FANUC 3000C

OPERATOR'S MANUAL

FUJITSU FANUC LTD

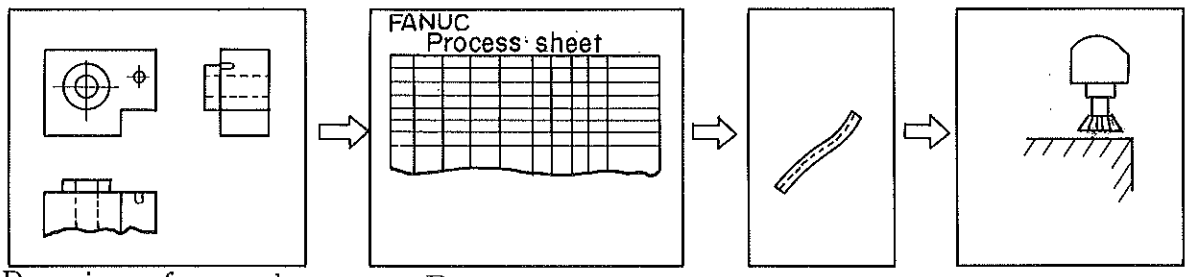
CONTENTS

B-51520E/06

APPENDIXES

1. OUTLINE

In a numerical control system, the machine tool operates according to the commands specified on a punched tape. In order to do some cutting, all the numerical values necessary for the machining, such as the dimensions, feed rates, etc., are punched from a drawing of the work to be cut on a tape according to a certain convention. The work of up to preparing an NC tape necessary for machining from this drawing is called the "programming."



Programming { Tool path (distance and direction)
Feed rate
Spindle-speed
Tool selection, etc.

Fig.1.1 Procedures in the NC system

Fig.1.1 shows the procedures in an NC system. First, the tool path, feed rate, etc. are determined from the drawing of the work and a process sheet is prepared according to the operational sequence. By operating a paper tape punch according to this process sheet, an NC tape is produced. Then the work, tool, etc. are fixed to the machine tool, the tool is set to the initial start point, the tape is loaded onto the control unit and actuated. The control unit reads in information from the NC tape, and drives the machine tool according to the commands punched thereon for machining.

This manual is a guidebook describing the method of preparing a process sheet from the drawing of a work and the method of operating an NC system.

A person who creates a process sheet from the drawing of a work is called a "part programmer."

This part programmer plays the most important role in operating an NC system, and must have a full knowledge of the abilities and limits of the NC machine tool and the control unit.

For example, he must

(1) select such a tool path as to reduce the machining time,

(2) select the optimum spindle speed, tool and feed rate,

(3) effectively use the M function or G function,

(4) give the operator information on the tool change, machine setup point, etc.,

(5) effectively use the Canned Cycle, etc.

Also, the operator of an NC system must, in order to produce high-accuracy products, pay attention to the following points in addition to the usual operations (loading and unloading the work and the tool, setting the tape, operating the control panel, etc.)

(1) Setting the Cutter Compensation amount (requiring the periodical resetting because of the tool wear, etc.)

(2) Giving instructions about regrinding the tool

(3) Daily maintenance (cleaning and inspection)

The operator does not need to operate the NC equipment with its front and back doors open except for the maintenance. Particularly in a usual operation, he rather operates it with the doors (including that of the tape reader) closed so as not to let the outside air enter directly into it.

This manual discusses the equipment with options for machining centers, milling machines, etc. Refer to the description issued by the machine tool builder as to which options can be used in practice.

The FANUC 3000C are NC equipped with 2-to 4-axis positioning and continuous cutting for milling machines, machining centers, etc, and used FANUC DC servo motor series as the servo.

2. SPECIFICATIONS

2.1 Basic Unit

Item	Name	Specifications
(1)	Controlled axes	3 axes (4 axes with additional options)
(2)	The number of simultaneously controllable axes	2 axes (3 axes for positioning with additional options) (4 axes for linear interpolation with additional options) In case of manual operation, three axes are not controlled at a time.
(3)	Maximum programmable dimension	See Section 3.7.
(4)	Tape command	8-channel black paper tape (EIA RS-227)
(5)	Tape format	See Section 3.3.
(6)	Increment system	See Section 3.4.
(7)	EIA code input ISO code input	See Appendix 1.
(8)	Absolute/Incremental programming	This permits specifying the Absolute and Incremental dimensions mixedly in one block of tape. The Preparatory function (G90, G91) identifies the Absolute and the Incremental programming, respectively.
(9)	Positioning	G00 permits individual positioning for each axis at the Rapid Traverse speed.
(10)	Linear interpolation	G01 specifies the movement of a tool to be controlled along a straight line up to a programmed end point.
(11)	4-digit F-code feed	See Section 3.11.
(12)	Automatic acceleration /deceleration	See Section 3.18.
(13)	Override	A override can be effected in increment of 10% in a range 0 to 200% by a rotary switch on the operator's panel.

Item	Name	Specifications
(14)	Programming of Absolute Zero point	G92 permits setting within the equipment such a coordinate system that the current position of a tool is a specified position
(15)	Miscellaneous function	This specifies the two-digit M00 to M99.
(16)	Backlash compensation	By this function, backlash is compensated in increments of one pulse in a range of 1 to 255 pulses.
(17)	Single block	This is a function of reading and executing only one block of NC tape.
(18)	Label skip	This function is used for naming the tape or for starting the tape at any place, and ignores the information up to the first EOB code in reading the tape.
(19)	Mirror image /Remote mirror image	Symmetrical cutting can be performed with respect to any axis by this function.
(20)	Interlock	This function stops the feed for each axis independently.
(21)	Dwell	G04 permits stopping the feed for a specified period of time.
(22)	Machine lock	This is a function of operating only the position display unit during an automatic or manual operation.
(23)	Auxilliary function lock	This function inhibits output of M, S, T and B code to Machine
(24)	Display lock	This is a function of stopping the position display unit while the machine tool is operating normally.
(25)	Optional Block Skip	This is a function of selectively skipping a block of NC tape punched with "/" (slash).

Item	Name	Specifications
(26)	Manual rapid traverse feed	This function can feed the machine tool rapidly while the RAPID TRAVERSE button is kept pushed in the manual continuous feed mode by the MODE SELECT switch.
(27)	JOG feed	This function can feed the machine tool while the JOG/STEP button is kept pushed in the manual continuous mode by the MODE SELECT switch.
(28)	STEP feed	This function can feed the machine tool to X1, X10, X100, X1000 pulses when the JOG/STEP button is pushed in the step feed mode by the MODE SELECT switch.
(29)	MDI & DPL	See Section 4.1.1
(30)	Override cancel	This enable the 4-digit F-code feed Override fix to 100%.
(31)	Z-axis command cancel	During a cycle operation or manual operation a command for the Z-axis can be made to be ignored.
(32)	Tool offset A	This is a function of compensating for a cutter length.
(33)	Sequence number search	This feature enables the operater to search any following sequence number on tape.
(34)	Remote deceleration	This function decelerates the feed rate from outside.
(35)	Remote power ON/OFF	The power is turned on and off by a contact signal from outside in addition to the POWER ON/OFF button on the control unit display panel.
(36)	Buffer register	This permits continuously transmitting pulses between blocks being executed, by reading one block ahead during the operation by an NC tape.

Item	Name	Specifications
(37)	Overtravel detection	This is a function of checking if a particular axis is in a dangeous area.
(38)	Power	AC 200/220V or AC 200/220/230/240/380/415/440/460/480/550V +10% -15%, 50/60Hz ±1Hz, 3 phases
(39)	Enviromental condition for operation	(a) Ambient temperature : 0 to 45°C (b) Humidity : Less than 95%(RH) (c) Vibration : Less than 0.5G (d) Atmosphere When the NC equipment is used in the atomosphere of relatively high density of dust, cutting oil or water, or organic solvent, please contact the NC builder. Also in case of the preservation or the transportation of the NC equipment under the worse condition than that of mentioned above, please contact the NC builder.

2.2 Basic Options

Item	Name	Specifications
(1)	Tape reader	This is an equipment to enter NC tape information into the NC equipment. (1) Main tape reader without reels Reading speed: 300 cps (60 Hz), 250 cps (50 Hz) Reading method: Photoelectric Tape capacity: Approx. 35m (2) Main tape reader with reels Reading speed: 300 cps (60Hz), 250 cps (50Hz) Rewinding speed: 600 cps (60Hz), 500 cps (50Hz) Reading method: Photoelectric Reels capacity: 150m
(2)	Built-in type position display unit	This unit calculats movement amount to display the position of the movement member of a machine tool. a) 3-axis display unit b) 4-axis display unit
(3)	1-digit F-code	The feed rate is specified in a 1-digit number following address F. The actual feed rate is set on the corresponding dial on the setting/display panel.
(4)	Built-in type manual operator's panel	The setting/display panel can be provided within this operator's panel.

2.3 Additional options

Item	Name	Specifications
(1)	Circular interpolation	G02 and G03 specify the movement of a tool to be controlled along an arc up to a programmed end point.
(2)	Cutter compensation	This is a function of compensation for a cutter diameter.
(3)	Tool offset B	Tool offset for Z axis (Tool length offset) can be performed at the command of G43 or G44.
(4)	Addition of the numbers of cutter compensation	As number of cutter compensation values can be designated maximum 64 adding the formerly offset number.
(5)	Canned cycles	G73, G76, G80 to G89 specify a series of drilling, boring and tapping operations, usually specified in several blocks of commands, to be specified in one block.
(6)	Program copy	This is a function of designation a part of tape commands and invoking that the part of repetitive excutions.
(7)	Pitch error compensation	This function compensates pitch error inherent in the feed screw of a machine tool.
(8)	Stored type pitch error compensation	This permits the function mentioned above by NC unit without dog. Compensation can be made up to 75 points.
(9)	Addition of the numbers of pitch error compensation points	In stored type pitch error compensation, pitch error compensation points can be added as follows. 0 up to 87 points (for 4 axes control) 0 up to 89 points (for 3 axes control)
(10)	Limitless overtravel	This function detects the machine's overtravel by NC unit without limit switch.

Item	Name	Specifications
(11)	Manual handle feed	This device is used in the manual handle mode operation. The machine tool can move the appropriate distance by turning the handle.
(12)	Rapid traverse override	A override can be effected in a range of 1, 10, 50 or 100% by a rotary switch provided on the operator's panel.
(13)	Manual zero return	This function permits making the movable member of a machine tool return to the zero point forcedly in a manual operation.
(14)	Scale return	This is a function to position the machine tool at a lattice point on the inductosyn scale or resolver.
(15)	Dry run	This is a function of switching a feed rate specified on an NC tape to a manual continuous feed.
(16)	ATC zero return	Zero Return for Automatic Tool Change can be specified on NC tape.
(17)	Automatic Zero Return Check	This function checks if the machine tool correctly return to the machine zero point.
(18)	S, T and B function	This function permits generating signals in BCD code to the machine tool to perform the appropriate operation.
(19)	Mid-tape start-up function	This function permits returning a machine tool to the start point of operation by a block currently being excuted for remachining.
(20)	Remote sequence number search	This function is the same as the sequence number search function except that search is made from the machine side.

Item	Name	Specifications
(21)	Inch/Metric conversion	This feature is for special increment system like MM input to the machine with inch ballscrew.
(22)	Manual pulse generator	This device permits efficiently, performing the sensitive feed of a machine tool manually. Next six kind of increment system can be selected. Manual pulse generator A ..... Increment system M1V Manual pulse generator B ..... M2V, M3V, M4V Manual pulse generator E ..... I1V Manual pulse generator H ..... I2V, I3V, I4V Manual pulse generator I ..... 0.01mm/0.001in double scale Manual pulse generator J 0.001mm/0.0001in
(23)	Remote type position display unit	This function is the same as the built-in type position display unit except that display unit is mounted in machine side. Next two types are there (1) remote type position display unit 3 axes. (2) remote type position display unit 4 axes.
(24)	Additional 4th axis	In addition to the X, Y, Z axes, an axis can be controlled (4 axes in total).
(25)	Circle cutting	Starting from the center of a circule, the cutter moves and cuts along a circle.
(26)	Part program storage and editing	Tape can be stored in memory and the contents of tape can be edited.

Item	Name	Specifications
(27)	Tape Memory	The following sorts of tape memory can be selected. (a) Tape Memory A (Capacity; Nearly 13m/26m/40m of tape information) Tape information stored in memory is clear in case of power off. (b) Tape Memory B (Capacity; Nearly 13m/26m/40m of tape information) Tape information stored in memory is held in case of power off. (c) Tape Memory C (Capacity; nearly 2.5m of tape information) Tape information stored in memory is clear in case of power off.
(28)	Programble interface with machine tool	This replace the conventional relay circuit for machine controls with logical circuit in NC unit.
(29)	Punch Function	NC command data stored in the memory or corrected NC command data can be punched out as NC command tape.
(30)	Herical Interpolation	Simultaneous 3-axis control is possible for orthogonal 3 axes, by which circular interpolation for optional 2 axes and linear interpolation for remaining axis can be conducted at a time. The tool can be moved following a spiral path, for threading of a large-diameter screw or 3-dimensional machining of cams.
(31)	Additional power supply unit	In case that built-in type position display unit is attached to NC unit, additional power supply unit must be specified.

Item	Name	Specifications
(32)	1 digit F-code unit scale	This is selected by increment system; Six kinds for linear axis, three kinds for rolling axis. For liner axis Increment system a) 1 digit F-code scale A ..... M1V, M2V b) 1 digit F-code scale B ..... M3V c) 1 digit F-code scale C ..... M4V d) 1 digit F-code scale D ..... 11V, 12V e) 1 digit F-code scale E ..... I3V f) 1 digit F-code scale F ..... I4V For rolling axis Increment system a) 1 digit F-code scale A ..... D1V, D2V b) 1 digit F-code scale B ..... D3V c) 1 digit F-code scale C D4V

2.4 Outer View

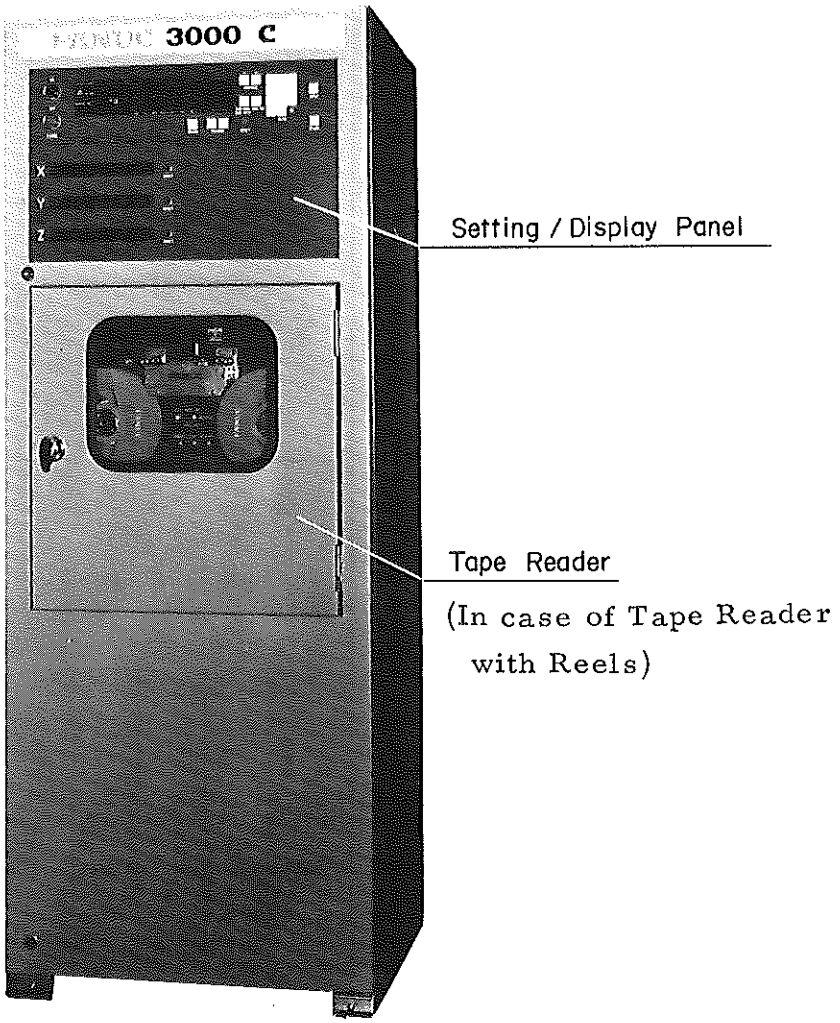
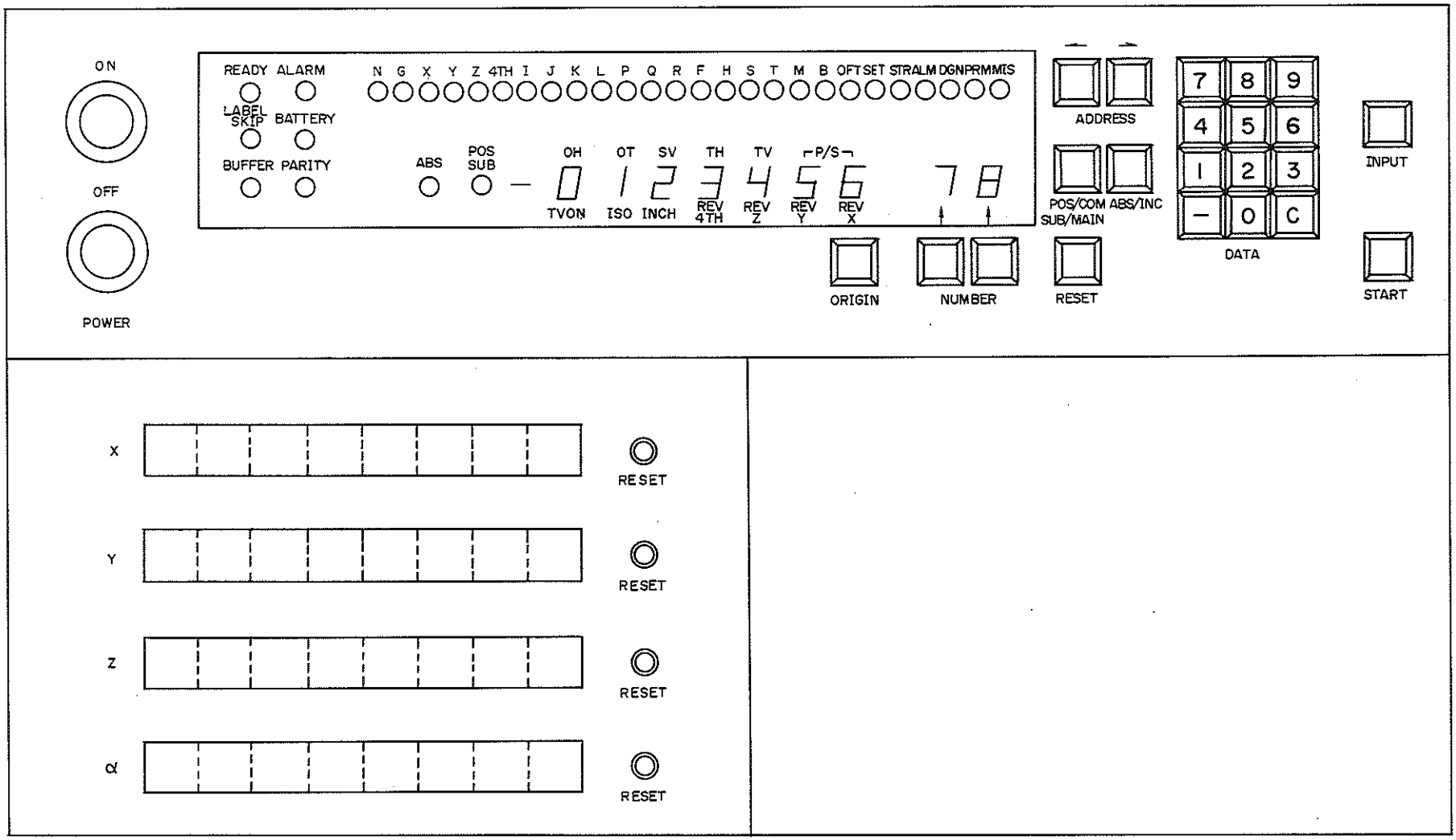


Fig.2.4.1 Outer View of FANUC 3000C
(In case of 3 axes control)



α .....A, B, C, U, V or W

Fig. 2.4.2 Setting/Display Panel (Example)

3. PROGRAMMING

3.1 General

Refer to the appendices of this manual for the following.

(1) Table of tape codes (appendix 1)
(2) Table of Characters (appendix 1)

3.2 Process Sheet

The format of the process sheet depends on the specifications of the NC equipment and machine tool. The most generic example is shown in Fig. 3.2.1.

PART NUMBER																						
PART NAME																						
PROGRMMER																						
DATE							PROCESS SHEET											PAGE		OF		

OPTIONAL BLOCK SKIP	SEQUENCE NUMBER	PREPARATORY FUNCTION	X-AXIS MOTION	Y-AXIS MOTION	Z-AXIS MOTION	4th-AXIS MOTION	ARC CENTER	ARC CENTER	ARC CENTER	CYCLE PARAM.	CYCLE PARAM.	CYCLE PARAM.	CYCLE PARAM.	FEEDRATE	SPINDLE SPEED	TOOL NUMBER	TOOL OFFSET NUMBER	CUTTER COMP. NUMBER	MISCELLANEOUS FUNCTION	END OF BLOCK	REMARKS
	N(:)	G	X	Y	Z	4th	I	J	K	P	Q	R	L	F	S	T	H	D	M	%	

Fig.3.2.1 Example of process sheet

3.3 Tape Format

The tape format conforms with the EIA RS 274-C and ISO 2539. The tape format for the F function, however, does not conform with the EIA recommendation. The format classification detailed shorthand varies with the increment system (described later) and options. It will be itemaized in Table 3.3.1.

Table 3.3.1 Format Classification Detailed Shorthand

Item		Shorthand of word	Section
Sequence Number		N 3	B
Preparatory Function		G 2	B
Move Command	Increment system M1V " M2V~M4V " I1V " I2V ~I4V " D1V " D2V~D4V	α + 42. β + 42 ..... α + 43. β + 43 ..... α + 33. β + 33..... α + 34. β + 34..... r + 42 r + 43	B, O
Dwell	Increment system M1V " M2V~M4V " I1V " I2V ~I4V	P 42 P 43 P 42 P 43	B
Canned cycle	Increment system M1V " M2V~M4V " I1V " I2V ~I4V	R+42. Q42. P42. L7 R+43. Q43. P43. L7 R+33. Q33. P42. L7 R+34. Q34. P43. L7	O
4-digit F code	Increment system M1V " M2V~M4V " I1V " I2V ~I4V " D1V " D2V~D4V	F 40 F 41 F 41 F 32 F 40 F 41	B
1-digit F code	Increment system M1V~M4V, I1V~I4V, D1V~D4V	F 1	O

Item	Shorthand of word	Section
Tool Offset/Cutter Compensation	H2 or D2	B, O
S Function	S2 or S4	O
T Function	T2 or T5	O
M Function	M2	B
B Function	B3	O

Note:

(1) α and β stand for any of X, Y, Z, U, V, W, I, J and K.

(2) r stands for any of A, B and C.

(3) Leading zeros can always be omitted except for Sequence Number (N3).

(4) Both H and D codes can be specified on the Tool Offset / Cutter Compensation number. Maximum 32 (0 ~31) or 64 (0 ~63) set can be commanded for Tool offset and Cutter Compensation.

(5) Among S function of 4 digits and T function of 5 digits and B function, only one function is selectable.

(6) With the optional B function added, the S and T functions can be expanded up to five digits. In this case, however, the B function is not effective.

(7) With the options of circule cutting, helical interpolation and program copy added, above tape format changes.

How to see a detailed shorthand.

Example: α + 52

α is any of the addresses X, Y, Z and etc.
+ is a sign.
5 means a maximum of five digits above the decimal point.
2 means two digits below the decimal point.

3.4 Increment System

The increment system is determined by the following elements:

(1) Least input increment

This is the least unit of movement amount that can be specified on NC tape or on the MDI unit. This is expressed in mm, inch or deg.

(2) Least command increment

This is the movement amount of machine tool for one command pulse transmitted from the NC equipment to the machine tool, and is the least unit in which it moves. This is expressed in mm/p, inch/p or deg./p.

Any of the increment systems shown in Table 3.4.1 is selected by MDI setting. Refer to the description issued by the machine tool builder for the actual increment system.

Table 3.4.1 Increment Systems

Increment system	Least input increment	Least command increment
M1V	0.01 mm	0.001 mm/p
M2V	0.001 mm	0.001 mm/p
M3V	0.001 mm	0.0005 mm/p
M4V	0.001 mm	0.0002 mm/p
I1V	0.001 inch	0.0001 inch/p
I2V	0.0001 inch	0.0001 inch/p
I3V	0.0001 inch	0.00005 inch/p
I4V	0.0001 inch	0.00002 inch/p
D1V	0.01 deg.	0.001 deg/p
D2V	0.001 deg.	0.001 deg/p
D3V	0.001 deg.	0.0005 deg/p
D4V	0.001 deg.	0.0002 deg/p

When cutter compensation or circular interpolation is selected, the increment system of X, Y and Z-axis must be same.

Increment system of X-axis is regarded as a fundamental. Therefore feed rate, offset amount, etc are changed into pulse speed or the number of pulses according to the increment system of X-axis.

Be careful of the designation of feed rate and offset amount for other than X-axis.

3.5 Buffer Register (Buffer)

One block ahead of the current block is read for storing in the buffer register. No interruption is caused due to a reading time of the tape reader, and the moving member of the machine can be smoothly moved between blocks.

Buffer capacity ..... 50 characters (include EOB)

When a command exceeding the buffer capacity is given, the reading of NC tape is stopped with the P/S alarm.

A command exceeding the buffer capacity is divided into two blocks. The next characters are not read into the buffer and, accordingly, are subject to no restriction by buffer capacity.

ISO code	EIA code
+	+
/	/
SPACE	Space
(	Tab
)	Del
HT	BS
DEL	Blank
NL	All mark
BS	
CR	

The following data are not stored in Buffer Register.

(1) from first character to first EOB code.
at the state of Label Skip ON.

(2) from control out "(" to control in ")".

(3) from next character of slash code to next EOB code.
at the state of optional block skip.

3.6 Absolute and Incremental Programming

Absolute or incremental programming can be used, and mixedly in one block.
G90 and G91 specify the subsequent dimension to be absolute and incremental, respectively. That is,

G90: Absolute Programming
G91: Incremental Programming

In incremental programming, as shown in Fig. 3.6.1 the end point of one block becomes the start point of the next block.

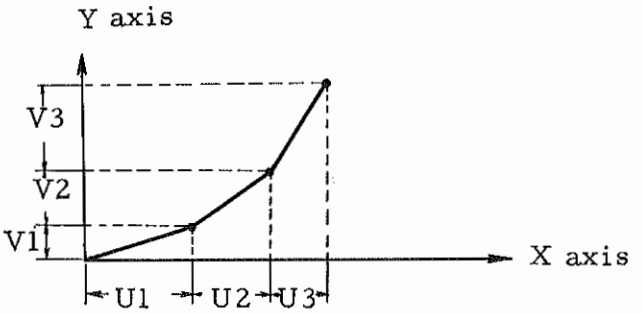


Fig. 3.6.1 Incremental coordinate values

In Absolute programming, as shown in Fig. 3.6.2 the point to which a tool is to move, is always given in a coordinate value from the zero point.

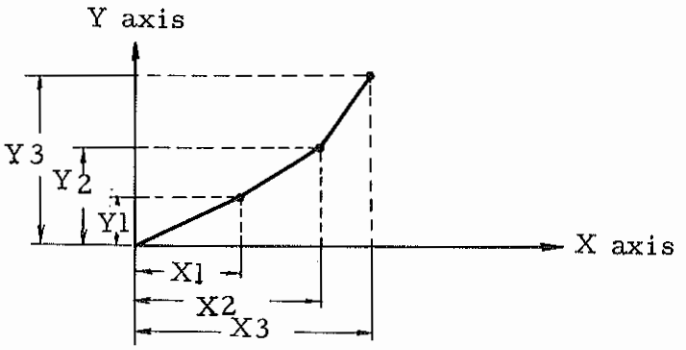


Fig. 3.6.2 Absolute coordinate values

3.7 Maximum Programmable Dimensions

Increment system	Maximum command value	Increment system	Maximum command value
M1V	± 8388.60 mm	I3V	± 419.4303 inch
M2V	± 8388.607 mm	I4V	± 167.7721 inch
M3V	± 4194.303 mm	D1V	± 8388.60 deg
M4V	± 1677.721 mm	D2V	± 8388.607 deg
I1V	± 838.860 inch	D3V	± 4194.303 deg
I2V	± 838.8607 inch	D4V	± 1677.721 deg

It is always forbidden to give the command over maximum programmable dimensions.

If over maximum command value is commanded, there is the possibility of changing the sign.

In case of circular arc, the radius is necessary under maximum command value.

In case of inch/metric conversion is selected, maximum command value is also restricted. (See 4.1.4 (9))

3.8 Tape Code

The ISO and EIA codes can be selected by MDI setting.

a) EIA code
EIA RS-244-A

b) ISO code
ISO 840

For more detail, refer to Appendix 1.

3.9 Address Codes

Address	Meaning	Section
A	Rotary axis around an axis in parallel with X axis	O
B	Rotary axis around an axis in parallel with Y axis, or table indexing function (B function)	O
C	Rotary axis around an axis in parallel with Z axis	O
D	Cutter compensation number selection	B
F	Feed function (F function)	B
G	Preparatory function (G function)	B
H	Tool offset number selection	B
I	X-axis coordinate of arc center, radius of circle cutting and helical interpolation A, vector of cutter compensation	O
J	Y-axis coordinate of arc center, vector of cutter compensation	O
K	Z-axis coordinate of arc center, lead of helical interpolation A, vector of cutter compensation	O

Address	Meaning	Section
L	Frequency number of canned cycle, Repeat number of helical interpolation	O
M	Miscellaneous function (M function)	B
N	Sequence number	B
O	Same with N (in EIA code only)	B
P	Dwell, starting block of programming copy and sequence number of sub tape control	O
Q	Step distance of shift value in canned cycle and sequence number of end block of programming copy	O
R	Point R in canned cycle and repeating times of programming copy or sub tape	O
S	Spindle-speed function (S function)	O
T	Tool function (T function)	O
U	Additional axis parallel with X axis	O
V	Additional axis parallel with Y axis	O
W	Additional axis parallel with Z axis	O
X	X axis	B
Y	Y axis	B
Z	Z axis	B

Section: B indicates a function of the basic control unit.

O indicates a function of an additional option.

Note) Address D and H are equivalent code,

so address D can be used instead of address H.

Selectable number of H and D code are up to 32. And this can be extended up to 64 using additional option.

3.10 Function Codes

EIA	ISO	Meaning
Blank	NUL	In EIA code, blank within significant information is detected as an error. In ISO code, NUL is ignored at all times.
BS	BS	Ignored
Tab	HT	Ignored
CR	LF/NL	End of block
	CR	Ignored
SP	SP	Ignored
ER	%	Absolute rewind stop & reset
	(	Control out
	)	Control in
+	+	Ignored
-	-	Negative sign
0 to 9	0 to 9	Numerals
a to z	A to Z	Alphabetic letters
o	:	Assumed to be N in forward reading. Ignored during rewind
/	/	Optional block skip
Del	DEL	Ignored

(1) Label Skip Function

If a tape is read in with the Label Skip Function on, all information is ignored until the End of Block code (EOB) is read. The information until a reset command is read after the first EOB code has been read, is called the Tape Significant Information. And at this state Label skip lamp turns off.
The label skip function becomes on and the lamp on the display panel lights up when
a) the power is turned on, or
b) the equipment is reset in a mode other than MDI mode.

(2) Control Out/In (Only at ISO code)

All information including EOB between left parenthesis and right parenthesis code is ignored, and TH and TV checks are not performed. When the power is turned on, or when the equipment is reset, Control In is effected. However Absolute Rewind Stop and Reset is not used in this section. And EOB code has no effects in this section.

(3) Absolute Rewind Stop and Reset

The code means an absolute rewind stop, and in addition, when this code is read within Significant Information, it means a reset. If this code is read during manual operation on the tape reader with reels, the tape feed is stopped unconditionally.

3.11 Feed Function

3.11.1 Rapid Traverse Speed

The axes will move at an internally preset rate of up to 250 kpps. The preset rate can be changed by 1 kpps step independently for each axis. However, the rapid traverse speed is depending on the type of servo motor, the condition at machine, etc.

Increment System	Rapid traverse range
M1V, M2V	60mm/min ~ 15,000mm/min
M3V	30mm/min ~ 7,500mm/min
M4V	12mm/min ~ 3,000mm/min
I1V, I2V	6 inch/min ~ 1,500 inch/min
I3V	3 inch/min ~ 750 inch/min
I4V	1.2inch/min ~ 300 inch/min
D1V, D2V	60 deg/min ~ 15,000 deg/min
D3V	30 deg/min ~ 7,000 deg/min
D4V	12 deg/min ~ 3,000 deg/min

3.11.2 1-Digit F-Code Unit

The feed rate is specified in a 1-digit number, 0 to 8, following the address F. The actual feed rate is set on the dial provided on the front of the control unit.

The Override dial, even if provided, is ineffective when a feed rate is specified on the 1-digit F-code unit.

When F1 is specified on tape, for example, the lamp F1 turns on, and the feed rate becomes the setting value on the dial F1.

(1) Command

F0 ..... Rapid traverse
F1～F8 ..... Feed rate
F9 ..... P/S alarm

(2) Feed rate range

Increment system	Feed rate range
M1V, M2V	5 ～ 1,200mm/min
M3V	2.5 ～ 600mm/min
M4V	1 ～ 240mm/min
I1V, I2V	0.5 ～ 120inch/min
I3V	0.25 ～ 60inch/min
I4V	0.1 ～ 24inch/min
D1V, D2V	5 ～ 1,200deg/min
D3V	2.5 ～ 600deg/min
D4V	1 ～ 240deg/min

(Note) : This value is limited by the gain of the servo system, eventually.

3.11.3 4-Digit F-Code Unit

The feed rate is specified directly in a number code in mm/min or inch/min following address F. The range of feed rates that can be specified is as in Table 3.11.1. The upper limit of the actual feed rate, however, is limited by the type and load condition of the motor used, or the time constant, etc. of the servo system, and is usually lower than these values. Refer to the description issued by the machine tool builder as to what the upper limit is.

Table 3.11.1 F code range

Increment system	Format	F code range
M1V	F40	F01 ~ F3600
M2V	F41	F01 ~ F36000
M3V	F41	F01 ~ F18000
M4V	F41	F01 ~ F 7200
I1V	F41	F01 ~ F 3600
I2V	F32	F01 ~ F 36000
I3V	F32	F01 ~ F 18000
I4V	F32	F01 ~ F 7200
D1V	F40	F01 ~ F 3600
D2V	F41	F01 ~ F36000
D3V	F41	F01 ~ F18000
D4V	F41	F01 ~ F 7200

└Decimal point expressed in mm/min, inch/min or deg/min

An Override can be applied in increments of 10% in the range of 0 to 200% to this feed rate on the FEED RATE OVERRIDE dial on the operator's panel.

With the dial at 100%, the feed rate is the same as specified. In a Fixed Cycle (tapping cycle G84), and with the OVERRIDE CANCEL switch on the operator's panel at ON, however, the Override is ignored (assumed to be 100%).

4-digit F-code should be commanded more than 2-digit in order to distinguish from 1-digit F-code.

Example 3.11.1 Correspondence between the F Codes and Feed Rates

F code	Increment system	Feed rate
F450	M1V	450 mm/min
	M2V~M4V	45.0 mm/min
	I1V	45.0 inch/min
	I2V ~ I4V	4.50 inch/min
	D1V	450 deg/min
	D2V~D4V	45.0 deg/min

Notes on the Feed function

Note 1: The 1-, and 4-Digit F-Code Feed functions can be used mixedly, and are identified by the number of digits of a number code following address F.

(Examples)

F3
- o With the 1-Digit F-Code Feed option, the 1-Digit F-Code Feed (F3) is effected.
- o Without the 1-Digit F-Code Feed option, the 4-Digit F-Code Feed (3 mm/min) is effected. Increment system M1V

F23, F3545 } The 4-Digit F-Code Feed is effected.

Note 2: Feed rate is as follows by G code command.

	G00	G01, G73, G76 G81~G89	G02, G03 G12-G15, G39
F0	RT	RT	P/S Alarm
F1-F8	RT	Cutting feed rate by 1-digit F-code	
4-digit F-code	RT	Cutting feedrate by 4-digit F-code	

RT: Rapid Traverse

Note 3: Acceleration and deceleration are effected automatically by the speed-up circuit near the start and end points of a move command. Refer to Section 3.18 Automatic Acceleration and Deceleration for details.

Note 4: The feed rate of each axis is controlled so that the tangent speed becomes the specified feed rate.

Note 5: The F code is modal, once specified, it is effective until another F code is specified.

3.12 Preparatory Function (G Function)

Table 3.12.1 lists G codes

Table 3.12.1 List of G codes

G code	Group	Function	Section
G00	A	Positioning	B
G01	A	Linear interpolation	B
G02	A	Circular interpolation CW	O
G03	A	Circular interpolation CCW	O
G04	*	Dwell	B
G12	*	Circle cutting CW	O
G13	*	Circle cutting CCW	O
G14	A	Herical interpolation CW	O
G15	A	Herical interpolation CCW	O
G17	B	XY plane selection	O
G18	B	ZX plane selection	O
G19	B	YZ plane selection	O
G25	*	Programming copy	O
G27	*	Zero return check	O
G28	*	Automatic zero return	O
G29	*	Return to zero return start point	O
G38	*	Cutter compensation vector set	O
G39	*	Corner offset	O
G40	C	Cutter compensation cancel	O
G41	C	Cutter compensation left side	O
G42	C	Cutter compensation right side	O
G43	*	Tool offset B + direction	O
G44	*	Tool offset B - direction	O
G45	*	Tool offset increase	B
G46	*	Tool offset decrease	B
G47	*	Tool offset double increase	B
G48	*	Tool offset double decrease	B
G49	*	Tool offset B cancel	O
G73	D	Canned cycle #10	O
G76	D	Canned cycle #11	O
G80	D	Canned cycle cancel	O
G81	D	Canned cycle #1	O
G82	D	Canned cycle #2	O
G83	D	Canned cycle #3	O
G84	D	Canned cycle #4	O
G85	D	Canned cycle #5	O
G86	D	Canned cycle #6	O
G87	D	Canned cycle #7	O
G88	D	Canned cycle #8	O
G89	D	Canned cycle #9	O

G code	Group	Function	Section
G90	E	Absolute programming	B
G91	E	Incremental programming	B
G92	*	Programming of absolute zero point	B
G98	F	Return to initial level in canned cycle	O
G99	F	Return to R point level in canned cycle	O

Notes on G code

(1) Generally speaking, the G code can be specified repeatedly in the same block.

(2) When the G codes belonging to the same group are specified mixedly, the G code specified later is effective.

(3) The G code having the alphabetic letter in Column Group in Table 3.12.1, is modal, and once specified, maintains its function, but is canceled by another G code of the same group. Furthermore, it is not canceled even if the control unit is reset.

(4) The G code having the * mark is not modal, and its function is effective only for the block in which it is specified.

(5) In Column Section of Table 3.12.1, B means Basic, and O means Additional option.

(6) G code's state at Power ON are as follows.

Group	G code
A	G00
B	unassigned
C	G40
D	G80
E	G91
F	G98

(7) When the A-group's G code is commanded during the canned cycle mode, the canned cycle is canceled automaticaly. And the A-group's G-code suffers no effects by the canned cycle's G code.

3.12.1 Positioning (G00)

The command

G00 α —— β —— * (Basic)
G00 α —— β —— γ —— * (Option)

(α , β , γ : X, Y, Z, U, V, W, A, B or C)

specifies positioning to be performed.

Any two axes are controlled at a time. With an additional option, 3 axes simultaneous control is provided.

Since the rapid traverse is made for each axis independently, the tool path does not necessarily become straight. Also, since the rapid traverse speed is set independently for each axis, refer to the description issued by the machine tool builder for that value.

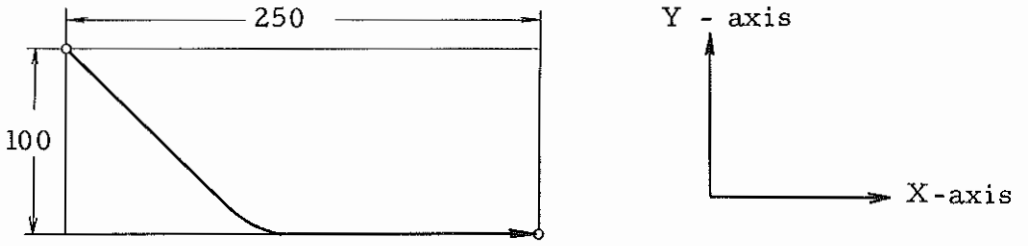
Example 3.12.1 Program example for positioning and tool path

(1) Increment system M1V

Rapid traverse feed rate X axis 9,600mm/min
Y axis 9,600mm/min

Incremental command

Program G00 X 25000 Y-10000 *

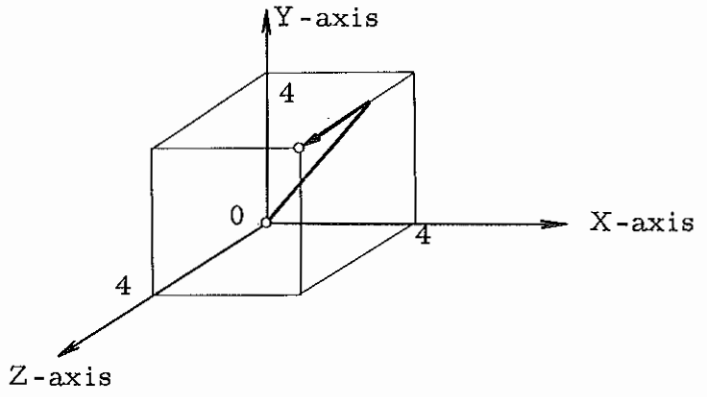


Note ; This command is incremental.

(2) Increment system I1V

Rapid Traverse feed rate X axis 960 ipm
Y axis 960 ipm
Z axis 480 ipm

Program G00 X 4000 Y 4000 Z 4000 *



Notes on positioning

(1) Since generally the tool path is not straight from the start point to the end point, make sure that there is no obstacle thereon. Particularly when a plurality of axes including the rotary axis are specified simultaneously, be careful enough so that the work does not interfere with the tool or other parts of the machine tool.

3.12.2 Linear Interpolation (G01)

The command

G01 α — β — F — * (Basic)
G01 α — β — γ — δ — F — * (Option)

(α, β, γ, δ: X, Y, Z, U, V, W, A, B or C)

Specifies linear interpolation.

Any two axes are controlled at a time. As an additional option, 3 axes or 4 axes control is provided.

The feed rate is specified in an F code.
In controlling the basic three axes (X, Y and Z) meeting at right angles, the feed rate of each axis is controlled so that the tangent speed becomes the specified feed rate. When axes with different increment systems, including rotary axes, move, the least command increments are considered to be equal, the feed rate is converted into the increment system for the basic axis (X axis as a rule), and the move is controlled so that the generalized feed rate F

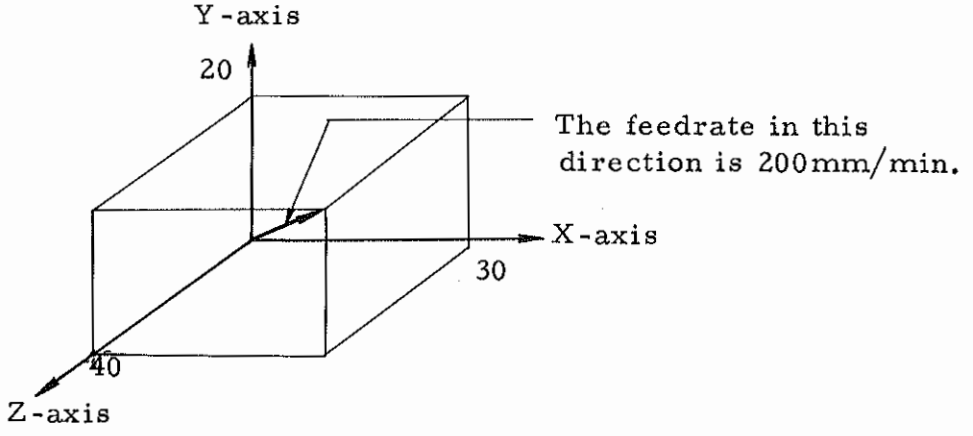
$$F = \sqrt{F_x^2 + F_y^2 + F_z^2 + F_\alpha^2} \qquad (\alpha: \text{U, V, W, A, B or C})$$

becomes the specified feed rate.

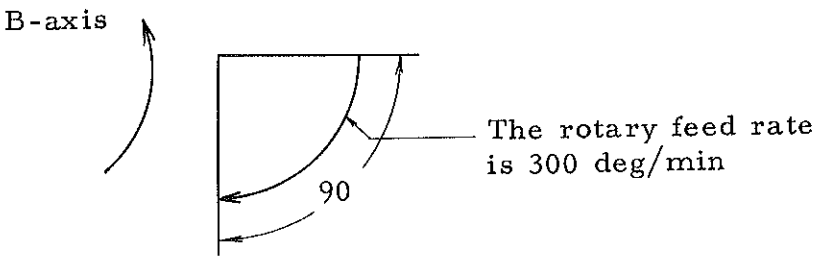
Example 3.12.1

The path, when programmed (in Incremental commands) under the following conditions, is as follows:

(1) Increment system M1V
Program G01 X3000 Y2000 Z4000 F200 *



(2) Increment system D1V (basic axis M1V)
Program G01 B-9000 F300 *



3.12.3 Circular Interpolation (G02, G03)

G02 and G03 specify circular interpolation.

G02 X —— Y —— I —— J —— F —— *
(G03)
G02 X —— Z —— I —— K —— F —— *
(G03)
G02 Y —— Z —— J —— K —— F —— *
(G03)

The plane on which an arc exists is limited to the following three planes, each of which must be specified by the appropriate G code. (Refer to Section 3.12.7 Plane Selection.)

XY plane	G17
ZX plane	G18
YZ plane	G19

Note 1: Plane selection is also selectable by the adress word X, Y, Z, I, J and K, when G17, G18 and G19 have not been commanded after power on;

When (X or I) and (Y or J) are commanded, the plane is regarded as X-Y plane.

When (Y or J) and (Z or K) are commanded, the plane is regarded as Y-Z plane.

When (Z or K) and (X or I) are commanded, the plane is regarded as Z-X plane.

Note 2: When the G code of plane selection is uncommanded and more than three axes address words are commanded or among four axes address words (X, Y, I, J), only one address is commanded, the plane is regarded as X-Y plane;

G02X — Y — Z — I — J — * ⇨ G02X — Y — I — J — *
(G03) (G03)

(The information of Z-axis is ignored.)

G02I — * ⇨ The plane is regarded as X-Y plane.
(G03)

G02K — * ⇨ This command is ignored.
(G03)

The rotary direction of the arc is defined by G02 and G03.

G02	clockwise (CW)
G03	counterclockwise (CCW)

Clockwise and counterclockwise employed here are defined, with respect to XY plane (ZX plane, YZ plane), looking toward the negative direction of Z axis (Y axis, X axis) from the positive direction in a right-hand rectangular coordinate system.
These are shown in Fig. 3.12.3(a).

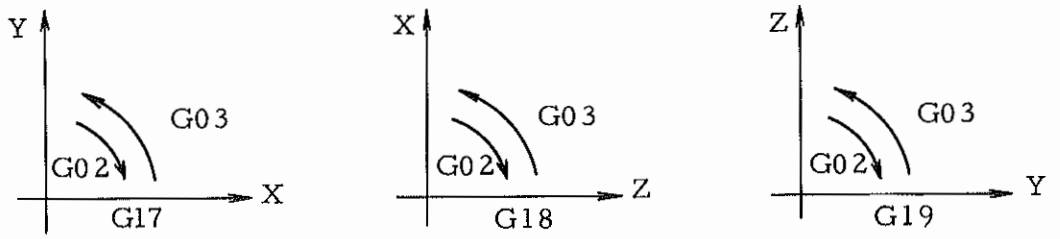


Fig. 3.12.3(a) Clockwise and counterclockwise

The end point of an arc is specified by the address X, Y or Z, and is expressed in an Absolute or Incremental value according to G90 or G91, respectively. In incremental expression, the coordinate of the end point is specified as seen from the start point of the arc. The arc center is specified by the address I, J and K for the axis X, Y and Z, respectively. The number following I, J or K, however, is a vector component looking toward the center of the arc from the start point, and is always specified in an Incremental value, independently of G90 and G91.
These are shown in Fig. 3.12.3(b).

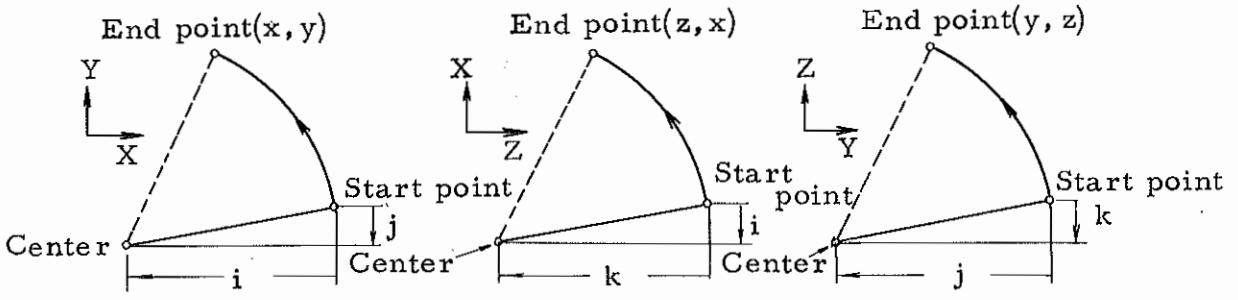
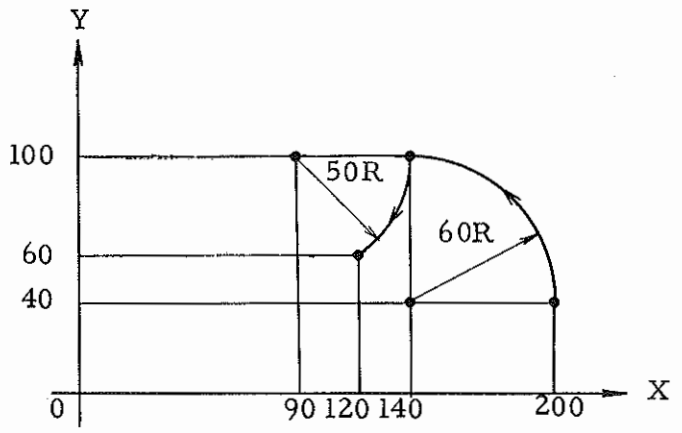


Fig. 3.12.3(b)

The feed rate is specified in an F code.
The actual feed rate is controlled so that the feed rate in the tangent direction is the specified feed rate.

Example 3.12.3(b) Specification in circular interpolation



The above noted path can be programmed in Absolute and Incremental values as follows:

Increment system M1V

a) Absolute programming

```
G92       X20000   Y 4000    Z0              *
G90 G03   X14000   Y10000    I-6000  F300    *
    G02   X12000   Y 6000    I-5000          *
```

b) Incremental programming

```
G91 G03 X-6000 Y 6000 I-6000 F300 *
    G02 X-2000 Y-4000 I-5000      *
```

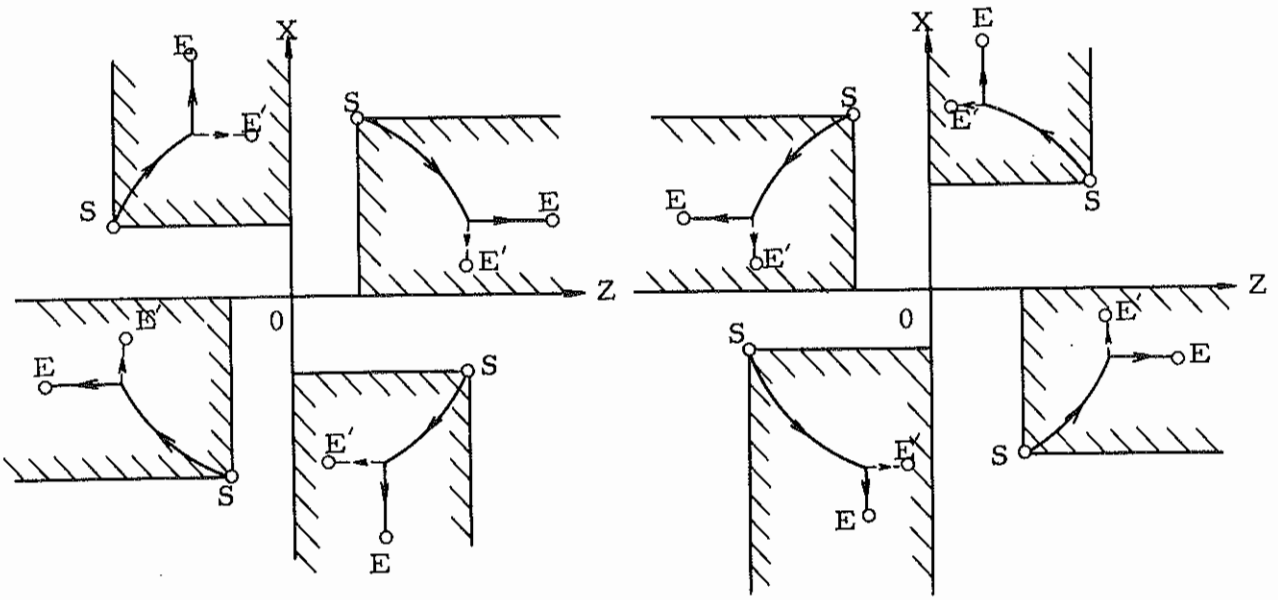
Refer to Section 3.12.14 for Absolute and Incremental programming.

(Note)

In case the end point of circular command is not on the circular arc drawn for the radius at start point, the interpolation is performed according to the following conditions.

In case of G02

In case of G03



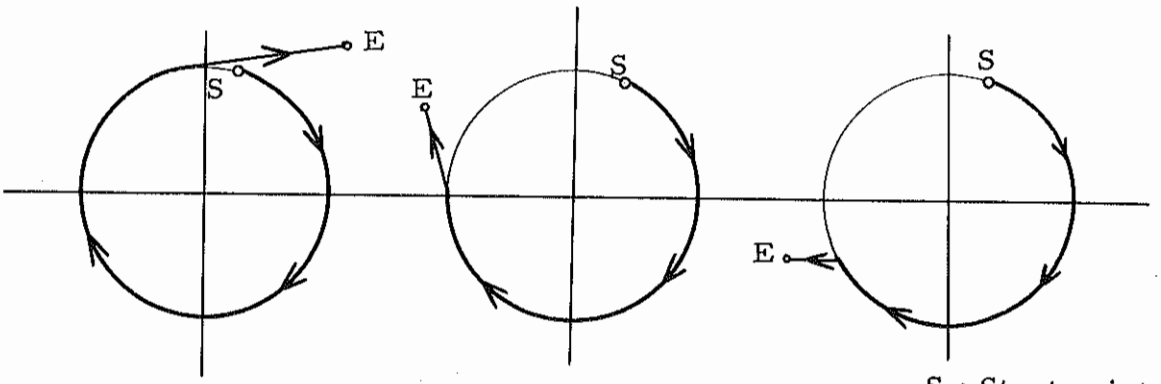
S : Start point

E, E′ : End point

(1) In case the end point is within the oblique line as shown in the above figures, a circular interpolation is performed up to the point where coordinate value of one of two axes on the arc plane coincides with coordinate value of the end point. The rest is distributed by a straight line.

(2) In case the end point is not within the same quadrant or not within the oblique line as shown in the above figures, a multi-quadrant circular interpolation is performed. In this case, the interpolation is performed in every quadrant according to the conditions in item (1). Unless the conditions in item (1) are applied, it is done by a straight line toward the end point in the quadrant in which the end point is.

(Example)



S : Start point

E : End point

3.12.4 Dwell (G04)

The command G04 P —— *
permits the NC command of the next block to be delayed by the time specified in a number code following address P.

The address can, instead of P, be X, Y, Z, F or α (with the axis added). α : U, V, W, A, B, C

Example 3.12.4 2.5 sec dwell

Increment System	NC command	Max. Command Valve
M1V	G04 P250	1048.57 sec.
M2V～M4V	G04 P2500	1048.575 "
I1V	G04 P250	1048.57 "
I2V～I4V	G04 P2500	1048.575 "

Note 1: Since the G04 command is not modal, the Dwell function is effective only for a block in which G04 is specified.

3.12.5 Circle Cutting (G12, G13)

The command

$$\begin{matrix} \text{G12} \\ (\text{G13}) \end{matrix} \text{ I } \text{———} \text{ D — F —} \text{ *}$$

specify circle cutting.

Starting from the center of a circle, the cutter moves and cuts along following circule.

This command is possible on X-Y.

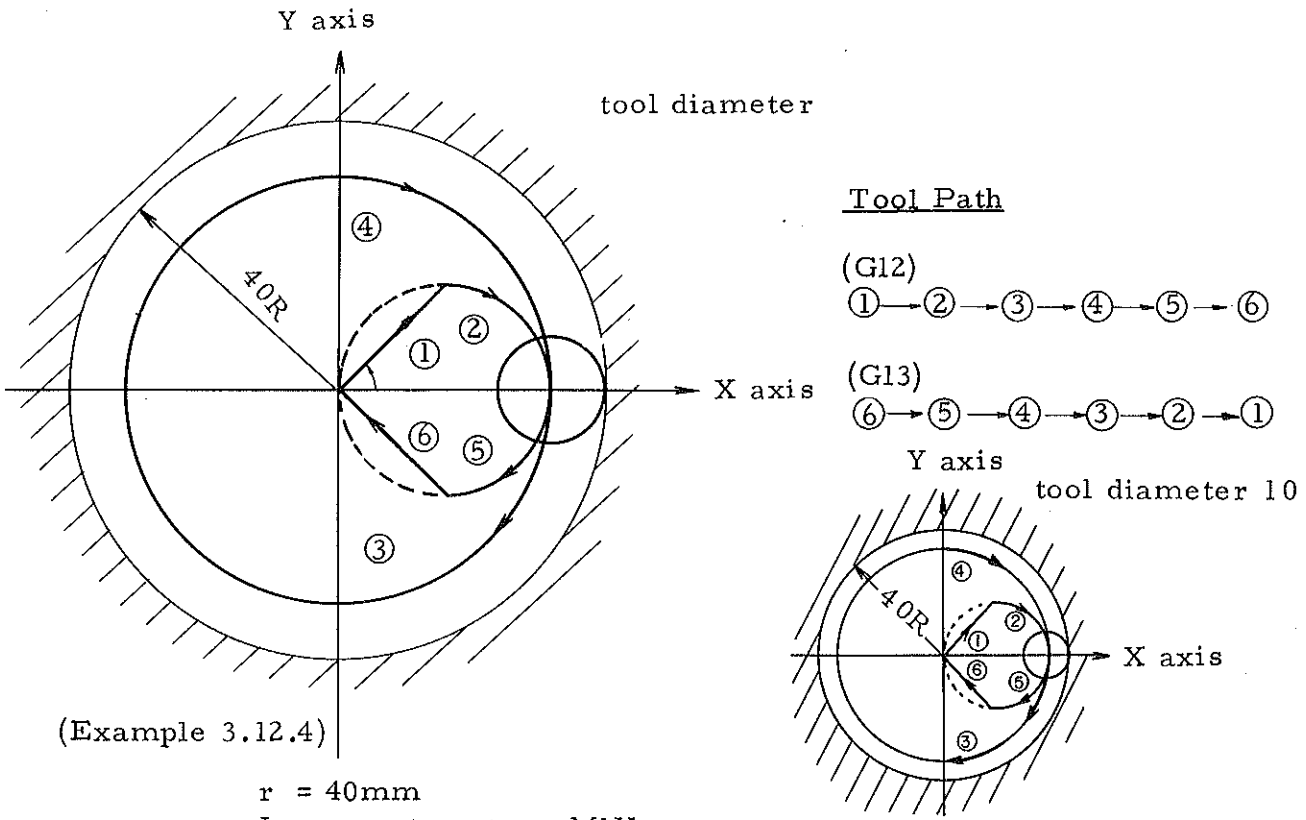
The starting point of machining circle is always on the X-axis independently the sign of address I.

The absolute value of address I's dimension is used as machining radius.

The rotary direction of the arc is defined by G12 and G13.

{ G12: Clockwise (CW)
G13: Counter clockwise (CCW)

Actual radius of a circule is decreased by offset amount.



(Example 3.12.4)

r = 40mm
Increment system: M1V
D (Offset number): 05 (offset amount 500)
NC command:
G12 I 4000 D05F200*

3.12.6 Herical Interpolation A (G14, G15), B (G02, G03)

Helical interpolation means circularly interpolating movement between two axes while simultaneously synchronizing the third axis with the angle of rotation.
Tool paths become spiral-shaped. This feature is most effective for large-diameter milling and cutting 3-dimensional cams.
The helical interpolation instruction is made by adding the traverse command of the third axis which is moving linearly to the commands to the two axes moving in curved paths.

(1) Helical interpolation A

By the 3-axis control of a circular interpolation in X-Y plane and a linear interpolation in Z axis, thread cutting of large diameter, etc can be easily machined.

The command

G14 I —— K —— L —— D —— F —— *
(G15)

G14: Clockwise (CW)
G15: Counter clockwise (CCW)

I : Radius (the value seen from a start point to an arc center)
If the value of I is commanded by the positive number, the helical interpolation begins on the -X axis.
And if the value of I is commanded by the negative number, the helical interpolation begins on the +X axis.

K : Lead (sign is plus at the time of threding to the positive direction of Z axis, minus is negative direction.)

L : Repeat number of times (positive number)

D : Tool offset number
If a tool offset amount is positive, an offset in made inside a circle, and if a tool offset amount is negative, an offset is made outside a circle.

F : Feed rate

Notes on the Herical interpolation

Note 1: This function can be only performed as following conditions.

- o X-Y plane : Circular interpolation
- o Z axis : : Linear interpolation

Note 2: Set the start point to the position on the +X axis.

Note 3: The feed rate designated F code is circular interpolation feed rate X-Y plane.

Note 4: When the repeat number of times doesn't become integer, this number can be set to integer by adjustment of Z-axis coordinate of start point. And the multiple thread screw cutting can be performed by this method.

Note 5: The relation between radius "r" of tool path and lead "K" must be satisfied as following condition.

$$K \leqq \frac{9}{10} r$$

Note 6: The lead range is as follows

$K \leqq 1646000p$ (p: Least command increment)

(2) Helical interpolation B

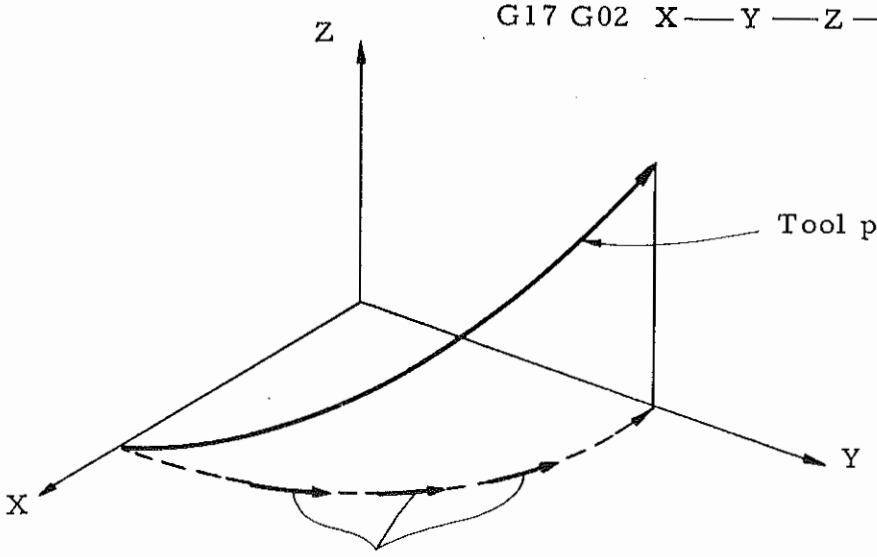
Necessary linear commands can be added to circular interpolation commands, effective within a 360 degree arc. The G02/G03 command (I, J, K) is used for the circular interpolation instruction. The X, Y, and Z axes specify the traverse of each axis, and are commanded by using the absolute or incremental programming in accordance with the G90/G91 command status at that time.
Whether any of the X, Y, and Z axes is circularly or linearly interpolated depends upon G17, G18, and G19.

$$\left.\begin{matrix} G17 \\ G18 \\ G19 \end{matrix}\right\} \left.\begin{matrix} G02 \\ G03 \end{matrix}\right\} \quad X\text{——}Y\text{——}Z\text{—}\left\{\begin{matrix} I\text{——}J\text{——} \\ I\text{——}K\text{——} \\ J\text{——}K\text{——} \end{matrix}\right\} F\text{—}*$$

G17 ..... Arc in the XY plane with Z axis linear movement.
G18 ..... Arc in the ZX plane with Y axis linear movement.
G19 ..... Arc in the YZ plane with X axis linear movement.

The helical interpolation feed rate becomes the cutting feed rate specified by the F code, and is controlled so that the circumferential velocity (tangential velocity) of the circularly interpolated two axes becomes the specified feed rate.

G17 G02 X—Y—Z—I—J—*



The tangential velocity of the circularly interpolated two axes becomes the specified velocity.

Restrictions on Helical Interpolation B

(a) Maximum command value

Circular traverse ..... Equal to the maximum value of the circular interpolation instruction.

Linear axis traverse ..... The following two conditions must be satisfied.

$\alpha \leqq 8\pi r$ (r : Radius* of circle after offset)

$\alpha \leqq 1647096P$ (P : Least command increment)

Where, α is the linear axis traverse in which the arc rotates through an angle of 360 degrees.

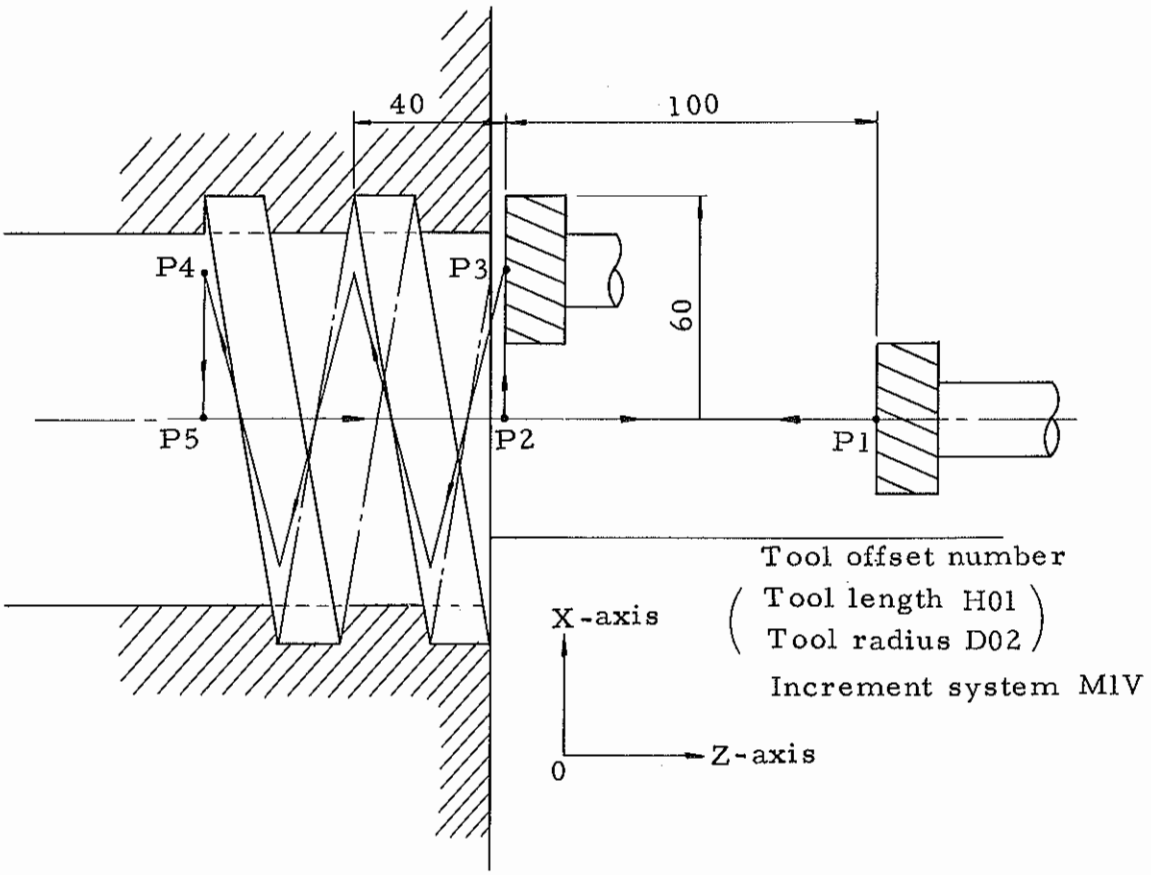
* Distance from the center of the arc to the tool center.

(b) Maximum feed rate

Least command increment × 60 kpps

(c) Offset is applied to the circular movement in the same way as in normal operation.
Linear movement is made while moving the offset arc.

Example 3.12.6 Example of helical interpolation A



Tool path P1 → P2 → P3 → P4 → P5 → P2 → P1

(Program example)

N123 G00 G45 G91 Z-10000 H01 *	P1 → P2	Approach
N124 G46 X 6000 D02 *	P2 → P3	Positioning to start point of machining
N125 G15 I-6000K-4000 1 L2 F150*	P3 → P4	Helical interpolation
N126 G00 G46 X-6000 *	P4 → P5	Positioning to center point "P5".
N127 Z 18000 H01 *	P5 → P2 → P1	Return to start point

Tool offset amount is programmed by positive number for male screw machining and negative number is for female screw machining.

3.12.7 Plane Selection (G17, G18, G19)

The plane on which circular interpolation is made, and the plane on which offset is made, are specified by G17, G18 and G19.

XY plane	G17
ZY plane	G18
YZ plane	G19

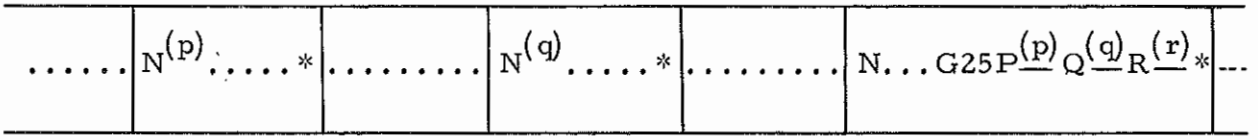
Note: Move command is indifferent to plane selection of G17, G18 and G19. For example Z-axis moves by the command G17 Z —— *

3.12.8 Programming Copy (G25)

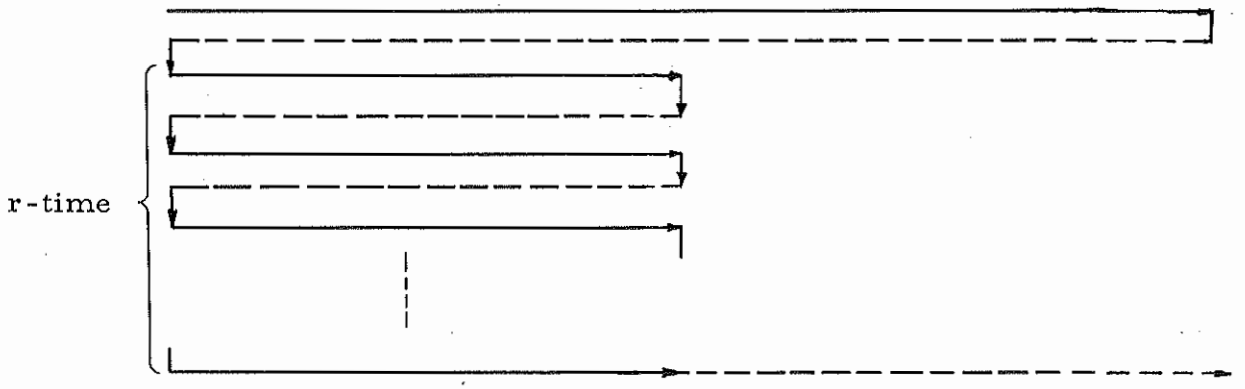
This function is that by designating a part of NC commands and rewinding the tape, that part is executed repeatedly (sometimes once).

The command N——G25 P(p)—— Q(q)—— R(r)—— *

specifies the NC commands of the Sequence Number p to q to be executed r times. Thereafter, execution is continued with the block next to these NC commands. That is, the tape is read in as shown in Fig. 3.12.8



Advancement of NC tape



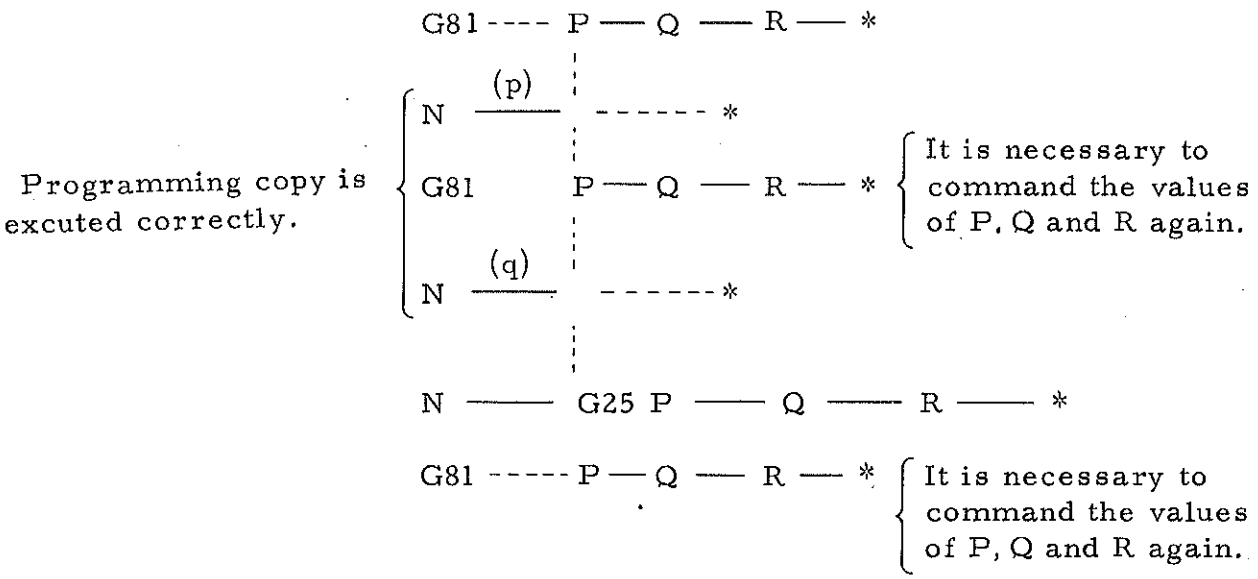
——→ .... These commands are executed
- - -→ .... NC tape is advanced, but no commands are excuted.

Fig. 3.12.8 Advancement of NC tape by Programming Copy

The Sequence Number must always be specified in the block of tape in which Programming Copy is specified. Further-more, the same tape cannot include the same Sequence Numbers, and Programming Copy cannot be specified from the Sequence Number p to q of the tape. In sub tape program, this programming copy is not available.

Note 1: When programming copy is commanded during fixed cycle mode, modal values of P, Q and R of a fixed cycle is destroyed by the block including G25. So it is necessary to command the values of R, Q and R again used by fixed cycle.

On the other hand, when the values of P, Q and R of the fixed cycle are commanded at the programming copy mode, programming copy is excuted correctly.



Note 2 : About modal data at the end of programming copy. When the programming copy finished by the next command,

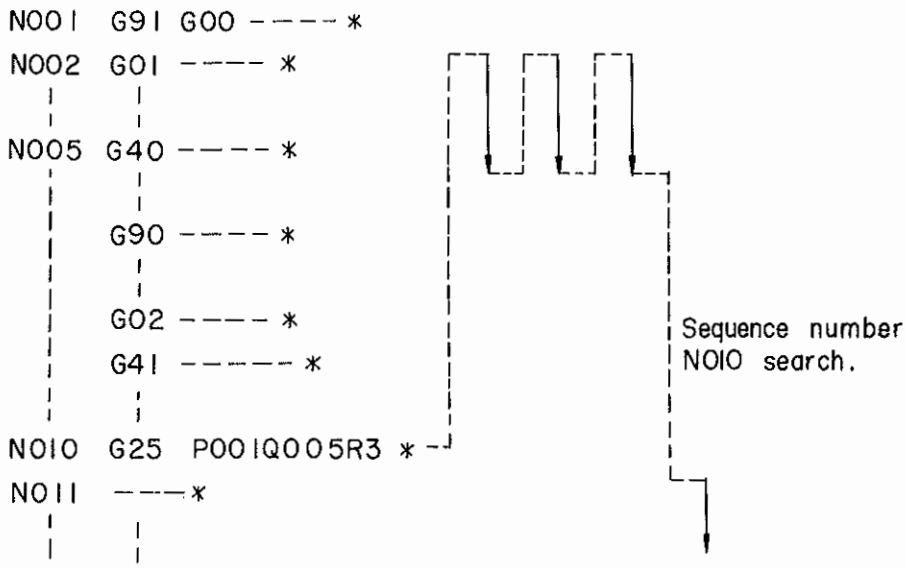
N (n) G25P (p) Q (q) R (r) *

sequence number search is made for the block N(n) to return the next block in which programming copy was commanded.

By this sequence number search, modal data are stored in memory during sequence number search. So, unexpectedly, modal data of the next block to N(n) differs from that of the block N(q) which finished

the programming copy.

For example at the next program



after excution of N005, the data is G91, G01, G40.

But the sequence number search is excuted thereafter, so at the N011 block, the data is G90, G02, G41.

By the above notation, modal data should be programmed again in the next block of the programming copy (in the N011 block for this example).

Note 3 : About the fixed cycles during programming copy

In the last block of the programming copy (block N (q)) fixed cycles must not be commanded.

If the state is fixed cycles mode, the block without move command for X, Y, Z and 4th axis (for example Dwell) should be commanded at N(q) block.

3.12.9 Zero Return (G27, G28, G29)

(1) Zero Return check (G27)

This is a function of checking if a program specifying a machine tool to start at and return to the zero point. This is executed as such.

The axis specified in the command

G27 α ____ β ____ γ ____ *

beings to move in Rapid Traverse, and at the same time the Zero Return sequence is executed. However, the movement amount is the value specified in the program, and even if the machine tool reaches the machine zero point, it is not stopped forcedly, but moves up to the programmed end point. When this end point coincides with the machine zero point, the Zero Return is displayed (lamp display, or the like). Thereafter, if an MDI command or the Signal Block mode has not been selected, the next NC tape is read in for continuing operation. In order to make the machine tool stop at the zero point, a Miscellaneous function (M00, M01, M02 or M30) must have been put in the same block or the next one. Tool compensation cannot be canceled at G 27, so tool compensation should be canceled before G27 command.

(2) ATC Zero Return (G28)

The command G28 α ____ β ____ γ ____ *

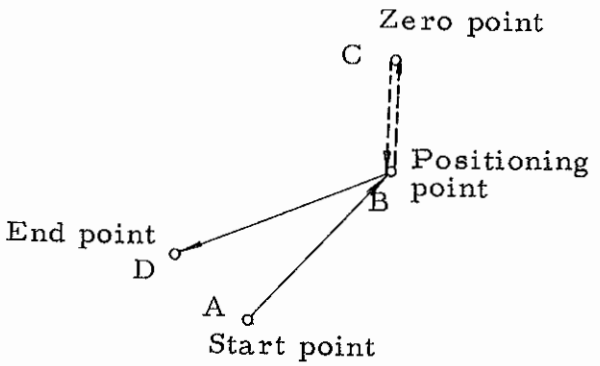
permits Zero Return for designeted axes to be performed automatically after performing positioning with the dimentions specified in α, β and γ .

On completion of the Zero Return, the next block command is executed. Tool compensation cannot be cancelled at G28, so tool compensation should be canceled before G28 command.

(3) Return to Zero Return start point (G29)

The command G29 α ____ β ____ γ ____ *

permits positioning to be performed from the zero point to the positioning point specified in the addresses α , β and γ in G28, and further positioning to be performed to the point specified in the α , β and γ .



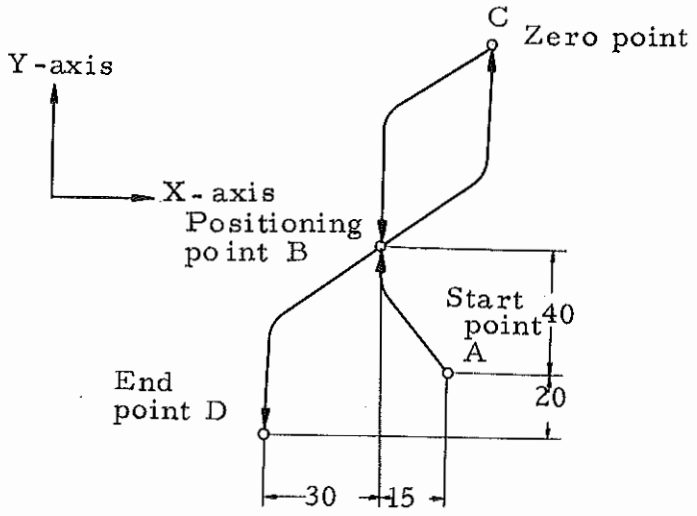
Program

G28X__ Y__ * A to B to C

G29X__ Y__ * C to B to D

This positioning point B must be a position in which Zero Return is possible (refer to Section 4.7.12 ZERO RETURN Switch). The zero point C is usually a position in which ATC (Automatic Tool Change) is possible. Consequently, programming does not require considering the dimension from point B to point C.

Example 3.12.9 Program example for ATC Zero Return and tool path



Tool change is performed at zero point C.

Increment system M1V

A to B Positioning
B to C Zero Return
C to B Return to start point
B to D Positioning

Program example (In case of incremental programming)

```
    :
G28X-1500  Y4000 *     Point A to B to C
M06 *                  Tool change
G29X-3000  Y-6000*     Point C to B to D
    :
```

The command of coordinate system setting or zero setting cannot be programmed between G28 and G29 block.

Note: In G28, G29 command, each axis moves at the rapid speed, so a tool path does not generally straight.

3.12.10 Cutter Compensation (G38 to G42)

This function permits a tool to be offset by the tool radius value given, that is, to pass through the offset path. This Offset command is specified by a G-code function on NC tape or on the MDI unit. The Offset amount (tool radius value) is previously stored in the memory correspondingly to the D code on the MDI unit. This memory is called the Offset amount memory, and it can be stored of maximum 32 or 64 kinds (option) offset value.

A memory address is specified by the D code or H code on the Offset amount memory.

Both D code and H code are modal.

The G functions related to this Offset are as in Table 3.12.10.

Table 3.12.10(a) G functions related to Cutter Compensation

G code	Group	Function
G38	*	Cutter compensation vector set
G39	*	Corner offset
G40	C	Cutter compensation cancel
G41	C	Cutter compensation left side
G42	C	Cutter compensation right side

If G41 or G42 is specified, the equipment is said to be in the Offset mode; if G40, in the cancel mode. When the power is turned on, and when the equipment is cleared, it is in the cancel mode. These Cutter Compensation modes are not affected by the non-modal G function (G38, G39).
G41 and G42 belong to group C. These codes are specified mixedly with G00, G01, G02 and G03, and cooperate with each other to define a mode related to tool movement (Offset).

The program always terminates with the cancel mode.

(1) Offset vector

In Fig. 3.12.10(a), in order to cut a work shaped as indicated by A with an R-radius tool, the path for the center of the tool to pass through must be the figure B which is separate R from the figure A. The tool being separate some distance like this, is called Offset. That is, the figure B is the path which is offset R from the figure A.

The Offset vector is a 2-dimensional vector equivalent in size to a specified Offset amount, and this value is kept stored in the control unit, and its direction is rewritten from moment to moment as cutter advances. This Offset vector (called simply vector later) is, in order to know how much Offset is to be made in the direction of a tool, etc., created in the control unit, and is used to calculate a path offset by the tool radius from a given figure.

This vector always belongs to and follows the tool as it advances, and it is very important in programming to know the behavior of the vector.

As shown in Fig. 3.12.10(a) the vector is usually at right angles with the direction in which a tool advances, and faces toward the tool center as seen from the work.

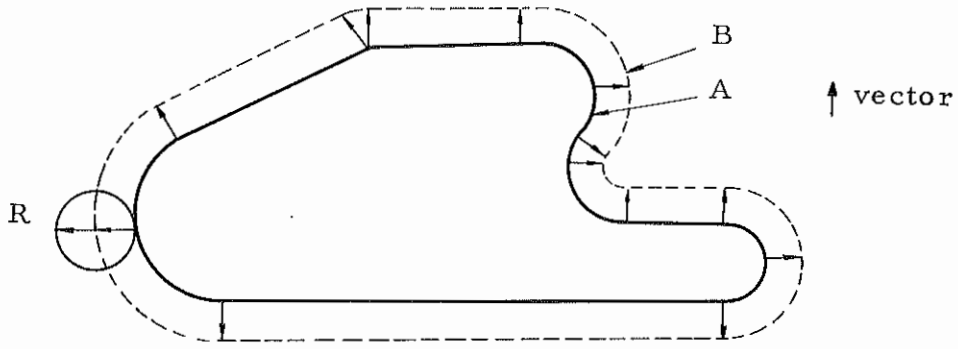


Fig. 3.12.10(a) Offset and vector

(2) Plane selection and vector

Offset calculation is made on a plane specified in plane selection. On an XY plane selected, for example, (X, Y) or (I, J) on a command tape is used to calculate the Offset to create a vector. An axis coordinate value outside the plane specified is not affected by Offset, but the value specified on the command tape is used as it is. In plane switching in the Offset mode, although the offset calculation of the selected new plane is not made, the offset vector of the old plane remains. So selecting the old plane again, offset calculation begins again using the offset vector of the old plane.

In the sequel, what vector is created, and what Offset calculation is made, by an Offset command, will be discussed on assumption that an XY plane is selected. This discussion applies also when another plane is selected.

Note: Plane selection is designeted by G17, G18 and G19. If these G code is unassigned, G17 code is selected.

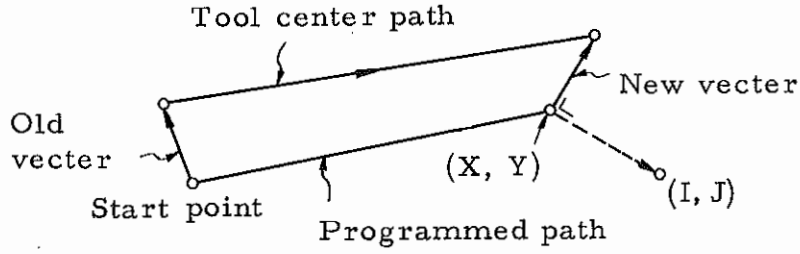
(3) Cutter Compensation left (G41)

(a) Case of G00, G01

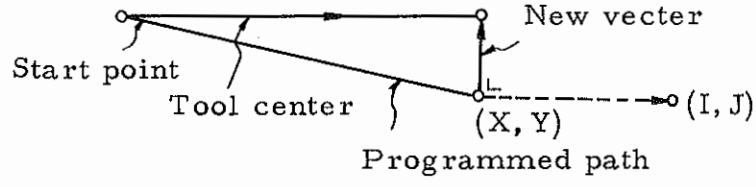
The command G41X ——Y —— I —— J —— D —— *

specifies a new vector to be created at right angles with the direction of (I, J) on the end point, and the tool center moves toward the point of the new vector from that of the old vector on the start point.

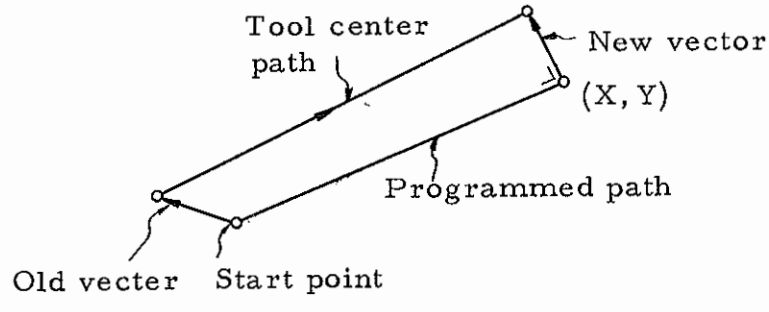
(I, J) is expressed in an Incremental value from the end point, and is significant only as a direction, and its amount is arbitrary.



In case the Old vector is zero, this command specifies the equipment to enter from the cancel mode into the Cutter Compensation mode. At this time, the Cutter Compensation amount memory address is specified by the D code.



If I, J or K is omitted, (I, J) is assumed to be the same as (X, Y). That is, the command G41 X —— Y —— * specifies a new vector at right angles with (X, Y) to be created.



If, however, G00 is specified, each axis moves independently at the Rapid Traverse speed.

(b) Case of G02, G03

The command
G41 ... *

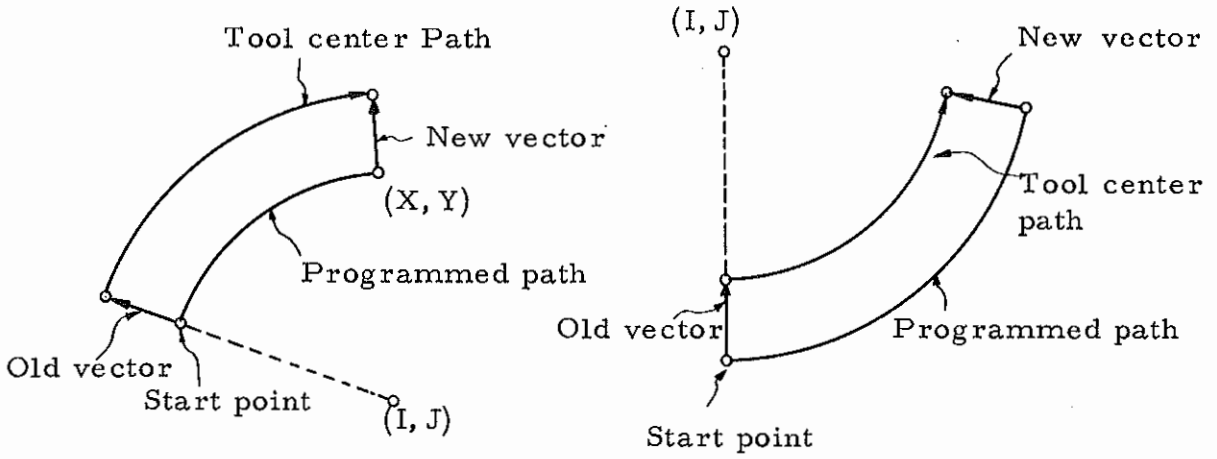
⋮

⋮

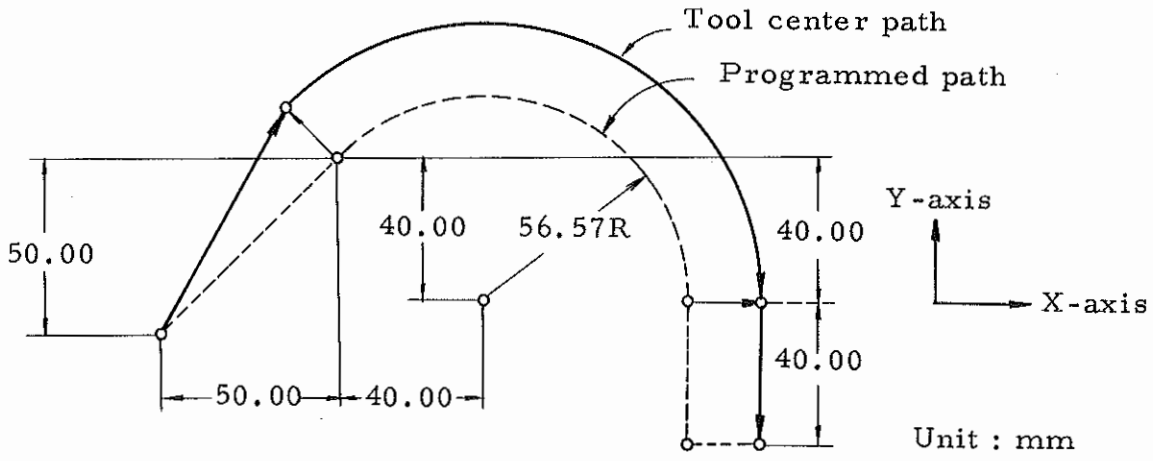
G02 (G03) X —— Y —— I —— J —— *

specifies a new vecter to be created to the left looking toward the direction in which an arc advances on a line connecting the arc center and the arc end point, and the tool center to move along the arc advancing from the point of the old vector on the arc start point toward that of the new vector. This is, however, established on assumption the old vector is created correctly.

In this case, (I, J) is specified as the arc center.



Example 3.12.10(a) Increment system M1V




```
G91 G17 G41 G01X5000 Y5000F150D06 *
    G02 X9657Y-4000 I4000 J-4000 *
    G01 Y-4000 *
```

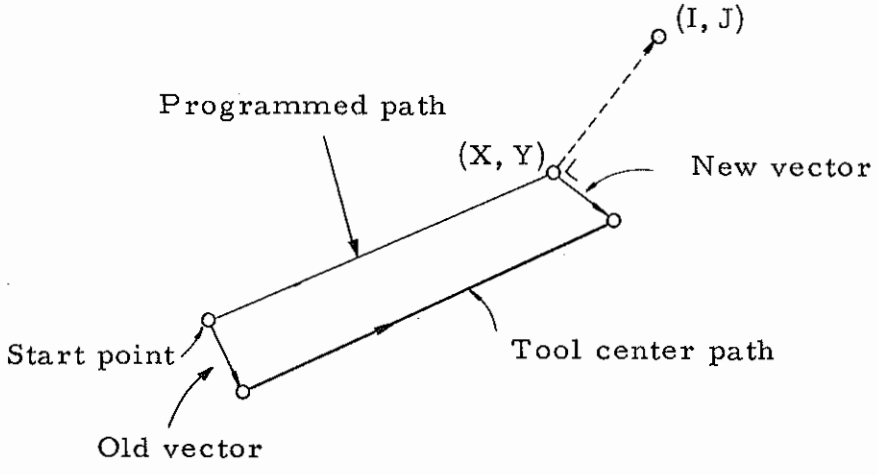
(Incremental command)

(4) Cutter Compensation right (G42)

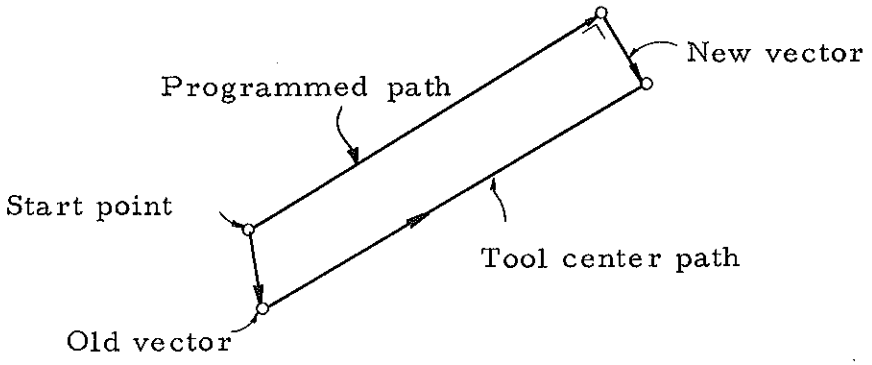
G42, contrary to G41, specifies a tool to be offset to the right of a work looking toward the direction in which the tool advances. That is, the vector created by G42 is toward the direction opposite to that created by G41. Except for this, the Offset method is quite the same as in the case of G41.

(a) Case of G00, G01

G42X —— Y —— I —— J —— *

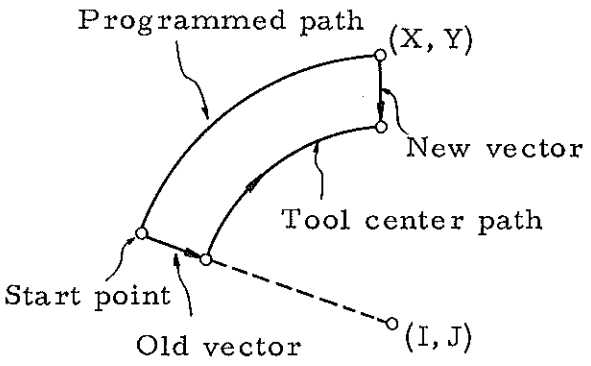


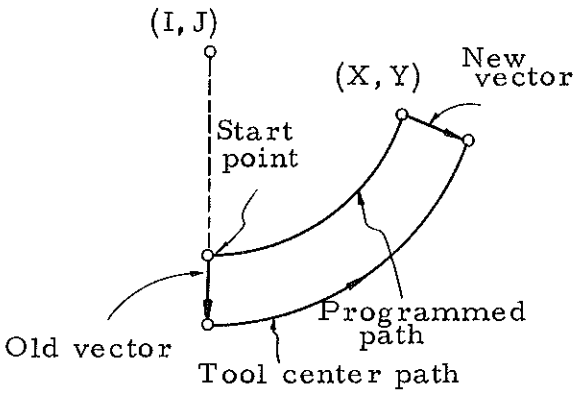
G42X ___ Y ___ *



In the case of G00, however, each axis moves independently at the Rapid Traverse speed.

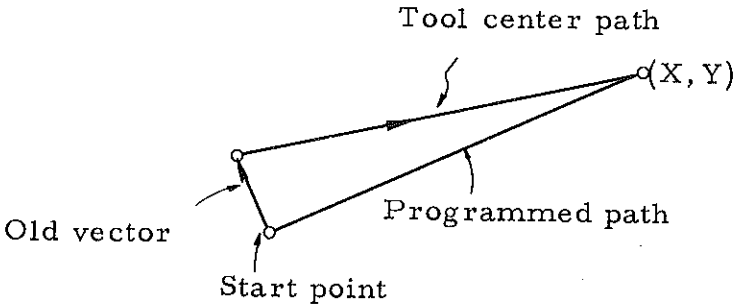
(b) Case of G02, G03





(5) Cutter Compensation cancel (G40)

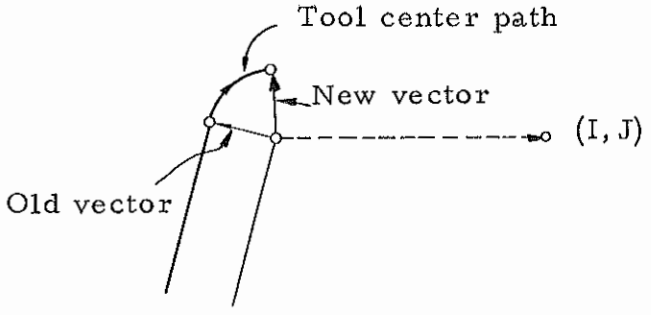
The command G40X ___ Y ___ * in G00 or G01 specifies each axis to move straightly from the point of the old vector on the start point toward the end point in G01 mode and each axis to move in Rapid Traverse in G00 mode. This command specifies the equipment to enter from the Cutter Compensation mode into the cancel mode.



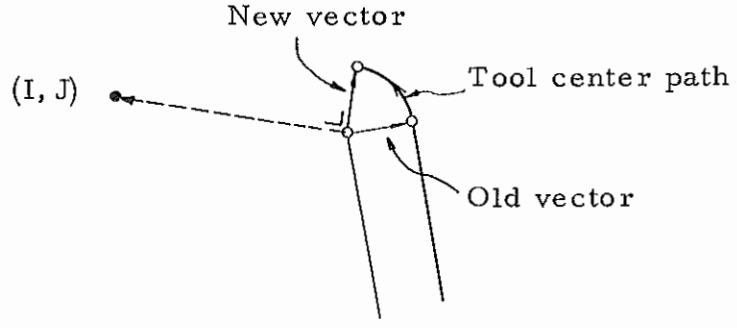
When only G40 * is specified, and X —— Y —— is not specified, the tool moves the old vector amount in the opposite direction.

(6) Corner offset Circular interpolation (G39)

The G39I —— J —— * in G01, G02 or G03 mode permits executing Offset circular interpolation with the tool radius at a corner. A new vector is created to the left (G41) or to the right (G42) looking toward (I, J) from the end point at right angles therewith, and the tool moves along the arc from the point of the old vector toward that of the new vector. (I, J) is expressed in an Incremental value from the end point.



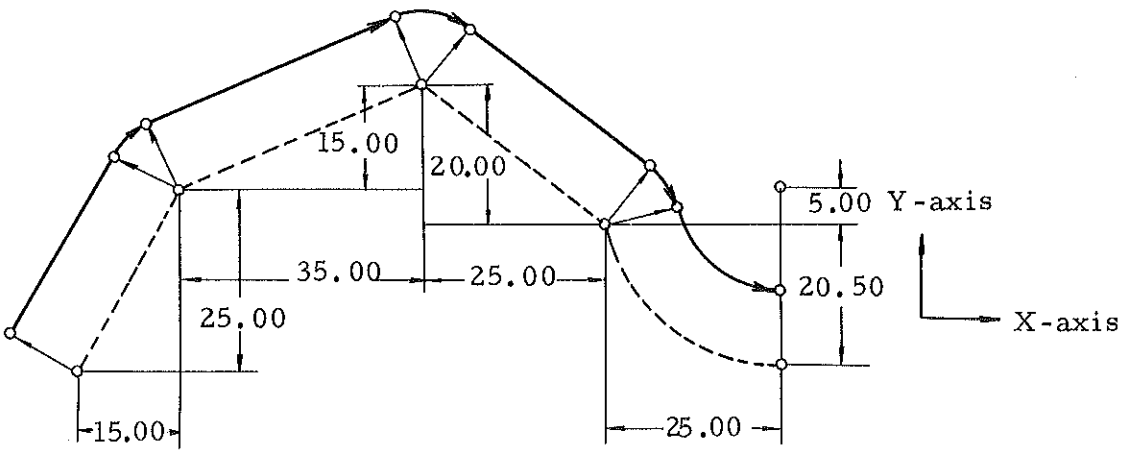
Case of G41



Case of G42

This command can be given in Offset mode, that is, only when G41 or G42 has already been specified. Whether the arc is to turn clockwise or counterclockwise, is defined by G41 or G42, respectively. This command is not modal, and executes circular interpolation, whatever the G function of group A may be. The G function of group A remains by this command.

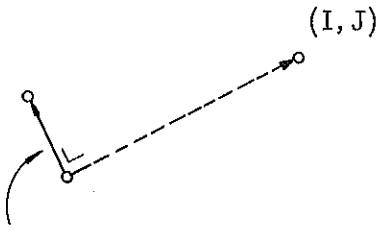
Example 3.12.10(b) Increment system M1V




```
G91 G17 G41 G01X1500Y2500F180D06*
G39I3500J1500*
    X3500Y1500*
G39I2500J-2000*
    X2500Y-2000*
G39I 500J-2500*
G03X2500Y-2050I2500J500*  (Incremental command)
```

(7) Vector rewriting (G38)

The command G38 I—— J —— D—— * in G00, G01, G02 or G03 mode
specifies a new vector to be created to the left (G41) or to the right (G42) looking toward the (I, J) at right angles therewith. The tool does not move, but a vector is only created.



New vector

This command can be given only in Offset mode, resulting in the equipment being still in Cutter Compensation mode.

(8) General notes on Offset

(a) Specification of the Offset amount memory
The Offset amount memory is specified by the D code or H code, and can be specified anywhere before the first change from the Offset cancel mode to the Cutter Compensation mode. Also, once specified, the Offset amount does not need to be specified again, unless it has to be changed.

(b) Change from the Offset cancel mode to the cutter Compensation mode

A move command at the time of change from the Offset cancel mode to the Cutter Compensation mode, must be positioning (G00) or linear interpolation (G01). The circular interpolation (G02. G03) cannot be used.

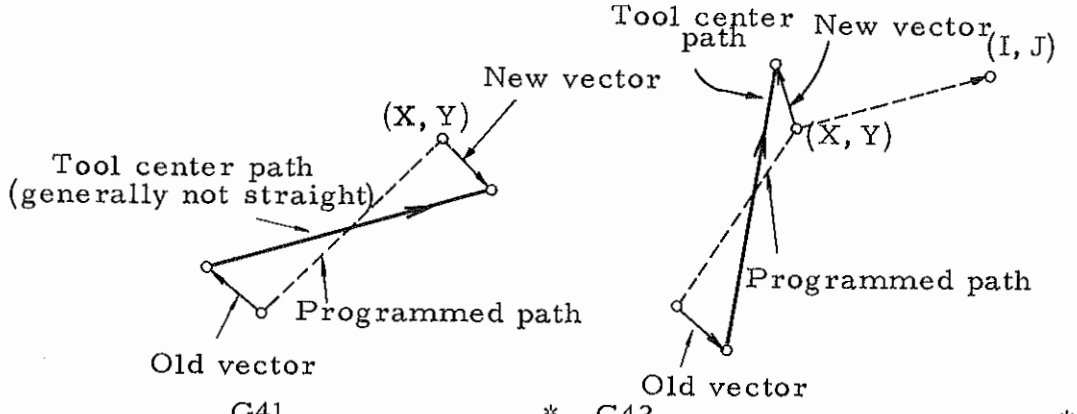
(c) Change from the Cutter Compensation mode to the Offset cancel mode

A move command at the time of change from the Cutter Compensation mode to the Offset cancel mode, must be positioning (G00) or linear interpolation (G01). The circular interpolation (G02, G03) cannot be used.

(d) Switch between Cutter Compensation left and Cutter Compensation right

The Offset direction is switched from left to right, or from right to left generally through the Offset cancel mode, but can be switch not through it only in positioning (G00) or linear interpolation (G01).

In this case, the tool path is as in Fig. 3.12.10(b)



G41 ..... * G42 ..... *
G00 G42 X ——Y —— * G01 G41X— Y —I —J— F— *

Fig. 3.12.10(b) Switch of the Offset Direction

(e) Change of the Offset amount

The Offset amount is changed generally when the tool is changed in the Offset cancel mode, but can be changed in the Offset mode only in positioning (G00) or linear interpolation (G01).

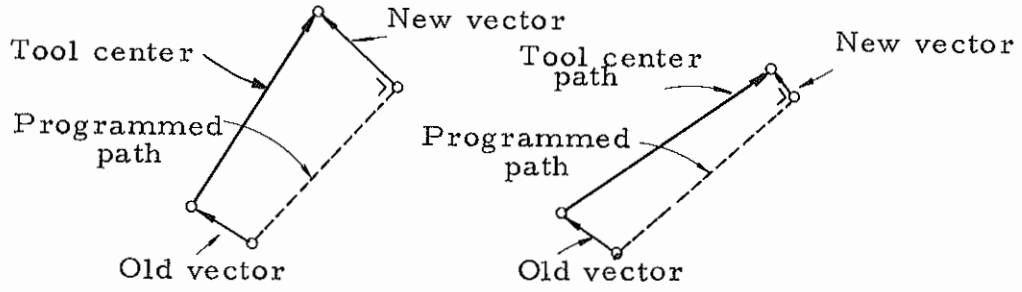


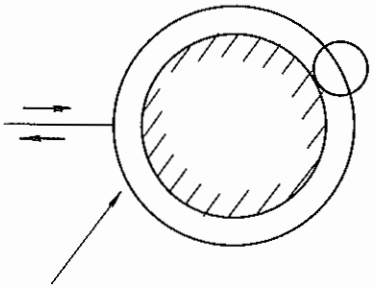
Fig. 3.12.10(c) Change of the Offset amount

Table 3.12.10(b) Tool length/offset amount

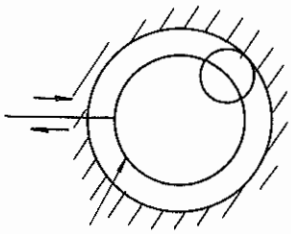
Increment system	Maximum command value
M1V	± 8388.60 mm
M2V M4V	± 999.999 mm
I1V	± 838.860 inch
I2V I4V	± 99.9999 inch

(f) Positive/negative Offset amount and tool center path

If the Offset amount is made negative (-), distribution is made for a figure in which G41's and G42's are all replaced with each other in the process sheet. Consequently, if the tool center is passing around the outside of the work, it will pass around the inside thereof, and vise versa.
Fig. 3.12.10(d) shows one example. Generally speaking, the Offset amount is programmed to be positive (+).
When a tool path is programmed as in (a), if the Offset amount is made negative (-), the tool center moves as in (b), and vise versa. Consequently, the same tape permits cutting both male and female shapes, and any gap between them can be selected.



(a) Tool center path



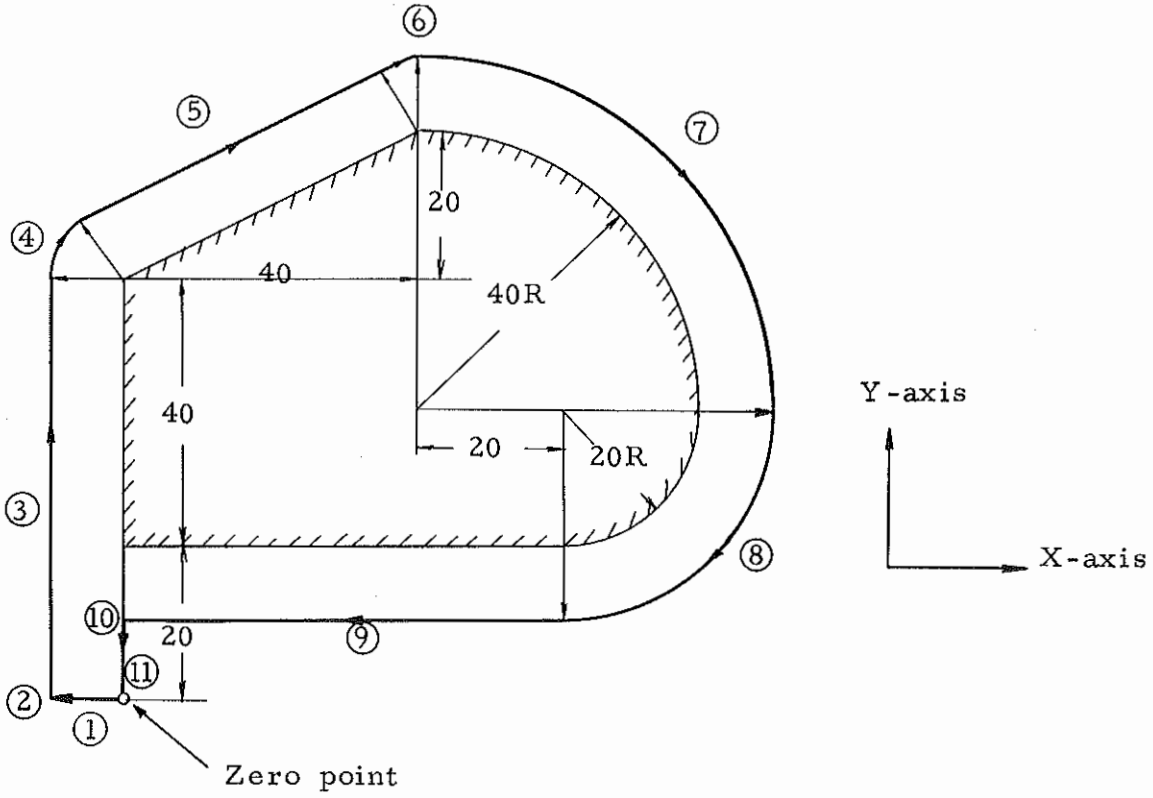
(b) Tool center path

Fig. 3.12.10(d) Tool center path by making the Offset amount positive or negative

For a cornered figure (involved in corner circular interpolation) in general, the Offset amount naturally cannot be made negative (-) to cut the inside. In order to cut the inside corner of a cornered figure, an arc with an appropriate radius must be inserted there to provide smooth cutting.

(9) Examples of the use of Offset

Example 3.12.10(c)

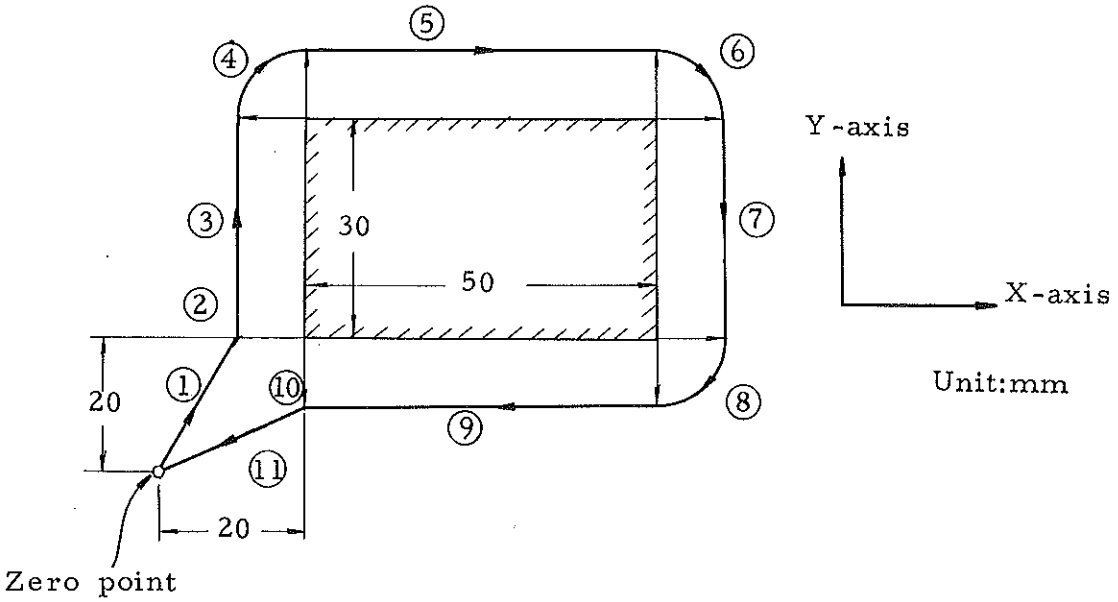


Increment system: M1V

① G91G17G00G41J6000D08 *
(D08 is an Offset number, and a tool radius value corresponding thereto is specified previously on the MDI unit.)

② G18 G01Z-2500F100 *
③ G17 Y6000F250 *
④ G39I4000J2000 *
⑤ X4000Y2000 *
⑥ G39I4000 *
⑦ G02X4000Y-4000J-4000 *
⑧ X-2000Y-2000I-2000 *
⑨ G01X-6000 *
⑩ G18 G00Z2500 *
⑪ G17 G40Y-2000M02 *

Example 3.12.10(d).



Increment system : MIV

① G91G17G00G41X2000Y2000J3000D06 *
(D06 is an Offset number, and tool radius value corresponding thereto is specified previously on the MDI unit.)

② G18 G01Z-2500F100 *
③ G17 Y3000F250 *
④ G39I5000 *
⑤ X5000 *
⑥ G39J-3000 *
⑦ Y-3000 *
⑧ G39I-5000 *
⑨ X-5000 *
⑩ G18 G00Z2500 *
⑪ G17 G40X-2000Y-2000M02 *

3.12.11 Tool Offset B

Tool offset for Z axis (Tool length offset) can be performed by the command of G43 or G44.

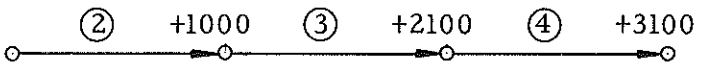
The offset amounts is selected by D or H code.

G43 specifies that tool is moved toward + direction by offset amount.

G44 specifies that tool is moved toward - direction by offset amount. Tool offset B is cancelled by G49.

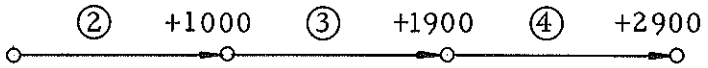
Example 3.12.11(a)

①	G92Z0 *	Programming of Absolute Zero Point.
②	G90G00Z1000 *	Absolute programming
③	G43G01Z2000H01F——*,	Tool offset + direction (Offset amount: + 100)
④	G00Z3000 *	



Example 3.12.11(b)

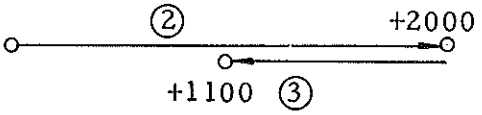
①	G92Z0*	Programming of Absolute Zero Point
②	G90G01Z1000F200*	Absolute programming
③	G44Z2000H01*	Tool offset - direction (Offset amount: +100)
④	G00 Z 3000 *	



Example 3.12.11(c)

①	G92 Z 0 *	
②	G90 G00 Z 2000 *	
③	G43 Z 1000 H01 *	(Offset amount: +100)

Tool path



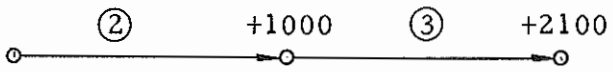
(Note 1) The offset data is modal, and this data is effective until cancel by other offset command.

(Note 2) This + or - direction of tool offset is effective to programmed end point and ineffective to movement direction.

Example 3.12.11(d)

① G92 Z0 *
② G91 Z1000 * Incremental programming
③ G43G01Z1000H01F200* (Offset amount: +100)

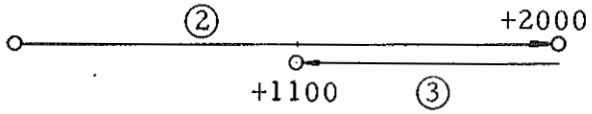
Tool path



Example 3.12.11(e)

① G92 Z0 *
② G91 Z 2000 *
③ G43G01Z-1000H01F200* (Offset amount : +100)

Tool path

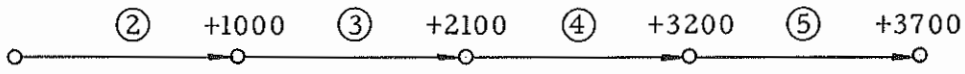


(Note 3) The offset is effective at absolute value of programmed end point in case of incremental mode.

Example 3.12.11(f)

① G92 Z0 *
② G90 Z1000 *
③ G43G01Z2000H01F200* (Offset amount: +100)
④ G43G00Z3000H02* (Offset amount: +200)
⑤ G44G01Z4000H03F200* (Offset amount: +300)

Tool path

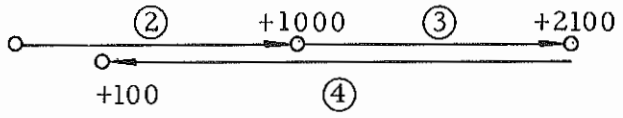


(Note 4) The new offset data is commanded in offset mode, the old offset data is canceled and for the positioning considering new offset data is performed.

Example 3.12.11(g)

① G92 Z0 *
② G90 Z1000 H01 * (Offset amount : +100)
③ G43 G01 Z2000 F200 *
④ Z0 H02 * (Offset amount : +200) H02 is ignored.

Tool path

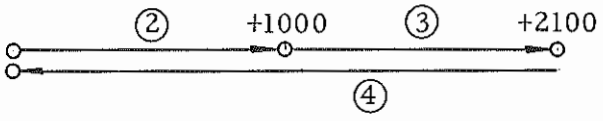


(Note 5) The G code of G43 and G44 is not modal.
As the H code is modal, the predetermined H code is effective.

Example 3.12.11(h)

① G92 Z0 *
② G90 G00 Z1000 *
③ G43 G01 Z2000 H01 F200 * (Offset amount : +100)
④ G49 G00 Z0*

Tool path



(Note 6) Tool offset B is canceled by G49 code.

(Note 7) These commands of G43, G44 and G49 can not be used in fixed cycle mode.

(Note 8) When the Z-axis command is not programmed after G43, G44 and G49 codes, the P/S alarm (No 35) is displayed on DPL unit.

(Note 9) G43, G44 and G49 can be used only in the G17 mode during the offset mode of G41 and G42. G43, G44 and G49 cannot be used in the G18 and G19 mode.

3.12.12 Tool Offset A (G45 to G48)

By specifying G45 to G48, the movement distance of the axis specified on NC tape, etc. can be increased or decreased by a setting in the Offset amount memory. Table 3.12.12 shows the G codes and their functions.

Table 3.12.12 Tool Offset and G Codes

G code	Function
G45	Tool offset increase
G46	Tool offset decrease
G47	Tool offset double increase
G48	Tool offset double decrease

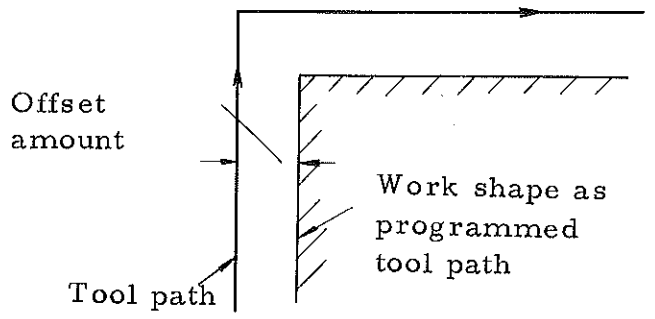
These G function are used for tool length compensation or for tool radius compensation of a simple drawing. Tool compensation of G41 and G42 can't be done during these function (G45 to G48) being excuted. Conversely, tool compensation of G45 to G48 can't be done during the function (G41 and G42) being excuted.

These G codes are not modal, and effective only for a block in which they are specified.

These offset amounts, once selected by the D, H codes and others, remain unchanged until other offset amounts are selected.

By setting the tool radius value in the offset amount memory, a work shape becomes the programing path.

By setting the tool wearing value in the offset amount memory, a tool path becomes the programing path.



Tool path increases or decreases to the direction toward which the tool advances.

At the time of absolute command, tool path increases or decreases to the direction toward which the tool advances from the end point of the last block to the position which is commanded by the block including G45 to G48.

(1) Command and Offset

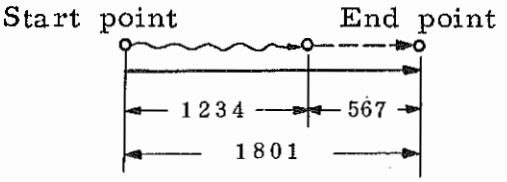
~~~~→ Move command data

- - - -→ Offset amount

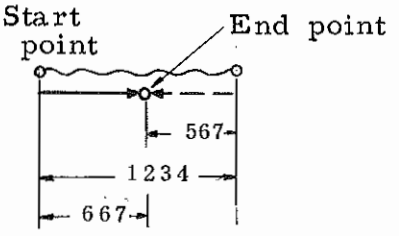
——→ Actual movement amount

(a) G45 command (increase by a offset amount memory)

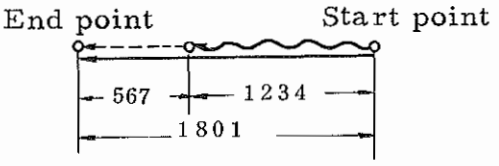
(i) Move command + 1234
Offset amount + 567



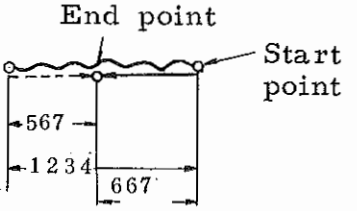
(ii) Move command + 1234
Offset amount - 567



(iii) Move command-1234
Offset amount + 567



(iv) Move command-1234
Offset amount - 567

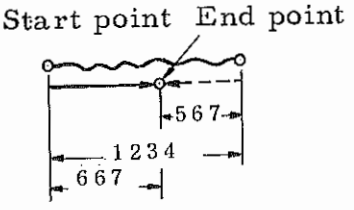


(b) G46 command (decrease by a offset amount memory)

The actual movement amount becomes equivalent to a setting in the switch with G45, the sign of which is reversed.

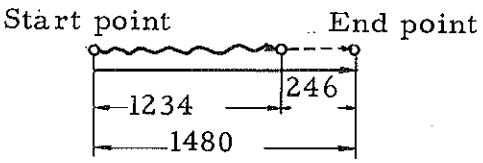
(i) Move command +1234
Offset amount +567

ii) to iv) are omitted.

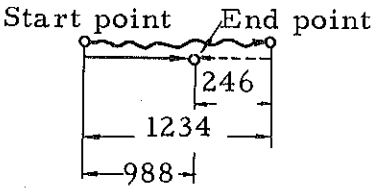

~~~~

(c) G47 command (increase by double offset amount memory)

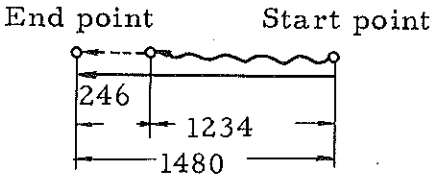
(i) Move command + 1234
Offset amount + 123



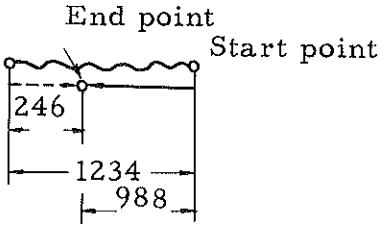
(ii) Move command + 1234
Offset amount - 123



(iii) Move command - 1234
Offset amount + 123



(iv) Move command - 1234
Offset amount - 123

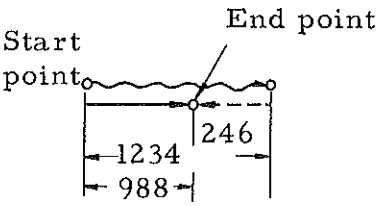


(d) G48 command (decrease by double offset amount memory)

The actual movement amount becomes equivalent to a offset amount memory with G47, with the sign of which is reversed.

(i) Move command + 1234 (ii) to (iv) are omitted.

Offset amount + 123



In a command in which a move is to be made by only a offset amount, the movement amount is specified as zero in incremental mode.

In case of incremental mode (G91), it can be made that movement amount is equal to offset amount by zero movement command.
In case of absolute mode (G90), it can't be made; no movement.

Offset amount + 01234 Offset No. 01

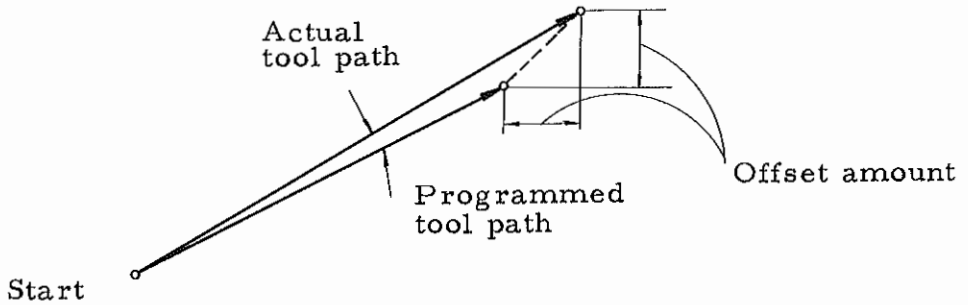
Programming command	G45X0 D01*	G46X0 D01*	G45X-0 D01*	G46X-0 D01*
Equivalent command	X 1234*	X-1234*	X-1234*	X 1234*

(2) Notes on Tool Offset

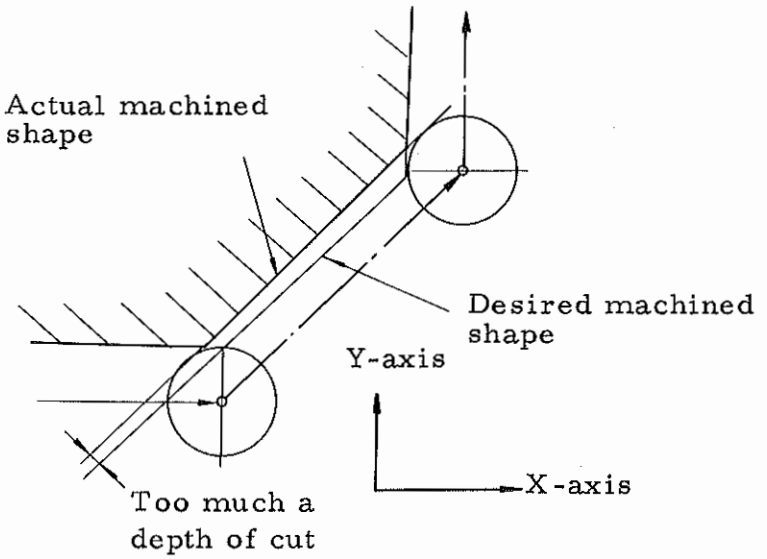
(a) In case G45 to G48 are specified for a 2-axis at-a-time control move command, Tool Offset is effected for both axes.

Case of G45

Move command	X10000 Y5000
Offset amount	+2000, Offset No. 02
Programmed command	G45 G01 X100000 Y5000 D02*



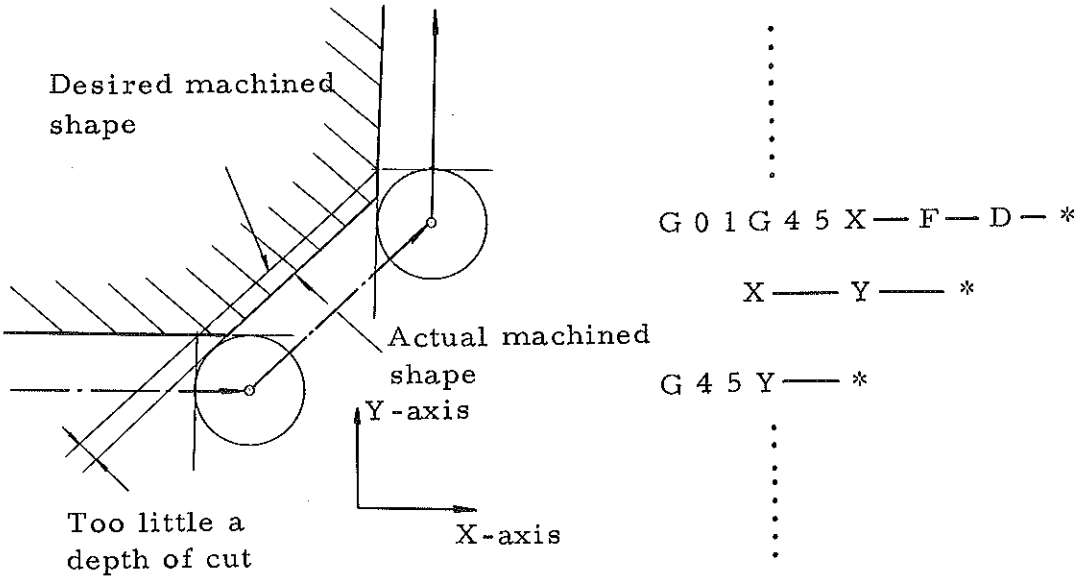
(b) If Tool Offset is effected for both axes, too much or little a depth of cut is generated.



G01X — F — *

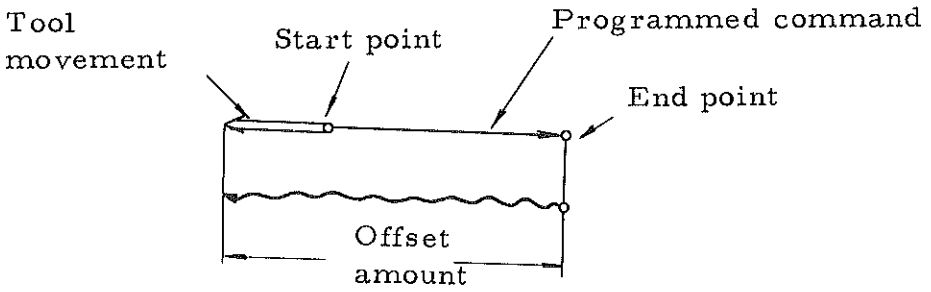
G45X — Y — D — *

Y — *



(c) Tool Offset can be effected by G45 to G48 commands for Circular Interpolation (G02, G03) only in the case of a one-quarter or three-quarter arc command. That is, Cutter Compensation is possible only in the case of a one-quarter or three-quarter arc command.

(d) In case the command is reversed due to the decrease as in the figure below, a move is made in the reverse direction.



(Example)

G46X250*

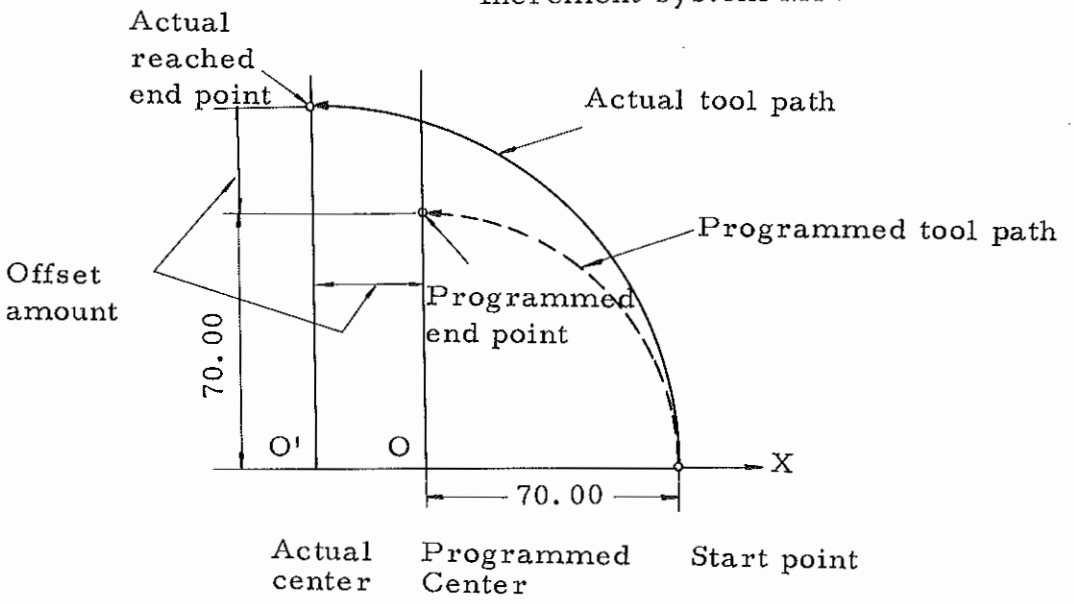
(incremental command) } Equivalent command
Offset amount +370 } X-120*

(e) G45 to G48 can't be used during tool compensation mode (G41 and G42). This becomes (P/S) alarm.

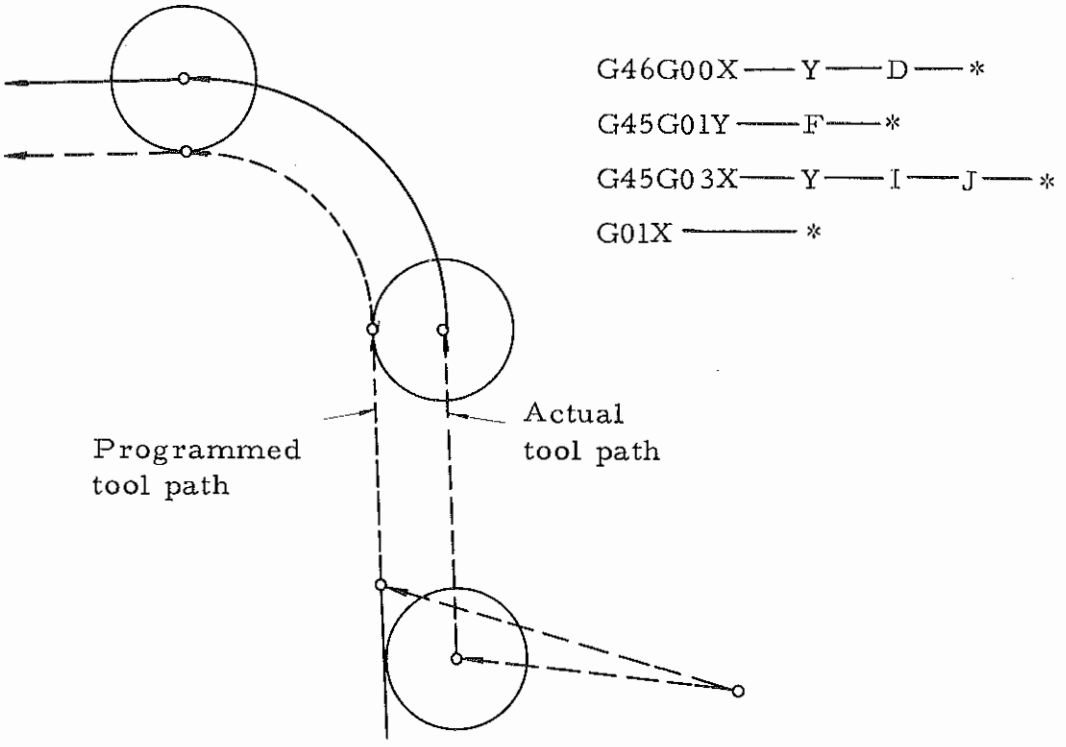
If you want to use both (G45 to G48) and (G41 or G42) mode, command G40 code and after that command G41 or G42.

(f) G45 to G48 are ignored during fixed cycle mode (G81 to G89). In this case, fixed cycle mode (G81 to G89) should be commanded after the G80 has commanded.

Example 3.12.12 (a) Offset amount +3000, Offset No. 01
Programmed command
G45G03X-7000Y 7000 I-7000 D01 *
Increment system M1V

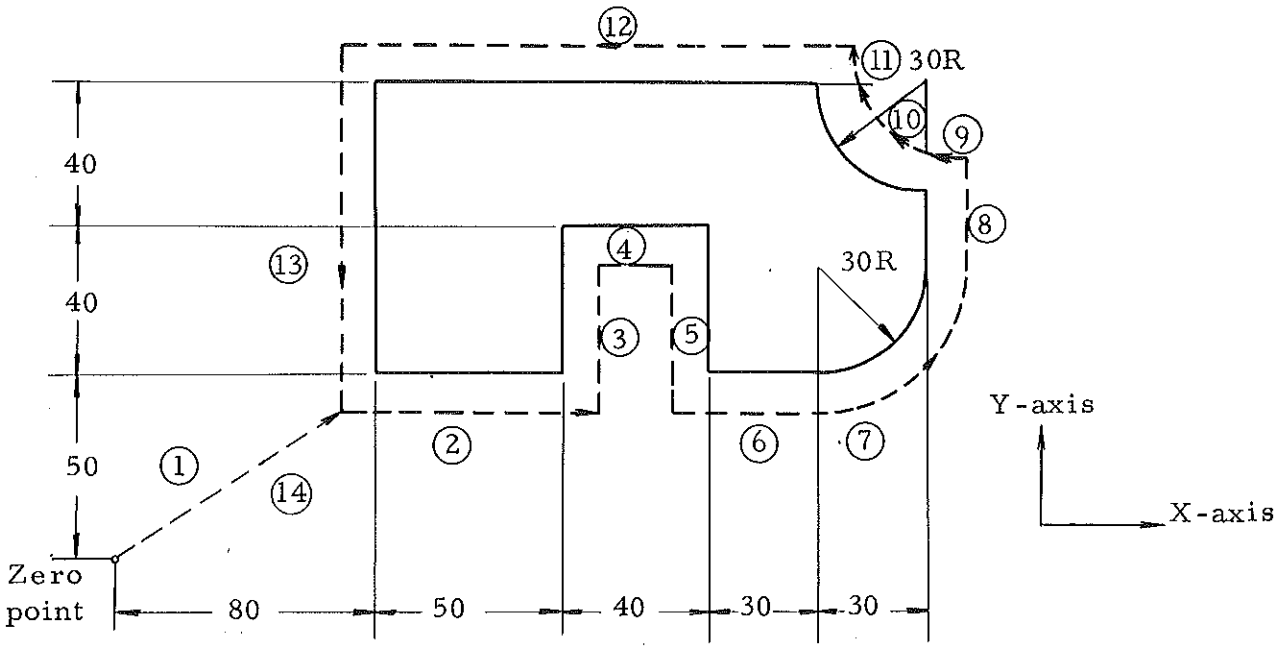


Example 3.12.12 (b) Cutter Compensation in Circular Interpolation



Example 3.12.12 (c) Program example using Tool Offset

Tool radius compensation



Tool diameter: 20

Offset No. 01 Offset amount +1000

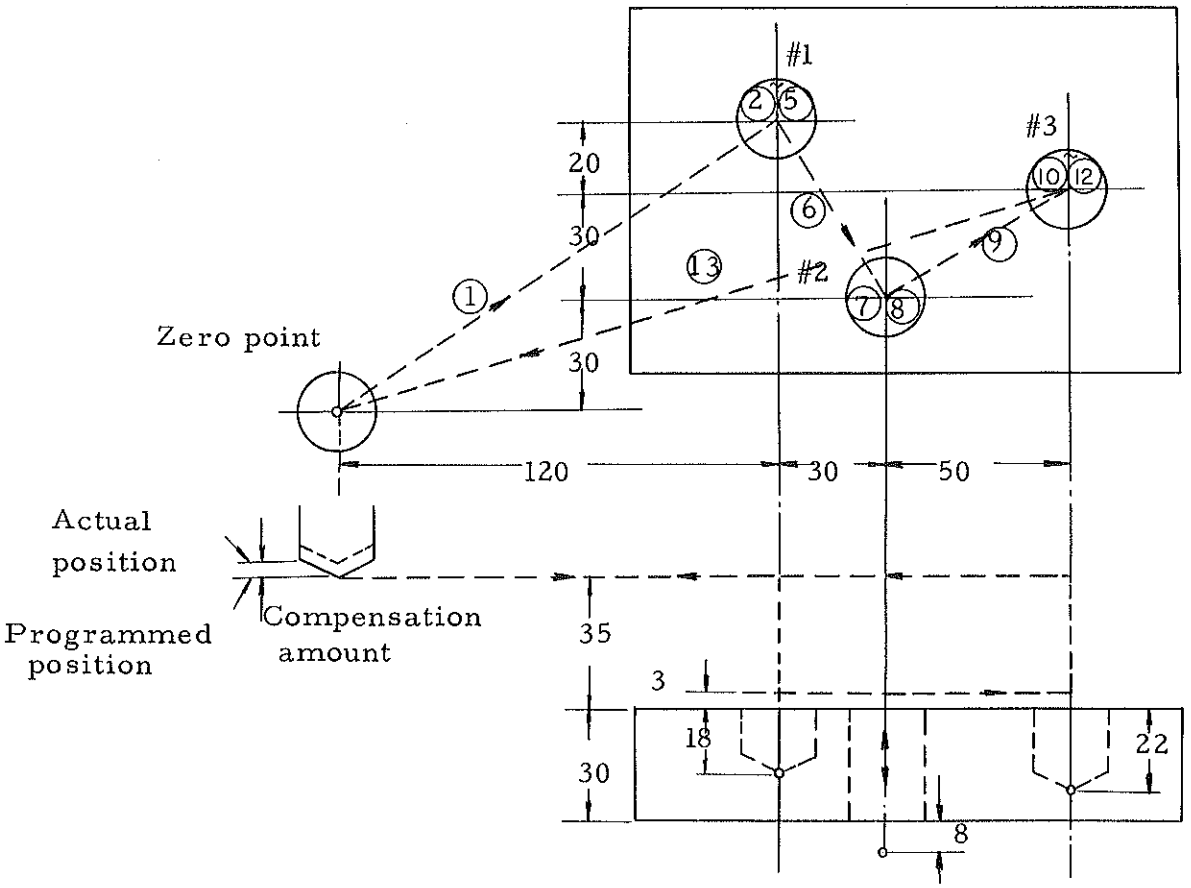
Increment system M1V

Program example

```
(1)  G91 G46 G00 X8000 Y5000D01 *
(2)  G47 G01 X5000 F120 *
(3)  Y4000 *
(4)  G48 X4000 *
(5)  Y-4000 *
(6)  G45 X3000 *
(7)  G45 G03 X3000 Y3000 J3000 *
(8)  G45 G01 Y2000 *
(9)  G46 X0 *
(10) G46 G02 X-3000 Y3000 J3000 *
(11) G45 G01 Y0 *
(12) G47 X-12000 *
(13) G47 Y-8000 *
(14) G46 G00 X-8000 Y-5000 *
```

Tool length compensation

Drilling of holes #1, #2 and #3.



Offset No. 02 Offset amount = +400

Increment system M1V

Program example

①	G91 G00 X12000 Y8000 *	
②	G45 Z-3200 H02 *	
③	G01 Z-2100 F100 *	
④	G04 P200 *	Dwell at hole bottom for 2 sec.
⑤	G00 Z2100 *	
⑥	X3000 Y-5000 *	
⑦	G01 Z-4100 *	
⑧	G00 Z4100 *	
⑨	X5000 Y3000 *	
⑩	G01 Z-2500 *	
⑪	G04 P200 *	Dwell at hole bottom for 2 sec.
⑫	G45 G00 Z5700 *	
⑬	X-20000 Y-6000 *	

3.12.13 Canned Cycles (G73, G76, G80 to G89)

The following Canned Cycles are available. When power is turned on, the state is of G80.

Table 3.12.13 Canned Cycles

G code	Drilling (-Z direc.)	Operation at hole botton	Retraction (+Z direction)	Usage
G73	Intermittent feed	——	Rapid traverse	High speed peck drilling cycle
G76	Feed	Oriented spindle stop	Rapid traverse	Fine boring cycle (For fixed cycle B Group only)
G80	——	——	——	Cancel
G81	Feed	——	Rapid traverse	Drilling cycle (spot drilling)
G82	Feed	Dwell	Rapid traverse	Drilling cycle (counter boring)
G83	Inter mittet feed	——	Rapid traverse	Peck drilling cycle
G84	Feed	Spindle reverse	Feed	Tapping cycle
G85	Feed	——	Feed	Boring cycle
G86	Feed	Spindle stop	Feed	Boring cycle
G87	Feed	Spindle stop	Manual operation or Rapid traverse	Boring cycle, Back boring cycle
G88	Feed	Dwell, spindle stop	Manual operation	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

(Note 1) Canned cycle A

Independently signals for spindle reverse and spindle stop are used.

Canned cycle B

M code signals for spindle reverse and spindle stop are used.

(Note 2) The different operations are performed for Canned cycle A and B in G87 mode.

"Feed" employed in Columns "Drilling" and "Retraction" means being dependent on the feed rate specified in an F code.

The Dwell time at the hole bottom is specified in a number code following address P.

Generally speaking, the Canned Cycle comprises a sequence of the following four operations:

(1) Positioning on X and Y axes.
(2) Rapid Traverse to point R
(3) Drilling
(4) Return

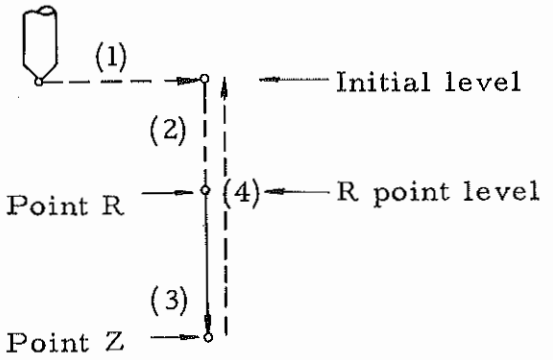


Fig. 3.12.13(a) Fixed Cycle Operation

During a canned cycle operation, the automatic acceleration/deceleration time constant is switched automatically according to the feed rate in each (1), (2), (3) and (4) operation.

After the last command pulse for a particular operation is transmitted, the next operation is proceeded at the next points of time;

(a) after completion of the operation (1)
(b) after completion of the operation (2)
(c) after completion of the operation (3)
(d) after completion of the operation (4)
(e) when cutting feed switches to Rapid Traverse at the R point level during the operation (4) in G98 mode (in the case of G84, G85 or G89).

If an auxiliary function (B, S, T or M function) is specified in a Canned Cycle command, an appropriate operation is performed simultaneously with the operation (1) of the Canned Cycle.

If you operates a Canned Cycle at the time of SINGLE BLOCK switch ON, the movement stops after completion of the operations (1) in the hold state and the movement stops after completion of

the operation (4) in the stopped state. But in the cycle including L command, the movement stops after completion of the operation (4) in the hold state up to (L-1) times and the movement stops after completion of the operation (4) in the stopped state at the Lth time.

These Canned Cycle operations, to be more strict, come in three modes, each of which is specified in a particular modal G code, as shown below.

(1)	Data format	G90	Absolute
		G91	Incremental
(2)	Return point level	G98	Initial level
		G99	R point level
(3)	Drilling mode	G73 G76 G80 G81 : G89	See Table 3.12.13

(Note) The initial lebel means the absolute value of Z-axis at the time of changing the fixed cycle cancel mode into the canned cycle mode.

Data is given as shown in Fig. 3.12.13(b), depending on G90 or G91 specified.

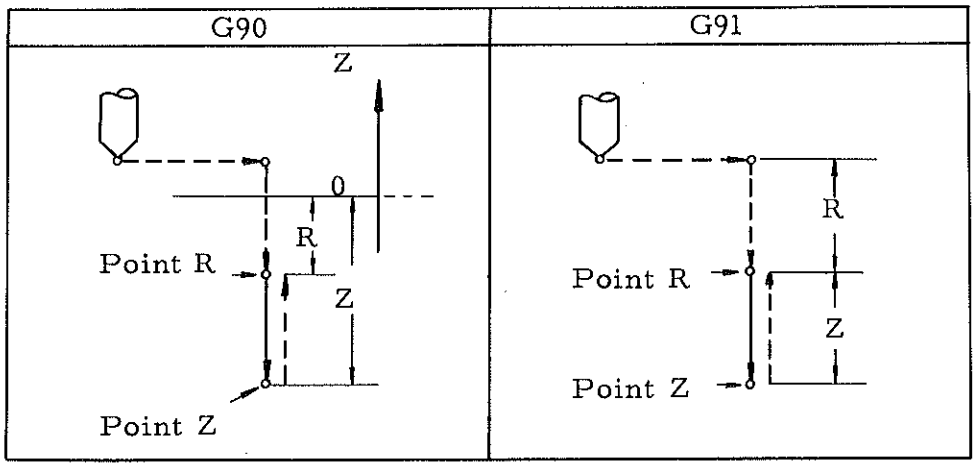


Fig. 3.12.13(b) Absolute and Incremental programming

By return operation, whether a tool is only returned to point R, or up to the initial level, depends on G98 or G99 specified. This is shown in Fig. 3.12.13(c).

Once a drilling operation is performed in G99 mode, since the initial level becomes the same as the R point level, be careful of a combination of G90 and G91.

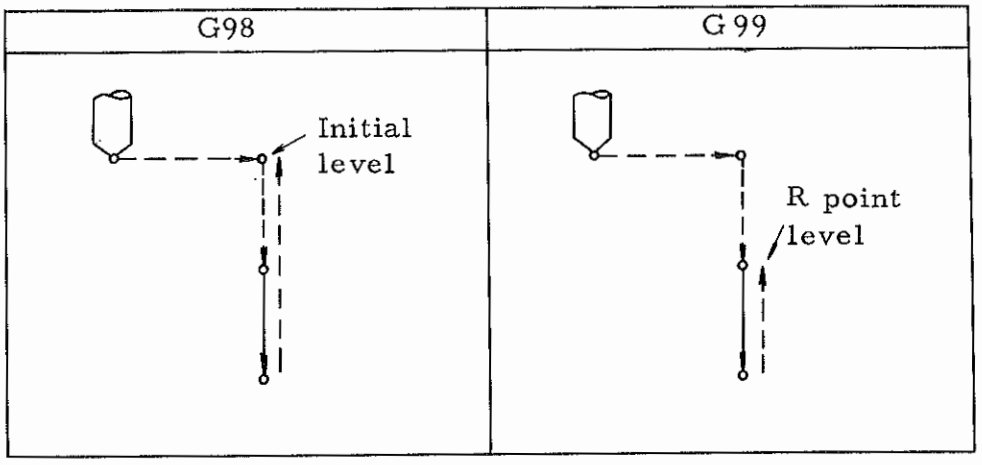


Fig. 3.12.13(c) Initial level and R point level

Data related to drilling operation specified subsequent to these G73, G76, G81 ~ G89, G98 and G99 constitute one block. This command permits the data to be stored in the control unit as a modal value.

$$\begin{pmatrix} G73 \\ G76 \\ G81 \sim G89 \end{pmatrix} \begin{pmatrix} G98 \\ G99 \end{pmatrix} \quad X — Y — Z — L — P — Q — R — F — *$$

X: Specifies a hole position (Absolute or Incremental).

Y: Specifies a hole position (Absolute or Incremental).

Z : Specifies a hole bottom position (Absolute or Incremental).

P : Specifies a Dwell time in the hole bottom position.

Q : Specifies a depth of cut at each time in G73 or G83 mode, and sift value in G76 or G87 (fixed cycle B) mode. (This is always given in an Incremental value.)

R : Specifies a R point lebel (Absolute or Incremental)

F : Specifies a feed rate in feed.

L : Specifies the number of repetitions of the same cycle.

An actual drilling operation is initiated by specifying a G code (G81 to 89) giving a drilling mode and hole position data. This mode is cancelled by G80, or a different G code of group A, or reset.

Among the address code X, Y, Z, P, Q, R, F and L used in the canned cycle, P, Q, R and F are modal.

Example 3.12.13(a)

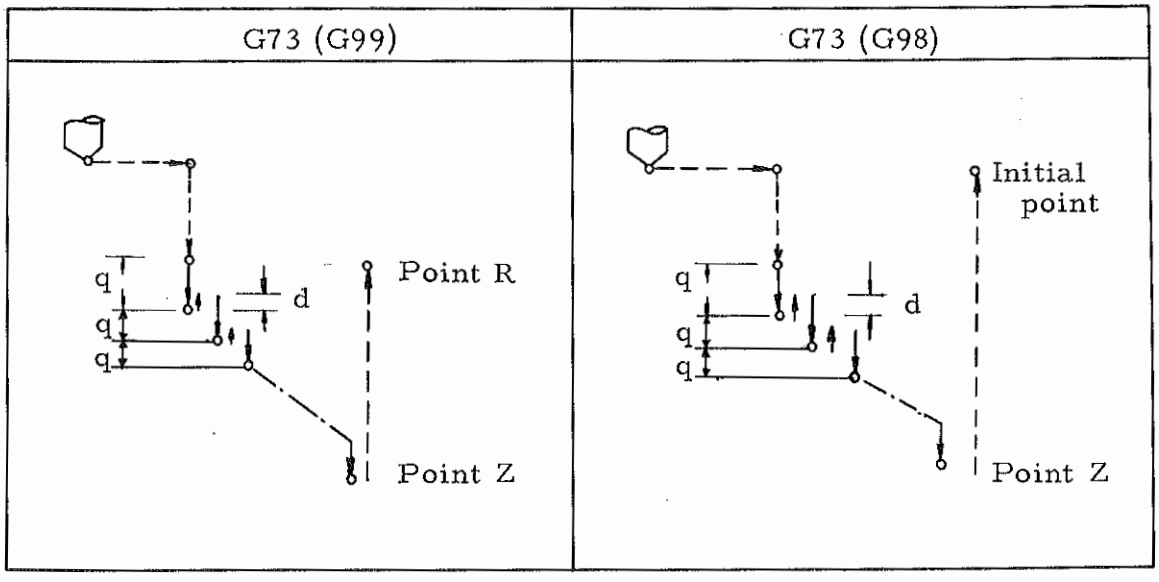
G90G81G99X ___Y ___ Z ___ R ___ F ___ *	Drills.
X ___ Y ___ *	Drills.
⋮	⋮
G80X ___ Y ___ *	Does not drill, but causes the tool to move.

Except for G87 (fixed cycle B), the absolute value of R point is necessary larger positive value than that of Z-point. That is, a drilling operation must be made at the negative direction toward Z point from R point. Otherwise G73 and G83 do not move correctly.

The tool offset commands are ineffective in canned cycle mode.

According to the state of "MANUAL ABSOLUTE" switch", R point and initial point in G87 and G88 mode become follows.
ON..... Designated value by programming.
OFF ..... Shifted value by manual feed.

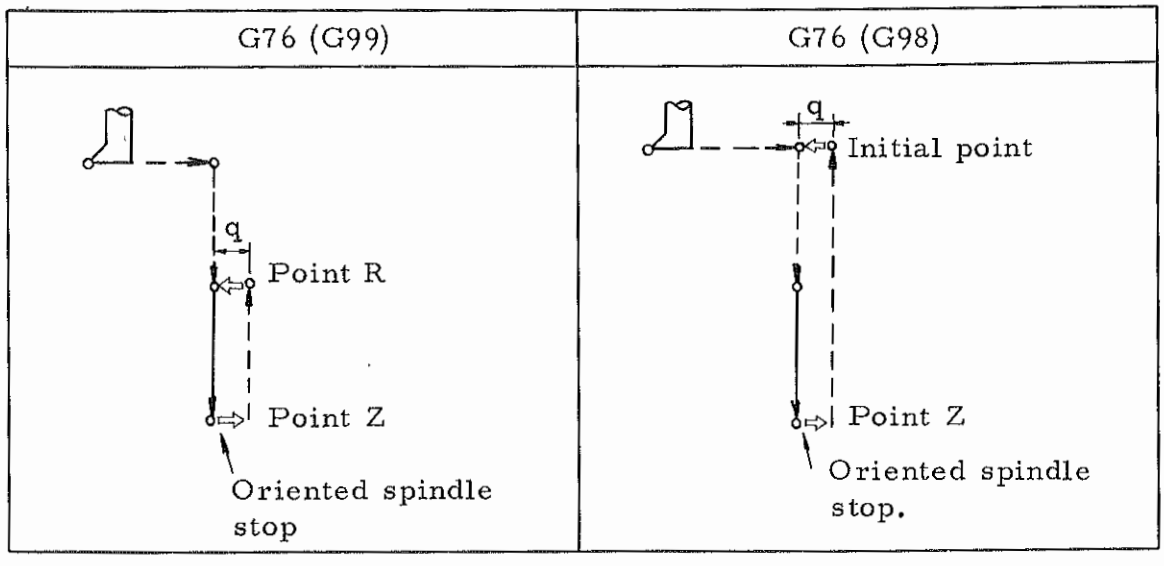
(1) G73 (High speed peck drilling cycle)

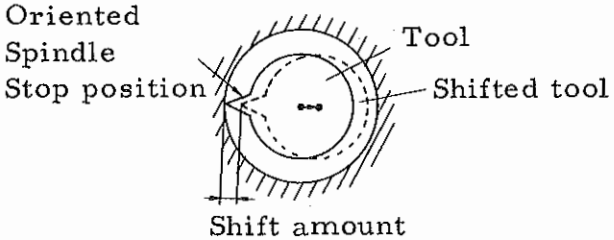


Return dimension
d= 1mm (M1V~ M4V)
d= 0.1 in (I1V ~ I4V)

----→ Rapid traverse
——→ Feed

(2) G76 (Fine boring cycle)





⇨ Shift of tool

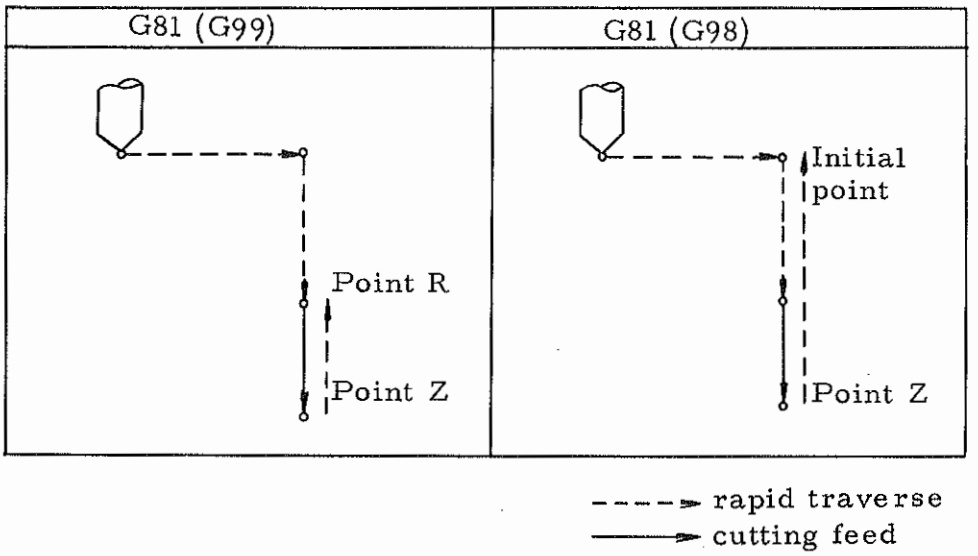
(designated by address Q)

Shift amount is speified by positive value. Shift direction is preset by parameter setting (+X, -X, +Y or -Y). This value is modal and is used for depth for cut in G73 and G83.

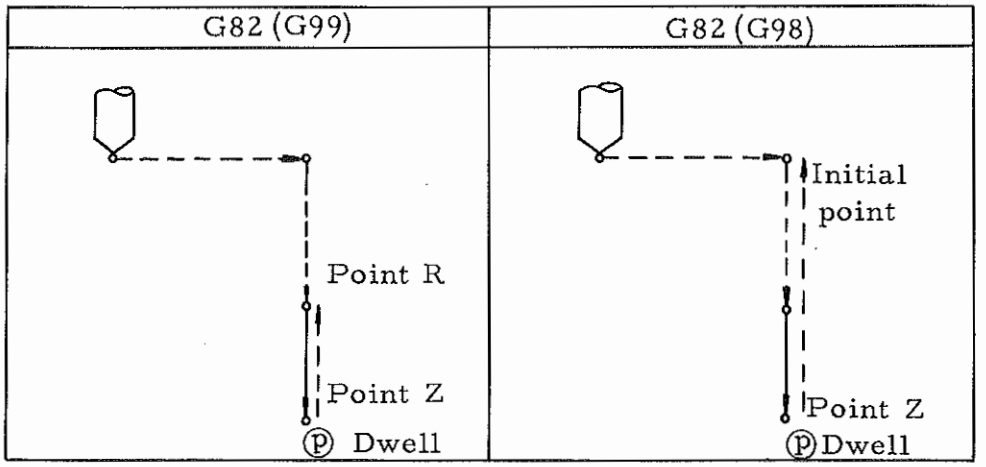
(3) G80 (Canned cycle cancel)

This command cancels a (Canned Cycle (G73, G76, G81 to G89), and subsequently causes ordinary operations to be performed. Also, point R and Z are canceled.

(4) G81 (Drilling cycle, spot drilling)

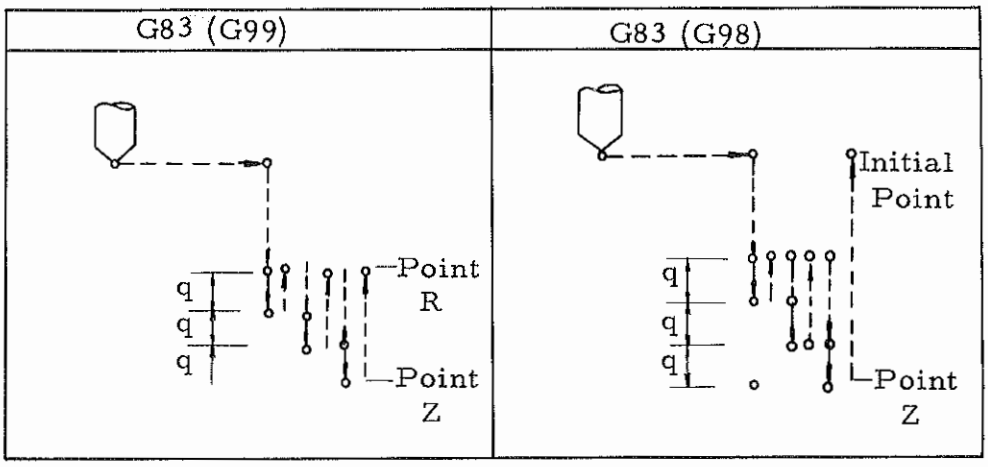


(5) G82 (Drilling cycle, counter boring)



This command functions the same as G81 except that the tool moves up after Dwell is performed at the hole bottom (specified in a P code).

(6) G83 (Peck drilling cycle)

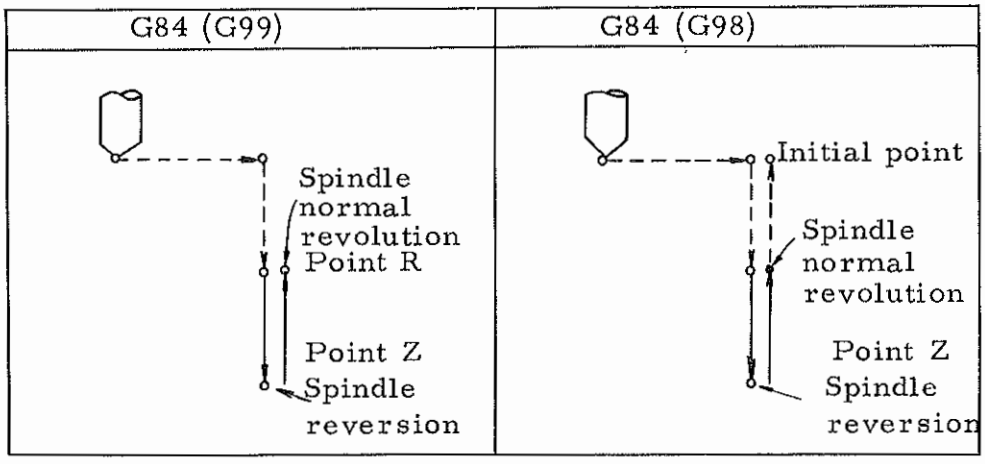


In this case, the command

G83 X ____ Y____ Z____ Q____ R____ F____ *

is programmed. Q represents a depth of cut at each time, and is always expressed in an Incremental value.
Rapid Traverse switches to cutting feed at one mm or one tenth inch ahead of the position where the immediately preceding cutting is made in the second cutting and after.

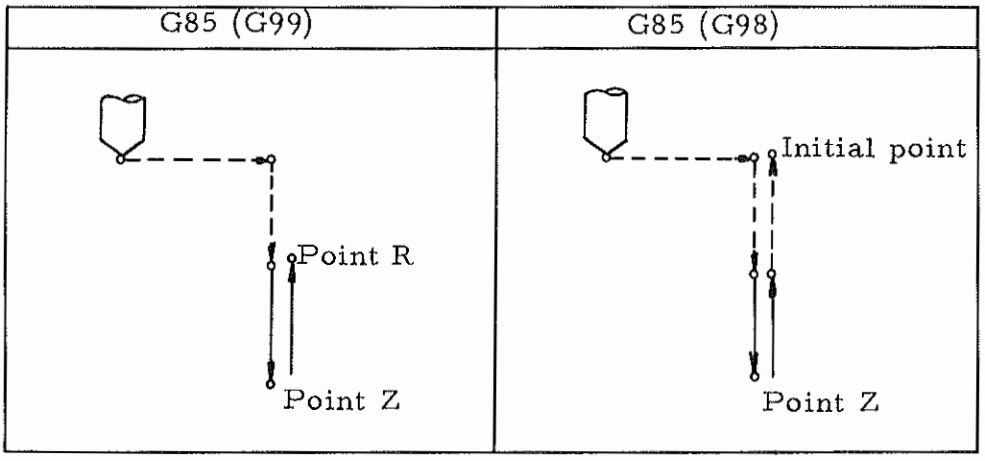
(7) G84 (Tapping cycle)



This command specifies the spindle to reverse at the hole bottom, and a tapping cycle to be performed.

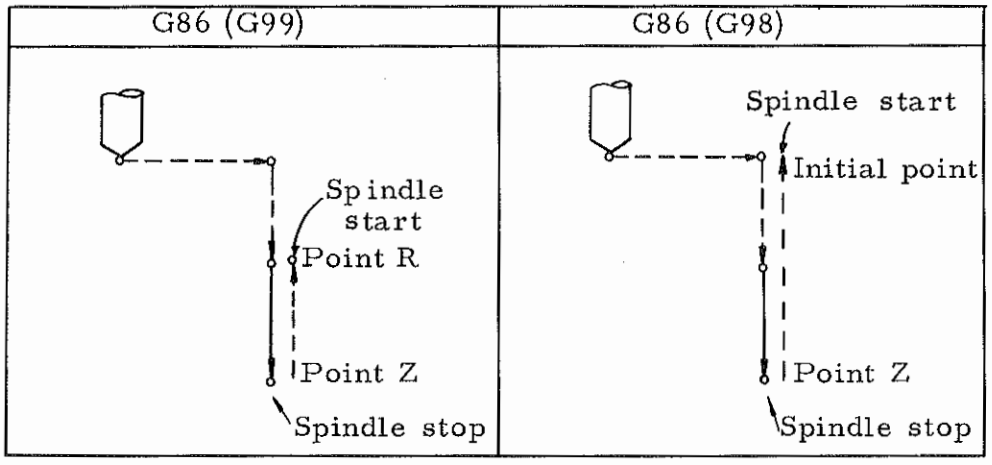
(Note) During tapping by G84, the Feed Hold and Feed rate Override are ignored.

(8) G85 (boring cycle)



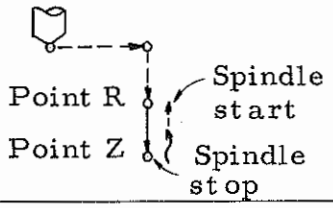
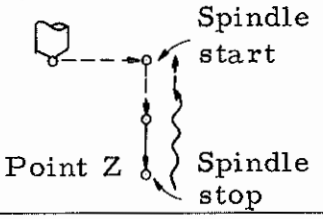
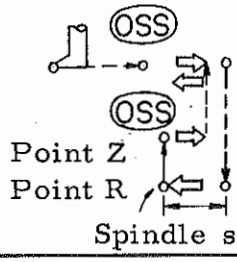
This command functions the same as G84 except that the spindle does not reverse at the hole bottom.

(9) G86 (boring cycle)



This command function the same as G81 except that the spindle stops at the hole bottom, and returns in Rapid Traverse.

(10) G87 (boring cycle/back boring cycle)

CYCLE	G87 (G99)	G87 (G98)
A		
B	This cycle should not be used.	

→ rapid traverse
→ cutting feed
~ Manual feed
⇨ shift

CYCLE A (boring cycle) (OSS) : Oriented Spindle Stop

A hold state is provided at the hole bottom after spindle stop. Consequently, at this time, the tool can be moved manually by a switch to manual mode. Any manual operation can be made, but it is safe to finally make a state in which the tool is drawn out of the hole.

When restarting maching, if the operation is initiated in the tape mode, it is restarted according to the commands in the next block of NC tape.

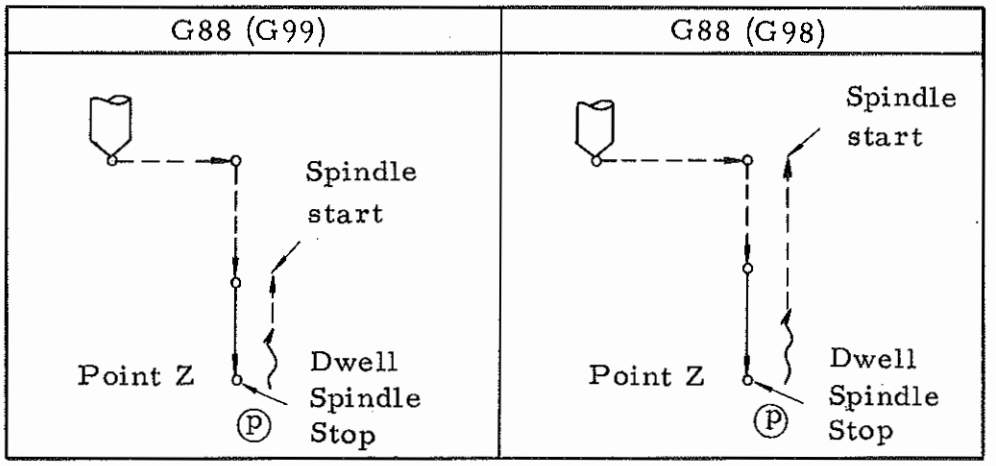
CYCLE B (back boring cycle)

The following operations are performed automaticaly by this command.

(1) Oriented spindle stop after positioning of X and Y axes.
(2) Shift of opposite tool tip direction.
(3) Positioning to point R.
(4) Return of shift amount.
(5) Spindle start (CW).
(6) Boring to point Z.
(7) Oriented spindle stop.
(8) Shift of opposite tool tip direction.
(9) Return to initial point.
(10) Return to positioning point.
(11) Spindle start (CW).

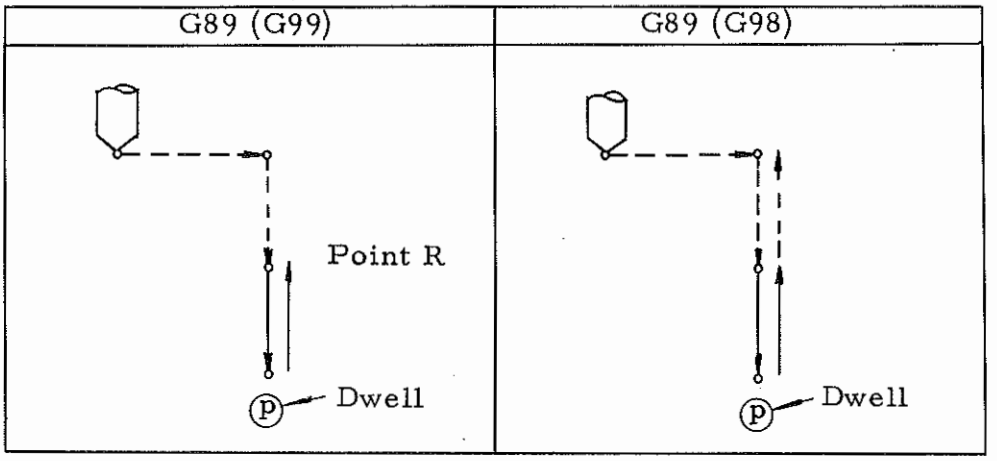
Note) Shift value and direction are same as G76 mode.

(11) G88 (boring.cycle)



This command functions the same as G87 (fixed cycle A) the spindle stops at the hole bottom after Dwell.

(12) G89 (boring cycle)



This command functions the same as G85 except that Dwell is made at the hole bottom.

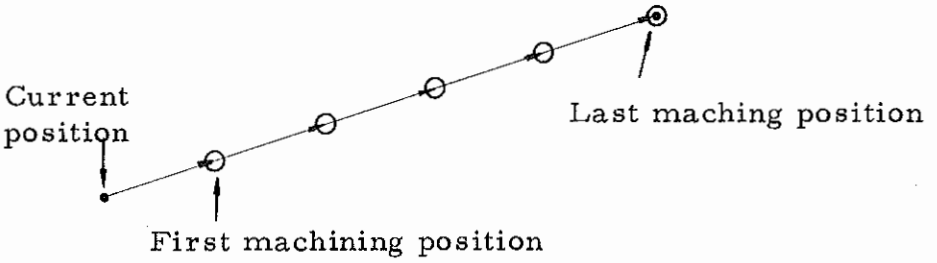
If drilling at equal intervals is to be repeated in the same cycle, the number of repetitions can be specified by the use of address L.

In the commands G98 or G99 including L designation, the return point is initial lebel or R point lebel respectively at the beginning.

The maximum value of L is 8388607.
The word L is only effective in programmed block.

The designation of repetition number designate the repetition number of positioning and drilling. If M, S and T functions are specified in the same block, these function are output only once at the same time with positioning.

Example 3.12.13(a)

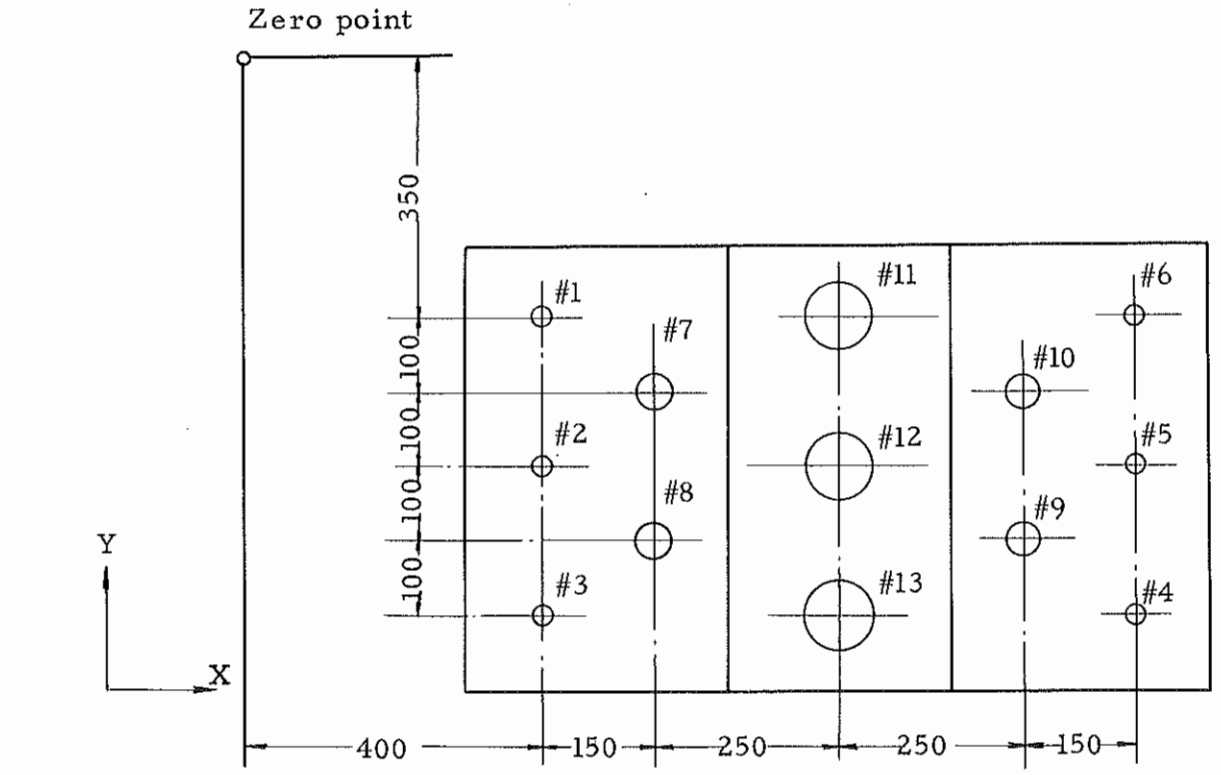


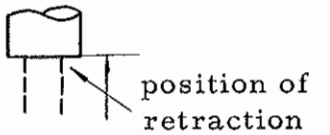
The command G98G81X__Y__ Z__ R__ L5F__ * or

G98*
G81X__Y__ Z__ R__ L5F__ *

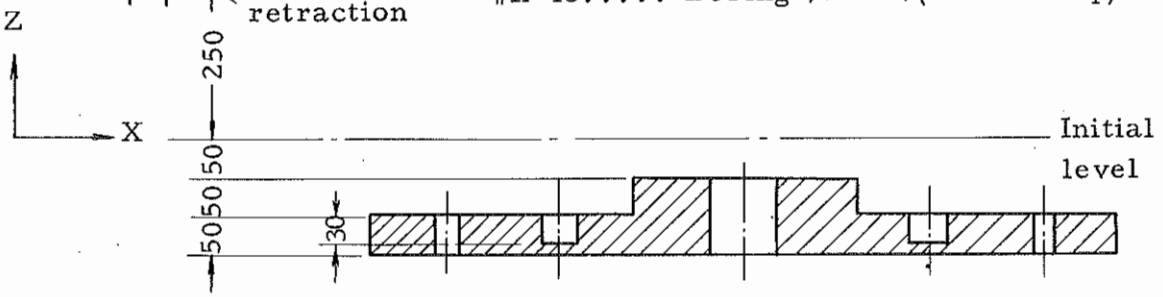
X——Y—— specifies the first machining position in Incremental values (in the case of G91). If this command is specified in Absolute values, drilling is repeated in the same position.

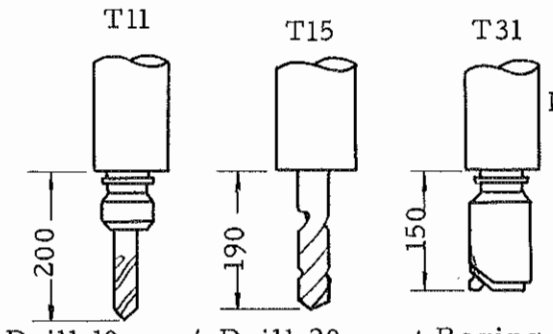
Example 3.12.13(b) Programming using Cutter Compensation and Canned Cycle





#1~6..... Drilling 10mmϕ
#7~10..... Drilling 20mmϕ
#11~13..... Boring 95mmϕ(50mm deep)





Note: The memory is assumed to store in advance

H11= 200mm
H15= 190 mm
H31= 150 mm

Drill 10mmϕ	Drill 20mmϕ	Boring bar 95mmϕ
H11	H15	H31

Tool length compensation number

Programming example (increment system M1V)

```
%
*
N001 G92 X0 Y0 Z0 *                          Programming of absolute zero
                                             point
                                               (with the current position
                                               as the zero point)

N002 G90 G00 Z25000T11M06*                   Tool change

N003 G46 Z 0H11*                             Tool offset, Initial lebel

N004 S0900M03*                               Spindle start

N005 G81 G99  X40000Y-35000                  Drilling #1 hole after
     Z-15300R-9700F120*                      positioning

N006 Y-55000*                                Drilling #2 hole after
                                             positioning, return to R point

N007 G98Y-75000*                             Drilling #3 hole after
                                             positioning, return to initial lebel

N008 G99X120000*                             Drilling #4 hole after
                                             positioning, return to R point

N009 Y-55000*                                Drilling #5 hole after
                                             positioning, return to R point

N010 G98Y-35000*                             Drilling #6 hole after
                                             positioning, return to initial lebel

N011 G00X0Y0M05*                             Zero Return & spindle stop

N012 G46 Z25000T15M06*                       Tool offset cancel &
                                             tool change

N013 G46 Z0H15*                              Tool offset, Initial lebel

N014 S0700M03*                               Spindle start

N015 G82G99X55000Y-45000                     Drilling #7 hole after
     Z-13000R-9700P300F70*                   positioning

N016 G98Y-65000*                             Drilling #8 hole after
                                             positioning, return to initial lebel
```

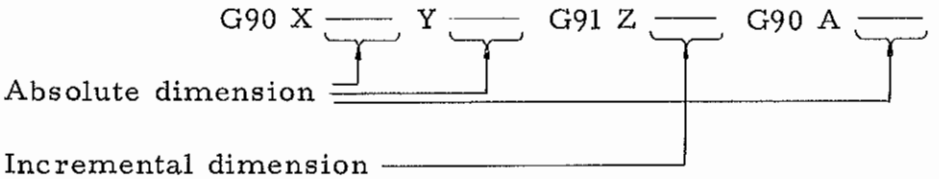
N017 G99X105000*	Drilling #9 hole after positioning, return to R point
N018 G98Y-45000*	Drilling #10 hole after positioning, return to initial lebel
N019 G00X0Y0M05*	Zero Return & spindle stop
N020 G46 Z25000T31M06	Tool offset cancel & tool change (T31 : 95mm-dia. boring bar)
N021 G46 Z0H31*	Tool offset, Initial lebel
N022 S0250M03*	Spindle start
N023 G85G99X80000Y-35000 Z-15300 R-4700F50*	Drilling #11 hole after positioning
N024 G91Y-20000L2*	Drilling #12 and #13 holes after positioning
N025 G00G90X0Y0Z0M05*	Zero Return
N026 G46G91Z0*	Tool offset cancel
M02*	Program stop

3.12.14 Absolute and Incremental Programming (G90 and G91)

G90 and G91 specify the subsequent dimensions to be Absolute and Incremental, respectively. That is,

G90	Absolute
G91	Incremental

These G codes, although modal, can be specified plural times in one block.
The following is one example.



Generally speaking, the G code can be specified repeatedly in the same block. When the G codes belonging to the same group are specified mixedly, the code specified later is valid.

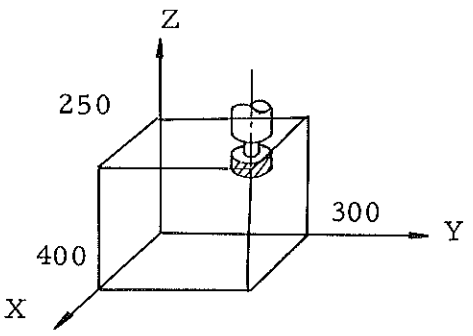
3.12.15 Programming of absolute zero point (G92)

The NC tape command or MDI

G92 X—— Y —— Z—— α —— *

permits setting in the control unit such a coordinate system that the current tool position is the specified position. This setting can be done at all times, regardless of the Absolute/Incremental (G90/G91) mode specified. A coordinate system, once set, is not cleared by reset button, but by the ORIGIN button on the setting/display panel. That is, this command is made with respect to all the axes. The absolute coordinate value of an axis that was not commanded does not change. When power is turned on, and after completion of automatic Zero Return, prior to entering into the next movement, the coordinate system setting must, as a rule, be redone. This G92 command can not be designated from G28 to G29 block. When this G92 command is made in offset mode, the top of the offset vector considered as the coordinate system.

Example 3.12.15 (a) Programming of absolute zero point (increment system M1V)



Setting a coordinate system as shown in the figure above, is programmed as follows:

G92 X 40000 Y 30000 Z 25000 *

3.12.16 Initial Level and R Point Level (G98 and G99)

G98 and G99 specify a return point level in Fixed Cycle (see Section 3.12.13) to be up to the initial level and up to the R point level, respectively, as shown in Fig. 3.12.16.

Refer to Fixed Cycle (G80 to G89)

As usual, G99 is used for the first drilling and G98 is used for last drilling.

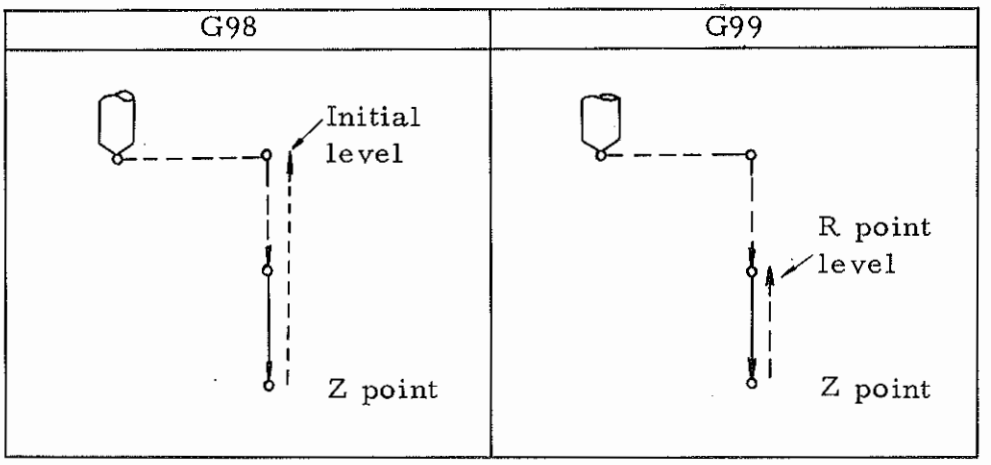


Fig. 3.12.16 Initial level and R point level

3.13 D and H Functions

Same offset amount are stored in the same number for both H and D codes.

D is used to compensate a tool diameter.

H is used to compensate a tool length.

The correspondence between codes and compensation amounts is previously stored in memory on the MDI & DPL unit. By specifying a 2-digit code from among them, an appropriate compensation can be performed. Refer to Sections 3.12.10 to 3.12.12.

The selectable number of H and D codes are up to 32 or 64 (option) put together. If you over there number, it becomes P/S alarm.

3.14 Spindle-Speed Function (S Function)

The Spindle-speed function specifies a spindle speed in a 2- to 4-digit number following the address S. Generally speaking, while the 2-digit S code specifies a speed corresponding to the code number, the 4-digit S code specifies a speed (rpm) directly. The range of numbers that can be used depends on the machine tool builder. In case a move command and a Spindle-speed function command are specified in the same block, there are available the following two ways of executing them, depending on the machine tool builder.

Refer to the description issued by the machine tool builder.

(1) A move command and a Spindle-speed function command start execution simultaneously.

(2) After completion of execution of a move command, a Spindle-speed starts execution.

3.15 Tool Function (T Function)

The Tool function is specified in a 2- to 5-digit number following the address T.

Among the T codes that can be specified, which code should correspond to which tool, depends on the machine tool builder.

In case a move command and a Tool function command are specified in the same block, there are available the following two ways of executing them, depending on the machine tool builder. Refer to the description issued by the machine tool builder.

(1) A move command and a T function command start execution simultaneously.

(2) After completion of execution of a move command, a T function command starts execution.

3.16 B Function

This function specifies positioning of an index table in a 3-digit number code following address B. Which code among B codes usable is to be made to correspond to which position, depends on the machine tool builder.

Among 4-digit S code and 5-digit T code and 3-digit B code, only one code is selectable.

Also when a move command and a B function command are specified in the same block, there are available the following two ways of executing them. Which of them to use depends on the machine tool builder. Refer to the description issued by the machine tool builder.

(1) The move command and the B function command start execution simultaneously.

(2) After completion of execution of the move command, the B function command starts execution.

3.17 Miscellaneous Function (M Function)

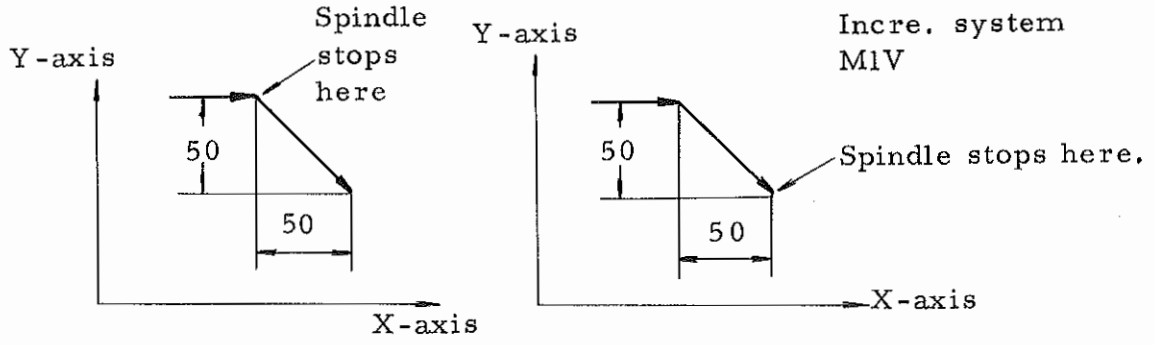
This function is specified in a 2-digit number code following address M. There are available 100 kinds of Miscellaneous functions of M00 to M99. How many codes (e.g., M10 to M15) are to be used among them, and which code is to be used for which function (e.g., M05 for spindle off), depend on the machine tool builder. Refer to the description issued by the machine tool builder. The M codes of M98 and M99 are used for sub tape control. For more detail, refer to section 3.24.

When a move command and a Miscellaneous function command are specified in the same block, there are available the following two ways of executing them. Which of them to use depends on the machine tool builder. Also, both ways may be used in the same machine tool.

(1) The move command and the Miscellaneous function command start execution simultaneously.

(2) After completion of execution of the move command, the Miscellaneous function command starts execution.

Example 3.17 G01X5000 Y-5000 M05 F200 * (spindle stop)



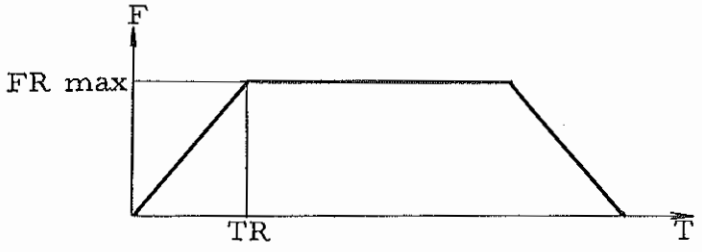
3.18 Automatic Acceleration and Deceleration

Acceleration and deceleration act automatically, requiring no programming.

Time constant of this automatic acceleration/deceleration can be set by MDI unit independently on each axes according to rapid traverse or manual feed. And for the cutting feed, time constant are set mutually on each axes.

Acceleration and deceleration behave exponentially or linearly as shown in following figurers.

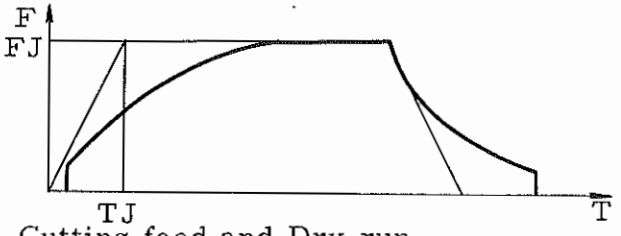
Rapid traverse (Auto and Manual)



FR max : maximum speed of rapid traverse.

TR: Acc/dec. time constant in rapid traverse

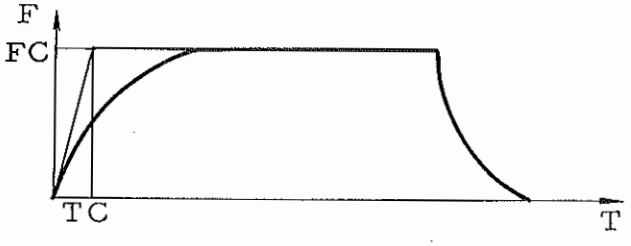
JOG



FJ : Speed in job feed.

TJ: Acc/dec. time constant in jog feed

Cutting feed and Dry run



FC: Speed of cutting feed dry run.

TC: Acc/dec. time constant in cutting feed and dry run.

Setting value of linear acc./dec. time

8 ms to 2040 ms (8 ms increment)

Setting value of exponential acc./dec. time constant (ms)

8.3	9.5	11.1	13.5
17.1	19.7	23.3	28.4
36.6	39.4	42.7	46.5
51.2	56.9	64.0	73.1
78.8	85.3	93.1	102.4
113.8	128.0	146.3	170.7
186.2	204.8	227.6	256.0
292.6	341.3	409.6	512.0

Programming notes on automatic acceleration and deceleration

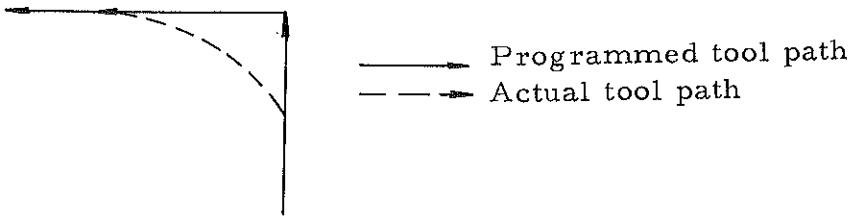
(1) Undershooting at a corner in cutting feed.

Old block \ New block	Positioning	Linear/Circular Interpolation	No movement
Positioning	○	○	○
Linear/Circular Interpolation	○	×	○
No movement	○	○	○

○ : Check of Error pulse
× : No check of Error pulse

If deceleration is effected near the end point of a block being executed, the NC equipment reads in the command of the next block, and begins to transmit command pulses; therefore, pulses for both blocks exist together for some time.

This, in a command specifying turning at a corner, causes the tool to move curvedly inside of the corner in practice.



Refer to Appendix 3.2 for this undershooting.

In order to eliminate this undershooting at the corner, a Dwell command (G04) is programmed.

The command

... *G04* .....

... *G04P0* .....

specifies the next block to be read in after the last command pulse of a block is transmitted. Undershooting at a corner may be harmful or useful.

3.19 Optional Block Skip

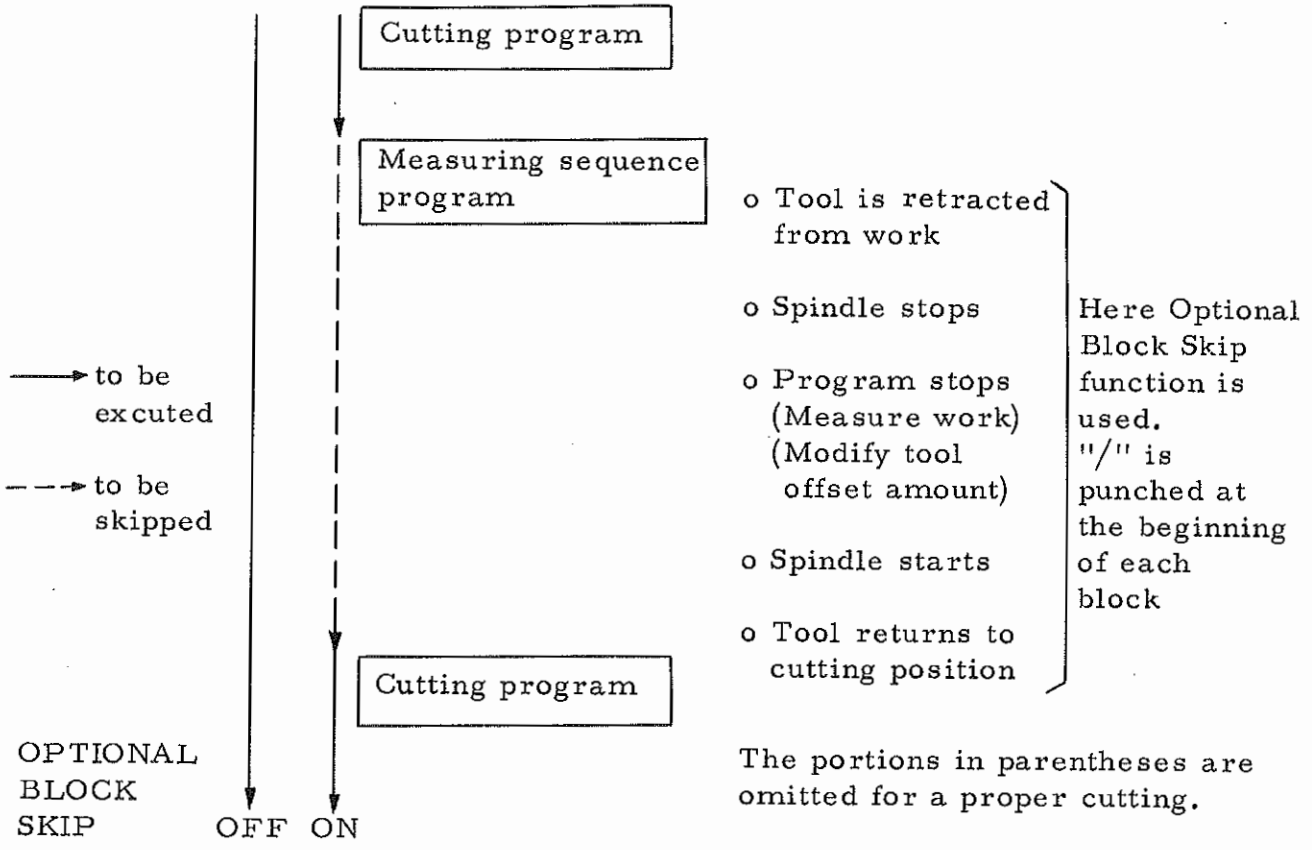
If the OPTIONAL BLOCK SKIP switch on the control panel is set to ON with a slash (/) punched at the beginning of a block, the information of that block is ignored. With the OPTIONAL BLOCK SKIP switch at OFF, this information is valid. In other words, the block containing this slash can be selectively skipped by the operator.

```
* /N123G01X4 .....  1 * Y78
```

With OPTIONAL BLOCK SKIP switch at ON, information in this range is ignored.

(Application example)

An NC tape containing a measuring sequence in a trial cutting can be applied to a proper cutting by use of this function.



Notes on Optional Block Skip

(1) This function is effective in sequence number search operation.

(2) When "/" code is programmed during one block, the commands before "/" code are effective and excuted if OPTIONAL BLOCK SKIP switch in ON.

But the command after "/" code is ignored.

(3) The skipped commands by optional block skip are not read in buffer register.

(4) If OPTIONAL BLOCK SKIP switch sets to ON after the commands is read in buffer register, these commands are excuted.

3.20 Absolute Rewind Stop & Reset

Percent (%) in ISO code and End of Record (ER) in EIA code stops a rewind.

If this code is specified within Significant Information of NC tape, a reset becomes effective.

If this code is read during manual operation with the tape reader with reels, the tape feed is stopped unconditionally.

3.21 Mirror Image

By using MDI unit, symmetrical cutting can be performed with respect to any axis. If only the X-axis is turned to REV (Reverse), a programmed drawing is reversed only with respect to X axis (+X to -X, -X to +X), and not with respect to Y and Z axes. This also applies for the each axis. Mirror Image function Fig. 3.20.1 illustrates the above description.
The setting of mirror image function is shown in section 4.1.4(4).
The mirror image function is effective from the next block.

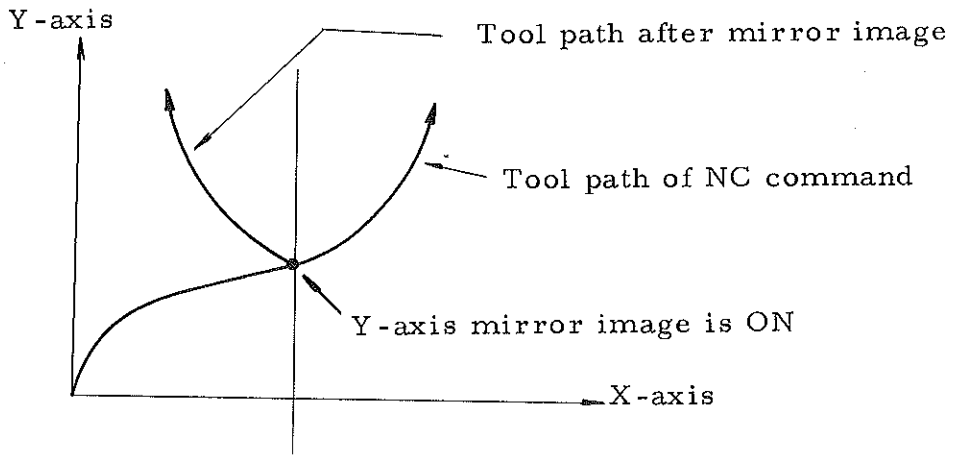
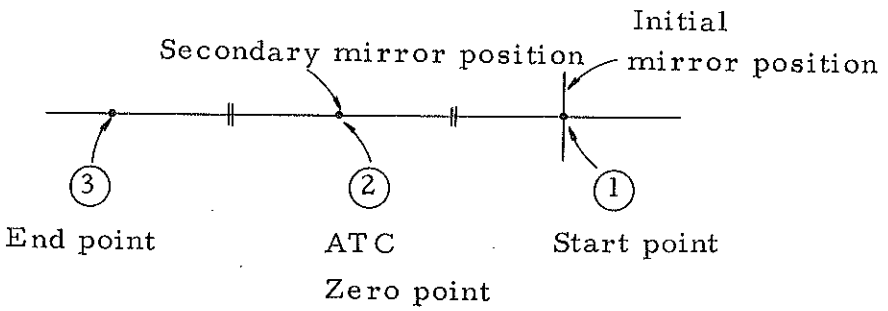


Fig. 3.20.1 Mirror image

When the Autonatic Zero return operation is performed by G28 command in REV position, the next return operation must be used by G29 command. If this operation is commanded by absolute programming, end point is shifted as follows.

(Example)

① N001 G92 Y0 *
② N002 G28 Y0 *
③ N003 G00 Y0 *



3.22 Sequence Number Display & Search

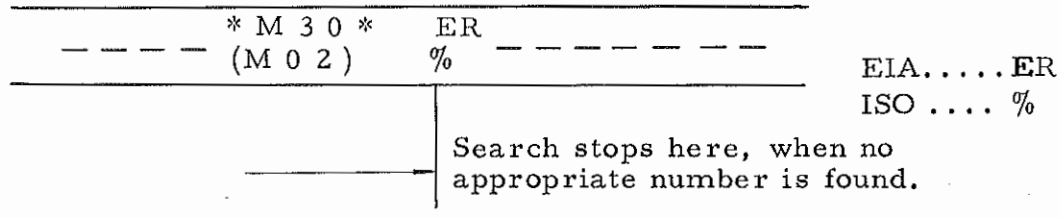
The Sequence Number is specified in a 3-digit number code following address N (: in ISO code or o in EIA code). The number must always be of three digits. The Sequence Number on NC tape is displayed on the DPL unit by selecting the address N using ADDRESS key. Also sequence number can be searched by MDI unit. The Sequence Number can be specified in each block as well as in blocks required only.

The Sequence Number is required for the blocks for which

* the Sequence Number Search is to be performed,
* the Program Copy is to be performed, where the beginning and the end of a portion to be repeated are included and the command block of Program Copy is specified, and
* the sub tape control is to be performed.

Programming considerations for efficient use of the Sequence Number function

(1) Since Sequence Number search is effected only in the forward (normal) direction, when the appropriate number is not found during search, it is necessary to punch the End of Record (ER or %) previously or the miscellaneous function (M30) for tape rewind.



(2) In order to restart maching with the block in which the Sequence Number was searched out, information necessary for restart must have been punched in the subsequent blocks. Also, the label skip function and the optional block skip function are effective. The information necessary for restart is as follows:

* Absolute coordinate value of each axis.
* B function
* M function (spindle on, coolant on, etc.)
* S function (spindle speed)
* T function (tool select)

Restart is, sa a rule, impossible with a block.
whereby offset is being executed.

(3) When the Sequence Number Search is effected, NC tape stops just on the next character.
At this time the Buffer lamp turns off, MDI operation is possible Setting TAPE mode and pushing the START button, the machine begins to excute from the block which includes the searched sequence number.

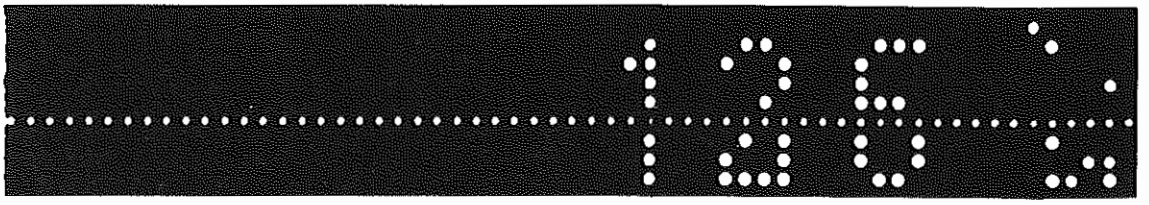
And when you push RESET button besides MDI mode, as the label skip function becomes effective automatically, the machine begins to excute from next block.

3.23 Label Skip Function and Control In/Out

These are functions for the NC equipment to partially ignore information punched on NC tape, and come in the following two kinds:

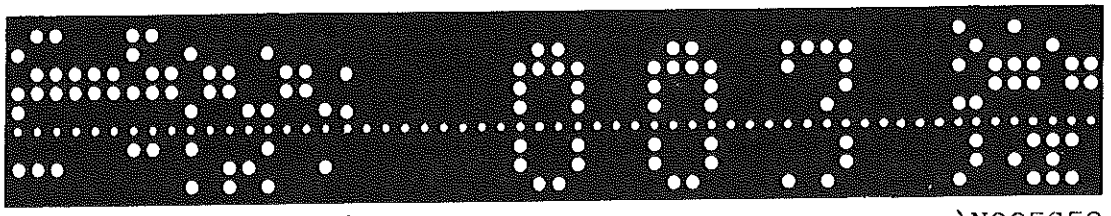
(1) Label Skip function

It is possible to punch an identifier at the head of an NC tape previously. The NC equipment ignores this identifier, and after reading the EOB code (*), operates according to the subsequent information. While this function is in effect, the LABEL SKIP lamp on the display panel is on (see Section 4.1.1 (8)).



(2) Control In/Out

The NC equipment ignores all information between Control In and Control Out on NC tape; therefore, it is possible to punch an identifier, etc. in any code in this range previously (for ISO code only).



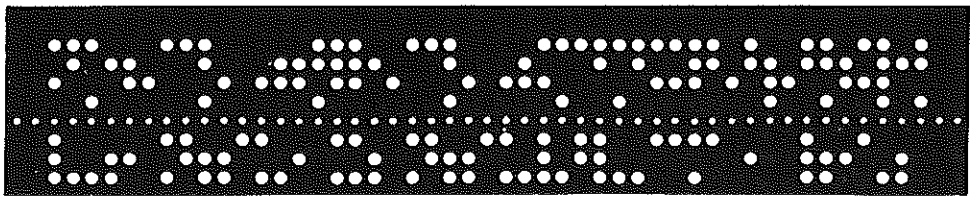
Z22000T40M03*M00*( )N025G50

Significant Information ——— Identifier etc. ——— Significant Information

This function is effective only within Significant Information. When power is turned on, and when the equipment is reset, it is in Control-In state. In this range, however, ER or % (Absolute Rewind Stop & Reset) must not be used. Also, * (EOB code), even if specified, is insignificant.

Examples using Label Skip function and Control In/Out

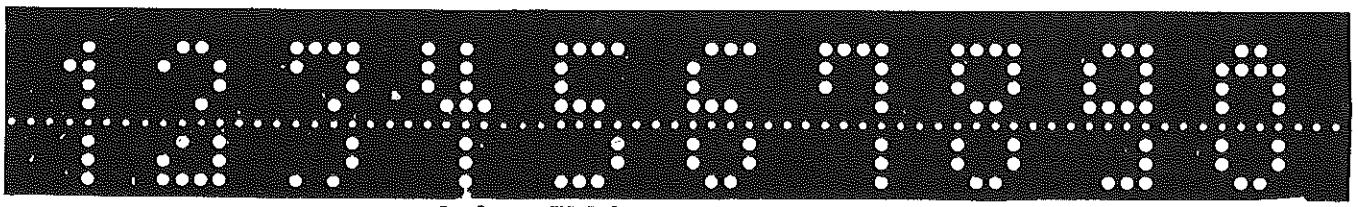
Example 3.23.1 Example of tape identifier for typing out



PARTS NO.345/SHEET NO.345/PROGRAMMED BY G.SMITH

(in EIA code)

Example of tape identifier for use of the punched pattern



Od TNRX A-RF +8d8 IRRO tRRO +--d FRRF YRRd tAAt

d = Delete

t = Tab

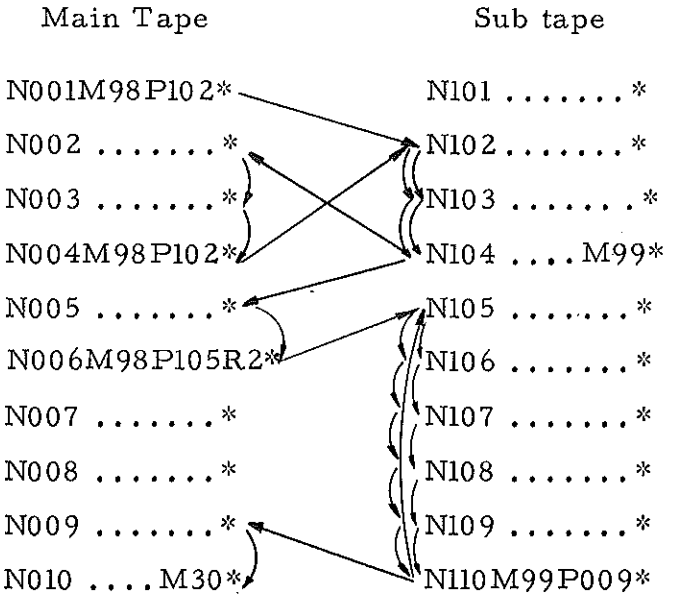
(in EIA code)

3.24 Sub Tape Control

The sub tape stored in the memory is called from main tape, and NC is controlled by it.

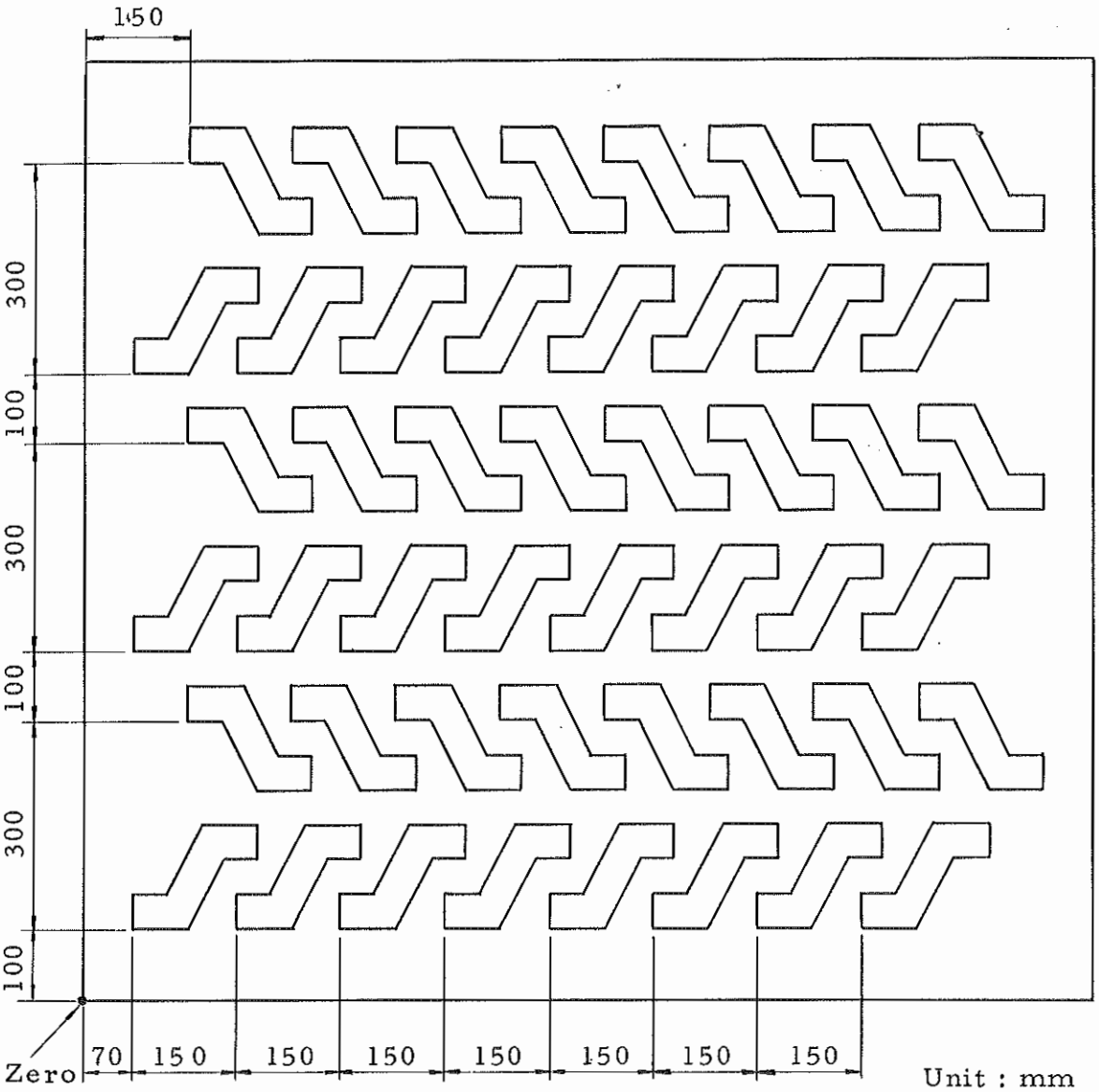
M98 specified on a main tape permits switching to a sub tape, and M99 specified on a sub tape permits returning to a main tape.

By specifying a Sequence Number in a 3-digit P code following M98/M99, execution is started with the block of that Sequence Number of the tape switched to.



The number of repetations can be specified by use of address R.

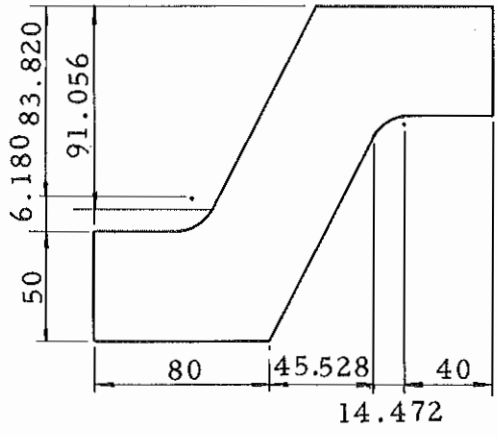
Example 3.24 Programming using sub tape control



Unit : mm

Increment system : M2V

(Detail of work)



(a) Programming of main tape

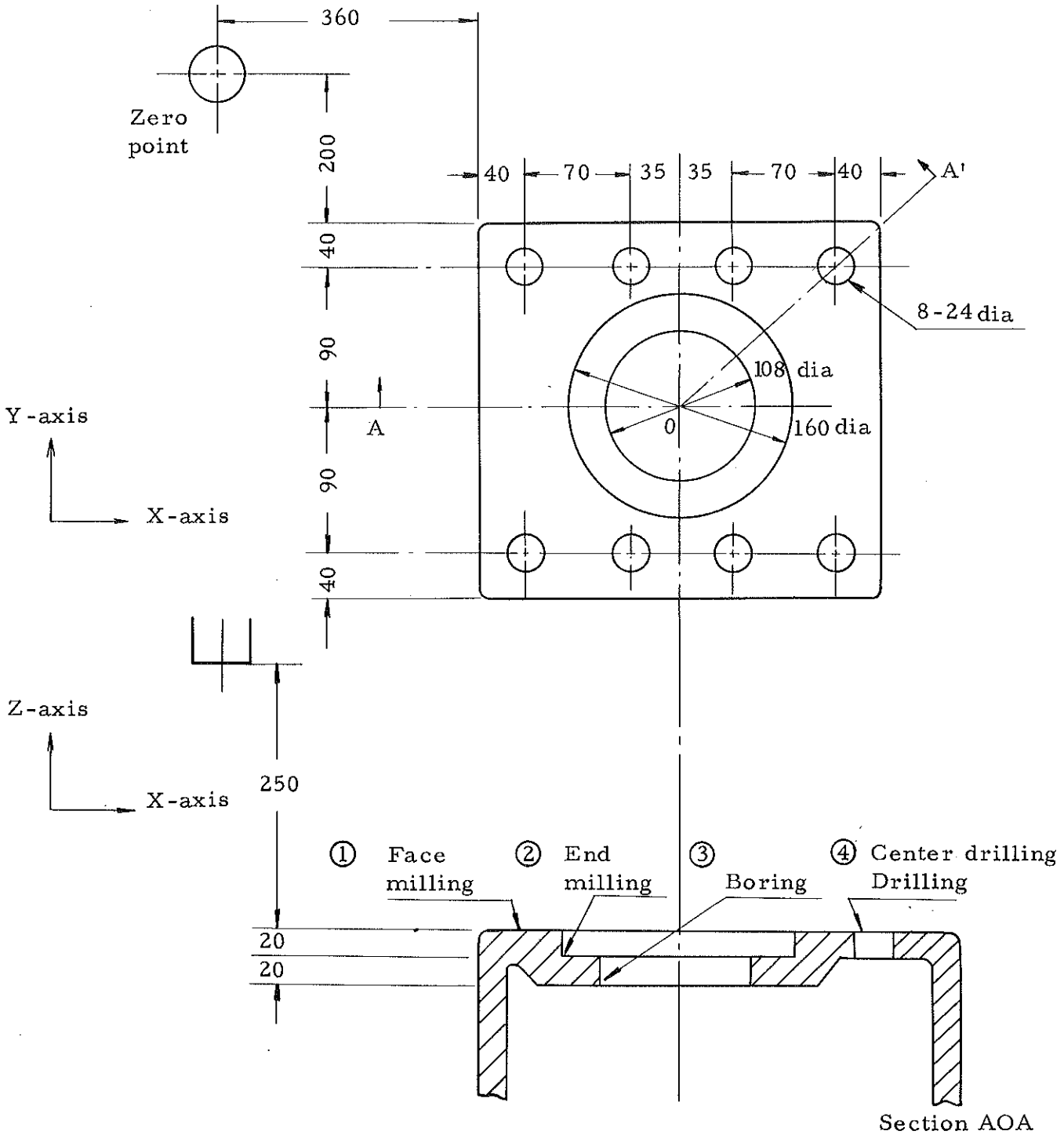
```
*

N001  T11 M06 *
N002  S1400 M03 *
N003  G92X0Y0Z0 *
N004  G90G45Z-123000H01 *
N005  G90G00X60000Y90000 *
N006  M98P100R8 *
N007  G90X140000G91Y300000 *
N008  M98P200R8 *
N009  G90X60000G91Y100000 *
N010  M98P100R8 *
N011  G90X140000G91Y300000 *
N012  M98P200R8 *
N013  G90X60000G91Y100000 *
N014  M98P100R8 *
N015  G90X140000G91Y300000 *
N016  M98P200R8 *
N017  G90X0Y0 *
N018  G45Z0 *
N019  M05 *
N020  M02 *
```

(b) Programming of sub tape

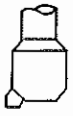
```
N100G91G01Z-45000 F80 *
N101G17G41X10000Y5000J10*
N102Y55000*
N103G39I10*
N104X40000*
N105G03X14472Y8944J16180*
N106G01X45528Y91056*
N107G39I10*
N108X80000*
N109G39J-10*
N110Y-50000*
N111G39I-10*
N112X-40000*
N113G03X-14472Y-8944J-16180*
N114G01X-45528Y-91056*
N115G39I-10*
N116X-85000*
N117G40X-5000Y-10000*
N118Z45000*
N119G00X150000 M99 *
N200G91G01Z-45000F80 *
N201G17G41X10000Y5000J10*
N202Y55000 *
N203G39I10*
N204X80000*
N205G39I10J-20*
N206X45528Y-91056*
N207G03X14472Y-8944I14472J7236*
N208G01X40000*
N209G39J-10*
N210Y-50000*
N211G39I-10*
N212X-80000*
N213G39I-10J20*
N214X-45528Y91056*
N215G03X-14472Y8944I-14472J-7236*
N216G01X-45000*
N217G40X-5000Y-10000 *
N218Z45000*
N219G00X150000 M99 *
```

3.25 Programming Examples

A programming example for the following work is shown as follows.



Types and specifications of the tools

Tool used (Tool No.)	Spindle speed code	Tool length/ diam. (mm)	Tool Offset No.	Offset amount
Face mill (01) 	50	Length 150 Diam. 120	00 01	+15000 + 6000
End mill (02) 	53	Length 140 Diam. 30	02 03	+14000 +1500
Boring bar (03) 	40	Length 160 Diam. 108	04 /	+16000 ———
Center drill (04) 	58	Length 120 Diam. 3	05 /	+12000 ———
Drill(05) 	54	Length 180 Diam. 24	06 /	+18000 ———

	ER*	
N001	G80T01*	Fixed Cycle cancel, tool selection (face mill)
N002	M06*	Tool change
N003	G92X0Y0 Z0*	Insert absolute preset
N004	G46G00Z-25000H00S50M03*	Positioning, Tool Offset, spindle-speed selection, spindle start
N005	X36000*	Positioning
N006	G46Y-19500D01*	Positioning, Cutter radius compensation
N007	G91G47G01Y-27000F380*	Face milling
N008	G45G00X0*	"
N009	G47G01Y27000*	"
N010	G45G00X0*	"
N011	G47G01Y-27000*	"
N012	G45G00X0*	"
N013	G47G01Y27000*	"
N014	G45G00X0*	"
N015	G47G01Y-27000*	"
N016	G46Z25000H00M05*	Tool length compensation, spindle stop
N017	G47G00X-0D01*	
N018	G47X-0*	
N019	G45Y-0*	
N020	G28Y26500*	ATC Zero Return
N021	T02*	Tool selection (end mill)
N022	M06*	Tool change
N023	G29X14500Y-13000S53M03*	Positioning, spindle-speed selection, spindle start
N024	G46Z-27000H02*	Positioning, Tool length compensation
N025	G13 I8000F130D03 *	End milling

N026	G46Z27000H02M05*	Positioning, Tool length compensation, spindle stop
N027	G28X-14500Y13000*	ATC Zero Return
N028	T03*	Tool selection
N029	M06*	Tool change
N030	G29X14500Y-13000S40M03*	Positioning, spindle-speed selection, spindle on
N031	G46Z-26500H04*	Positioning, Tool length compensation
N032	G01Z-3000F50*	Linear Interpolation (drilling)
N033	Z3000F200*	Linear Interpolation (retraction)
N037	G00G46Z26500M05*	Positioning, Tool length compensation spindle stop
N035	G28X-14500Y13000*	ATC Zero Return
N036	T04*	Tool selection (center drill)
N037	M06*	Tool change
N038	G29X-3000Y-4000S58M03*	Positioning, spindle-speed selection, spindle start
N039	G46Z-24800H05*	Positioning, Tool length compensation
N040	G81X7000Z-500R0L4F300*	Fixed Cycle (drilling cycle)
N041	X-21000Y-18000*	"
N042	X7000L3*	"
N043	G80G46G00Z24800*	Fixed Cycle cancel, positioning, Tool length compensation
N044	G28X-25000Y22000M05*	ATC Zero Return, spindle off
N045	T05*	Tool selection (drill)
N046	M06*	Tool change
N047	G29X-3000Y-4000S58M03*	Positioning, spindle-speed selection, spindle on
N048	G46Z-24800H06*	Positioning, Tool length compensation
N049	G81X7000Z-3000R0L4 F350*	Fixed cycle (drilling)
N050	X-21000Y-18000*	"
N051	X7000L3*	"

N052	G80G46G00Z24800*	Fixed Cycle cancel, Positioning, Tool length compensation
N053	G28X-25000Y22000M05*	ATC Zero Return, spindle stop
N054	M02*	Reset & Rewind

The M, S and T functions, or adequate machining conditions vary with the machine tool used, work material, preprocessing, etc. Refer to the description issued by the machine tool builder for details.

4. OPERATION

4.1 MDI & DPL Unit

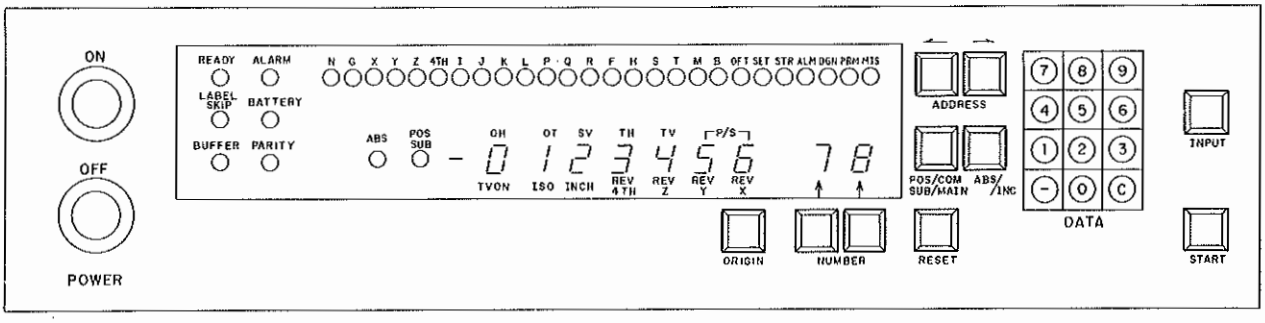


Fig. 4.1.1 MDI & DPL Unit

4.1.1 Display Section

(1) READY Lamp

This lamp lights up when the control unit and the servo unit wave become ready.

(2) LABEL SKIP lamp

This lamp is on when the Label Skip function is in effect. That is, it is on in cases when

(a) the power is turned on, and
(b) the control unit is reset (in a mode other than MDI mode) .

(3) BUFFER lamp

This lamp is on when data is contained in the buffer, and goes out by reading out of data.

(4) BATTERY lamp

This lamp is on when battery voltage become under set value. These batteries are used for memory protection at power off.

(5) PARITY lamp

This lamp is on when data is party-checked at NC.

(6) ALARM lamp

This lamp is on when the alarm (OH, OT, SV, TH, TV or P/S) is detected, and "0" is displayed under the appropriate error label in DPL unit.

(7) ABS lamp

By ABS/INC switch, this lamp is on when the ABS position is selected.

(8) POS, SUB lamp

By POS/COM, SUB/MAIN switch, this lamp is on when the POS or SUB position is selected.

4.1.2 Switch and Button Section

(1) POWER ON/OFF button

If this button is pushed, the power supplies turn on or off in a determined order.

(2) RESET button

Generally speaking, this button is used to release an alarm. The modal commands among the commands already executed are not canceled. If this button is pushed while a move command or an auxiliary function (B, S, T or M) command is being executed, the other move commands in the block concerned are canceled, the machine tool stops with deceleration, and the command for auxiliary function operation is reset. Also, the contents of the buffer are canceled except for MDI or EDIT mode, and the BUFFER lamp goes out. Further, the position display unit displays the position of the machine tool.
Also the offset vector is canceled by this button.

(3) ORIGIN button

At the very time this button is pushed, the position of a tool is assumed to be the coordinate zero point in an Absolute coordinate system. That is, this action provides the same effect that the following NC command is executed.

G92 X0 Y0 Z0 α 0*

where α stands for additional axis.
This button is effective only when in an automatic operation stop state or in a reset state.
In cutter compensation mode (G41, G42), this button must not be pushed.

(4) ABS/INC switch

When MDI mode and COM position are selected, this switch is effective as follows.

ABS: Absolute values are inputted and displayed for address X, Y, Z and 4th-axis.

INC: Incremental values are inputed and displayed for address X, Y, Z and 4th-axis.

(5) POS/COM, SUB/MAIN switch

This switch has two functions.

(a) This switch is used for selecting the display of an absolute coordinate value or a command value (Refer to Item 4.1.3)

(b) This switch is used for selecting the main tape (MAIN) or sub tape (SUB).

(6) ADDRESS key

Address and functions are selected. By deppressed these buttons, the address are sifted to arrow direction.

(7) NUMBER switch

These switches are used for tool offset number input, display of G code, tape editing and diagonose.

Two switches are related with 2-digit number.
By pushing this button, the displayed value is added of value "+1".

(8) INPUT button

This is a push button for data input.

(9) START button

This is a push button for starting.

4.1.3 Display Unit Section

You can display the value of any addresses by selecting the appropriate address using the ADDRESS key.
However the meaning of the displayed value differs from each other as the table shown below depending on the state of POS/COM and ABS/INC switches.

<table>
<tr><th rowspan="2">POS/COM
ADDRESS</th><th rowspan="2">POS</th><th colspan="2">COM</th></tr>
<tr><th>ABS</th><th>INC</th></tr>
<tr><td>X, Y, Z, 4TH</td><td rowspan="2">The absolute value of X, Y, Z or 4th-axis coordinate which was set using the G92 is displayed.</td><td>The input data in the MDI mode is displayed in absolute value.</td><td>The input data in the MDI mode is displayed in incremental value.</td></tr>
<tr><td colspan="2">In other case, blank is displayed</td></tr>
<tr><td>I, J, K,</td><td colspan="3">The input data in the MDI mode is displayed.
In other case, Blank is displayed.</td></tr>
<tr><td>L</td><td colspan="3">Those input data in the MDI mode is displayed.
In other case, the rest value is displayed, and blank is displayed at last cycle.</td></tr>
<tr><td>P, Q, R</td><td colspan="3">The input data in the MDI mode is displayed.
The modal value of P, Q, R in fixed cycle is displayed.
In other case, blank is displayed.</td></tr>
<tr><td>N, G, F, H,
S, T, M, B</td><td colspan="3">The input data in the MDI mode is displayed.
In other case, the model value is displayed.</td></tr>
<tr><td>OFT</td><td colspan="3">The offset value of the specified number on NUMBER key is displayed.</td></tr>
<tr><td>SET</td><td colspan="3">The state of the setting is displayed</td></tr>
<tr><td>STR</td><td colspan="3">The information of NC tape is read in and is stored in memory.</td></tr>
<tr><td>ALM</td><td colspan="3">When ALARM lamp is on, "0" is displayed under the appropriate error label (OH, OT, SV, TH, TV).
In the case of the P/S error, error number is displayed under P/S label. When ALARM lamp is off, blank is displayed.</td></tr>
<tr><td>DGN</td><td colspan="3">The state of the signal to be checked is displayed.
For details, refer to the maintenance manual.</td></tr>
<tr><td>PRM</td><td colspan="3">The parameters such as time constant values etc are displayed.</td></tr>
<tr><td>MIS</td><td colspan="3">Blank</td></tr>
</table>

Note 1) "The input data in the MDI mode" is the state of the data after setting the data and before pushing the START button in the MDI mode.

Note 2) G code of each group can be displayed independently. In this case, designation of group is used of NUMBER key.

NUMBER key	Group
00	*
01	A
02	B
03	C
04	D
05	E
06	F
Blank	the other

(Note 3) In EDIT mode, the contents of addresses X, Y, Z, 4TH, I, J, K, L, P, Q, R, N, G, F, H, S, T, M and B are displayed according to the POS/COM switch as follows.

POS : Contents of sub tape.
COM: Contents of main tape.

(Note 4) Address 4TH is additional axis (A, B, C, U, V and W axis). Address B is B 3-digit.

(1) OH (Over heat) alarm

When the temperature in the control unit or servo motor exceeds a tolerance, Alarm lamp lights up, address ALM position is selected by ADDRESS key and "0" is displayed on OH position. In this case, the equipment is placed into the Single Block mode, and stops after completion of a block currently being excuted.

While the Alarm lamp is on, a restart by the CYCLE START button on the control panel is impossible.
By reset after the temperature becomes lower than standard the lamp goes out. At this time, a restart is effected by pushing the CYCLE START button on the control panel.

(Fault causes and remedies)

(a) When the temperature in the control unit rises

(i) If the ambient temperature is in excess of 45°C, lower it.

(ii) If the air filter of the servo unit clogged, clean the air filter (refer to Section 6.1).

(iii) If the cooling fan motor is in failure, communicate to a maintenance engineer or Fujitsu FANUC.

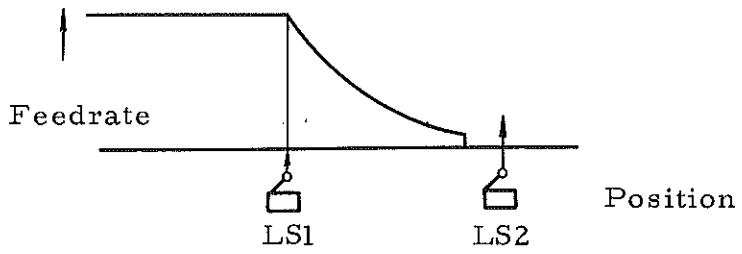
(b) When the servo motor is overheated.

(i) If the machining requirement is too severe, reexamine the program, or cool the servo motor.

(2) OT (Over Travel) alarm

When the movable member of the machine tool reaches the stroke end, this OT alarm is on as same as OH alarm.

Two limit switches (LS1, LS2) are provided on each axis in each direction (at the stroke end on +X, -X, +Y, -Y, etc.), as shown in the figure below, to stop the movable member of the machine tool.



When LS1 operates, the feed is stopped after deceleration, and the OT lamp on the display panel lights up. When LS2 operates, the feed is emergency stopped (LS2 operates only if LS1 should malfunction).
operates, the feed stops emergently. (LS2 operates only if LS1 should malfunction).

(Fault causes and remedies)

(i) Fault of setting zero point. → alteration of programming

(ii) Programming error → alteration of programming

Release

(a) When only LS1 operates

Move the movable member of the machine tool by manual operation (Jog, Step or Handle) in the opposite direction (safe side) to separate it from the limit switch, and then push the RESET button on the display panel.

Note: In this case, the movable member of the machine tool can be moved only in the opposite direction.

(b) When both LS1 and LS2 operate

(i) Set the 2ND L.S REMOVE switch to ON.

(ii) Conduct the same operation as in releasing LS1.

Note: In the equipment in which LS1 is not in operation when LS2 is in operation, the movable member is movable in both directions by manual operation. Be careful of the direction in which it is to be moved.

(3) SV (Servo) Alarm
The SV alarm is on for one of the following causes:

(a) The detect level is lowered (the signal level from the detector such as the inductosyn, resolver, etc. is lowered).

(b) The error register overflows (an error between a specified value and the machine position by the detector becomes excessive).

(c) The servo motor is overloaded.

Remedies

Communicate to a maintenance engineer. At this time, observe the following conditions.

* Did the machine tool malfunction?

* Could the alarm be released by the RESET button on the display panel?

* Could the alarm be repeated by any means?

* Has any fuse in the servo unit blown out?

(4) TH (Tape Horizontal Check) alarm

If any of the tape codes that are not included in the list of functional codes (Appendix 1) is detected within Significant Information (except a range from Control Out to In) on NC tape, tape reading is stopped (the tape stops on a character to the erroneous one), the control unit, when executing the preceding block, stops after completion thereof, and the TH alarm is on as same as OH alarm. A block having an erroneous character is all invalid and is not executed.

The TH alarm goes out by reset.

(Causes and remedies)

* If the NC tape or tape reader is dirty, clean them.

* If an ISO NC tape is loaded on the equipment for EIA code, replace the tape. EIA/ISO Switching (see 4. 1. 4)

* If an NC tape is put on its front side, set the tape again correctly.

* If an NC tape is punched erroncously, correct the tape. Examine the hole pattern of a character detected as an error, and further display the information contained in the buffer on the DPL unit, to study the error cause.

(5) TV (Tape Vertical Check) alarm

If in parity checking a block of NC tape in the vertical direction, an odd number of characters are detected therein (from a character next to *(EOB) to * , the control unit, when executing the preceding block, stops after completion thereof, and the TV alarm is on.

TV check is also effective for the block to which the optional block skip is ON.

A block having an error is all invalid.

The TV alarm goes out by reset.

This TV checking function can be set effectively or non effectively by setting of MDI unit.

Remedies

* Punch any code (Spáce, for example) ignored by the NC equipment before *EOB so that the number of characters is even.

* If this lamp lights up with an even number of characters punched, since the tape reader is considered to read erroneously, clean the reading section thereof, or the NC tape.

(6) P/S (Program/Setting Check) alarm

This alarm is on when there is generated any contradiction in executing processing, due to an error in programming, or in setting on the MDI unit.

The P/S alarm goes out by reset.

Remedies

It is possible to display a number indicating the error contents on the DPL unit.

Table 4.1.1 shows the error numbers and their corresponding error contents.

Table 4.1.1 P/S alarm

Error No.	Error contents	Remarks
03	Data exceeding the allowable number of digits is entered. (7 digits)	For the tape only
04	There is no address word, but a number or a sign (-) at the beginning of a block.	For the tape only
05	There is no data, but the next address word or * EOB immediately after an address word.	For the tape only
08	The one block characters exceed 50ch.	
10	An unusable G code is specified.	
15	The simultaneous controlled axes exceed the max. programmable axes.	
21	A vector or axis for the other plane is specified.	
25	In circular interpolation, zero is specified for 1-digit F code (F0).	
26	F9 is designated in 1-digit F code	

Error No.	Error contents	Remarks
30	A D code for Cutter Compensation or an H code for Tool offset is specified to be too large.	
31	G38 or G39 is specified in Cutter Compensation cancel mode or out of offset plane.	
35	The Z-axis commands are not specified after G43, G44 or G49 codes.	
36	Any one of G45 to 48 is specified in Cutter Compensation mode.	
37	G40 is specified in outside mode of a specified plane.	
44	Any one of G27 to G29 is specified in Fixed Cycle mode.	
48	After power on, move command is performed without zero return. This check is only effective with option "Stored Pitch Error Compensation".	
52	A block cannot be found having a Sequence Number specified by P for Program Copy.	
60	In Sequence Number Search, a specified Sequence Number is not found.	
70	Memory overflow	
71	In tape edit mode, a specified block is not found.	
72	In tape edit mode, an unusable address is seleded by ADDRESS key.	
73	In tape edit mode, data is not key input or only minus sign	
74	The words or blocks for deletion is not found.	
75	The insertion of block is not operated.	

Error No.	Error contents	Remarks
76	In tape read or punch mode, error is found in ADDRESS key or NUMBER key setting.	
77	The sequence number is not designated in 3-digit.	
79	In tape edit mode, OPTIONAL Block Skip is ON.	
90	Lead (K) of helical interpolation is not designated.	
91	Repeat number (L) of helical interpolation is not designated or is not positive.	

(Note)

During tape operation when the TV alarm or the P/S alarm except for error No. 8 have generated, the tape stopps at the next place of the EOB including error block. And at this time buffer content is null.

Pushing reset button for alarm release, as the Label Skip lamp turns on, the next block of alarm generated cannot be read at the start. So next operation is necessary, that is,

(1) Turn to the MDI mode not to light up the Label Skip lamp.

(2) When the label Skip lamp turns on, shift the tape before the EOB point by manual operating.
For the TH alarm or the P/S alarm of error No. 8, as the tape stops just the next character point of the error including (during that block), by turning on the Label Skip button the tape continue to read. But when the point of error generated is the end of block, the condition is same as the case of TV, P/S alarm.

4.1.4 Manual Data Input (MDI) Section

The Manual Data Input provides key board, rotary switch, two digits thumbwheel switch and a button to input command or data mounted on the Setting/Display panel.

The key board (DATA key) is used to set numerical data, the key board switch (ADDRESS key) is used to specify the functions or the meanings of the numerical data, the key board switch (NUMBER key) is used to set a tool number for tool offset etc., and the input button is used to store the data set from the switches in memory. These switches are only enabled, or active, when the Mode Select switch on machine side is in the Manual Data Input mode.
Each operation is explained as follows:

(1) One block operation

The block which can be commanded on input paper tape can be commanded using MDI operation also. Through the block including N can not be commanded because N is used for the sequence number search. The operation is performed as follows:

(a) Select the address using ADDRESS key to a desired address.

(b) Key in data using DATA key starting from the sign (if minus) to the least significant digit number. At this time the input data of X, Y, Z or 4th axis is displayed in COM mode.

(c) Push the INPUT button. If the address is not G, X, Y, Z or 4-th, display is unchanged.

When the address of G code is selected and group number of input G code is set by NUMBER key, the input data can be displayed. The Input of G code is possible independent of the displayment of number key.

(d) Repeat (a) (b) (c) for each address word in the block.

(e) Push START button to start the command.

When the mode select switch is changed to the other mode before pushing START button, all input data are canceled by (a) to (c) operation.

(2) The tool offset or cutter compensation data input

The tool offset or cutter compensation data is inserted as follows:

(a) Select the address using ADDRESS key to OFT.

(b) Set offset number on the NUMBER switch.
Old offset value is displayed.

(c) Key in offset data using DATA key.
New offset value is displayed.

(d) Push the INPUT button.

This operation is available at MDI or other mode and also during automatic feed state.

Example of setting

Tool offset value 10mm

Increment system	Input value
M1V	1000
M2V-M4V I1V	10000
I2V-I4V	100000

In case of cutter compensation, tool radius value should be set.

(3) Sequence number search

The block with specified sequence number can be searched.

(a) Select the address using ADDRESS key to the position N.

(b) Key in sequence number using DATA key.

(c) Push the INPUT button to search the sequence number.

(4) Setting

This function is used to switch TV ON/OFF, ISO/EIA, INCH/MM and/or mirror image ON/OFF.

(a) Select the address using ADDRESS key to SET position.

(b) Key in number 1 or 0 in the position TV ON, ISO, INCH, REV 4TH, REV Z, REV Y and REV X.

(Example : in order to set TV ON=1 (ON), ISO=0 (EIA), INCH=1 (INCH), and no mirror image, 1010000 should be keyed in)

The numbers keyed in are displayed.

(c) Push INPUT button.

Note: If you use the remote mirror image as anadditional option, when either mirror image is REV, the function of mirror image is effective.

(5) Diagnose input/output signals.

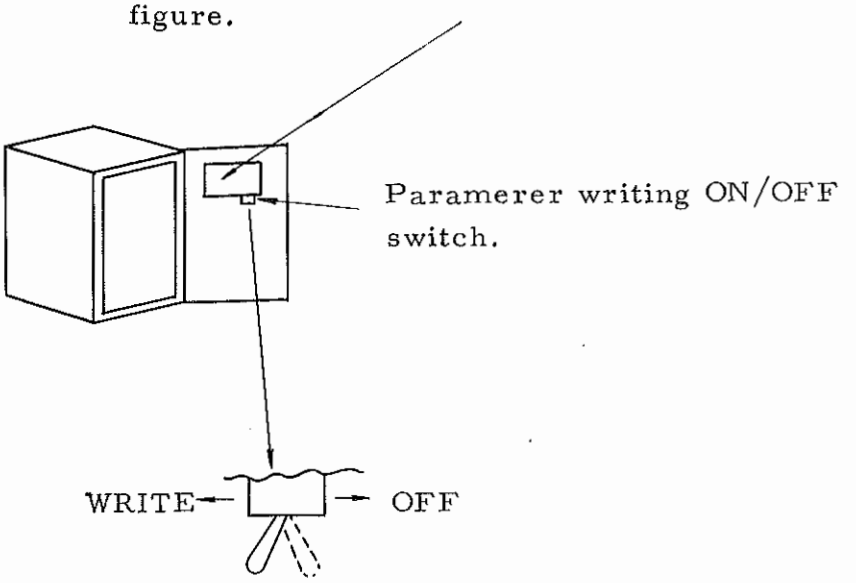
The DGN position is used to check Digital input/output signal. The operation for DGN is explained in the maintenance manual.

(6) Setting parameters

The PRM position is used to set parameter, time constant values, rapid travers speed, backlash pulse number, etc.

(a) The process of parameters setting

(i) Turn on the parameter writing switch as next figure.



(ii) Turn on the mode select switch to MDI

(iii) Select the address to the position PRM

(iv) Set the parameter number using Number Key. (At this time previous setting parameter is displayed)

(v) Input the parameter using DATA switch. Every time DATA key is pushed, the number shifts left to left end.

(vi) After admitting displayed data number, push the input button.

(vii) Turn off the parameter writing switch.

Note of parameter setting

(Note 1) If you mistake the data number, pushing [C] Key of DATA switch and put the correct data number in.

(b) The method of displayment of parameter.

(i) Select the address to the position PRM

(ii) Setting the parameter number to be displayed using Number Key.

(c) The method of parameters setting. (Backlash pulse number)

(i) Set samely as (i) to (iii) of sequence (a)

(ii) Set the parameter number using NUMBER KEY

Backlash pulse number	Parameter number
X axis backlash	No. 20
Y axis backlash	No. 21
Z axis backlash	No. 22
4th axis backlash	No. 23

(iii) Set samely as (i) to (iii) of sequence (a).

Pulse	0	1 (CH)	2 (OT)	3 (SV)	4 (TH)	5 (TV)	6 ⌐ (P/S) ¬	7
0		0	0	0	0	0	0	0
1		0	0	0	0	0	0	1
2		0	0	0	0	0	1	0
3		0	0	0	0	0	1	1
4			0	0	0	1	0	0
				0	0	1	0	0
252	1	1	1		0	1	0	1
253	1	1	1					
254	1	1	1	1	1	1	1	0
255	1	1	1	1	1	1	1	1

0 to255 pulses of backlash can be set.

(d) The method of parameters setting. (Backlash initial direction)

(i) Set samely as (i), (ii), (iii) of sequence (a).

(ii) Set the parameter number using number key.

Backlash initial direction	Parameter number
X axis back lash initial direction	No. 4
Y axis "	No. 5
Z axis "	No. 6
4th axis "	No. 7

(iii) Set samely as (v), (vi), (vii) of sequence (a).

Number Key	0	1 (OH)	2 (OT)	3 (SV)	4 (TH)	5 (TV)	6 ┌P/S┐	7
04		ISNX						
05		ISNY						
06		ISNE						
07		ISN4						

ISN2 : Setting of backlash initial direction of α-axis.

0 : Initial direction at Power ON is +
(After movement of setting amount to the + direction it stops to move.)

1 : Initial direction at Power ON is -
(After movement of setting amount to the - direction; it stops to move.)

(7) Others

At the time of ALARM lamp lights up, selecting the address to the position ALM, the kinds of alarm can be displayed.

Manual Data Input operation can not be used at the middle of execution execution except offset value input.
It can be operated at the following time:

(a) After stop by single block.
(In case of feed hold during fixed cycle, ATC zero return, MDI operation is not available.)

(b) After pushing RESET button on the machine side.

(8) Part program storage and editing

Tape can be stored in memory and the contents of tape can be edited. Tape program stored and editing is available

both main tape and subtape. These are changed by POS/COM, SUB/MAIN switch.

COM, MAIN : Main tape

POS, SUB : Sub tape

(a) The NC tape can be stored in memory

(i) Check ER or % code on NC tape end, and set the NC tape on tape reader unit.

(ii) Set the Mode select switch to EDIT.

(iii) Set the POS/COM switch as follows.

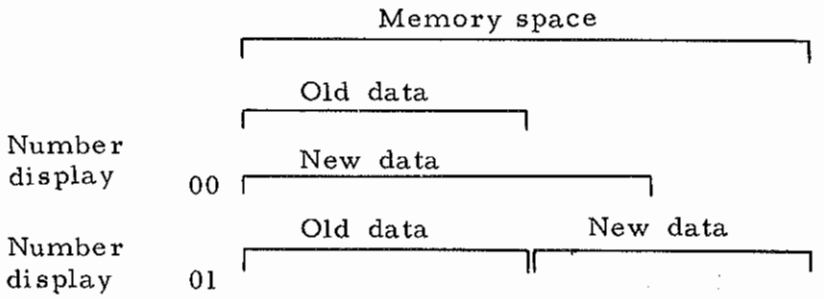
MAIN, COM : Main tape

SUB, POS : Sub tape

(iv) Select the address using ADDRESS key to STR position.

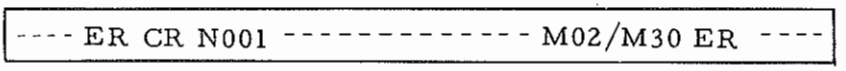
(v) Set the NUMBER display 00 or 01.

At this time memory contents ara as follows.



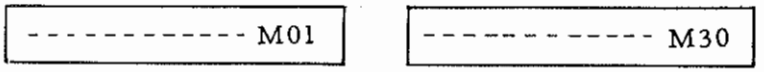
When you want to store your NC tape in memory, the NC tape should be prepared as follows.

An ER code (% code for ISO) must be punchecl at end.



If you want to store the new data follwing the old data by setting "01" on the number display, not the reset & rewind (M02 or M30) but M00 or M01 must be punched at the old data.

NC tape of old data NCtape of new data



(vi) Press the START button

(b) The MDI data can be stored in memory

(i) Set the Mode select switch to EDIT

(ii) Check OFF position of optional Block Skip switch.

(iii) Set the POS/COM switch as follows

MAIN, COM : Main tape

SUB, POS : Sub tape

Clear the NC tape's command stored in memory by next sequence.

(iv) Select the address using ADDRESS key to STR position.

(v) Set the NUMBER key to 00

(vi) Push the START button without tape setting.

(vii) Push the RESET button.
At this time, the data in memory are cleared and the indication lights out.

(viii) A block can be inserted by repeating the operation of 4.1.4 (8) (i) on the MDI & DPL unit.

Advantage

NC controll is possible without making tapes for simple program by this method.
When you want to memorize the plural blocks of commands, operation (viii) is necessary at the beginning of the block.

(c) The tape informations stored in memory can be operated.

(i) Set the Mode select switch to MEMORY.

(ii) Push the START button after confirming the start position of excution.
Start position of excution is as follows.

After the RESET in EDIT mode ⇒ From Top of NC command stored in memory.

After the Reset & rewind (M02 or M30 ....
refer to the description issued by machine tool builder ⇒ From top of NC command stored in memory.

After the operation of item (d) to (i) ⇒ From operated block.

(d) Block(s) of the tape can be searched

In case of the block having sequence number

(i) Set the Mode select switch to EDIT mode.

(ii) Check the OFF position of the Optional Block Skip switch.

(iii) Set the POS/COM switch as follows.
MAIN, COM : Main tape
SUB, POS : Sub tape

(iv) Set the address using ADDRESS key to N position.

(v) Key in the sequence number of the block using DATA key. If the sequence number is 055, key in 055.

(vi) Set the NUMBER key to 00.

(vii) Push the INPUT button. Then the search is excuted, and the sequence number is displayed.

(viii) And at this time you want to examine the data of this block, by setting the address to your desired position, its data is desplayed on DPL panel.

In case of the block without sequence number

(i) Set the Mode select switch to EDIT mode.

(ii) Check the OFF position of the Optional Block Skip Switch.

(iii) Set the POS/COM switch as follows.

MAIN, COM : Main tape
SUB, POS : Sub tape

(iv) Set the address using ADDRESS key to N position.

(v) Key in the sequence number of the nearest block from the block to the searched, using DATA key.

(vi) Key in the number counted from the nearest block with sequence number using NUMBER key.

(vii) Push the INPUT button. Then the search is excuted, and the desplay becomes blank.

Example :

```
N120 X 123 X 123 *  ┐ 1
     X456*          ┘
     Z 789          ┐ 2
     S 53 *         ┘ 3
```

When the S53 is searched, 120 is keyed in using DATA key and 3 is set to the NUMBER key.

(viii) At this time, by selecting ADDRESS key to the appropriate address, the data in the address is displayed if the address exists in the block.

(e) The deletion of the block.

In order to delete the block after the block is searched, confirming the (i) to (iii) operation, operate the (iv) to (vi).

(i) Set the Mode select switch to EDIT position.

(ii) Check the OFF position of the Optional Block Skip switch.

(iii) Set the POS/COM switch as follows.

MAIN, COM : Main tape
SUB, POS : Sub tape

(iv) Set the address using ADDRESS key to N position.

(v) Push C "Cancel" button on the DATA key.

(vi) Push the INPUT button. Then the block is deleted.

(f) The deletion of the address word.

In order to delete the address word after the block is searched, confirming the (i) to (iii) operation, operate the (iv) to (vi).

(i) Set the Mode select switch to EDIT position.

(ii) Check the OFF position of the Optional Block Skip switch.

(iii) Set the POS/COM switch as follows.

MAIN, COM : Main tape
SUB, POS : Sub tape

(iv) Set the address using ADDRESS key to desired address position.

(v) Push the C "Cancel" button on the DATA key.

(vi) Push the INPUT button. Then the address word is deleted.

(g) The correction of the address word

In order to correct the address word after the block including this word is searched, confirming the next (i) to (iii), operate (iv) to (vi).

(i) Set the Mode select switch to EDIT mode.

(ii) Check the OFF position of the Optional Block Skip switch.

(iii) Set the POS/COM switch as follows.

MAIN, COM : Main tape
SUB, POS : Sub tape

(iv) Set the address using ADDRESS key to specified address position.

(v) Key in the correct data using DATA key.

(vi) Push the INPUT button. Then the address word is corrected.

(h) The insertion of the address word.

In order to insert the address word after the block including this word is searched, confirming next (i) to (iii), operate (iv) to (vi).

(i) Set the Mode select switch to EDIT mode.

(ii) Check the OFF position of the Optional Block Skip switch.

(iii) Set the POS/COM switch as follows.

MAIN, COM : Main tape
SUB, POS : Sub tape

(iv) Set the address using ADDRESS key to specified address position.

(v) Key in the insert data using DATA key.

(vi) Push the INPUT button. Then the address word is inserted, in the block tail.

(Note) If you want to insert the address word in the middle point of the block, it is necessary to delete the address word after this point. And you must insert the deleted address word.

(i) Insertion of the block

In order to insert the block after the block which is just before this block is searched, confirming next (i) to (iii), operate (iv) to (x).

(i) Set the Mode select switch to EDIT mode.

(ii) Check the OFF position of the Optional Block Skip switch.

(iii) Set the POS/COM switch as follows.

MAIN, COM : Main tape
Sub, POS : Sub tape

(iv) Set the address using ADDRESS key to STR position.

(v) Key in the sequence number using DATA key, if necessary.

(vi) Push the INPUT button.

(vii) Set the address using ADDRESS key to specified address position.

(viii) Key in the new data using DATA key.

(ix) Push the INPUT button.

(x) By repeating operation (vii) to (ix), one block data can be inserted.

(j) The corrected tape can be operated.

(i) Set the Mode select switch to MEMORY position.

(ii) Push the START button.
Refer to 4.1.4 (8) (c) for start point of excution.

Tape Memory

The following sorts of tape memory can be selected by option. Confirm which tape is selected.

(a) Tape Memory A (Capacity; Nearly 13m/26m/40m of tape information)
Tape information stored in memory is clear in case of power off.

(b) Tape Memory B (Capacity: Nearly 13m/26m/40m of tape information)
Tape information stored in memory is held in case of power off.

(c) Tape Memory C (Capacity; Nearly 2.5m of tape information)
Tape information stored in memory is clear in case of power off.

Memory type	Character number	Tape length
Tape Memory A, B	5114 ch	Nearly 12.9 m
	10574 ch	" 26.8 m
	16034 ch	" 40.7 m
Tape Memory C	1000 ch	" 2.5 m

(9) Inch/Metric Conversion

The metric machine tool can be used with the inch program, and to the contrary the machine tool of inch system can be used with the metric program. This function can be performed by MDI setting.

Increment system	Least input increment		Least command increment
	(mm)	(inch)	
M1V	0.01 mm	0.001 inch	0.001 mm/p
M2V	0.001 mm	0.0001 inch	0.001 mm/p
M3V	0.001 mm	0.0001 inch	0.0005 mm/p
M4V	0.001 mm	0.0001 inch	0.0002 mm/p
I1V	0.01 mm	0.001 inch	0.0001 inch/p
I2V	0.001 mm	0.0001 inch	0.0001 inch/p
I3V	0.001 mm	0.0001 inch	0.00005 inch/p
I4V	0.001 mm	0.0001 inch	0.00002 inch/p

Increment system	Max. command value		Max. feedrate		Jog feedrate	
	mm	inch	mm	inch	mm	inch
	mm	in.			mm/min	ipm.
M1V	8388.60	209.715	F3600	F900	2-1600	0.2-160
M2V	8388.607	209.7151	F36000	F9000	2-1600	0.2-160
M3V	4194.303	104.8575	F18000	F4500	1-800	0.1-80
M4V	1677.721	41.9430	F7200	F1800	0.4-320	0.04-32
I1V	8388.60	838.860	F3600	F3600	2-1600	0.2-160
I2V	8388.607	838.8607	F36000	F36000	2-1600	0.2-160
I3V	4194.303	419.4303	F18000	F18000	1-800	0.1-80
I4V	1677.721	167.7721	F7200	F7200	0.4-320	0.04-32

4.2 Position Display Unit (Inside/Outside)

This unit integrates movement value from the control section and displays the position of the movable member (tool) of a machine tool. The movement value integrated are command pulses except movement value for various compensations (backlash compensation). and pitch error compensation).
Position Display Unit can be attached either inside or outside of NC machine.

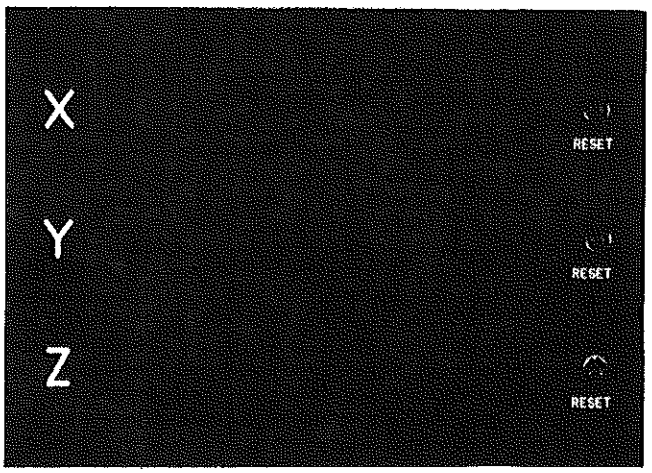


Fig. 4.2.1 Position Display Unit

There are the following switches on the front panel.

X, Y, Z .. If one of these push buttons is pushed, display for the appropriate axis is cleared.

Display

(1) When the power is turned on, the display is zero.

(2) The display is not affected by programming zero point.

(3) The display is updated each time approx. 100 msec.

Usage

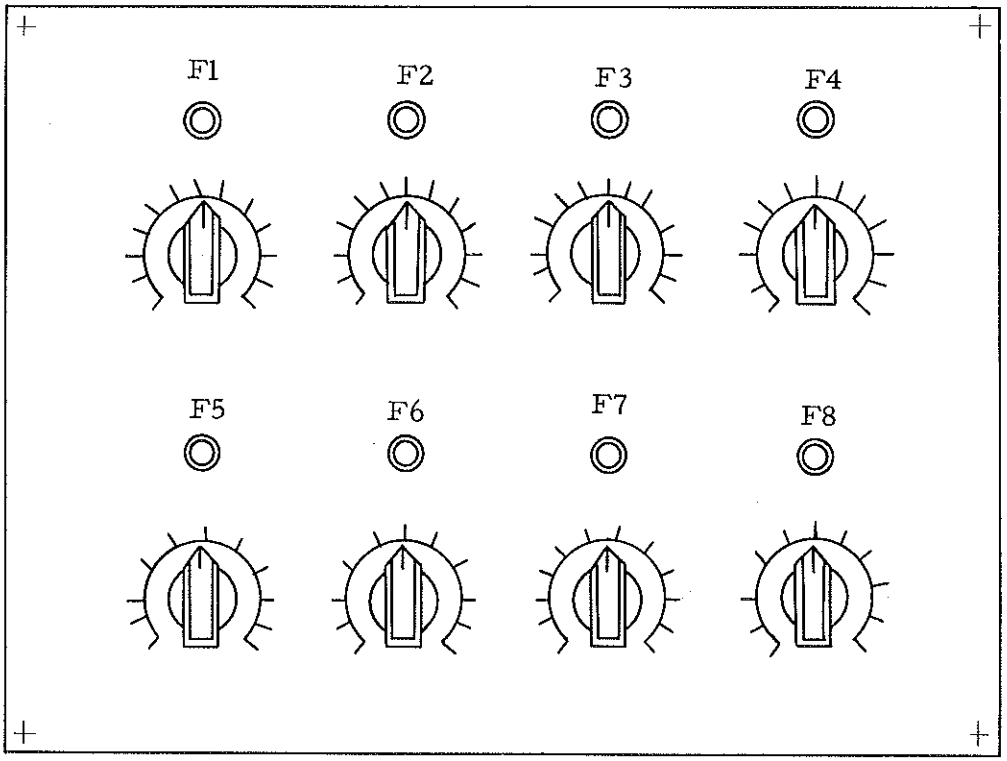
(1) Use of the DISPLAY LOCK/MACHINE LOCK switch (Refer to Section 4.3.14) permits only the display on the position display unit to be updated, or only the machine tool to be moved.

(2) Suppose inserting a manual operation during a tape operation. Record the display before entering into the manual operation, and when resuming the tape operation, it will be able to start with the position recorded.

(3) Positioning in any position is possible by use of Jog or Setp operation and the position display unit.

4.3 1-Digit F-Code Unit

These units are for putting in effect the 1-Digit F-Code in Section 3.11.



4.4 Built-in Type Manual Operator's Panel

The control panel usually provided on the machine side, can be mounted on the setting/display panel.
For more detail, refer to the MTB's manual.

4.5 Tape Reader

4.5.1 Tape Reader without Reels

If the front door of the tape reader section is opened, a tape reader shown in Fig. 4.5.1 is found. Data entry can be accomplished by setting an NC tape onto this tape reader.

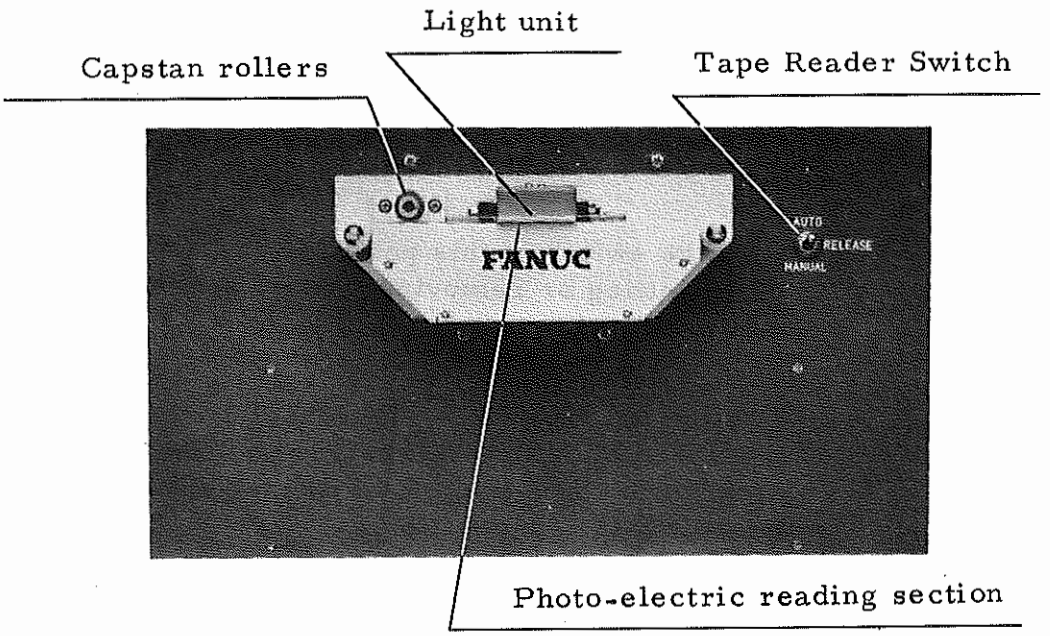


Fig. 4.5.1(a) Tape Reader without Reels

(1) Light source section

This section contains nine light emitting diode every channel. And this section contains stop shoe to stop the tape. Also this section is attached to photoelectric reading section by magnet for tape holding.

(2) Tape holder

This device acts to make an NC tape come into close contact with the aperture of the photoelectric reading section. If the tape holder is pushed upward lightly, it can be opened. If an NC tape is set with it opened and it is pushed downward lightly, the tape comes into close contact with the photo-electric reading section.

(3) Capstan roller

This device acts to feed a tape at a command from the control section.

(4) Photoelectric reading section

This section reads a tape, and has a glass aperture. Dust or flaws on this aperture will cause reading error.

(5) Tape reader switch

AUTO ... With the switch at this position, the tape can be started or stopped at a command from the control section. For tape operation, with the switch at this position, the stop shoe is put in effect to fix the tape.

RELEASE ... With the switch at this position, the tape is released. When setting or removing a tape, set the switch to this position.

MANUAL ... With the switch at this position, the tape is fed at a speed of 200 ch/sec. This position is selected to check the tape reader. For tape operation, don't set the switch to this position.

(6) Tape box

The tape box can accommodate approx. 30 m of NC tape. There is a polyested tape in this box, so NC tape inside this box can be easily drawn by drawing this polyestel.

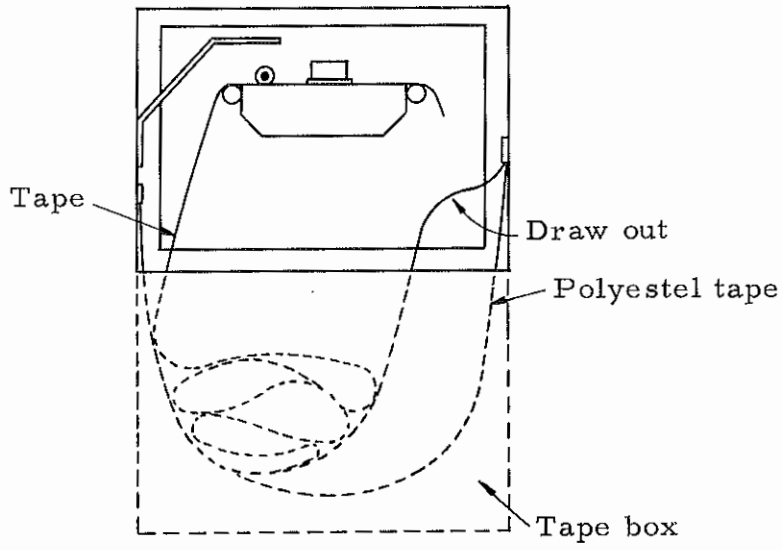


Fig. 4. 5. 1(b) Polyestel tape position

Note on operation

Do not set the switch to RELEASE or MANUAL while the tape is at stoppage, for purposes other than of feeding backward or replacing the tape for any due reason. Even while the tape is at stoppage, if the switch is set to the position other than AUTO, the tape may be put out of position, perhaps causing malfunctioning.

4.5.2 Tape Reader with Reels

This is a tape reader equipped with reels capable of accommodating 150 m of NC tape, and has an outer view as in Fig. 4.2.3.

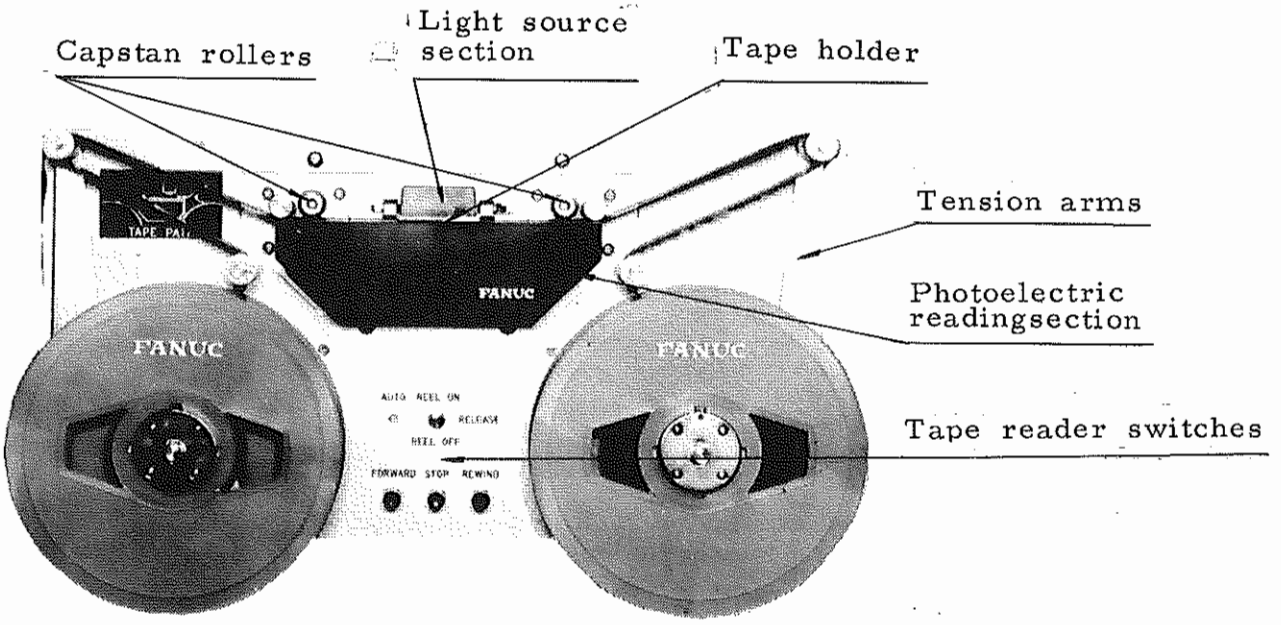


Fig. 4.2.3 Tape Reader with Reels

(1) Tape reading switch

Switches and a lamp are arranged as shown in Fig. 4.2.4.

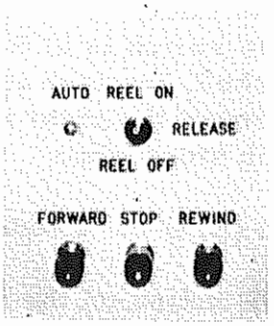


Fig. 4.2.4 Tape Reading Switches

a) AUTO lamp

This lamp is on during Cycle operation (tape operation, MDI operation, or the like).

While this lamp is on, the operational buttons (FORWARD, REWIND and STOP) on the tape reader are ignored.

b) REEL ON/OFF and RELEASE (positions on toggle switch)

REEL ON	When using the reels, set the switch to this position, after setting a tape.
REEL OFF	When using no reels, set the switch to this position, after setting a tape.
RELEASE	When loading or unloading a tape, set the switch to this position.

c) FORWARD (push button switch)

This switch is effective while the AUTO lamp is off, and the toggle switch is at REEL ON/OFF. If this switch is pushed, the tape is fed in the forward direction and wound around the left reel. The tape feed initiated by this switch stops when

* ER code (EIA) or % code (ISO) is read,

* the STOP button on the tape reader is pushed, or

* the NC equipment is reset by pushing the RESET button on the display panel, etc.

d) REWIND (push button switch)

This switch is effective while the AUTO lamp is off, and the toggle switch is at REEL ON/OFF. If this switch is pushed, the tape is fed in the reverse direction and wound around the right reel: The tape feed initiated by this switch stops when

* ER code (EIA) or % code (ISO) is read,

* the STOP button on the tape reader is pushed, or

* the NC equipment is reset by pushing the RESET button on the display panel, etc.

Furthermore, the tape does not stop just on the character, but a bit past it.

e) STOP (push button switch)

The tape feed, when initiated by the FORWARD/REWIND switch, can be stopped by pushing this switch.

(2) Light source

Refer to 4.5.1 (1)
Remove this cover before wiping duest on the lamp and lens.

(3) Tape holder

Refer to 4.5.1 (2)

(4) Capstan roller

This roller acts to feed an NC tape at a command from the control section, and comes for forward feed and reverse feed.

(5) Photoelectric reading section

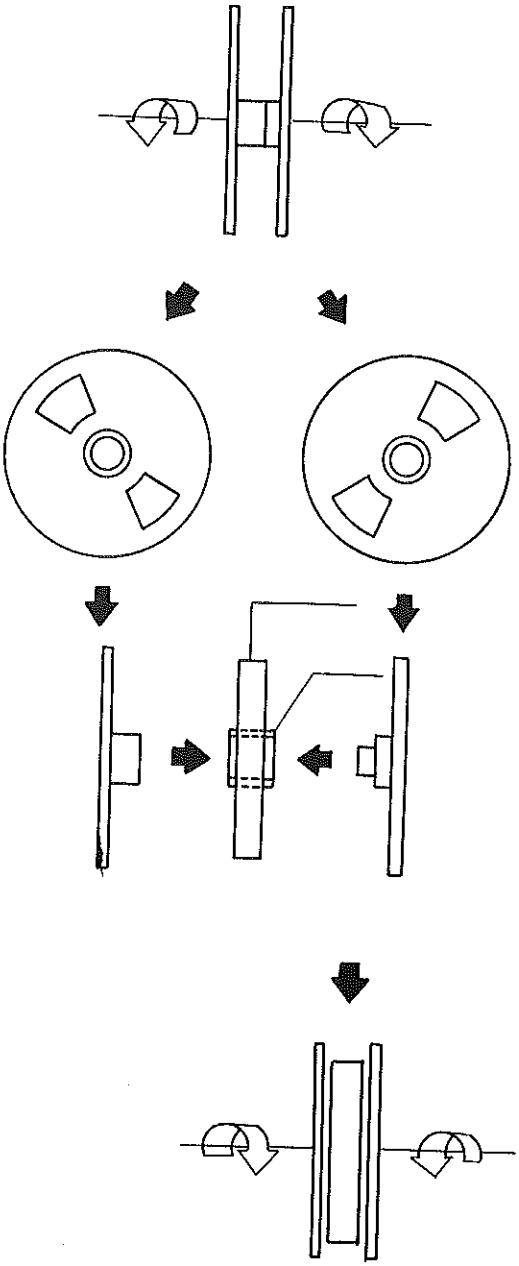
Refer to 4.5.1 (5)

(6) Tension arms

In order to quickly start or stop an NC tape, a buffer is required before it is wound around the reel. Thus, the tension arms are provided. By setting an NC tape through these tension arms, a delay in the start and stop time of the reel motor can be absorbed.

4.5.3 Procedure of Setting an NC Tape (for the Tape Reader with Reels)

Set a tape on the reel as follows:



(1) Twist the reel in the directions indicated by arrows to separate it.

(2) The reel is separated in the left figure.

(3) Set the tape on the reel

(4) Twist the reel in the directions indicated by arrows to fix the reel and the tape.

(5) Be sure that the protuberance of the reel boss is inside.

(6) If the protuberance is out, turn the lever of the reel boss in the opposite direction.

(7) Make one of the two grooves of the reel meet the position of the protuberance to set the reel on the reel boss.

(8) Turn down the lever of the reel boss in the opposite direction to fit the protuberance into the groove of the reel (See Fig. 4. 2. 5).

(9) Set the tape reader switch to RELEASE.

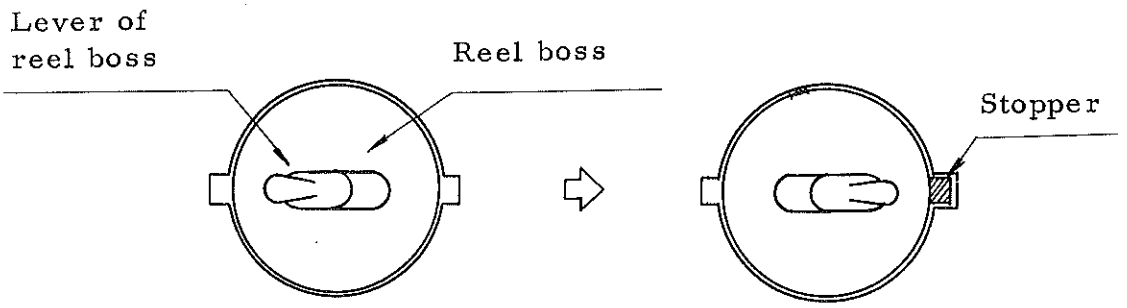


Fig. 4. 2. 5 Setting the Reel

(10) When using the reels, set the tape as shown in Fig. 4. 5. 3(a) And set the tape reader switch to REEL ON. Now the tape is ready to be read.

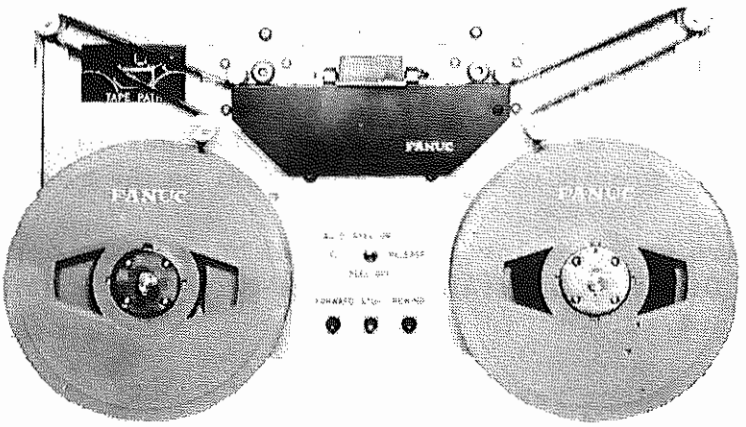


Fig. 4. 5. 3(a) Setting a Tape, When Using the Reels

(11) When using no reels, or when using an endless tape, set the tape as shown in Fig 4. 5. 3(b). Now, the tape is ready to be read.

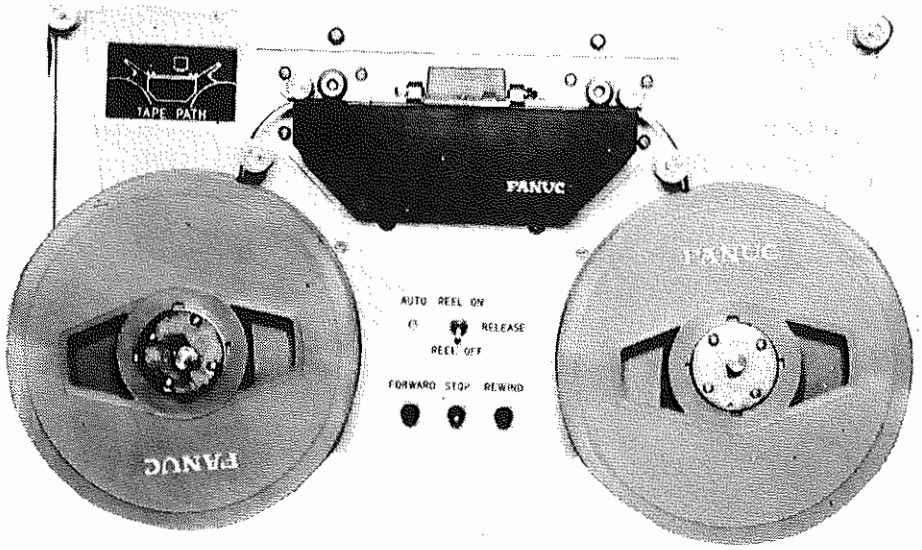


Fig. 4. 5. 3(b) Setting a Tape, When Using No Reels

4.6 Procedure of Punch Function

The NC command in memory is punched out by tape puncher and NC tape can be made.

(1) Check the punch unit ready. (Power, punched tape and etc.)

(2) Set the Mode select switch to EDIT position.

(3) Set the address using ADDRESS key to STR position.

(4) Set the POS, SUB/COM, MAIN switch to as follows.

Main tape punch out : COM, MAIN
Sub tape punch out : POS, SUB

(5) Set the NUMBER key to 99.

(6) Push the START button.

In this time, punch out operation is started in ISO code or EIA code by setting ISO or EIA respectively.

For punch unit operation, refer to the attached manual on puncher. (Instruction book/tape punch FACIT 4070)

4.7 Operator's Panel

The control panel varies in function and arrangement of the switches with the NC machine tool. Fig. 4.6.2 illustrates a typical control panel for explanation of its use. Refer to the description issued by the machine tool builder for details.

4.7.1 MODE SELECT Switch

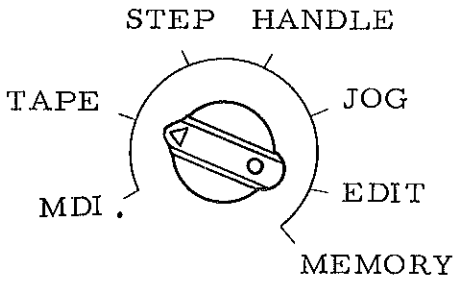


Fig. 4.6.1 Example of MODE SELECT Switch

This switch is used to specify an operational mode.

TAPE : Set the switch to this position to perform an operation by an NC tape.

STEP : Push one of the JOG & STEP buttons (+X, -X, +Y, -Y, +Z, ...) with the switch at this position, and the machine tool will be able to be moved step by step in the corresponding direction.

HANDLE: The equipment with a manual pulse generator has this position. Set the switch to this position for use of the manual pulse generator.

JOG : Push one of the JOG buttons with the switch at this position, and while the JOG & STEP button is kept depressed, the machine tool will be able to be moved continuously in the corresponding direction. The feed rate is set on the JOG FEED RATE switch (see Section 4.7.9).

MDI : Set the switch to this position to perform Manual Data Input.

EDIT : Set the switch to this position to use tape editing function.
Refer to Section 4.1.4(4) for the tape editing function.

MEMORY : When executing NC commands stored in core memory, set the switch to this position. Refer to Section 4.1.4(7) Tape Edit Function for execution of NC commands stored in memory.

(Note 1) When NC machine becomes in the state of automatic holding state caused by feed hold or mode change etc. during cycle operation, machine tool cannot activate in the different mode.

(Note 2) When mode select switch are changed to STEP, HANDLE or JOG during cycle operation, NC machine becomes in the state of automatic stopping state immediately. (In case of drilling of G84, after finishing drilling it become in this state)

And when mode select switch are changed to TAPE, MDI, MEMORY, or EDIT, NC machine becomes in the state of automatic stopping state. (Refer to appendix 8)

4.7.2 CYCLE START Button

Push and release this button in Cycle Operation mode (Tape, MDI or Memory) to activate the machine tool.

This button is effective when

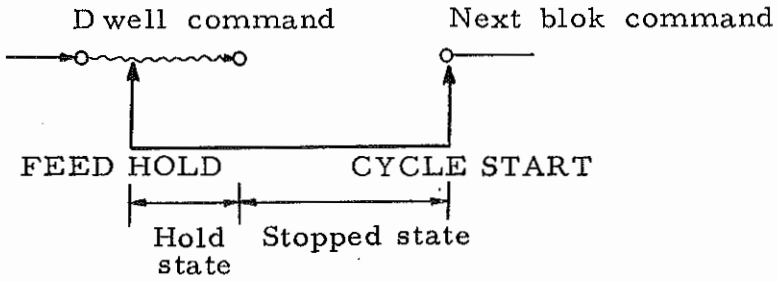
- in a reset state (caused by the RESET button on the display panel or just after turning on power, etc)
- in a hold state (caused by the FEED HOLD button, etc. during execution of a block), or
- in a stopped state (state after completion of execution of a block by Single Block).

If this button is pushed, the display lamp lights up, and an appropriate operation is executed.

4.7.3 FEED HOLD Button , FEED HOLD Lamp

If this button is pushed during a Cycle Operation activated by the CYCLE START button.

- o the feed is stopped after decleration if the machine tool is moving,
- o Dwell is continued even in a feed hold state if the Dwell is being executed, or



- o a stopped state is created after completion of the M, S, T or B function, when being executing.

If this button is pushed, the display lamp (FEED HOLD) lights up and the display lamp (CYCLE STARY) out. Restart can be made by the CYCLE START button.

4.7.4 JOG & STEP Button

If this button is pushed in a Jog or Step mode, the machine tool can be moved in a apprpriate direction.

For safty, this function does not become efficient if reset state is released at this button on.

This function becomes efficient when this button is set on after this button setting off.

Feed rate can be changed by JOG FEED RATE dial.

Jog mode

A move is made continuously while the button is depressed.

The feed rate is increased or decreased (in twenty-two stages) by use of the JOG FEED RATE dial. With the RAPID TRAVERSE switch at ON, the Rapid Traverse speed is created.

Step mode

A move is made by one step each time the button is pushed.

The movement amount per step is as shown in the following table, depending on the condition of the MANUAL PULSE MULTIPLY switch and the increment system.

Increment system	When multiplied by 1	When multiplied by 10	When multiplied by 100	When multiplied by 1000
M1V	0.01 mm	0.1 mm	1.0 mm	10 mm
M2V～M4V	0.001 mm	0.01 mm	0.1 mm	1.0mm
I1V	0.001 inch	0.01 inch	0.1 inch	1.0 inch
I2V～I4V	0.0001 inch	0.001 inch	0.01 inch	0.1 inch
D1V	0.01 deg	0.1 deg	1.0 deg	10 deg
D2V～D4V	0.001 deg	0.01 deg	0.1 deg	1.0 deg

4.7.5 RAPID TRAVERSE Switch

If this switch is set to ON in Jog or STEP mode, a move is made in Rapid Traverse.

4.7.6 DRY RUN Switch

If this switch is set to ON in the Cycle Operation of Tape, Memory or MDI, an F function specified on NC tape is ignored, and a move is made (This function is effective for each block)

at a cutting feed command

at a speed of a setting on the JOG FEED RATE dial with the RAPID TRAVERSE switch at OFF, and at the maximum feed rate of Jog feed with the RAPID TRAVERSE switch at ON,

at a Rapid Traverse Command

(In case that DRY RUN is effective,)
at a speed of a setting on the JOG FEED RATE dial with the RAPID TRAVERSE switch at OFF, and at a Rapid Traverse speed with the RAPID TRAVERSE switch at ON, and

at a Rapid Traverse command (Dry Run B)

(In case that DRY RUN is ineffective,) at a Rapid Traverse at all times.
(Application example)

This switch can be used to conduct efficient tape checking. That is, with a command of lowspeed feed rate in a cutting area specified on NC tape as a setting on the above switch, a move can be made at the appropriate high speed. Since the F code is read in also during Dry Run operation, the display can be made on the display unit.

Note : This function must be used with the work removed.
At this time the lamp lights up for warning.

4.7.7 OPTIONAL BLOCK SKIP Switch

With this switch set to ON or OFF, commands of a block containing "/" (slash) specified on NC tape can be made effective or not, respectively.
When the operation in Edit mode is excuted,
this switch must be turned to OFF potion

4.7.8 F4 FEED RATE OVERRIDE (4-digit F-code feed rate Over ride) Dial

An Override can be applied in increments of 10% in a range of 0 to 200% to the feed rate specified by the F function during a Cycle Operation (see Section 3.11.3). With the dial at 100%, the feed rate is the same as specified. In Fixed Cycle tapping (G84), however, the Override is ignored.

4.7.9 JOG FEED RATE Dial

This dial can be used to set a feed rate in twentytwo stages in a Jog or Dry Run operation. Correspondence between the actual feed rates and the scales depends on the machine tool builder.

Position of Rotary switch	Increment System					
	M1V, M2V (mm/min)	M3V (mm/min)	M4V (mm/min)	11V, 12V (inch/min)	13V (inch/min)	14V (inch/min)
1	2	1	0.4	0.2	0.1	0.04
2	4	2	0.8	0.4	0.2	0.08
3	6	3	1.2	0.6	0.3	0.12
4	8	4	1.6	0.8	0.4	0.16
5	10	5	2.0	1.0	0.5	0.20
6	12	6	2.4	1.2	0.6	0.24
7	14	7	2.8	1.4	0.7	0.28
8	16	8	3.2	1.6	0.8	0.32
9	20	10	4.0	2.0	1.0	0.40
10	40	20	8.0	4.0	2.0	0.80
11	60	30	12.0	6.0	3.0	1.20
12	80	40	16.0	8.0	4.0	1.60
13	100	50	20.0	10.0	5.0	2.00
14	120	60	24.0	12.0	6.0	2.40
15	140	70	28.0	14.0	7.0	2.80
16	160	80	32.0	16.0	8.0	3.20
17	200	100	40.0	20.0	10.0	4.00
18	400	200	80.0	40.0	20.0	8.00
19	600	300	120.0	60.0	30.0	12.00
20	800	400	160.0	80.0	40.0	16.00
21	1000	500	200.0	100.0	50.0	20.00
22	1200	600	240.0	120.0	60.0	24.00
23	1400	700	280.0	140.0	70.0	28.00
24	1600	800	320.0	160.0	80.0	32.00

4.7.10 EMERGENCY STOP Button

This button is pushed to make a stop at emergency time. If this button is pushed, the feed is stopped, and a reset state is created.

Reset state

This refers to a state in which all NC tape information except Sequence Numbers executed earlier is canceled. (Refer to Appendix 8.)

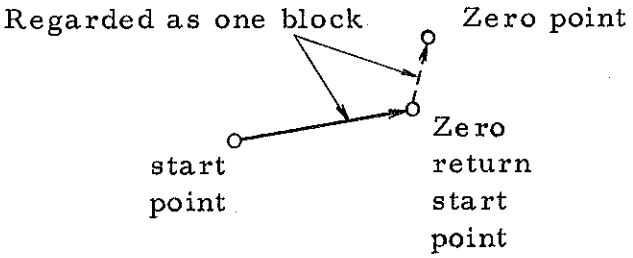
4.7.11 SINGLE BLOCK Switch

If a start is effected by the CYCLE START button with this switch at ON, a stopped state is created after the commands of one block are executed. At this time, the commands of the next block are not contained in the buffer register.

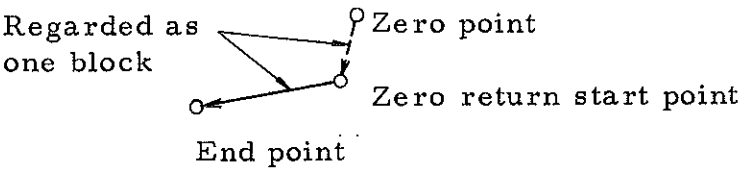
If this switch is switched to ON during execution of NC tape commands, after completion of the block currently being executed, a stopped state is entered. At this time the commands of the next block remain in the buffer register. If a start is effected by the CYCLE START button in this state, the commands in the buffer register are executed, and after completion of execution, a stopped state is entered.

(a) In a Fixed Cycle operation, positioning and drilling are each assumed as one block for execution. Therefore, in operation with the switch at ON, machine tool becomes cycle operation hold state after completion of the positioning. If a restart is effected (by pushing the CYCLE START button), machine tool becomes cycle operation stop state after completion of the drilling.

(b) At the time of ATC zero return, positioning to Zero Return start point and zero return are regarded as a single block respectively, during Single Block state.



(c) At the time of ATC return to Zero Return start point, return to zero return start point and zero return are regarded as a single block respectively, during Single Block.



(Application example)

It is convenient to use this switch in checking an NC tape.

While making the NC equipment read the NC tape with the SINGLE BLOCK switch at ON, observe the movement on the position display unit, etc. to check if the movement is made as programmed. At this time, do not move the movable member of the machine tool with the switch described in Section 4.6.14 at MACHINE LOCK.

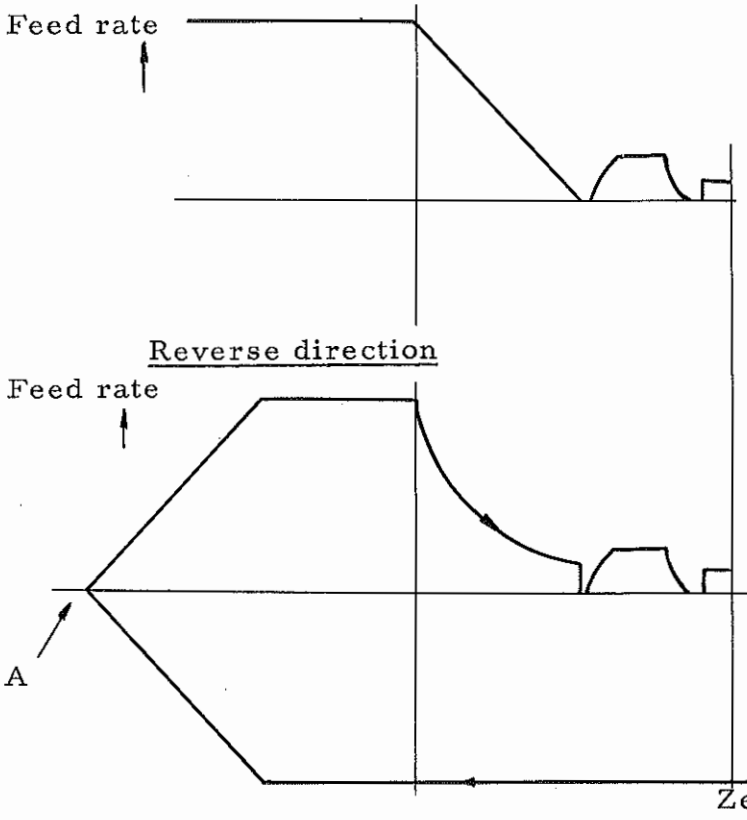
4.7.12 ZERO RETURN Switch and ZERO POSITION Lamps

With this switch at ON, and with the MODE SELECT switch at JOG, each axis, if moved in the zero point direction by the JOG button, moves at rapid traverse and stops at the zero point, and the ZERO POSITION lamp lights up.
When moving each axis in the zero point direction, move the machine tool toward the zero point from a point distant to some extent.
"A point distant to some extent" means the place where the deceleration is possible. The Zero Return Operation is possible from both direction.

For the beginning point of deceleration, refer to the manual issued by machine tool builder.

Zero positioned lamp also lights up when the machine tool come on the lattice point of resolver or inductosyn, scale during SCALE RETURN.

Normal direction



Once the zero point has been returned you must set the ZERO RETURN switch to OFF, or the machine tool will not be able to be moved again by manual operation.

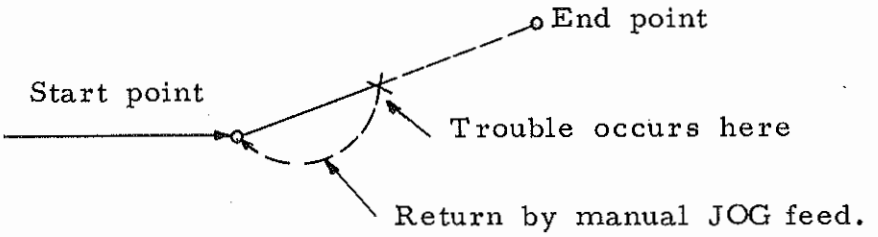
In the case of ATC Zero Return (G28), it is not necessary to operate this ZERO RETURN switch.

4.7.13 BLOCK RETURN Switch

This switch is used to perform the Mid-Tape Start-Up function.

Mid-Tape Start-Up function

When a trouble such as tool damage occurs during Cycle Operation, for example, this function permits the movable member of a machine tool to be returned to the start point of operation of the block currently being executed for re-machining.



Operational procedure

(1) When a trouble occurs, stop the movable member of the machine tool by any of the following ways:

Push the RESET button on the display panel.

Push the FEED HOLD button on the control panel.

Enter a signal into the External Reset (ERS) terminal of the NC equipment.
In this case, check the BUFFER lamp ON/OFF

(2) With the MODE SELECT switch at JOG, let the tool go from the work to the safe side by Jog operation.
In this case, MANUAL ABSOLUTE switch set to ON.

(3) Dispose of the trouble.

For tool change, correct the setting in the Tool Offset.

For program checking, correct the program, and set the FEED RATE OVERRIDE dial.

(4) Reset the NC equipment.

Push the RESET button on the display panel.

Enter a signal into the External Reset (ERS) terminal of the NC equipment.

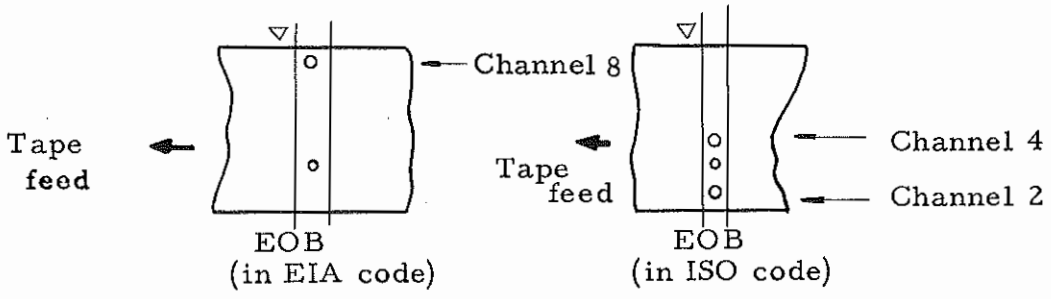
(5) Set the BLOCK RETURN switch to ON.

(6) Move the movable member of the machine tool on each axis in the direction of the start point of the block by Jog operation. At this time, if the start point is in the cutting area, when a tool aproaches a start point, lower the Jog feed rate for safety.

(7) When the tool reaches the start point, it stopps automatically. And BLOCK RETURN lamps light on.

(8) Return the NC tape on the tape reader, (up to before the EOB code).

If BUFFER lamp is ON ... Two block return
If BUFFER lamp is OFF ...One block return



(9) Check the all BLOCK RETURN lamps light on.

(10) Set the MODE SELECT switch toTAPE again.

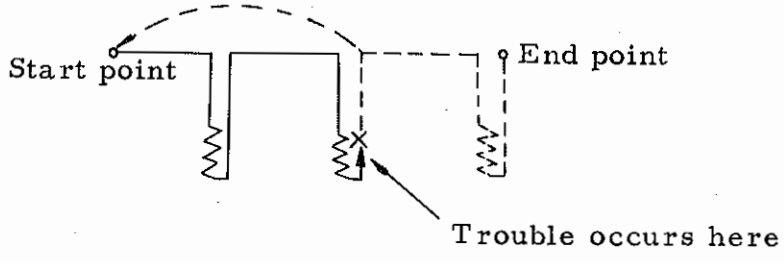
(11) Check the BLOCK RETURN switch to ON.

(12) Restart the operation by the CYCLE START button.

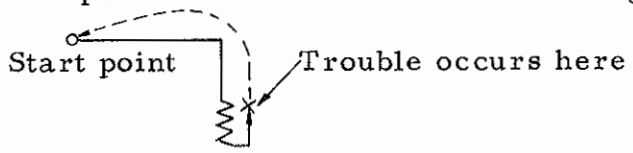
(13) Change the BLOCK RETURN switch to OFF.

Note

(a) When this function is performed during Fixed Cycle operation (having L command), the start point is the initial point of this block shown in the figure below.

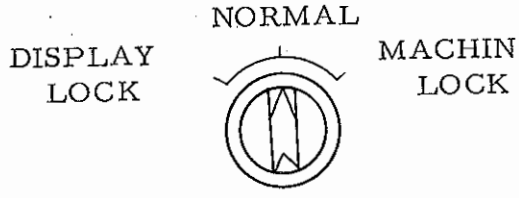


(2) When this function is performed during Fixed Cycle operation (not having L command), the start point is the initial point of this block shown in this figner below.



4.7.14 DISPLAY LOCK/MACHINE LOCK Switch

This switch functions to stop pulses to the position display unit, and to suppress move command pulses to the machine side.



With the switch at NORMAL, the machine tool and the position display unit operate according to the command by Cycle Operation or manual operation.

With the switch at DISPLAY LOCK, pulses are stopped to the position display unit. Consequently, by Cycle Operation or manual operation, the machine tool operates, but the display is not updated.

Typical application

When shifting a coordinate system by manual operation, use this switch not to enter this shift amount into the display.

With the switch at MACHINE LOCK, move command pulses are suppressed to the machine side. Consequently, by Cycle Operation or manual operation, the position display is updated as specified, but the machine tool does not move. The M, S, T and B functions, however, are executed.

Typical applications

* For presetting an appropriate value in the position display unit

* For checking an NC tape (in which case with the SINGLE BLOCK switch at ON, tape check is conducted while seeing the coordinate value)

Note: This function becomes efficient even at the midpoint of the block.

4.7.15 Z-AXIS FEED NEGLECT (Z-Command Cancel) Switch

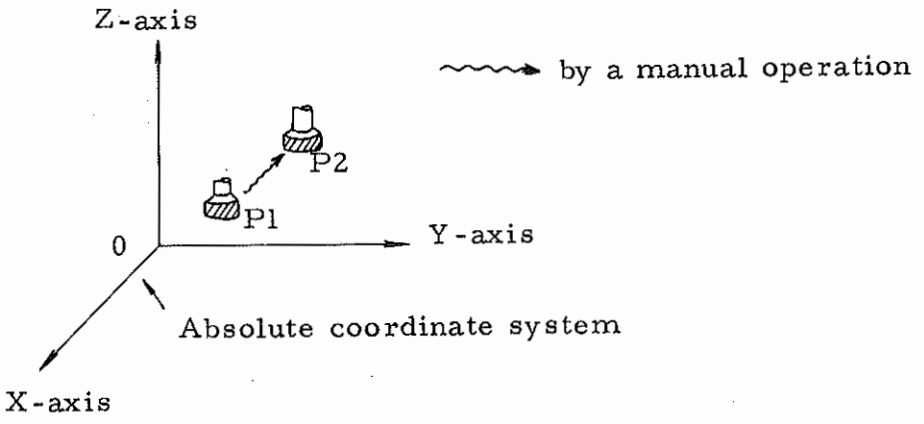
If this switch is set to ON, a Z axis command can be ignored.

This function is effective when checking the contents of an NC tape command.

4.7.16 MANUAL ABSOLUTE Switch

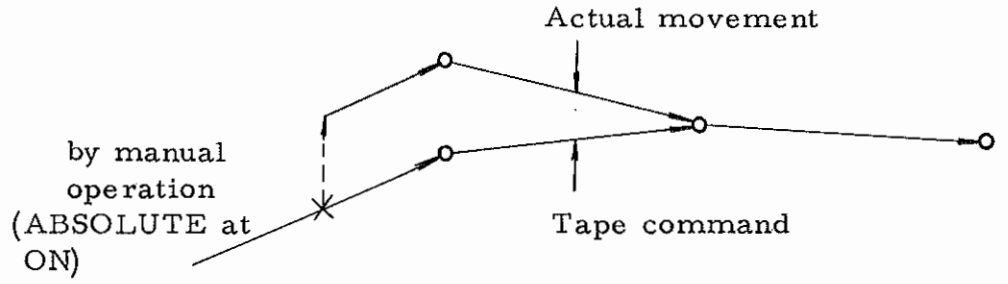
By setting this switch to ON or OFF, adds a movement amount by a manual operation to an Absolute coordinate value or ignores, respectively.

at On

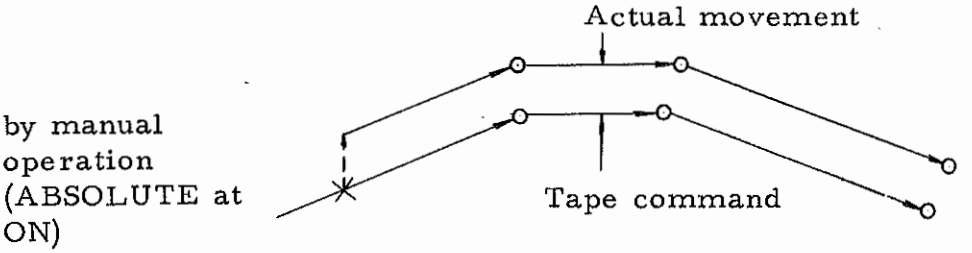


When a tool is moved manually, the Absolute coordinate system does not shift. That is, the Absolute coordinate value of this tool changes by the movement amount by the manual operation.

By a manual operation in a hold state of Cycle Operation (by use of the FEED HOLD, etc.) movement for the rest of the block is in parallel aparting from the manual movement amount.

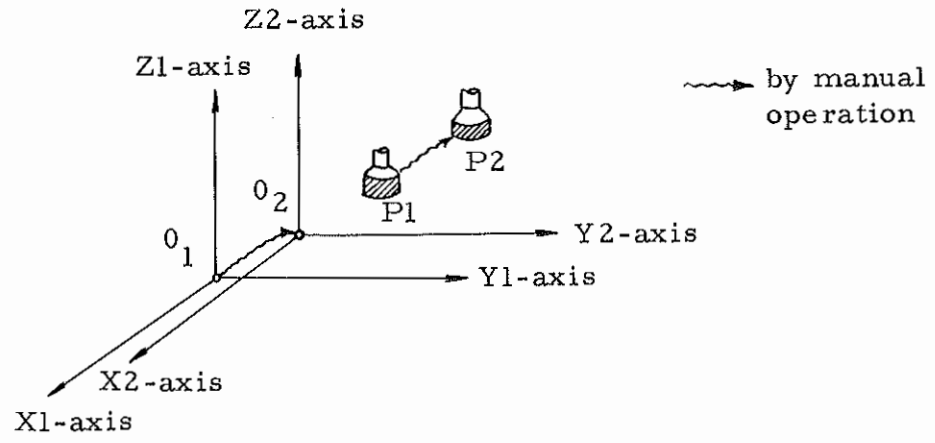


In case of the next block being absolute.

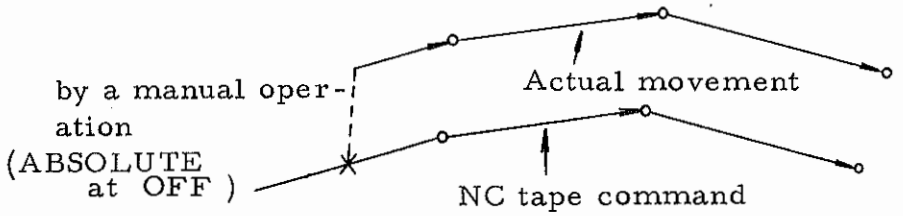


In case of the next block being incremental.

at OFF



When a tool is moved manually, the Absolute coordinate system shifts by the movement amount. That is, the absolute coordinate value does not change.



The Block Return function must be done at the state of MANUAL ABSOLUTE being ON.

The function of MANUAL ABSOLUTE ON/OFF becomes efficient at the time of setting RESET after manual operation or at the time of setting the CYCLE START. Therefore absolute coordinate value changes by manual operation even if the MANUAL ABSOLUTE switch is ON. And at the time of Reset or CYCLE START the absolute coordinate changes immediately.

4.7.17 SCALE RETURN Switch

This is a function to position at a lattice point on the inductosyn scale or resolver.
With this switch at ON, and the MODE SELECT switch at JOG, positioning is made at the lattice point nearest to a direction specified by that button from the present position. When the positioning is completed, ZERO POSITIONED lamp lights up. Once the positioning has been completed, set the SCALE RETURN switch to OFF, or machine tool will not be able to be moved again by manual operation.

By setting the SCALE RETURN switch to OFF, ZERO POSITION lamp is out.

4.7.18 AUXILIARY FUNCTION LOCK Switch

With this switch at ON, the B, S, T and M function commands are ignored. The M00, M01, M02 and M30, however, are normally processed. This switch, after set to ON, is effective for the B, S, T and M function commands, and the command that have already been commanded to the machine tool, are executed.

Application example

This function is used with the MACHINE LOCK function to check the path of an NC tape. With only the Machine Lock function made effective, the B, S, T and M functions are executed. When these functions are not desired, the AUXILIARY FUNCTION LOCK switch must be made effective simultaneously.

4.7.19 HANDLE (Manual Pulse Generator) and AXIS SELECT Switch

By turning the handle of the manual pulse generator with the MODE SELECT switch at HANDLE, the sensitive feed can be performed. One hundred pulses are generated at a full one turn of the handle, and the machine tool moves the distance for 100 pulses correspondingly to the rotary direction. The movement amount per pulse is the same as in the case of the Step feed, as in the following table.

The HANDLE AXIS SELECT switch is used to select an axis on which the machine tool is to be moved. Furthermore, the movement amount per pulse can be multiplied by 10 or 100 by use of the MANUAL PULSE MULTIPLY switch.

Increment system	When multiplied by 1	When multiplied by 10	When multiplied by 100	When multiplied by 1000
M1V	0.01mm	0.1mm	0.1mm	0.1mm
M2V ~ M4V	0.001mm	0.01mm	0.1mm	0.1mm
I1V	0.001inch	0.01inch	0.01inch	0.01inch
I2V ~ I4V	0.0001inch	0.001inch	0.01inch	0.01inch
D1V	0.01deg	0.1deg	0.1deg	0.1deg
D2V ~ D4V	0.001deg	0.01deg	0.1deg	0.1deg

Application example

Use of these functions permits easily and effeciently performing positioning at a comparatively short distance. This is very useful for setting at the tool change for wear or damage, manual setting to the zero point, instructions about a depth of cut at the trial cutting, etc.

4.7.20 MANUAL PULSE MULTIPLY Switch

With the MODE SELECT switch at STEP or HANDLE, this switch is used to multiply a movement amount per pulse by 10, 100 or 1000. 1000 multiply is only effective for STEP operation.

4.7.21 SEARCH START Button

This button is used to search for a Sequence Number from machine side.

4.7.22 OVERRIDE CANCEL Switch

With this switch at ON, the function of the OVERRIDE dial in Section 4.6.8 is ignored (assumed to be of 100%), and the command value by the F function remains effective.

4.7.23 POWER ON/OFF Button

These buttons function the same as the POWER ON/OFF button on the display panel, and are used to power on and off on the control panel.

4.7.24 2ND L.S. REMOVE Switch

When the movable member of a machine tool reaches the stroke end, and the second limit switch operates, with this switch at ON, it is moved in the safe direction by manual operation for release.

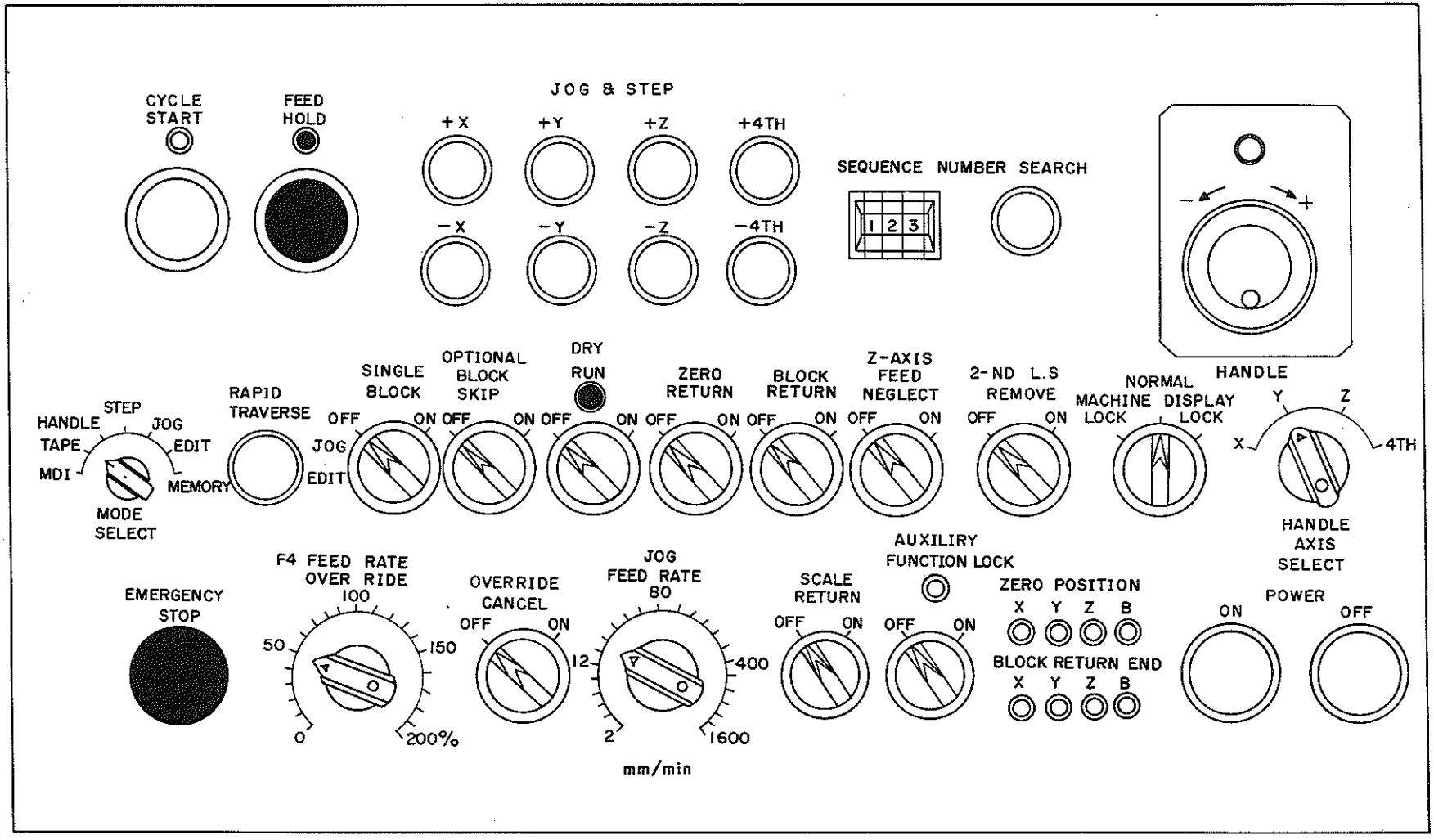


Fig.4.7.2 Operator's Control Panel (Example)

5. OPERATIONAL PROCEDURE

5.1 Preparation and Inspection before Turning on Power

(1) Make sure that the front and back doors of the equipment are closed.

Note: This NC equipment is of sealed construction not to intrude outdoor flaw directly into it. Consequently, when turning on power, do not keep the doors, etc., including the door of the tape reader, open for a long time. Since there are portions partially applied with high voltage within the servo unit in particular, the power is designed so as to be turned off if the door (front and back) at the servo unit side is opened.

5.2 Turning on Power

(1) After making sure the main power is on, push the POWER ON button on the display panel (control panel).

(2) Make sure that the tape reader lamp and the READY lamp and the motor fan are on.

5.3 Setting a Tape

5.3.1 For the Tape Reader without Reels

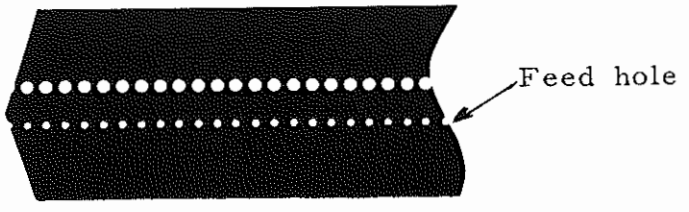
(1) Open the front door of the tape reader section.

(2) Set the tape reader switch to the neutral position.

(3) Wipe dust on the tape with a soft cloth.

(4) Push upward the tape holder lightly and insert the tape thereunder.

(5) Insert the tape as follows as viewed from above.



← Tape forward direction (in EIA code)

(6) Put the tape so that its head is to the left of the capstan roller.

(7) Make sure that the tape is exactly in the tape guide.

(8) Hold the tape with the tape holder.

(9) Set the tape reader switch to Auto.

(10) Close the front door of the tape reader section.

Furthermore, when unloading the tape, set the tape reader switch to OFF.

5.3.2 When Using the Reels with the Tape Reader with Reels

(1) Open the front door of the tape reader section.

(2) Make sure that the tape reader switch is at RELEASE.

(3) Wipe dust on the tape with a soft cloth.

(4) Set the tape according to Section 4.5.3.

(5) Make sure that the tape is exactly in the tape guide.

(6) Hold the tape with the tape holder.

(7) Set the tape reader switch to REEL ON.

(8) Close the front door.

When unloading the tape, set the tape reader switch to RELEASE.

5.3.3 When Using No Reels with the Tape Reader with Reels

(1) Open the front door of the tape reader section.

(2) Make sure that the tape reader switch is at RELEASE.

(3) Wipe dust on the tape with a soft cloth.

(4) Push upward the tape holder lightly, and insert the tape thereunder as in Fig. 4.5.7.

(5) Make sure that the tape is exactly in the tape guide.

(6) Hold the tape with the tape holder.

(7) Set the tape reader switch to REEL OFF.

(8) Close the front door.

When unloading the tape, set the tape reader switch to RELEASE.

5.4 Manual Operation

Rapid (rapid traverse) operation

(1) Set the MANUAL ABSOLUTE Switch to ON or OFF according to Section 4.7.16.

(2) Set the MODE SELECT switch to JOG, and set the RAPID TRAVERSE switch to ON.

(3) Push the JOG & STEP button (see Section 4.7.4).

Note: Three axes are not controlled at a time.

Jog (Jog feed) operation

(1) Set the MANUAL ABSOLUTE Switch to ON or OFF according to Section 4.7.16.

(2) Set the MODE SELECT switch to JOG.

(3) Set the feed rate on the JOG FEED RATE dial (see Section 4.7.9).

(4) Push the JOG & STEP button (see Section 4.7.4).

Note: Three axes are not controlled at a time.

Step (Step feed) operation

(1) Set the MANUAL ABSOLUTE Switch to ON or OFF according to Section 4.7.16.

(2) Set the MODE SELECT switch to STEP.

(3) Set the MANUAL PULSE MULTIPLY switch to any of X1, X10, X100 and X1000.

(4) Push the JOG & STEP button (see Section 4.7.4).

Note: Three axes are not controlled at a time.

Handle (manual pulse generator) operation

(1) Set the MANUAL ABSOLUTE Switch to ON or OFF according to Section 4.7.16.

(2) Set the MODE SELECT switch to HANDLE.

(3) Select the axis by the HANDLE AXIS SELECT switch.

(4) Set the MANUAL PULSE MALTIPLY Switch to any of X1 and X10.

(5) Turn the handle of the manual pulse generator (see Section 4.7.19).

5.5 Setting the Zero Point

In order to perform an NC tape operation after turning on power, return the movable member of a machine tool to the zero point at the beginning. In this case, there are the following two methods of Zero Return available:

5.5.1 Automatic Zero Return

Program the Zero Return at the beginning of the NC tape previously or use the MDI unit, for automatic Zero Return.

(1) Positioning to a position where Zero Return is possible

(2) ATC Zero Return command (G28) See Section 3.12.9

5.5.2 Manual Zero Return

Perform Zero Return by manual operation by use of the switch on the control panel (see Section 4.6.1).

5.6 NC Tape Operation / Memory Call Operation

(1) Make sure that the alarm display lamps (TH, TV, ..., P/S) on the display panel are not on. If they are on, release the alarm according to Section 4.1.3.

(2) Check and correct, if required, the data in the Tool Offset switch.

(3) Set the switches on the control panel as follows:

MODE SELECT to TAPE/MEMORY (For memory operation)
DRY RUN to OFF or ON

F4 FEED RATE OVERRIDE (100% when not specified)
OPTIONAL BLOCK SKIP to OFF or ON
SINGLE BLOCK to OFF or ON
DISPLAY LOCK/MACHINE LOCK to NORMAL or others
AUXILIARY FUNCTION LOCK to OFF or ON
(___ indicates the position where the switch is usually set.)

(4) Set the mirror image by MDI, if necessary.

(5) Set the NC tape onto the tape reader. (Not necessary in Memory operation)

(6) Push the RESET button on the display panel.

(7) Push the ORIGIN button on the display panel to make the current position the Absolute Zero point.

(8) Push the CYCLE START button on the control panel for NC tape operation.

(9) In order to effect a temporary stop during the operation, push the FEED HOLD button on the control panel. Also, when an unexpected event occurs, push the EMERGENCY STOP button to make a stop.

(10) During the operation using the tape reader with reels, a rewind is effected automatically by the Miscellaneous function command (M02, M03) after completion of the machining, and the tape is reversed until the Absolute Rewind Stop code (ER in EIA code, % in ISO code) is read in.

5.7 Manual Operation during NC Tape Operation

(1) Push the FEED HOLD button on the control panel, or set the SINGLE BLOCK switch to ON, to make a temporary stop.

(2) Record the coordinate value of the stop position from the position display unit.

(3) Perform the manual operation (see Section 5.4).

(4) Return the tool to the recorded coordinate value (the start point of the manual operation).

(5) In order to resume the NC tape operation, set the MODEL SELECT switch to TAPE.

(6) Push the CYCLE START button on the control panel.

5.8 Operation by the MDI Unit

(Refer to Section 4.1.4).

5.9 Operation by the Manual Data Input Unit during NC Tape Operation

(1) Set the SINGLE BLOCK switch to ON (see Section 4.7.11).

(2) Perform the operation by the Manual Data Input unit (see Section 4.1.4).

(3) In order to resume the NC tape operation, set the MODE SELECT switch to TAPE.

(4) Push the CYCLE START button on the control panel.

5.10 Preparation before Turning Off Power

(1) Make sure that the CYCLE START display lamp is off.

(2) Set the tape reader switch to OFF.

5.11 Turning Off Power

(1) Push the POWER-OFF button on the display panel.

(2) After turning off power, make sure that the tape reader lamp is out.

6. MAINTENANCE

6.1 Daily Maintenance

The operator maintains the following points:

6.1.1 Cleaning the Tape Reader

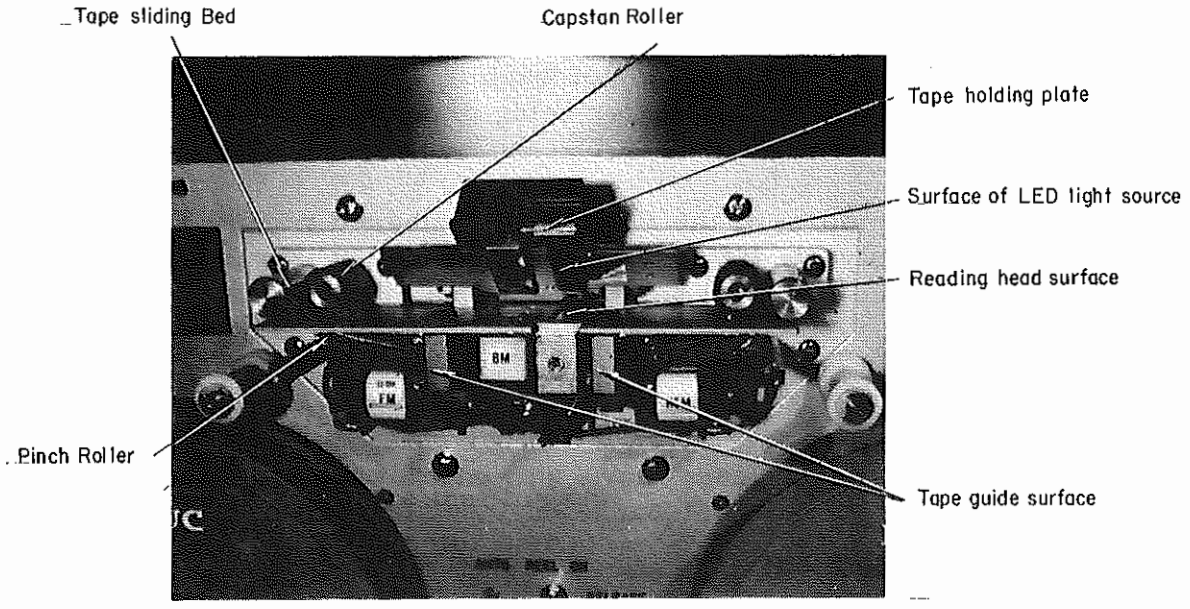


Fig 6.1.1 Tape Reader section

Item	Cycle	Cleaned portion	Cleaning method
1	Daily	Tape guide surface	Clean with gauze with absolute alcohol.
2	Daily	Read head surface	Clean with gauze or a thin brush with absolute alcohol. A thin brush is more effectively used to clean the recess of the read head.
3	Weekly	Tape box bottom	Clean with a cloth, brush or cleaner.
4	Weekly	Magnet section for tape holder	Clean with gauze with absolute alcohol. In order to wipe out iron powder, etc. in the magnet section, remove the cover of the reading section and use an adhesive tape.
5	Weekly	Tape path (1) Tape holder (2) Stop shoe (3) Capstan roller (4) Pinch roller	Clean with gauze with absolute alcohol.

6.1.2 Cleaning the Air Filter

The air filter mounted on the lower part of the back side of the servo unit, requires cleaning, since its dust catching efficiency is worsend due to much dust on it. For cleaning the filter, after removing it from the unit with the two set screws loosened, usually give it light shakes, and apply a compressor to its inside for dedusting. When it is very dirty, after putting it in a 2 to 4 g/liter synthetic detergent solvent for more than several minutes, wash it with pressure in the liquid, taking advantage of the cushion of its material. At this time, do not wash it with rubbing. And then, dry it out in the shade. After being dried out, mount it on the servo unit.

Furthermore, if the filter mat is cleaned repeatedly in a detergent solvent, its dust catching efficiency is worsend; therefore, replace it before being cleaned so many times.

6.2 Power Fuses

There are power fuses in the control section and the servo unit. If any of the fuses blows out, after examining its cause, communicate to a maintenance engineer to replace it with a spare.

6.3 Fault Causes and Remedies

Many fault causes of the equipment can be discovered by a simple inspection; therefore, when the equipment malfunctions, inspect the following points:

6.3.1 Power cannot be turned on totally.

* The service power is not on.

6.3.2 Power cannot be turned on partially.

(1) The tape reader lamp does not light up.

* Inspect the lamp

(2) LEADY lamp does not light.

* If the alarm lamp is on, inspect the alarm referencing 6.3.3.
* If the alarm lamp is not on, inspect the LEADY lamp.

6.3.3 The alarm lamp is on.

(1) OH alarm

* If the ambient temperature is in excess of 45°C, lower it.
* If the DC servo motor is overloaded, review the machining requirements.
* The cooling fan is in failure.

(2) OT alarm

* When the machine tool is at the stroke end, refer to Section 4.1.1 (3) for the treatment.

(3) SV alarm

* Communicate to a maintenance engineer. Refer to Section 4.1.3.

(4) P/S alarm

* There are some erroneous data in the program or MDI. Refer to Section 4.1.3 for errors.

(5) * If any tape code that is not found in the functional code list (Appendix 1) is detected within the significant Information on NC tape, take an action according to Section 4.1.3.

(6) TV alarm

* If an error is detected in a block of NC tape in parity checking in the vertical direction, refer to Section 4.1.3 for the treatment.

6.3.4 PARITY alarm lamp lights up

* There occurs any fault therein.

6.3.5 BATTERY alarm lamp light up

* Battery voltage become under set value.

6.3.6 NC tape operation is impossible.

* If any of the alarm lamps lights up, see Section 6.3.3.

* If the MODE SELECT switch is not at TAPE or MEMORY, switch it properly.

* If the tape reader switch is not at the specified position, set the switch properly.

AUTO for the tape reader without reels

REEL ON/OFF for the tape reader with reels

* If the NC tape cannot be fed, check if the tape is set correctly.

6.3.7 Manual operation is impossible.

* If any of the alarm lamps lights up, refer to Section 6.3.3.

* If the MODE SELECT switch is not at the manual operation position (JOG, STEP or HANDLE), switch it properly.

* If there is an error in the manual operation, do it according to Section 5.4.

6.3.8 The machine tool malfunctions

* If there is any error in NC setting or in programming, correct the NC tape.

If the fault still remains unrepaired even after the above treatments done, communicate to a maintenance engineer, or to Fujitsu FANUC. In this case, check the following points previously:

* When did the fault occur? (What is the operation/machining during which the fault occurred?)

* Can the fault be realized again? (Can the same fault occur again under the same condition?)

* What is the kind of fault? (Did not many kinds of faults occur simultaneously?)

APPENDIX

Appendix 1 List of Tape Codes

The ISO and EIA codes explaind in the table below can be used in this FANUC.

Which codes can be used in the table below, refer to the manual issued by machine tool builder.

ISO code										EIA code										Meaning
	8	7	6	5	4		3	2	1		8	7	6	5	4		3	2	1	
0			○	○		○				0			○			○				Numeral 0
1	○		○	○		○			○	1						○			○	" 1
2	○		○	○		○		○		2						○		○		" 2
3			○	○		○		○	○	3				○		○		○	○	" 3
4	○		○	○		○	○			4						○	○			" 4
5			○	○		○	○		○	5				○		○	○		○	" 5
6			○	○		○	○	○		6				○		○	○	○		" 6
7	○		○	○		○	○	○	○	7						○	○	○	○	" 7
8	○		○	○	○	○				8					○	○				" 8
9			○	○	○	○			○	9				○	○	○			○	" 9
A		○				○			○	a			○	○		○			○	Angular dimension around X axis
B		○				○		○		b			○	○		○		○		Angular dimension around Y axis
C	○	○				○			○	c			○	○	○	○			○	Angular dimension around Z axis
D		○				○	○			d			○	○		○	○			Cutter conpensation number
E	○	○				○	○		○	e			○	○	○	○	○		○	
F	○	○				○	○	○		f			○	○	○	○	○	○		F function (feed function)
G		○				○	○	○	○	g			○	○		○	○	○	○	G function (preparatory function)
H		○			○	○				h			○	○		○	○			Tool offset number
I	○	○			○	○			○	i			○	○	○	○	○		○	X-axis element of arc center, etc.
J	○	○			○	○		○		j			○		○		○		○	Y-axis element of arc center, etc.
K		○			○	○		○	○	k			○		○		○	○		Z-axis element of arc center, etc.
L	○	○			○	○	○			l			○				○	○	○	Fixed cycle number
M		○			○	○	○		○	m			○		○		○	○		M function (miscellaneous function)
N		○			○	○	○	○		n			○				○	○	○	Sequence number
O	○	○			○	○	○	○	○	o			○				○	○	○	Same as N in normal direction (EIA)
P		○		○		○				p			○		○		○	○	○	Dwell in Fixed cycle, Dwell, Program copy
Q	○	○		○		○			○	q			○		○	○	○			Step dimension or sift value in Fixed cycle, Program copy
R	○	○		○		○		○		r			○			○	○		○	Point R in Fixed cycle, Program copy or sub tape
S		○		○		○		○	○	s				○	○		○		○	S function (spindle-speed function)
T	○	○		○		○	○			t				○			○		○	T function (tool function)
U		○		○		○	○		○	u				○	○		○	○		Secondary motion dimension parallel to X
V		○		○		○	○	○		v				○			○	○	○	Secondary motion dimension parallel to Y
W	○	○		○		○	○	○	○	w				○			○	○		Secondary motion dimension parallel to Z
X	○	○		○	○	○				x				○	○		○	○	○	Primary X motion dimension
Y		○		○	○	○			○	y				○	○	○	○			Primary Y motion dimension
Z		○		○	○	○		○		z				○		○	○		○	Primary Z motion dimension
DEL	○	○	○	○	○	○	○	○	○	Del			○	○	○	○	○	○	○	** Delete (punching errors)
NUL						○				Blank							○			** No hole
BS	○				○	○				BS				○		○	○		○	** Back space
HT					○	○			○	Tab				○	○	○	○	○		Tablator
LF or NL					○	○		○		CR or EOB	○						○			End of Block (*)
CR	○				○	○	○		○											
SP	○		○			○				SP					○		○			** Space
%	○		○			○	○		○	EOR						○	○		○	○	Starting program, Stopping search
(			○		○	○														Control Out
)	○		○		○	○			○											Control In
+			○		○	○		○	○	+			○	○	○		○			** Pogitive direction
−			○		○	○	○		○	−			○				○			Negative direction
:			○	○	○	○		○												Same as N in normal direction (ISO)
/	○		○		○	○	○	○	○	/				○	○		○		○	Optional block skip

* Can be used in this FANUC system with options.

** Ignored by NC equipment. An alarm is given for a blank character position in significant information in the EIA code.

Appendix 2 Range of Various Command Values

The range of command values that can be feed to this device is shown below.

2.1 Sequence Number

Address	Range of Command Value
N	001 - 999

2.2 Preparatory Functions

G code	Group	Function	Section
G00	A	Positioning	B
G01	A	Linear interpolation	B
G02	A	Circular interpolation CW	O
G03	A	Circular interpolation CCW	O
G04	*	Dwell	B
G12	*	Circle cutting CW	O
G13	*	Circle cutting CCW	O
G14	A	Herical interpolation CW	O
G15	A	Herical interpolation CCW	O
G17	B	XY plane selection	O
G18	B	ZX plane selection	O
G19	B	YZ plane selection	O
G25	*	Programming copy	O
G27	*	Zero return check	O
G28	*	Automatic zero return	O
G29	*	Return to zero return start point	O
G38	*	Cutter compensation vector set	O
G39	*	Corner offset	O
G40	C	Cutter compensation cancel	O
G41	C	Cutter compensation left side	O
G42	C	Cutter compensation right side	O
G43	*	Tool offset B + direction	O
G44	*	Tool offset B - direction	O
G45	*	Tool offset A increase	B
G46	*	Tool offset A decrease	B
G47	*	Tool offset A double increase	B
G48	*	Tool offset A double decrease	B
G49	*	Tool offset B cancel	O
G73	D	Fixed cycle #10	O
G76	D	Fixed cycle #11	O
G80	D	Fixed cycle cancel	O
G81	D	Fixed cycle #1	O

G code	Group	Function	Section
G82	D	Fixed cycle #2	O
G83	D	Fixed cycle #3	O
G84	D	Fixed cycle #4	O
G85	D	Fixed cycle #5	O
G86	D	Fixed cycle #6	O
G87	D	Fixed cycle #7	O
G88	D	Fixed cycle #8	O
G89	D	Fixed cycle #9	O
G90	E	Absolute programming	B
G91	E	Incremental programming	B
G92	*	Programming of absolute zero point	B
G98	F	Return to initial level in fixed cycle	O
G99	F	Return to R point level in fixed cycle	O

2.3 Dimension Word

X, Y, Z, U, V, W, A, B, C, I, J, K

Increment system	Maximum command value	Increment system	Maximum command value
M1V	8388.60 mm	I3V	419.4303inch
M2V	8388.607mm	I4V	167.7721inch
M3V	4194.303mm	D1V	8388.60 deg.
M4V	1677.721mm	D2V	8388.607 deg.
I1V	838.860 inch	D3V	4194.303 deg.
I2V	838.8607inch	D4V	1677.721 deg.

2.4 Feed Function

2.4.1 Rapid Traverse (G00 Positioning)

Setting can be effected at any value in a range of 1 to 250 kpps (in increments of 1 kpps) independently for each axis. An optimum value, however, is fixed by allowing for the requirement of the servo and mechanical systems.

2.4.2 Cutting Feed (G01, G02 etc.)

(1) 1 - digit F-code Feed

Increment system	Setting range	
M1V, M2V	5 ~ 1,200	mm/min
M3V	2.5 ~ 600	mm/min
M4V	1 ~ 240	mm/min
I1V, I2V	0.5 ~ 120	inch/min
I3V	0.25 ~ 60	inch/min
I4V	0.1 ~ 24	inch/min
D1V, D2V	5 ~ 1,200	deg/min
D3V	2.5 ~ 600	deg/min
D4V	1 ~ 240	deg/min

(2) 4-digit F-code Feed

Increment system	Format	F-code range
M1V	F40	F01 ~F3600
M2V	F41	F01 ~F36000
M3V	F41	F05 ~F18000
M4V	F41	F02 ~F 7200
I1V	F41	F01 ~F 3600
I2V	F32	F01 ~F 36000
I3V	F32	F01 ~F 18000
I4V	F32	F01 ~ 7200
D1V	F40	F01 ~F3600
D2V	F41	F01 ~F36000
D3V	F41	F01 ~F18000
D4V	F41	F01 ~F 7200

decimal point

2.5 Dwell, Dwell in Fixed Cycle (G04, G82, G89)

Increment system	Maximum command value (sec)
M1V, I1V	1048.57
M2V~M4V, I2V~I4V	1048.575

2.6 Auxiliary Function

SOO, SOOOO
TOO, TOOOOO
MOO
BOOO
} Refer to MTB's manual

2.7 Tool Offset/Cutter Compensation

Increment system	Maximum Command value
M1V	± 8388.60 mm
M2V~M4V	± 999.999 mm
I1V	± 838.860 inch
I2V~I4V	± 999.999 inch

2.8 Repeat Number

L : Fixed cycle, helical interpolation
R : Program copy, sub tape control

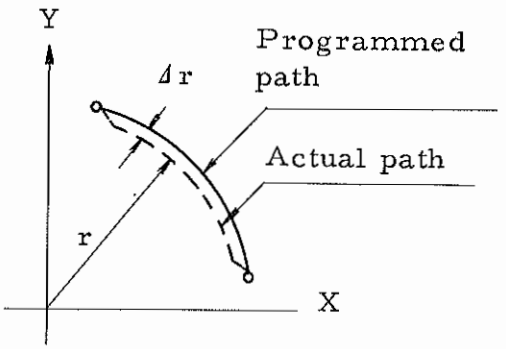
Increment system	Maximum Command value
M1V, M2V	8388607
M3V	4194303
M4V	1677721
I1V, I2V	8388607
I3V	4194303
I4V	1677721

Appendix 3 Nomographs

3.1 Error in the Radius Direction in Circular Cutting

In this FANUC, a speed-up circuit is used for smooth acceleration and deceleration, with the output pulse train following the command pulse train with a primary lag system.

When a DC servo motor is used, a lag is caused between the input and output axes by the hydraulic motor. In linear interpolation the tool moves on a specified segment of a line; thus no error is caused. In circular interpolation, especially in high-speed circular cutting, an error is caused in the radius direction, and such error can be calculated by the procedure described below.



Δr	:	Maximum error in radius direction (mm)
v	:	Feed rate (mm/sec.)
r	:	Radius of arc (mm)
T_1	:	Time constant for speed-up circuit (sec.)
T_2	:	Time constant for DC motor (sec.)

For a DC motor (secondary lag system)

$$\Delta r = \frac{1}{2} (T_1^2 + T_2^2) \cdot \frac{v^2}{r} \quad \ldots\ldots\ldots\ldots\ldots (1)$$

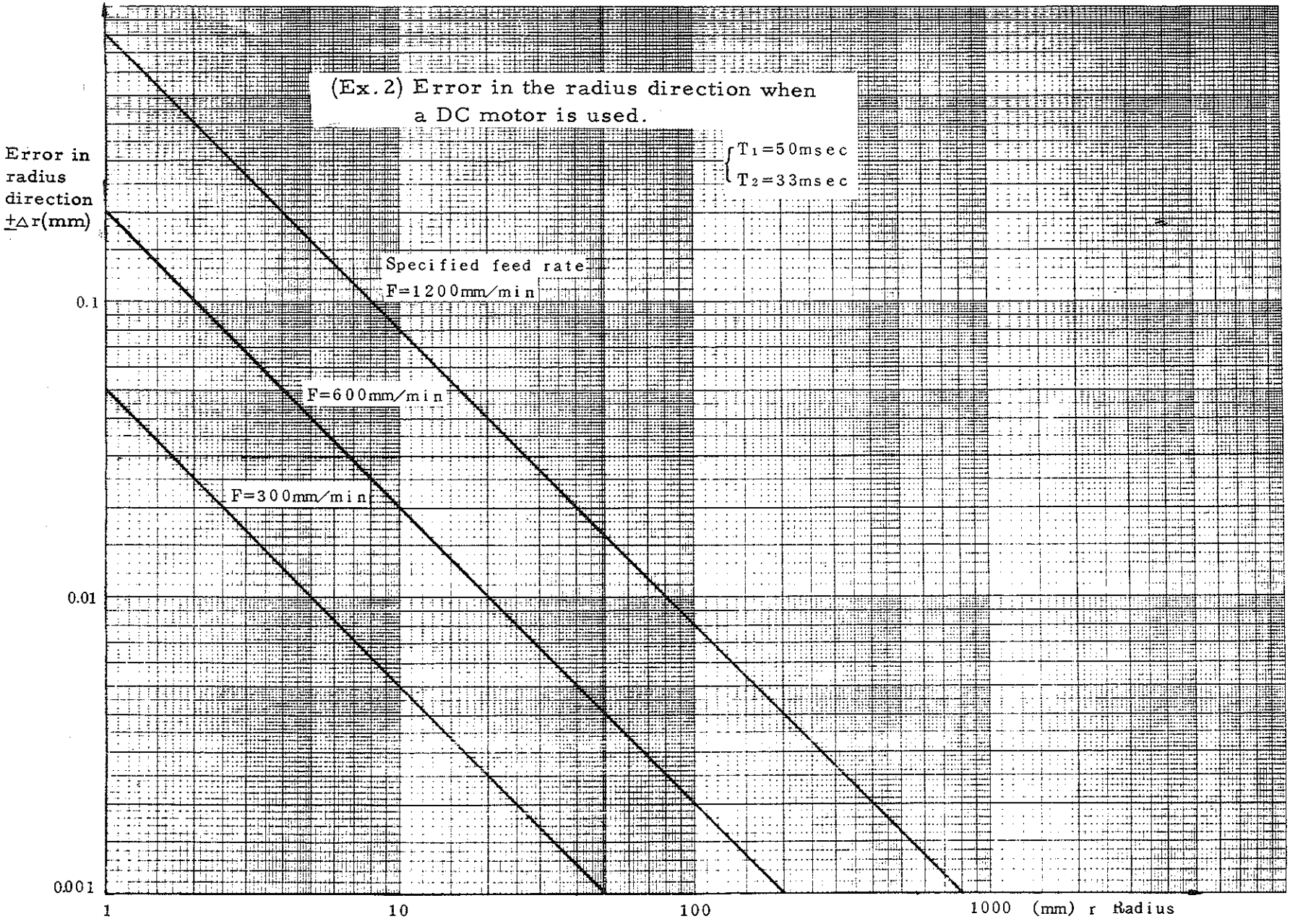
For actual processing, a processing radius, r(mm), of the workpiece and an allowable error, Δr (mm), are given, and allowable speed limit, v (cm/sec.), can be calculated by formulas (1).

The time constant of the speed-up circuit set in this FANUC varies with the machine. Refer to the manual issued by the machine tool builder.

Time constant for the DC motor

$$T2 = 33 \times 10^{-3} \text{ (sec)}$$

Thus a graph which allows a specified feed rate, F, to be easily read for a particular processing radius, r, and a particular allowable error, Δr, will be quite useful.


(Ex.2) Error in the radius direction when a DC motor is used.
T1=50msec
T2=33msec
Error in radius direction ±△r(mm)
Specified feed rate
F=1200mm/min
F=600mm/min
F=300mm/min
0.1
0.01
0.001
1
10
100
1000 (mm) r Radius

3.2 Tool Path at a Corner

3.2.1 Outline

Due to a lag of the servo system (attributable to the speed-up circuit or a motor when a DC servo motor is used), the actual path of the tool (to be more exact, the locus of the center of the tool) at a corner slightly deviates from the programmed path as shown in Fig. 3.2.1.

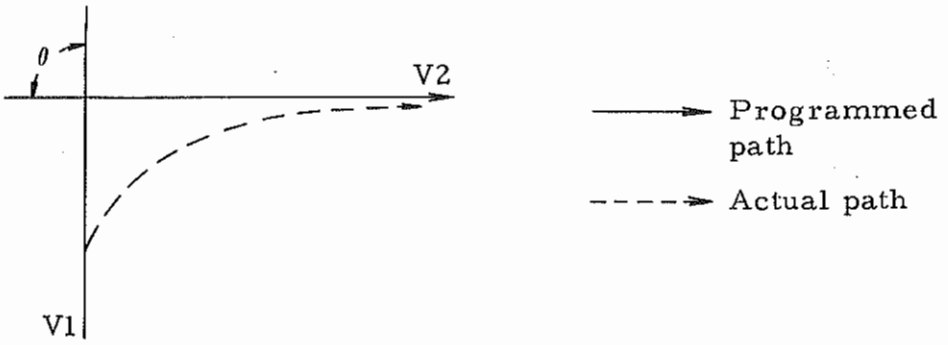


Fig. 3.1 Tool Path at Corner

A path of the tool is determined by the next parameters.

(1) Feed rate (V1, V2)

(2) Angle of corner (θ)

(3) Time constant of speedup circuit (T_1)

(4) Type of motor used

(5) Use or non use of buffer register

In this supplement, the tool path is theoretically analyzed by use of the above parameters, and a path based on supposed parameters is presented in graph form. The programming for a tool path must allow for the deviation of path discussed above and ensure that the workpiece will be processed to a desired accuracy.

3.2.2 Analysis

The tool path shown in Fig. 3.2 is analyzed under the following conditions.

(1) The feed rate is constant for blocks before and after the corner.

(2) The control section is provided with a buffer.

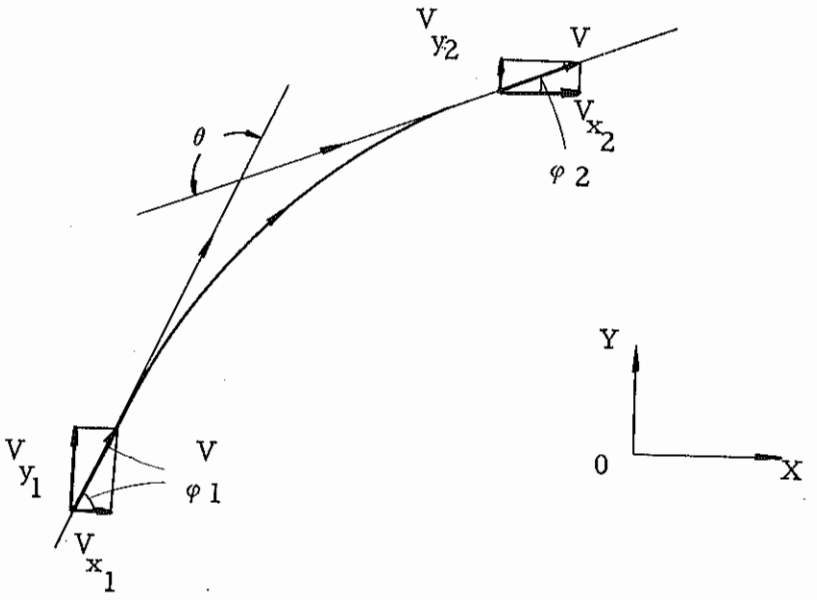


Fig. 3.2 Command

[Conditions]

$$Vx1 = V \cdot \cos \varphi 1$$

$$Vy1 = V \cdot \sin \varphi 1$$

$$Vx2 = V \cdot \cos \varphi 2$$

$$Vy2 = V \cdot \sin \varphi 2$$

$$\pi - (\varphi 1 + \varphi 2) = \theta$$

Explanation of Symbols

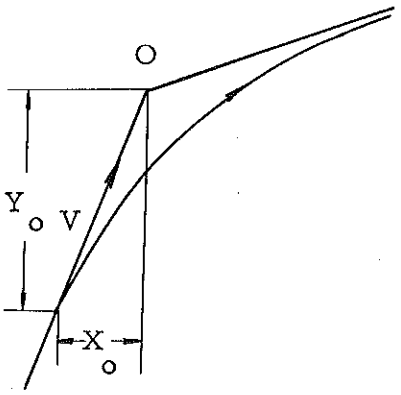
V	Feed rate for blocks before and after a corner
Vx1	X-axis element of the feed rate for the preceding block
Vy1	Y-axis element of the feed rate for the preceding block
Vx2	X-axis element of the feed rate for the next block
Vy2	Y-axis element of the feed rate for the next block

θ Angle of the corner

$\varphi 1$ Angle between the programmed tool path of the preceding block and the X axis,

$\varphi 2$ Angle between the programmed tool path of the next block and the X axis

(1) Calculation of Initial Value



X and Y coordinates upon completion of distribution by the control section, are determined by the feed rate, time constant of the speed-up circuit and time constant of the DC motor.

Fig. 3.3 Initial Values

$$Xo = Vx1\,(T_1 + T_2) \quad \ldots\ldots\ldots \quad (1)$$

$$Yo = Vy1\,(T_1 + T_2) \quad \ldots\ldots\ldots \quad (2)$$

T_1: Time constant of the speed-up circuit

T_2: Time constant of the DC motor

(2) Analysis of tool path

The X - and Y-axis (feed rates at a corner are calculated as follows.

$$Vx(t) = (Vx2 - Vx1)\left[1 - \frac{Vx1}{T_1 + T_2}\left\{T_1 \exp\left(-\frac{t}{T_1}\right) - T_2 \exp\left(-\frac{t}{T_2}\right)\right\}\right] + Vx1$$

$$= Vx2\left[1 - \frac{Vx1}{T_1 + T_2}\left\{T_1 \exp\left(-\frac{t}{T_1}\right) - T_2 \exp\left(-\frac{t}{T_2}\right)\right\}\right] \quad \ldots\ldots\ldots \quad (3)$$

$$Vy(t) = \frac{Vy1 - Vy2}{T_1 + T_2} \left\{ T_1 \exp\left(- \frac{t}{T_1} \right) - T_2 \exp\left(- \frac{t}{T_2} \right) \right\}$$

$$+ Vy2 \qquad \ldots\ldots (4)$$

The coordinates of the tool path at time t are calculated as follows.

$$X(t) = \int_o^t Vx(t)dt - Xo$$

$$= \frac{Vx2 - Vx1}{T_1 + T_2} \left\{ T_1^2 \exp\left(- \frac{t}{T_1} \right) - T_2^2 \exp\left(- \frac{t}{T_2} \right) \right\}$$

$$- Vx2 (T_1 + T_2 - t) \qquad \ldots\ldots (5)$$

$$Y(t) = \int_o^t Vy(t)\, dt - Yo$$

$$= \frac{Vy2 - Vy1}{T_1 - T_2} \left\{ T_1^2 \exp\left(- \frac{t}{T_1} \right) - T_2^2 \exp\left(- \frac{t}{T_2} \right) \right\}$$

$$- Vy2 (T_1 + T_2 - t) \qquad \ldots\ldots\ldots (6)$$

3.2.3 Calculation Examples

Examples of tool locus are shown in Fig. 3-4 to 3-6 by use of formulas (5) and (6).

Conditions

$Vx1 = 0$pps

$Vy1 = 1,000$ pps = 10 mm/sec. (A least input increment of 0.01 mm/sec. is presupposed.)

$T_2 = 33 \times 10^{-3}$ sec.

Vx2 and Vy2 vary with corner angle

θ and T_1 change as shown in Table 3-1.

Table 3-1

θ	T_1
45°	25×10^{-3} sec.
90°	50×10^{-3} sec.
135°	100×10^{-3} sec.
	200×10^{-3} sec.

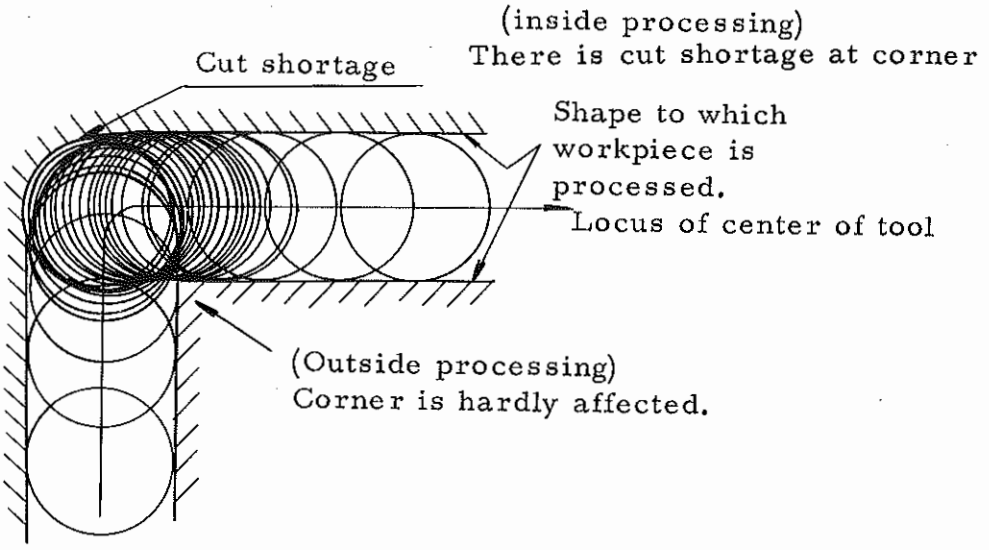
(Note on reading drawings)

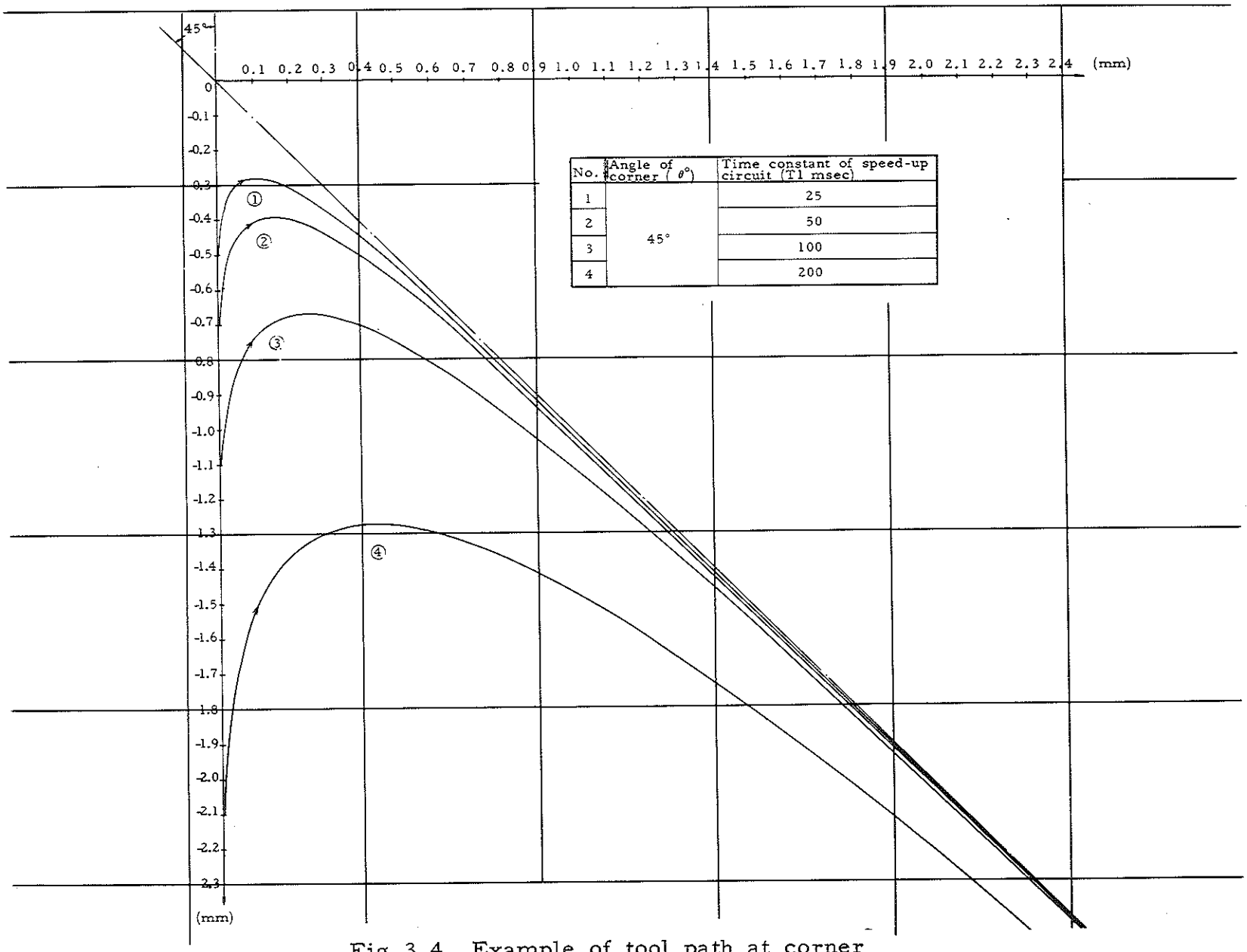
o Feed rate

The feed rate of 1,000 pps is used as the basis of calculations. For other feed rates, change the scale proportionally to such feed rates.

o' If a corner angle and a time constant of the speed-up circuit are other than those used in the graph, a calculation can be made using the graph as a reference.

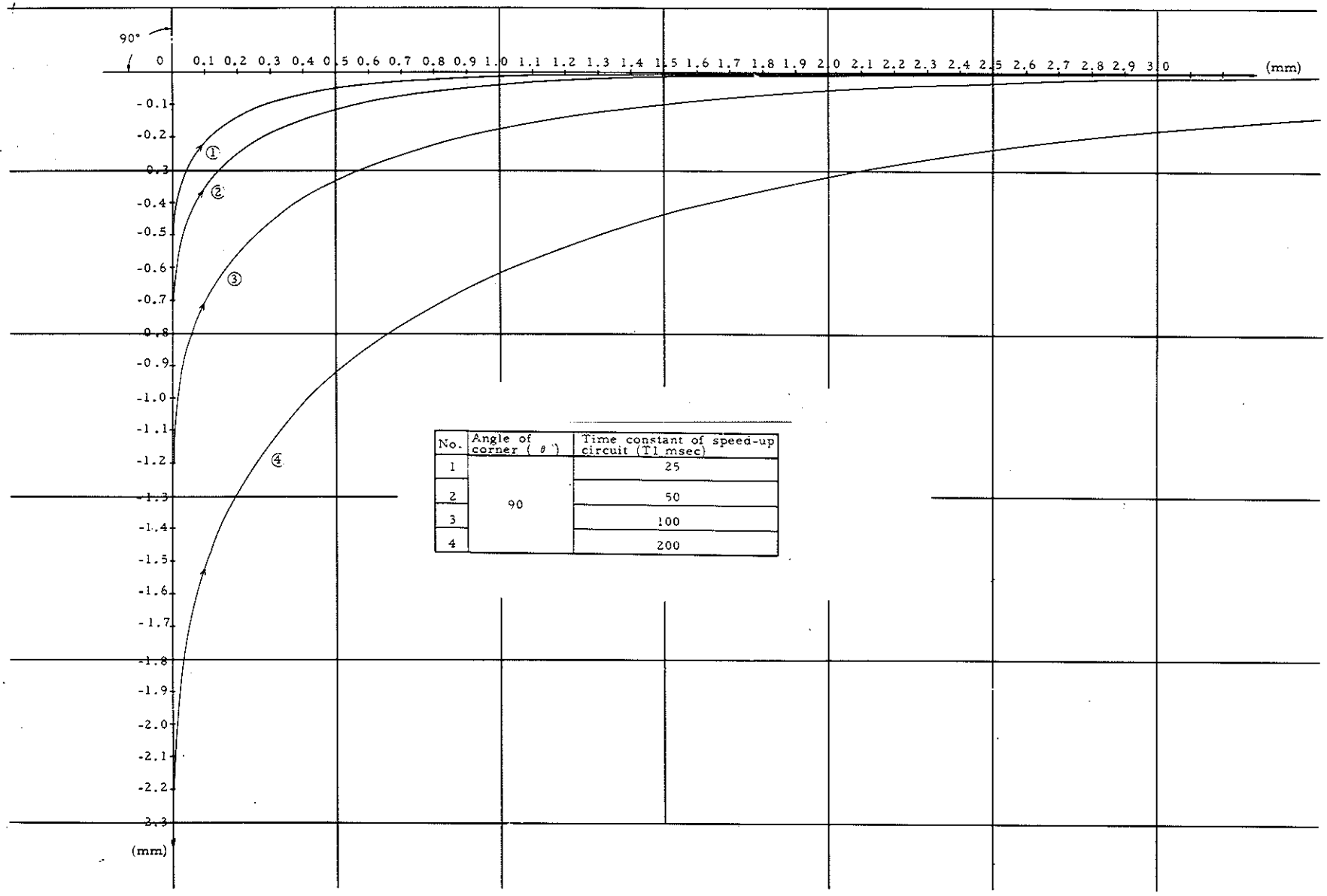
The graph-analyzes the locus of the center of the tool, which is slightly different from the shape to which the workpieces is processed.





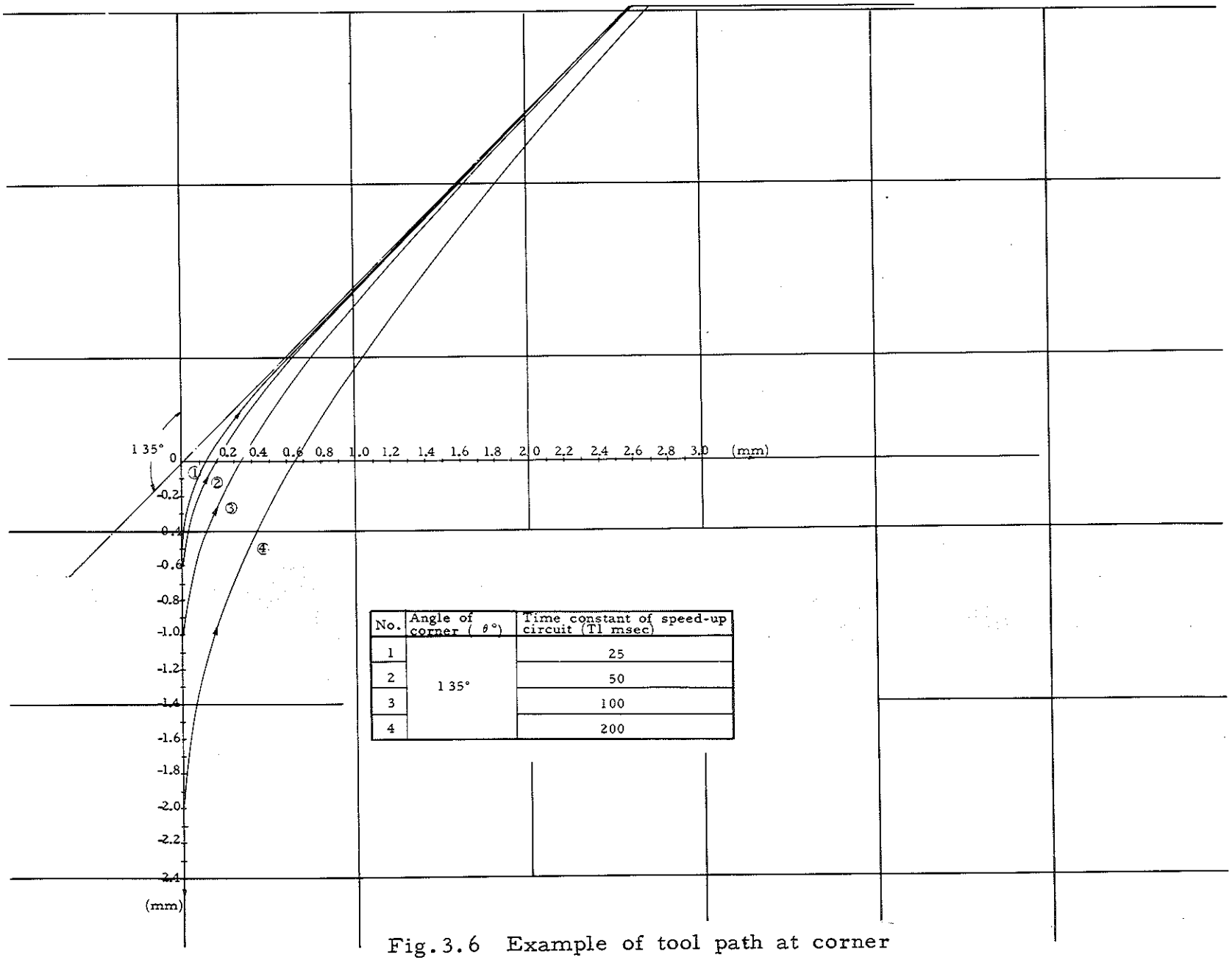
No.	Angle of corner (θ°)	Time constant of speed-up circuit (T1 msec)
1	45°	25
2		50
3		100
4		200

Fig.3.4 Example of tool path at corner



No.	Angle of corner (θ)	Time constant of speed-up circuit (T) msec
1	90	25
2		50
3		100
4		200

Fig.3.5 Example of tool path at corner

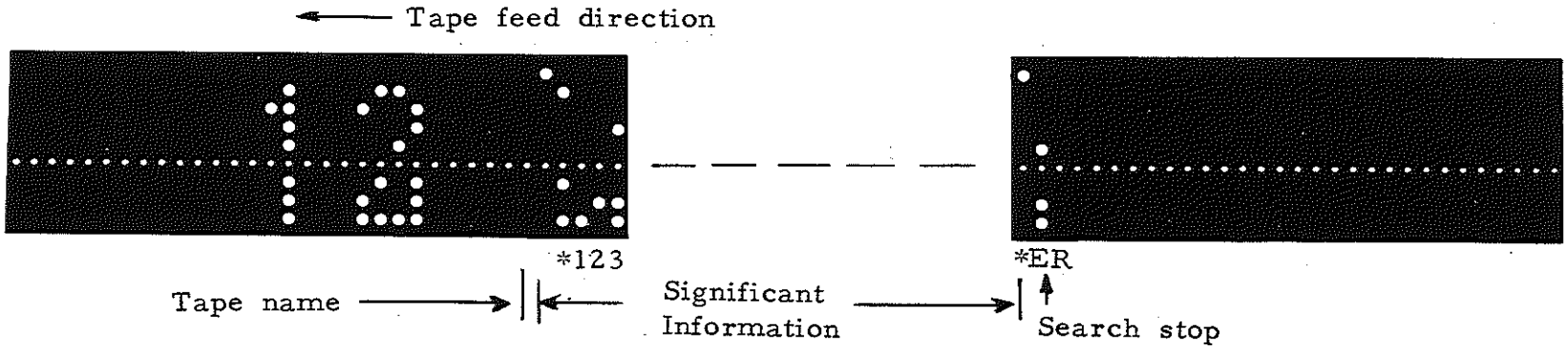


No.	Angle of corner (θ°)	Time constant of speed-up circuit (T1 msec)
1	135°	25
2		50
3		100
4		200

Fig.3.6 Example of tool path at corner

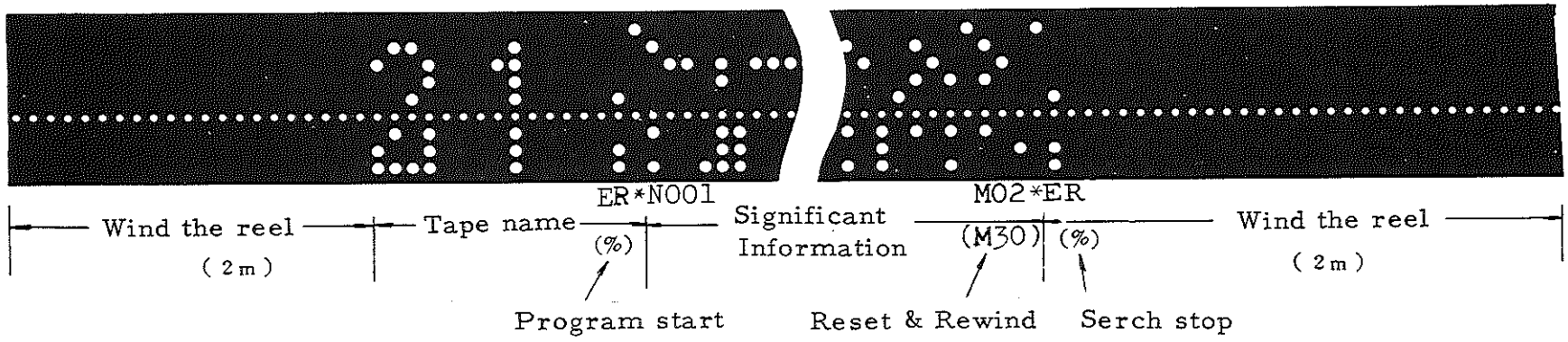
Appendix 4 How to Prepare an NC Tape

NC tape for the tape reader without reels should be prepared as illustrated below.



(EIA code)

NC tape for the tape reader with reels should be prepared as illustrated below.



Program start ..... This is used for rewind stop.

Reset & .... The control unit is reset by the Miscellaneous functions (M02 or M30) and the tape reader continues to rewind up to read in ER(%) code.

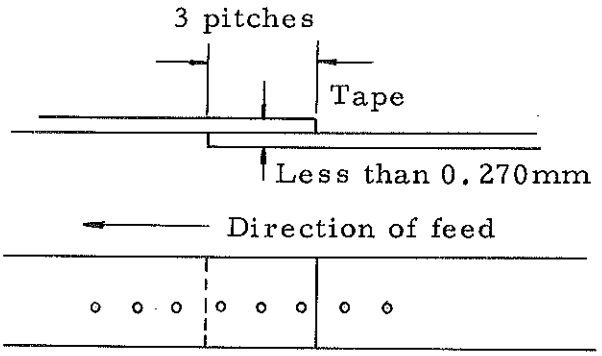
Appendix 5 How to Join NC Tapes

If a loop of punched tape is to be made or tape is broken or in other similar cases, two tape sections must be jointed.

Join such tape sections as described below.

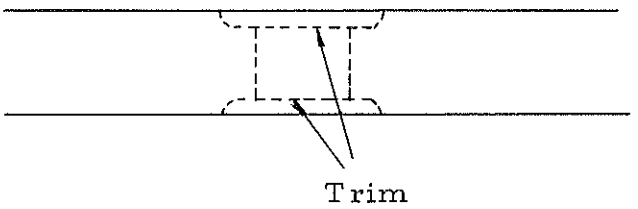
(a) But the two tape sections and paste them together, placing the section on that side to which the tape is fed, on the other.

(b) The joint at which the two tape sections overlap must be approx. 3 pitches.



(c) Make sure the punched holes of the overlapped tape sections at the joint are correctly aligned. Use special care with the feed holes.

(d) Trim both edges of the joint to smoothen the edges.



(e) Make sure punched holes are not closed by the paste.

Appendix 6. RESET

Item	Reset
Sequence Number display	Unchanged
Modal G codes	"
Move command (In operation, move command in Feed Hold)	Canceled (In motion, stop after deceleration and canceled)
Fixed cycle parameter	P, Q and R ... Unchanged L ... canceled (P, Q and R are canceled by G80 or Group A G-code)
Feed Function	Unchanged
Tool Offset number	"
Cutter Compensation vector	"
Offset Amount memory	"
Auxiliary Function code	M: Canceled S, T: Unchanged
Alarm lamp	If alarm state is released, this lamp is turn off.
READY lamp	Unchanged
ZERO POSITIONED, BLOCK RETURN lamps	Unchanged
LABEL SKIP lamp	In MDI mode : Unchanged In other MDI mode : Light up
BUFFER lamp	In MDI mode : Unchanged In other MDI mode : Light out
Position display	Unchanged
Cycle start lamp	Light out
Feed Hold lamp	Light out

Appendix 7. State and Display of Control Unit/Mode Selection and Operation

The control unit includes various states, and the display varies according to the state. Also, it should be noted that there are operational restrictions varying with the state. The state of the control unit varies according to the operation. Refer to the section on operations as to how it changes.

7.1 State

(1) Reset State

This is a state in which the control circuitry of the equipment is completely reset, and all tape information except Sequence Numbers is canceled.

(2) Cycle Operating State

This is a state that is not in a reset state, and is switched to from a reset state by pushing the Cycle start button or by inputting data on the MDI unit. This state returns to a reset state by reset.

a) Cycle Operating State

This is a state in which the equipment is in operation (pulse distribution, dwelling, or Miscellaneous function operation) by other than manual operation.

b) Cycle Operation Hold State

This is a state in which the equipment interrupts pulse distribution temporarily, that is, resets during execution of one block.

c) Cycle Operation Stop State

This is a state in which execution of one block is completed and stopped normally or forcibly.
The operational mode is roughly divided into the Cycle Operation mode and the manual operation mode.

7.2 Mode

(1) Cycle Operation Mode

This is a mode in which operation is carried out based on the data entered according to a predetermined input format. There are the following four types.

a) Tape Mode

This is a mode in which operation is carried out based on the data on a tape set on a paper tape reader. With Single Block on, one block's operation is executed each time the Cycle start button is pushed.

b) Manual Data Input Mode

This is a mode in which operation is carried out based on the data set on the MDI unit. The Cycle Operation stop state is entered after the execution. This is basically the same as the Single Block mode.

c) Memory Mode

This is a mode in which operation is carried out based one block's operation is executed each time the Cycle start button is pushed.
time the Cycle start button is pushed.
For operation in the Cycle Operation mode, there are the Cycle Start terminal to effect a start and the Feed Hold terminal to effect a hold.

(2) Manual Operation Mode

This is a mode in which operation is made by a manual switch, and includes the following four types. (As for the items a) and b), either of them is selectable.)

a) Step Feed Mode

This is a mode in which machine tool is moved by one pulse each time the push button is pushed.

b) Manual Handle Feed Mode

This is a mode in which pulses are generated by turning a manual handle, thereby moving a machine tool.

c) Jog Mode

This is a mode in which pulses are generated while the push button is pushed, thereby moving a machine tool continuously.

(3) Special Mode

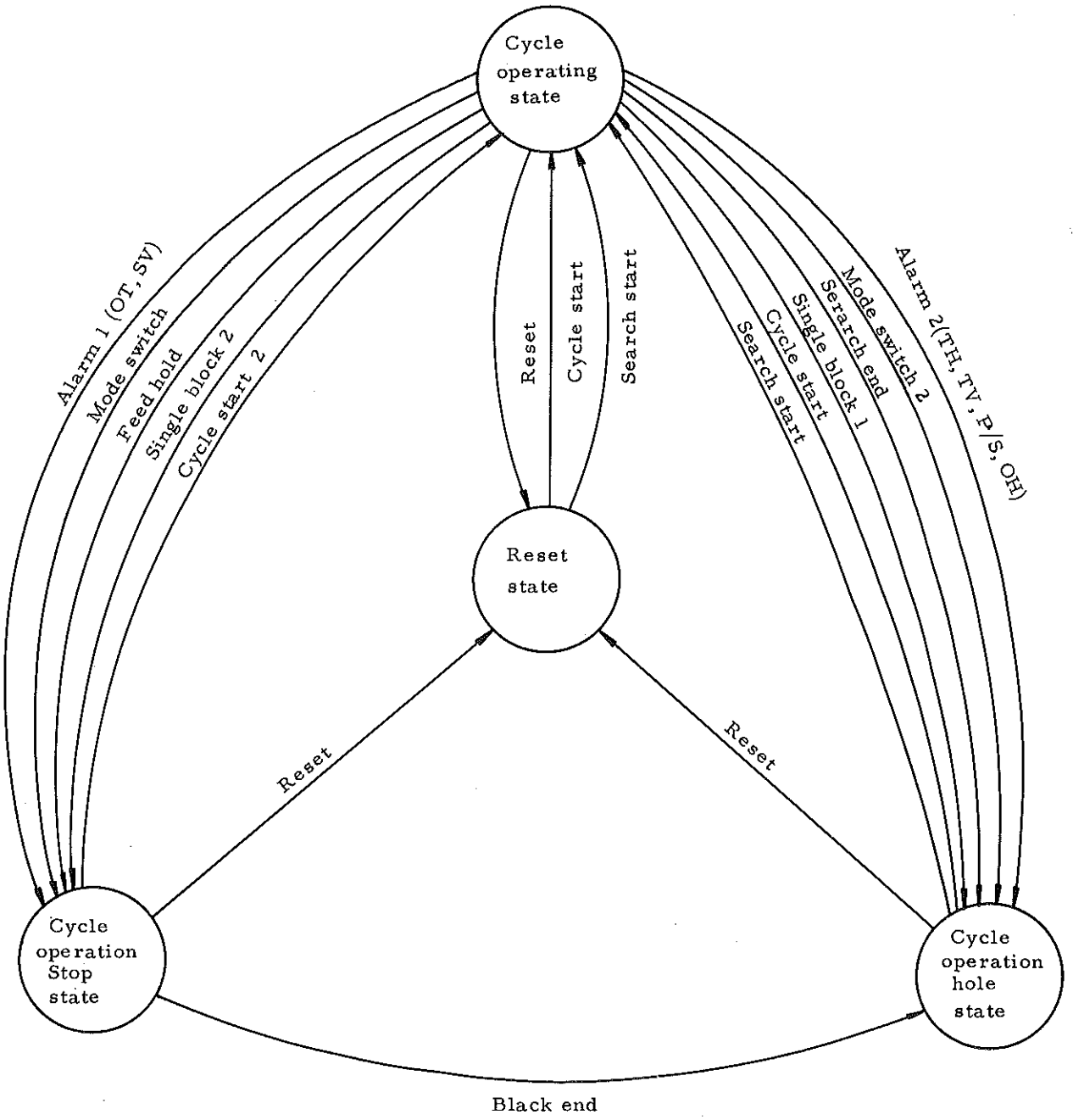
This is a mode not relating directly to the operation of a machine tool, including the following mode.

a) Tape Edit Mode

This is a mode to modify the contents of a program stored in the memory.

7.3 Variety of state

The state of the control unit varies with the mode of operation.



(1) Alarm 1

This is caused when the OT or SV alarm occurs.

(2) Alarm 2

This is caused when the OH, TH, TV or P/S alarm occurs.

(3) Cycle Start

This is made by pushing the CYCLE START button in the Cycle Operation mode and not in an alarm state.

(4) Search Start

This is made by pushing the SEARCH START button for Sequence Number search.

(5) Feed Hold

This is made by pushing the FEED HOLD button during Cycle Operation.

(6) Mode Switch 1

This is caused when the manual operation mode is switched during Cycle Operation.

(7) Mode Switch 2

This is caused when another Cycle Operation mode is switched to during Cycle Operation. In this case, stopped state is entered after a block currently being executed is completed.

(8) Single Block 1

When execution of one block by Single Block is completed.

(9) Single Block 2

This is caused when excution of each operation in Fixed Cycle mode.

(10) Block End

This is caused when excution of one block command or dwell is completed.

(11) Search End

This is caused when a proper Sequence Number is detected during Sequence Number search.

(12) Reset

This is made by pushing the RESET button, remote reset, emargency stop or reset & rewind.