MORI-YASNAC 2000B

DE6425439

Sept 25 '78 E. CHIBAI

CONTENTS

1. INTRODUCTION

The YASNAC 2000B is a high-precision NC control designed exclusively for turret lathes.

The logic section consisting of a micro-processor and nondestructive memory is highly flexible and reliable with sophisticated specifications.

Substantial consideration has been given to ease of operation and maintenance.

The YASNAC series achieves smooth feeding and high-precision machining because of the transistorized servo unit driving the DC servo motor.

This manual describes the programming of the YASNAC 2000B and NC machining operations.

2. SPECIFICATIONS

2.1 Basic Specifications

Item No.	Item	Specifications
0	Model	YASNAC 2000B
1	Use	For turret lathe
2	Dimensions/weight	650(W) x 1,400(H) x 700(D)mm/300 kg
3	Structure	Enclosed, dustproof type
4	Ambient temperature	0°-45° C (32-113°F)
5	Power supply	AC 200/220 V +10%/-15%, 50/60Hz ± 1 Hz. 3-phase, 5KVA (with built-in transformer: 100 V AC, 1 kVA)
6	Servo control unit	Transistor type X-axis: CPCR-MR05N Z-axis: CPCR-MR15N
7	Servo motor	DC servo motor with tacho-generator and pulse encoder X-axis: UGCMEM-04DB/SX Z-axis: UGCMEM-15DB/SX
8	Spindle position coder	PC-1024Z-MFT 1,024 pulses/rev: A and B phases 1 pulse/rev: A phase
9	Controlled axes	2 axes (X and Z)
10	Simultaneously controllable axis	2 axes/1 axis by manual operation
11	Least command increment	X-axis: 0.001 mm (radius) Z-axis: 0.002 mm
12	Least input increment	0.01 mm (X-axis: diameter)
13	Maximum programmable dimension	±8,388.606 mm (±330.260 in.)

Item No.	Item	Specifications
14	Tape reader	Photoelectric type, 200 char/sec paper tape: up to 25 m
15	Command tape	8-channel, black paper tape EIA RS277
16	Tape code	EIA RS244-A
17	Programming method	Absolute and/or incremental
18	Tape format	N3G2X(U)±4.2 Z(W) ±4.2 I±4.2 K±4.2 F2.2(E2.4) M2 S2 T±(2+2) P3 Q3 D4.2 *
19	Positioning	G00 code
20	Rapid traverse rate	X-axis: max. 4,800 mm/min Z-axis: max. 9,600 mm/min
21	Acceleration/ deceleration	Linear for rapid traverse
22	Interpolation	Linear: G01 Circular: G02(CCW), G03(CW) Multiquadrant circular command possible
23	Feedrate range	F.01-40.95 mm/rev
24	Maximum feedrate	4,095mm/min (68.25 kpps)
25	Thread cutting	G32 code F.01-40.95 mm lead E.0001-40.95 mm precision lead maximum thread cutting speed: 4,095 mm/min
26	Feedrate override	0-200%, in steps of 10%
27	Manual feed	1 axis simultaneously step JOG (Hi-Lo) rapid
28	Miscellaneous function	Controlled by incorporated mechanical sequence

Item No.	Item	Specifications
29	Spindle-speed function	Specified the surface speed (m/min) directly in an S code
30	Tool function	T(2 digits + 2 digits) Tool offset No. Tool No. BCD code output
31	Tool offset	X-axis: ± 999.998 mm 16 pairs Z-axis: ± 999.998 mm 16 pairs in internal memory
32	Backlash compensation	0-127 pulses independently for X- and Z-axes (1 pulse = Least command increment)
33	Buffer storage	Looks 1 block ahead
34	Dwell	G04 U2.2
35	Absolute coordinate preset	G50
36	Universal display & MDI	. Displays current position and Command data . Inputs and displays manual data . Inputs and displays tool offset-value
37	Parameter storage	Rapid traverse rate, acceleration and deceleration constant backlash compensation value
38		P
39	Single block	Provided (switch input)
40		
41	Optional block skip	Provided (/ code)
42	Machine lock	Provided (switch input)
43	Fixed cycle	G90 straight/taper cutting cycle G92 straight/taper thread cutting cycle G94 straight/taper facing cycle
44	Overtravel	Provided (input for LS)

Item No.	Item	Specifications
45	Manual absolute	Fixed at off
46	Error detection	G00: fixed at on G01-G03: fixed at off
47	Label skip	provided
48	Multiple Repetitive Cycles	provided (G70 TO G76)
49	Constant surface Speed control	provided
50	Inch/Metric Conversion	provided
51	ISO/EIA Conversion	provided
52	Procedure of punch function	provided
53		
54	Diagnostic function (I)	On-line diagnosis by displaying 2-digit error code
55	Diagnostic function (II)	Off-line diagnosis to check system/memory/tape reader/interface

2.3 View

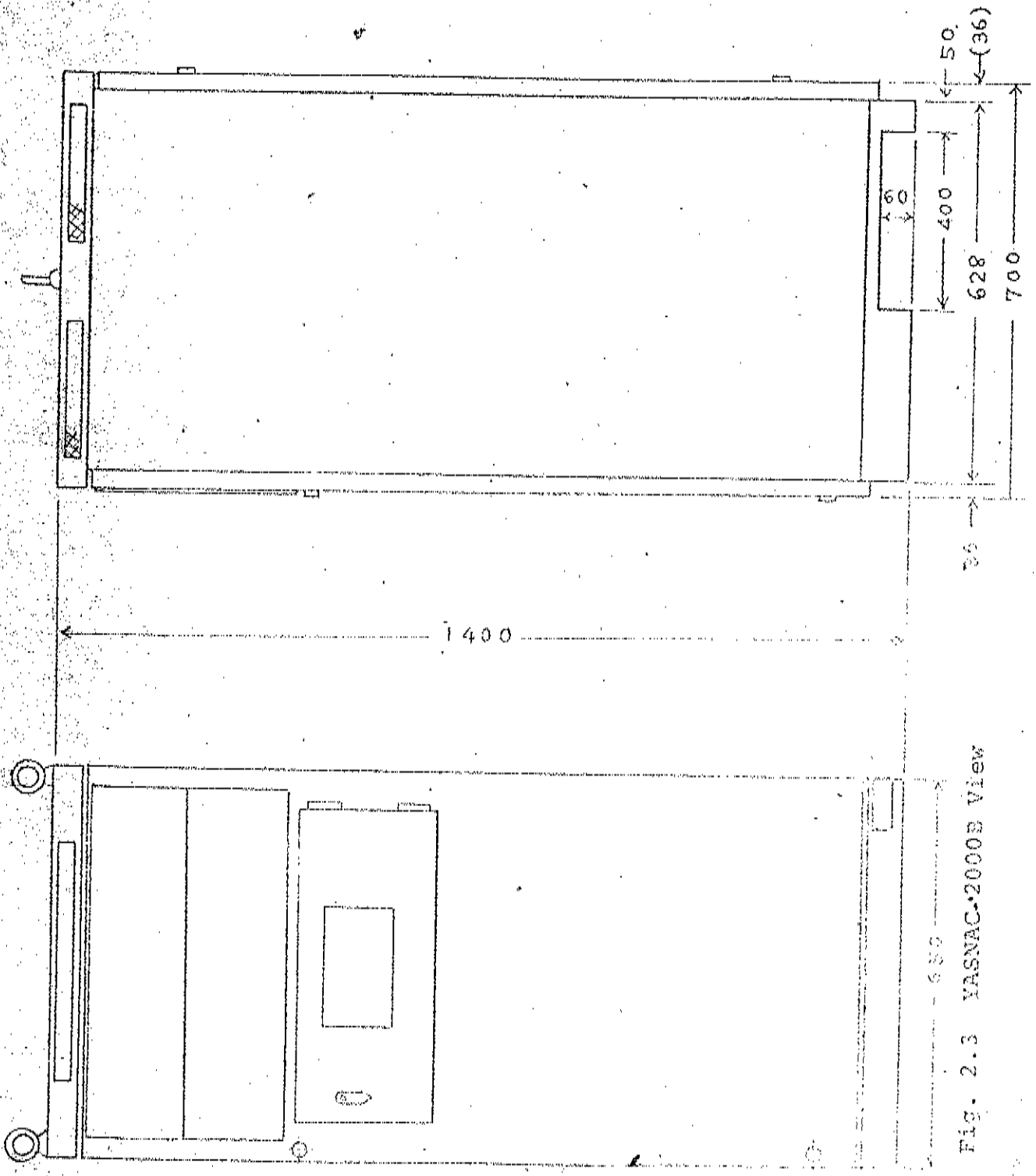


Fig. 2.3 YASNAC-2000S View

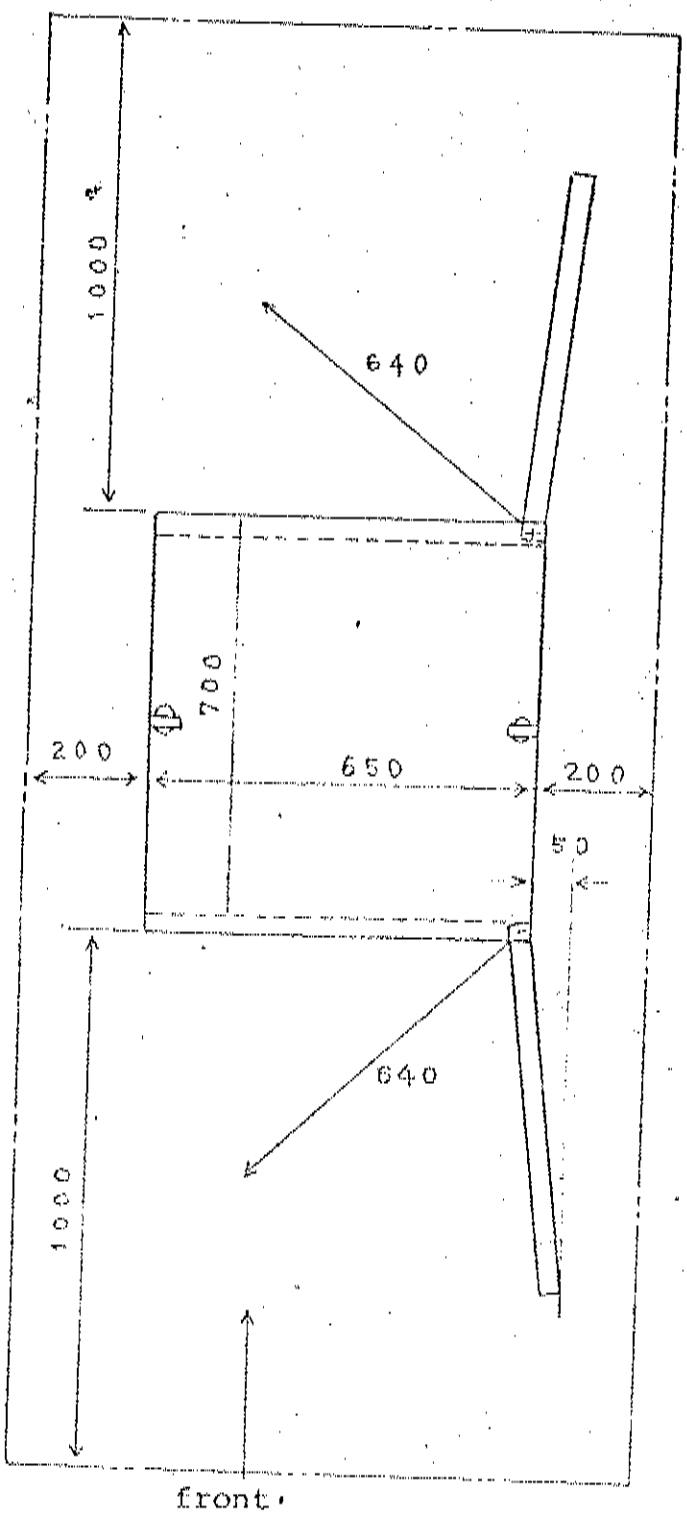
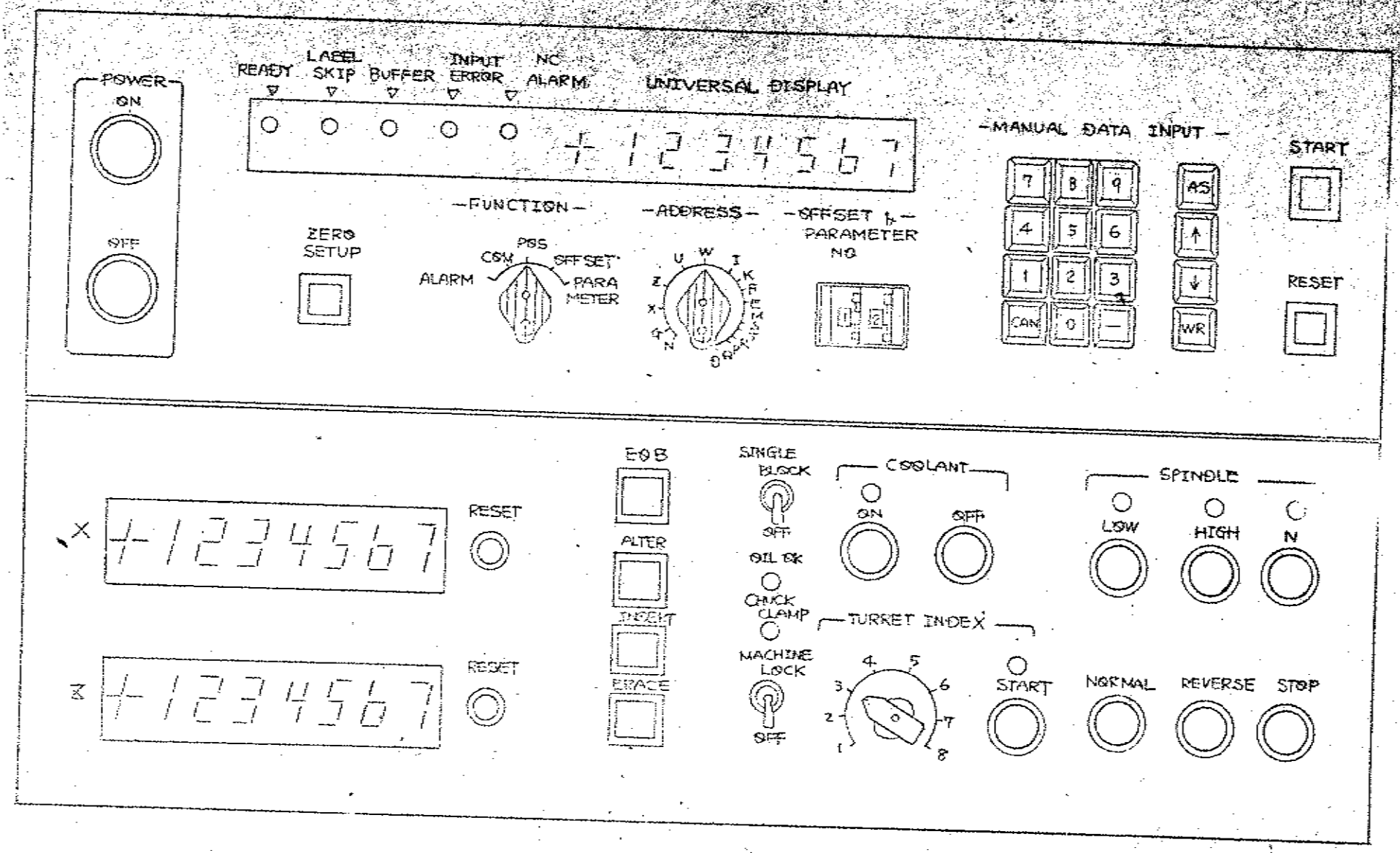


Fig. 2.4 YASNAC 2000B maintenance area



(control panel of optional unit)

Fig. 2.5 YASNAC 2000B control panel

3. PROGRAMMING

3.1 Address code and function code

1) The EIA RS244-A code is used.

Address code table

Address	Significance
N	sequence number
G	preparatory function (G-function)
X	absolute X-coordinate (diameter)
Z	absolute Z-coordinate
U	incremental X-coordinate (diameter), dwell
W	incremental Z-coordinate
I	incremental X-coordinate of circle center, parameter of fixed cycle
K	incremental Z-coordinate of circle center, parameter of fixed cycle
E	precise designation of thread lead
F	feedfunction (F-function), thread lead designation
M	miscellaneous function (M-function)
S	spindle-speed function (S-function)
T	tool function (T-function)
P	sequence number for multiple repetitive cycles
Q	sequence number for multiple repetitive cycles
D	depth of cut for multiple repetitive cycles

Function code table

EIA	Significance	ISO*
Blank	parity error in significant data area	NUL
BS	disregarded	BS
Tab	disregarded	HT
CR	end of block	LF/NL
SP	disregarded	SP
ER	rewind stop	%
UC	disregarded	/
LC	disregarded	/
+	disregarded	+
-	negative sign	-
0-9	numerals	0-9
a-z	address codes (other letters are error)	A-Z
/	optional block skip	/
Del	disregarded (including All Marks)	DEL

2) Codes not listed above are regarded as error in significant data area.

3) Leading zeros may be suppressed for numerals following an address.

3.2 S-function

S-function can be specified in a 4-digit figure following address S. Usually, the spindle speed (rpm) can be specified directly.

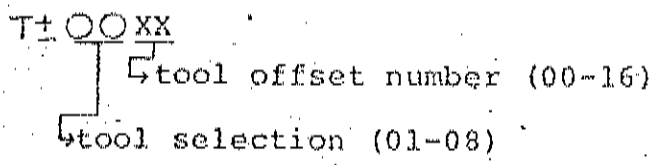
When performing constant speed control, the surface speed (m/min) is specified directly.

See page 42 for the details of the constant surface speed control.

3.3 T-function

T-function consisting of T followed by a 4- or 3-digit figure determines a tool and tool offset number at the same time.

1) Significance of 4-digit T command



a) Tool selection
The leading 2- or 1-digit figure determines the tool number to be used. If the figure is zero or omitted, it is regarded as input error because no tool is specified.

b) Tool offset number
The following 2 digits determine a tool offset number of which offset value is given in advance by manual data input. See page 67 for the details of setting tool offset.

2) Actual coordinates of tool tip
When a tool offset number is given, programmed coordinates are compensated for tool offset. Compensation is done not only for tool offset but for tip nose radius. Compensation for tool offset becomes effective when a tool offset number is given. See page 33 for the details of compensation of tip nose radius (cutter compensation).

a) An offset value is added to programmed coordinates and the resulting coordinates give the actual track of tool tip.

b) Tool offset becomes effective immediately when a tool offset number is given, and it

is held until a new tool offset number is given.

c) When the tool offset number given is 00, tool offset is cancelled.

d) With M02 or reset operation, tool offset is cancelled without the tool traveling.

e) When a T-function and coordinate command are given in the same block, the tool moves to the command value compensated and therefore it is possible to correct taper angle.

f) When a T-function alone is given, the tool is moved by the amount of tool offset.

g) For the compensation of tool offset, the tool travels at the feedrate currently given. Before issuing a T-function, therefore, a feedrate command (G00 or G01 and F) is needed in the same block or a preceding one.

h) A tool selection code other than 01-08 is regarded as input error.

i) Tool offset is regarded as zero for tool offset numbers other than 01-16.

3.4 M-function

An M-function consists of M followed by 2 digits. More than one M-function cannot be issued in the same block.

3.4.1 M00 (program stop)

(1) When M00 is issued, the spindle and coolant stops and the program is interrupted. To restart operation, depress the CYCLE START switch provided on the control pendant of the machine tool.

(2) When operation is resumed and the program has been restarted with the CYCLE START switch depressed, the S- and F-commands given in the program remain effective but it is required to reissue a spindle start command, M03 or M04, and a coolant start command, M08.

(3) ~~An M-function lock switch, M00 DEL, is provided in the relay unit at the rear of the panel. If M00 is issued when the switch is turned ON, the spindle and coolant do not stop but the NC alone comes to program stop.~~ ② "M00-DELETE" is provided for external input or 'SETTING' NO '50'. If M00 is issued when "M00 DELETE" is ON, the spindle and coolant do not stop but the NC alone comes to program stop. (If memory mode, executing NC data area is rewinded.)

(4) A program that attempts to change the direction of spindle revolution after stopping the spindle in such a way as M03 - M00 - M04 or M04 - M00 - M03 is regarded as program error. Such a program should be corrected as M03 - M05 - M00 - M04, for example.

3.4.2 M02 (program end)

(1) Issue this command in a single block at the end of a program.

(2) When M02 is executed, the spindle and coolant stop running and the NC processor is cleared.

(3) Then label skip function becomes effective at the same time and tape information is skipped until the first EOB is encountered.

(Example)

```
G50 X -20000 Z 6000
G00 T0404 S100 M03
X -4500 Z500
```



```
G00 U-500 W 500
    X -20000 Z6000
    T0400
    M02
```

3.4.3 M03 (normal spindle revolution)
M04 (reverse spindle revolution)
M05 (spindle stop)

(1) Stop the spindle before reversing its revolution.

(Example)

```
G50 X -20000 Z6000
G00 T0101 M03          ———— Spindle runs in
    X -7000 Z500             the normal direction.
```



```
G00 X -20000 Z6000 M05 ———— Spindle stops.
    T0100
G50 X -24000 Z10000    _____ Spindle revolution
G00 T0202 M04                is reversed.
```

A program that attempts to reverse the spindle revolution without stopping it is regarded as input error.

(2) Do not issue M05 in the same block with G01, G02, and G03, but in the same block as G00 or in an independent block.

(3) M00 or M05 (or M02) stop the spindle. To restart it, reissue M03 or M04.

3.4.4 M08 (coolant ON)
M09 (coolant OFF)

(1) M08 turns on coolant and M09 turns it off. With the coolant on/off switch on the NC control panel, it is also possible to turn on and off coolant even during tape operation.

(2) M00 or M02 issued in the middle of a program stops coolant. To restart coolant supply, issue M08.

3.4.5 M23 (chamfering ON)
M24 (chamfering OFF)

(1) Issue them before executing a thread cutting program using fixed cycle G92.

(2) M23 commands thread chamfering.
M24 cancels M23.

(3) Chamfering is turned off when power has been turned on or reset.

3.4.6 M10 (chuck clamp)
M11 (chuck unclamp)

When "M00 DELETE" on control panel (setting function 'SA') or external input is "1", chuck is clamped or unclamped by NC data (M10, M11). When M10 is issued, chuck is clamped and chuck is unclamped by M11.
While spindle is revolved, chuck is clamped or unclamped by M10 or M11.

3.5 G-function (preparatory function)

3.5.1 General description on tool travel commands

G00, G01, G02 and G03 are used for positioning, linear interpolation and circular interpolation. Before explaining these commands, interpolation is described below.

(1) There are six characters; X, Z, U, W, I and K, prefixed to coordinate commands. They indicate whether the coordinates given are absolute or incremental values, as shown below.

They may be mixed in a single block as X ... and W ... for example.

address	command type		related axis
X Z	absolute	diameter	X Z
U W	incremental	increment in diameter	X Z
I K	incremental	radius	direction of X direction of Z

(2) The feedrate specified with an F-command indicates the feedrate in the tangential direction of a figure.

(3) Acceleration and deceleration of tool travel is automatically controlled and need not be programmed.

3.5.2 G50 (Absolute coordinate preset)

(1) This command establishes the origin of the coordinate system of the workpiece as the current position of the tool tip.

Program the current tool position as G50X... Z... and all of the succeeding commands are processed on this newly established coordinate system.

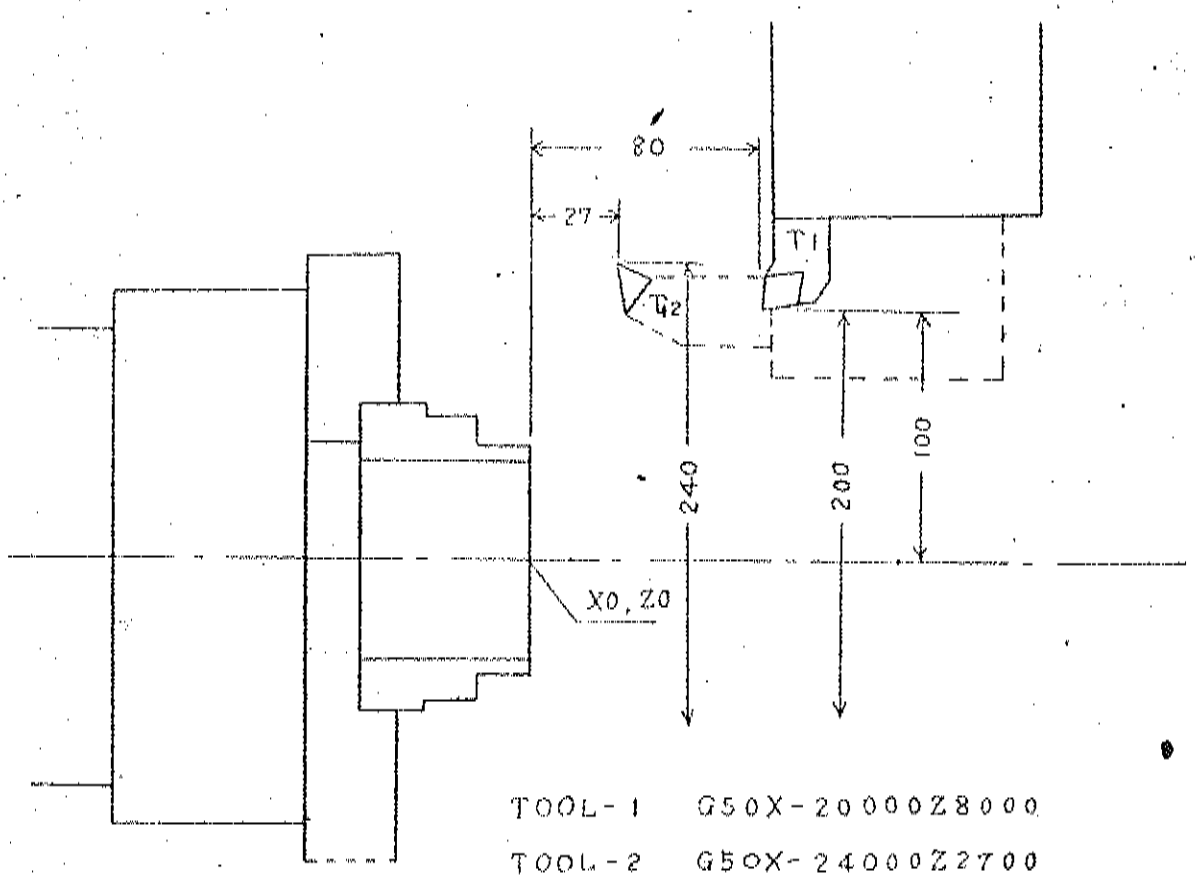
Establish a new coordinate system if the origin has been moved in relation to the workpiece since the tool has been replaced without the tool position being changed.

(2) Be sure to use absolute coordinates for G50.

(3) No other function may be included in the same block as G50.

Issue G00 or G01 whenever an M- or T-function is delivered in the block next to that of G50.

(Example)



3.5.3 G00 (positioning)

(1) Command format: G00 X(U) ..., Z(W)... *

(2) This command causes the tool to travel fast to the point (X, Z) in the coordinate

system established by the absolute coordinate preset command (G50).

It is also possible to control cutter travel incrementally using prefixes U and W.

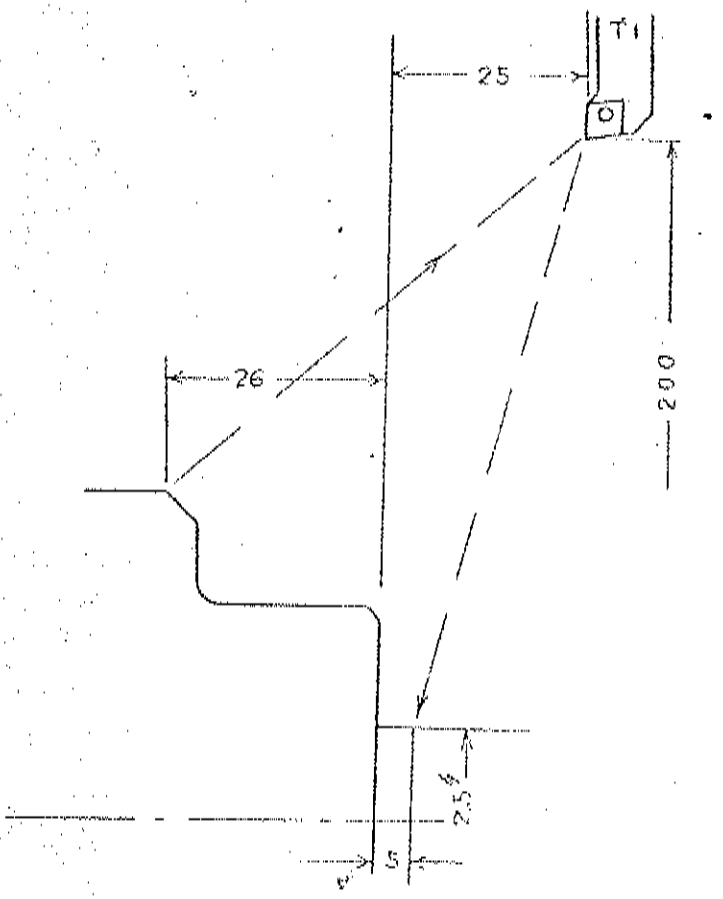
(3) The speed of rapid traverse with G00 is
X: 4,800 mm/min
Z: 9,600 mm/min

(4) G00 is also used together with M- and T-functions.

(Example) G00 T0101 M03* .

(5) G00 is a modal command like G01, G02 and G03 and it can be omitted in the following blocks.

(Example)



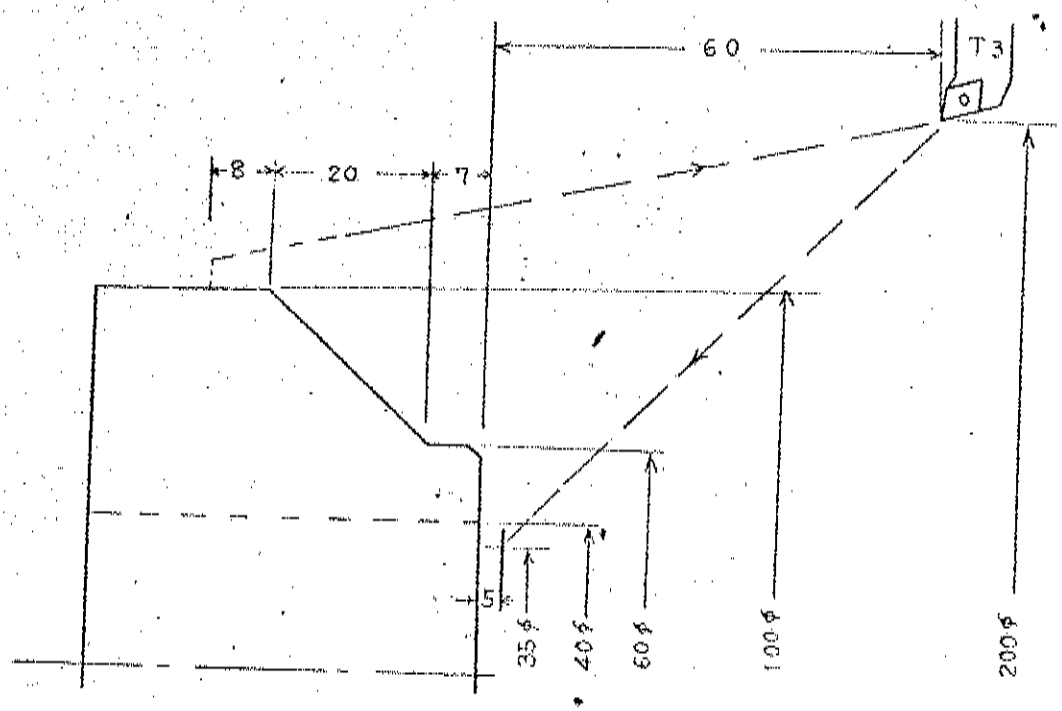

```
G50X-20000Z2500
G00 T0101 M03 S80
    X-2500 Z500
G01,Z 0 , F200
.
.
.
G00 X-20000 Z2500
    T0100 M09
    M05
    M02
```

3.5.4 G01 (linear interpolation)

(1) Command format: G01X(U) ....Z(W).... *

(2) A feedrate must be specified following the G01 command. If the same feedrate is employed for continuous cutting, the following blocks need not involve any F-command. To change the feedrate, issue a new F-command.

(Example)




```
G50 X-20000Z6000
G00 T0303 M03 S100
    X-3500 Z500
G01 Z0      F15        ———— feedrate 0.15mm/rev
    X-5800 F20         ————
    X-6000 Z-100               feedrate 0.20mm/rev
    Z-700              ————
    X-10000 Z-2700 F30 ————
    Z-3500                     feedrate 0.30mm/rev
    X-10500            ————
G00 X-20000 Z6000 T0300 M05
```

3.5.5 G02 (circular interpolation for counterclockwise cutting)

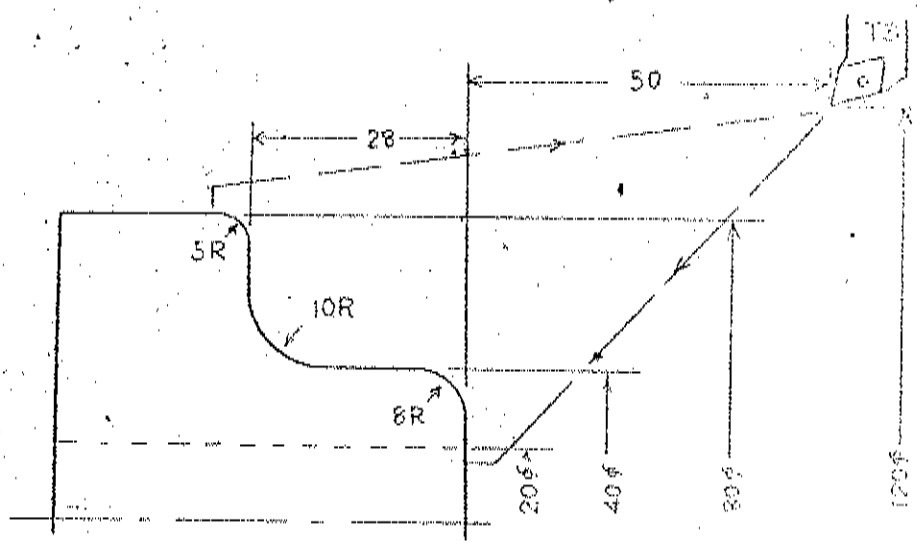
G03 (circulation interpolation for clockwise cutting)

(1) Command format: G02, G03 } X(U)...Z(W)... I... K...

(2) X... and Z... (or, U... and W...) give the end point on a circle. I... and K... are the incremental coordinates of the circle's center taken from the start point. I... is the circle's radius lying along the X-axis while K... lies along the Z-axis.

(3) It is possible to program circular interpolation over more than one quadrant in a single block.

(Example)



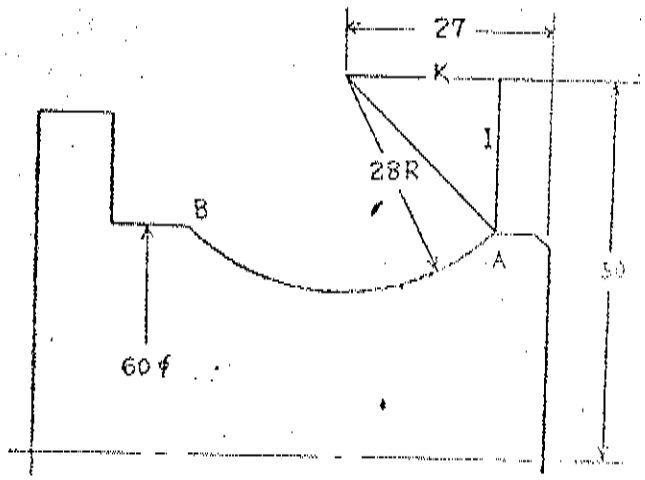

```
G50X-1.2000 Z5000
G00 T0303 M03 S100
    X-1500 Z500
G01 Z0 F50
    X-2400 F15
```

```
G02 X-4000 Z-800 K-800      ——— counterclockwise
G01 Z-1800                      travel
G03 X-6000 Z-2800 I-1000    ——— clockwise travel
G01 X-7000
G02 X-8000 Z-3300 K-500     ——— counterclockwise
G00 U-500                       travel
    X-12000 Z5000 T0300
```

Circular interpolation may be done on two or more quadrants.

In such a case as shown below where the figure extends over two quadrants, one block is enough for programming.

(Example)



- o center of the circle X - 10000 Z-2700
- o dimension I $\frac{100 - 60}{2}$ 20
- o dimension K $\sqrt{28^2-20^2} = \sqrt{384} = 19.5959$

point A X-6000 Z-740

point B G03X-6000 Z-4660
I-2000 K-1960

3.5.6 Dwell

Command format: G04 U...CR

Reading this command, the processor suspends operation for the duration specified. The unit of time is 0.01 second.

(Example) To suspend operation for 5 seconds, program G04 U 500CR.

(Example)

```
G00 X-6000
G04 U 200          ..... 2-second dwell
    X-10500 W500
    X-20000
    T0300
```

Another function existing in the block of G04 is regarded as input error.

3.5.7 G32 (thread cutting)

Command format: G32X(U) ... X(W) ... F.... CR
The G32 command allows cutting of any of straight, taper or scroll threads.

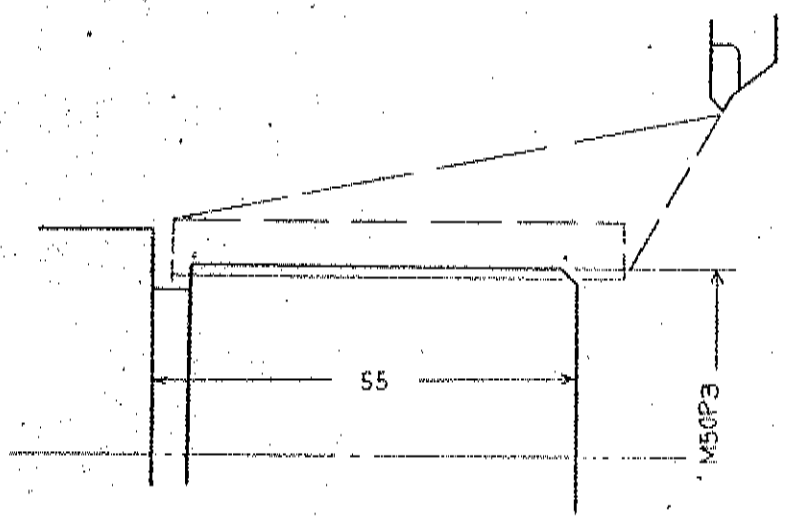
Note 1: During thread cutting, feedrate override is ineffective.

Note 2: During thread cutting, halt operation is disregarded.

Note 3: The spindle speed must remain unchanged until a screw has been cut.

Note 4: G32 does not perform chamfering.

(Example)




```
G50 X-20000 Z6000
G00 T0202 S80 M03
    X-4840 Z1000
G32 Z-5200          F300 ... screw pitch 3.00 mm
G00 X-6000
    Z1000
    X-4720
G32 Z-5200
G00 X-6000
    Z1000

G00 X-20000 Z6000
    T0200
```

G32 (accurate thread cutting) with E assignment
Command format: G32X(U) ... Z(W) ... E ... CR
This command is used to cut precision thread by specifying its pitch in units of 1/10,000 mm for which the F assignment is not enough.
For thread cutting : F has A 4 Digit format (F ****)
E has A 6 Digit format (E ******) in The inch system.

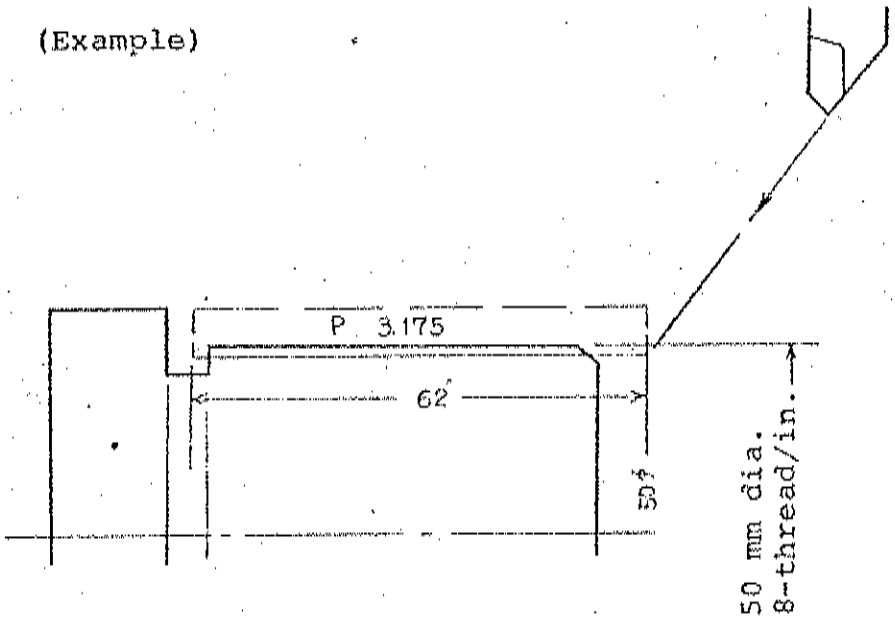
Note 1: During thread cutting, feedrate override is ineffective.

Note 2: During thread cutting, halt operation is disregarded.

Note 3: The spindle speed must remain unchanged until a screw has been cut.

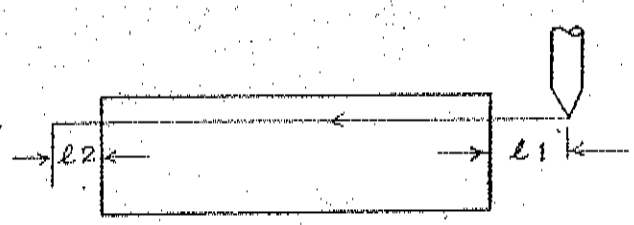
Note 4: G32 does not perform chamfering.

(Example)




```
G50 X-20000 Z6000
G00 T0505 S80M03
    X-4840    Z1000
G32           W-6200 E31750 ... pitch 3.175mm
G00 X-6000
              Z1000
    X-4720
G32           W-6200
G00 X-6000
              Z1000
```

Note 5: To prevent incomplete thread cutting: Because of automatic acceleration and deceleration, the lead would become incorrect at both ends of a screw for the lengths of $\ell 1$ and $\ell 2$ as shown below. It is therefore required to program with sufficient margins for $\ell 1$ and $\ell 2$.



L1 and L2 can be approximately obtained from the fellowing formulas.

$$L1 = \frac{N.P}{60.K}\left(\ln\frac{1}{a} - k\right) = L2\left(\ln\frac{1}{a} - 1\right)$$

$$L2 = \frac{N.P}{60.K}$$

where N: spindle speed (rpm)
P; screw pitch (mm/rev)
K: constant = 30
a: screw pitch accuracy
ln: natural logarithm

Example

If a = 1/100 when N = 500 rpm and P = 3mm/rev,

L1 = L2 x 3.61 = 3.00mm

L2 = 500 x 3/(60 x 30) = 0.83mm

a	$(\ln\frac{1}{a} - 1)$
$\frac{1}{50}$	2.91
$\frac{1}{100}$	3.61
$\frac{1}{200}$	4.30
$\frac{1}{300}$	4.70

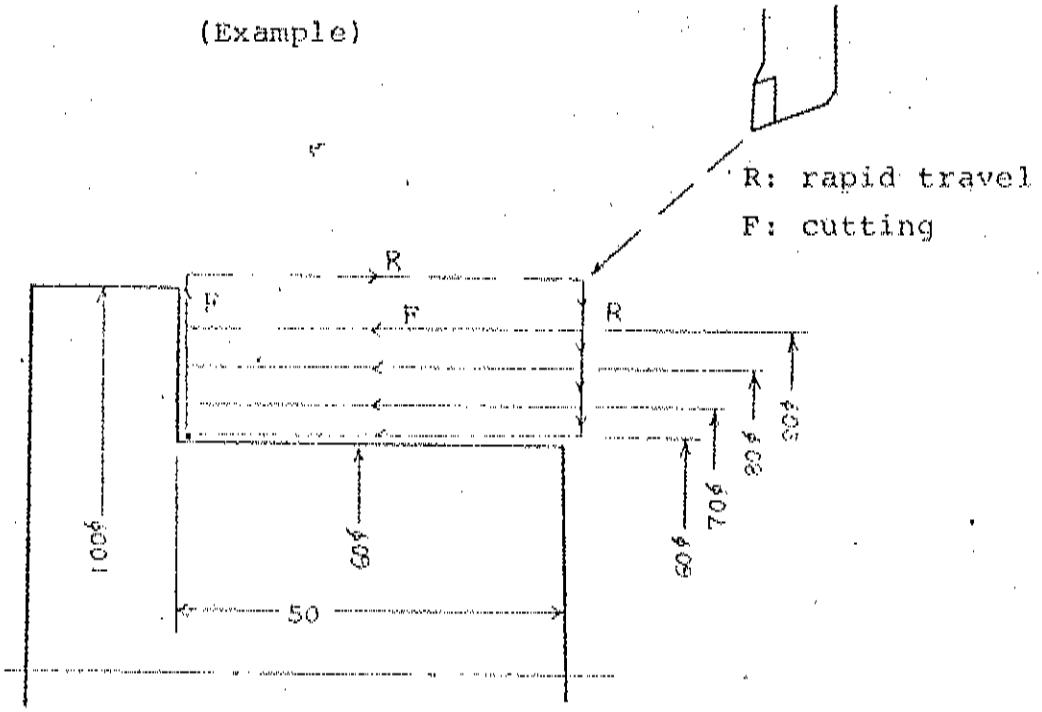
3.5.8 G90 (Straight/taper cutting cycle)

(1) Straight cutting

Command format: G00 X... Z... F...CR

Once a G90 command has been issued, it is enough to give the depth of cutting in the X-direction.

(Example)

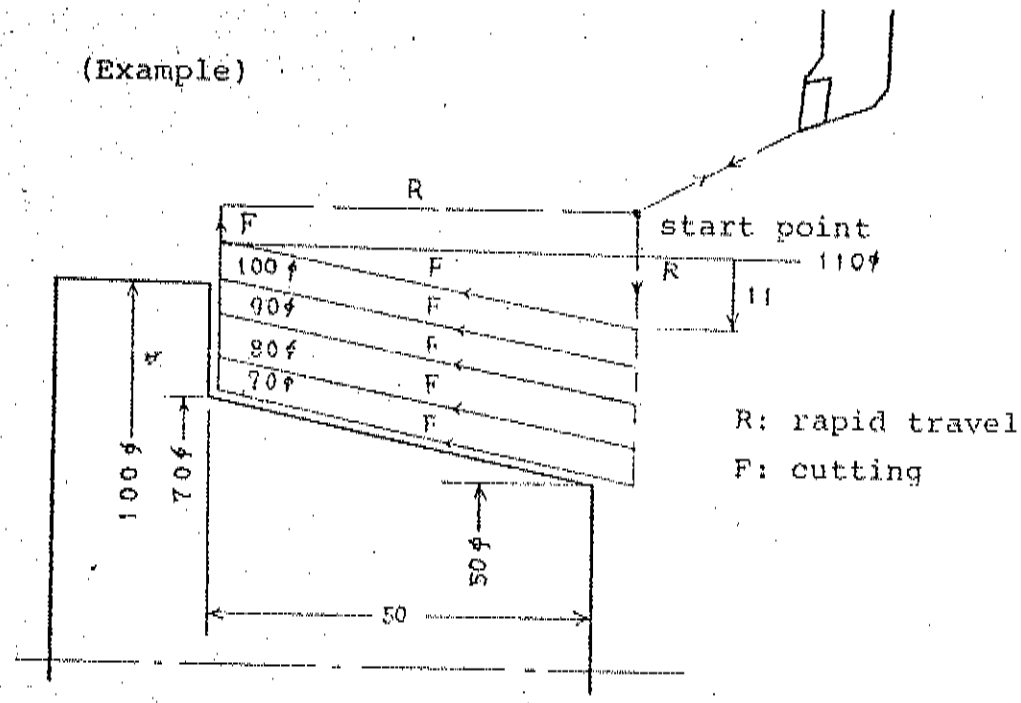



```
G50 X-20000 Z7000
G00 T0101 S80 M03
    X-10500  Z200
G90 X-9000   Z-5000 F035
    X-8000
    X-7000
    X-6000
G00 X-20000  Z7000
    T0100.
```

(2) Taper cutting

Command format: G90 X... Z... I... F...CR

(Example)




```
G50 X-20000 Z7000
G00 T0101 S90M03
    X-11200  Z500
G90 X-11000  Z-5000 I  1100  F035
    X-10000
    X- 9000
    X- 8000
    X- 7000
G00 X-20000 Z7000
    T0100
```

Note: The value of I is the increment from the end of taper to the start (i.e. the increment of radius).

3.5.9 G92 (Straight/taper thread cutting cycle)

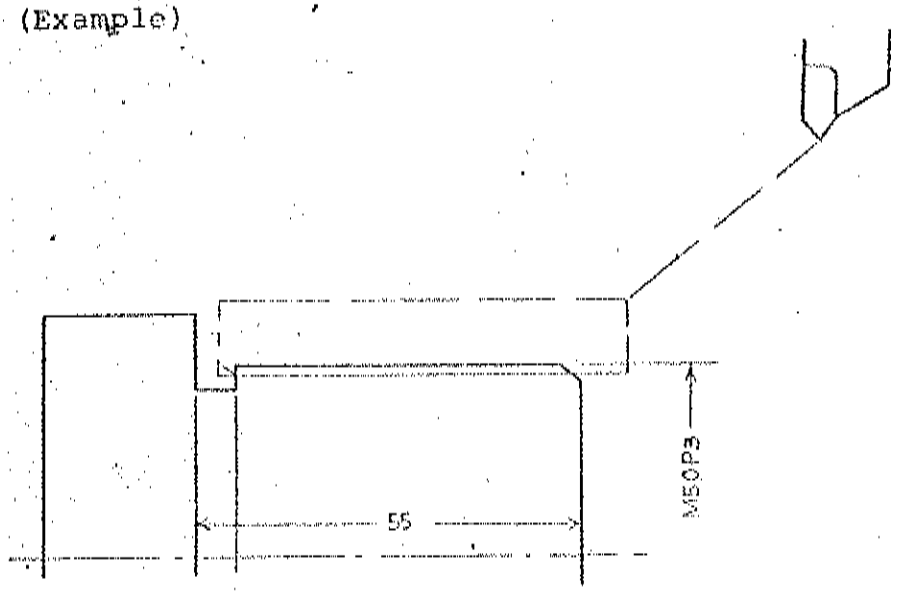
(1) Straight thread cutting

Command format: G92X... X... F... OR E...

M23 is the chamfering command that chamfers screws at 45° for 0.8 pitch at the end of screw.

The E assignment is to cut precision thread.

(Example)




```
G50 X-20000  Z6000
G00 T0505 S80 M03
    X-7000   Z1000  M23 - chamfering command
G92 X-4840   Z-5200 F309
    X-4720
    X-4620
      :
      :
G00 X-20000  Z6000  M24 - cancels M23
    T0500
```

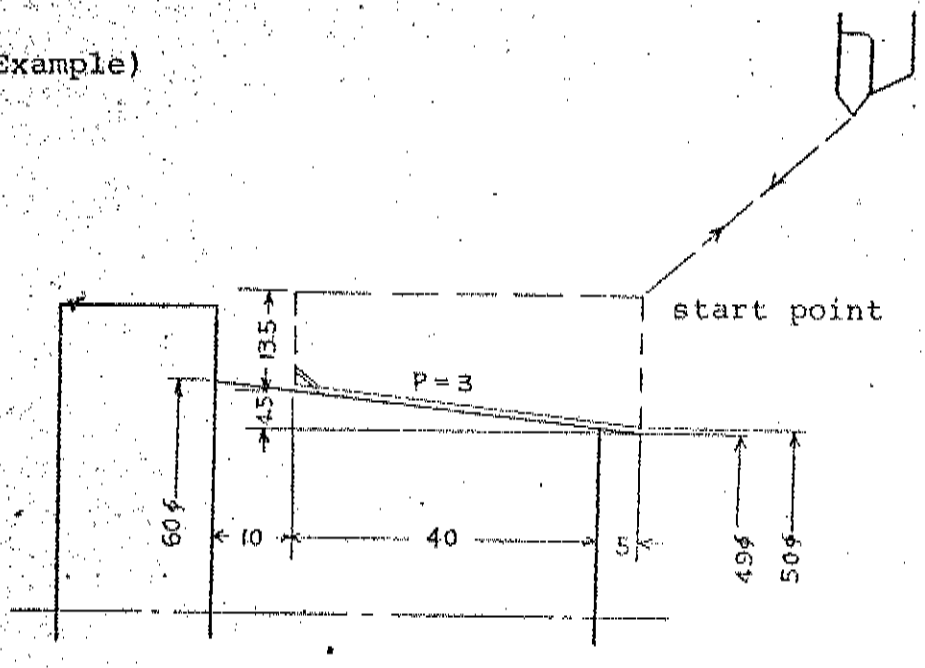
(2) Taper thread cutting

Command format: G92X... Z... I... F... CR
E...

M23 is the chamfering command for screw.

The E assignment is to cut precision thread.

(Example)




```
G50 X-20000   Z6000
G00 T0505 S90 M03
    X-8500    Z500    M23 - chamfering command
G92 X-5800    Z4000   I450   F300
    X-5000
    X-4400
      :
      :
```

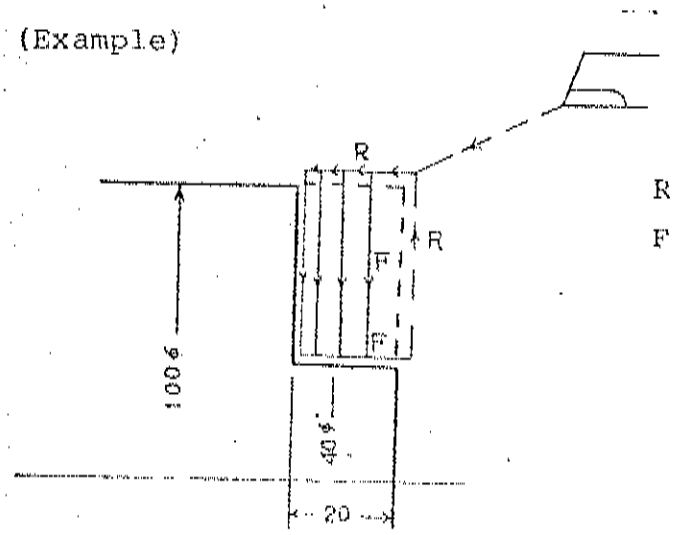
3.5.10 G94 (Straight/taper facing cycle)

(1) Straight facing

Command format: G94X... Z... F... CR

Once a G94 command has been issued, it is enough to give the depth of cutting in the Z-direction.

(Example)



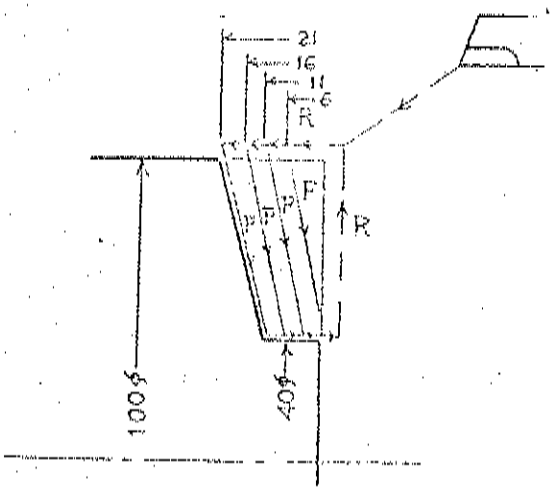
R: rapid travel
F: cutting

```
G50  X-20000   Z7000
G00  T0202  S70 M03
     X-10500   Z200
G94  X-4000    Z-500   F035
               Z-1000
               Z-1500
               Z-2000
G00  X-20000   Z7000
     T0200
```

(2) Taper facing

Command format: G94X... Z... K... F... CR

(Example)



R: rapid travel
F: cutting

```
G50 X-20000  Z7000
G00 T0202 S70 M03
    X-10600  Z200
G94 X-4000   Z0  K-600  F035
             Z-500
             Z-1000
             Z-1500
G00 X-20000  Z-7000
    T0200
```

3.6 F-function

This is the function to specify the tool feedrate. It consists of address F followed by a 4-digit figure that specifies the tool feedrate for each revolution of the spindle.

(1) The 4-digit figure is in units of 1/100 mm/rev.
Ex. F1234 = 12.34 mm/rev

(2) The feedrate commanded by tape is that of 100-per cent override.

(3) The feedrate is restricted according to the spindle speed.

[example]

	spindle speed	maximum feedrate
	80 rpm	40.95mm/rev
	125	32.76
	220	18.61
	330	12.40
	500	8.19
	750	5.46
	1300	3.15
	2000	2.04

(4) Remarks for F-function

(a) In the first block of thread cutting, it is required to designate the lead with an F- or E-command. In the first block where thread cutting has just been switched to simple cutting, it is required to designate the feedrate with an F-command.

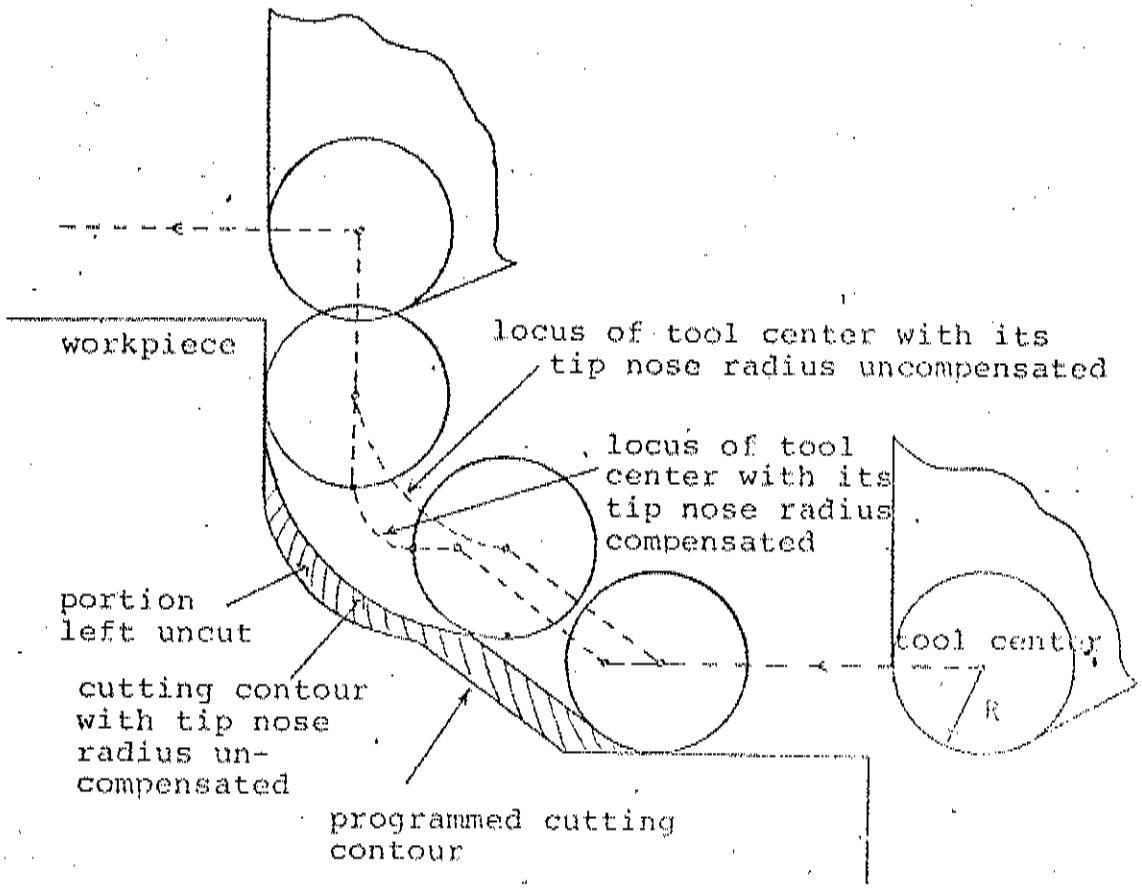
(b) The F-function gives the feedrate in the tangential direction.

(c) A block which involves G01, G02 and/or G03 to cut a workpiece but does not involve any F-command is regarded as input error.

(d) During thread cutting, feedrate override is taken as 100 percent irrespective of the override switch setting.

3.7 Compensation for tip nose radius (cutter compensation)

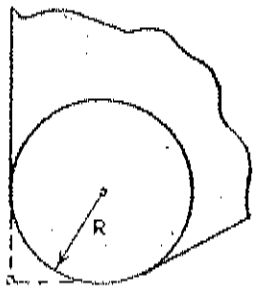
Because of the round end of tool tip, tool offsetting is sometimes not enough for taper and circular cutting and the compensation for tip nose radius aims at automatically compensating for the error left uncompensated.



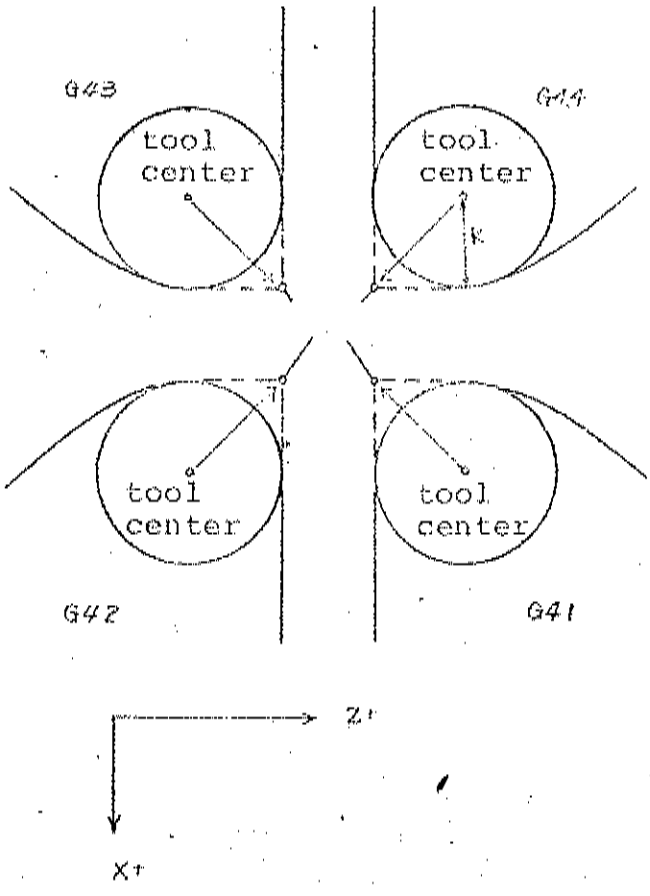
3.7.1 Virtual tool tip

In the above figure, the path of the tool is indicated as the locus of the center of the tool tip. Actually, however, the coordinates of a point called virtual tool tip are displayed as the command or current coordinates.

The virtual tool tip is the position of tool tip on the assumption that the tool is infinitely sharp.



The relationship between the virtual tool tip and the actual center of the tool tip is determined by the shape of the tool and there are four possible cases as shown below. To compensate for the tip nose radius, use one of G41, G42, G43 and G44 according to the tool-workpiece relationship.

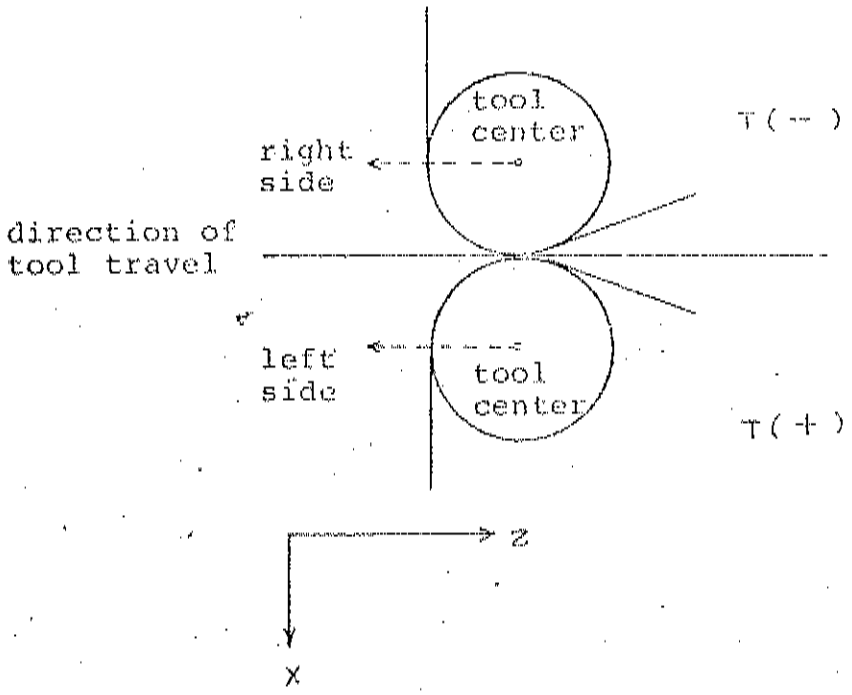


3.7.2 Sign of T-code

Compensation for tip nose radius is given through the MDI unit as for tool offset (see pages 67 and 68).

To compensate for the tip nose radius, a tool offset number has to be actually specified in the program and the corresponding value determined though the MDI unit is used for actual compensation.

Attention must be paid to the sign of T-code. It is related to the direction in which the tool center is offset with respect to the direction in which the tool travels, and it must be determined according to the shape of the tool and the cutting direction.



sign of T-code	direction to offset tool center
T-...	right side viewed in the direction of tool travel
T+...	left side viewed in the direction of tool travel

in the coordinate system shown above

3.7.3 Precautions for programming

(1) To compensate for the tip nose radius, issue G41, G42, G43 or G44 together with a tool offset number.

(2) Compensation for the tip nose radius comes into effect when one of G41 through G44 and a tool offset number have been encountered.

(3) Issue G40 to cancel compensation for the tip nose radius.

(4) The tool travel command involved in the block where cutter compensation is started or cancelled must be G00 or G01. Otherwise the program is regarded as input error.

(5) Before switching one of G41 through G44 to another, issue G40 to cancel cutter compensation.
If a new G-command is issued without cancelling the current one, the program is regarded as input error.

G41 - G42 incorrect
G41 - G40 - G42 correct

(6) Issue each of G40, G41, G42, G43 and G44 in an independent block as a rule.

(Example)

```
    :
G00T0100
G40
G44
G50X0Z0
G00T-202
    :
G00T0200
G40
M02
```

(7) To change the value of cutter compensation, cancel the current value using G40 or the tool offset number of zero before giving a new value. Otherwise the current value would remain effective.

(8) In the following cases, cutter compensation will be cancelled and remain ineffective until a new cutter compensation command is encountered.
(a) Power has been turned on.
(b) The equipment has been reset.
(c) M02 has been issued.
(d) G40 has been issued.

(9) Even when one of G41 through G44 has been issued, cutter compensation remains ineffective in the following cases.
(a) The value of cutter compensation is zero.

(b) Zero is assigned to the storage of the OFFSET & PARAMETER No. "99" when the FUNCTION selector set to OFFSET. (Figure 8 should be normaly assigned.)

(10) In the following cases, cutter compensation is temporarily cancelled but is resumed.
(a) The block that requests cutter compensation does not involve any tool travel command.
(b) M00 (or M01) has been issued.

(11) Fixed cycles: G90, G92 and G94, and multiple repetitive cycles: G74, G75 and G76, do not perform cutter compensation.
G70, G71, G72 and G73 perform cutter compensation if specified.

(12) Cutter compensation is temporarily cancelled when the tool offset number of 00 is encountered. When a new tool offset number is given, cutter compensation restarts with a new tool offset.

(13) Special shapes programmed may be regarded as error.
(a) Reverse cutting
(b) The intersection of the loci of tool center cannot be obtained.

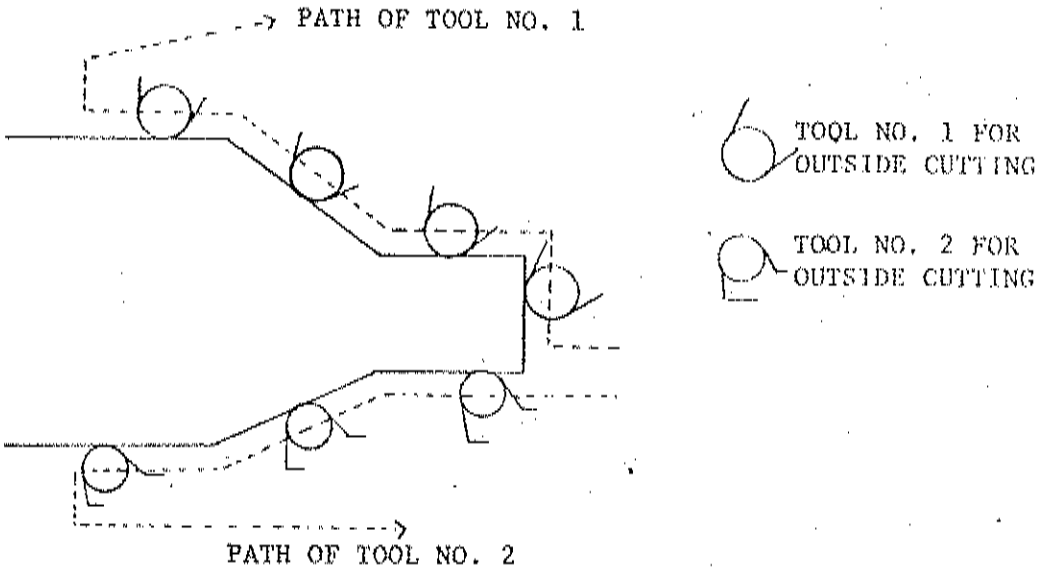
3.7.3 Input in the MDI mode

When cutter compensation is in effect, manual data input is impossible without resetting the equipment.

Cautions

(1) To make compensation for tip nose radius, G41, G42, G43 or G44, and a tool offset number must be issued. When both a tool offset number and a command (G41 through G44) are issued, compensation for tip nose radius is started. And when any one of them is cancelled, the compensation is stopped.

- G41 to G44 and T codes programming for outside or inside cutting.



N001

G50X··· Z···

G00 M03S06 T-0101: Selection of tool offset number No. 1. Specification of direction of tool center (Right side viewed from the direction of tool travel).

G44: Specification of direction of virtual tool tip (↙).

G00X··· Z···

G01Z··· F···

⋮ (Execution of tip nose radius compensation)

G00X

X··· Z··· T0100: Zero return and cancel of tool offset number.

G40: G code to cancel the tip nose radius compensation.

N002

G50X··· Z···

G00 T0202: Selection of tool offset number No. 2.
Specification of direction of tool center
(Left side viewed from the direction of tool travel).

G41: Specification of direction of virtual tool tip (↖).

G00X··· Z···

G01Z··· F···

(Execution of tip nose radius compensation.)

G00X···

X··· Z···· T0200: Zero return and cancel of tool offset number.

G40: G code to cancel the tip nose radius compensation.

M02

(2) Each of command G41, G42, G43 or G44 should be issued in an independent block next to the block in which tool number and tool offset number are specified. (Refer fo (5) - (b))

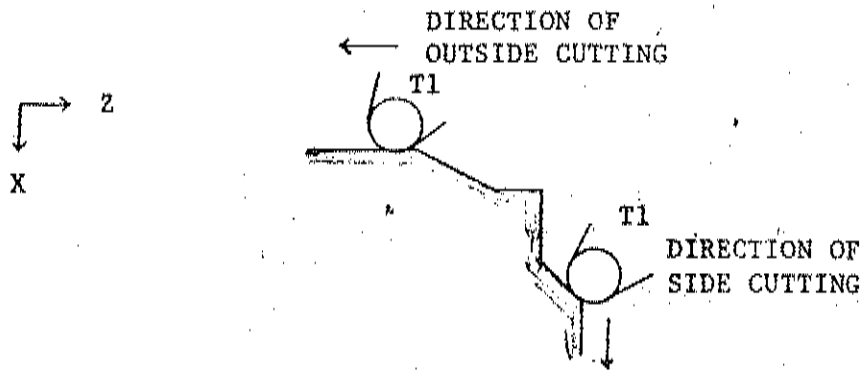
(3) Take the following procedure when the code G41, G42, G43 or G44 is required to change.

1 Cancel of tool offset number
2 G40 command specification
3 Specification of new tool number and tool offset number
4 Specification of new G command. (G41 to G44)

When G40 command is not issued, it is regarded as input error.

(4) How to change the code of tool command

When outside cutting is made in the minus direction of Z-axis after side cutting in the plus direction of X-axis using one tool, the tool center is changed in the direction of tool travel. Therefore, the codes of tool command must be changed (Side cutting: Left side, Outside cutting: Right side).



N00:

G5C ·· Z···

G0[illegible]0101 M03 S[illegible]: Selection of tool offset number No. 1. Specification of direction of tool center (Left side viewed from the direction of tool travel).

G44: Specification of direction of virtual tool tip (↙).

G00X··· Z···
G01X··· F···
⋮
G00X···

} Side cutting program (Execution of tip nose radius compensation).

G40
G00T0100

} Tip nose radius compensation and tool offset number are once cancelled.

T-0101: Selection of tool offset number No. 1 Specification of direction of tool center (Right side viewed from the direction of tool travel).

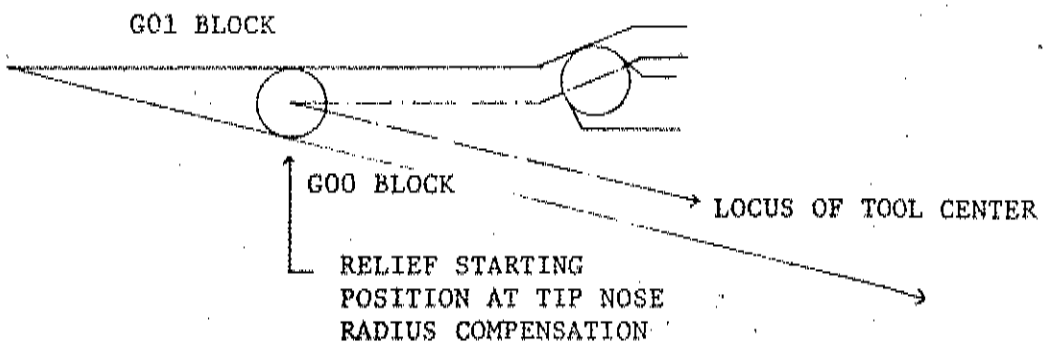
G44: Specification of direction of virtual tool tip (↙).

G01 Z···

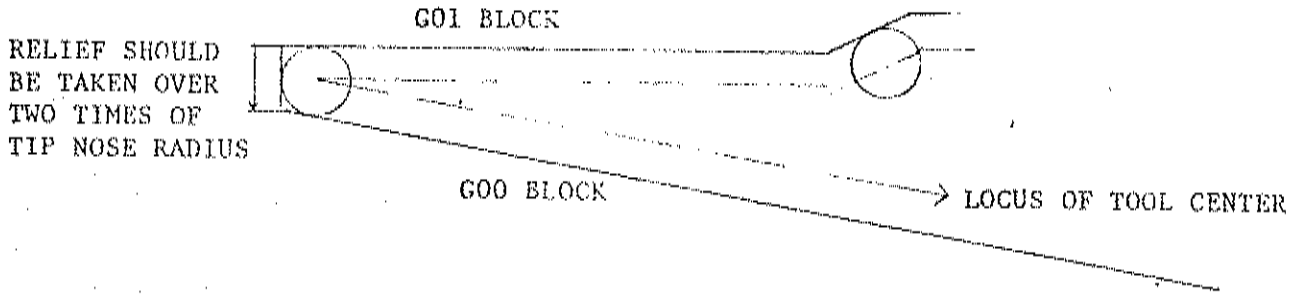
Outside cutting program.

(5) Tip nose radius compensation of YASNAC is made by operation of the intersecting point of tool center locus. When the move of acute angle is programmed, the intersecting point is away from the original figure. Therefore, be careful in the following case.

(a) When the relief of X axis in the inside cutting is small:



In this case, make program as follows

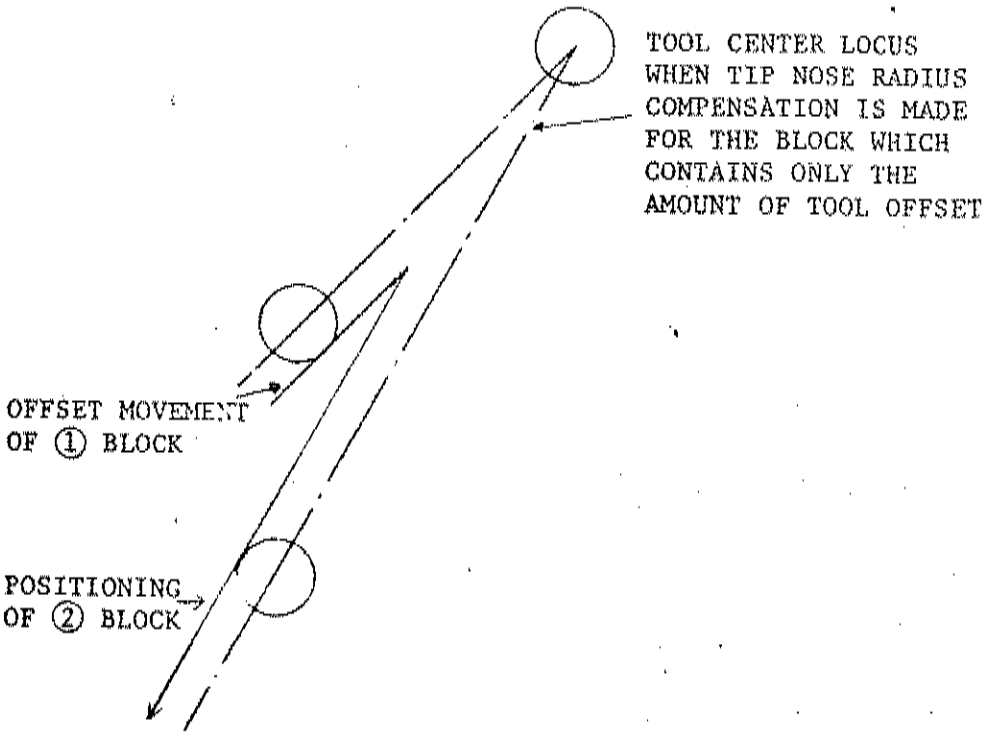


(b) When there is some change only in the amount of tool offset in the block which executes tip nose radius compensation.

```
G50X··· Z···
G41
G00T 0101 M03S04        ①
X··· Z···               ②
```

When the program as mentioned above is executed, the tip nose radius compensation is made only for the move of offset in ① block.

The following case may occur, depending upon the amount of offset and move of X and Z in ② block.



Therefore, when tip nose radius compensation is made, the block only for position compensation should be avoided.

For program above mentioned, G41 must be command after ① block.

3.7.4 Display of virtual tool tip

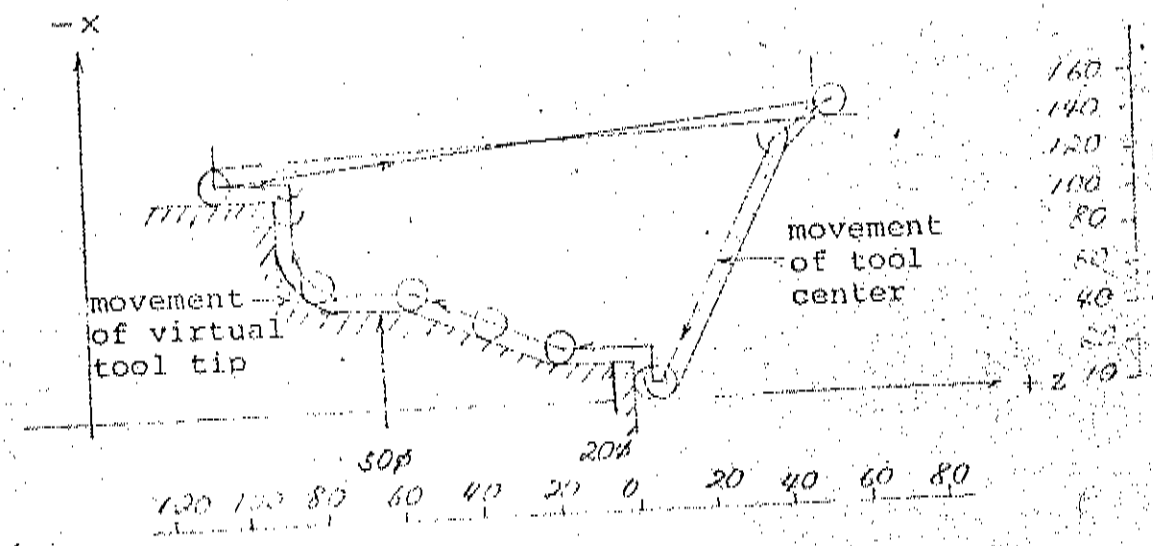
The current direction of virtual tool tip can be displayed in the following manner.

(1) Set the FUNCTION switch on the control panel to PARAMETER.

(2) Set the parameter and offset number to 99.

(3) Then the current direction of virtual tool tip is displayed on the universal display.

3.7.5 An example of program




```
      :
     G00 T0100
     G40
      :
N001 G44
N002 G50X-14000 Z5000
N003 G00 S4M03 T-0202
N004 X0Z1000
N005 G01Z0F20
N006 X-2000
N007 Z-2000
N008 X-5000Z-5500
N009 Z-7000
N010 G03I-2000KOU-4000W-2000
N011 G01X-11000
N012 Z-11000
N013 X-13000
N014 G00X-14000 Z5000 T-0200
N015 G40
N016 M05
      :
```

3.8 MULTIPLE REPETITIVE CYCLES

3.8.1 Multiple Repetitive Cycles (G70 to G76)

1) YASNAC 2000B is provided with the following optional functions.

<table>
<tr><th>G Code</th><th>Cycle Name</th><th colspan="2">Remarks</th></tr>
<tr><td>G70</td><td>Finishing</td><td colspan="2"></td></tr>
<tr><td>G71</td><td>Stock removal in turning</td><td rowspan="3">Finishing possible with the code G70</td><td rowspan="3">Tip nose radius compensation possible</td></tr>
<tr><td>G72</td><td>Stock removal in facing</td></tr>
<tr><td>G73</td><td>Pattern repeating</td></tr>
<tr><td>G74</td><td>Peck drilling in Z axis</td><td colspan="2" rowspan="3">Tip nose radius compensation impossible</td></tr>
<tr><td>G75</td><td>Grooving in X axis</td></tr>
<tr><td>G76</td><td>Thread cutting</td></tr>
</table>

2) The programs specified by G71, G72, or G73 are stored in the internal memory of NC unit. The maximum capacity to input is 25 blocks which can store a set of program. So when the following command is issued,

```
G71 Pα Qβ  ... *Cycle command for stock removal in turning
Nα ...  *
 :
 :                 } Program (A)
                     (25 Blocks max.)
Nβ ...  *
G70 Pα Qβ  ... *Finishing for program (A)
G72 Pα'Qβ' ... *Cycle command for stock removal in facing
Nα' ...  *
 :  ...  *        } Program (B)
 :                  (25 Blocks max.)
Nβ' ...  *
```

program (B) remains in the internal memory after erasing program (A). Therefore the finishing cycle specified by G70 is executed to the program (B) after completing G71 cycle.

3) Cycle G70 to G73 can execute the tip nose radius compensation and viceversa.

4) For G74 to G76, tip nose radius compensation is not effective.

3.8.2 Stock Removal in Turning (G71)

1) The code G71 can command the stock removal in turning for inside or outside diameter.

2) Command format

G71 ··· P ··· Q ··· U ··· W ··· I ··· K ··· D ··· F ··· S ··· *

Nα ..... *
..... *
F ··· *
S ··· *
F ··· *
Nβ ..... *

The cutting shape for finishing

Tool path: A → B → C
(See next figure)

where

- P – Sequence number (α) for cycle start
- Q – Sequence number (β) for cycle finish
- U – Finishing allowance in X-axis direction: diameter designation
- W – Finishing allowance in Z-axis
- I – Rough finishing allowance in X-axis direction: radius designation
- K – Rough finishing allowance in Z-axis direction
- D – Depth of cut: radius designation
  Designation without sign

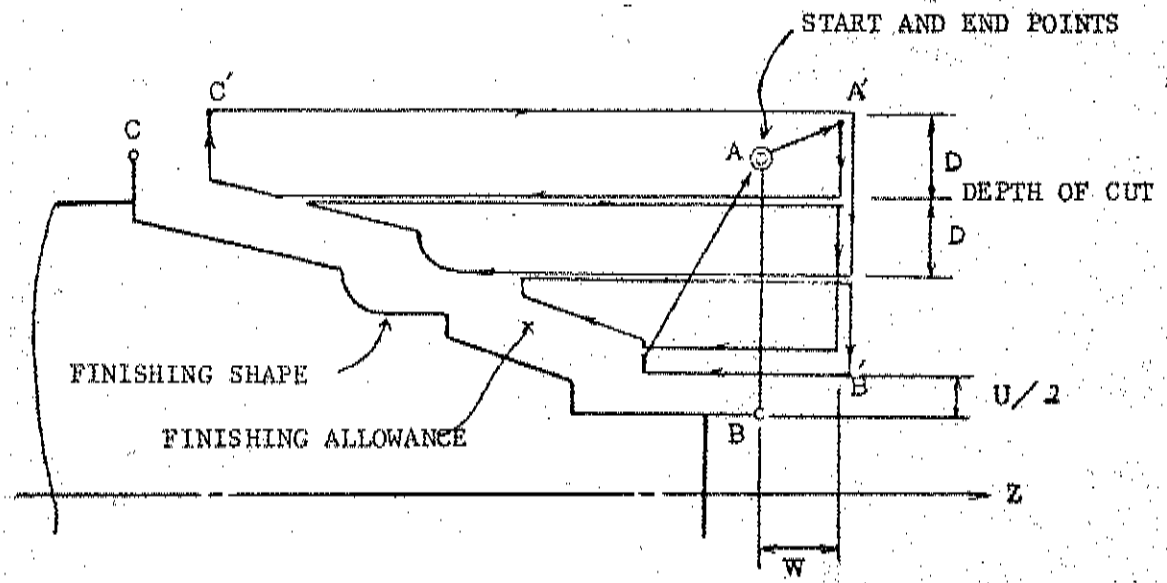
(U, W, I, K, D: Incremental value)

- F, S – Feedrate or spindle-speed designation in rough cutting cycle.

  Without this code, the previously specified code F or S becomes effective.

Note) Codes F and S specified in the sequence numbers Nα to Nβ are ignored and becomes effective for G70 finishing cycle.

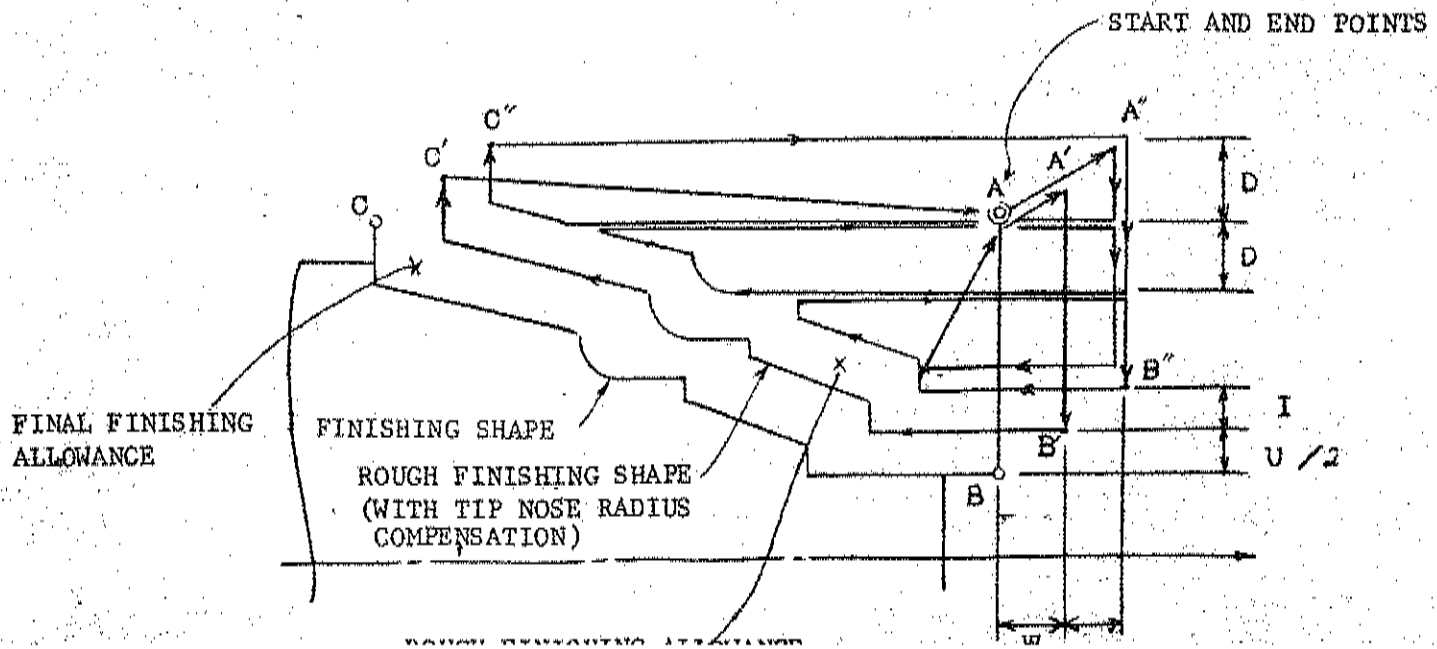
3) When I=0 and K=0, the tip nose radius compensation is not executed, and the cycle completes without cutting the finishing allowance as shown in the following figure. (See the next page)



4) When I≠0 and K≠0, after the rough cutting without the tip nose radius compensation, the rough finishing is made without cutting final finishing allowance while executing the tip nose radius compensation.

In this case, G codes (G41 to G44) for tip nose radius compensation and compensation number is necessary to specify in the preceding block in advance.

When the cancell mode of tip nose radius compensation is specified in this cycle, the rough finishing is executed without the tip nose radius compensation.



5) Regulations on programming Nα to Nβ are shown as follows:

a) Programmed order for finishing is A → B → C.

b) The tool path between A and B is specified by the block of sequence number α. In any case A to B must parallel to the x axis.

c) The final block is specified by the block of sequence number β. In any case straight line CA must parallel to the Z axis.

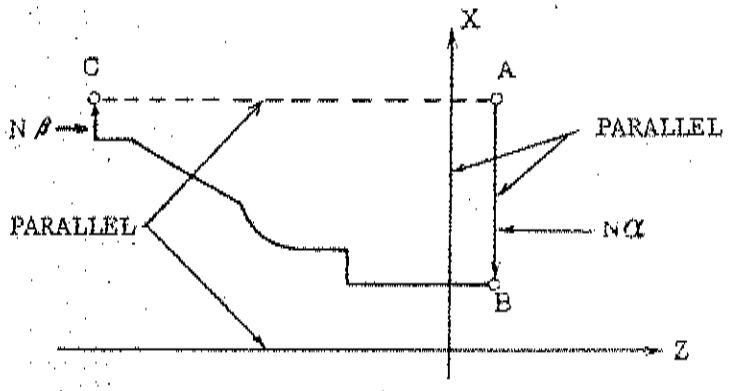
d) The tool path between B and C is specified by one of four codes G00, G01, G02 and G03. Otherwise the Command becomes format error.

e) The tool path between B and C should be monotonous increase or decrease to X or Z axis.

f) T, M and G codes except such as G00, G01, G02, and G03 should not be specified between the sequence number Nα and Nβ.

g) The number of blocks between the sequence number Nα and Nβ should not exceed 25 blocks. (If not, format error occurs.)

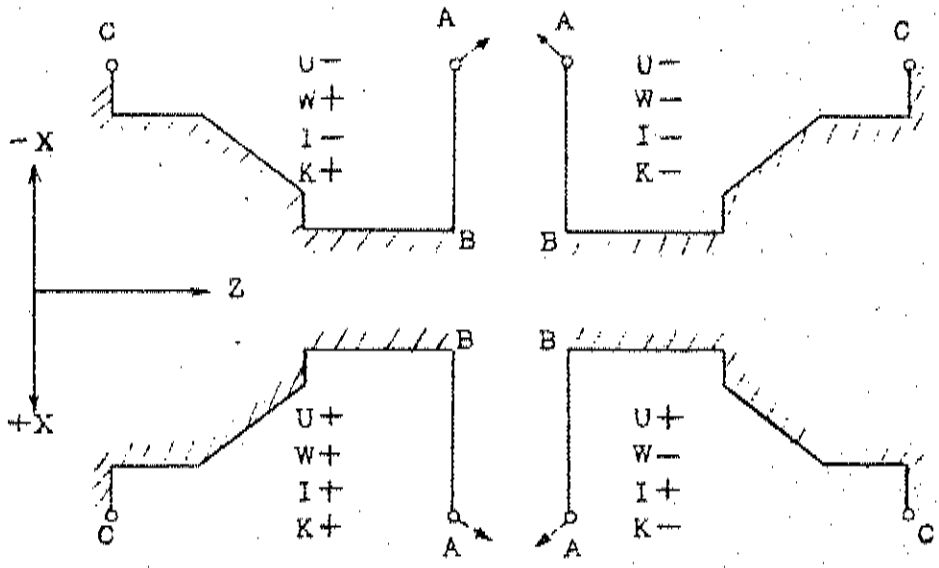
h) When executing tip nose radius compensation, follow the program regulations of tip nose radius compensation.



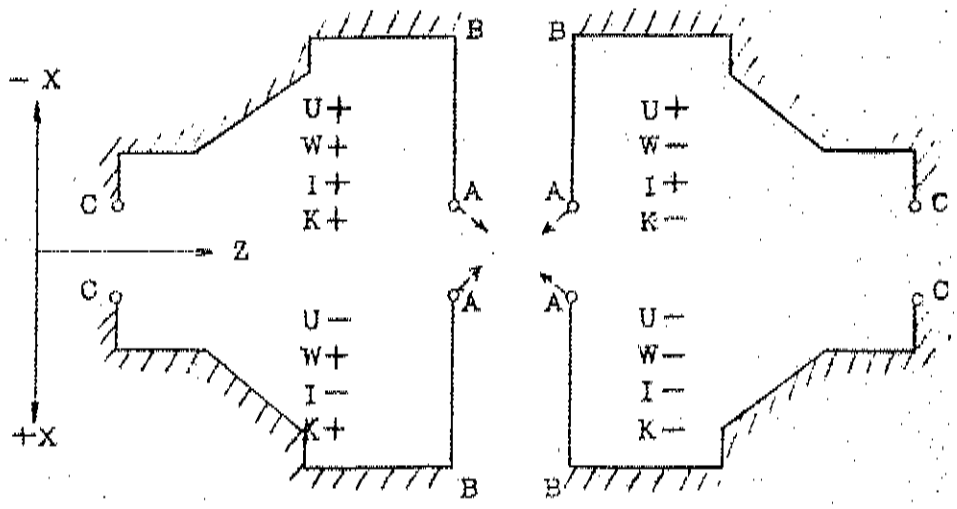
6) G71 starts the cycle from point A and finishes at point A. Therefore before executing G71 function, the cycle should be returned to point A.

7) The cutting shape specified with G71 can be consider four patterns of internal or external cutting respectively.

Signs such as U, W, I and K should be specified properly as shown in the following figure. (See the next page)



(a) TURNING (OUTSIDE)



(b) TURNING (INSIDE)

G71 CYCLE PATTERNS

3.8.3 Finishing (G70)

1) After executing G71 cycle, finishing cycle can be specified by the code G70 as described at the end of previous example.

2) The G72 and G73 cycles described later can also execute the finishing by the code G70.

3) Command format

G70 P$\overset{\alpha}{\downarrow}$. Q$\overset{\beta}{\downarrow}$. *

where

P--- Sequence number (α) for finishing cycle start

Q--- Sequence number (β) for finishing cycle end

4) Finishing cycle G70 should be followed after executing the program Nα to Nβ in the codes such as G71, G72 and G73.

5) Unless the reset operation or M02 command is executed, the program Nα to Nβ is still retained. Therefore G70 is not necessary to command just after ending G71, G72, or G73. Finishing cycle G70 can execute even after shifting the tool bite for rough cutting into one for finishing.

6) The cutting is specified by the F and S codes in the block Nα to Nβ.

7) G70 also starts the cycle from point A and returns to point A consequently, after executing G70, the next block should be programmed to start with point A.

8) When starting G70 command, the tip nose radius compensation is executed for the mode G41 to G44, but is cancelled for the mode G40.

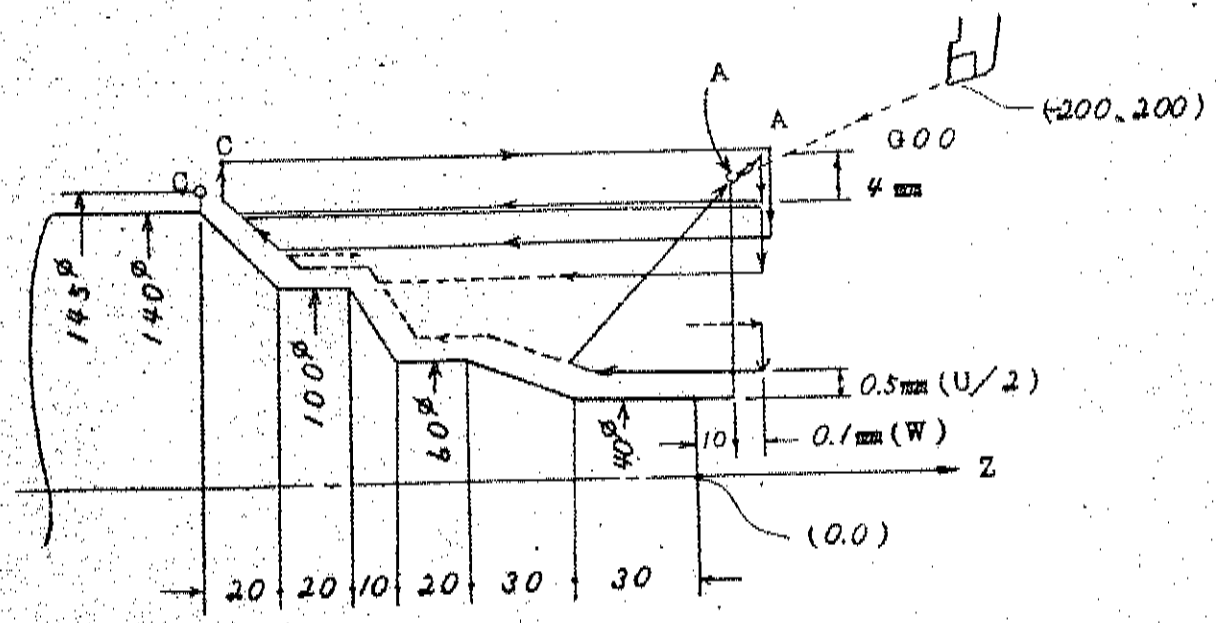
8) Program examples

ex. 1) Where I=0, K=0. (Without tip nose radius compensation)

```
N001 G50 X-20000 Z20000  *
N002 G00 T-0101880 M03 *
N003     X-14500 Z1000  *
N004 G71 P005 Q012 U-100 W10  D400 F30 S90 *
N005 G00 X-4000 F15 S120 *
N006 G01          Z-3000  *
N007     X-6000   W-3000  *
N008              W-2000  *
N009     X-10000  W-1000  *
N010              W-2000  *
N011     X-14000  W-2000  *
N012     X-14500          *
(N013 G70 P005 Q012 *)
```

G71 Cutting cycle (8 Blocks)

Finishing cycle



ex. 2) Where I=0, K=0 (with tip nose radius compensation)

```
N001 G50 X-20000 Z20000  *
N002 G00 T-0101 S80 M03 *
N003 G41 *
N004 G00 X-14500 Z1000  *
N005 G71 P006 Q013 U-100 W10  I-200 K200
                          D400  F30 S90 *
N006 G00 X-4000 F15 S120 *
  .
  .
  .
N013 (G01) X-14500 *
N014 G00 X-20000 Z20000  T0100*
N015 T-0202*
N016      X-14500 Z1000  *
N017 G70 P006  Q013 *
N018 G40 *
```

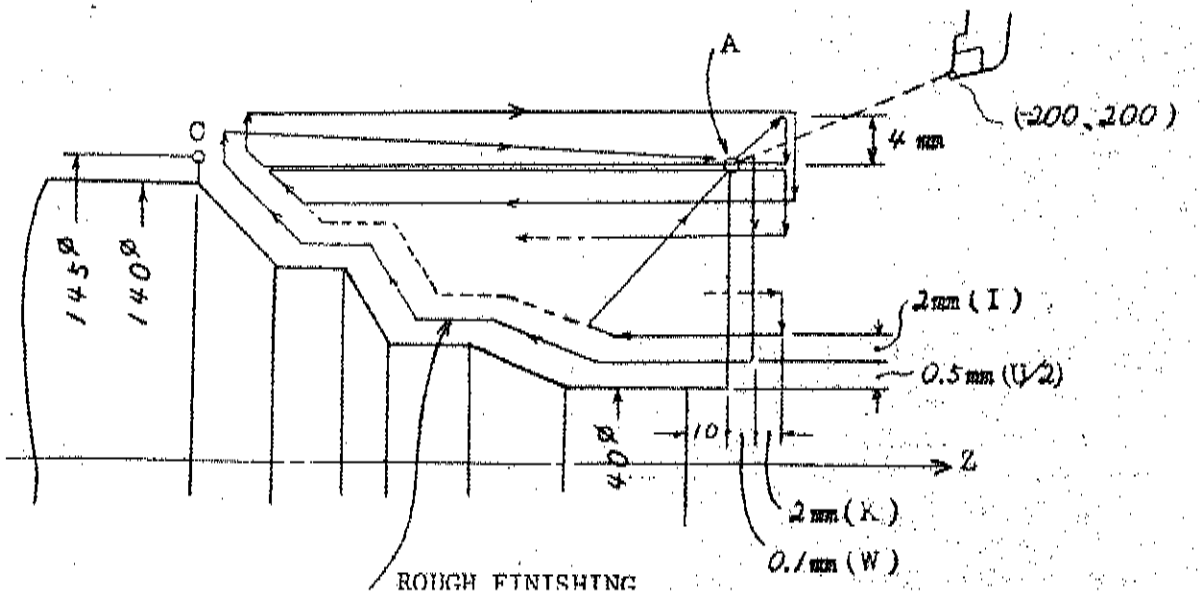
G71 Cutting cycle

(8 Blocks)

Same program with example 1

← For the change of tool T0202

Finishing cycle



3.8.4 Stock Removal In Facing (G72)

1) The code G72 can command the stock removal in facing.

2) The cutting of G71 cycle should be made parallel to the Z axis, but the cutting of G72 cycle should be made parallel to the X axis. This cycle is just the same as G71 except the above mentioned.

3) Command format

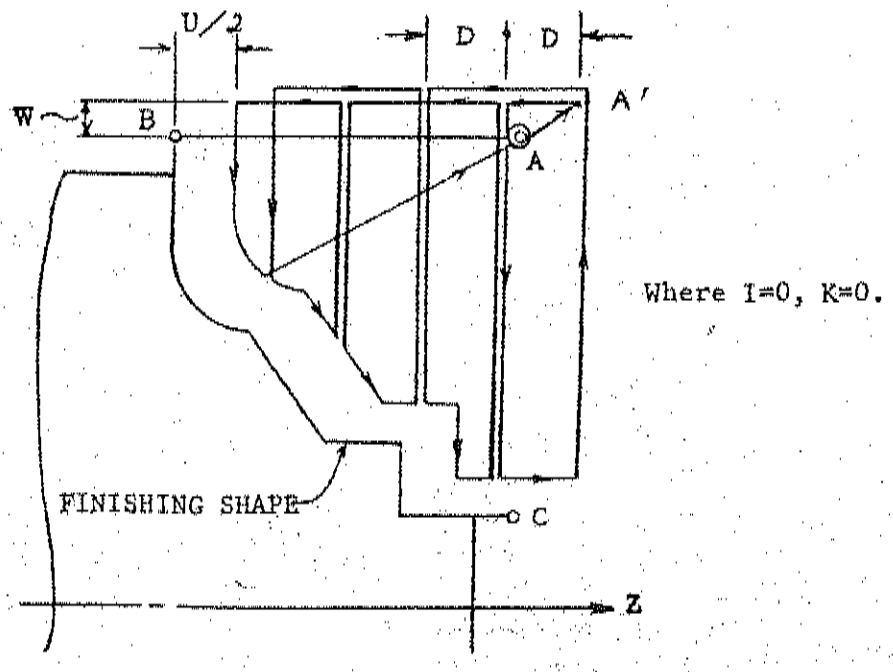
```
G72··· P(α)··· Q(β)··· U··· W··· I··· K··· D··· F··· S··· *
Nα·····  *
  ·····  *
       F ···  *
       S ···  *
Nβ·····  *
```

Program for finishing
Tool path A → B → C

where

D ------ Depth of cut in Z axis
Designation without sign.

Others are identified with G71 command



Where I=0, K=0.

3.8.5 Pattern Repeating (G73)

1) The code G73 can command the fixed cutting pattern by shifting the position of tool little by little. This function is available for the cutting of previously formed forging and cast iron.

2) Command format

G73 ... P(α) ... Q(β) ... U ... W ... I ... K ... D ... F ... S ... *

Nα ..... *
F ... *
S ... *
... *
Nβ ..... *

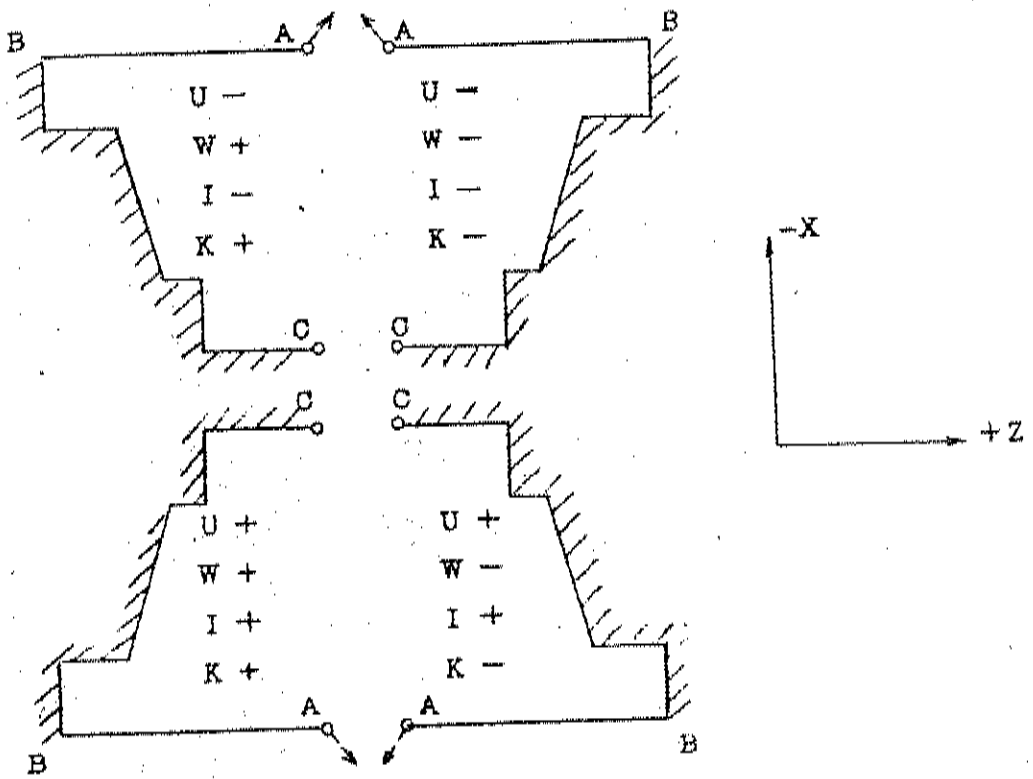
Program for finishing
Tool path: A → B → C

where

- P - Sequence number (α) for cycle start
- Q - Sequence number (β) for cycle finish
- U - Finishing allowance in X axis direction: diameter designation
- W - Finishing allowance in Z axis
- I - Total displacement in the direction of X axis: radius designation
- K - Total displacement in the direction of X axis
- D - Each moved displacement is determined by dividing I or K into (D-1) equal parts ie: the cutting is made D times. Designation without sign.
- F, S - Practical feedrate or command of this cycle. Without this code, the preceding specified code F or S becomes effective.

Note) The code F or S specified in the program Nα to Nβ becomes effective for finishing cycle G70.

4) G72 is provided with the following 4 different patterns. The signs such as U, W, I, and K should be specified properly as shown below.



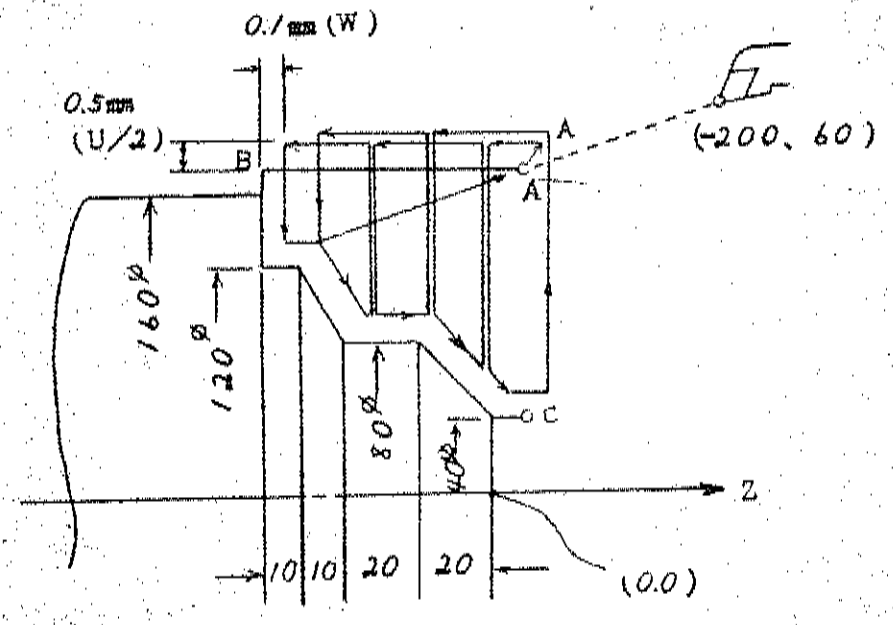
5) G72 starts the cycle from point A and returns to input A.

6) When I=0 and K=0, tip nose radius compensation is not executed. When I≠0, K≠0 and the mode G41 to G44, tip nose radius compensation is executed for the G72 cycle.

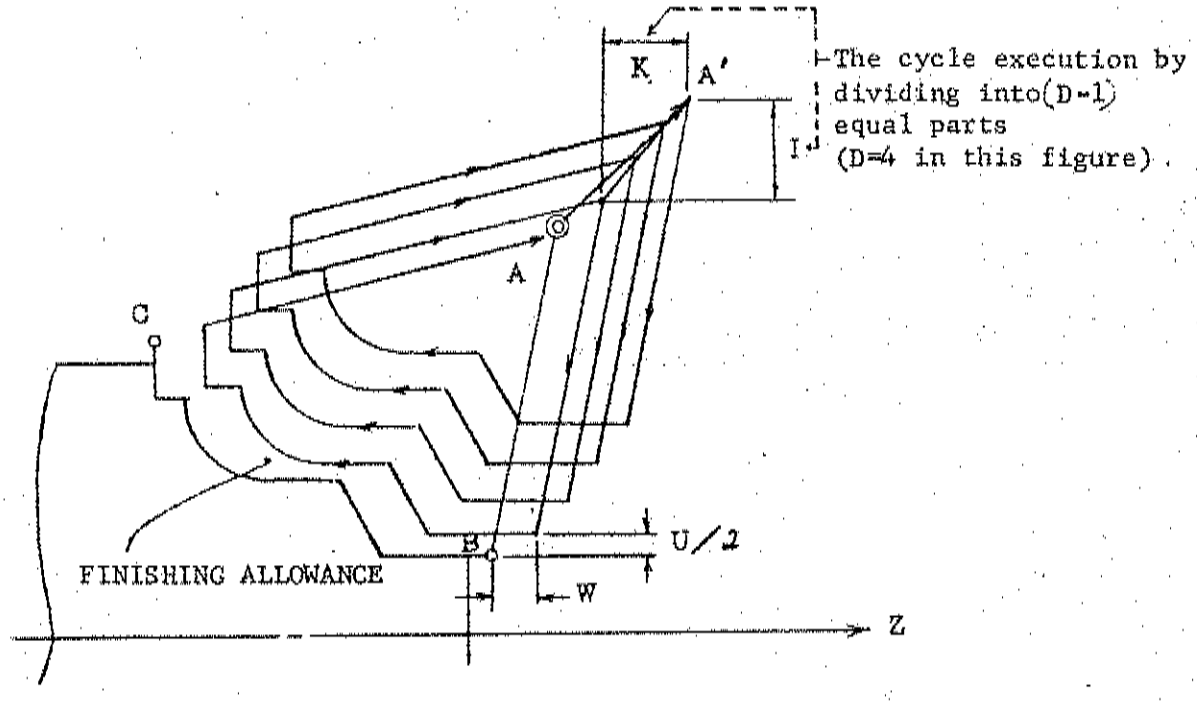
7) Program example

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

```
N001 G50 X-20000 Z6000  *
N002 G00 T0101 S80 M03 *
N003     X-17000 Z500  *
N004 G72 P005 Q011 U-100 W10  10 K0
                    D400  F30 S80 *
N005 G00           Z-6000 F15 S120*
N006 G01 X-12000 *                          G72 cycle
N007             Z-5000  *
N008     X-8000  Z-4000  *
N009             Z-2000  *
N010     X-4000  Z    0  *
N011             Z  500  *
N012 G70 P005 Q011 * ..... Finishing cycle
```



3) The cycle completes without cutting finishing allowance (U/2, W) as shown in the figure below.



4) If the tip nose radius compensation is specified during the G73 cycle, it becomes effective throughout the whole cycle.

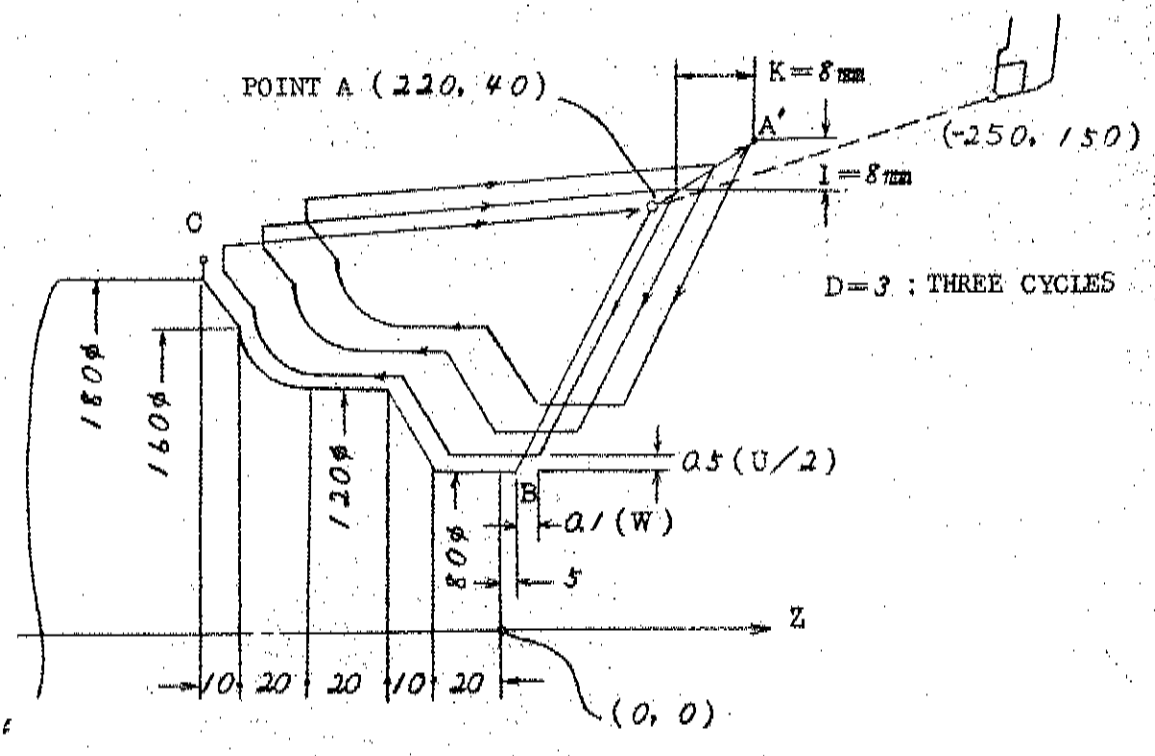
5) G73 starts the cycle from point A and finishes at point A.

6) Four cutting patterns can be considered about G73 cycle as well as G71 cycle. This signs such as U, W, I and K should be properly specified.

7) Program example

```
N001 G50 X-25000 Z1500  *
N002 G00 T-0101 S80 M03 *
N003     X-22000 Z4000  *
N004 G73 P005 Q011 U-100 W10  I-800 K800
                          D3 F30 S80 *
N005 G00 X-8000 Z500  S120 *
N006 G01          Z-2000  F15 *
N007     X-12000 W-1000  *
N008              W-2000  *
N009 G03 X-16000 W-2000  I-2000 *
N010 G01 X-18000 W-1000  *
N011     X-19000 *
N012 G70 P005 Q011 *  ..... Finishing cycle
```

G73 cycle



3.8.6 Peck Drilling In Z axis (G74)

1) The code G74 can command the peck drilling in Z axis. This cycle in also available for the turning (outaide) while breaking tips.

2) Command format

G74 ... X(U) ... Z(W) ... I ... K ... D ... F ... *

where

X - X coordinate of point C

U - Incremental amount from A to C (Diameter difference)

Z - Z coordinate of point B

W - Incremental amount from A to B

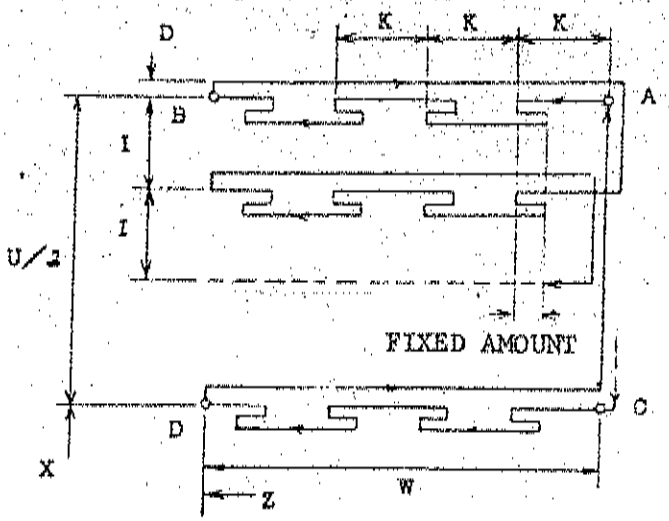
I - Displacement in X-axis (Designation without sign)

K - Depth of cut in Z direction (Designation without sign)

D - Retract amount of the tool at the cutting bottom. Designation without sign. When there is no D, D is assumed to be zero.

F - Feedrate designation

Fixed amount - The fixed value can be set as incremental unit 0.001 mm by parameter setting



3) Tool path of the cycle (G74): A → B → C → D → A

4) If X(U) or I or D is omitted, only the operation on Z axis results, to be used for drilling.

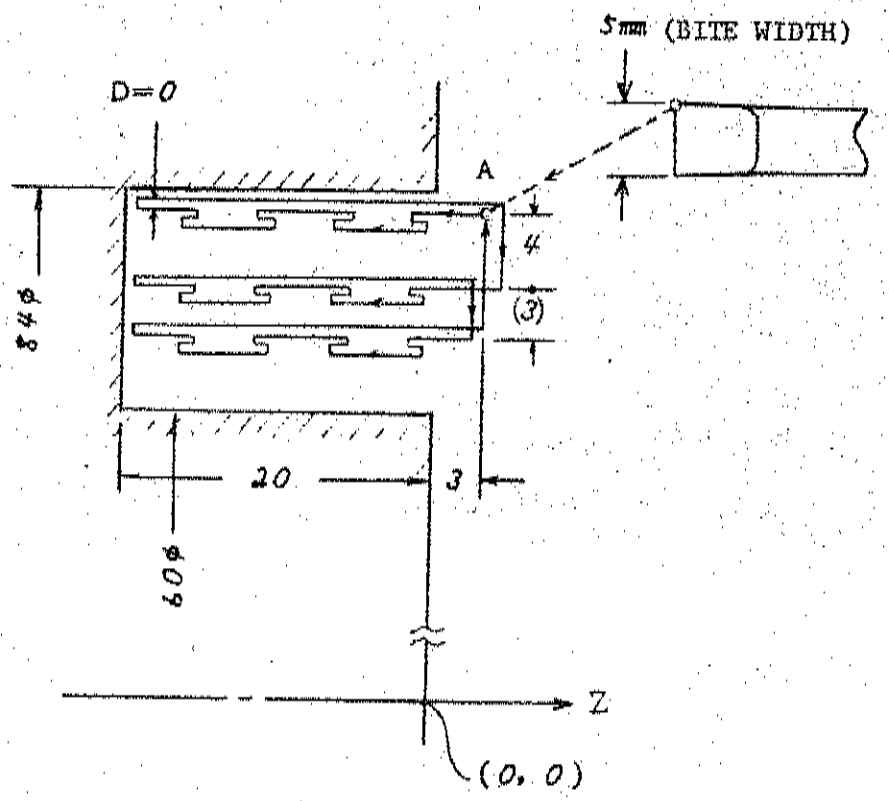
5) When a retract is made by D at the cutting bottom, the next cutting is made by (D + I)

6) Finishing depth of cut in Z direction and finishing movement amount

7) Program example

G00 X-8400 Z300 *

G74 U 1400 Z-2000 I400 K500 D0 F30 *



3.8.7 Grooving in X axis (G75)

1) The code G75 can command the grooving in X axis. This function is available for the grooving while breaking tips.

2) This is equivalent to G74 in which X is replaced with Z.

3) Command format

G75 X(U) ... Z(W) ... I ... K ... D ... F ... *

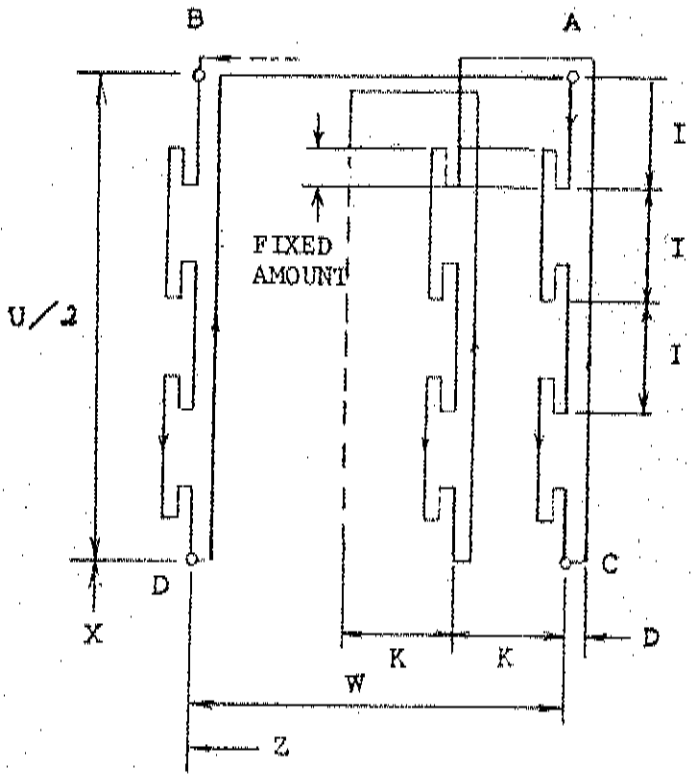
where

I - Depth of cut in X-axis

K - Displacement in Z-axis (Designation without sign)

Other designations are equal to G74.

4) Tool path of cycle: A → C → B → D → A



3.8.8 Thread Cutting (G76)

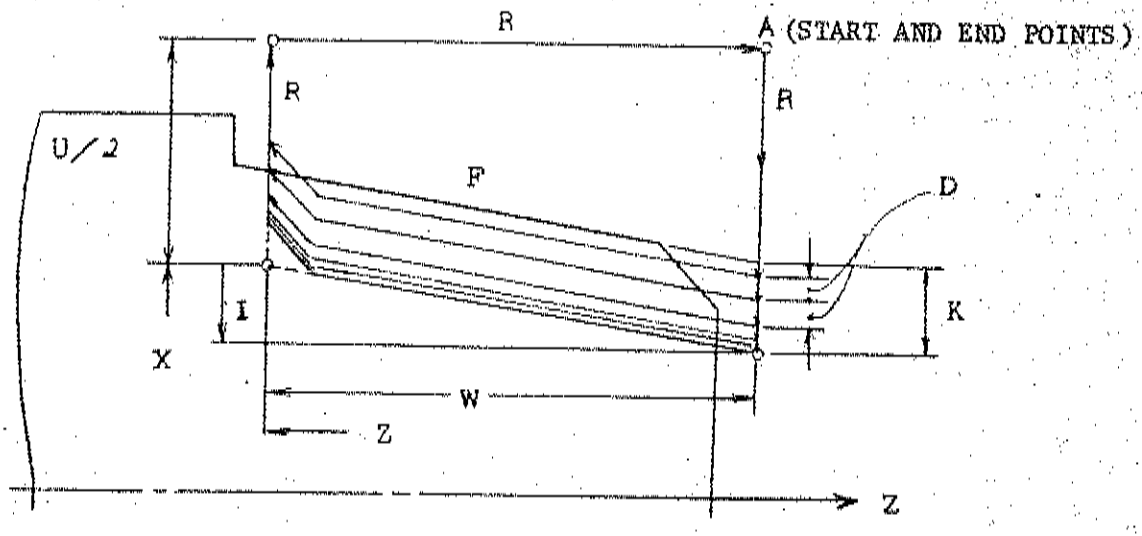
1) The code G76 can command straight or taper thread cutting.

2) Command format

G76 X(U)·· Z(W)··· I··· K··· D··· F(E)··· * ——precision screw

where

- X(U), Z(W) } Terminating portion coordinates of thread. (or movement amount of tool)
- I - Difference of thread radius (X-axis derection) If I=0, ordinary straight thread cutting can be made.
- K - Height of thread (X-axis direction, without sign)
- D - Depth of cut (without sign)
- F(E) - Lead of thread



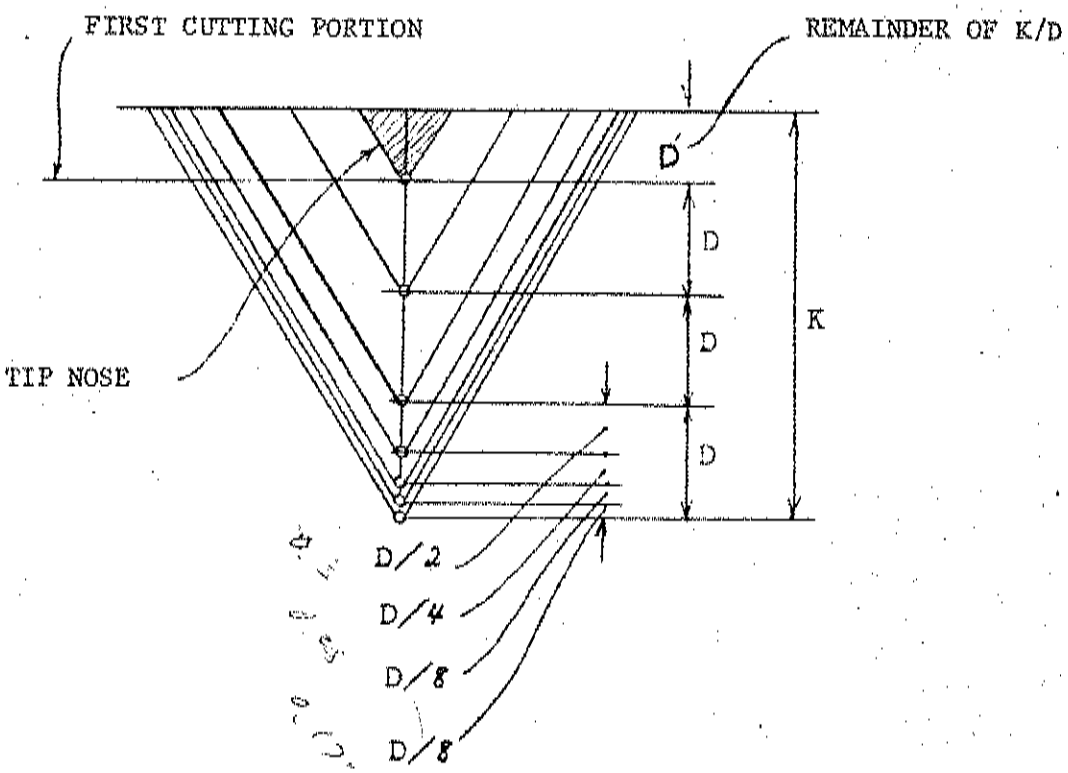
3) If M23 (Chamfering) is specified previously, chamfering is made at angle 45° from the 0.8 lead ahead of terminating portion of thread.

4) The fixed depth of cut is specified by D.

The final D is divided into the four cuts, consequently, the following equation is obtained:

$$\frac{D}{2} + \frac{D}{4} + \frac{D}{8} + \frac{D}{8} = D$$

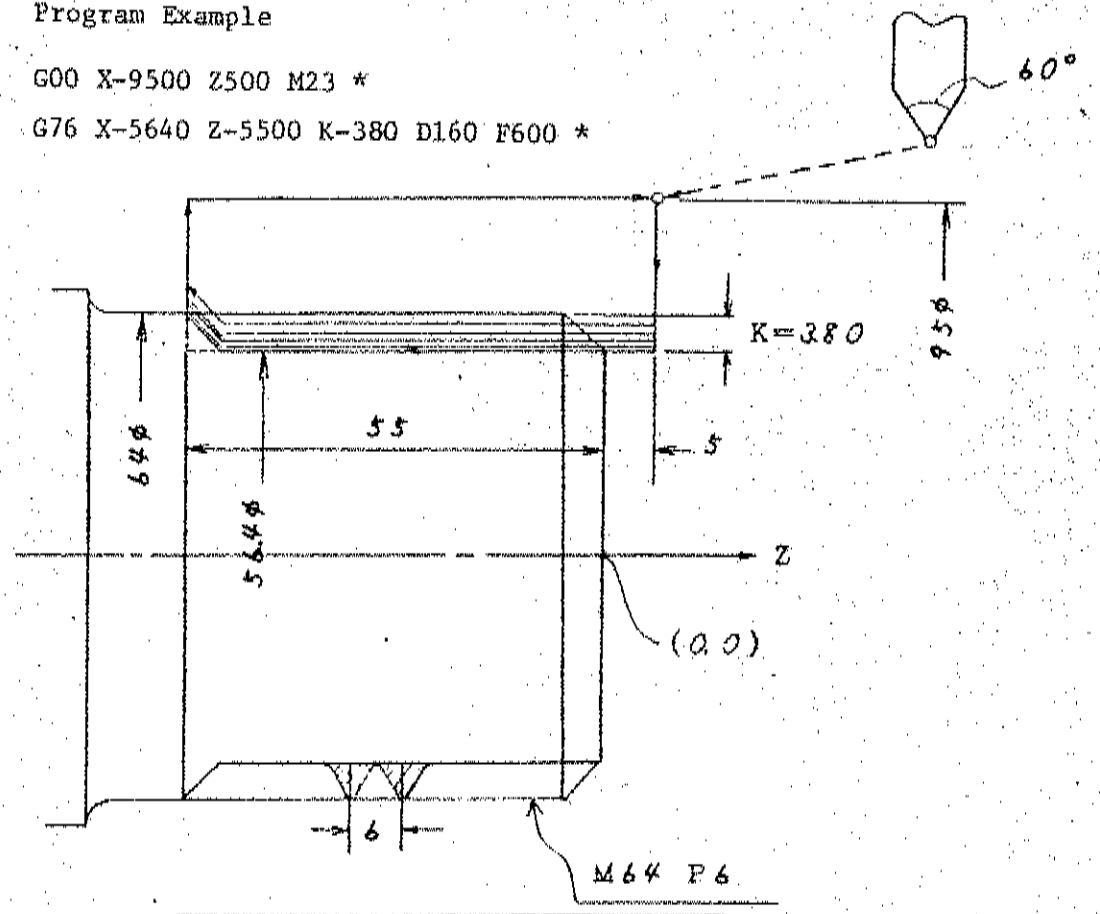
The remainder D' after dividing K height of thread by D becomes the first depth of cut.



(D/8 can be replaced with D/16 by setting...)

5) Program Example

```
G00 X-9500 Z500 M23 *
G76 X-5640 Z-5500 K-380 D160 F600 *
```



The Number of Cut	Depth of Cut
1	0.6 mm
2	1.6
3	0.8
4	0.4
5	0.2
6	0.2
Total	3.8 mm

3.9 Constant surface speed control (G96, G97)

G96: performs constant surface speed control
G97: does not perform constant surface speed control

(1) The spindle-speed range is switched over as necessary.

M41: LOW range
M42: HIGH range

(a) It is M41 when power has been turned on.
(b) Once M41 has been issued, it remains effective unless M42 is issued. Once M42 has been issued, it is held in memory unless M41 is issued.
(c) Issue the spindle-speed range selection command, M41 or M42, in a single block before the S-command.

```
M41
G96S120M03
  :
```

(2) Determine the maximum speed of the main spindle as a parameter.

The parameter number for M41 (LOW range) is 86.
The parameter number for M42 (HIGH range) is 87.

(3) With G50 S0000 , it is possible to clamp the maximum speed of the spindle. If the S-command does not follow G50, the maximum speed will be clamped at that determined by the parameter.

3.9.1 G96

Together with G96, S-command followed by 4-digit figure determines the constant surface speed (m/min) in cutting. The spindle speed varies

with the radius of the workpiece to keep the surface speed constant.

```
      :
   G96 S150          S = 150 m/min
     * S100          S = 100 m/min
```

3.9.2 G97

Use G97 if constant surface speed control is not desired. The figure following code S determines the spindle speed (rpm) directly.

To advance the sequence, SF signal is delivered about 100 ms after delivery of S-binary signal. When the spindle motor has reached a commanded speed, turn on FIN signal. Then SF will be turned off. When SF has been turned off, turn off FIN signal and then it will advance to the next block. It is G97 at turning on of power or reset.

```
   G97 S500          S = 500 (rpm)
      :
      :
     S80            S = 80 (rpm)
      :
```

3.9.3 Processing of constant surface speed control

(1) Output data for G96

$$Sn = \frac{4095}{Np} \times \frac{V}{\pi \cdot U}$$

where V: Constant surface speed commanded by S-code (m/min)
U: diameter in x-direction (m)
No: maximum spindle speed set with parameter (rpm)

(2) Output data for G97

$$Sn = \frac{4095}{Np} \times S$$

where S: Spindle speed commanded by S-code (rpm)
Np: maximum main spindle speed set by

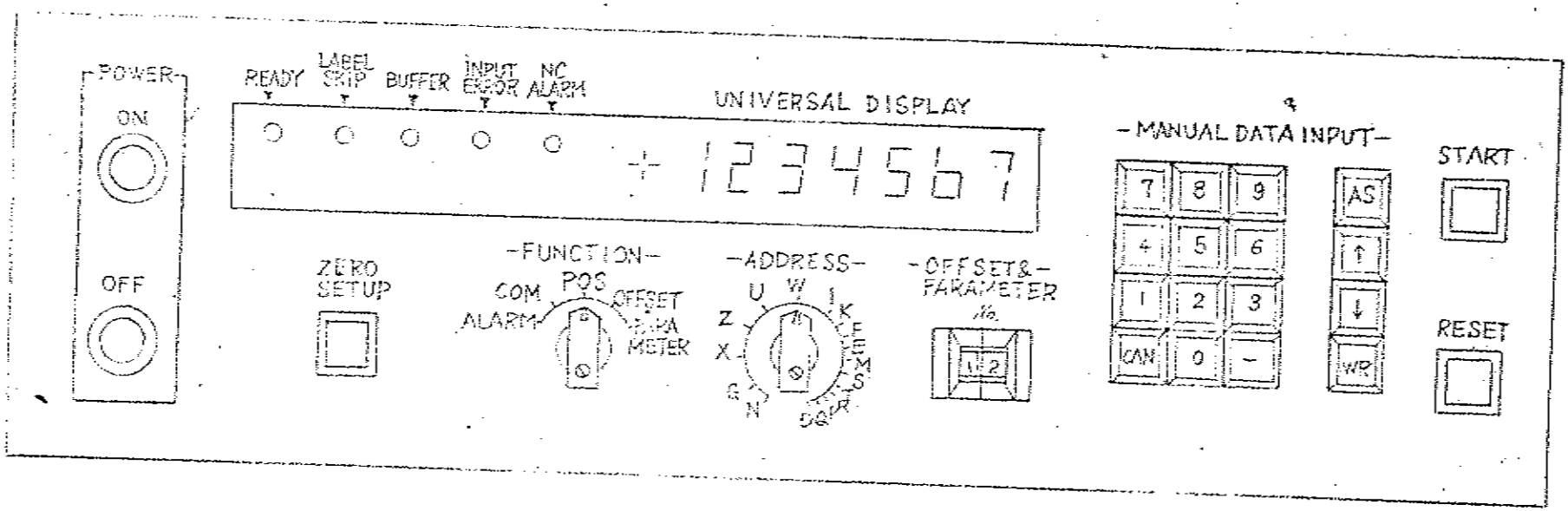


Fig. A.1 Control panel

4. OPERATION

4.1 Control Panel

4.1.1 Pushbuttons and select switches on the control panel

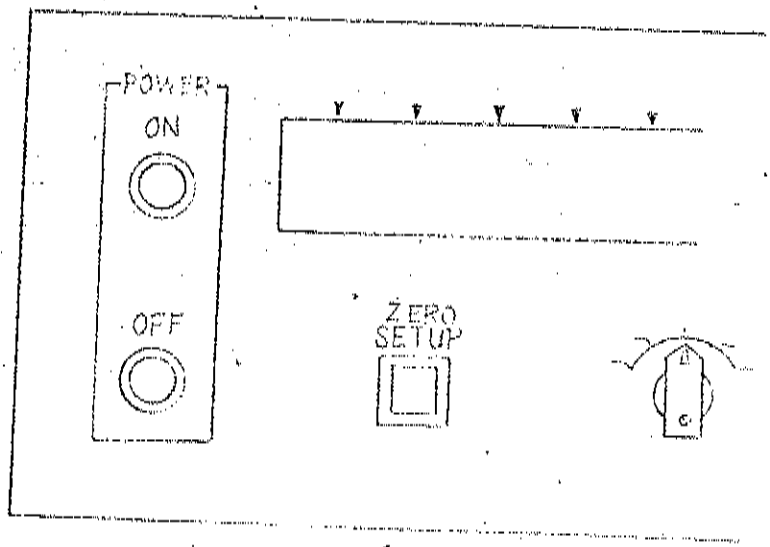
(1) POWER ON button

Depress it to turn on the NC system.

(2) POWER OFF button

Depress it to turn off the NC system.

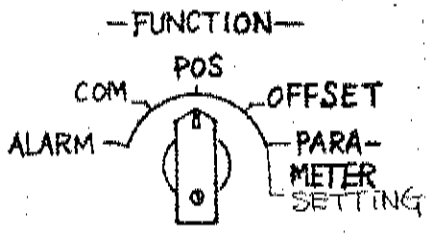
(3) ZERO SETUP button (ineffective for this NC system)



(4) FUNCTION selector

This selects one out of five functions for the operation of the display and MDI.

◈ SETTING:
Select this position to displ
& write such setting data
rapid thayerse speed Hi-l
Switching on Moo DELETE



FUNCTION selector

ALARM: Select this position to display an alarm code. Four decimal points appear on the display.

COM: Select this position to display or write a command value.

POS: Select this position to diaplay the current tool position.

OFFSET: Select this position to display or write the value of cutter compensation.

PARAMETER: Select this position to display or write such parameters as backlash compensation and rapid traverse speed.

(5) ADDRESS selector

This selects an address character to display or write internal data using the control panel.

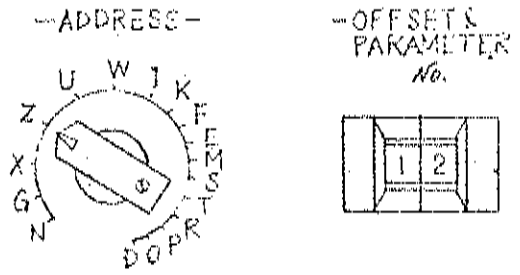
(6) OFFSET & PARAMETER NO. switch

This thumbwheel switch is used in the following cases.

1) To display or write the values of tool offset and cutter

compensation, it designate a tool offset number.

2) To display or write a parameter, it indicates a parameter number.



ADDRESS selector and OFFSET & PARAMETER NO switch

(7) DATA keys

Twelve keys, 0 through 9, - (minus) and CAN (cancel), are used for manual data input.

(8) WR key (WRITE)

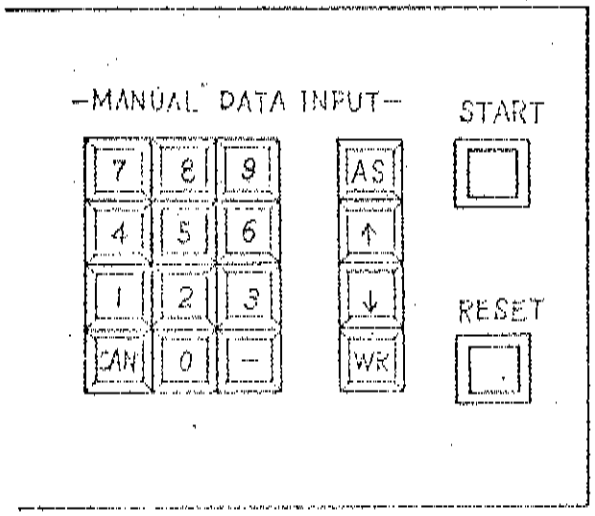
Depress this key to store the data input by means of the DATA keys in the MDI mode.

(9) START button

Depress it to start the system. It operates only in the MDI mode.

(10) RESET button

Depress it to reset the NC system. The internal state of the NC system is reset but the current tool position and sequence number is reinitialized. The values of tool offset and cutter compensation, parameter and the contents of program memory remain unchanged.



(11) SYSTEM NO. switch

This switch, out of sight at the back of the front door, is not operated normally. It is fixed at 0 during normal operation. Its setting is as follows.

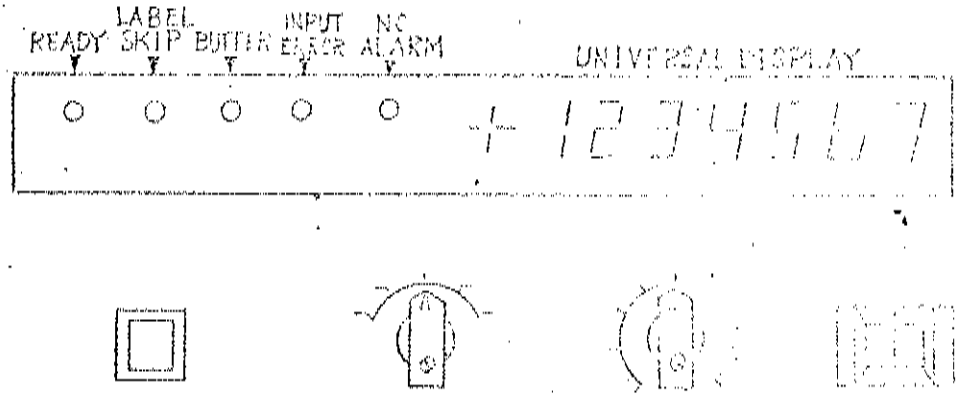
Setting	Purpose
7	MAKE: to totalize the system program stored
8	LOAD: to store the system program in own memory
9	CHECK: to collate the system program stored with the system tape
0	SYSTEM: This is the normal setting. It cannot write parameters.
1	PARAMETER: to write parameters. It also allows normal operation. Reset it to 0 when parameters have been written.

4.1.2 Display

The indicate lamps (LEDs) and display units mounted in the control panel are:

1) READY lamp
2) LABEL SKIP lamp
3) BUFFER lamp
4) INPUT ERROR lamp
5) NC ALARM lamp
6) signed 7-digit digital display

Each of them is described below.



(1) READY lamp

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

(2) LABEL SKIP lamp

It lights up when the label skip function

that makes tape setting easy by disregarding information until the first EOB is encountered is effective. It goes off when EOB has been read from a tape.

It goes on when

1) power has been turned on, or

2) the control system has been reset.

This display is related only to tape information and not to MDI operation.

(3) BUFFER lamp

It lights up when data of the next block are held in the buffer storage. It goes off when the buffer has been evacuated by cycle start or reset operation.

Command position written through the MDI unit to the buffer causes the display to go on.

(4) INPUT ERROR lamp

It goes on when an error is detected in input information. Operation, if it is automatic, is suspended immediately when the current block has been executed, and cycle start is then inhibited. Possible causes for the lamp to go on are as follows.

1) Mispunched information on paper tape

2) Use of wrong characters

3) Wrong tape format

4) Misreading of paper tape reader

5) Contents of memory destroyed

Set the FUNCTION selector to ALARM to obtain the detailed information of error in alarm code.

Take the following measures according to the error information.

1) Correct NC tape.
2) Clean NC tape.
3) Clean the sensor of the paper tape reader.
4) Check the contents of memory.

The lamp goes off when the system has been reset.

(5) NC ALARM lamp

It goes on when an error other than the above-noted input error has been detected in the NC system. If the system is in automatic operation, it stops immediately or when the end of a block is encountered, depending on the error, and cycle start is then inhibited.

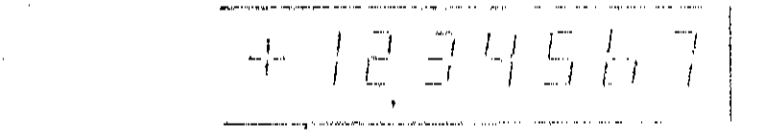
As in the case of input error, it is possible to obtain detailed information of the error by means of alarm code.

See "3.2 Alarm code display" for the measures to take.

(6) UNIVERSAL DISPLAY

This is a signed 7-digit display used when displaying and writing various data.

UNIVERSAL DISPLAY



4.1.3 Operation detailed

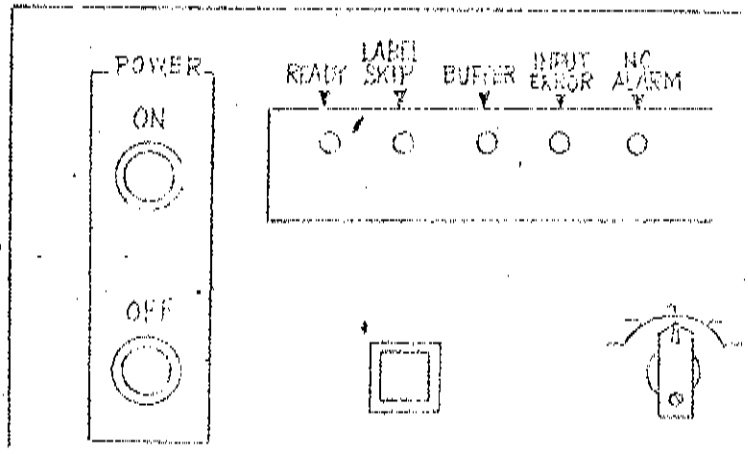
(1) Power on/off operation

(a) Turning on power

1) Depress the POWER ON button to turn on control power.
2) After the initial timer has been reset in about 2 seconds, depress the POWER ON button again to turn on servo power.
3) The READY lamp goes on when the system is ready.

(b) Turning off power

1) Depress the POWER OFF button.



Note: Power can be turned off simply by depressing the POWER OFF button. But the turn-off operation we recommend in order to allow the system to operate stably over a long period is as follows. Depress the EMERGENCY STOP button to cut off servo power before depressing the POWER OFF button.

(2) Alarm code display

The system is always performing diagnosis even during machining. When an

error is detected, the INPUT ERROR or NC ALARM lamp lights and the system stops operation. The corresponding alarm code will be displayed when the FUNCTION selector on the control panel is set to ALARM.

However, CPU error (alarm code: 81) and memory check error (alarm code: 82) are displayed at whatever position the FUNCTION selector may be set. The alarm code is distinguishable from other data because four decimal points are added. Alarm codes are described below together with typical causes and countermeasures.

Alarm code "11": excessive temperature rise in the panel

Possible causes

a) Ambient temperature exceeding 45° C

b) Cooling system including fan motor in trouble

Correction

a) Reduce ambient temperature and reset system.

b) Contact service personnel

Alarm code "12": tape horizontal parity error

Possible causes

a) Even holes perforated with [illegible] code

b) Blank detected with [illegible] code

c) Odd holes perforated with [illegible] code

Correction

a) Correct tape.

b) Clean paper tape reader sensor.

Then reset the system.

Alarm code "13": tape vertical parity error (option)

Possible cause

a) Number of characters in block containing EOB is odd when TV check is on.

Correction

a) Correct tape.

b) Clean paper tape reader sensor.

Then reset the system. If TV check is not required, turn it off.

Alarm code "14": format error

Possible causes

a) Illegal characters used.

b) Illegal G-command used.

c) Number of significant characters in block containing EOB exceeds 64.

Correction

a) Correct tape.

b) Clean paper tape reader sensor.

Then reset the system.

Alarm code "15": data error

Possible causes

a) Feedrate not given for cutting.

b) G92 (thread cutting cycle) requests stroke smaller than 0.8 pitch when chamfering is on.

Correction

a) Correct program.

Then reset the system.

Alarm code "16": offset error

Possible cause

a) Values of tool offset and cutter compensation destroyed.

Correction

a) Contact service personnel, though it is enough to rewrite the values of tool offset and cutter compensation and reset the system.

Alarm code "17": parameter error

Possible cause

a) parameter destroyed

Correction

a) Contact service personnel though it is enough to rewrite parameter and reset the system.

Alarm code "18": program memory error

Possible cause

a) Contents of program memory used for memory operation are destroyed.

Correction

a) Contact service personnel though it is enough to rewrite program and reset the system.

Alarm code "21": overtravel

Possible cause

a) Movable part of mechanism has come to the end of its stroke.

Correction

a) Move it back by manual operation and reset the system.

Alarm code "27": positioning error

Possible cause

a) Positioning error due to trouble of servo unit or mechanism.

Correction

a) Check servo unit or mechanism and reset the system.

Alarm code "26": system unready

Possible causes

a) System is waiting for MRD (machine ready) signal after power has been turned on.

b) MRD signal has been lost due to wrong machine control sequence.

Correction

a) When power has just been turned on, system becomes ready automatically on receiving MRD signal.

b) Check machine control sequence to recover MRD signal and reset the system.

Alarm code "31": servo power dead

Possible cause

a) Servo power has not yet been turned on.

Correction

a) Depress POWER ON button.

Alarm code "32": control unit not ready

Possible cause

a) Control unit is not yet ready for turning on servo power.

Correction

a) When the unit is ready, alarm code turns to "31" automatically.

b) Contact service personnel if alarm code does not switch to "31".

Alarm code "33": emergency stop

Possible cause

a) Emergency stop button depressed.

Correction

a) Resetting the system, depress POWER ON button to turn on servo power. Alarm code turns to "31" when reset.

Alarm code "34": servo error

Possible cause

a) Excessive velocity error due to defective servo system or mechanism.

Correction

a) If alarm code has turned to "31" when resetting the system, turn on servo power.

b) Check servo system or mechanism.

Alarm code "35": overload

Possible causes

a) Thermal trip due to motor overload.

b) Fuse blown due to motor overload.

c) Excessive temperature rise in drive unit

Correction

a) Remove cause and reset the system. Then alarm code turns to "31" and turning on servo power becomes possible.

b) Check program.

c) Check servo system or mechanism.

Alarm code "51": M, S or T function in execution (status display)

System is executing M, S or T function and waiting for FIN signal to be sent from machine.

Alarm code "52": pulse distribution (status display)

This indicates the sequence status that command pulse is being processed. It is displayed even during dwell time but not displayed during manual operation.

Alarm code "53": M-, S- or T-function being executed and pulse being processed (status display)

This is the compound status of codes "51" and "52".

Alarm code "54": tape being read (status display)

This indicates that paper tape is being read.

Alarm code "55": M-, S- or T-function in execution and tape being read (status display)

This is the compound status of codes "51" and "54".

Alarm code "56": pulse being processed and tape being read (status display)

This is the compound status of codes "52" and "54".

Alarm code "57": M-, S- or T-function being executed and pulse being processed and tape being read (status display)

This is the compound status of codes "51", "52" and "54".

Alarm code "81": CPU error

Possible cause

a) NC system CPU in trouble and not capable of processing.

Correction

a) Contact service personnel.

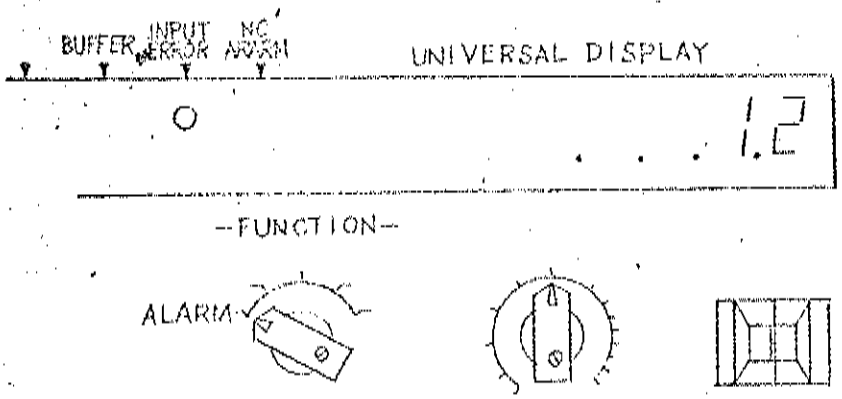
Alarm code "82": memory error

Possible cause

a) System program stored in built-in memory destroyed.

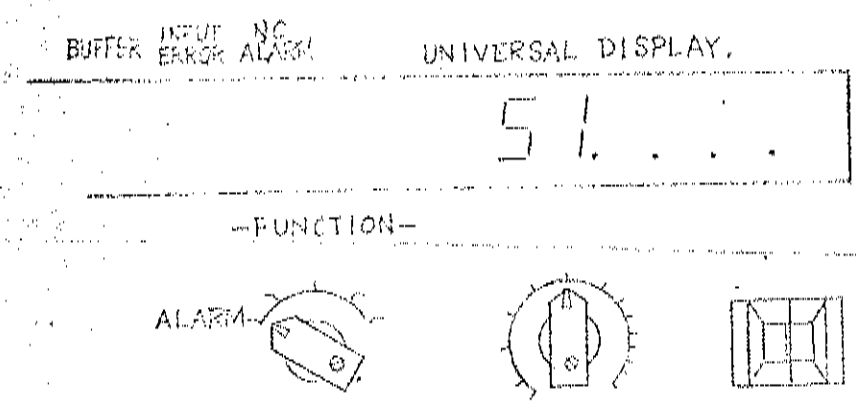
Correction

a) Contact service personnel.



Example of alarm code display

"12" ... tape horizontal parity (input error)



Example of status code display

"51" ... M-, S- or T-function being executed

List of alarm codes

alarm code	cause	lamp display: INPUT ERROR	lamp display: NC ALARM	result
..... 1.1	excessive temperature rise in panel		o	stops at end of block
..... 1.2	tape horizontal parity error	o		
..... 1.3	tape vertical parity error	o		
..... 1.4	format error	o		
..... 1.5	data error	o		
..... 1.6	offset error	o		
..... 1.7	parameter error	o		
..... 1.8	program memory error	o		
..... 2.1	overtravel		o	immediately decelerates and stops
..... 2.7	positioning error		o	
..... 2.8	system unready		o	
..... 3.1	servo power dead		o	cuts off servo power and emergency-stops
..... 3.2	control unit unready		o	
..... 3.3	emergency stop		o	
..... 3.4	servo error		o	
..... 3.5	overload		o	
..... 8.1	CPU error		o	cuts off servo power and emergency-stops: CPU comes to complete halt
..... 8.2	memory error		o	
..... 9.1	contents of tape and memory differ		o	at off line diagnostics
..... 9.2	tape misreading		o	

Note: If two or more alarm statuses occur at the same time, the larger or largest alarm code is displayed in preference to the others.

List of status codes

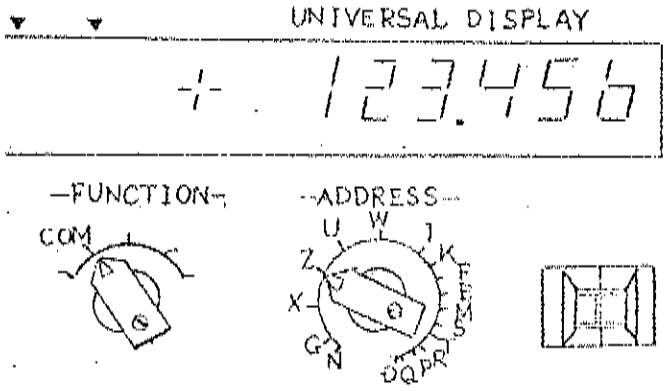
status code	status	remark	
51	M, S or T being executed		These codes simply indicate the status of a sequence.
52	pulse being processed (or in dwell)		
53	M, S or T being executed and pulse being processed	51+52	
54	tape being read		
55	M, S or T being executed and tape being read	51+54	
56	pulse being processed and tape being read	52+54	
57	M, S or T being executed, pulse being processed and tape being read	51+52+54	

(3) Command data display

Command data in a block can be displayed on the universal display in the following way.

1) Set the FUNCTION selector to COM.
2) Select an address character with the ADDRESS selector.

This operation is always possible independently of the operating mode. Contents of the register are displayed when the system is in operation (or at a temporary stop), and contents of the lock-ahead buffer when it is idle. For coordinate commands, the data displayed are modified with cutter compensation.



Example of command data display
(Z-axis)

(4) Writing command data

Command data of a block can be written by manual operation in the following way. It is possible only in the MDI mode and when the data written enter the buffer.

1) Set the system to the MDI mode.
2) Set the FUNCTION selector to COM.
3) Select an address character with the ADDRESS selector, Then command data already entered are displayed.
4) Input a figure through the DATA keyboard. Then the data just keyed in is displayed. To correct the data just keyed in, depress the CAN key and then key in the correct figure.
5) Depress the WR key. The data just keyed in is stored as a new command data. When cutter compensation is effective, the input data will be modified.
6) Repeat steps 3 through 5 until data in a block have been written.

7) Depress the START button and the input data are executed.

Note: During manual data input, the least significant digit of the input data lights on and off.

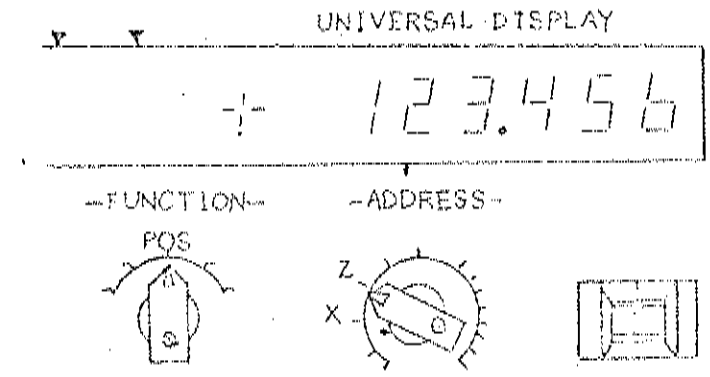
(5) Current position display

The current position of X- or Z-axis can be displayed on the universal display in the following way.

1) Set the FUNCTION selector to POS.
2) Set the ADDRESS selector to X or Z.

This operation is always possible independently of the operating mode, and the absolute coordinate of the X- or Z-axis is displayed in the coordinate system set up by G50.

If an address other than X or Z selected, nothing is displayed.



Example of current position display (Z-axis)

(6) Current position display in incremental mode

The incremental value given below of the X-or Z-axis can be displayed on the universal display in the following way.

1) Set the FUNCTION selector to POS.

2) Set the ADDRESS selector to U or W.

Then,

a) the difference between the command data and the current position will be displayed in automatic operation. It is zero if the current position coincides with the command data.

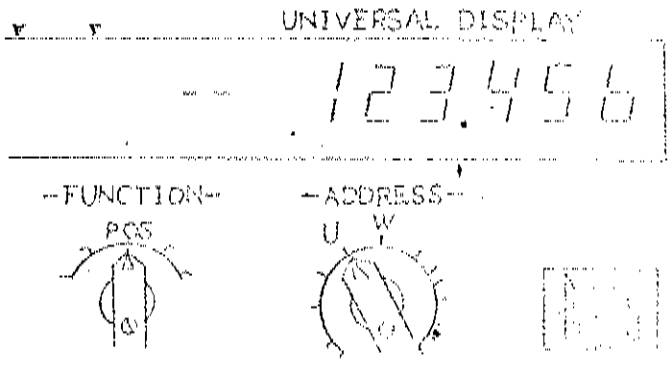
b) For manual operation, the value of travel will be displayed.

c) The result displayed is zero in the following cases.

. The command data coincides with the current position.

. The operating mode has just been switched from manual to automatic or vice versa.

. The system has just been reset.



Example of current position display in incremental mode (U)

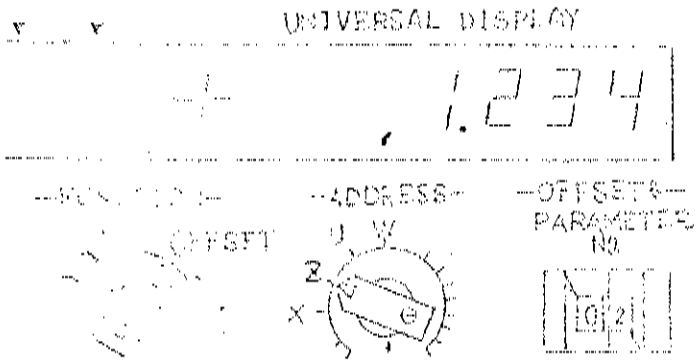
(7) Tool offset display

The value of tool offset along the X- or Z-axis can be displayed on the universal display in the following way.

1) Set the FUNCTION selector to OFFSET.
2) Set the tool offset number on the OFFSET & PARAMETER NO switch.
3) Select an axis with the ADDRESS selector.

This operation is always possible independently of the operating mode.

If an unused tool offset number has been set or the ADDRESS selector has been set to a position other than X, Z, U or W, nothing is displayed. The positions of X and U as well as Z and W give the same result.



Example of tool offset display
(tool offset number: 2, Z-axis)

(8) Writing the value of tool offset

Write the value of tool offset in the following way. It is always possible independently of the operating mode even during operation.

1) Set the FUNCTION selector to OFFSET.
2) Set a tool offset number on the OFFSET & PARAMETER NO switch.
3) Select an axis with the ADDRESS selector. Then the current value of tool offset will be displayed.
4) Key in a figure from the DATA keyboard. Then the data just keyed in will be displayed.
   To correct the data keyed in, depress the CAN key and then key in the correct one.
5) Depress the WR key.
   Then the figure just keyed in will replace the old one.
6) Repeat steps 3 through 5 for the other axis if necessary.
7) Select X or Z on the ADDRESS selector to input an absolute value. Select U or W to input an incremental value.

Note: While a data is being input, the least significant digit just keyed in will light on and off.

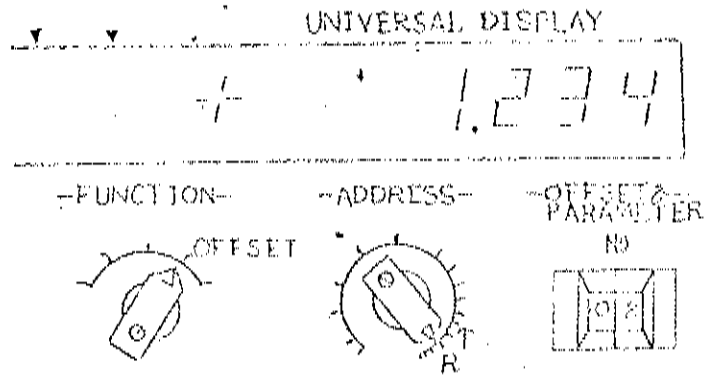
(9) Display of the value of compensation for tip nose radius (cutter compensation)

The value of cutter compensation can be displayed on the universal display in the following way.

1) Set the FUNCTION selector to OFFSET.

2) Set a tool offset number on the OFFSET & PARAMETER NO switch.
3) Set the ADDRESS selector to R.

This operation is always possible independently of the operating mode.



Example of the display of cutter compensation

tool offset number: 2

cutter compensation: R = +1.234 mm

(10) Writing the value of compensation for tip nose radius (cutter compensation)
Write in the radius of tool tip in the following way. This operation is always possible independently of the operating mode even during operation.

1) Set the FUNCTION selector to OFFSET.
2) Set a tool offset number on the OFFSET & PARAMETER NO switch.
3) Set the ADDRESS selector to R.
Then the current radius of tool tip already registered will be displayed.

4) Key in a figure from the DATA keyboard.
The data just keyed in will be displayed.
To correct the data just keyed in, depress the CAN key and key in the correct value.

5) Depress the WR key.
Then the data just keyed in will replace the old one.

Note: While the data is being input, the least significant digit just keyed in will light on and off.

(11) Parameter display

The value of any parameter can be displayed on the universal display in the following way. This operation is always possible independently of the operating mode.

1) Set the FUNCTION selector to PARAMETER.
2) Set a parameter number on the OFFSET & PARAMETER NO switch.
3) For parameters of 90 or greater, select an axis with the ADDRESS selector.

Refer to the list of parameters for the parameter number.
If an unused parameter number has been selected or an address character other than X and Z has been selected for a parameter number of 90 or larger, nothing will be displayed.

Parameter No.	function of parameter	input
74	fixed amount (G74)	give amount taking 1 = 0.001 [mm]
75	fixed amount (G75)	give amount taking 1 = 0.001 [mm]
76	Division number (G76)	
81	tape vertical parity check ON/OFF	0 : OFF 8 : ON
82	ISO/EIA CONVERSION	0 : EIA 8 : ISO
83	INCH/METRIC CONVERSION	0 : MM 8 : INCH
86	Low range maximum spindle speed (Constant surface speed control)	give the spindle speed (rpm) taking 1 = 1[rpm]
87	High range maximum spindle speed (Constant surface speed control)	
89	Compensation for tip nose radius at G00 ON/OFF	0 : OFF 8 : ON

EXAMPLE - 1

for parameter No. 94 (rapid traverse speed)

input data	5	10	15	225	300	450	600
speed	80	160	240	3600	4800	7200	9600 (mm/min)

EXAMPLE - 2

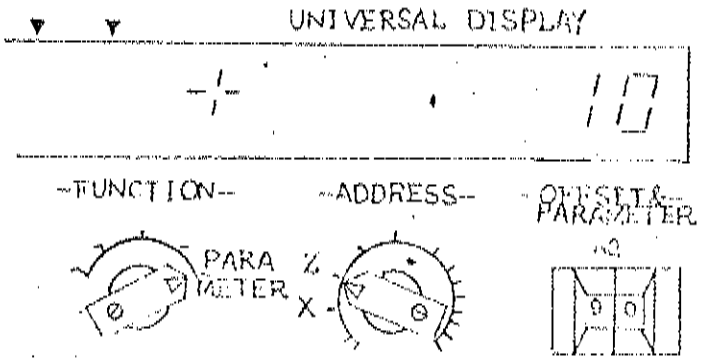
for parameter No. 95 (acceleration and deceleration time)

$$ta = \frac{V}{\frac{128}{0.03} \times n} \text{ (sec)}$$

n: input data
V: rapid traverse (mm/min)

V = 4800 mm/min

input data	1	2	3	4	5	6
ta	1.125	0.563	0.375	0.281	0.225	0.187 (sec)



Example of parameter display

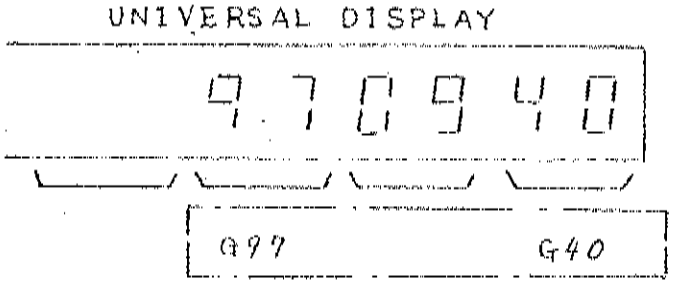
Parameter No. 90: backlash compensation

Z-axis, result = 10 pulses

List of parameters

parameter No.	address	function of parameter	input
90	X	backlash compensation	Give number of pulses taking 1 pulse = least increment
	Z		
94	X	rapid traverse speed	Give speed taking 1= 16 mm/min
	Z		
95	X	acceleration/ deceleration	Acceleration and deceleration taking 1 = change of 128mm/min in 30 msec
	Z		
98		time span between issuing of M-(S- or T-) and MF- (SF- or TF-) signals	1 = 1 msec

Parameter number 99, if selected, allows a special G-command to be displayed.

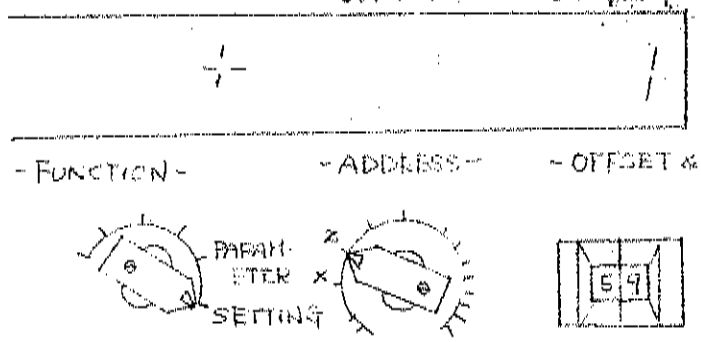


(12) Writing parameters

To do this operation, set the SYSTEM NO switch provided inside the panel to "1". The switch is normally set to "0" inhibiting parameter writing.

Follow the procedure given below.

1) Set the SYSTEM NO switch inside the panel (at the back of the front door) to "1".
2) Select the MDI mode.
3) Set the FUNCTION selector to PARAMETER.
4) Set a parameter number on the OFFSET & PARAMETER NO. switch.
5) For a parameter of 90 or greater, select an axis with the ADDRESS selector.
   Then the current value of the parameter designated will be displayed.
6) Key in a figure from the DATA keyboard. The figure just keyed in will be displayed.
   To correct the figure, depress the CAN key and key in the correct one.



Example of setting data display.

(12)^2. 2 Writing setting data

Follow the procedure given bellow

1. Set the FUNCTION selector to SETTING
2. Set the setting number on the OFFSET & PARAMETER NO. switch.
3. Key in a figure from the DATA keyboard.
   ON : 1
   OFF : 0

   To correct the figure depress the CAN key and key in the correct one.
4. Depress the WR key.
   Then the data just keyed in will replace the old one.
5. Repeat steps 2 through 4 for another setting data as necessary.

Note : While data is being input, its least significant digit will light on and off

7) Depress the WR key.
Then the data just keyed in will replace the old one.

8) Repeat steps 4 through 7 for another parameter as necessary.

9) Reset the SYSTEM NO switch to "0".

Note: While data is being input, its least significant digit will light on and off.

(12)-2 Display and Writing of SETTING data

(12)-2-1 setting data display

The value of any setting data can be displayed on the universal display in the following way. This operation is always possible independently of the operating mode.

1 Set the FUNCTION selector to SETTING.

2 Set a setting number on the OFFSET & PARAMETER NO switch.

Setting number	Function
50	Rapid traverse. speed 0:H, 1:L
51	Reserve
52	Reserve
53	Reserve
54	Reserve
55	Reserve
56	Reserve
57	Reserve
58	Reserve
59	M00 DELETE

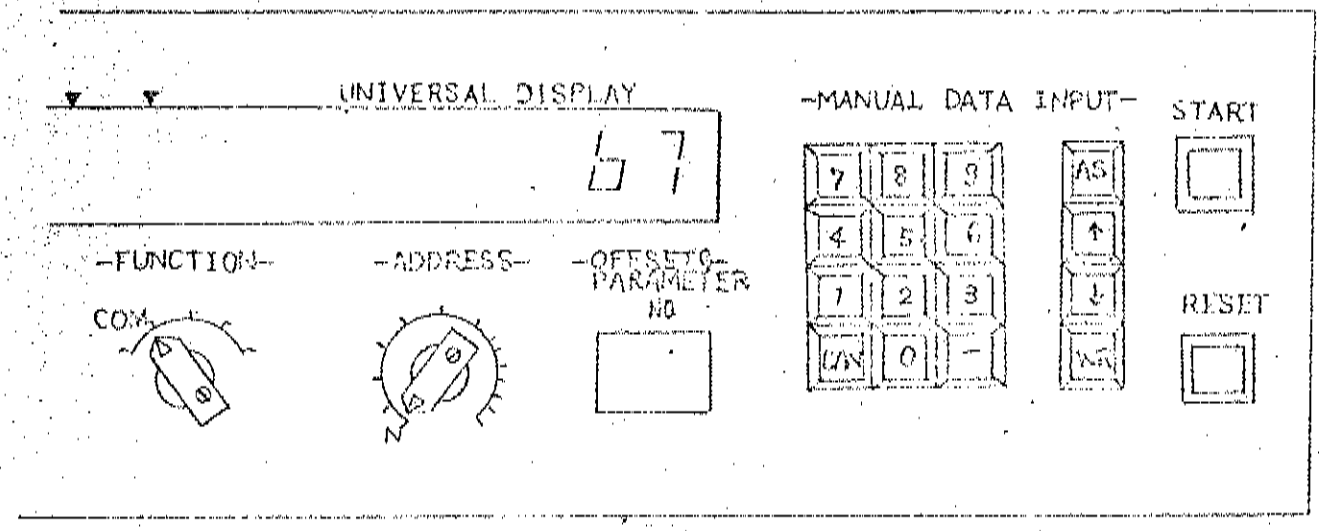
(13) Address search

The contents of a data block in tape or memory can be searched through the key of sequence number or address in the following way.

1) Select the TAPE (or MEM) mode.
2) Set the FUNCTION selector to COM.
3) Set the ADDRESS selector to a desired position.
4) Key in a figure from the DATA keyboard.
   To correct the figure just keyed in, depress the CAN key and then input the correct one.
5) Depress the AS key, and then address search starts. When the block containing desired address data is encountered, its contents will be moved to the buffer.

Note 1: If the CAN key or RESET button is depressed during search, search operation will be cancelled.

Note 2: If M02 is encountered during search, search operation will be cancelled.



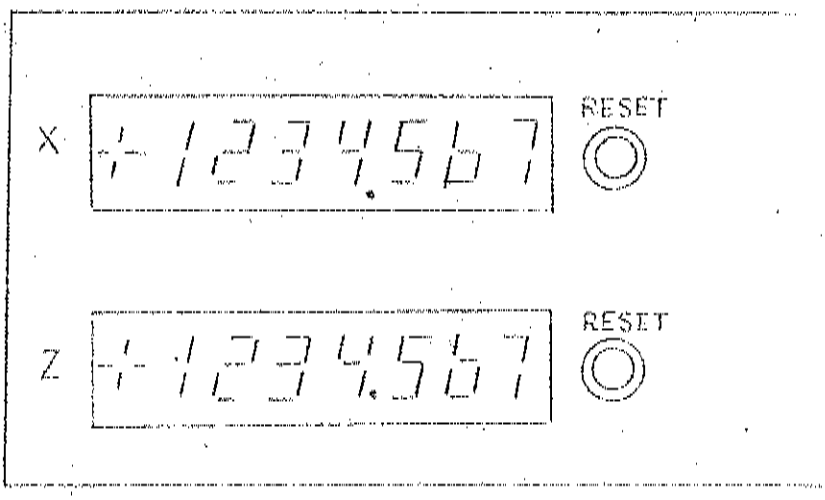
Example of block search of sequence number 67

4.2 Aux. Control Panel

4.2.1 Current position display unit

The processor sums up incremental movements of the tool and the display indicates the current position of the tool thus worked out.

1) It displays zero just after turning on power.
2) Depress the RESET button to clear the display.
3) Even when the tool has come up to the end of stroke or it has been emergency-stopped, the exact value of its current position will be displayed.



4.2.2 OIL OK lamp

Turn on the no-fuse breaker on the machine tool's distribution panel, the motor that drives hydraulic unit starts running and the OIL OK lamp goes on.

4.2.3 CHUCK CLAMP lamp

While the spindle remains stationary, it is possible to release the tool chuck by depressing the machine tool's footswitch. The CHUCK CLAMP lamp lights up while the chuck is closed.

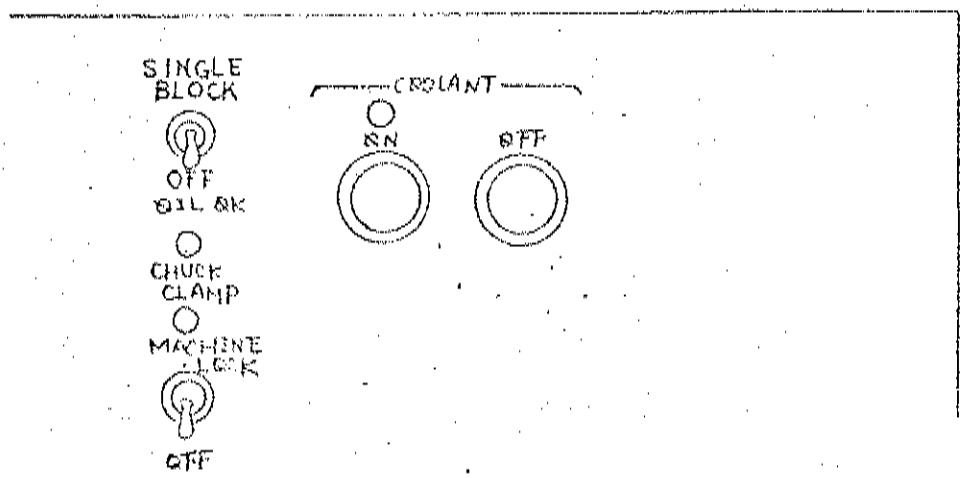
While the lamp remains off, the machine tool cannot be started automatically.

4.2.4 MACHINE LOCK switch

When the switch is turned on, the machine tool will be locked but current position commanded by tape or manual operation will be displayed.

Turn it on to check tape or set a figure on the current position display unit.

Turn it off, and then both the current possition display and motion of machine tool will assume a dynamic status.



4.2.5 SINGLE BLOCK switch

Turn on the switch and start the machine tool by depressing the CYCLE START button on the machine tool's pendant. The block commands on the NC tape will be executed and then the machine tool will come to a stop.

At this time, the commands of the next block are in the buffer registers and they can be displayed by using the universal display as the command data display.

If the switch is turned on while commands on an NC tape are being executed, the machine tool will come to a stop after executing the current block leaving the commands of the next block in the buffer registers. Depress the CYCLE START button at this time, and the commands contained in the buffer registers will be executed and the machine tool will come to a stop after reading the data of the succeeding block into the buffer.

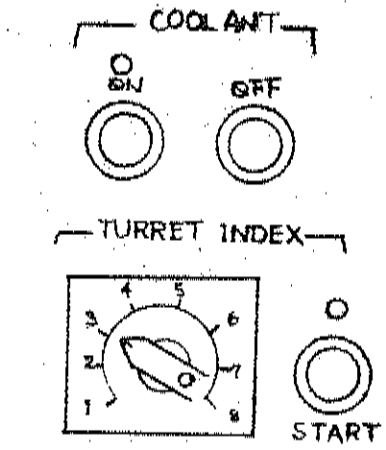
Note 1: If the SINGLE BLOCK switch is turned on during thread cutting (G32), the machine tool will not stop immediately on the spot but after the block of thread cutting has been executed.

Note 2: If the SINGLE BLOCK switch is turned on during a fixed cycle (G90 - G94), the machine tool will come to a stop at the end of the fixed cycle.

4.2.6 COOLANT buttons

Depress the COOLANT ON button. Then coolant starts running and the indicator lamp lights up. The lamp also lights up when coolant starts running controlled by the command of M08 on tape.

Depress the COOLANT OFF button. The coolant stops and the lamp goes off.



4.2.7 TURRET INDEX selector

This selector is used to select a tool by manual operation.

Select a desired tool number and depress the START button. Then the tool head will be elevated and start turning.

When the desired tool has been selected, the tool head stops revolving and goes down, and the indicator lamp goes on at the same time.

4.2.8 SPINDLE control switches

These are used to control spindle rotation in manual operation.

1) Depress the NORMAL button and the spindle will run in the normal direction. Depress the STOP button to stop the spindle.

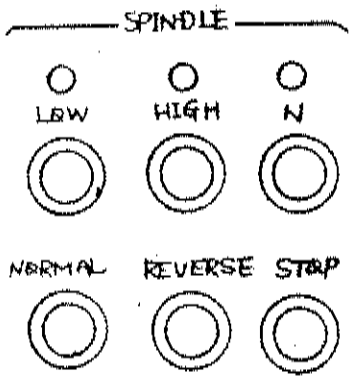
Depress the REVERSE button and it will run in the reverse direction.

Bring the main spindle revolution to a stop before reversing its direction.

2) The spindle-speed range is switched over as necessary. Depress the SPINDLE LOW or the SPINDLE HIGH button. Then, the spindle-speed range is switched over and the indicator lamp lights up.

3) If button N is depressed while the spindle is stationary, the power transmission system comes to the neutral position with lamp N becoming dark and the chuck can be turned by hand.

While the power transmission system is neutral, neither the spindle nor automatic operation can be started.



4) If automatic operation is switched to the manual mode during operation, the spindle will come to a stop. If the automatic mode is resumed, the main spindle automatically restores the operating condition just before the manual mode was entered.

5) If operation is switched to the automatic mode after the spindle has been driven in the manual mode, the spindle will come to a stop unless M03 or M04 has been issued in advance.

4.3 Tape reader

4.3.1 Construction

As shown in Fig. 4.3.1, a photoelectric sensor is installed inside the front door of the paper tape reader. Data is transfered from the NC tape to the system via the paper tape reader.

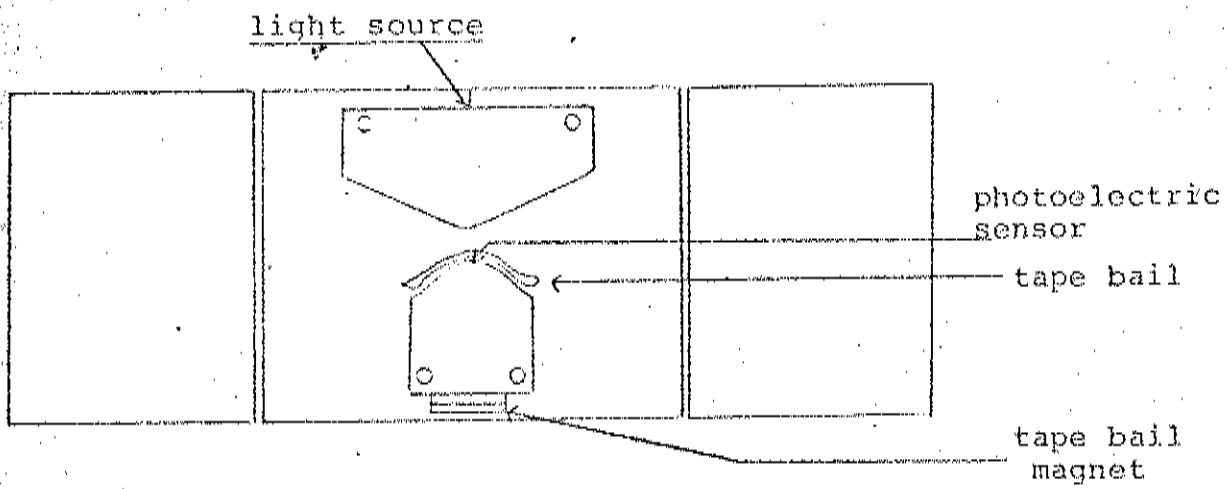


Fig. 4.3.1 Construction of paper tape reader

1) Light source
It consists of a lamp and condenser. The lamp working voltage is adjusted to +15 V ± 3% with an adjustable resistor.

2) Photoelectric sensor
It reads out NC tape. There is a glass window covering the sensor. Be careful not to scratch it and to keep it clean, because otherwise misreading can result.

3) Tape bail

Holding up the tape bail magnet, mount an NC tape and then push down the tape bail.

4) Tape compartment

It can accommodate an NC tape of about 25 meters in length. There is a tape outlet on the front door as shown in Fig. 4.3.2. A polyester tape installed inside the compartment will help take out NC tape.

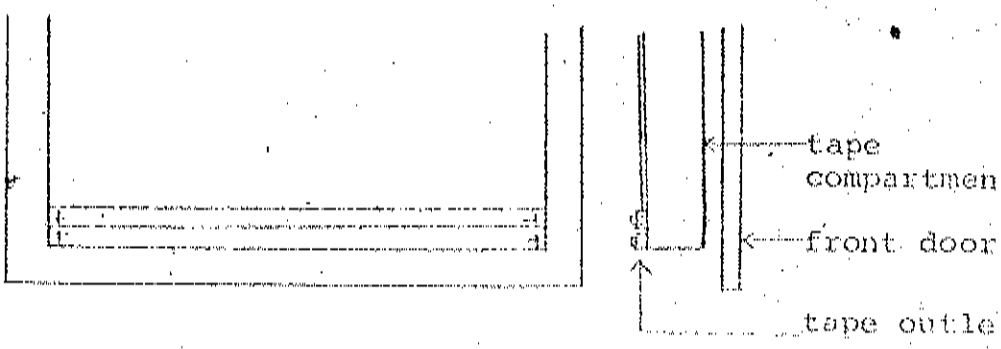
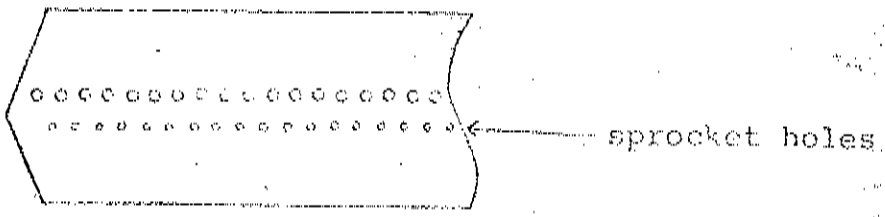


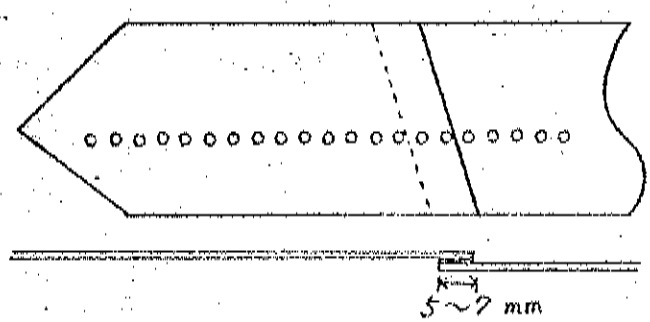
Fig. 4.3.2 Tape outlet

5) Set the NC tape in the direction as shown below viewed from the topside.



tape runs in this direction
(space marks of EIA code)

To splice tapes, cut the ends of both tapes slantwise and cement them together.



←——tape runs in this direction

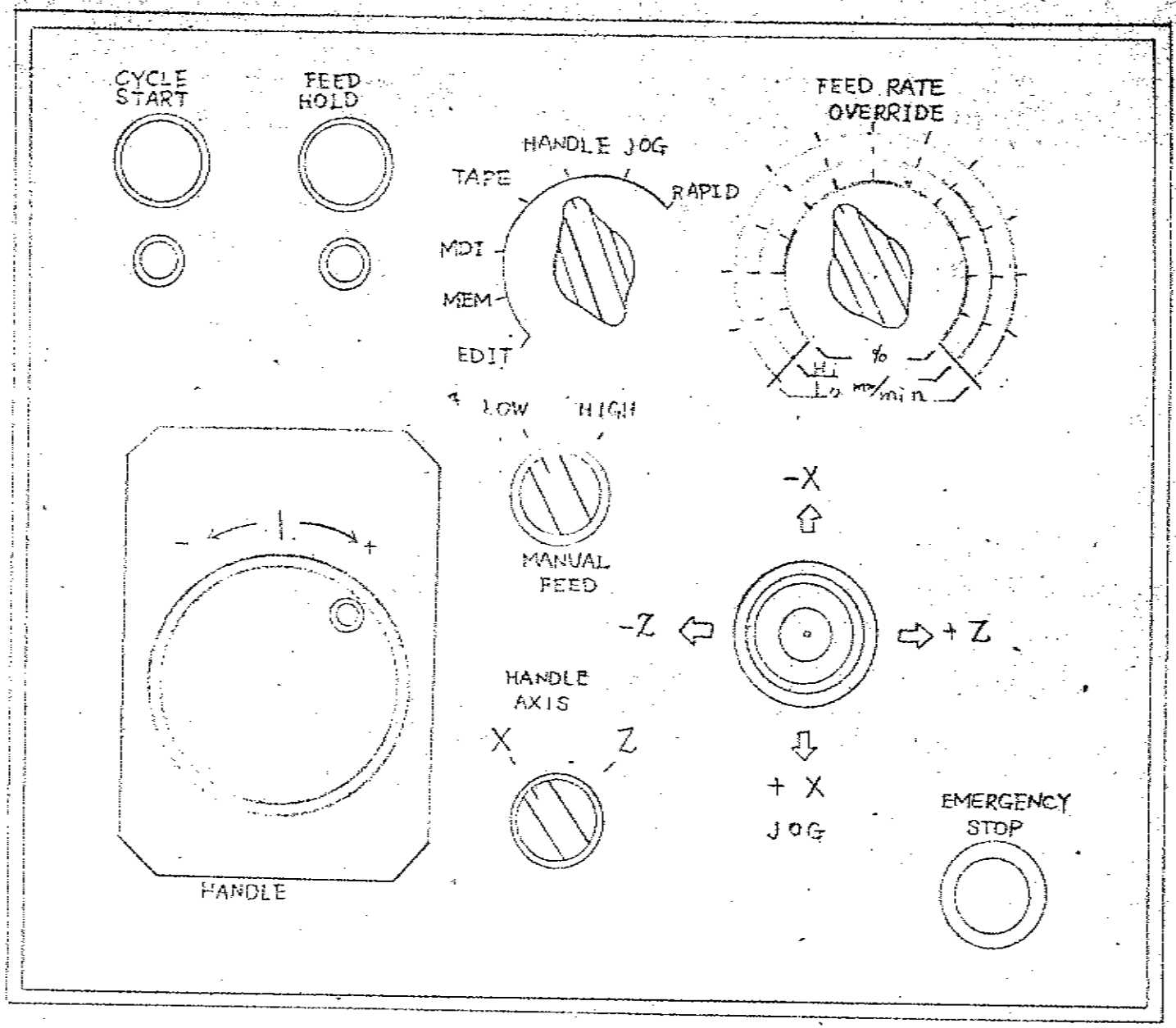
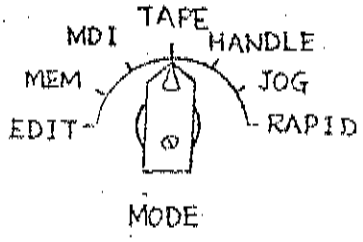


Fig. 4.5.1 Machine tool's operating pendant (example)

4.5 Machine tool's pendant switch

The controls on the machine tool's pendant switch depend somewhat on the machine tool. Refer to the instruction manual of the machine tool details.

4.5.1 Mode selector



This is a selector that has seven positions and allows selection of the operating mode of the NC system.

RAPID	The tool traverses rapidly by manual operation.
JOG	The tool is fed continuously by manual operation at the feedrate determined by the FEEDRATE OVERRIDE switch.
HANDLE	The tool is fed with the aid of a manual pulse generator.
TAPE	The NC system is automatically controlled with NC tape.
MDI	It is permitted to input data manually.
MEM	The system performs automatic operation using memory.
EDIT	Select this position to load a program to the memory or edit it.

4.5.2 CYCLE START (auto-start) button

Depress this switch to start the system in the

automatic operation mode (TAPE, MDI and MEM). The indicator lamp lights up when automatic operation starts.

Note: The system cannot be started with this button when the chuck is released or the spindle is at neutral.

4.5.3 FEED HOLD (temporary stop) button

Depress this button to temporarily suspend automatic operation.

1) If the tool is traveling, it is decelerated and then comes to a stop.
2) This button is ineffective during thread cutting by G32 or G92.
3) If it is depressed while an M, S or T function is being executed, the temporary stop lamp will light up but execution of commands will continue until the end of a block is encountered.
4) For the operation of tool offset designated by a T function, the tool movement will be stopped temporarily.

4.5.4 FEEDRATE OVERRIDE switch

1) During automatic operation, the feedrate determined by an F function can be overridden in the range of 10% to 200% in steps of 10% using this switch.
2) Thread cutting of G32 or G92 will be performed at the feedrate specified with an F function at whatever position the switch may be set.
3) This switch can also be used to set the JOG feedrate.

4.5.5 HIGH-LOW switch

1) It selects the range of JOG feedrate.

LOW: low-speed range

HIGH: high-speed range

LOW	HIGH
1.5 mm/min	30 mm/min
3.0	60
4.5	90
6.0	120
7.5	150
9.0	180
10.5	210
12.0	240
13.5	270
15.0	300
16.5	330
18.0	360
19.5	390
21.0	420
22.5	450
24.0	480
25.5	510
27.0	540
28.5	570
30.0	600

2) It can also be used to select the value of step feed by handle operation.

	LOW	HIGH
X-axis	1 μ/step (radius)	5 μ/step (radius)
Z-axis	2 μ/step	10 μ/step

4.5.6 Manual pulse generator

It is used to operate the system in the HANDLE mode. See page 87 for operation.

4.5.7 EMERGENCY STOP button

Depress it to emergency-stop the system or turn off power. Then servo power will be cut off and feeding will stop immediately.

It is possible to turn on servo power again by depressing the POWER ON button but the system will not start working until reset button is depressed.

5. OPERATIONAL PROCEDURE

5.1 Preparation before turning ON power

(1) Make sure that the front and rear doors of the panel are closed.

Note: The NC system is of a enclosed structure that does not allow air to freely enter.

Do not leave the doors including that of the paper tape reader open for a long time.

5.2 Turning ON power

(1) Making sure that the main power has been supplyed, depress the POWER ON button.

(2) The initial timer will be reset in about 2 seconds. Then depress the POWER ON button again to turn on servo power.

(3) When the system is ready for operation, the READY lamp will go on.

5.3 Manual operation

(1) Checking operation of manual feed.

a) HANDLE operation (using manual pulse generator)

- o Set the mode selector to HANDLE.
- o Select the X- or Z-axis by means of the HANDLE AXIS SELECT switch.
- o Set step feed by means of the HIGH/LOW selector.

  LOW: 2 μ/step

  HIGH: 10 μ/step
- o Turn the handle of the manual pulse generator.

b) JOG operation (manual continuous feed)

- o Set the mode selector to JOG.
- o Set feedrate by means of the FEEDRATE OVERRIDE switch.
- o Select a feedrate range by means of the HIGH-LOW selector.
- o Tilt the JOG switch in the desired direction.

c) RAPID traverse

- o Set the mode selector to RAPID.
- o Tilt the JOG switch in the desired direction.

(2) Checking operation using Aux. control panel.

When the mode selector is set at HANDLE, JOG or RAPID, it is possible to check the machine operation using the Aux. control panel.

a) Checking coolant

- o Depress the COOLANT ON button and coolant will start running.
- o Depress the COOLANT OFF button and coolant will stop.

b) Tool head selection

- o Select a tool number with the TURRET INDEX selector.

- o Depress the START button and the tool head will revolve and come to the desired tool, and an indicator lamp will light up.

c) Spindle

- o Set main spindle speed by means of the SPINDLE SPEED selector.
- o Select speed range A or B.
  This operation is possible only when the spindle is at rest
- o Depress the NORMAL button and the main spindle will start running in the normal direction. Depress the STOP button to stop it.
  Depress the REVERSE button to reverse its direction of rotation.
  Stop the main spindle once before reversing its direction of revolution.
- o If button N is depressed while the spindle is at rest, power will not be transmitted to the main spindle (neutral). Depress button N again to get out of the neutral condition (an indicator lamp N will light up).
  The spindle and automatic operation cannot be started in the neutral condition.

d) Opening and closing chuck

While the main spindle is stationary, every depressing of the machine tool's footswitch will cause the chuck, if it is open, to close, and if it is closed, to open.
Automatic operation cannot be started when the chuck is left open.

5.4 NC tape operation

(1) Check that the alarm display lamp on the control panel remains off.

(2) Check the tool offset and correct it if necessary.

(3) Set the mode selector to TAPE.

(4) Load an NC tape onto the paper tape reader in the following way.

a) Open the front door of the paper tape reader.

b) Remove dust off the surface of the tape with a soft cloth.

c) Pushing up the tape bail, insert the tape beneath it.

d) Make sure that the tape is held by the guide.

e) Press down the tape bail onto the tape.

f) Close the reader's front door.

(5) Depress the RESET button on the control panel.

(6) Depress the CYCLE START button on the machine tool's pendant switch, and then NC tape operation will start.

(7) To suspend operation temporarily, depress the FEED HOLD button on the control panel.

5.5 MDI (manual data input) operation

(1) Set the mode selector to MDI.

(2) Set the FUNCTION selector on control panel to COM.

(3) Select an address character by means of the ADDRESS selector.

(4) Key in a data from the DATA keyboard. To correct the wrong data keyed in, depress the CAN key and then key in the correct figure.

(5) Depress the WR key.

(6) Repeat steps 3 through 5 until data of a block have been input.

(7) Depress the CYCLE START button on the machine tool's pendant switch, and the data just keyed in will be executed.
The NC system cannot start operation when the chuck is open or the main spindle is in neutral.

5.6 Memory operation

The NC system incorporates a memory having a capacity of 4,000 characters or 10-meter long tape. Programs stored in it make automatic operation possible.

5.6.1 Putting NC tape into memory

Follow the procedure given below to store the contents of NC tape into the memory.

1) Set the system to the EDIT mode.

2) Depress the RESET button.

3) Load an NC TAPE onto the paper tape reader. Now it is ready for transferring the contents of the NC tape into the memory. Then the LABEL SKIP lamp lights up and data are disregarded until the first EOB is encountered.

4) Depress the START button (on the control panel).
Now the system starts reading the tape and its contents are moved to the memory.
If M02 is encountered on tape, the operation stops at the end of the block containing the command.

5) If the SINGLE BLOCK switch is turned on, data will be moved block by block as the START button is depressed.

Precautions

1) Be sure to place M02 in the last block to be stored in the memory.
2) If the number of significant characters (including EOB mark) on tape exceeds the capacity of the memory, it is regarded as input error.
3) Displaying in the EDIT mode is done just as it is given on tape, in units of 1/100 mm for coordinate commands.

5.6.2 Checking contents of memory

The contents of the memory can be checked in the following way.

1) Select the EDIT mode.
2) Depress the RESET button.
3) Set the FUNCTION selector to COM.
   It is ready for checking the contents of the memory starting from the first block.
   The display is cleared by the resetting operation.
   The LABEL SKIP lamp lights up but it is not significant.
4) Depress the "↓" key.
   Now the first block in the memory has been picked up.
5) Select an address character with the ADDRESS selector.
   Now the command data corresponding to the selected address will be displayed one by one.
   Nothing will be displayed if the memory is blank.
6) Depress the "↓" key to advance to the next block. Depress the "↑" key to go back to

the preceding block. Set the ADDRESS selector for each block.

7) It is possible to localize a desired block simply by means of address search (see page 72) in this mode.

8) In order to return to the starting address of the memory, depress the RESET button,

Note: If address search cannot find the desired data block or the program area has been exceeded by the depressing of the "↓" or "↑" key, the INPUT ERROR lamp will go on. It will go off on resetting.

5.6.3 Automatic operation in memory mode

Automatic operation with a program stored in the memory is performed in the following way.

1) Select the MEM mode.

2) Depress the RESET button.
This is the step that is required to cause the program to be executed from the first block.
The LABEL SKIP lamp goes on but it goes off when the first block is read out.

3) To execute the program from the middle position, read out the desired block using address search or the "⦃" and "⦄" keys.
In the MEM mode, the block thus selected will be moved to the buffer and executed when activated.

4) Depress the CYCLE START button.

5) The succeeding operation is just the same as with tape operation.

Precautions

1) If M30 is given, operation will be repeated

from the beginning of the program without interruption.

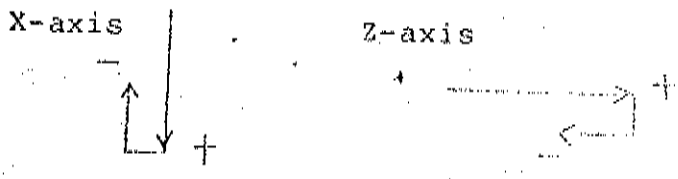
2) If M02 is given, the main spindle and coolant will stop and the system will be reset. When restarted, it will start execution from the beginning of the program.

5.7 Preparation for turning OFF power

(1) Make sure that the CYCLE START lamp on the machine tool's pandant switch is off.

(2) To adjust the backlash condition, move both axes in the negative direction.

With power on, the NC system when recieving the first tool travel command in the positive direction starts compensation for backlash. Therefore, adjust the tool backlash in the negative direction by JOG as shown below before cutting off power.



* Move 1 mm or more.

5.8 Turning off power

(1) Depress the EMERGENCY STOP button to cut off servo power.

(2) Depress the POWER OFF button on the control panel.

(3) When power has been cut off, the positional relationship between the tool and motor shaft may be varied by reaction. So the origin must be readjusted whenever power has been turned on as a rule.

When the EMERGENCY STOP button has been depressed, the current position is strictly held and no readjustment for the origin is required.

(4) If the shift of origin is small, the origin adjustment can be simplified in the following way.

(a) Depress the EMERGENCY STOP button to cut off servo power.

(b) The current position being displayed will vary as the tool moves and the value of change is written down.

(c) Depress the POWER OFF button to cut off control power.

(d) Turn on power in the given the procedure.

(e) Turn on the MACHINE LOCK switch on the aux. control panel.
Adjust the current position by manual operation so that it becomes equal to the value recorded.

(f) Turn off the MACHINE LOCK switch when the current position displayed has become the desired value.

This operation eliminates origin adjustment.

5. 9 Tape editing

Data stored in memory from NC tape (see page 90) can be edited and NC data can be stored by the MDI operation in the EDIT mode in the following way.

5. 9 .1 ERA (erase)

(1) Search the block to be erased using "AS", "↓" and "↑".

(2) Depress "ERA" button, and the current block will be erased and the contents of the next block will be displayed.

```
G00 x 123 Z456
    x 789                 Depress ERA when
G01       Z012 F34        this block is not
                          needed.
```

5.9.2 ALT (alteration)

(1) Search the block to be altered using "AS", "↓" and "↑".

(2) Make alteration in units of address character by MDI operation.

(a) To add an address character which is not involved in the block searched, input a data from the DATA keyboard by MDI operation and depress "WR".

(b) To change a figure of address character in the block searched, input new data by MDI operation and depress "WR".

(c) To erase a address character in the block searched, depress "CAN". The data just displayed will be erased and nothing will be displayed.

(d) To cancel data being input, depress "CAN".
Then the data displayed just before the input will be restored on the display.

(3) Depress "ALT" button, and the contents of the block just altered will replace the old one and the BUFFER lamp will go off.

(Example)

```
G00 X 123 Z456
    X 789             To replace with
G01       Z12F34      G01 X 789 F34
```

(a) Setting the FUNCTION selector to COM, and the ADDRESS selector to X, depress "7", "8", "9" and "AS".

(b) Setting the ADDRESS selector to G, depress "0", "1" and "WR" on the DATA keyboard.

Setting , the ADDRESS selector to F, input "3", "4" Depress "WR".

(c) Depress "ALT".

5. 9.3 INS (insertion)

(1) Search the last one block ahead to insert using "AS", " ↓ " and " ↑ ".

(2) Depress "INS" button. Then the display will be cleared and the BUFFER lamp will go on. The block having EOB alone will be secured.

(3) Key in the data to be inserted by MDI operation.

(4) Depress "INS" button. Then the contents of the block just input will be inserted next to the block seached and the BUFFER lamp will go off.

(Example)

G00 X 123 Z456

X 789 To insert Z987 S05 M03

G01 Z012 F34

(a) Setting the FUNCTION selector to COM and the ADDRESS selector to X, depress "7", "8", "9" and "AS".

(b) Depress "INS" button.

(c) Resetting the ADDRESS selector to Z, key in "9", "8", "7" and "WR" from the DATA keyboard. Setting the ADDRESS selector to S, key in "0", "5" and "WR", and at M, "0", "3" and "WR".

(d) Depress "INS" PB.

5.9.4 EOB (loading)

(1) Depress "RESET". (Then the starting address of the memory area will be set.)

(2) Prepare a block's data to be loaded by MDI operation.

(3) Depress "EOB" button. Then the data just input will be loaded, the BUFFER lamp will go off, and the display will be cleared.

(4) Return to step 2 and repeat the same operation, and NC data can be loaded one by one.

6. MAINTENANCE

6.1 Routine maintenance

6.1.1 Cleaning paper tape reader

The paper tape reader does not require mechanical service and lubrication since it has a pulse motor but no movable parts subject to wear.

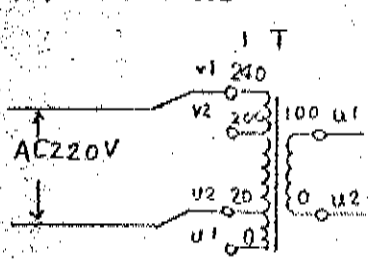
(1) Cleaning tape guide and sensor (every day)
Remove dust from the lamp, lens and photo-transistor using a soft brush.

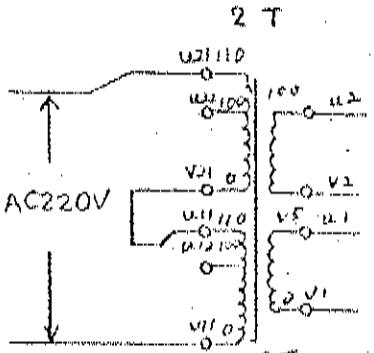
(2) Cleaning tape-guide polyester tape and tape compartment (every week)
Clean them with a brush and cloth.

6.2 Precautions for connecting to power source

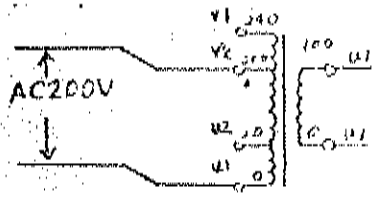
On installation, adjust the taps of transformers 1T and 2T according to the source voltage.

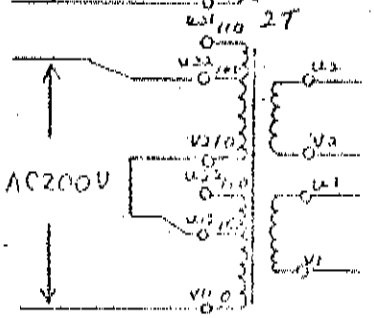
(1) 220 V AC





(2) 200 V AC





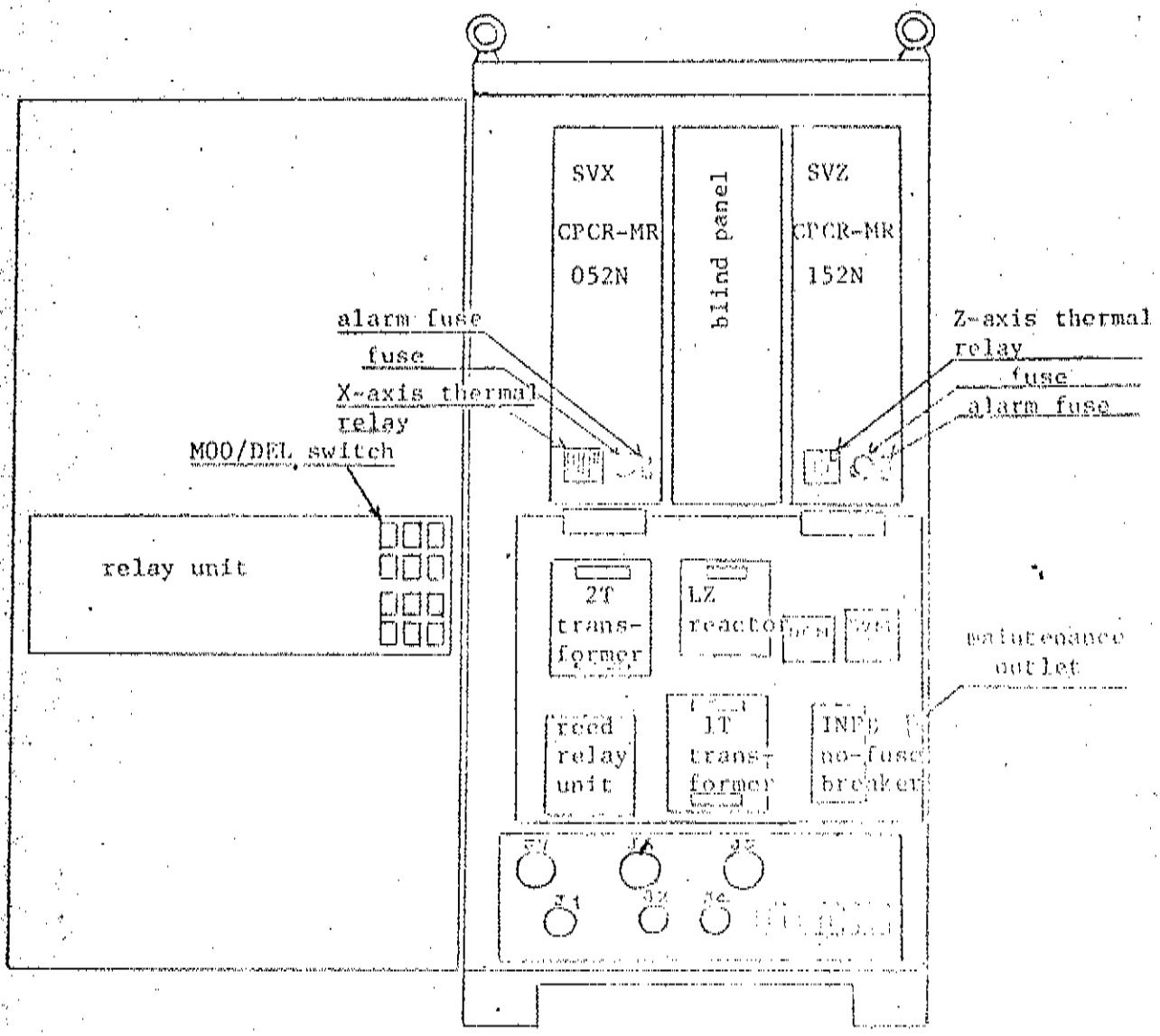
(3) Molded-case circuit breaker (INFB)

Keep the no-fuse breaker in the control unit turned on unless it is required to turn it off for servicing etc.

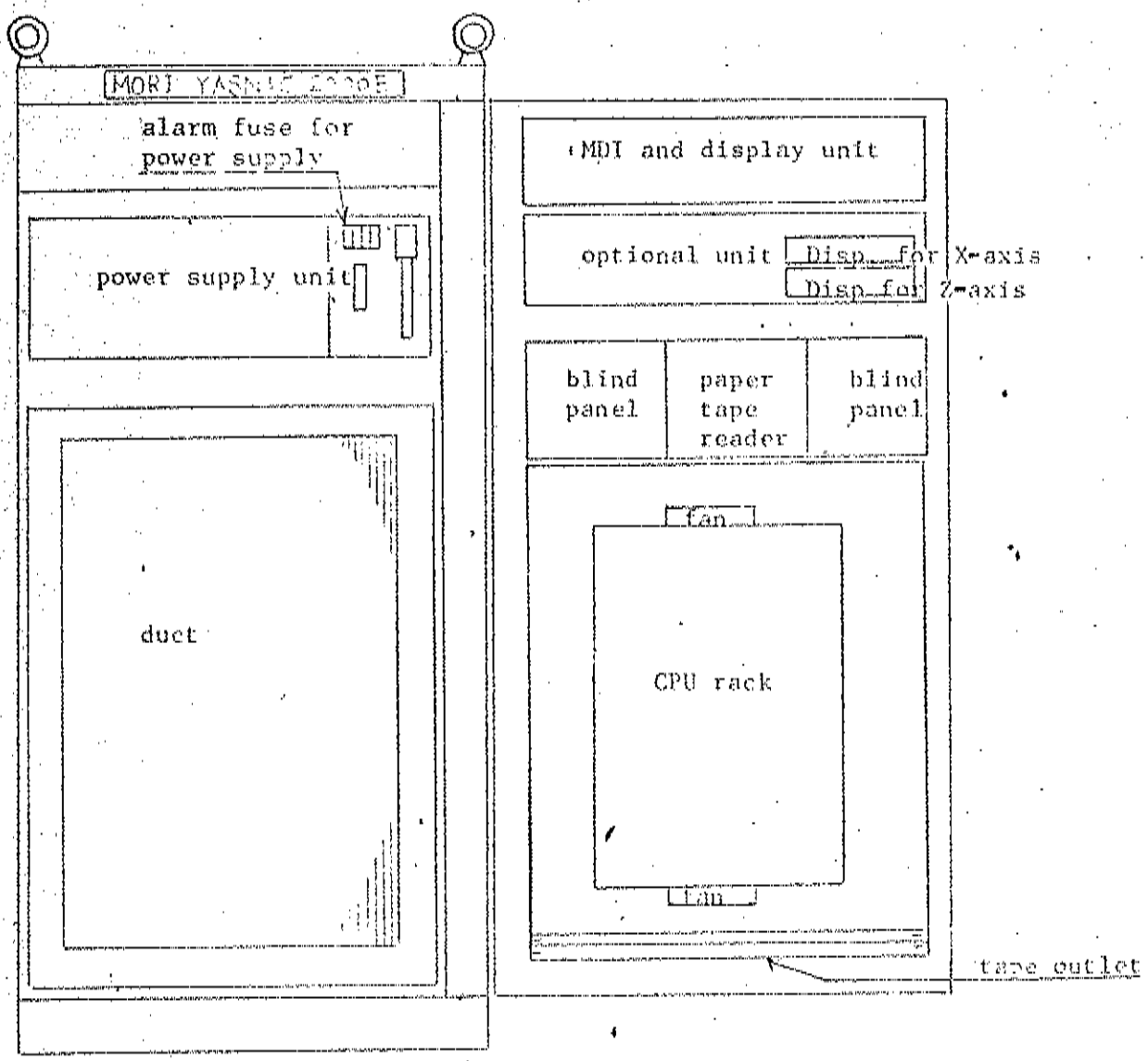
(4) Maintenance outlet

It can supply power when the no-fuse breaker is turned on. Its capacity is 1 ampere at 100 V AC.

6.3 Component layout



Component layout at the back of the unit



Component layout at the front of unit

6.4 Fuse and thermal relay

(1) Fuse

Fuses are provided separately for the control power supply and servo unit. If a fuse has been blown, localize the cause and replace it with a spare one.

(a) Alarm fuses of control power supply

. 100 V AC	15 ampere fuse
. +24 V	3.5 ampere fuse
. +15 V	1.3 ampere fuse
. -15 V	1.3 ampere fuse

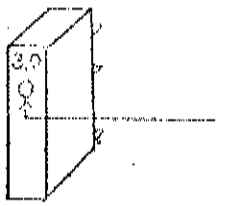
The power supply has a protective device for +5 V and 12 V.

(b) Fuses of servo unit (X- and Z-axis)

. Plug in fuse 600 V AC, 20 ampere (X-axis) 15 ampere (Z-axis), (Fuji Electric BLA type)

. Plug-in alarm fuse

3 ampere (Daito Tsushiuki P-430 type)



When the fuse is blown, a white mark appears.

alarm fuse

If the fuse of the servo unit is blown, error code 35 lights up.

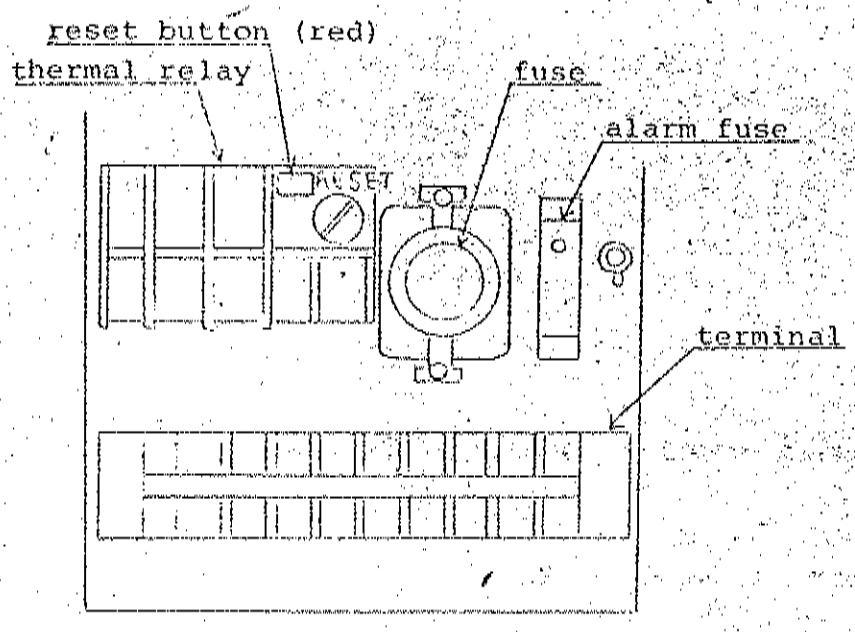
(2) Thermal relay

If the machine tool is operated when locked or overloaded, error code 35 will light up and servo power will be cut off.

Remove the cause and depress the RESET button on the control panel. Error code will turn to

31 and it becomes possible to turn on servo power again. If the RESET button is depressed without the cause being removed, error code 35 remains lit. (See page 57)

(a) Thermal relay to detect motor overload
It is equipped for the servo unit (CPCR-MR052N, CPCR-MR 152N).

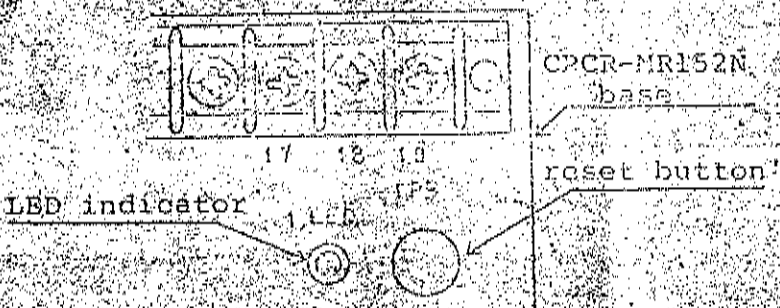


Parts layout

After cooling the motor, depress the RESET button.

(b) Temperature rise in drive unit
The Z-axis drive unit (CPCR-MR152N) is provided additionally with the following detectors.

o Power transistor temperature sensor



1 LED lights up if the power transistor temperature rises abnormally. When the temperature has lowered, depress LEB to reset the detector.

o Temperature rise in servo circuit resistor

The servo circuit resistor is fitted to the air duct at the top, and when the detector fitted to the resistor operates, its temperature rises abnormally and is reset automatically when the temperature lowers.

6.5 Troubleshooting

Most trouble can be isolated by simple checking. Check the system referring to the alarm codes and corrections described in 4.4.3. If trouble cannot be corrected, contact the service personnel or the service section of Yasukawa Electric Mfg., giving the following information.

o Time of occurrence of the trouble

Date of occurrence

Operation

o Frequency of occurrence of trouble

Does the same trouble occur many times under the same conditions?

o Kinds of trouble

Is there any other trouble occurring at the same time?