipped at 1000 rpm.
S96  S150  M03  *                 ---------- Constant surface speed 150 m/min.
G01  Z0  F15  *
     X8000  Z-3000  *
     W-1000  *                    Constant surface speed control
G22  X12000  W-2000  R2000  *
G01  U1000  *
     M05  *
G50  S2000  *                     ---------- Release of spindle speed clipping
G97  S500  *
```

POS value is displayed by depressing POS key

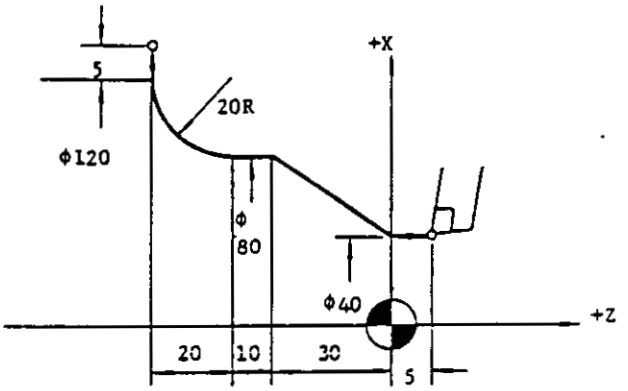


Fig. 2.10.16.3

3. Cancel of constant surface speed control (G97) · G99 *

G97 S ..... (M03) *

This command cancels the Constant Surface Speed Control. The spindle runs at the speed directly specified in rpm by the four digits following the address S

2.10.17 FEED FUNCTION DESIGNATION (G98, G99)

G98 designates the feed function specifying feedrate in millimeters per minute G99 designates the feed function specifying feedrate in millimeter per revolution.

G98 *

This command permits the control to execute the F command in millimeters (or inches) per minute.

This command permits the control to execute the F command in millimeters (or inches) per revolution.

Since G98 and G99 are modal, they are effective until erased by the command of new modal G code.

F code specified in the previous block will be cancelled when G98/G99 are switched. Therefore, new F command should be commanded

NOTE When power is turned on, G99 is in effect However, when the contents of parameter No 71 are "1," G98 is in effect.

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 3.1.1.

Before programming, select the code to be used.

Table 3.1.1 Tape Codes

EIA Code									Character		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		○	○	○	○	○	○	○	○	○

3.1.2 EIA/ISO/AUTO-SELECT

Before operating the NC system with punched tape, accomplish the changing operation of the tape code in accordance with 4.3.9 Writing Parameters.

Where the contents of parameter No. 80 are 1, the control will be automatically adjusted so as to read the tape with EIA or ISO coding. The tape code to be used is sensed in reading the first EOB code at the beginning of the tape with label skipped.

With the contents of parameter No. 80 set to 0, the control does not perform Auto-Select function. In this case, the tape code is determined according to the contents of parameter No. 82 as shown below.

0 means EIA code.

1 means ISO code.

If tape reader reads the NC tape of unset code, INPUT ERROR lamp on the operator's panel will be illuminated, and the universal display will indicate alarm code "12 "

NOTE: NC tape must be punched out with EIA or ISO code depending on the contents of parameter No. 82.

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.

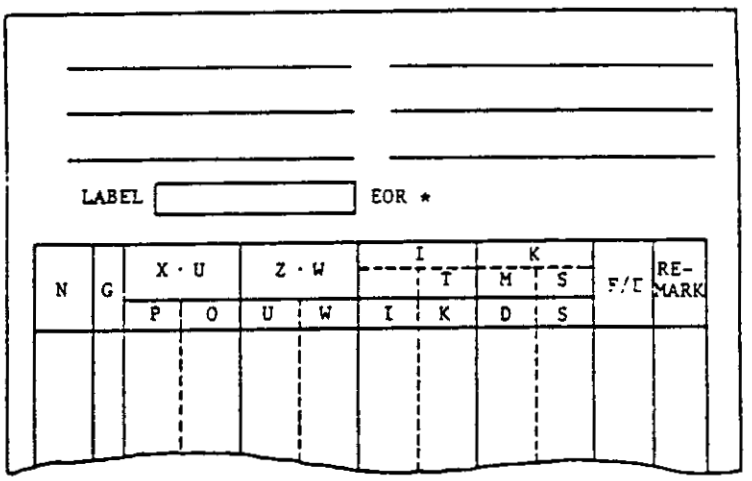


Fig. 3.2.1 An Example of the Process Sheet

3.2.2 GENERAL PROGRAM FORM

A part program will be generally made in the following form.

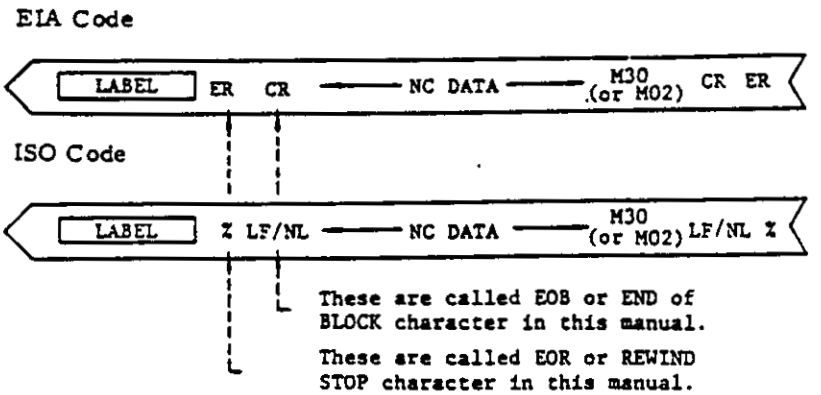


Fig. 3.2.2

Any LABEL can be written at the beginning of tape to classify easily the tapes. The label skip function has the data from LABEL to the first EOB code skip readily Therefore, the undesignated address or function characters can be used as LABEL. In addition, the modified code which disregards parity is also available.

EOR code at the next of LABEL means the stop point of tape rewinding.

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.

3.2.3 PRECAUTION IN PROGRAMMING

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.

A part program ends with the block including M02 (End-of-Program) or M30 (End-of-Tape).

When M02 or M30 is commanded, automatic operation# 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.

NOTE See 2.7 MISCELLANEOUS FUNCTIONS

Slash "/" character means the optional block skip function. This code must be surely punched out at the head of block (before the address N of sequence number). If this is punched out on the way of the block, this function is disregarded even if the optional block skip switch is ON.

The character specified on 2.1.2 Address and Function Characters should be used for programming, but others should not.

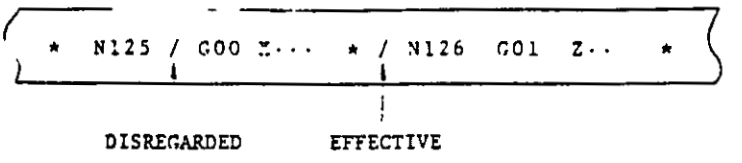
Where the tape vertical parity check (TV check) is made, number of characters in a block must be even. If odd, it should be made even by using "SP" character.

The disregarded characters such as "BS, Tab, SP, UC, LC and Del" should be avoided from the significant data area, if unnecessary.

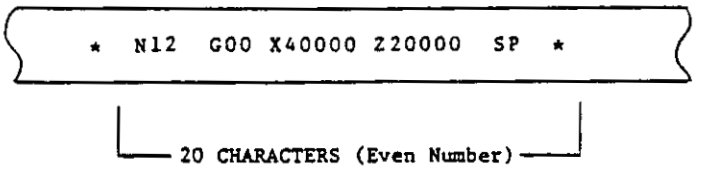
The maximum allowable number of characters in a block is 64. The disregarded characters such as "Del, BS and Tab" are not included in them.

"Automatic operation" means operation in TAPE, MDI or MEM mode

a. Position of Optional Block Skip



b. Making the Number of Characters even for TV Check



(Odd number causes tape vertical parity error.)

C. The Maximum Allowable Number of Characters in a Block

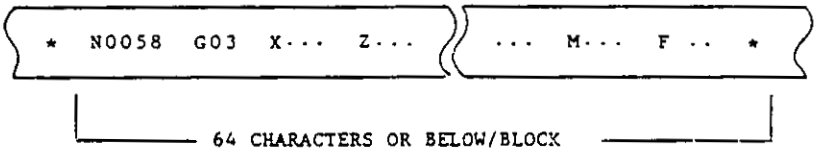


Fig 3.2.3

3.3 NC TAPE

3.3.1 PAPER TAPE

Eight-channel paper tape for computers complying with JIS#-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 PUNCHING OF NC TAPE

NC tape must be punched out with the tape puncher 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 CHECKING OF NC TAPE

NC tape can be checked by using the following function.

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

Japanese Industrial Standard

3.4 NC TAPE HANDLING

3.4.1 TAPE FOR SPLICING NC TAPE

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.

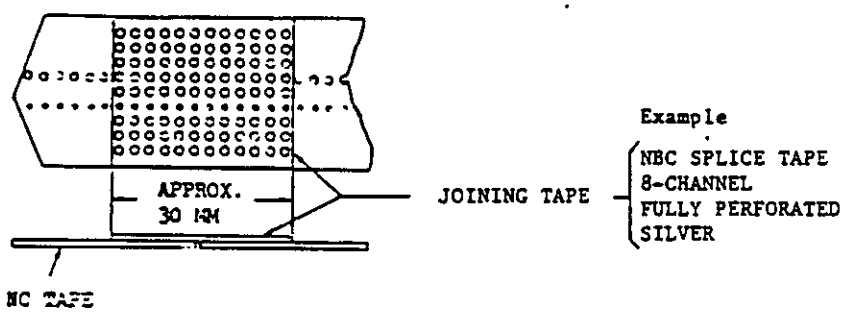


Fig. 3.4.1 Splicing of NC Tape

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.

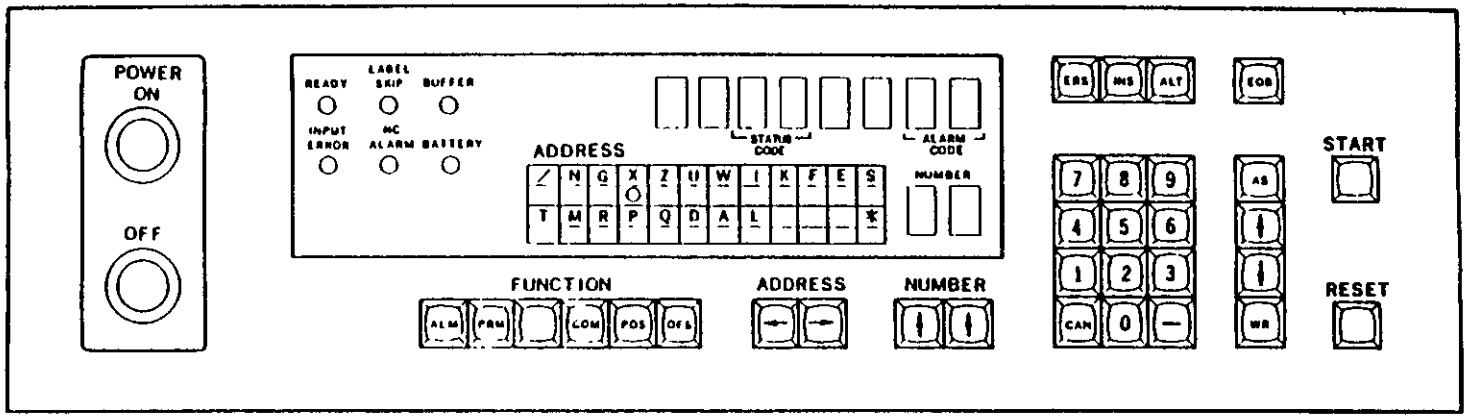


Fig. 4.1.0 Standard NC Operator's Panel

4. NC OPERATOR'S PANEL

4.1 PUSHBUTTONS, KEYS, AND LAMPS

4.1.1 POWER ON/OFF PUSHBUTTONS

· POWER ON pushbutton

To turn on the control: Depress the button first to turn on control power and depress it again to turn on servo power. Push the button to recover servo power after emergency stop.

· POWER OFF pushbutton

Depress it to remove servo power and control power.

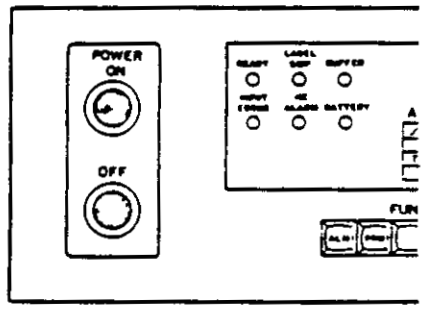


Fig. 4.1.1

4.1.2 INDICATING LAMPS

· READY lamp

It lights up when the control is ready to operate with power normally supplied to the control and servo units. While it remains off, the control cannot be operated either manually or automatically.

· LABEL SKIP lamp

It is on when the Label Skip function is effective when power has been turned on or the control has been reset. The Label Skip is the function that makes tape setting easy by ignoring all tape information until the first EOB is encountered. It goes off when EOB has been read.

In the MEM or EDT mode, the illuminated lamp indicates that memory or tape is rewound. This display is related to only tape or memory information but not to MDI operation.

· BUFFER lamp

It is on when data in the next block are held in the buffer register. It goes off when the buffer has been evacuated by depressing the CYCLE START or RESET button. In the automatic operation#, a block of data is read in advance, and the lamp goes on and off according to buffer storing conditions.

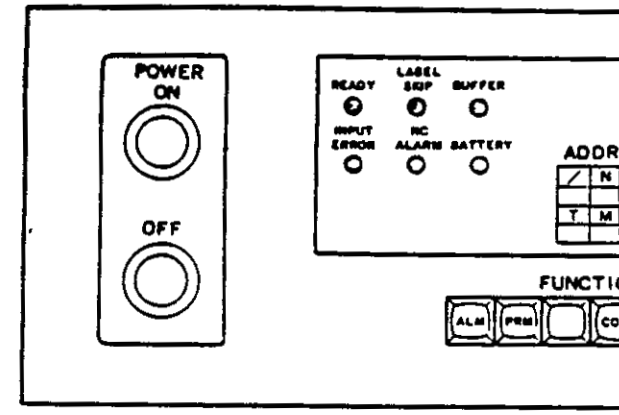


Fig. 4 1.2

· INPUT ERROR lamp

It goes on when an error is detected in input information. At that time, automatic operation is suspended immediately after the current block has been executed, and Cycle Start is then prevented. Possible causes for the lamp to go on are as follows:

- Tape format error
- Use of wrong characters
- Mispunched information on paper tape
- Dirty tape
- Misreading of paper tape reader
- Destroyed memory contents

Set the FUNCTION select key to ALM to obtain the detailed information of error in alarm code. The lamp goes off when the control has been reset.

Automatic operation is defined as the operation in auto-mode (TAPE, MDI, or MEM) and manual operation, as the operation in HANDLE, JOG, or RAPID mode in this manual

· NC ALARM lamp

It is on when an error other than input error mentioned above has been detected in the control. If the control is in automatic operation, it stops immediately or at the end of a block, depending on the error. Then Cycle Start is prevented.

The lamp goes off when the control has been reset after taking the correct measures according to the error. See 4.3.12 Alarm Code Display.

· BATTERY lamp

It is on when battery voltage is below a safe level. Then the battery must be replaced with a new one within a month. Contact the maintenance personnel for battery change. Battery is used for protection of parameters, tool offset values and tape memory at power OFF

4.1.3 FUNCTION SELECT KEYS

The key selects one of five functions for the operation of the display and MDI Pushing a key makes it light up.

· ALM (Alarm) key· To display an alarm code or I/O signal.

· PRM (Parameter) key· To display or write parameters. With parameter No. set at 00, operation time is displayed.

· COM (Command) key To display or manually write a command value for automatic operation.

· POS (Position) key: To display the current tool position.

· OFS (Offset) key· To display or write tool offset value.

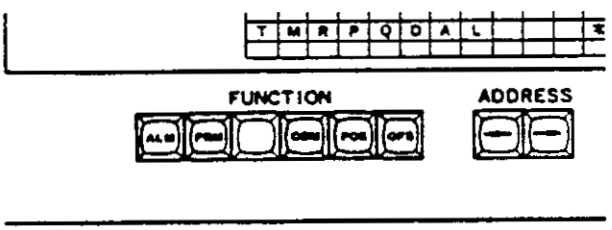


Fig. 4.1.3

4.1.4 ADDRESS SELECT KEYS AND DISPLAY

The keys select an address character to display or write command data.

· ⊟ Key

Each depression of the key moves the address indicator to the right by one address. If the key is held in for more than 0 5 second, the indicator will continue to move automatically until the key is released.

· ⊞ Key

Each depression of the key moves the address indicator to the left by one address. If the key is held in for more than 0 5 second, the indicator will continue to move automatically until the key is released.

Pushing the both ADDRESS keys makes the indicator go back to "X."

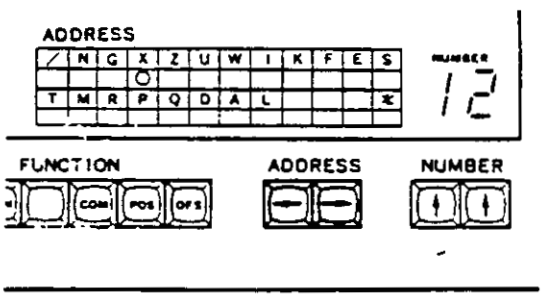


Fig. 4.1.4

4.1.5 NUMBER SETTING KEYS AND DISPLAY

The NUMBER keys are used to set a tool offset number or a parameter number when displaying or writing tool offset value or parameter.

· ⊞ Key at right

Each depression of the key increases the digital display by 1. If the key is held in for more than 0 5 second, the number will continue to increase automatically until the key is released.

· ⊡ Key at left

Each depression of the key increases the digital display by 10. If the key is held in for more than 0.5 second, the number will continue to increase automatically until key is released.

By pushing the both NUMBER keys, NUMBER is set back to "00."

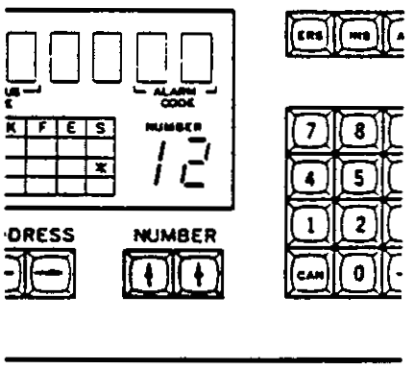


Fig. 4.1.5

4.1.6 DATA KEYS

Twelve keys of 0 through 9, - (minus), and CAN (cancel) are provided for manual data input. Any command value, tool offset value, and parameter data is built on the universal display. To clear the input value, use the CAN key.

4.1.7 WR (WRITE) KEY

Depress it to store the input data displayed into the buffer register.

4.1.8 AS (ADDRESS SEARCH) KEY

Depress it to start searching tape or memory contents. For details, see 4.3.14 Address Search.

4.1.9 SEQUENTIAL SEARCH KEYS ⊡ AND ⊡

Sequential search keys " ↓ " and " ↑ " located between AS and WR keys are to search memory or tape contents in the MEM and EDT modes.

" ↓ " Key is to search the data of one block before, and " ↑ " key is to search the data of one block advanced.

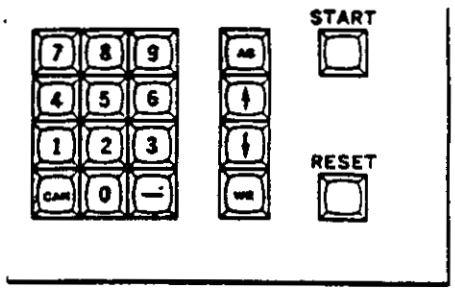


Fig. 4.1.9

4.1.10 ERS (ERASE), INS (INSERT), ALT (ALTER), AND EOB (END OF BLOCK) KEYS

These keys are for storing and editing a block of data and effective only in the EDT mode.

· ERS (erase) key

It is to delete a block of data in the memory in the EDT mode.

· INS (insert) key

It is to store a block of data into memory. Depress the key first, and it lights up indicating that the operator may start MDI operation. After a block is built through the DATA keyboard, depress the key again to store the built data into the memory. Then the indicating lamp goes off.

· ALT (alter) key

It is to modify address data in a block.

· EOB (end of block) key

It is to store a block of data written in the buffer register into memory in the EDT mode

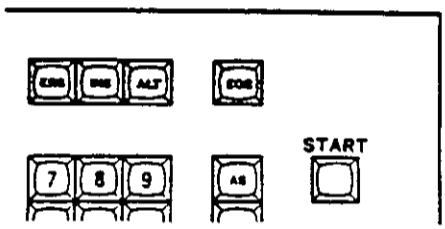


Fig. 4.1.10

4.1.11 START KEY

It is used to start tape punching and program storing from tape+ in the EDT mode. In the MDI mode, it may be used for Cycle Start operation. For details, refer to the machine tool builder's manual.

4.1.12 RESET KEY

The key resets the control.
Operations executed by RESET key (or remote reset button connected to ERS input terminal) are·

- Move command cancel
- Buffer register clear
- Alarm code release if the cause eliminated
- Tool offset cancel
- Miscellaneous function cancel
- Label Skip function ON
- Memory pointer rewind
- Sequence number reset
- RST signal transmission to the machine

The following data remain unchanged after depressing the RESET key (or remote reset button).

- Current position value of X-, and X-axis
- Modal G-codes (except G-codes of B group)
- F commands
- S 4-digit commands
  S 2-digit commands are affected
- Memory contents such as tool offset values and parameter data

NOTE: Depressing the RESET key or the remote reset button is defined as Reset operation in this manual.

4.1.13 UNIVERSAL DISPLAY

The display unit shows all the data other than tool offset number and parameter number. Usually shown is the data consisting of a sign and seven digits. On-line diagnostics such as I/O signals or the result of a part of off-line diagnostics are displayed in 8 digits.

F.g. 4.1.13

4.1.14 TAPE FEED AND SYSTEM NO. SWITCHES

These switches are mounted above the tape reader.

- TAPE FEED switch

  It is to feed and rewind the tape manually with the control at standby. Setting the switch to F (forward) causes the tape to feed. To rewind the tape, set the switch to R (reverse).

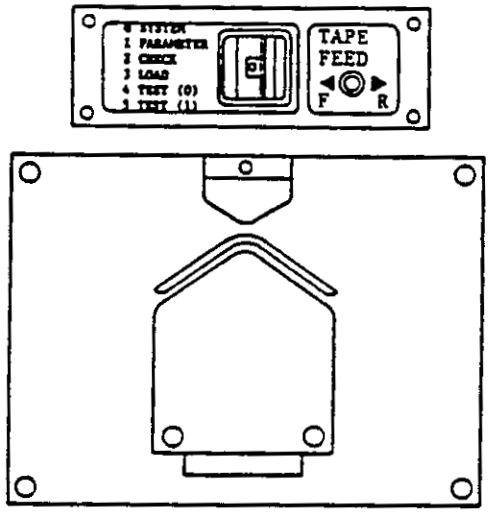


Fig. 4.1.14

· SYSTEM NO switch

Set the switch at "0" during usual operation. Functions to its setting are as follows.

Setting at.

"0" SYSTEM--

For usual operation. Writing parameters is prevented.

"1" PARAMETER--

To write parameters. At this position, Cycle Start is prevented. Set the switch back to "0" when parameters have been written.

"2" CHECK--

To collate the system program stored with the system tape.

"3" LOAD--

To store maintenance tape data into the control.

"4" TEST (0)--

Usual operation is permitted. Diagnosing of the memory contents and checking of Zero Return Position are omitted.

"5" TEST (1)--

Writing parameters is effective. Diagnosing of the memory contents and Zero Return position check are omitted.

4.2 POWER ON/OFF OPERATION

4.2.1 TURNING ON POWER

Before turning on power, check the machine referring to the machine tool builder's manual.

1. Depress the POWER ON button to turn on control power. The initial timer will be reset in about two seconds. Then the servo unit is ready for turning on power, which can be shown in alarm code "31."

2. Depress the POWER ON button again to turn on servo power. The NRD (NC READY) signal is given which indicates that the control is ready.

3. When the NRD (NC READY) signal turns on the power of the machine and the MRD (MACHINE READY) signal is given back to the control, the READY lamp will be lit. The NC machine is ready to operate.

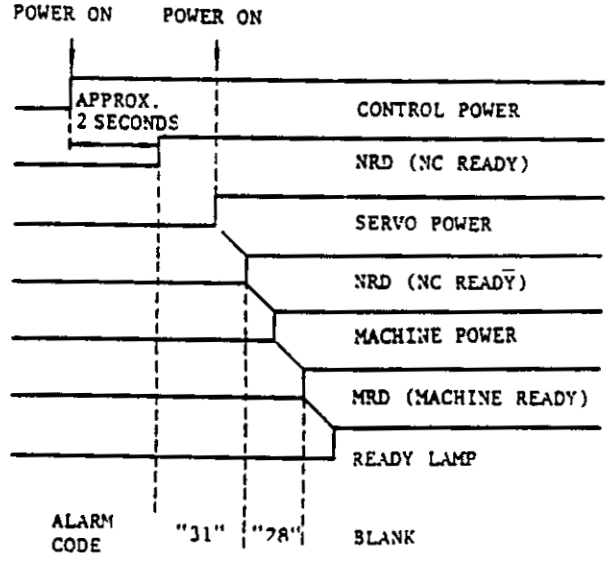


Fig. 4.2.1 Sequence of Turning on Operation

4.2.2 TURNING OFF POWER

Depressing the POWER OFF button causes servo power and control power to be turned off simultaneously. However, for stabler system operation, take the following procedure.

1. Depress the EMERGENCY STOP button to cut off servo power. NRD signal is interrupted, which results in turning off the machine power, too.
2. Depress the POWER OFF button to cut off control power.

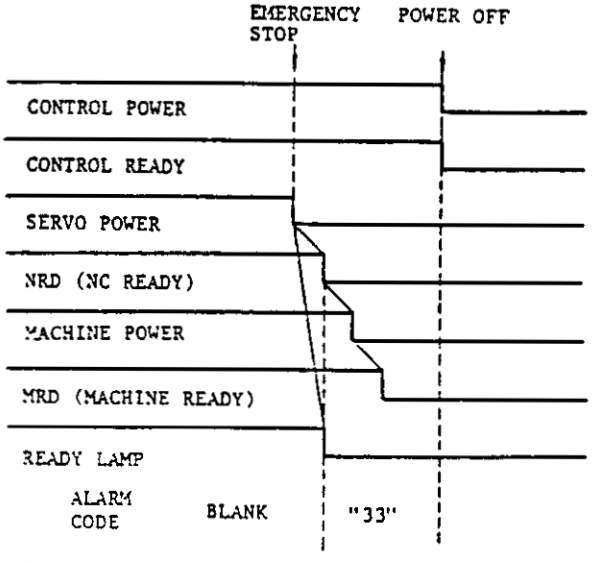


Fig. 4 2.2 Sequence of Turning Off Operation

4.2.3 REMOTE TURNING ON/OFF BUTTONS

Connect power on/off buttons to EON, EOF, and COM terminals of the control as shown below, then remote turning on/off operation can be made.

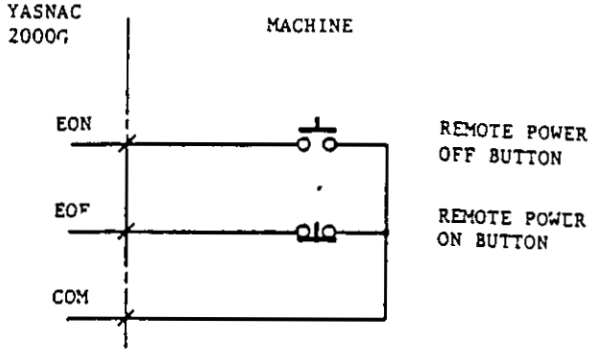


Fig. 4.2.3 Connections of Remote Power ON/OFF Buttons

4.3 DISPLAY AND WRITING OF COMMAND DATA

4.3.1 DISPLAY OF COMMAND DATA

Command data can be displayed in any mode using the following procedure.

1. Depress the COM key, and it lights up
2. Select an address character with the ADDRESS key

   Then command data already entered is displayed. The data shows the contents of the active register when the control is in automatic operation. With the control stopped at block end, displayed is the contents of the buffer register. The coordinate values displayed are modified with tool offset value.

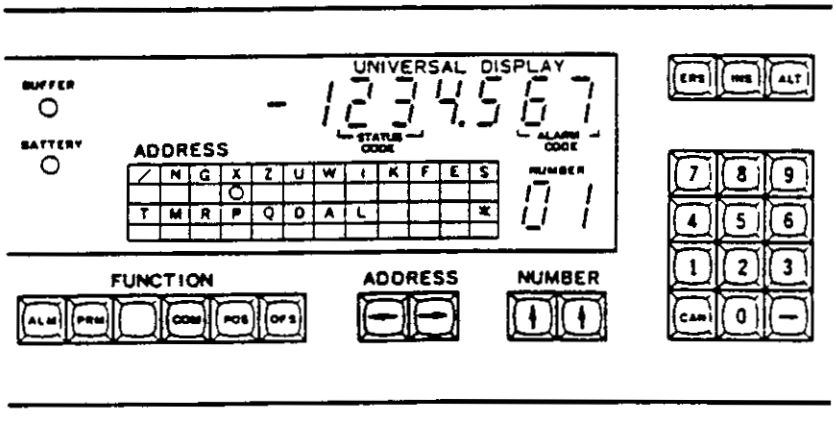


Fig. 4.3 1.1

Use the following steps following step 2 to facilitate the display operation.

3. To display all address characters in the selected block

   All the address characters in the block can be displayed while the COM key is held in. By releasing the COM key, the command data obtained by step 2 appears again on the universal display.

4. To display all G-codes

   G-codes are classified into two types, modal and non-modal. Modal are G-codes of A, B, C, and D groups and non-modal, of * group. Refer to Table 2.10.1.2. Usually displayed is a G-code of A group or non-modal type. All the current G-codes are displayed by setting NUMBER at "99" and selecting address G. See Fig. 4.3.1.2.

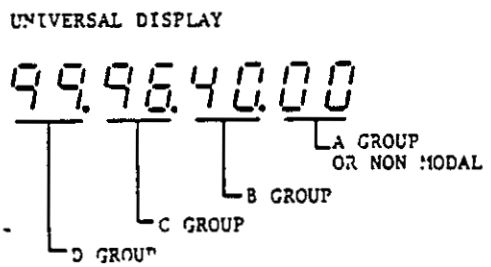


Fig. 4.3.1.2 Example of All G-code Display

5. To display two types of S-codes

   S-code is to specify surface speed(m/min) for constant surface speed control and maximum spindle speed (rpm--set by G50 S · · · *).

   Usually displayed is surface speed. Both S-codes are displayed by setting NUMBER at "99" and selecting address S.

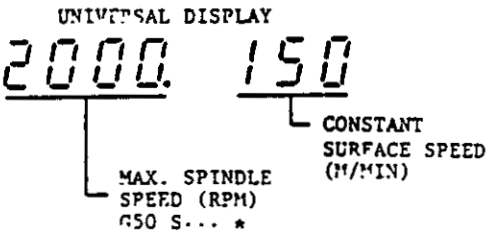


Fig. 4.3.1.3 Example of All S-codes Display

4.3.2 WRITING COMMAND DATA BY MDI

Command data of a block can be written manually in the MDI mode when the control is stopped at block end. Writing operation cannot be allowed if the data remains in the active register during automatic operation or after a temporary stop by FEED HOLD pushbutton.

1. Set the MODE select switch to the MDI mode
2. Depress the COM key and it will light up
3. Select the address character with ADDRESS key

   Modal command data already executed is displayed on the universal display.

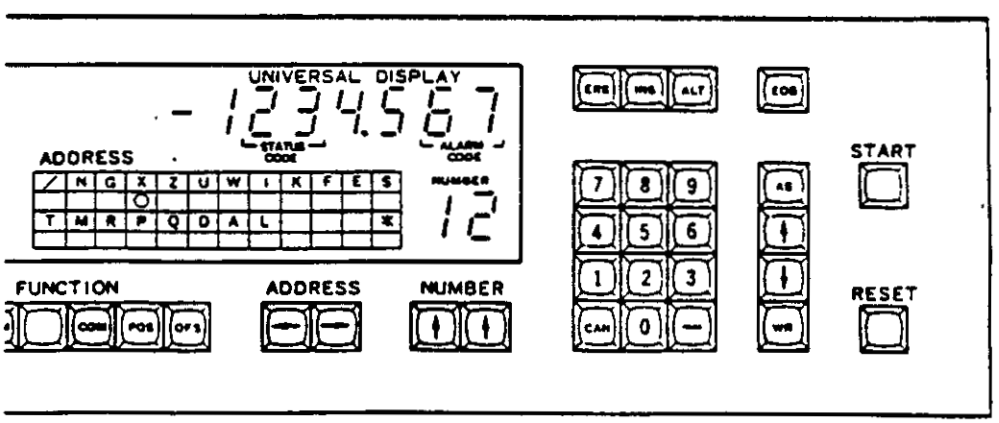


Fig. 4.3.2

4. Key in the new data through the DATA keyboard.

   The display shows the data as it is entered, flickering the last significant digit. To correct the data just keyed in, depress the CAN key and key in the correct data.

5. Depress the WR key.

   The data just keyed in enters the buffer register as a new command data. The coordinate values displayed are modified with the tool offset value.

6. Repeat steps 3 through 5 until a block has been written.

7. Depress the CYCLE START pushbutton, and the input commands are executed.

NOTES.

- Depressing the RESET key during writing operation by MDI deletes all the data already written.
- G-codes for Tip Nose Radius Compensation (G41 to G44) and Multiple Repetitive Cycles (G70 to G76) cannot be changed by MDI operation. New G-code can be displayed on the universal display, but never entered the buffer register by the WR key.
- When writing G-codes of B to D groups, check the written value on the universal display by setting the number at "99" with the NUMBER key. G-codes of other than A group and non-modal type are not usually displayed. See 4.3.1 Display of Command Data.
- Also check S-code (G50 S · · · *) specifying maximum spindle speed on the universal display by setting the number at "99" with the NUMBER key.
- All the address characters already written in a block can be displayed on the address display by depressing the COM key during MDI operation. See 4.3.1 Display of Command Data.

4.3.3 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, and it lights up.

2. Set the ADDRESS to X or Z using ADDRESS keys.

   Then current position of the selected axis is displayed on the universal display

Either total amount of movement or absolute coordinate value can be displayed by parameter setting.

Where parameter No. 72 = "0"

- Displayed current position is the same as that on the current position display unit.
- Universal display shows total amount of movement of the tool by manual and automatic operation. It cannot be reset by G50.
- The display is updated even if the LOCK MODE switch is set to MACHINE LOCK position.

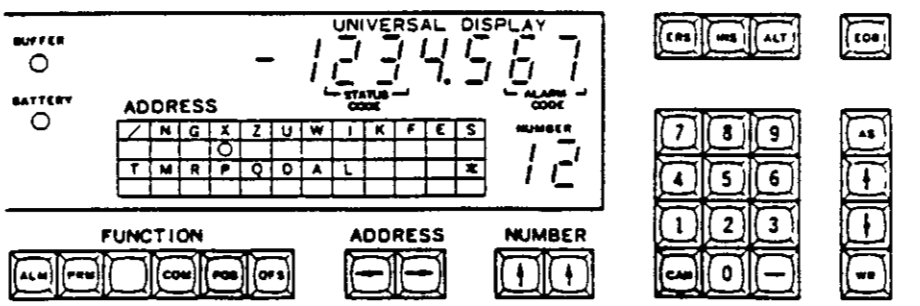


Fig. 4.3.3

- To reset the displayed data, depress the POS and CAN keys simultaneously.
- The RESET buttons on the current position display unit* do not affect the universal display.

Where parameter No. 72 = "1"

- Displayed data is automatically set up by G50 command. The universal display shows the current position based on the programmed coordinate system as far as manual operation does not interrupt.
- Displayed data cannot be cleared by any buttons on the operator's panel.
- The display is updated even if the LOCK MODE switch is set to MACHINE LOCK position.

4.3.4 INCREMENTAL VALUE DISPLAY

Incremental values of X-, and Z-axis movement are displayed by depressing POS key and selecting the addresses U and W respectively.

Display shows:

- Updated distance to the end of the block being executed in automatic operation.
- Updated distance from the manual operation starting point in manual operation mode. The displayed incremental value in manual operation is cleared by switching the MODE SELECT switch to TAPE, MDI, or MEM.

4.3.5 DISPLAY OF TOOL OFFSET VALUE

The values of tool offset are stored in memory of the control. The display of the value can be made at any time in every mode even during automatic operation.

Operating procedure is as follows:

1. Depress the OFS key, and it lights up.
2. Set tool offset number with NUMBER key.
3. Select the address character from X, Z, U, W, and R with ADDRESS select key.

   The offset value for the selected address is displayed.

If an unused offset number or an address other than described above has been selected, the display will be blank.

Table 4.3.5 Addresses for Tool Offset Values

Addresses	Display
X	Tool offset value for X axis
Z	Tool offset value for Z axis
U	The same as the display for X (To write incremental tool offset value for X axis)
W	The same as the display for Z (To write incremental tool offset value for Z axis)
R	Radius value for tip nose radius compensation*

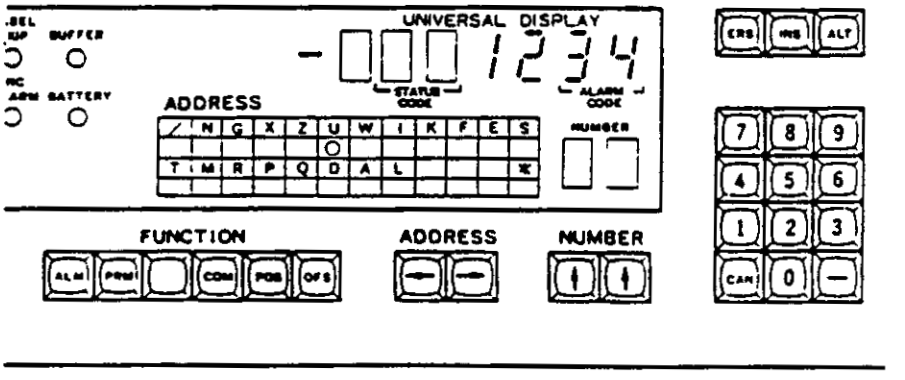


Fig. 4.3.4

4.3.6 WRITING OF TOOL OFFSET VALUE

Writing or modification of the tool offset value is always possible independently of the operating mode. The procedure is as follows:

1. Depress the OFS key, and it lights up.
2. Set the tool offset number using the NUMBER key.
3. Select the address character using the ADDRESS key.

   The present offset value for selected address is displayed.
4. Key in new offset value from the DATA keyboard. The display shows the data just keyed in, flickering the last significant digit. To correct the data just keyed in, depress the CAN key and input correct data.
5. Depress the WR key. Then the displayed data will be stored into memory as a new tool offset value

NOTES:

- Tool offset values stored in memory will not be erased by turning off the power.
- Writing and modification of tool offset values can be always possible in any mode including automatic operation mode.
- When the offset value is changed during automatic operation, the blocks in buffer register and active register are executed with an old one. The new offset value is effective in the next block.
- Current radius value cannot be modified during Tip Nose Radius Compensation+ until it is cancelled by G40 or T00. When zero is specified for radius, the compensation will be cancelled

4.3.7 WRITING OF INCREMENTAL VALUE OF TOOL OFFSET

To add the incremental value to the stored tool offset value, select address U or W in the procedure of 4.3.6 Writing of Tool Offset Value. Then the summed offset value will be stored into memory as a new offset data.

4.3.8 PARAMETER DISPLAY

Parameters stored in memory are to determine the operating conditions such as rapid traverse rate and tape code. Parameter display can be made in any mode of operation using the following procedure. For meanings of parameters, see Table 4.3.2 List of Parameters.

1. Depress the PRM key, and it lights up.
2. Set the parameter number using NUMBER key
3. For parameter numbers of 20's, 30's, 60's and 90's, select an axis (X or Z) with the ADDRESS key. Then the stored data will be shown on the universal display.

   If an address character other than X and Z has been selected for a parameter number in 20's, 30's, 60's and 90's, the display will be blank.

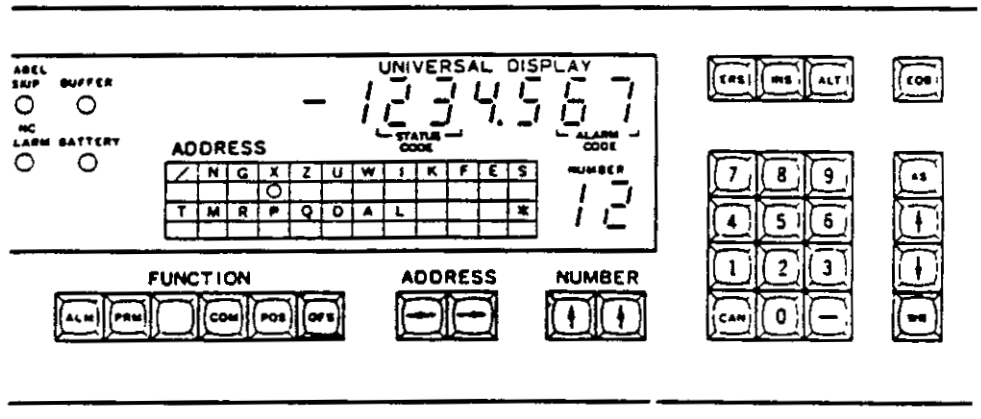


Fig. 4.3.6

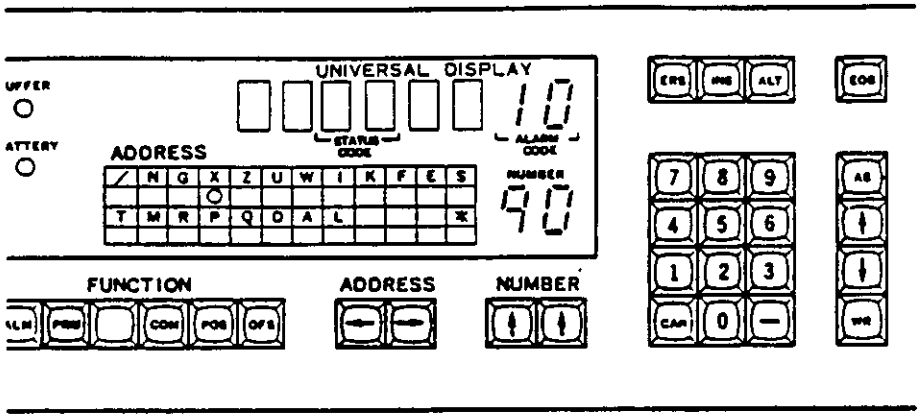


Fig. 4.3.8

4.3.9 WRITING PARAMETERS

Optimum data of parameters other than the ones for setting function have been set according to machine performance and applications. Parameters are interlocked by setting SYSTEM NO switch at "0" so that the data are not accidentally erased or changed. Where it is desired to modify the data of parameters, consult the machine tool builder. Procedure is as follows.

1. Set the SYSTEM NO. switch at "1."
2. Depress the PRM key, and it it illuminated.
3. Set the parameter number using the NUMBER key.
4. For parameter numbers in 20's, 30's, 60's, and 90's, select an axis (X or Z) with the ADDRESS key. Then the stored data will be shown on the universal display.
5. Key in the parameter data from the DATA keyboard.

   The display shows the data as it is entered, flickering the last significant digit. To correct the data just keyed in, depress the CAN key and key in the correct one.
6. Depress the WR key. The data just keyed in will be stored into memory as a new parameter.
7. Repeat steps 3 to 6 until the necessary parameters have been set.
8. Set back the SYSTEM NO switch to "0 "
9. Depress the RESET key.

NOTES

- Data of parameters, once stored in memory, are not erased by turning off the power
- With parameter No. at 00, totalized time of automatic operation will be displayed. For details, refer to 4.3.10 Operation Time Display.
- Data of parameter numbers 01 to 09 can be changed without setting the SYSTEM NO switch at "1." For details, refer to 4.3.11 Setting Function.
- Setting parameter number at 99 displays active address of the tape memory with number of characters on the universal display. See 4.5.4 Address Display of Tape Memory.

  Data of parameter numbers 10 to 98 can be displayed at anytime. They cannot be changed unless SYSTEM NO switch is set at "1 "
- If any of parameter data is accidentally destroyed, alarm code "17" will be displayed.
- SYSTEM NO. switch is provided above the tape reader.

Table 4.3.9 List of Parameters

第 4・3・9 表　パ ラ メ ー タ ー 覧 (1/3)

NO.	ADD-RESS	意味 MEANING
00		稼動時間表示 Operation Time Display 9999. 時 (H) 59 分 (M) 59 秒 (S)
01		早送り速度レンジ Rapid Traverse Rate Range '0' = Hi, '1' = Lo
02		ドライラン時早送り速度 Rapid Traverse Rate for Dry Run '0' = Hi/Lo, '1' = JOG
03		ストロークチェック入切 Stroke Check ON-OFF '0' = OFF, '1' = ON
04		マシンロック時, ストロークチェック入切 Stroke Check ON-OFF for Machine Lock '0' = OFF, '1' = ON
05		周速制御時, 工具補正量キャンセル Tool Offset Cancel for Surface Speed Control '0' = OFF, '1' = ON
06		未使用 Not Used
07		
08		
09		

セッティング機能
Setting Storage 備考1)
Notice 1)

NO.	ADD-RESS	意味 MEANING
30	X	固有原点からのストロークチェック座標 (+) Stroke Check Point (+) from Zero Point '1' = 0.001 mm
	Z	
	I	未使用 Not Used
	K	
31	X	固有原点からのストロークチェック座標 (−) Stroke Check Point (−) from Zero Point '−1' = −0.001 mm
	Z	
	I	未使用 Not Used
	K	
32	X	
	Z	
	I	Param 39 → Memory OPTION
	K	
33	X	
	Z	未使用 Not Used
	I	
	K	
34	X	
	Z	
	I	
	K	

NO.	ADD-RESS	意味 MEANING
40		
41		
42		
43		※シーケンス制御用パラメータ Parameter for Optional
44		Machine Interface '0' = OFF, '1' = ON
45		
46		
47		
48		未使用 Not Used
49		

Table 4.3.9 List of Parameters

第 4・3・9 表　パ ラ メ ー タ ー 覧 (2/3)

NO.	ADDRESS	意味 MEANING
50		
51		
52		
53		※ シーケンス制御用タイマ定数 Timer Constant for Optional
54		Machine Interface '1' = 16/80 msec (Hi/Lo)
55		
56		
57		
58		未使用 Not Used
59		

NO.	ADDRESS	意味 MEANING
60	X	バックラッシュ補正開始方向 Backlash Comp. Starting Direction
	Z	'0' = from (+), '1' = from (−)
61	X	原点復帰方向 Zero Return Direction
	Z	'0' = to (+), '1' = to (−)
62	X	原点復帰最終速度 Zero Return Final Speed
	Z	'1' = 7.5 mm/min
63	X	ポジションエラー領域 Position Error Zone
	Z	'1' = 最小移動単位 Least Command Increment
64	X	サーボエラー領域 Servo Error Zone
	Z	'1' = 最小移動単位 Least Command Increment
65	X	未使用 Not Used
	Z	
66	X	サイクルスタート前の原点復帰 Zero Return before Cycle Stare
	Z	'0' = 不要, '1' = 要 Not Required Required
67	X	手動ハンドル送り最大速度 Max. Feedrate for Handle
	Z	'1' = 7.5 mm/min
68		切削送り時定数 Time Constant for Feed '1' = 32 msec
69		切削送り速度バイアス Velocity Bias for Feed '1' = 2 Kpps

NO.	ADDRESS	意味 MEANING
70		直径／半径指定 Diameter/Radius Designation '0' = 直 径, '1' = 半 径 Diameter Radius
71		G99/G98 イニシャルセット Initial Set '0' = G99, '1' = G98
72		G50 表示プリセット入切 Display-Preset ON-OFF '0' = OFF, '1' = ON
73		GR0 主軸DA出力値 Spindle DA Output '2047' = 10 V
74		G74 引戻し量 (一定量) Retracting Value '1' = 0.001 mm or 0.0001 inch
75		G75 引戻し量 (一定量) Retracting Value '1' = 0.001 mm or 0.0001 inch
76		G76 仕上げ代 (一定量 a) Finishing Value (a) '1' = 0.001 mm or 0.0001 inch
77		G92, G76 チャンファ幅 (γ) Chamfering Width '1' = 0.1 リード Lead
78		GR3 主軸最大回転数 Spindle Max. RPM '1' = 1 RPM
79		GR4 主軸最大回転数 Spindle Max. RPM '1' = 1 RPM

Table 1.3.9 List of Parameters

第4・3・9表　パラメーター一覧 (3/3)

NO.	ADDRESS	内容 MEANING	NO.	ADDRESS	内容 MEANING
80		EIA/ISOの自動判別入切 - Auto Select '0' = OFF, '1' = ON	90	X Z	バックラッシュ補正量 Backlash Value '1' = 最小移動単位 Least Command Increment
81		TVチェック入切 TV Check ON - OFF '0' = OFF, '1' = ON	91	X Z	未使用 Not Used
82		EIA/ISOコード指定 Code Dignation '0' = EIA, '1' = ISO	92	X Z	
83		MM/INCH指定 Designation '0' = MM, '1' = INCH	93	X Z	早送り速度 (Lo) Rapid Traverse Rate (Lo) '1' = 7.5 mm/min
84		タッチブザー入切 Touch Buzzer ON - OFF '0' = OFF, '1' = ON	94	X Z	早送り速度 (Hi) Rapid Traverse Rate (Hi) '1' = 7.5 mm/min
85		DNC結合入切 DNC Interface ON - OFF '0' = OFF, '1' = ON	95	X Z	早送り加減速定数 Accel./Decel. Time Const for RT '1' = 125/8 mm/sec²
86		GR1　1軸最大回転数 Spindle Max. RPM '1' = 1 RPM	96	X Z	原点復帰クリープ速度 Zero Return Approaching Speed '1' = 7.5 mm/min
87		GR2　1軸最大回転数 Spindle Max. RPM '1' = 1 RPM	97	X Z	原点復帰最終距離 Zero Return Final Stroke '1' = 0.001 mm
88		入力指令10倍入切 Command data × 10 ON - OFF '0' = OFF, '1' = ON	98		MF, SF, TF送出遅れ時間 Delay. Time for MF, SF, TF '1' = 1 msec
89		G00　刃先R補正入切 Tip Nose R Comp - ON - OFF '0' = OFF, '1' = ON	99		メモリポインタ表示 Memory Pointer Display '1' = 1 ch

備考 2)
Notice 2)

備　考
NOTICE

1) No. 01～09はSYSTEM No.スイッチのインタロックなしに書込み可能です。

No. 01～09 can be changed without the interlock of SYSTEM No. switch.

2) No. 00および99は表示専用であり，書込みはできません。

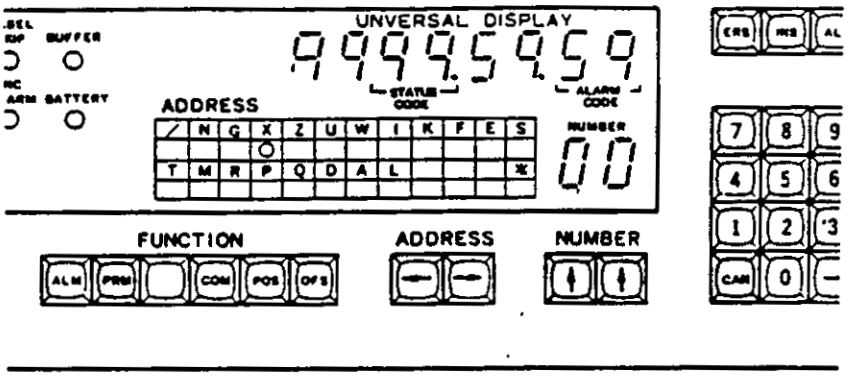
No. 00 & 99 are for display only.
These cannot be written.

3) No. 30～98はSYSTEM No.スイッチによってインタロックされており，同スイッチを'1'の位置にして書込みを行ないます。

No. 30～98 are interlocked by SYSTEM No. switch, and can be changed only when the position of SYSTEM No. switch is '1'.

4) ◤ 印の付されたパラメータはオプションです。

Parameters with ◤ Mark are optional.



(Operation time: 9999 hours, 59 minutes, and 59 seconds)

Fig. 4.3.10

4.3.10 OPERATION TIME DISPLAY

Operation time displayed shows the totalized time of automatic operation of machine. It may be used to know the working time to finish a workpiece or total operation time of the system.

1. Depress the PRM key, and it lights up
2. Set the parameter No. at 00 with NUMBER key.

   Operation time is displayed in hours, minutes, and seconds. Operation time means the totalized time while CYCLE START lamp is on. It is not cleared by turning off power.
3. To reset the display, depress the PRM key and CAN key simultaneously.

4.3.11 SETTING FUNCTION

Parameter numbers 01 to 09 contain switching function and can be written without operation of SYSTEM NO switch. For their meanings, see Table 4.3.9 List of Parameters.

Procedure of changing operation is as follows.

1. Depress the PRM key, and it will light up
2. Set the parameter number (01 to 09) using NUMBER key. The stored parameter data is shown on the universal display.
3. Input "1" or "0" through the DATA keyboard. The display shows the input figure and flickers it. To correct the figure, depress the CAN key and input the correct one
4. Depress the WR key. The displayed data is stored into memory as a new setting parameter data.

NOTE Data of parameter numbers 01 to 09 can be changed in any mode including automatic operation mode.

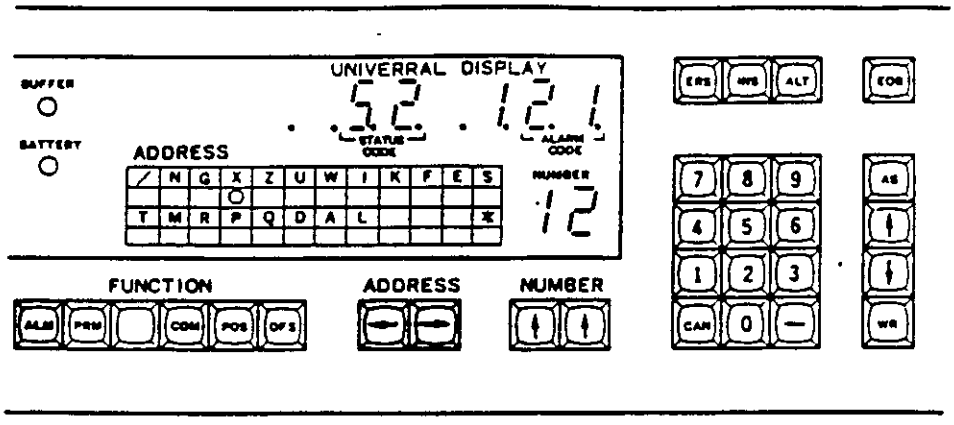


Fig 4.3.12

4.3.12 ALARM AND STATUS CODE DISPLAY

The control is always diagnosing even during machining. When the control detects an error, INPUT ERROR or NC ALARM lamp lights up and it stops operation. The corresponding alarm and status code will be displayed on the universal display when ALM key is pushed and lights up. Alarm code signifies the cause of the error and status code indicates the operating condition. See Table 4.3.12.1 List of Alarm and Status Codes.

NOTES

- CPU error in code "81" and memory check error in code "82" are displayed without operating ALM key.
- Alarm code is displayed with eight dots to distinguish from the other displayed data.
- Alarm code is usually shown in two digits. Errors on axis are displayed in three digits, first of which shows the axis whose error is detected.
- When more than one error occurs at the same time, alarm codes are displayed in numerical order.
- Displayed alarm code is cleared by depressing the RESET key after removing the cause of the alarm

  Refer to 8 5 Trouble Causes and Remedies.
- Status code will be displayed, if any, simultaneously with alarm code
- Where the SYSTEM NO. switch is set at other than zero, the set number is shown on the extreme left of the universal display.
- I/O signal display will replace alarm code display by operating the NUMBER key Alarm code will be displayed again by depressing the ALM key again. See 4.3.13 Display of I/O Signals.

Table 4.3.12.1 List of Alarm Codes and Status Codes

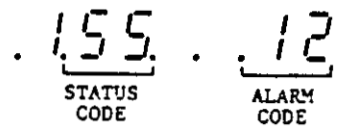
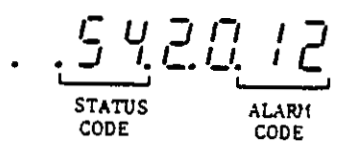
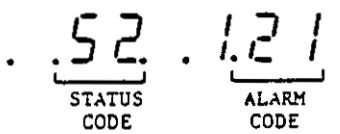
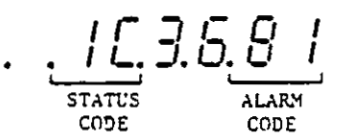
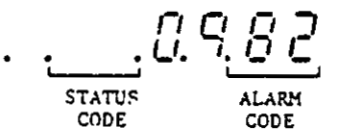
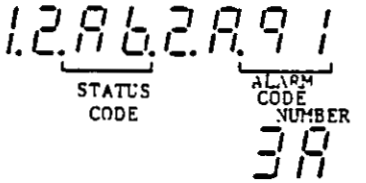
Alarm code	Causes	Alarm code	Causes
11	Excessive temperature rise in the panel.	27	Positioning error.
□12	Tape/Memory horizontal parity error.	28	Machine unready.
13	Tape vertical parity error.	31	Servo power unsupplied.
14	Format error.	32	Control unit unready.
15	Data error.	33	Emergency stop.
16	Offset error.	□34	Servo error.
17	Parameter error.	35	Overload.
18	Tape memory error.	36	Feedback error.
□21	Overtravel.	37	Hardware error (FG).
□22	Zero return area error	38	Hardware error (RPG).
□23	Zero return unready.	81	CPU error.
□24	Zero return position error.	82	Memory collating error.
25	Sequence error.	91	Contents disagreement between tape and memory. (For off-line only.)
26	Spindle error.	92	Tape reading error. (For off-line only.)

Status code	Status	Remarks
□51	Performing M-, S- and/or T-function.	-
52	Distributing pulses.	-
□53	Performing M-, S- and/or T-function and distributing pulses.	51 + 52
54	Reading tape.	-
□55	Performing M-, S- and/or T-function and reading tape.	51 + 54
56	Distributing pulses and reading tape.	52 + 54
□57	Performing M-, S- and/or T-function, distributing pulses and reading tape.	51 + 52 + 54

Notes:

1. □ Of alarm code is filled with a digit indicating the axis whose error is detected. 1: X-axis, 2 Z-axis, 3 X and Z-axis.
2. For alarm code "12," two digits are added to indicate the memory IC number
3. □ Of status code is filled with a digit indicating either M, S, or T function which is being executed.

Table 4.3.12.2 Display Example of Alarm Code Combined with Status Code

Alarm and Status Codes	Meanings
UNIVERSAL DISPLAY  	Horizontal parity error (alarm code "12") occurs while executing M-function and reading tape (status code: "55" = "51" + "54") in the TAPE mode.
 	Horizontal parity error (alarm code "12") occurs while reading tape memory (status code "54") in the MEM mode. Defective IC number is 20.
 	Overtravel (alarm code "21") of X axis (addition of alarm code "1") occurs in the automatic operation.
 	Operation cannot be continued due to erroneous operation of CPU (alarm code: "81"). The active address is 1C36.
 	Diagnostics for memory detect memory error (alarm code "82") Defective ROM number is "09."
 	Contents of system memory are different from those of source tape (alarm code: "91") during collating operation with SYSTEM NO. switch set at 2. Memory address is "12AB," memory contents, "2A," and tape contents "3A."

4.3.13 DISPLAY OF INPUT/OUTPUT SIGNALS

All the input/output signals can be checked on the operator's panel at any time even during automatic operation.
Procedure is as follows:

1. Depress the ALM key, and it lights up.
   Alarm and status codes will be shown on the universal display.
2. Select diagnostic number of I/O signals using NUMBER key and ADDRESS key.
   Then I/O signal "1" or "0" replaces the displayed alarm code and status code on the universal display. Operation of NUMBER key makes I/O signal display mode automatically.

NOTES.

- For diagnostic number of each signal, refer to Table 8.5.3 List of Input/Output Signals.
- Signal is shown by "1" or "0."
  "1": contact close, "0": contact open.
- In the I/O signal display mode, I/O signal diagnostic number can be input from data keyboard, too. Inputting the diagnostic number shifts the displayed diagnostic number to left and makes room for new number on the NUMBER display. Operation of the WR key is not necessary.
- I/O signal display mode is cancelled by depressing the RESET key or FUNCTION key.

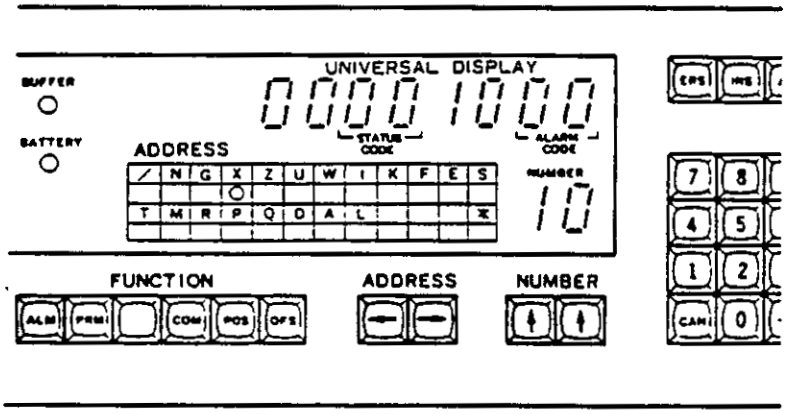
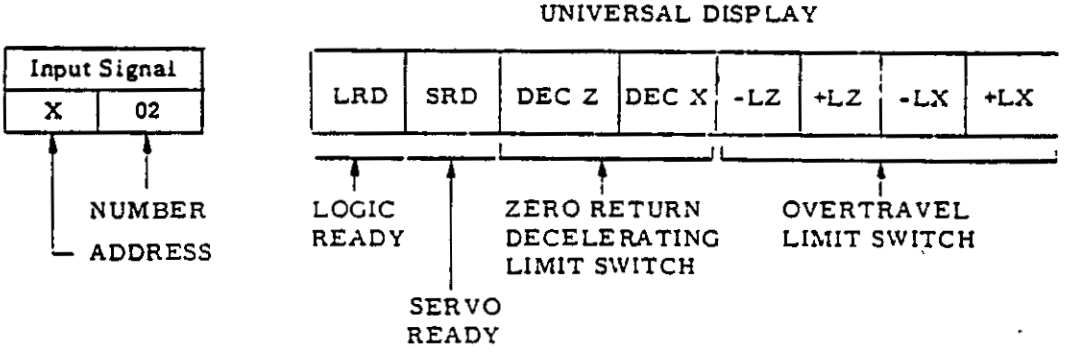


Fig. 4.3.13.1



Selecting address X and number "02" causes input/output signals shown above to be displayed by "1" or "0" on the universal display.

Fig 4 3 13.2 Example of Input/Output Signal Display

4.3.14 ADDRESS SEARCH

The block with specified data can be searched by designating every address character as well as N. Search may be through the tape data and the stored part program data respectively in the TAPE mode and in the MEM and EDT modes. In the EDT mode searched block data does not enter the buffer register. See NOTES.

1. Set the MODE SELECT switch to TAPE, MEM, or EDT.

2. Depress the COM key, and it is lit.

3. Depress the RESET key. Then the LABEL SKIP lamp will be on, and the memory is rewound.

4. Set the address character by ADDRESS key.

   In the EDT mode, set the NUMBER key at other than "00."

5. Set the data through the DATA keyboard.

   The display shows the data as it is entered, flickering the last significant digit. To correct the data just entered, depress the CAN key and key in the correct one.

6 Depress the AS key to start address search.

   When the block containing the selected address data is encountered, BUFFER LAMP lights up indicating that search operation is completed.

7. To stop the search operation, depress the RESET key

8 In the MEM and EDT modes, block-by-block search is allowed by using ↓ and ↑ keys.

NOTES:

In the TAPE and MEM modes searched block of data enters the buffer register. Coordinate values are modified with tool offset values.

- In the EDT mode, searched block enters edit-buffer register without being modified with tool offset value and so on. But BUFFER lamp is on likewise in Tape and MEM mode.
- Leading zeros may be suppressed for all address characters including N in address search operation. For example, N12 means N0012.
- All commands in the searched blocks including modal one are not recognized, and only update data enters in the buffer register.
- Operation begins with searched block by depressing CYCLE START pushbutton after address search in the TAPE or MEM mode has been accomplished.
- Reset the control before Cycle Start after address search in the EDT mode. Failure to do so causes format error (alarm code: "14") in any operating mode.

4.3.15 TV CHECK (VERTICAL PARITY CHECK)

TV check is used to make the vertical parity check in each block during tape reading operation in TAPE mode. TV check ON or OFF can be selected with parameter No. 81. For parameter setting, see 4.3.9 Writing Parameters.

With No "81" at "0" . . . TV Check OFF
With No. "81" at "1" . . . TV Check ON

With TV Check ON, if the number of characters including EOB code in the block is odd, INPUT ERROR lamp lights up during tape reading operation in the TAPE mode Alarm code "13" (TV parity error) is displayed To adjust the number of characters in a block to be even, use a space code as additional character

4 3.16 CURRENT POSITION DISPLAY UNIT

The movement of the tool is summed up and the current position of each axis is diplayed on the current position display unit.

- Depressing the RESET button causes the display to be zero

  The indication is not affected by the G50 command.

- The indication is updated even with the MACHINE LOCK switch on, but not changed with the DISPLAY LOCK switch on.
- Even if servopower is OFF by pressing the EMERGENCY STOP button, the indication follows tool movement.

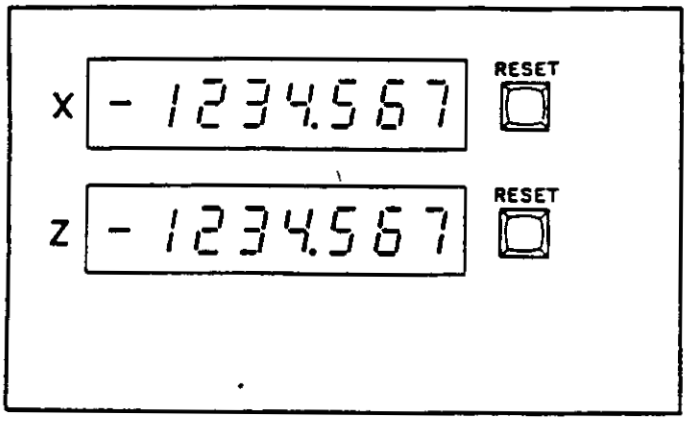


Fig. 4.3.16 Current Position Display Unit

4.4 STORING TOOL OFFSET VALUES FROM NC TAPE†

Tool offset values are usually written through the DATA keyboard. It can be input from the punched tape, too.

Tape format for tool offset values is

```
LABEL *
T1 X···  Z···  R···  *
T2 X··   Z···  R···  *
T3 X···  Z··   R···  *
 .
 .
 .
ER (or %) .... Rewind stop code
```

Procedure for storing the tool offset values from the punched tape is as follows.

1. Set the MODE SELECT switch to EDT.
2. Depress the RESET key. The LABEL SKIP lamp is illuminated.
3. Depress the OFS key. Then it lights up.
4. Set the NUMBER at other than "61" and "62" using the NUMBER key.
5. Set the punched tape on the tape reader. Be sure that Label Skip function is effective.
6. Depress the START key with the OFS key held in. Tape reader starts and tool offset values are stored into memory. It automatically stops when rewind stop code is encountered.

4.5 PART PROGRAM STORAGE†

4.5.1 STORING PART PROGRAM FROM NC TAPE†

Part program punched on the tape can be stored into memory through the tape reader. Maximum capacity of the memory is shown in Table 4.5.1. If the number of significant characters (including EOB character) on the tape exceeds the capacity of memory, INPUT ERROR and alarm code "18" (tape memory error) will be indicated.

Table 4.5.1

	Memory Capacity	Tape Length
A	4000 char.	Approx. 10 m
B	8000 char.	Approx. 20 m
C	12000 char.	Approx. 30 m
D	16000 char.	Approx. 40 m

Part program punched should be sandwiched with rewind stop code (EIA: "ER," ISO: "%") M02 or M30 command must be programmed in the final block of the program.

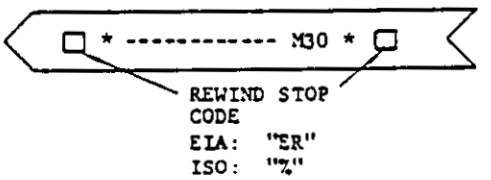


Fig. 4.5.1.1

Tape code is specified by parameter No. 82. Where automatic selection of tape code is effective with parameter No. 80 = "1," the control will automatically adjust to read tape with either EIA or ISO character format.

Follow the procedure below to store the punched tape data.

1. Set MODE SELECT switch to EDT

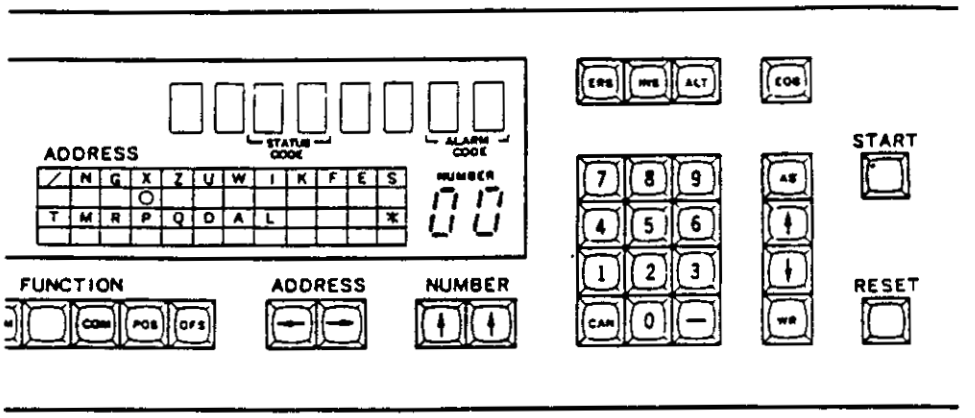


Fig. 4.5.1.2

2. Depress the RESET key. Then memory is rewound and LABEL SKIP lamp is on. This is a preparation to store part program orderly from the first address of the memory
3. Depress the COM key, and it lights up. Any FUNCTION key except OFS is allowed to be set. However, use COM key normally
4. Set the NUMBER at other than 61 and 62, using the NUMBER key
5. Set the punched tape onto the tape reader unit. Be sure that Label Skip function is effective
6. Depress the START key.
   Then tape reader starts and part program is stored into the memory. It automatically stops when rewind stop code has been read.

4.5.2 PART PROGRAM MODIFICATION FROM NC TAPE

To modify the block already stored following the desired block, use the following procedure.

The already stored data starting from the desired block is deleted by storing the new data.

1. Set the MODE SELECT switch to EDT
2. Search the block in which the new program will be stored. See 4 3.14 Address Search.
3. Depress the COM key, and it lights up
4. Set the NUMBER at other than "61" and "62" using the NUMBER key
5. Apply the new tape onto the tape reader unit.
   Be sure that Label Skip function is effective.
6. Depress the START key
   New tape data is stored into memory beginning from the searched block continuously until a rewind stop code is read.

NOTE
If storing operation is interrupted by parity error or depressing the RESET key, restore the new data.

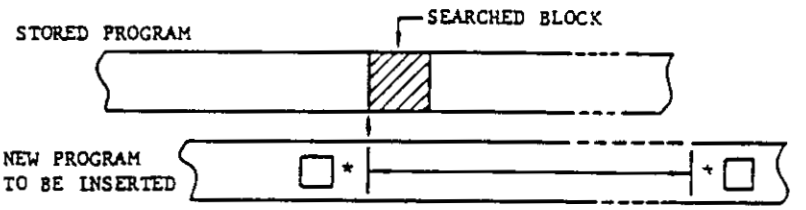


Fig. 4.5.2

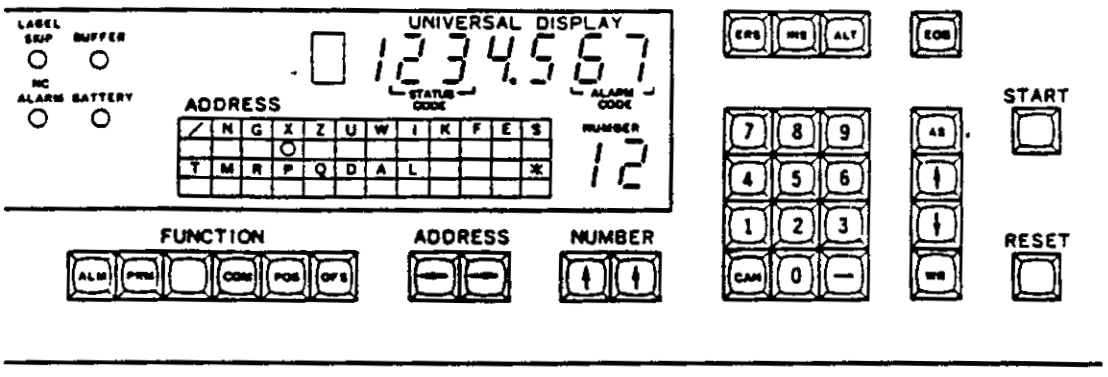


Fig. 4.5.3

4.5.3 STORING PART PROGRAM FROM MDI

1. Set the MODE SELECT switch to EDT
2. Depress the RESET key.

   Memory is rewound, and LABEL SKIP lamp is on. It indicates that the control is ready to store the programs from the first address in the memory.
3. Depress the COM key, and it lights up.
4. Set the NUMBER at other than "00."

   With the NUMBER set at "00" in the EDT mode, free address setting is not allowed because the addresses already stored in the buffer register are scanned.
5. Select the address character using the ADDRESS key.
6. Key in the data through the DATA keyboard.

   The display shows the data as it is entered, flickering the last significant digit. To correct the data just keyed in, depress the CAN key, then key in the correct data.
7. Depress the WR key

   The data just keyed in is registered as a new command data, and flickering stops. BUFFER lamp lights up
8. Repeat steps 4 through 6 until a block of data has been written.
9. Check the input data, referring to 4.3.1 Display of Command Data. Then depress the EOB key.

   The data of a block in buffer register is stored into memory, and BUFFER lamp goes out
10. Repeat steps 4 through 9 to complete the part program storage Store the block including "M02" or "M30" command as the final block of the part program.
11. Check the program stored after depressing the RESET key.

    See 4.6.1 Display of Stored Part Program.
12. Correct the data, if necessary, referring to 4 6.2 Editing Stored Part Program.
13. Depress the RESET key again to rewind memory. LABEL SKIP lamp will light up

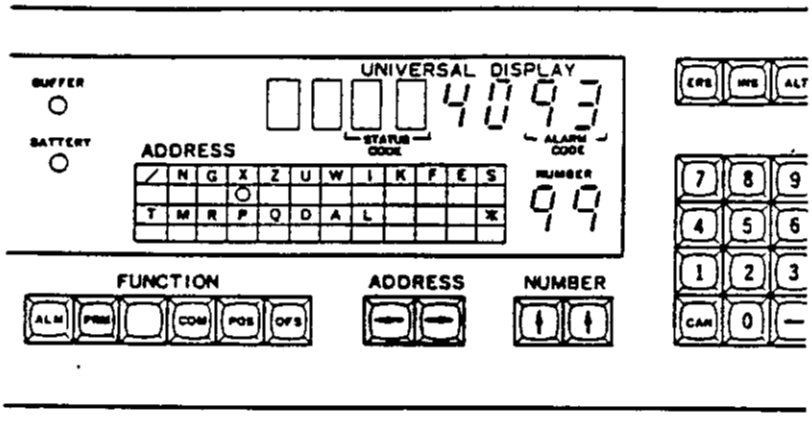


Fig. 4.5.4

4.5.4 ADDRESS DISPLAY OF TAPE MEMORY (PARAMETER NO. 99)

Parameter No. 99 is used to display the address of part program in memory which is being executed With parameter No. set at 99, the number of characters from first address to the address which the pointer in the control indicates is shown on the universal display.

EXAMPLE

· Set the parameter No at 99 after storing the tape data in the EDT mode, and the total number of characters in the tape is displayed on the universal display

Set the parameter No. at 99 during operation in the MEM mode, and the position of active address in memory is displayed continuously. The display shows the number of characters in the blocks from the first of the program to the block preceding the latest one in buffer register

4.6 EDIT †

4.6.1 DISPLAY OF STORED PART PROGRAM

Stored program contents can be displayed, and checked by the operator. The display operation is made in the EDT mode Setting the NUMBER at 00 permits scanning of effective addresses in a block. Addresses not commanded are skipped and only effective address data is displayed. This makes check of stored part program easy. Take the following procedure.

1. Set the MODE SELECT switch to EDT
2. Depress the COM key, and it lights up.
3. Set the NUMBER at 00.

   Depressing the both NUMBER keys makes the NUMBER display 00
4. Depress the RESET key.

   Memory is rewound and LABEL SKIP lamp lights up
5. Depress sequential search key ⊡ to call the first block of the program.

   BUFFER lamp lights up and address indicator automatically indicates the first address of the block.
6. Check the command data shown on the universal display. Then depress the ADDRESS key ⊡ .

   Address indicator stops at the next effective address, skipping the addresses not commanded.
7. Repeat step 6 to check all the command data in the block

   At the block end, address indicator indicates * (end of block code) on the ADDRESS display.

   Notes

   · Depressing ⊡ ADDRESS key at the block end causes the first address of the block to appear again

- Depressing [←] ADDRESS key makes the scanning of the addresses from the last.
- Depressing both ADDRESS keys always moves address indicator to the first address of the block.

8. Depress the [↓] key to read out the next block. Check the command data repeating steps 6 and 7.
9. Depressing [↑] key calls the preceding block. Selected block can be searched with the AS key.
10. On completion of reading out the memory and checking the command data, depress the RESET key to rewind the memory.

4.6.2 EDITING STORED PART PROGRAM+

Stored part program can be edited and modified in the EDT mode using the following keys.

ERS (erase) . . . To delete block

INS (insert) . . . To insert block

ALT (alter) . . . To modify block: erasing, inserting, and modifying address data in a block.

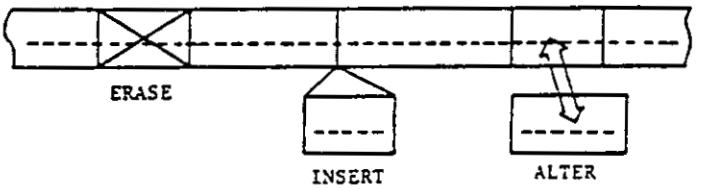


Fig. 4.6.2.1

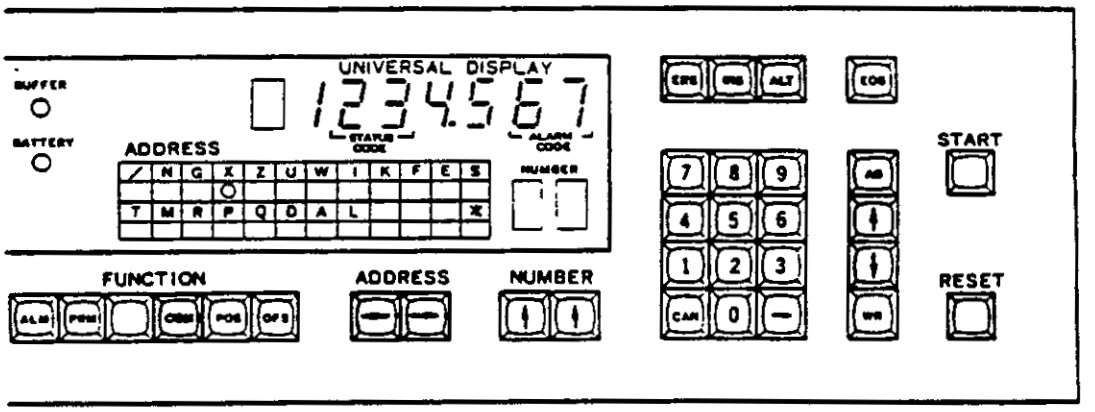


Fig. 4.6.2.2

· Deleting part program block (ERASE key)

1. Check the contents of the block to be deleted after searching with ↓, ↑ or AS key. See 4.6.1 Display of Stored Part Program. Then BUFFER lamp will be illuminated.

2. Depress the ERS key to delete the searched block from the memory. The next block is read out and its data is automatically displayed on the universal display. BUFFER lamp remains on.

   The block is deleted each time the ERS key is depressed.

3. If the block is accidentally erased, insert the deleted data again following the procedure of "Insertion of block" given below.

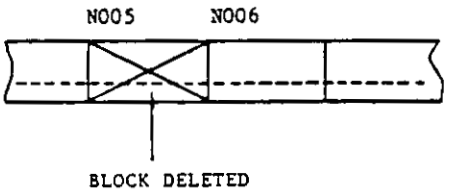


Fig. 4.6.2.3

NOTE Where the NUMBER is set at 00 in the EDT mode, address selection cannot be allowed because scanning of effective addresses is performed.

· Insertion of block (INSERT key)

1. Select the block preceding the new block to be inserted using ↓, ↑ or AS key and check the block contents.

   The new data is to be entered immediately following selected block. Depress the RESET key to insert the data of the first block. See Fig 4.6.2.4.

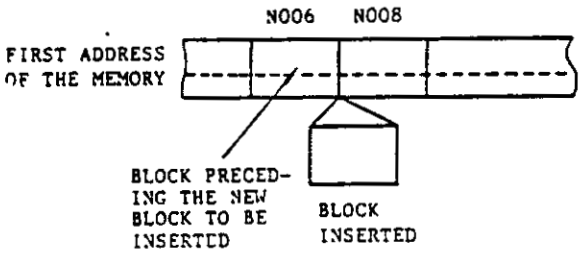


Fig. 4.6.2.4

2. Depress the INS key, and it is illuminated and BUFFER lamp is extinguished. Universal display will be blank.

3. Insert the data of a block according to steps 3 and 4 in 4.3 2 Writing Command Data by MDI.

   BUFFER lamp lights up, and the display shows the data as it is entered.

4. Depress the INS key again.

   Block insertion is executed, and the light is extinguished. The display shows the data of the inserted block. BUFFER lamp goes out.

5. Repeat steps 2 through 4 to insert new blocks in sequence.

6. Check the new data by displaying the inserted block and preceding and following blocks. Setting the NUMBER at 00 makes checking easy.

· Modifying part program block (ALTER key)

1. Select the block to be altered using ↓ and ↑ keys or AS key and check it.

   BUFFER lamp will be lit.

2. Select the address of the data to be corrected with the ADDRESS key Last command data is displayed on the universal display. To select the new address character, set the number except 00 by operating NUMBER key.

3. Input new command data from DATA keyboard. Then depress the WR key.

   When the selected address data should be erased, depress CAN key and then depress WR key.

4. Repeat steps 2 and 3 until desired data of a block are built.

5. Check the corrected data. Then depress the ALT key.

   New block is stored into memory after the old block is deleted.

6. Check the new data by displaying the inserted block and its preceding and following block. Setting the NUMBER at 00 makes checking easy.

4.6.3 OUTLINE OF EDIT OPERATION

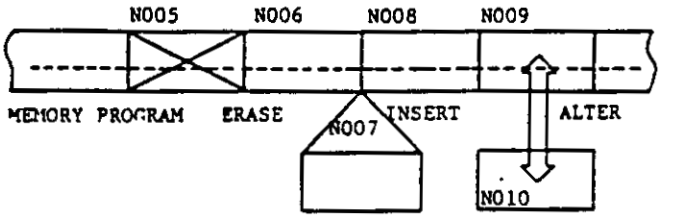


Fig. 4.6.3

Table 4.6.3

Operation		ERASE	INSERT	ALTER
Display before edit operation		○ Block to be deleted (ex. N005)	○ Block preceding new data to be inserted (ex. N006)	○ Block to be corrected (ex. N009)
Operation procedure	1	○ ERS key	× INS key (light ON)	○ Data modification by MDI operation
	2	——	○ Writing command data by MDI operation	× ALT key
	3	——	× INS key (light OFF)	——
Display after edit operation		○ Block immediately following the deleted block (ex. N006)	× Inserted new block (ex. N007)	× Modified block (ex. N010)
Status of BUFFER lamp after edit operation		ON	OFF	OFF

Notes:

1. Edit operation should be made in the EDT mode.
2. "○" or "×" in each column shows "on" or "off" conditions of BUFFER lamp, respectively.
3. For operating procedure, see 4.6.2 Editing Stored Part Program.

4.7 PUNCHOUT OPERATION †

4.7.1 TAPE PUNCHER+

Part program or tool offset values stored in memory can be punched out in tape using tape puncher. The tape puncher should be separately provided.

Tape Puncher
Type FACIT 4070

Punching Speed
75 characters/sec

Dimensions·
432 wide x 220 deep x 198 high (mm)

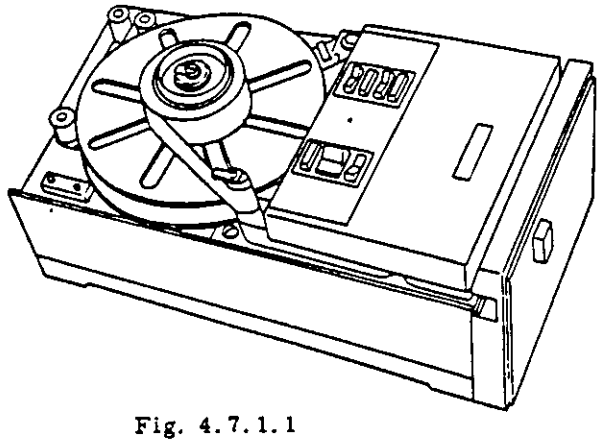
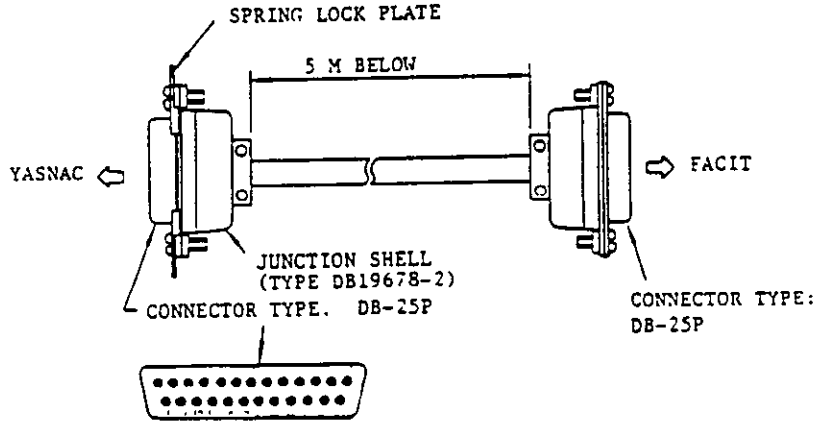


Fig. 4.7.1.1



CONNECTIONS BETWEEN THE CONTROL AND THE PUNCHER

YASNAC PIN NO.		FACIT PIN NO.
1	Ch1	1
2	Ch2	2
3	Ch3	3
4	Ch4	4
5	Ch5	5
6	Ch6	6
7	Ch7	7
8	Ch8	8
9	Ch9	9
10	SD	10
11	P1	11
12	PR	12
20	Err. 1	20
21	TL	21
25	0V	25

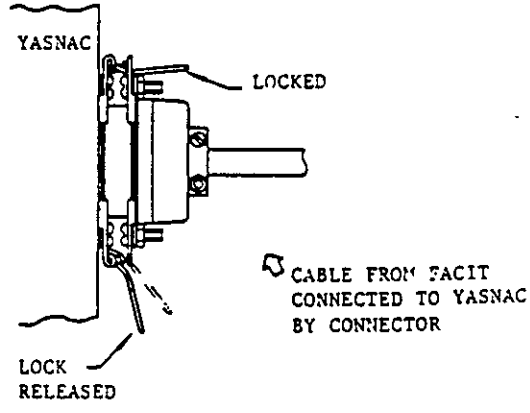


Fig. 4.7.1.2

Operation of FACIT Punchers

Before punching NC tape, take the following procedure. For details, refer to the instructions for FACIT 4070 punchers.

1. Set the supply voltage selecting switch to the AC voltage applied.

Fig. 4.7.1.3

2. Adjust the tape width setter to eight-channel tape width.

3. Set the eight-channel paper tape to the tape puncher.

4. Turn off the control.

   Connect the FACIT 4070 to the control using the cable provided. Receptacles for the puncher cable are provided in the tape reader box.

5. Connect AC power supply to FACIT 4070.

6. Turn on the FACIT power switch, and READY lamp will light up.

7. Feed the tape by depressing FEED HOLES SWITCH on the FACIT.

8. Turn on the control.

   The FACIT puncher is ready to operate.

4.7.2 PUNCHOUT OF NC TAPE†

1. Set the MODE SELECT switch to EDT.

2. Depress the RESET key, and LABEL SKIP lamp lights up.

3. Depress the COM key, and it lights up.

   Any FUNCTION key is available except the OFS key. However, select the COM key in general.

4. Set the NUMBER at 61 which is the function number for punch operation.

5. Check to see the puncher is ready to operate.

6. Depress the START key.

   Tape puncher starts punching operation and automatically stops, when memory contents have been punched out on the tape.

7. To interrupt punchout operation, depress the RESET key.

   To resume the operation, take the steps from 5.

NOTES

- Tape is punched out according to the coding selected by parameter No. 82.

  Where parameter 82 is "0," ··· EIA code,
  parameter 82 is "1," ··· ISO code.

- If the number of punched out characters in a block is odd, a space code for TV check is automatically punched.

- Each end of the NC tape is provided with feed holes.

- If the Error lamp on control panel of the FACIT lights up due to the trouble in the FACIT 4070, the control automatically stops punching operation. After clearing the troubles such as excessive tape tension and tape shortage, start punching operation following procedures described above.

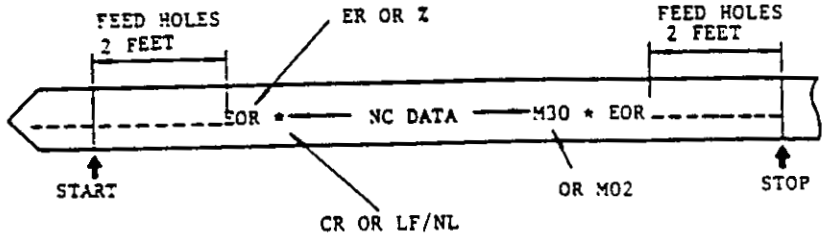


Fig. 4.7.2

4.7.3 PUNCHOUT OF TOOL OFFSET VALUE†

1. Set the MODE SELECT switch to EDT.
2. Depress the RESET key, and LABEL SKIP lamp lights up.
3. Depress the OFS key, and it lights up.

   If the other FUNCTION key than the OFS key is selected, part program is punched out.
4. Set the NUMBER at "61" for punching operation.
5. Check to see that the tape puncher is ready to operate.
6. Depress the START key with the OFS key held in.

   The puncher starts and automatically stops when the offset value has been punched out.
7. To interrupt the punch operation, depress the RESET key.

   To resume the operation, take the step from 1.

NOTES

- Tape is punched out according to the coding selected by parameter No. "82."

  Where parameter No. "82" is set at "0," EIA code is selected and "1," ISO code
- If the number of punched out characters in a block is odd, a space character for TV check is automatically punched. Each end of the NC tape is provided with feed holes, Fig. 4.7.2.
- If the Error lamp on the FACIT control panel lights up due to the trouble in the FACIT 4070, the control automatically stops punching operation. After clearing the troubles such as excessive tape tension and tape shortage, start punching operation following procedures in 4.7.2 Punchout Operation of NC Tape.

4.7.4 OUTLINE OF TAPE DATA STORING AND PUNCHING OPERATION†

Select EDT mode.

↓

Depress RESET key.

↓

Selection of FUNCTION keys.

- Any key other than OFS key for NC tape.
- OFS key for offset value stored tape

↓

Set the NUMBER at

- Other than "61" and "62" for storing
- "61" for punching
- "62" for collating.

↓

- Depress the START key for punching NC tape
- Depress the START and OFS keys at the same time for punching offset value stored tape.

Fig. 4.7.4

4.8 COLLATING OF STORED PROGRAM AND OFFSET VALUE

4.8.1 COLLATING OF STORED PROGRAM

To check whether the memory contents agree with NC tape contents, proceed as follows:

1. Set the MODE SELECT switch to the EDT.
2. Depress the RESET key.

   Memory is rewound and LABEL SKIP lamp is illuminated.

3. Depress the COM key, and it will light up.

4. Set the NUMBER at "62" which is the function number for collating operation.

5. Set the NC tape to the tape reader.

   Be sure that LABEL SKIP lamp reamains on.

6. Depress the START key.

   The tape reader starts reading, and the memory contents are collated with the tape data. When it reads rewind stop code, it automatically stops.

7. If disagreement with NC tape is detected, INPUT ERROR lamp lights up and tape reader stops. Alarm code "18" is displayed.

NOTES:

- Collation is made only on significant information. Disregarded characters such as space, tab, and ALL MARK are ignored during collating operation.
- If the stored data is different from programmed data because of omitted leading zero, INPUT ERROR lamp lights up.

4.8.2 COLLATING OF STORED OFFSET VALUE

To check whether the memory contents agree with offset value stored in tape proceed as follows·

1. Set the MODE SELECT switch to the EDT.

2. Depress the RESET key, and LABEL SKIP lamp will go on.

3. Depress the OFS key, and it will be on.

4. Set the NUMBER at "62" which is the function number for collating operation.

5. Set the source tape to the tape reader.
   Be sure that LABEL SKIP lamp remains on.

6. Depress the START key with the OFS key held in.
   Tape reader starts reading, and memory contents are collated with the tape data. It automatically stops when rewind stop code is encountered.

7. If disagreement with the tape data is detected INPUT ERROR lamp lights up and tape reader stops. Alarm code "16" is displayed.

NOTES Tool offset number not effective in the control is ignored, if commanded in tape.

4.9 OUTLINE OF OPERATION IN THE EDT MODE †

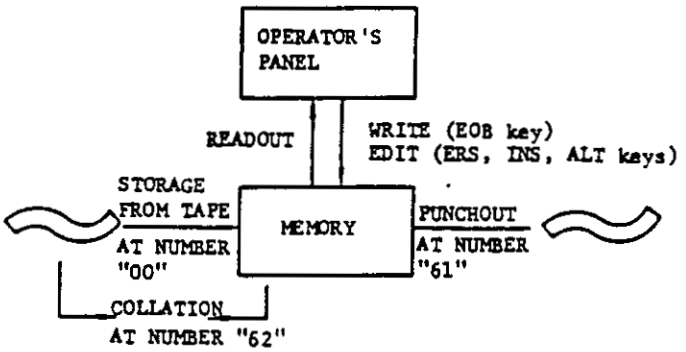


Fig. 4.9.1

Table 4.9.1

	Function key	NUMBER	Keys for starting
Storing NC data from tape	COM	Other than "61" and "62"	START
Storing offset value from tape	OFS	Other than "61" and "62"	OFS & START
Punchout of stored program	COM	"61"	START
Punchout of tool offset value	OFS	"61"	OFS & START
Collating of stored program	COM	"62"	START
Collating of stored offset values	OFS	"62"	OFS & START
Readout of stored part program	COM	—	—
Storing part program through DATA keyboard	COM		
Editing stored part program	COM		

5. TAPE READER COMPARTMENT

5.1 TAPE READER

5.1.1 TAPE FEED AND SYSTEM NO. SWITCHES

These switches provided above tape reader are exposed by opening door for tape reader compartment.

· TAPE FEED switch

This is a spring-return switch and used to feed and rewind the tape manually. Setting it to FORWARD feeds the tape. To rewind the tape, set it to REVERSE. The switch cannot be activated during automatic and manual operation or with tape bail pushed up.

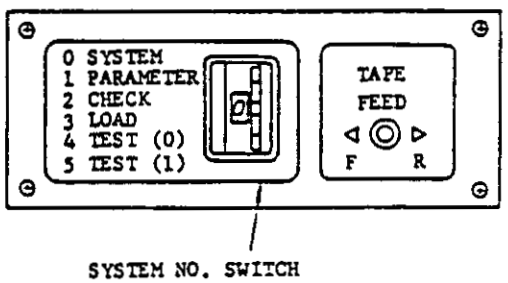


Fig. 5.1.1

· 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 4.1.14 TAPE FEED and SYSTEM NO. switches and 4.3.9 Writing Parameters.

5.1.2 TAPE READER

· Light source

LED is used for light source. It does not need maintenance operation except for removal of dust.

· 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.

· 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.

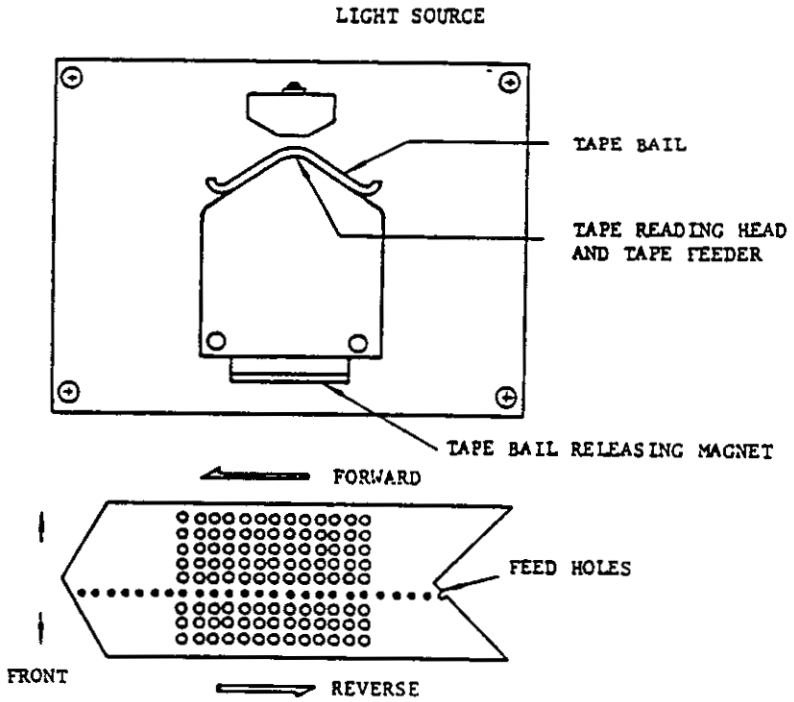


Fig. 5.1.2

5.1.3 TUMBLE BOX

Tumble box is provided below the tape reader to accomodate NC tape. The NC tape is easily taken out by pulling a polyester 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.

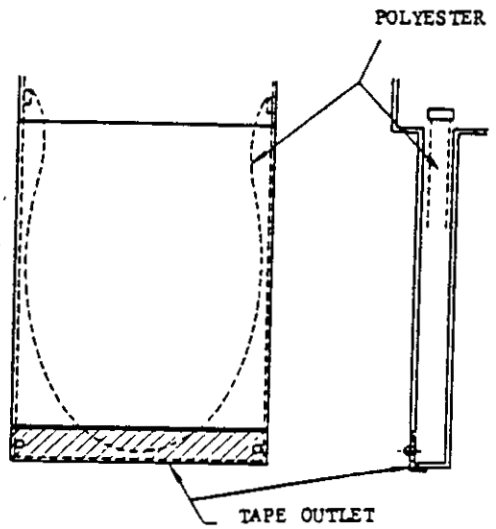


Fig. 5.1.3 Tumble Box

5.2 TAPE REELS

Tape reader with 6 inches diameter reels is available as an option. It accomodates an NC tape 260 feet (80 meters) long in case of 0.0042 inch (0.108 mm) thick.

To mount the NC tape, proceed as follows.

1. Take out the right hand side reel, pressing a stopper pin.
2. Set the tape on the reel. Be sure the tape does not slide.
3. Mount the reel on rotating shaft at right with the stopper pin positioned in the slot of handler.
4. Draw out the tape 2 feet (70 cm) or more, load it through the tape reader and set it to the left hand side reel. Wind the tape three times or more so that the tape does not slide.
5. Holding the reel not to loosen the tape, detach the tension arm from the arm rest and push down slowly. Fig. 5.2.2 shows the tape properly mounted on the tape reels. Then tape reader is ready to operate.

   When the reels are not used, place the tension arms on the arm rests as shown by broken lines in Fig. 5.2.2.

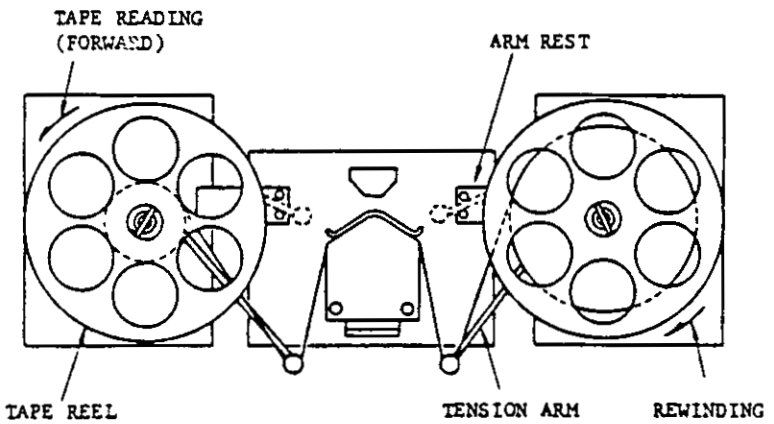


Fig. 5.2.2 Tape Reader with Tape Reels

6. PENDENT CONTROL STATION FOR MACHINE

6.1 SWITCHING UNITS ON THE PENDENT CONTROL STATION

Fig. 6.1.0 shows the layout of switching units on the pendant control station. For details, refer to the machine tool builder's manual.

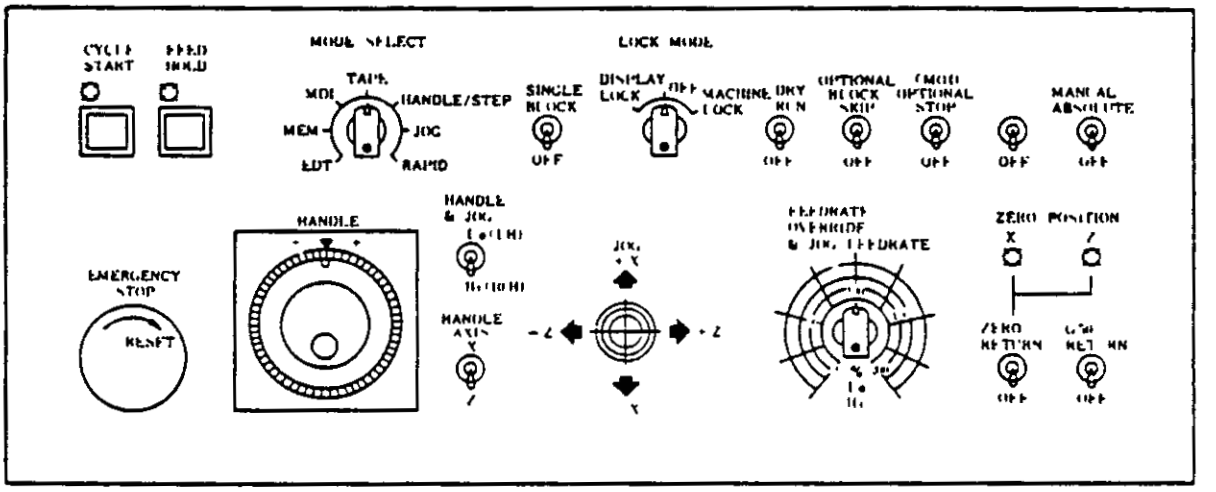


Fig. 6.1.0 Layout Example of Switching Units on Pendent Control Station

6.1.1 MODE SELECT SWITCH

This switch selects operation mode of the NC system and consists of 7 positions (RAPID, JOG, HANDLE, TAPE, MDI, MEM, EDI). RAPID, JOG, and HANDLE modes are called manual operation mode, and TAPE, MDI, and MEM, automatic operation mode in this manual.

RAPID· To make the tool traverse rapidly or return to zero by manual operation.

JOG. To feed the tool continuously by manual operation. Feedrate is set by FEEDRATE OVERRIDE switch.

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 lever.

TAPE To automatically control the NC system with NC tape.

MDI. To enter the block of data through the DATA keyboard and control the system automatically with the data.

MEM To automatically control the system with the stored part program.

EDT. To store the part program into memory and edit the part program.

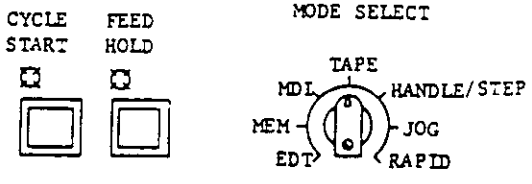


Fig. 6.1.1

6.1.2 CYCLE START PUSHBUTTON

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

Depress this pushbutton to temporarily suspend 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. Chamfering and retracting motion at G92 and G76 commands cannot be interrupted.

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 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

6.1.5 EMERGENCY STOP PUSHBUTTON

Depress this pushbutton to emergency-stop the machine. The servo power is turned off and the machine is stopped immediately by dynamic brake. The NC ALARM lamp lights and alarm code "33" 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 "31" replaces "33."
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.

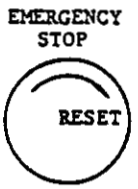


Fig. 6.1.5

6.1.6 HANDLE DIAL⁻ (MANUAL PULSE GENERATOR)

The dial is used as a manual pulse generator to feed the tool manually with the MODE SELECT switch set to the HANDLE. HANDLE operation is effective for an axis. Procedure of HANDLE operation is as follows:

1. Set the MODE SELECT switch to the HANDLE
2. Select the axis to be operated with HANDLE AXIS X-Z select switch.
3. Set the move amount per graduation of the dial by setting HANDLE & JOG Lo-Hi select switch to Lo or Hi.

   Lo: 0.002 mm/graduation (0.0001"/graduation)
   Hi: 0.01 mm/graduation (0.001"/graduation)
4. Rotate the dial to move the selected axis.

   Turning it clockwise causes the axis to move in the plus direction. The axis moves in the minus direction by turning it counterclockwise.

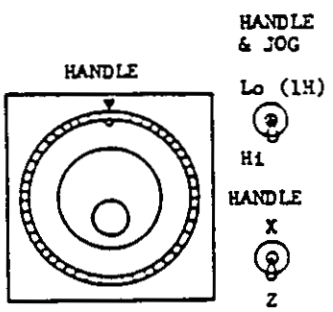


Fig. 6.1.6

6.1.7 HANDLE AXIS X-Z SELECT SWITCH⁺

This switch is used to select an axis to be operated.

6.1.8 HANDLE & JOG LO-HI SELECT SWITCH

This switch is used to:

- Select the range of JOG feedrate in the JOG mode.
- Select the value corresponding to a single graduation of the HANDLE dial in the HANDLE mode.

6.1.9 JOG LEVER

This lever is used to feed the tool manually.

- With this lever set to +X, -X, +Z or -Z, the tool can be moved rapidly to the set direction in the RAPID mode
- This lever moves the tool at the speed set by JOG FEEDRATE switch in the JOG mode.
- Where the control is not provided with a HANDLE dial⁻ (manual pulse generator), the tool is moved by the value set by HANDLE & JOG Lo-Hi switch each time the JOG lever is activated in the HANDLE mode.

6.1.10 FEEDRATE OVERRIDE & JOG FEEDRATE SWITCH

In case of automatic operation mode (TAPE, MEM and MDI), this switch is used to adjust the feedrate by 10% from 0 to 200% of the programmed feedrate by F-code. Threadcutting of G32, G92, G76 will be performed at the feedrate specified with an F function at whatever position the switch may be set.

In the JOG mode, jog feedrate is selected with this switch. Feedrate is available in Hi and Lo range. For each range, 21 steps are available. Hi and Lo range selection is made by HANDLE & JOG Lo-Hi select switch.

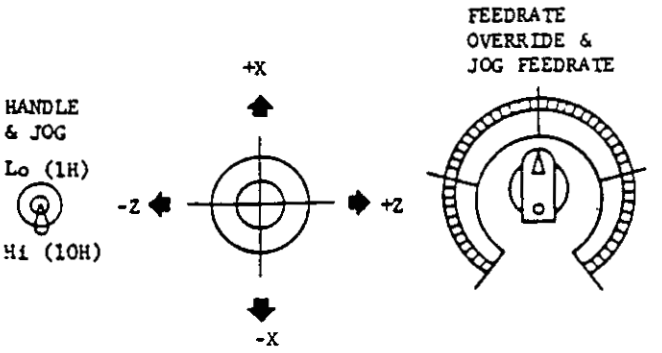


Fig. 6.1.10

Table 6.1.10

STEP	FEEDRATE OVERRIDE	JOG FEEDRATE (Lo)	(Hi)
0	0 %	mm/min	mm/min
1	10		
2	20		
3	30		
4	40		
5	50		
6	60		
7	70		
8	80		
9	90		
10	100		
11	110		
12	120		
13	130		
14	140		
15	150		
16	160		
17	170		
18	180		
19	190		
20	200		

Note: JOG FEEDRATE depends on the machine tool. For definite values, refer to the machine tool builder's manual.

6.1.11 ZERO RETURN SWITCH (MANUAL ZERO RETURN)

Turning on this switch causes the tool to return to the fixed original point by manual operation. For operation, refer to 6.2.1 Manual Zero Return.

6.1.12 ZERO POSITION LAMPS FOR X AND Z AXIS

These lamps indicate that the tool is positioned at the fixed original point. When the tool has been positioned at fixed original point manually or by G28 automatically on each axis, the lamps light up. The lamps go off when the tool leaves the point.

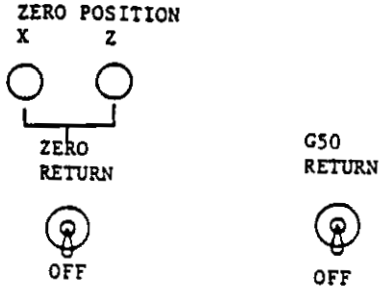


Fig. 6.1.12

6.1.13 G50 RETURN SWITCH*

This switch is used to return the tool manually to setup point (programmed point of G50). For operation of the switch, see 6.2.2 G50 Return.

6.1.14 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.

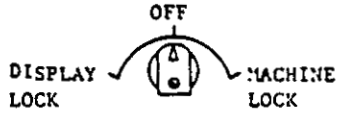


Fig. 6.1.14

"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.

"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.

6.1.15 DRY RUN SWITCH

With the DRY RUN switch turned on in the TAPE MDI or MEM mode, the tool moves at the speed selected by the FEEDRATE OVERRIDE switch and HANDLE & JOG Lo-Hi switch, ignoring all programmed T-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 No "02."

Where parameter No. 02 = "0"

Rapid traverse (G00) is made at usual rapid traverse rate. Range of rapid traverse rate can be selected by setting parameter No "01."

Parameter No. "01"	Rapid Traverse Rate Range
"0"	Hi
"1"	Lo

Where parameter No. 02 = "1"

Rapid traverse (G00) is made at any speed by FEEDRATE OVERRIDE switch.

NOTES

- Setting the DRY RUN switch during automatic operation causes the next block to be executed in the DRY RUN mode
- During dry run operation, the traverse speed set by parameter No "01" and "02" is effective immediately after the parameter setting.

6.1.16 OPTIONAL BLOCK SKIP SWITCH

This switch, when set on, ignores the block starting from optional block skip character "/" in the automatic operation mode (TAPE and MEM). To cancel this function, turn off the switch. The "/" character is placed before the sequence number.

Operation of this switch is not effective for the block stored in the buffer register and active register. During the automatic operation Block Skip is effective for the block to be entered in the buffer register.

6.1.17 OPTIONAL STOP SWITCH

This switch is to execute M01 command in automatic operation mode (TAPE, MEM or MDI).

When the switch is on, the program stops on completion of M01 command, while CYCLE START pushbutton remains illuminated. When the control catches FIN signal, the light is extinguished. To restart the program, depress the CYCLE START button. When the switch is off, M01 command is ignored.

Operation of the switch is not effective for the block being executed. During the automatic operation, the switch acts for the next block.

6.1.18 MANUAL ABSOLUTE SWITCH†

· When MANUAL ABSOLUTE switch is on.

When automatic operation is restarted after interrupted by manual operation, the tool performs the rest of the command in the interrupted block from the end point of manual operation. The tool moves in parallel with the path specified by the program.

When the command of the next block is G00 or G01, the tool moves automatically to the target coordinate specified by the program. Then the operation is performed according to block of data.

When the command of the next block is G02 or G03 (circular interpolation), the interpolation is performed in parallel with program command The tool automatically returns to the target coordinate when G00 or G01 is commanded after the interpolation.

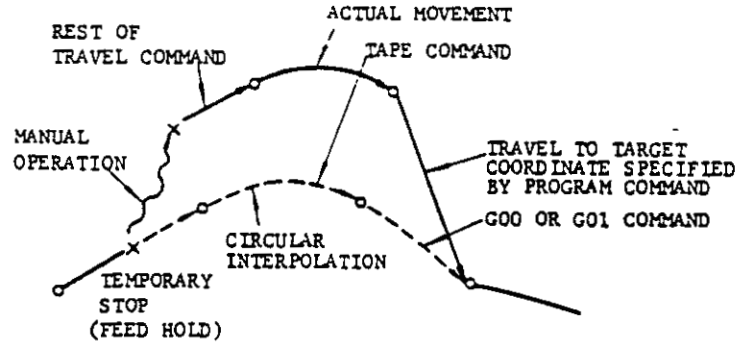


Fig. 6.1.18.1 Tool Movement with MANUAL ABSOLUTE Switch On

· 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.

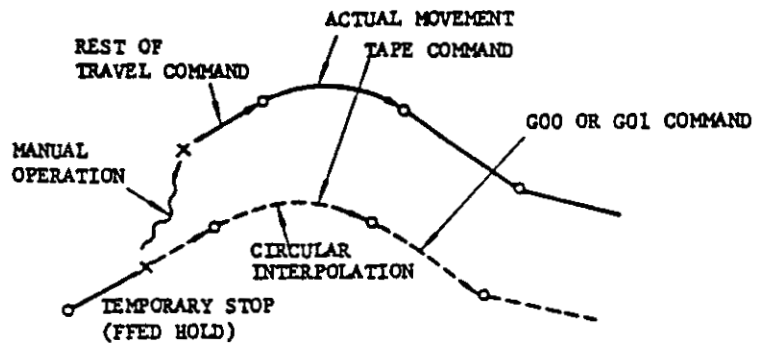


Fig. 6.1.18.2 Tool Movement with MANUAL ABSOLUTE Switch Off

6.2 OPERATION FOR MANUAL ZERO RETURN AND G50 RETURN

6.2.1 MANUAL ZERO RETURN

To return the tool to the fixed original point manually, proceed as follows.

1. Set the MODE SELECT switch to RAPID or JOG
2. Move the tool to the point which is out of deceleration range manually. If the tool is in the deceleration range, Zero Return is not executed (alarm code "22" is displayed).

   Care should be taken in moving the tool so that the tool does not contact the workpiece.
3. Turn on ZERO RETURN switch.
4. Move the tool to the fixed original point on each axis by operating the JOG lever.

   The tool is moved in the same manner as manual feed operation, slowed down after deceleration starting point, and automatically stopped at the zero point.
5. ZERO POSITION lamp (APX or APZ) indicates the axis on which the tool has reached the zero point.

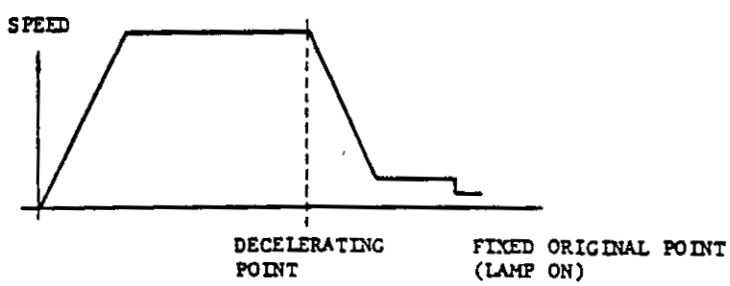


Fig. 6.2.1

NOTES

- The tool cannot be advanced manually unless the ZERO RETURN switch is turned off, where Zero Return operation has been completed. Moving back the tool from the fixed original point is possible.
- If the tool is in the deceleration range, the tool cannot be moved by Zero Return operation, and NC ALARM lamp lights up (Alarm code "22"--Zero Return error--is indicated).

  The error is eliminated by depressing the RESET key.
- With the MACHINE LOCK switch ON, Zero Return operation is not effective.
- After completion of Zero Return, the tool offset cancelled.

6.2.2 G50 RETURN+

To return the tool manually to the setup point of coordinate system (point commanded by G50), proceed as follows.

1. Set the MODE SELECT switch to RAPID or JOG
2. Turn on G50 RETURN switch.
3. Operate the JOG lever to move the tool to G50 setup point and keep the lever operated. The tool is moved in the same manner as manual feed operation and automatically stopped at the setup point. If the JOG lever is set to the opposite direction of the setup point, the tool does not move.
4. To move the tool manually on the axis which the tool has reached setup point, turn off the switch.
5. Move the tool on the other axis to the setup point by operating the JOG lever. On completion of G50 Return, turn off the G50 RETURN switch.

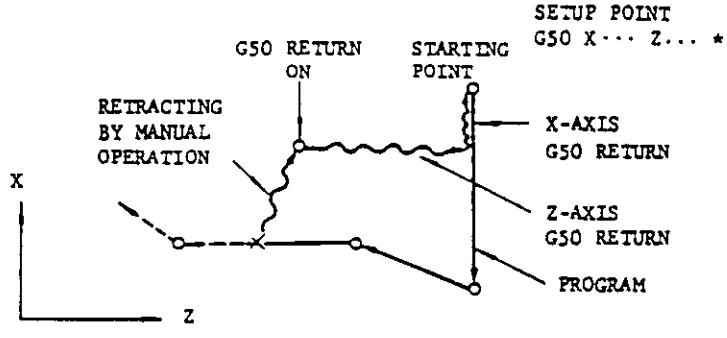


Fig. 6.2.2

NOTES·

- With MACHINE LOCK switch ON, G50 Return is effective. Though the tool does not move, the display is operated.
- Tool offset is cancelled by G50 Return. When MANUAL ABSOLUTE switch is on, returning motion of manual-move amount is cancelled by executing G50 Return.
- Where the several G50 commands are programmed in the program, the tool is moved to the latest setup point.
- To restart the tool in the automatic operation from the G50 setup point, address search operation should be made after depressing the RESET key

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 totally-enclosed, dustproof enclosure to shut out surrounding air. If the door is open, lock closely it by turning two door lockers with a large screwdriver (minus). 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 POWER ON pushbutton on the operator's panel, and the control power is supplied and then the cooling fans will be started. Make sure that wind blows out from the exhaust ports of the upper side of the control.
3. Depress 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 by the indication of alarm code. Refer to 4.3.12 Alarm Code Display.

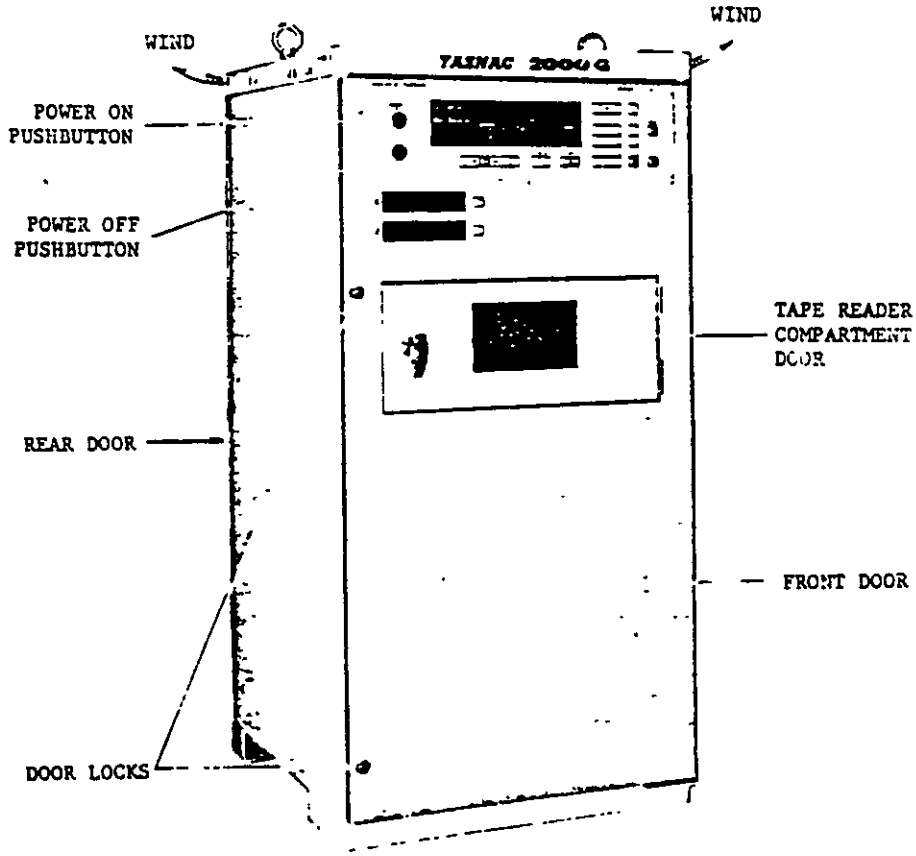


Fig. 7 2 0

7.3 MANUAL OPERATION

When the machine is operated manually, MODE SELECT switch on the pendant control station must be set to RAPID, JOG or HANDLE mode according to the object of operation.

· Operation in RAPID Mode

1. Set MODE SELECT switch to RAPID position.
2. Operate the JOG lever in any direction of +X, -X, +Z and -Z, and the machine will be moved in the specified direction of axis at rapid traverse rate.
3. Rapid traverse rate is interchangeable either Hi (High) or Lo (Low) by setting parameter No. 01 at 0 or 1.

   0 means Hi (High Speed)
   1 means Lo (Low Speed)

   The contents of parameter No 01 can be written without operating SYSTEM NO. switch above tape reader.

· Operation in JOG Mode

1. Select JOG mode using MODE SELECT switch.
2. Set the feedrate range with HANDLE & JOG Lo-Hi switch.
3. Determine the feedrate with FEEDRATE OVERRIDE & JOG FEEDRATE switch.
4. Move the machine with JOG lever operation. The machine can be moved at the specified speed in the axis direction of the JOG lever.

· Operation in HANDLE Mode

1. Select HANDLE mode of MODE SELECT switch.
2. Select the axis with HANDLE AXIS X-Z switch.
3. Determine the move amount of the machine corresponding to one scale of HANDLE dial by using HANDLE & JOG Lo-Hi switch.

   Lo: 0.002 mm (0.0001") per graduation
   Hi: 0.01 mm (0.001") per graduation
   (X-axis in diameter)

4. Rotate HANDLE dial, and the machine will move to the specified direction. Turning the dial clockwise causes the machine to move in the plus direction, and turning it counterclockwise, in the minus direction.

7.4 PREPARATION FOR PROGRAMMING OF ABSOLUTE ZERO POINT

In order to start the machine by automatic operation, the machine must be set to the programmed starting position. The procedure for this operation is determined by the engaging program. For the details, refer to the machine tool builder's manual.

· In the case using Automatic Zero Return (G28)

Where G28 is programmed at the beginning of program, move the machine manually into the area where Zero Return can be performed. If the starting position of the machine is between decelerating point and fixed original point, it causes Zero Return error and the alarm code "22" will be displayed in CYCLE START operation.

EXAMPLE

```
EOR  *
N001 G28  *
N002 G50  X__ Z__*
```

· In the case using Manual Zero Return

Where G28 is not programmed and the coordinate system is set to the fixed original point, move the machine manually to the fixed original point. Refer to 6.2.1 Manual Zero Return.

EXAMPLE

```
EOR *
N001 G50  X__ Z__*
  .
  .
  .
```

· In the other case

Where the starting point of tool is set up on the basis of the workpiece without using fixed original point, operate the control as follows

1. Select the reference tool, and then set the sample workpiece.

Manual operation means the operation in RAPID, JOG or HANDLE mode in this manual.

2. Set the tool to the end of workpiece (reference face) by manual operation to decide the absolute zero point of Z axis.
3. Reset the current position display of Z axis, and the position is set as the absolute zero point of Z axis.
4. Set the tool to the definite point of peripheral surface (reference surface) of workpiece manually to make the center line of the workpiece the absolute zero point of X axis. Cut off the workpiece, if necessary.
5. Turn on MACHINE LOCK switch. Then set the diameter value of workpiece on the current position display, and the absolute zero point of X axis is on the center line of the workpiece.
6. Turn off MACHINE LOCK switch.
7. Move the each axis (X and Z) to setup point respectively by manual operation.

NOTE: Where the above operation is correctly done, the tool position offset value for the tool is zero.

7.5 PREPARATION FOR AREA CHECK OPERATION

Before operating the machine, the Area Check operation is made to detect automatically without the motion of tool whether tool position commanded in the block to be executed is within the specified range of tool or not. When the control is operated in TAPE, MEMORY or MDI mode, with the contents of parameter No. 03 set at "1," the Area Check is executed. If the tool is out of the range, INPUT ERROR lamp lights and then the machine stops (Alarm code "15" is displayed). Before starting the automatic operation, the proper area for the machine must be specified.

The area is specified with parameter 30 and 31 in the following procedure.

- Designate the distance with "+" sign from fixed original point to the plus end of the area for each axis by using parameter No. 30-X and 30-Z.
- Designate the distance with "-" sign from fixed original point to the minus end of the area for each axis by using parameter No. 31.
- The sign ("+" or "-") is determined by a position of programmed boundary line of the area If the line is in the positive side of the fixed original point, the sign is "+," and the line is in the negative side, the sign is "-."
- The data to be written is the distance from the fixed original point to the boundary line. Therefore, the data for X axis is not a value of diameter.

Precausion of Area Check Operation

- Where the Area Check operation is not made, set the contents of parameter No. 03 at "0."
- If MACHINE LOCK is turned on, the Area Check is not performed. But where the Area Check is required to check the NC tape, with MACHINE LOCK turned on, set the content of parameter No. 04 at "1."

Table 7.5.0

Parameter No. 03 / Parameter No. 04	"0"	"1"
"0"	Area Check is not performed.	Where MACHINE LOCK is off, Area Check is performed.
"1"	Area Check is not performed.	Area Check is performed regardless of on-off status of MACHINE LOCK

- Area check is not performed in manual operation even if the machine is operated out of the area.

7.6 OPERATION IN TAPE AND MEMORY MODE

1. Make sure that NC ALARM lamp is not illuminated. If illuminated, detect and eliminate the cause by the indication of alarm code. Refer to 4.3.12 Alarm Code Display.
2. Check and correct the stored offset values, and then put the machine in the correct start point.
3. Set the switches on the pendant control station to the proper positions
   - MODE SELECT switch
   - SINGLE BLOCK toggle switch
   - LOCK MODE (DISPLAY LOCK and MACHINE LOCK) 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 operator's panel. Then LABEL SKIP lamp will be illuminated and the memory will be rewound.

6. Depress CYCLE START button to give the 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. Refer to 4.3.3 Current Position Display

3. Set MODE SELECT switch to manual operation mode (HANDLE, JOG or RAPID), and the machine will be manually operated.

4. Return the tool 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 restart with the automatic operation.

NOTES

- Where MODE SELECT switch is changed without depressing FEED HOLD pushbutton.
  a. When the automatic-mode (TAPE, MDI or MEM) is changed to the manual-mode (HANDLE, JOG or RAPID), the machine is rapidly slowed down and stopped.
  b. When the automatic-mode is changed to other automatic-mode, the machine is stopped at the block end.
- Where the machine is restarted by depressing CYCLE START button, the tool path shifted during manual operation will be changed by ON-OFF operation of MANUAL ABSOLUTE switch. Refer to 6.1.18 MANUAL ABSOLUTE Switch.

7.8 AUTOMATIC OPERATION IN MDI MODE

1. Set MODE SELECT switch to MDI position.

2. Write the one block data by MDI operation, and BUFFER lamp on operator's panel lights. Refer to 4.3.2 Writing Command Data by MDI.

3. Depress CYCLE START button, and automatic operation can be executed in MDI mode. BUFFER lamp will be off.

7.9 MDI OPERATION INTERRUPTING AUTOMATIC OPERATION

When modifying the block data interrupting operation in TAPE or MEM mode, the following operation should be done.

1. Turn on SINGLE BLOCK switch, and the operation is stopped temporarily after the completion of executing block. At the same time, the next block data is stored in the buffer register.

2. Display the contents of the data on UNIVERSAL DISPLAY according to 4.3.1 Display of Command Data, and check it.

3. Set MODE SELECT switch to MDI position.

4. Modify the data referring to 4.3.2 Writing Command Data by MDI

5. After modifying the data, set 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 by the modified data.

NOTES:

- While the tip nose radius compensation (G41, G42, G43 and G44) is executed, MDI operation can not interrupt automatic operation, since the next two or three blocks are stored in the buffer register.
- While the multiple repetitive cycle (G70 to G76) is commanded, MDI operation is also impossible.
- Excepting the above two cases, MDI operation is possible. Although, the next block of data is read ahead in the buffer register usually, additional or altered data can be written into the buffer register by MDI operation.

7.10 PREPARATION FOR TURNING OFF POWER

1. Make sure that the machine is at a standstill and CYCLE START lamp is extinguished.
2. Check to see that NC ALARM and INPUT ERROR lamps are not illuminated. If illuminated, detect the causes by the indication of alarm code and eliminate them. Refer to 4.3.12 Alarm and Status Code Display.
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.

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 optimum condition for extended period.

Table 8.1.0 Inspection Schedule

Components	Inspection Items	Schedule						Remarks
		Daily	Weekly	Monthly	Every three months	With the system-on	With the system-off	
Tape reader	Cleaning of reading head.	○					○	Including light source part.
	Cleaning of tape tumble box.		○				○	
Control panel	Tight close of the doors	○					○	
	Loose fit and gap of side plates and worn door gaskets.			○			○	
Servo motor, and DC motor for spindle	Vibration and noise.	○				○		Feel by hand, and do the audible inspection.
	Motor contamination and breakage	○				○	○	Visual inspection.
	Clearance of ventilation openings.	○				○	○	Inspect mainly spindle DC motor
	Burned spots, cracks, wear, and pressure of brushes.				○		○	Check the length of brushes.
	Roughened commutator surface.				○		○	Check dark bar, threading and grooving of commutator.
	Dirt of motor interior.				○		○	Clean with compressed air
	Battery	○				○	○	See if BATTERY lamp on operator's panel lights

The detail of inspecting operation for each component is as follows.

8.1.1 TAPE READER

1. Cleaning of head of tape reader

   A. Remove tape rubbish and dust on the glass with a blower brush. If the glass is stained oil or oily dust, wipe it with a gauze or soft cloth soaked with absolute alcohol. Also clean the tape guide and the tape retainer.

   B. Remove the dust, if any, on LED (light source) with a blower brush.

2. Cleaning of tape tumble box

   A. Clean the polyester leading tape with a clean, soft cloth.

   B. Remove the tape outlet cover (See Fig. 5.1.3) by loosening two mounting screws and clean the bottom of the tape tumble box with cloth or brush.

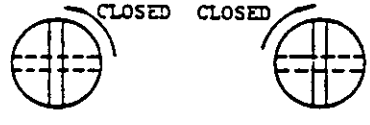
8.1.2 CONTROL PANEL

The control panel is dustproof, sheet-steel enclosure with gasketed doors.

A. Front and rear doors of the control should be always shut tightly, even if the control is not operating.

B. Where inspecting the control with the door open, after it is over, lock the door positively by turning two door locks with a large screwdriver (minus).

   Turning direction of door locks is as follows.



With the door lock on right side of the door

With the door lock on left side of the door

Fig. 8.1.2

   The vertical groove on the door lock means the door locked, and the horizontal, the door unlocked.

   NOTE: With the control equipped with the door interlocking switch as an option, opening the door shuts off the main power supply and stops all operations.

C. Check gaskets on the brims of front and rear doors.

D. See if the inside of enclosure is dusty. Clean it, if necessary.

E. Check for any opening to the door base with the doors shut tightly.

8.1.3 SERVOMOTOR AND DC MOTOR FOR SPINDLE

1. Vibration and noise.

   Vibration can be checked by hand resting on the motors, and to hear the 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 keeping the excellent performance of the control. Inspection work to be executed is described in the following three items. For detailed instructions, refer to YASNAC MAINTENANCE MANUAL.

/Three-Month Inspection of Commutators and Brushes/

The carbon dust from brushes, scattered 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 the control to a fatal damage. To avoid this, be sure to give an inspection on the commutators and blushes at least every three months.

4. Carbon brushes

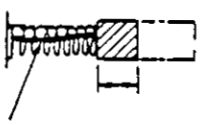
A. Under normal operating conditions, brush wears by 2 to 4 mm per 1000 operating hours. If wear is excessive, check to see if oil contaminates armature surface, or abnormal overcurrent flows motor circuit.

B. When brush length becomes shorter than those shown below, replace the brush with new one.

Cup motor· 6 mm or below

DC motor for spindle· 17 mm or below

C. If either of brush or pigtail is broken, brush assembly must be replaced as a whole unit with new one.



PIGTAIL

Fig. 8.1.3

NOTE When replacing the brush assembly, consult the company.

5. Commutator surface

A. Check visually surface roughness of the commutator throughout inspecting window

The commutator should take on a polished light brown or chocolate color after 100-200 operating hours in the process the motor ideal development of the commutator film. Such commutator needs no attention other than to be kept clean.

B. See if blackened bar, threading (or grooving) is on the commutator. If any of them is detected, investigate the cause of trouble.

Threading or grooving on the commutator surface may be due to too small motor load. Blackened bar is responsible to carbon dust in commutator slots, or incidentally aroused sparkings. If it appears that the carbon dust is a cause of blackened bar, finish the commutator with a clean dry cloth to smooth the surface. If the cause seems to be responsible for sparking, consult the company.

6. Motor inside (dirty)

A. Check visually the motor interior through inspection window.

Though the dried carbon dust is virtually not harmful to the correct motor running, clean the inner parts such as commutator, brushholders and brushes with 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, consult the company.

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 as well as the control. If possible, yearly inspection taking less than 8 hours is recommended.

8.1.4 BATTERY

Make sure that the battery lamp on operator's panel is turned off. If it is turning on, consult maintenance personnel. The battery must be replaced with new one within a month.

8.2 REPLACEMENT OF BATTERY

While power is off, the batteries are 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 going to be discharged after a long period of use, BATTERY lamp on the operator's panel lights to give warning for replacement. In such occasion, consult the company The battery must be changed with new one before a month passes Where replacing, never remove the old battery, with power off, otherwise the data stored in memory may be cleared away

Replacing Procedure

1. Depress POWER OFF pushbutton on the operator's panel.

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. If the memory (printed circuit) board is mounted behind the CPU module, loosen two CPU module set screws, and the CPU is hinged open.

3. Depress POWER ON pushbutton. Where the control is equipped with a door interlock switch, draw it out with the movable section by hand. The power can be turned on, with the door open.

4. Check if LEDs (1LED and 2LED) on memory board are lighting. Fig. 8.2.1 shows the arrangement of LEDs and the battery.

1LED:	If lighting, replace the large battery "660S" as memory power of part program with new one.
2LED:	If lighting, replace the small battry "400S" as memory power of parameter and tool offset with new one.

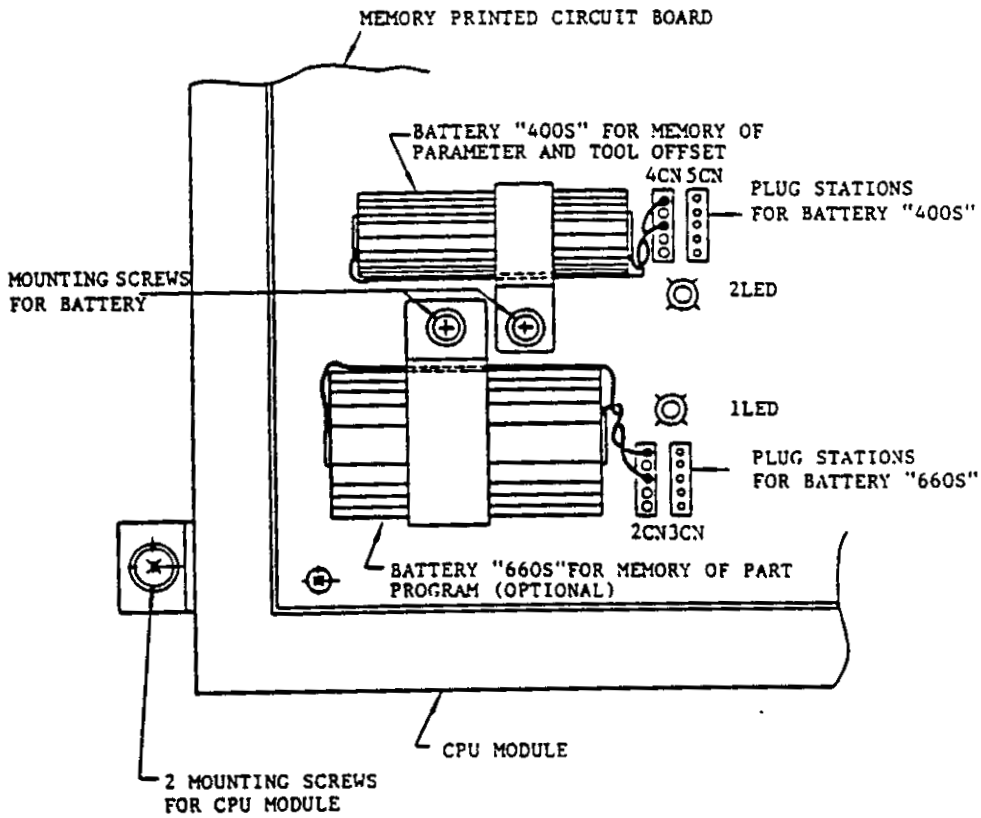


Fig. 8 2.1 Arrangement of LED and Battery

5. With the control power turned on, connect the receptacles of the new battery into the plugs (3CN or 5CN) on memory circuit board, and LED will be turned off. See Fig. 8.2.1. If LED is still lighting, it is due to the wrong insertion of battery connectors, or poor battery charge.

   IMPORTANT: Two plug stations 4CN and 5CN (or 2CN and 3CN) are connected together with common leads. Where an old battery is replaced with a new one, connect the new battery first to the plug station not occupied, and remove the receptacles of the old battery.

6. Depress POWER OFF pushbutton.

7. Remove the mounting screw of old battery, and then replace the battery with new one. In this case, pass the battery lead through gaps between the battery and the battery clamp, and use care not to contact the lead with memory circuit board.

   Where the control is provided with a door interlock switch, push it back in place with the movable section, and power cannot be turned on with the door open.

   Where CPU module is open, fasten it with two mounting screws.

8. Close tightly the front door.

9. Depress POWER ON pushbutton.

10. Make sure that BATTERY lamp on the operator's panel is turned off.

NOTES:

1. While battery is being replaced, exercise utmost care to prevent the oil mist from coming into the control, and to accomplish the work as quickly as possible.
2. Use special care so that no water drop, no oil or dust sticks to the devices (printed circuit board, connectors, cables, etc.) inside the control.
3. Never leave any screws or washers in the control. If left, take them out.

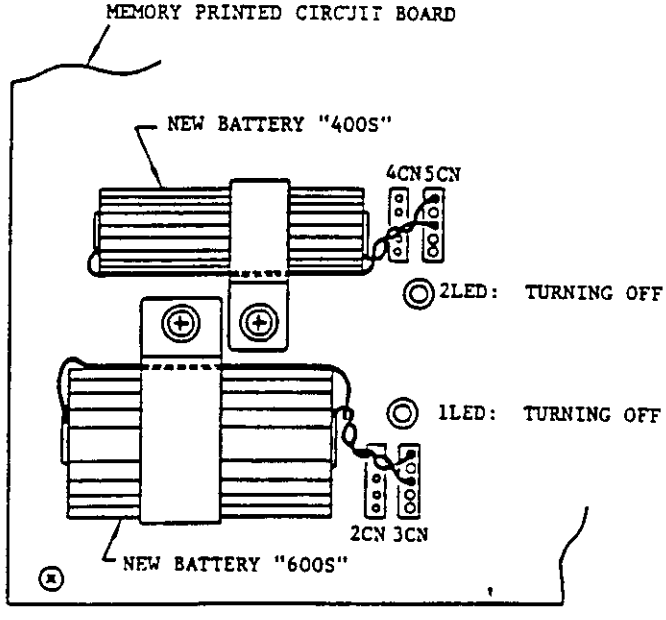


Fig. 8.2.2 Arrangement of Replaced Battery

8.3 POWER FUSES

The power fuses are provided for the safety of the control; five in the composite control power supply unit and two in the servo unit for X- and Z-axis. If they are blown off, consult maintenance personnel.

8.3.1 FUSES OF COMPOSITE CONTROL POWER SUPPLY UNIT

If any of the fuses are blown off, all power supplies are turned off. Reset the control according to the following procedure.

1. Remove the blown-off fuse that is indicated with white mark coming on the fuse casing window.
2. Inspect for the cause of the control power supply, I/O interface and servo unit, and remove, if any.
3. Replace the blown-off fuse with new one.
4. Depress POWER OFF pushbutton on the operator's panel to reset the control and then turn on the power.
5. If the fuse is blown off again, consult the company.

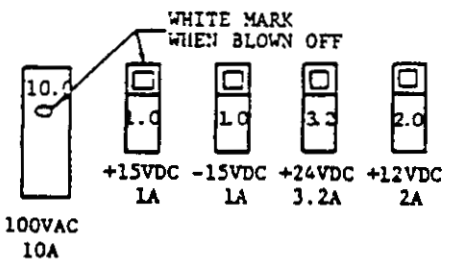


Fig. 8.3.1 Fuses of Composite Control Power Supply Unit

8.3.2 FUSES OF SERVO UNIT FOR X- AND Z-AXIS

Two plug-in fuses (1FU and 2FU) are mounted on the lower part of the each servo unit. Small capacity servo unit is equipped with only 1FU.

If any of the fuses are blown off, the system becomes overloaded (shown by alarm code "35"), and servo power supply is turned off. Reset the system using the following procedure.

1. Check for a blown-off fuse which has white mark at the enclosure window, and remove it, if found.
2. Remove the cause of overcurrent.
3. Replace the blown-off fuse with new one.
4. Depress RESET key on the operator's panel to release the alarm status.
5. Depress POWER ON pushbutton to retrieve servo power supply.
6. If any of the fuses are blown off again, consult the company.

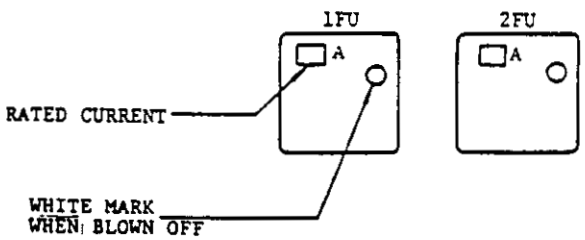


Fig. 8.3.2 Fuses of Servo Unit

8.4 OTHERS

8.4.1 MOLDED-CASE CIRCUIT BREAKERS (MCB)

With turned off the molded-case circuit breaker (1MCB) in the NC control panel, power is not supplied to the circuit even if POWER ON push-button on the operator's panel is depressed. Be sure to keep the breaker turned on except it is turned off at inspection and maintenance.

NOTE: The NC control may sometimes not be furnished with MCB. Or some types of the control allow to operate the MCB from outside the enclosure as the all machine interface is housed in the NC control enclosure. For operation instructions on these panels, refer to the machine tool builder's manual.

8.4.2 POWER RECEPTACLES FOR MAINTENANCE TOOLS

· The receptacle for powering the devices for maintenance is provided in the enclosure. It can be used for maintenance devices of ratings shown below.

100 VAC, 1 A or below

Note that the receptacle is not provided with the fuse.

· When the breaker is turned on, the receptacle becomes live.

8.5 TROUBLE CAUSES AND REMEDY

8.5.1 ON-LINE DIAGNOSTICS

Malfunction and operation status of YASNAC 2000G control system and the machine are automatically detected while the control system is on-line and machining. To display them, the control is provided with the displaying function shown below.

· Three-digit alarm code display

Alarm code shows the cause of the error, and is usually shown in two digits. Errors on the axis are displayed in three digits, and the first digit indicates the axis.

· Three-digit status code display

Status code shows the operating condition and is displayed simultaneously with alarm code by the operation for alarm code display. The first digit indicates the function code, M, S or T.

· Input/Output signal display

These displaying function can be always used even if the machine is running. For the displaying operation, refer to 4.3.12 Alarm and Status Code Display and 4.3.13 Display of Input/Output Signals.

8.5.2 LIST OF ALARM AND STATUS CODES AND REMEDIES

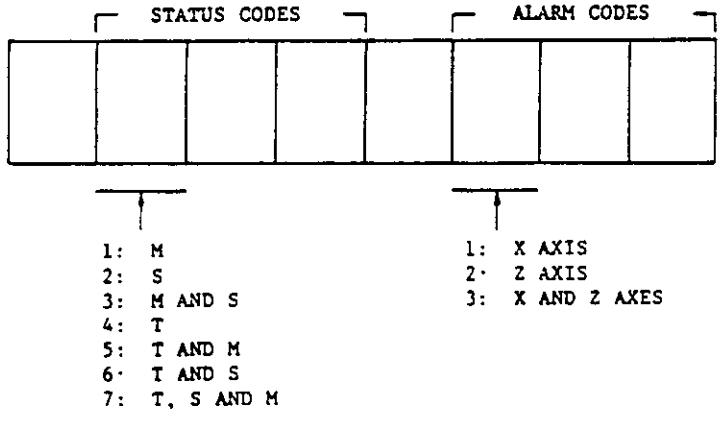


Fig. 8.5.2

Table 8.5.2 Alarm Codes and Remedies

Alarm code	Causes	What to do
11 Excessive temperature rise in the panel.	· Ambient temperature exceeding 45°C (113°F). · Failure in cooling system such as fan.	· Depress RESET button after lowering ambient temperature, and alarm status is released. · Contact maintenance personnel, if necessary.
12 Tape/Memory horizontal parity error.	· Mispunched or dusty tape. · Reading error of tape reader. · Even holes in EIA code. · Blank in EIA code. · Odd holes in ISO code. · Memory error due to the poor battery, in MEM mode.	· Depress RESET button, and alarm status is released. · Correct the tape. · Clean the head of tape reader. · Where the error occures in MEM mode or in EDT mode, inspect the control, especially, battery.
13 Tape vertical parity error.	· Odd number of significant characters in one block when TV check is on (parameter No. 81 = "1"). · Dusty tape. · Reading error of tape reader.	· Depress RESET button, and alarm status is released. · Correct the tape. · Clean the head of tape reader. · If TV check is not required, set the content of parameter No. 81 to "0."
14 Format error.	· Illegal characters are used. · Illegal G code is used.	· Depress RESET button, and alarm status is released. · Correct the tape. · Clean the head of tape reader. · Correct the program.
15 Data error.	· Feedrate F (E) is not given in cutting block. · Number of significant characters in one block exceeds 64. · In EDT mode, the input data exceeds maximum capacity of memory when storing them in the program memory. · Length of threaded portion including the chamfering width specified by G92 and G76 is shorter than chamfering width set by parameter No. 77.	Depress RESET button, and alarm status is released. · Correct the program.

Table 8.5.2 Alarm Codes and Remedies (Continued)

Alarm code	Causes	What to do
16 Offset error.	· Tool offset values are destroyed due to poor battery.	· Make sure that BATTERY lamp lights or not. If lighting, contact maintenance personnel.
17 Parameter error.	· Contents of parameter are destroyed due to poor battery.	· If BATTERY lamp is lighting, contact maintenance personnel.
18 Tape memory error.	Disagreement between tape and memory at NC tape collating. · Memory capacity over.	· Depress RESET button, and alarm status is released.
21 Overtravel.	· The machine is at the end of their stroke.	· Move it back in stroke by manual operation and depress RESET button.
22 Zero Return area error.	· Zero Return action is commanded in the area where decelerating switch is effective. · The third digit of ALARM CODE indicates the axis.	· Depress RESET button, and alarm status is released. · Put the machine out of the area by manual operation.
23 Zero Return unready.	· CYCLE START pushbutton is depressed before performing Zero Return with the control interlocked for Zero Return. (parameter No. 66 = "1")	· Depress RESET button, and alarm status is released. · Perform Zero Return. · If interlocking is unnecessary, set the contents of parameter No. 66 to "0."
24 Zero Return position error.	· Disagreement between fixed original point and programmed point at G27. · In manual Zero Return or G28 for automatic Zero Return, positioning error.	· Depress RESET button, and alarm status is released. Check the program. · For the error in automatic operation, contact maintenance personnel
25 Sequence error	· The program which is prohibited by the machine interface is executed. For example, when the spindle is running, the reverse running of the spindle is commanded.	· Depress RESET button, and alarm status is released. · Correct the program.

Table 8.5.2 Alarm Codes and Remedies (Continued)

Alarm code	Causes	What to do
26 Spindle error.	· Spindle drive is affected due to failure in fuse or thermal relay, and ambient temperature rise.	· After eliminating the cause, depress RESET button. · Contact maintenance personnel.
27 Positioning error.	· Failure in mechanical or servo system.	· Check their systems. Then depress RESET button. · Contact maintenance personnel.
28 Machine unready.	· MRD (machine ready) signal is not received after power is turned on. · MRD signal is temporarily eliminated on the way because of abnormal mechanical sequence.	· If MRD signal come when turning on power, the alarm is soon released automatically. · On the other causes besides above, check the machine and machine sequence · If temporarily eliminated signal, reset operation is available. In addition, contact maintenance personnel.
31 Servo power unsupplied.	· Servo power is not supplied.	Depress the POWER ON button according to procedure turning on servo power, and the alarm is released automatically.
32 Control unit unready.	· For diagnostic of control system, the control is not yet ready for turning on servo power supply. · Servo system is not sufficiently adjusted.	When the control is ready, alarm code turns to "31" automatically. · Contact maintenance personnel if alarm code "32" remains on the display for several seconds and above.
33 Emergency stop.	· Emergency stop button is depressed.	· Reset the system, and alarm code turns to "31." · Depress POWER ON button for turning on servo power supply. · Servo power can be turned on before reset operation. But the alarm code remains on the display until performing reset operation.

Table 8.5.2 Alarm Codes and Remedies (Continued)

Alarm code	Causes	What to do
34 Servo error.	· Excessive cutting command. · Excessive servo lag. · Defective servo system and mechanism.	· Correct the program. · Reset the system, and alarm code turns to "31" automatically. Servo power can be turned on. · Check servo system and mechanism.
35 Overload.	· Excessive cutting command. · Thermal overload relay tripped due to motor overloaded. · Blown out fuses. · Temperature in servo control unit rises excessively. · Defective servo system or mechanism. The reset operation is executed during rapid traverse operation.	· Check the thermal overload relay and fuses in drive unit. · After cooling, reset the relay or replace the fuse with new one. · Eliminate the cause and then reset the system, and alarm code turns to "31." Servo power supply can be turned on. · Correct the program · Inspect servo system and mechanism.
36 Feedback error.	· Poor feedback encoder for X- or Z-axis. · Cable for encoder is disconnected.	· Contact maintenance personnel.
37 Hardware error. (FG error)	· Failure in package of type GCP01 or GFG01, printed circuit board in the control.	· Contact maintenance personnel.
38 Hardware error. (RPG error)	· Failure in package of type GCP01, printed circuit board in the control.	· Contact maintenance personnel
81 CPU error.	· Impossible to perform the operation due to CPU trouble.	· Make the memorandum of contents indicated on UNIVERSAL DISPLAY, and contact maintenance personnel.
82 Memory collating error.	· System program stored in the memory is destroyed.	· Contact maintenance personnel. · The first and second digits show the number of poor ROM.

Table 8.5.2 Alarm Codes and Remedies (Continued)

Alarm code	Causes	What to do
91 Contents disagreement of tape and memory. (For off-line only.)	· Contents of system memory and source tape are not equal. SYSTEM NO. "2": CHECK	· Replace or repair ROM or the package. · The first and second digits show the contents of memory. · The numbers displayed in NUMBER mean the tape contents.
92 Tape reading error. (For off-line only.)	· Source tape is misread. SYSTEM NO. "2": CHECK SYSTEM NO. "3": LOAD	· Check the tape. · Clean the head of tape reader.

8.5.3 LIST OF INPUT/OUTPUT SIGNALS

UNIVERSAL DISPLAY

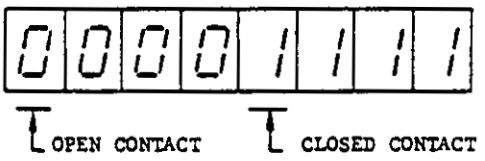


Fig. 8.5.3 Status Display of Input/Output Signals

INPUT/OUTPUT SIGNAL
Shows input/output signal name and its displaying position, and status of signal is displayed with a numeral 1 or 0.

ADDRESS:
Shows the distinction of input and output signal X is input signal and Z is output signal.

NUMBER
Shows diagnostic number which is specified by OFFSET and PARAMETER No. switches.

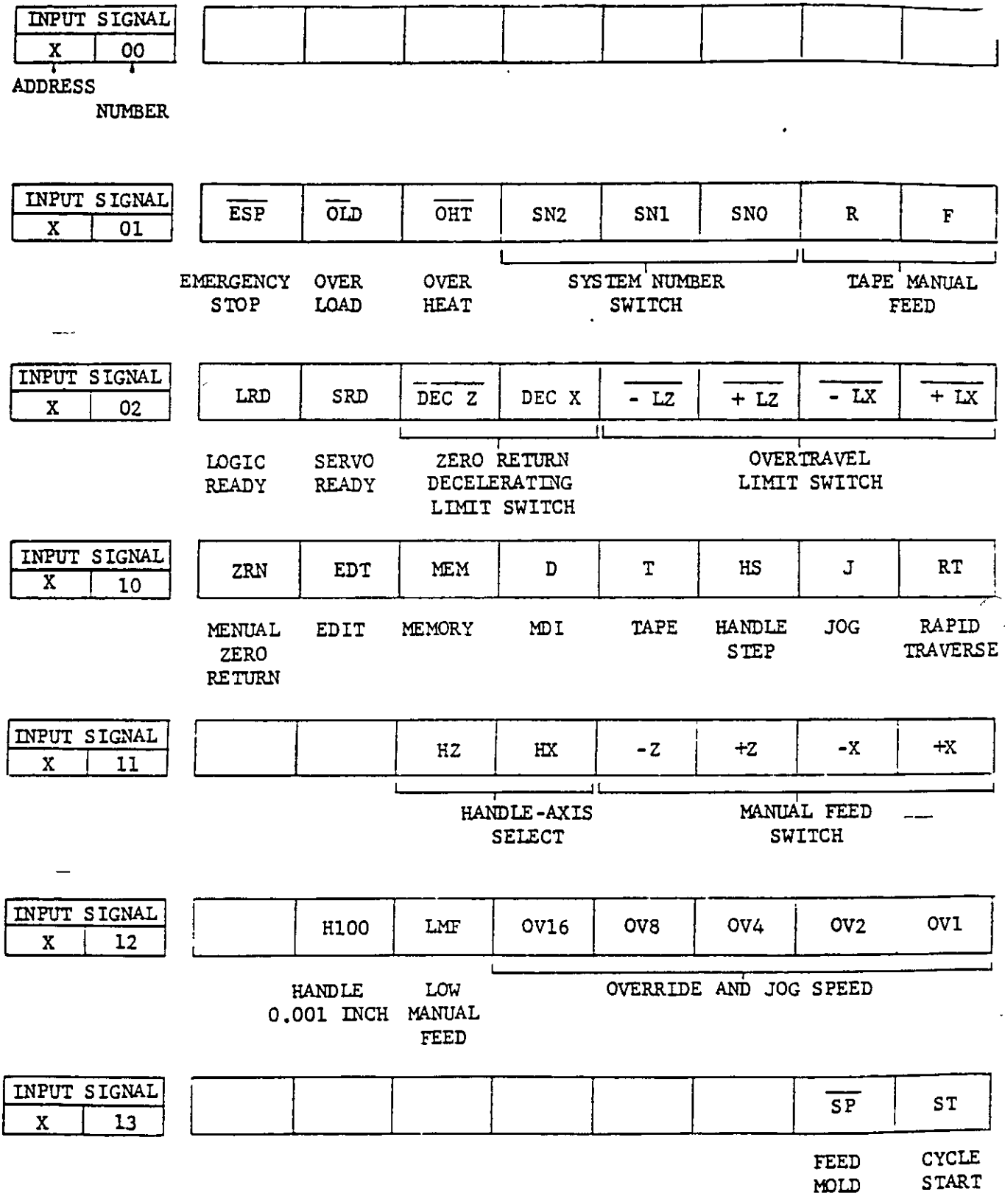


Fig. 8.5.3 List of Input/Output Signals

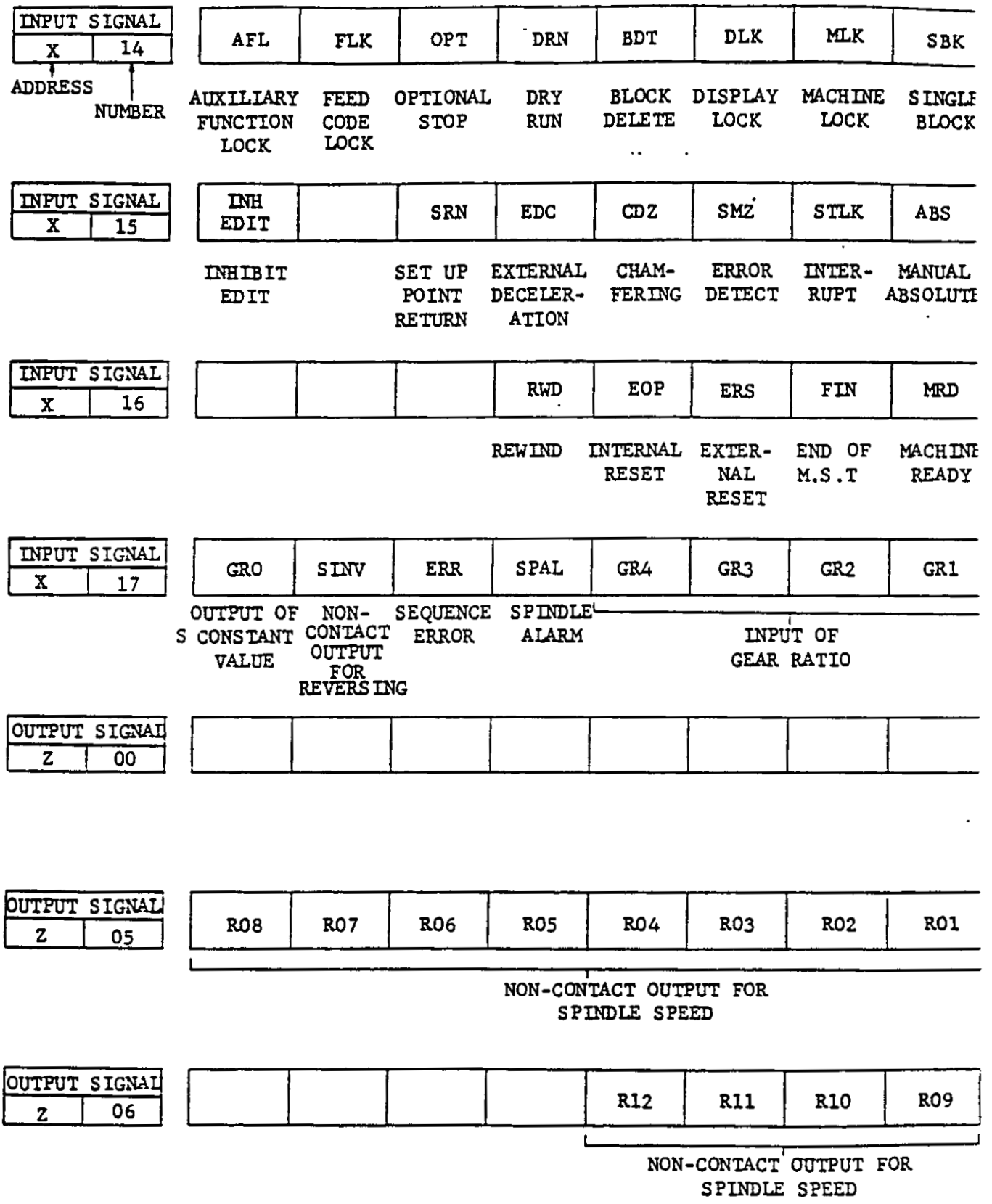


Fig. 8.5.3 List of Input/Output Signals (Cont'd)

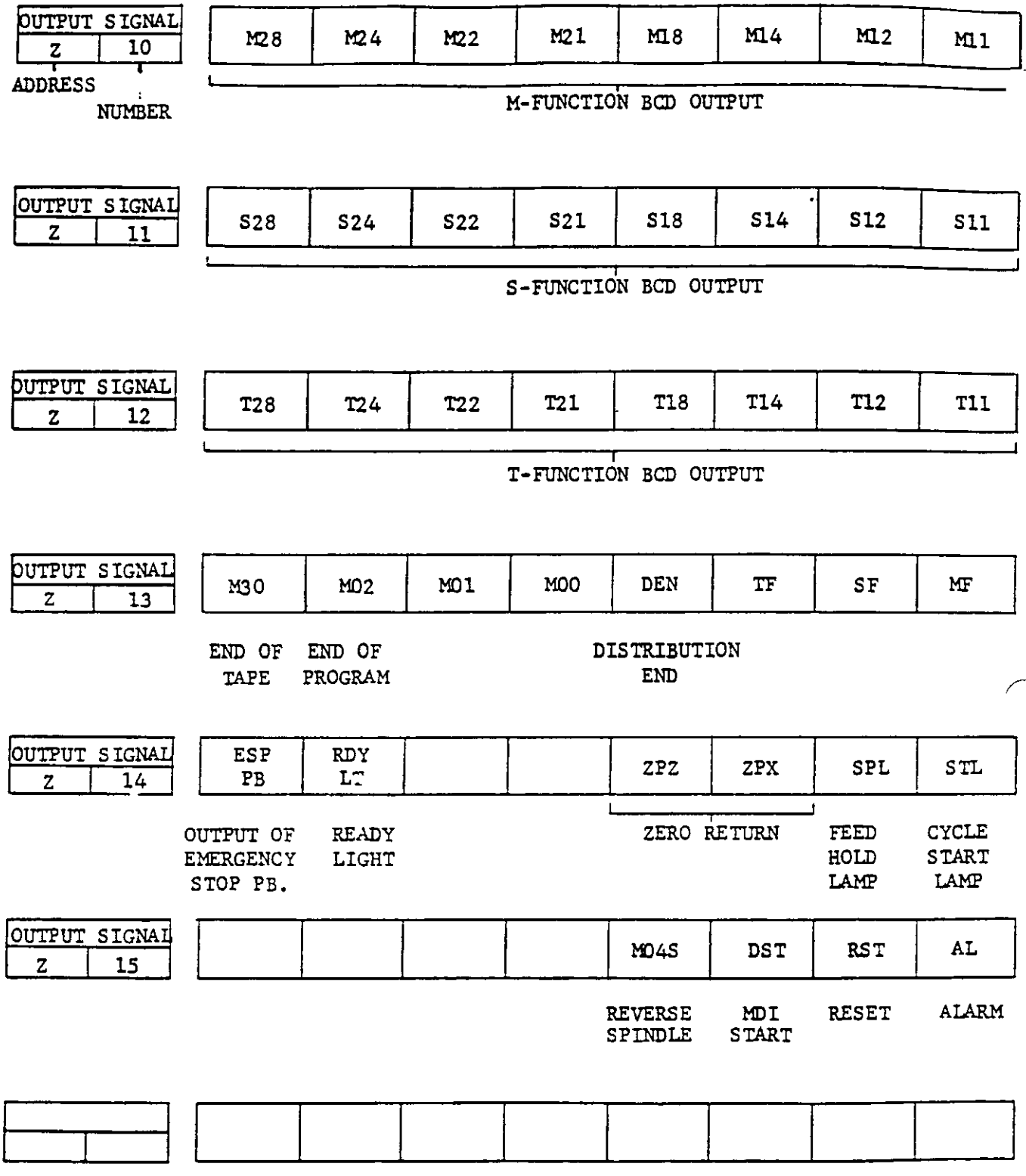


Fig. 8.5.3 List of Input/Output Signals (Cont'd)

8.5.4 BEFORE MAINTENANCE CALL

If the cause of trouble cannot be found by using alarm codes or I/O signals (described in 8.5.1 to 8.5.3), or correct action fot the trouble cannot be taken, record the following items, and notify the company as immediately as possible.

- Alarm code and status code or unusual indication on the operator's panel.
- The state and phenomenon of the trouble.
- The operational procedures just before the trouble have occured and number of applied tape.
- Whether the trouble occurs again or not in each time the operation is repeated after depressing the RESET key.
- Date and time when the trouble occured.
- Name of the discover of the trouble and the operator.

While you are contacting the company, maintain the control in the condition when the trouble has occured as same as possible. Avoid turning off control power by depressing POWER OFF button, if the situation allows.

APPENDIX-1

INTERFACE FOR M-, S-, AND T-CODE

1. M CODE

YASNAC 2000G equipped with a standard interface sends M-BCD code and M- decode output signals to the machine. In addition, DEN output and FIN input signals for the common signal of M-, S-, and T- code are also fed.

- DEN: Completion of tool move
- FIN Completion of M-, S-, and T-function

NOTE. Where the control is equipped with a machine interface", the above signals are processed by the sequence control section of the interface, and then are transmitted to the machine. Signals for S- and T- code are also processed by the same way.

- M-BCD Output

a. When the block containing M code is executed, M-BCD output signal is sent to the the machine But the command to read the code, MF is sent after "t" msec. This delay time, "t" msec is set by parameter No. 98.

b. Sending of M-BCD signal is stopped when FIN signal is fed back to the control. The reset or mode select operation also stops sending of M-BCD signal.

c. To execute M-function after completion of a move command in the same block, use DEN and MF signal as the command to read the code.

- M-Decoded Output (M00, M01, M02 and M30)

a. When a move command and M command are given in the same block, M-decoded signal is fed to the machine after the completion of tool move.

b. If a move command is not given in the block, M-decoded signal and M-BCD signal are fed together.

c. Sending of M-decoded signal is stopped by the Cycle Start or reset operation. But feedback of FIN signal does not stop M-decoded signal.

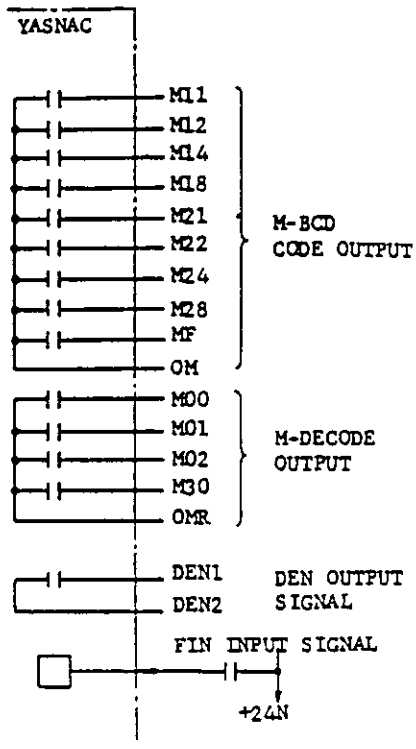


Fig. 1.1

- DEN Output Signal for Completion of Move

a When a move command and M-, S-, or T-code are given in the same block, DEN signal is fed to the machine after completion of move command. If FIN signal is fed back to the control during a movement of tool, DEN signal is not sent out

b. DEN signal is fed to the machine together with BCD code when M-, S-, or T- code is commanded without move command.

c. Sending of DEN signal is stopped by the feedback of FIN signal, the reset operation or the mode select operation.

- Precaution

Since M-cdoes shown below are processed by the control, these BCD and decoded output are not sent to the machine.

M23/M24, M51/M52, M94/M95, M98/M99

Example of M00

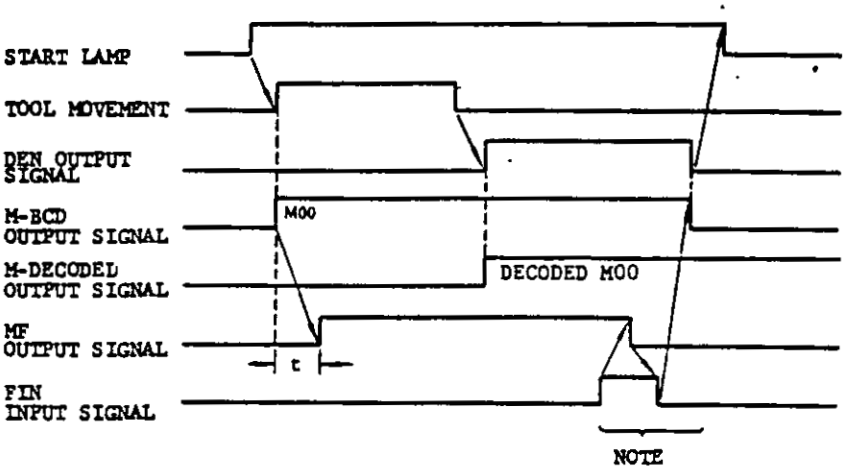


Fig. 1.2

NOTE: For adjustment of the timing of FIN signal, stop FIN signal when MF signal is stopped at the rising portion of FIN input signal. M-BCD and DEN signals are reset and the block is completed at the falling portion of FIN input signal.

Example of M30 (Where including the reset or rewinding operation)

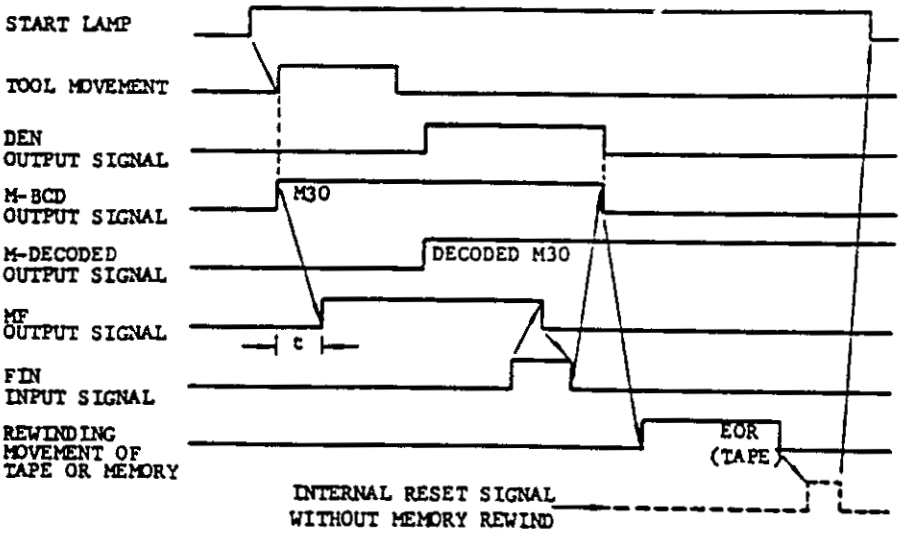


Fig. 1.3

It is determined in accordance with the following condition whether reset or rewind is executed or not by M02 or M30 command.

At the rising portion of FIN input signal,

The input signal EOP is turned on:
The internal reset is executed without a memory rewind, with LABEL SKIP lamp turned off.

The input signal RWD is turned on:
With MODE SELECT switch set to TAPE or MEM mode, the memory rewind is executed and LABEL SKIP lamp lights. In TAPE mode, the tape rewind is executed.

Fig. 1.4 shows an example of sequence control which uses M02 and M30.

M02· For resetting

M30· For resetting and rewinding

By means of the delay timer, EOP and RWD input signals should be set to ON or OFF at least 100 msec before the control receives FIN signal.

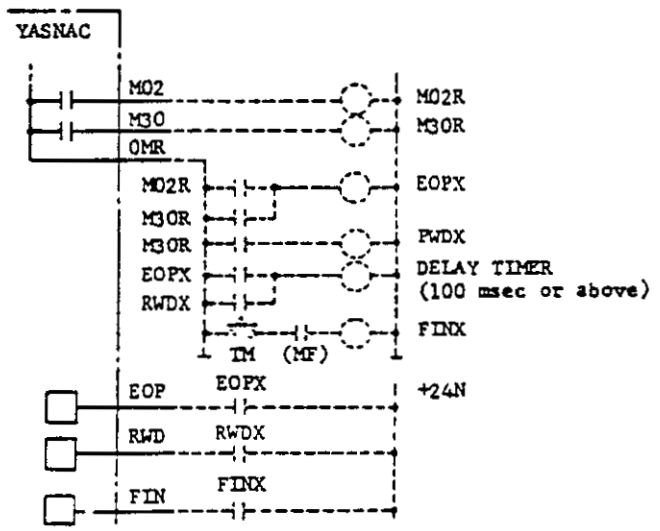


Fig. 1.4

2. S CODE

· S2-Digit Command

S2-digit BCD output signal is sent to the machine at the equal timing as that of M-BCD output. Sending of the BCD signal continues until a new S2-digit command is given, even if the control receives FIN signal.

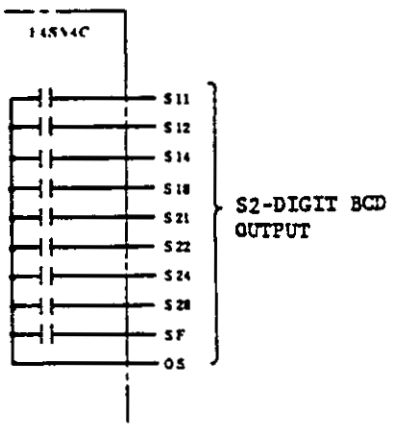


Fig. 1.5

· S4-Digit Command

Where S4-digit is used, 12 bits binary non-contact output or spindle D/A output signal is sent to the machine.

a. 12 Bits binary non-contact output (Sn)

Where Constant Surface Speed Control option is not equipped or G97 (cancel) is commanded,

$$S_n = \frac{4095}{S_M} \times S$$

S_M: Maximum spindle speed (rpm) of each gear

S: Spindle speed (rpm) by S command

NOTE: 11 Bits binary output with ± sign is also available.

$$Sn = \pm\frac{2047}{S_M} \times S$$

When input signal SINV is turned on, Sn becomes the minus value.

When the block containing S4-digit command is executed, 12 bits binary output signal is fed to the machine. In this case, FIN signal need not be fed back to the control, since sending of the signal is not stopped by FIN signal or reset operation. Sending of the signal continues until a new S code is commanded. .

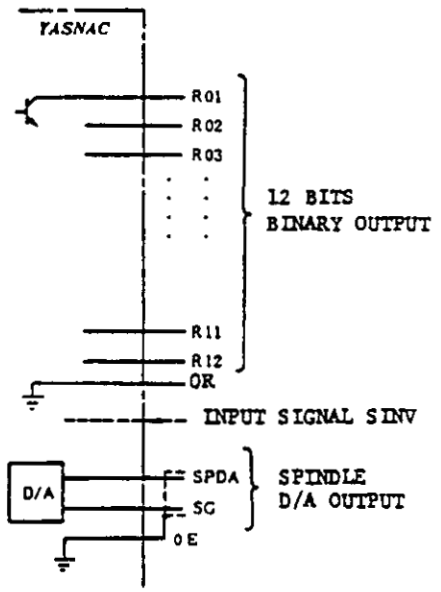


Fig. 1.6

b. Spindle D/A output (Vs)

Where Constant Surface Speed Control option is not equipped or G97 (cancel) is commanded.

$$Vs(\text{voltage}) = \pm\frac{10}{S_M} \times 6$$

With the input signal SINV turned on, Vs is the minus voltage.

When the block containing S4-digit command is operated, analog output is sent to the machine. D/A signal continues until a new S command is given, even if FIN signal is fed back, or reset operation is made. Therefore, FIN signal need not be fed back to the control.

3. T CODE

T1-digit BCD output signal for T2-digit command or T2-digit BCD output signal for T4-digit command is fed for tool selection with the equal timing as that of BCD output for M code. This BCD signal continues until a new T command is given, even if FIN signal is fed back to the control.

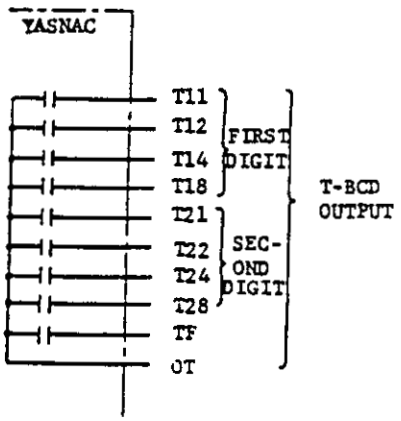


Fig. 1.7

Table 1.1 List of M-, S-, and T-code Output

Output / Signal	DEN	M-decoded	M-BCD	S-BCD	S12 Bits or D/A Output	T-BCD
Command	M-, S-, and T-code	M00, M01 M02, M30	M code Excluding internally processed code	S2-digit command code	S4-digit command code	T-code for selecting the tool
Start Timing of Signal	After executing the tool move command in the same block.		At the start of the execution of commanded block.			
By FIN Signal	Reset	Held	Reset	Held	Held	Held
By RESET Operation	Reset	Reset	Reset	Reset	Held	Reset
By MODE SELECT Operation	Reset	Held	Reset	Held	Held	Held
By CYCLE START Operation		Reset				

APPENDIX-2

INTERFACE FOR CONSTANT SURFACE SPEED CONTROL†

1. INTERFACE FOR CHANGING SPINDLE GEAR RATIO

Example of 2-stage gear change

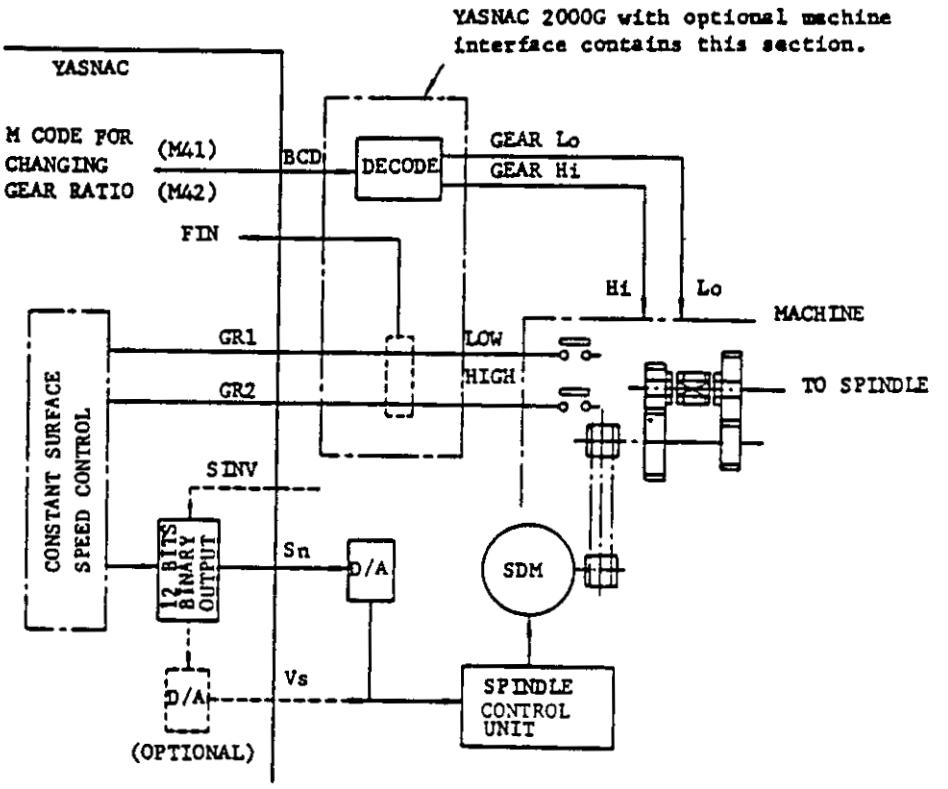


Fig. 2.1

- For M-BCD signal to be used for changing the gear ratio, refer to the machine tool builder's manual. The answer for the M-BCD signal, FIN should be fed back to the control after completion of gear ratio change.
- The gear ratio signal, GR1 (Low) or GR2 (High) must be held until it is changed the other ratio.

· For Constant Surface Speed Control, output Sn (or Vs) of spindle drive unit is calculated as follows.

a. Where using 12 bits binary non-contact output,

$$Sn = \frac{4095}{S_M} \times \frac{S}{\pi L X}$$

S_M Maximum spindle speed (rpm) for each gear

When using GR1, S_M is specified by the content of parameter No. 86.

When using GR2, S_M is specified by the content of parameter No. 87.

S Surface speed by S command (M/min or Feet/min)

X. Current value of X coordinate

NOTE 11 Bits binary output with ± sign is also available.

$$Sn = \pm \frac{2047}{S_M} \quad \frac{S'}{\pi L X}$$

When input signal SINV is turned on, Sn becomes the minus value

b Where using spindle D/A output,

$$Vs \text{ (voltage)} = \pm \frac{10}{S_M} \quad \frac{S}{\pi L X}$$

With input signal SINV turned on, VS becomes the minus value. Therefore, while M04 command for reversing the spindle is given, SINV input should be turned on.

- Maximum number of gear stage to be used is 4 (GR1 to GR4).

2. CONSTANT SURFACE SPEED CONTROL WITHOUT GEAR RATIO CHANGE

· The interface for changing the gear ratio shown below is not required.

* M code processing interface (including the feedback of FIN signal)
* Interface of gear ratio signal (GR1 and GR2).

· The output for spindle control unit, Sn is obtained by the above equation. But, as to S_M, the maximum spindle speed must be set by the parameter No. 86.

APPENDIX-1 LIST OF PARAMETERS

YASNAC 2000G パラメーター一覧 (1/8)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 INITIAL VALUE
00		稼動時間表示 Operation Time Display ○○○○. ○○. ○○ 時(H) 分(M) 秒(S)	
01		早送り速度レンジ Rapid Traverse Rate Range '0' = Hi, '1' = Lo	0
02		ドライラン時早送り速度 Rapid Traverse Rate for Dry Run '0' = Hi/Lo, '1' = JOG	1
03		ストロークチェック入切 Stroke Check ON-OFF '0' = OFF, '1' = ON	0
04		マシンロック時, ストロークチェック入切 Stroke Check ON-OFF for Macine Lock '0' = ON, '1' = OFF	0
05		周速制御における工具補正量キャンセル Tool Offset Cancel for Surface speed Control '0' = OFF, '1' = ON	0
06		未使用 Not Used	0
07		未使用 Not Used	0
08		未使用 Not Used	0
09		未使用 Not Used	0

Notes: 1. Parameter numbers 01 to 09 can be changed without operating of SYSTEM No. switch.
2. Parameter number 00 is only for display.
3. Parameter numbers with mark ◤ are optional.

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 INITIAL VALUE
◤30	X	固有原点からのストロークチェック座標(+) Stroke Check Point (+) from Zero Point "1" = 0.001 mm	0
	Z		0
	I	未使用 Not Used	0
	K		0
◤31	X	固有原点からのストロークチェック座標 (−) Stroke Check Point (−) from Zero Point "−1" = −0.001 mm	0
	Z		0
	I	未使用 Not Used	0
	K		0
32	X	未使用 Not Used	0
	Z		0
	I	未使用 Not Used	0
	K		0
33	X	未使用 Not Used	0
	Z		0
	I	未使用 Not Used	0
	K		0
34	X	未使用 Not Used	0
	Z		0
	I	未使用 Not Used	0
	K		0

Note: Parameter numbers with mark ◤ are optional.

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 INITIAL VALUE
40			
41			
42			
43			
44			
45			
46			
47			
48		未使用 Not Used	0
49		未使用 Not Used	0

Notes: 1. Parameter numbers 40 to 47 are for optional machine interface.
"0": OFF "1": ON
2. Parameter numbers with mark ◤ are optional.

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 INITIAL VALUE	
◤ 50				
◤ 51				
◤ 52				
◤ 53				
◤ 54				
◤ 55				
◤ 56				
◤ 57				
58		未使用 Not Used	0	
59		未使用 Not Used	0	

Notes: 1. Parameter numbers 50 to 57 are for timer constant for optional machine interface. "1": 16/80 msec (Hi/Lo)
2. Parameter numbers with mark ◤ are optional.

YASNAC 2000G パラメーター一覧 (5/8)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 INITIAL VALUE	
60	X	バックラッシュ補正開始方向 Backlash Comp. Starting Direction '0' = 正から (+)　'1' = 負から (−)	0	
	Z		0	
61	X	原点復帰方向 Zero Return Direction '0' = 正方向 to (+)　'1' = 負方向 to (−)	0	
	Z		0	
62	X	原点復帰最終速度 Zero Return Final Speed '1' = 7.5 mm/min	10	
	Z		20	
63	X	ポジションエラー領域 Position Error Zone '1' = 最小移動単位 Least Command Increment	32	
	Z		32	
64	X	サーボエラー領域 Servo Error Zone '1' = 最小移動単位 Least Command Increment	8192	
	Z		8192	
65	X	未使用 Not Used	0	
	Z		0	
66	X	サイクルスタート前の原点復帰 Zero Return before Cycle Start '0' = 不要 Not Required, '1' = 要 Required	0	
	Z		0	
67	X	手動ハンドル送り最大速度 Max. Feedrate for Handle '1' = 7.5 m/min	400	
	Z		800	
68		切削送り時定数 Time Constant for Feed '1' = 32 msec	6	
69		切削送り速度バイアス Velocity Bias for Feed '1' = 2 Kpps	10	

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 INITIAL VALUE
70		直径／半径指定 Diameter / Radius Designation '0' = 直径, '1' = 半径 Diameter Radius	0
71		G99/G98 イニシャルセット Initial Set '0' = G99, '1' = G98	0
72		G50 表示プリセット入切 Display - Preset ON - OFF '0' = OFF, '1' = ON	0
73		GR0 主軸DA出力値 Spindle DA Output '2047' = 10 V	0
74		G74 引戻し量（一定量） Retracting Value '1' = 0.001 mm or 0.0001 inch	0
75		G75 引戻し量（一定量） Retracting Value '1' = 0.001 mm or 0.0001 inch	0
76		G76 仕上げ代（一定量a） Finishing Value (a) '1' = 0.001 mm or 0.0001 inch	0
77		G92, G76 チャンファ幅（γ） Chamfering Width '1' = 0.1 リード Lead	8
78		GR3 主軸最大回転数 Spindle Max. RPM '1' = 1 RPM	0
79		GR4 主軸最大回転数 Spindle Max. RPM '1' = 1 RPM	0

Note: Parameter numbers with mark ◤ are optional.

YASNAC 2000G パラメーター一覧 (7/8)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 INITIAL VALUE
80		EIA/ISOの自動判別入切 - Auto Select '0' = OFF, '1' = ON	1
81		TVチェック入切 TV Check ON-OFF '0' = OFF, '1' = ON	0
82		EIA/ISOコード指定 Code Disignation '0' = EIA, '1' = ISO	0
83		MM/INCH指定 Designation '0' = MM, '1' = INCH	0
84		タッチブザー入切 Touch Buzzer ON-OFF '0' = OFF, '1' = ON	1
85		DNC結合入切 DNC Interface ON-OFF '0' = OFF, '1' = ON	0
86		GR1 主軸最大回転数 Spindle Max. RPM '1' = 1RPM	0
87		GR2 主軸最大回転数 Spindle Max. RPM '1' = 1RPM	0
88		入力指令10倍入切 Command Data×10 ON-OFF '0' = OFF, '1' = ON	0
89		G00 刃先R補正入切 Tip Nose R Comp. ON-OFF '0' = OFF, '1' = ON	1

Note: Parameter numbers with mark ◤ are optional.

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 INITIAL VALUE
90	X	バックラッシュ補正量 Backlash Value "1" = 最小移動単位 Least Command Increment	0
	Z		0
91	X	未使用 Not Used	0
	Z		0
92	X	未使用 Not Used	0
	Z		0
93	X	早送り速度 (Lo) Rapid Traverse Rate (Lo) "1" = 7.5 mm / min	60
	Z		120
94	X	早送り速度 (Hi) Rapid Traverse Rate (Hi) "1" = 7.5 mm / min	600
	Z		1200
95	X	早送り加減速定数 Accel. / Deccl. Time Const. for RT "1" = 125/8 mm / sec²	20
	Z		40
96	X	原点復帰クリープ速度 Zero Return Approaching Speed "1" = 7.5 mm / min	20
	Z		40
97	X	原点復帰最終距離 Zero Return Final Stroke "1" = 0.001 mm	500
	Z		500
98		MF, SF, TF送出し遅れ時間 Delay Time for MF, SF, TF "1" = 0.001 sec	200
99		メモリポインタ表示 Memory Pointer Display "1" = 1 ch	

Note: Parameter number 99 is only for display.

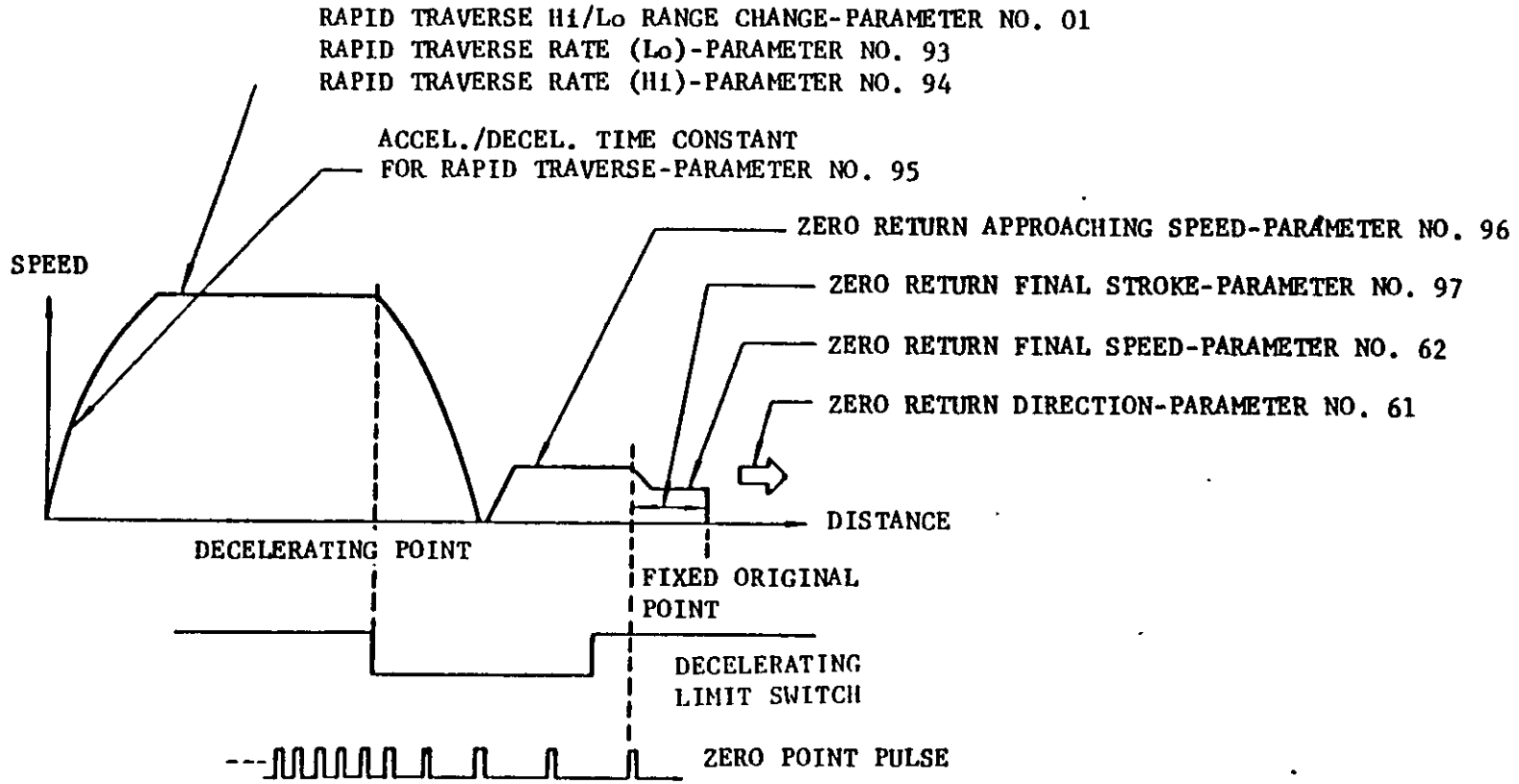

RAPID TRAVERSE Hi/Lo RANGE CHANGE-PARAMETER NO. 01
RAPID TRAVERSE RATE (Lo)-PARAMETER NO. 93
RAPID TRAVERSE RATE (Hi)-PARAMETER NO. 94
ACCEL./DECEL. TIME CONSTANT
FOR RAPID TRAVERSE-PARAMETER NO. 95
ZERO RETURN APPROACHING SPEED-PARAMETER NO. 96
ZERO RETURN FINAL STROKE-PARAMETER NO. 97
ZERO RETURN FINAL SPEED-PARAMETER NO. 62
ZERO RETURN DIRECTION-PARAMETER NO. 61
SPEED
DISTANCE
DECELERATING POINT
FIXED ORIGINAL POINT
DECELERATING LIMIT SWITCH
ZERO POINT PULSE

Fig. 3.1