FANUC SYSTEM 6M-MODEL B

OPERATOR'S MANUAL

B-54044E/02

CONTENTS

I. GENERAL

1. INTRODUCTION

The FANUC SYSTEM 6M, a high-accuracy, high-performance fixed-software CNC for milling machines and machining centers has been developed by FANUC Limited to satisfy the needs of NC markets in the world. The control circuit uses high-speed microprocessors, custom LSIs, semiconductor memory, and the latest storage elements to provide high reliability and significantly improve the performance/cost ratio.

The FANUC SYSTEM 6M is closed-loop CNC using FANUC DC servo motors developed by Fujitsu FANUC Limited itself, and a high-performance pulse encoder, a resolver or an inductosyn scale as the detector.

This manual explains programming, operation, and routine maintenance. For convenience the basic and optional specifications are explained first. The following detailed descriptions can be referred to as required. This manual covers everything pertaining to the system 6M and, also gives the specifications for the machine operator's panel. Also refer to the manual issued by the machine tool builder.

2. NOTES ON READING THIS MANUAL

2.1 The function of an NC machine tool system depends not only on the NC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the NC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand-point of the NC. So, for details on a particular NC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.

2.2 This manual addresses as many subjects as possible. But it would become too voluminous to point out everything that should not or cannot be done. Functions which are not specifically stated as possible are impossible.

2.3 Notes refer to detailed and specific items. So, when a note is encountered, terms used in it sometimes are not explained. In this case, first skip the note, then return to it after having read over the manual for details.

II. SPECIFICATION

1. BASIC SPECIFICATION

<table>
<tr><th>No.</th><th>Item</th><th>Specification</th><th>Reference Item</th></tr>
<tr><td>(1)</td><td>Controlled axes</td><td>Three axes (X, Y and Z). 4th and 5th axes can be controlled as an option.</td><td>III 3.1</td></tr>
<tr><td>(2)</td><td>Simultaneously controlled axes</td><td><table>
<tr><th rowspan="2">No. of controlled axes</th><th colspan="5">No. of simultaneously controlled axes</th></tr>
<tr><th>Standard</th><th>Additional axis simultaneous control option</th><th>Simultaneous 3 axes option</th><th>Simultaneous 3 axes option + additional axis simultaneous control option</th><th>Simultaneous 4 axes option</th></tr>
<tr><td>3</td><td rowspan="3">2</td><td></td><td rowspan="3">3
(X, Y, Z)</td><td></td><td rowspan="3">4</td></tr>
<tr><td>4</td><td rowspan="2">2
(Includes additional axis)</td><td rowspan="2">3
(Includes additional axis)</td></tr>
<tr><td>5</td></tr>
</table>
(Note) Simultaneous 2 axes in manual operation.</td><td>III 3.1</td></tr>
<tr><td>(3)</td><td>Increment system</td><td><table>
<tr><td>Least input increment</td><td>0.001 mm</td><td>0.0001 inch</td><td>0.001 deg</td></tr>
<tr><td>Least command increment</td><td>0.001 mm</td><td>0.0001 inch</td><td>0.001 deg</td></tr>
</table>
Least input increment in metric input can be changed to 0.01 mm by a parameter setting.
Least input increment in mm and inch is reduced to one-tenth by a parameter.
Least command increment can be halved by another parameter.</td><td>III 3.2</td></tr>
<tr><td>(4)</td><td>Position detector</td><td>Pulse coder, Resolver, Inductosyn scale</td><td></td></tr>
<tr><td>(5)</td><td>Maximum programmable dimensions</td><td>±99999.999 mm , ±9999.9999 inch
±99999.999 deg</td><td>III 2.5</td></tr>
<tr><td>(6)</td><td>Command tape</td><td>8-channel opaque paper tape or color tape (colored tape is for tape reader without reels only) (EIA RS-227, ISO 1154, JIS C6246)</td><td>III 13.1</td></tr>
<tr><td>(7)</td><td>Tape code</td><td>EIA RS-244-A</td><td>III 13.2</td></tr>
<tr><td>(8)</td><td>Tape format</td><td>Variable block, word and address format.</td><td>III 2.3</td></tr>
<tr><td>(9)</td><td>Decimal point programming</td><td>The data can be inputted with a decimal point. The addresses which can use the decimal point are X, Y, Z, A, B, C, U, V, W, I, J, K, Q, R and F.</td><td>III 2.4</td></tr>
<tr><td>(10)</td><td>Rapid traverse rate</td><td>Rapid traverse rate can be set up to 24,000 mm/min, or 960 inch/mm for each axis, by parameters. An optional override can be applied to the rapid traverse rate (Fo, 25, 50 or 100%).</td><td>III 4.1</td></tr>
</table>

No.	Item	Specification	Reference Item
(11)	Feed rate	The feed rate can be set in the following range. 1 to 15,000 mm/min 0.01 to 600.00 inch/min The upper limit of the feed rate can be set by parameters. An override from 0 to 200% in increments of 10% can be effected. Feed rate unit can be changed to 0.1 mm/min, 0.001 mm/min or 0.001 inch/min by parameter.	III 4.2
(12)	Automatic acceleration and deceleration	In rapid traverse, regardless of whether the control is in manual or automatic mode, the acceleration/deceleration ramps are linear and therefore shortening the positioning time.	III 4.6
(13)	Absolute/incremental programming	G codes select either absolute or incremental programming. G90 : Absolute programming G91 : Incremental programming	III 3.8
(14)	Programming of absolute zero point (G92)	A G92 command with addresses X, Y and Z establishes the coordinate system that the coordinate values of the present tool position become commanded value.	III 3.6
(15)	Positioning (G00)	A G00 command moves the machine at the rapid traverse rate in each axis. The machine decelerates to a stop at the end point, and the in-position check (A check of whether the machine has reached the specified position) is performed by parameter setting.	III 5.2
(16)	Linear interpolation (G01)	Linear interpolation is performed at the feed rate commanded by a F code.	III 5.4
(17)	Buffer storage	One block of tape information is read into the buffer register while the previous block is being performed. This is to avoid interruptions between blocks caused by reading time of the tape reader. While data is in the buffer register, BUF is displayed on the screen of CRT.	
(18)	Dwell (G04)	A G04 command prevents the control from preceding on to the next block of information for a period of time specified by address P or X.	III 5.10
(19)	Exact stop check (G09)	In the block with a G09 command, deceleration is made at the end of execution of the block and the in-position check is made.	III 5.11
(20)	Exact stop check mode/ cutting mode (G61, G64)	If G61 is specified, in the subsequent programmed motion, deceleration is effected near the end of each block, and after an in-position state is reached, the next block is executed. If G64 is specified, in the subsequent programmed motion, no deceleration is effected, but the next block is executed. This is generally used in the cutting mode.	III 5.12

No.	Item	Specification	Reference Item
(21)	Miscellaneous function (M 2-digit)	Control (ON/OFF) of function on the machine side can be performed by the use of a 2-digit numerical command following the address "M". One M code can be specified in a block.	III 8.4
(22)	Dry run	The feed rate becomes jog feed rate in the dry run mode. Rapid traverse rate remains effective at the rapid traverse command (G00) and the optional rapid traverse override is also effective. It is possible to effect dry run in G00 by parameter setting.	IV 4.6.9
(23)	Interlock	The feeds of all axes can be stopped in common. If the interlock is set while the axes are moving, the movable member of the machine decelerates to a stop. If the interlock signal is released, the machine accelerates to start feeding.	
(24)	Servo off	The signals, which turn the servo unit output ON/OFF can be controlled separately. This allows each axis of the machine to be clamped independently.	
(25)	Single block	The commands may be executed one block at a time.	IV 4.6.3
(26)	Optional block skip	Turning the optional block skip switch on the machine operator's panel ON, blocks containing a slash code (/) at the beginning of the block are ignored.	III 2.8
(27)	External mirror image	The signs of X, Y and 4th axes's move commands on tape and MDI input can be reversed. The setting of mirror image can be made by a switch on the machine control panel or from the MDI keyboard.	IV 4.6.12
(28)	Manual absolute ON/OFF	By turning the manual absolute switch at the machine side ON/OFF, the amount of tool travel during manual operation is, or is not added to the absolute coordinate values. Manual absolute switch ON : added OFF : not added	IV 4.4.4
(29)	Auxiliary function lock	This function prevents the sending of BCD and strobe signals for the M, S, T and B functions to the machine side.	
(30)	Machine lock	In machine lock condition, the machine does not move but the position display updates as if the machine were moving. Machine lock is effected even in the middle of the block.	IV 4.6.10

No.	Item	Specification	Reference Item
(31)	Z axis command cancel	Same as if only the Z axis were in machine lock. This is useful when the contents of an NC tape are checked by writing with a pen.	
(32)	Feed hold	The feeds of all axes can temporarily be stopped. Pushing the cycle start button resumes the feed. Manual operation is allowed in the manual mode before resuming the feed.	IV 4.6.2
(33)	Override cancel	Feed rate override is clamped at 100% by a signal from the machine side.	
(34)	Emergency stop	By pushing the emergency stop button, all commands are suspended and the machine stops instantaneously.	IV 4.6
(35)	Remote reset, reset signal	The NC can be reset from outside. All commands are suspended and the machine decelerates to a stop. The reset signal is output when the reset button on the NC, the emergency stop or the remote reset is in operation.	
(36)	Remote power supply ON/OFF	It is possible to turn the power supply ON and OFF by a switch on the machine tool (such as from the operator's panel at the machine side) as well as by means of the power supply ON/ OFF buttons on the front panel of the NC.	
(37)	Overtravel	By receiving a signal informing that one of the machine's movable parts have reached the stroke end, the machine decelerates to stop and the corresponding alarm number is displayed.	
(38)	NC ready signal	This signal is sent to the machine side when the NC power supply is turned on and the control is ready to operate. The signal output stops when the power is turned off.	
(39)	Servo ready signal	This signal is sent to the machine side when the servo system has become operational. When the signal cuts off, the READY indication on the CRT turns off and any axis that is in motion will brake to a stop.	
(40)	Rewind signal	The NC outputs this signal while the tape reader is rewinding.	
(41)	NC alarm signal	The NC outputs this signal in the alarm condition.	
(42)	Distribution end signal	The NC outputs this signal when a move command has ended. If M, S, T or B function and a move command are commanded in the same block, the M, S, T or B function can be executed after the move command has completed by using this signal.	

No.	Item	Specification	Reference Item
(43)	Cycle operation signal	The NC outputs this signal during cycle operation.	
(44)	Cycle start lamp signal	The NC outputs this signal at cycle start.	
(45)	Feed hold lamp signal	The NC outputs this signal during stop state that was entered by feed hold.	
(46)	Manual continuous feed	(1) Jog feed Jog feed rate can be changed in 24 steps by a rotary switch. The ratio of the 24 steps is geometric progression. (2) Manual rapid traverse Rapid traverse is also effected by manual operation. Override can be applied to the manual rapid traverse rate set by parameter. However, rapid traverse override is optional. The manual continuous feed can be effected to any two axes simultaneously.	IV 4.4.1
(47)	Incremental feed	Positioning in incremental amounts can be performed, permitting high-efficiency manual positioning. The incremental feed can be effected to any two axes simultaneously. Incremental amounts: 0.001, 0.01, 0.1, 1, 10 and 100 mm (Input in metric) 0.0001, 0.001, 0.01, 0.1, 1 and 10 inch (Input in inch)	IV 4.4.3
(48)	Origin button	This creates the work coordinate system such that the position of the tool in the axis selected by the MDI keyboard becomes the zero point of that axis.	
(49)	Sequence number search	A sequence number in the program concerned is searched for, by using the MDI & CRT panel.	IV 5.26
(50)	Program number search	A 4-digit program number following address O is searched for by using MDI & CRT panel.	IV 5.16
(51)	Backlash compensation	This function compensates lost motion inherent in a machine system. The compensation amount can be set by parameters in the range from 0 to 255 pulses in least command increment.	
(52)	Key lock of programs	This function can inhibit display, setting, or edition of programs No. 9000 to 9899 by key locking.	Appendix. 11
(53)	Weight	Free standing type : Max. 300 kg. Built-in type 1 : Max. 120 kg. Built-in type 2 : Max. 200 kg. Unbundled type : 55 kg. Above values are subject to change due to options and motors equipped.	

No.	Item	Specification	Reference Item
(54)	Self Diagnostic function	(i) Servo system (a) When the error of the error register is greater than the setting in a stopped state, the alarm is generated. (b) When the value of the error register is greater than the maximum setting, the alarm is generated. (c) When the position detecting system fails, the alarm is generated. (d) When the voltage command drift is too great, the alarm is generated. (e) When the DC motor is overheated, the alarm is generated. (f) When the velocity control unit fails, the alarm is generated. (ii) NC (a) When the bubble memory fails, the alarm is generated. (b) When ROM, RAM memory fail, the alarm is generated. (c) When the microprocessor fails, the alarm is generated. (d) When the NC is overhead, the alarm is generated. (iii) State display (a) The state of the NC is displayed on the CRT. (b) The state of I/O signals is displayed on the CRT.	
(55)	Environmental conditions	(1) Ambient temperature During operation: 32 ~ 113°F (0 ~ 45°C) During housing or transportation –4 ~ 140°F (–20 ~ 60°C) (2) Temperature fluctuations Max. 2.0°F/min. (Max. 1.1°C/min.) (3) Humidity Usually: 75% or less (Relative humidity) Short period of time: Max. 95% (4) Vibration During operation: 0.5G or less (5) Atmosphere When NC equipment is used in an atmosphere with a relatively high density of dust, cutting oil, water, or organic solvents, please contact the NC manufacturer.	

2. BASIC OPTIONS

<table>
<tr><th>No.</th><th>Item</th><th>Specification</th><th>Reference Item</th></tr>
<tr><td>(1)</td><td>MDI & CRT unit</td><td>A MDI & CRT unit is built-in the NC or is mounted on the machine.</td><td>IV 5</td></tr>
<tr><td>(2)</td><td>Connection unit</td><td>An I/O interface circuit between the NC and the machine tool can be mounted either inside the NC or on the machine side.</td><td></td></tr>
<tr><td>(3)</td><td>Part program storage & editing</td><td>
<table>
<tr><th>Name</th><th>Memory capacity</th></tr>
<tr><td>Part program storage & editing A</td><td>20 m* **</td></tr>
<tr><td>Part program storage & editing B</td><td>40 m</td></tr>
<tr><td>Part program storage & editing C</td><td>80 m</td></tr>
<tr><td>Part program storage & editing D</td><td>320 m</td></tr>
<tr><td>Part program storage & editing E</td><td>640 m</td></tr>
<tr><td>Part program storage & editing F</td><td>1280 m</td></tr>
</table>

* This capacity is reduced by 5m when optional stored pitch error compensation function or menu switch is equipped.

** The capacity is reduced by 1m with custom macro option.</td><td>IV 5.32</td></tr>
<tr><td>(4)</td><td>Tape reader</td><td>Tape reader is used for input the NC tape data to NC equipment.

(1) Tape reader without reels

(a) Reading speed:

300 ch/sec. (60 Hz)

250 ch/sec. (50 Hz)

(b) Reading method:

Photoelectric (light emitting diode)

(c) Tape box capacity:

30m (Free standing type and built-in type 2)

10m (Built-in type 1)

(2) Tape reader with reels

(a) Reading speed:

300 ch/sec. (60 Hz)

250 ch/sec. (50 Hz)

(b) Rewinding speed:

600 ch/sec. (60 Hz)

500 ch/sec. (50 Hz)

(c) Reading method:

Photoelectric (light emitting diode)

(d) Reel capacity: 150m long in tape length (for 0.108mm tape thickness and 187mm reel dia.)

(e) Tape box capacity 30m (Free standing type and built-in type 2 only)

(f) Tape rewind:

Tape is automatically rewond by a signal from the machine side until % (ISO code) or ER (EIA code).</td><td>IV 3</td></tr>
</table>

No.	Item	Specification	Reference Item
(5)	Input unit	The four kinds of input unit are provided depending on the type, provision of input circuit for servo and the capacity of input fuse for servo.	

22

3. ADDITIONAL OPTION

No.	Item	Specification	Reference Item
(1)	Simultaneous 3 axes control	Simultaneous 3 axes positioning and linear interpolation is possible.	
(2)	Simultaneous 4 axes control	Simultaneous 4 axes control can be able. 4 axes option or 5 axes option is required.	
(3)	4 axes control	An additional axis can be controlled in addition to X, Y, Z axes. The 4th axis address is selected among A, B, C, U, V or W. Whether the 4th axis is linear or rotary is selected by parameter.	
(4)	5 axes control	The 5th axis can be controlled. The 5th axis address is selected among U, V, W, A, B and C. Whether the 5th axis linear or rotary is selected by parameter.	
(5)	Additional axis simultaneous control	Simultaneous 2 axis control between one of X, Y and Z axes and an additional axis is available. Also, when another simultaneous 3 axes control option is provided, simultaneous 3 axes control among two of X, Y and Z axes and an additional axis is possible.	
(6)	S & T function (BCD 2-digit)	When 2-digit numerical command following the address S or T is specified, a 2-digit BCD signal is transmitted being separated from the other codes and the commanded code remains effective until another S or T code is commanded.	III 8.1, 8.2
(7)	4-digit S code A (12 bit binary signal output) / (analog output).	A 12 bit binary signal or an analog signal corresponding to a spindle speed is output to the machine side. Analog voltage: Max. ±10V, 20mA The spindle speed (rpm) is directly specified by a 4-digit S code. Spindle speed override can be applied by contact signals from the machine side at the following values: 50, 60, 70, 80, 90, 100, 110 and 120%.	III 8.1
(8)	4-digit S code B (12-bit binary signal output)/ (analog output)	When the main spindle speed (rpm) is specified directly by S 4-digit code, the voltage which corresponds to the spindle speed is output depending on the gear number (1 to 4) that is selected. Change of gear is supposed to be performed in power magnetics circuit, and its result should be input to the NC with the signal of GRA or GRB. As the information for decision to change the gear in power magnetics circuit, NC outputs the higher 2-digit or lower 2-digit of S4 digits of the program command with BCD.	III 8.1

No.	Item	Specification	Reference Item
(9)	Thread cutting/ Synchronous feed	It is available to provide a position coder to the main spindle, and to perform thread cutting synchronized with the pulse from the position coder.	III 4.4 III 5.8
(10)	Position coder	A position coder is connected to the spindle and generates square wave voltage signals with a frequency proportional to the spindle speed to synchronize the feed with the spindle revolution. (1) Position coder A Max. revolution per min: 4000 rpm (2) Position coder B Max. revolution per min: 6000 rpm	
(11)	Constant surface speed control	The correct surface speed (relative speed between the tool and workpiece) can be obtained by changing the spindle speed according to the tool position so that the surface speed is always equal to the value designated with the S code in the program.	III 8.2
(12)	2nd auxiliary function (3-digit B code)	When a 3-digit numerical command following address "B" is specified, 3-digit BCD signal is transmitted. The B code can be used for indexing a table.	III 8.5
(13)	T function (BCD 4-digit)	When a 4-digit numerical command following address T is specified, a 4-digit BCD signal is transmitted being separated from the other code, and the commanded code remains effective until another T code is commanded.	III 8.3
(14)	ISO code input	Both ISO (ISO 840) and EIA (RS-244-A) codes can be used for tape code. Distinction between ISO and EIA codes can automatically be made.	
(15)	Rapid traverse override.	An override is applied to the automatic or manual rapid traverse rate. (Fo, 25, 50 and 100%). "Fo" is set by parameter.	IV 4.6.13
(16)	Reference point return A	Reference point return A includes following functions. (1) Manual reference point return (2) Reference point return check (G27) (3) Automatic reference point return (G28)	III 5.9
(17)	Reference point return B	In addition to the functions of reference point return A, this function will move the machine to the 2nd reference point (G30).	III 5.9
(18)	3rd and 4th reference point return	Through performing parameter setting for the distance from the 1st reference point, it is possible to set 3rd and 4th reference points and return to them.	III 5.9

24

No.	Item	Specification	Reference Item
(19)	Stored stroke limit 1, 2	The stored stroke limit 1 stores in parameters the maximum/minimum travel in any axis beyond which the machine is forbidden to move. In stored stroke limit 2, inside or outside the area specified by parameters or program is the forbidden area. Stored stroke limit 2 can be made effective or ineffective by G codes. G22 : effective G23 : ineffective	III 5.17
(20)	Stored pitch error compensation	This is the function to compensate for the pitch error of the feed screw. The function can improve accuracy of machining and increase mechanical life. The pitch error is compensated for by storing within the NC unit the position where the compensation is made and the compensation value at the position without placing a dog on the machine.	Appendix. 9
(21)	Work coordinate system select	Corresponding to 6 G codes including G54 to G59, one of 6 work coordinate systems that have been set in advance can be selected. The program thereafter is carried out in the selected coordinate system.	III 5.14
(22)	Tool offset (G45 to G48)	By specifying G45 to G48 codes, the tool offset is performed. Tool offset increases or decreases the program values by an offset amount specified by D or H code. The number of D or H code available is 32. Max. offset value is ±999.999mm or ±99.9999 inch. G code: Function G45: Increase by set value G46: Decrease by set value G47: Double increase by set value G48: Double decrease by set value	III 6.2
(23)	Automatic coordinate system setting	The coordinate system which is in accordance with the parameter set in advance is established at manual reference point return time. Namely, it performs automatically the same thing as to command G92 at the reference point.	III 5.16
(24)	Tool length compensation (G43, G44 and G49)	By the use of G43 or G44 command, tool offset only in the Z axis (tool length compensation) can be performed. Up to 32 offset number is specified by H code. Max. offset value is ±999.999mm, or ±99.9999 inches.	III 6.1

25

No.	Item	Specification	Reference Item
(25)	Cutter compensation B, C (G40 to G42)	The cutter compensation can be effected by the G40 to G42 commands. Up to 32 offset amounts can be specified by D code. The max. offset amounts is ±999.999mm, or ±99.9999 inches. Cutting of inner corners of less than 90 degrees is impossible in cutter compensation B. Cutting of inner corners of less than 90 degrees is possible in cutter compensation C.	III 6.3
(26)	Tape input of offset data	It is possible to read a tape in the following format from a tape reader and store it in offset memory. G10P – R – ; G10P – R – ; . . . ; % P ; Offset No. R ; Offset value	IV 5.29
(27)	Tool length measurement	After letting a standard tool run against the machine fixed point by manual operation, let the tool to be measured run against the same machine fixed point. When [Z] [INPUT] is pushed, the tool length compensation value is entered as the offset value.	IV 5.14
(28)	Tool life management	The tools which can be set on the tool magazine are divided into several groups, the tool life (operation time or frequency) is set for each group. A tool is selected by designating the group No., and the operation time or frequency is counted each time the tool is used. When the set life is reached, the next tool of the same group is automatically selected.	III 11
(29)	Additional offset memory A	Total of 64 tool offset and cutter compensation can be specified.	
(30)	Additional offset memory B	Total number of tool offset and cutter compensation is increased up to 99. (Part program storage & editing B to F is required.)	
(31)	Additional offset memory C	Up to 200 pcs of tool compensation values can be used. (Part program storage & editing C to F is required.)	
(32)	F 1 digit feed	When 1 digit No. from 1 up to 9 following F is specified, it follows that the feed rate set corresponding to the No. is commanded. If F0 is commanded, it causes rapid traverse. Sending an input signal for speed change from the machine side and revolving the manual pulse generator, the feed rate of the No. that has been currently selected is increased or decreased.	III 4.5

26

No.	Item	Specification	Reference Item
(33)	Remote operation function	After positioning of X and Y axes, the remote operation signal is issued by G81 code. G80 cancels this function.	III 7.1
(34)	Canned cycle A (G80, G81, G82, G84, G85, G86, and G89)	Six types of canned cycle are available. (These include drilling, tapping and boring cycles.)	III 7.2
(35)	Canned cycle B (G73, G74, G76 and G80 to G89)	Twelve types of canned cycle are available. (These include peck drilling, fine boring, tapping and left-handed tapping cycles).	III 7.2
(36)	Inch/metric conversion (G20, G21)	With G code switching, selection between the inch and metric system for input can be made. G20 : input in inch system G21 : input in metric system	III 5.18
(37)	Circular interpolation (G02, G03)	Any arc between 0 and 360 degrees may be programmed by G02 or G03. G02 : clockwise (CW) G03 : counterclockwise (CCW)	III 5.5
(38)	Helical cutting	By commanding one axis that moves synchronously with circular interpolation to a circle arc command, helical interpolation becomes possible. Namely, machining a large diameter screw or solid cam is available by moving the tool spirally.	III 5.6
(39)	Sine curve interpolation	In helical cutting command, it is possible to conduct sine curve interpolation between two axes by effecting interpolation for the other axis (setting it as the imaginary axis) in the arc plane without moving it.	III 5.7
(40)	Circular interpolation by radius programming	In circular interpolation, the radius of an arc can be specified directly, instead of specifying it by address I, J and K. Either an arc of up to 180° or an arc more than 180° can be commanded.	III 5.5
(41)	Part program collation	Programs registered on the memory and the program on a tape are collated.	IV 5.20
(42)	External deceleration	With this function, jars to the machine caused by instantaneous stop at the stroke end can be held to the minimum and the effective stroke range is widened to the maximum. It is not available for the movement along 4th axis.	
(43)	External work No. search A	It is the function to give any program No. among 1 to 31 from outside such as the machine side or the like, and select the program in the memory of the NC.	IV 4.6.14

No.	Item	Specification	Reference Item
(44)	External data input	It sends data to the NC from the external such as the machine side or the like to perform a specified operation. In accordance with type of the external data, it includes followings: (1) External work No. search C (2) External tool compensation C (3) External alarm message (4) External operator message.	
(45)	Automatic acceleration/ deceleration for feed	By parameter setting, it is possible to apply acceleration/deceleration of the exponential function type having the time constant from 8 msec. up to 4,000 msec. on cutting feed and manual continuous feed.	III 4.6
(46)	Additional optional block skip	Write a numerical value from 1 up to 9 following/ at the start of a block. Also provide 9 optional block skip switches in the machine side. When optional block switch n is put ON, the block having/n is not executed but skipped.	III 2.8
(47)	Skip function (G31)	By the command of X, Y, Z 4th or/and 5th axes following G31, linear interpolation can be obtained as in the case of G01. If a skip signal is input during this command, the remaining of this command is cancelled and the next block is executed.	III 5.19
(48)	Restart of program	This is the function to specify a sequence No. to be restarted and restat the program from there.	IV 5.27
(49)	Single direction positioning	In order to perform precision positioning eliminating backlash, it is possible to perform positioning only from 1 direction.	III 5.3
(50)	Addition of registerable programs	96 programs are added to the standard resulting 191 programs in total.	
(51)	Scaling	Scaling by a factor between 0.001 and 99.999 times can be applied to the programmed tool path.	III 6.7
(52)	Handle interruption	When the tool is to be slightly fed in or relieved during automatic operation, this can be done without interrupting the machining by turning the manual pulse generator.	IV 4.8
(53)	Automatic corner override	When the inside of a corner is cut by applying the cutter compensation, the set override is activated automatically and machining occurs at a low speed within the region set by the parameter.	III 4.7

28

No.	Item	Specification	Reference Item
(54)	Manual arbitrary angle feed	When the angle with respect to the X-axis positive direction is set with the dial on the machine operator's panel is set and the start button is pressed, the machine moves at the jogging rate in the set direction. This operation is possible only in the XY plane, and the angle can be set between 0° and 360° in 5° increments.	IV 4.7
(55)	FANUC 3000C Format	A program created for FANUC 3000C (hereinafter referred to as F3000C) can be executed with FANUC SYSTEM 6M – MODEL B (hereinafter referred to as FS6M-B). The following option is necessary for each function: Circular cutting: Circular interpolation option Helical cutting: Helical cutting option Cutter compensation B: Cutter compensation B/C option	
(56)	Sequence No. comparison & stop	When a block of the sequence No. set in advance appears during the execution of a program, the machine stops after executing that block. The sequence No. can be set from the MDI & CRT by the operator. With this function, it is possible to stop the operation at an arbitrary point without changing the program. Therefore, this function is useful for checking the program.	IV 5.28
(57)	Run hour display	It is possible to display run hour of the NC on the CRT in the unit of sec., min. and Hr.	
(58)	Menu switch	Functions of the switches existing on the machine operator's panel can be turned on and off from the MDI & CRT. As a result, it is possible to reduce the switches on the machine operator's panel.	IV 5.34
(59)	Custom macro A, B	It is available to create specific function by the machine tool builder or the user. There are A and B due to the limit of function.	III 10
(60)	Conversational automatic programming	Milling programs such as contour program and pocket machining program and hole machining program can be easily created at the job site by using the MDI & CRT.	Separate volume B-54044E-1
(61)	Graphic display	The tool locus can be drawn on CRT screen. This facilitates checking the program.	〃

No.	Item	Specification	Reference Item
(62)	Coordinate system rotation (G68, G69)	A programmed shape can be rotated. When a workpiece is placed at a rotated position, from programmed position or when a pattern to be machined is composed of the same figures with some angles rotated, this function will be available.	III 6.8
(63)	Local coordinate system setting (G52)	Child coordinate systems, or, local coordinate systems are settable in all work coordinate systems (G54 – G59) by specifying G52.	III 5.17
(64)	Direct input of measured work zero point offset values	Relative coordinate values displayed on WORK OFFSET screen are set as work zero point offset values as they are, by simple key operation.	IV 5.35.3
(65)	Input transformer	To accept the following input power supplies: AC 200/220/230/240/380/415/440/460/480/550V (The transformer for the servo is not included.)	
(66)	Input/output	Programs, tool offset amount, parameters, etc. stored in the memory can be punched out. Selection between EIA and ISO codes is made by a setting on the MDI panel. The punched out paper tape is TV checked. ASR33, FACIT 4070 and other I/O devices with RS232C interface are available as punch units.	
(67)	FANUC CASSETTE B1/B2 (Bubble cassette)	It is possible to perform input/output of data from the bubble cassette built in with bubble memory. (1) Outline dimension : 120 x 64 x 22 (mm) (2) Weight : 235 g (3) Memory capacity : converted in tape length 80m (B1)/ 160m (B2)	IV 7
(68)	FANUC CASSETTE P1 (LSI cassette)	It is possible to input or output data from a LSI cassette built-in LSI memory. (1) External dimensions : 120 x 64 x 24 (mm) (2) Weight : 150 g (3) Memory capacity : converted to tape length 20m	IV 7
(69)	Portable tape reader	A tape reader without reels with RS232C interface is available. It is portable type and can be used for plural NCs by connecting to each NC.	
(70)	FANUC PPR	FANUC PPR is an I/O device of NC data and is composed of paper tape reader, tape punch and printer. Tape reader reading speed: 150 characters/sec or more. Tape punch punching speed: 50 characters/sec Number of characters per line printed on printer: 40 characters (Dot impact method) Printing speed: 1.2 lines/sec.	

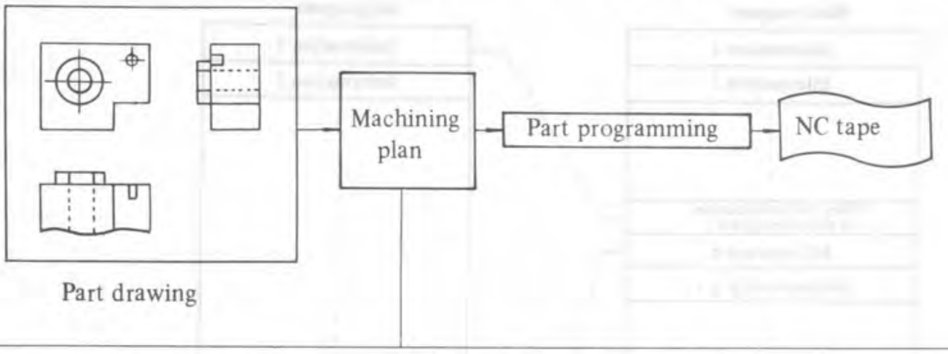
<table>
<tr><th>No.</th><th>Item</th><th>Specification</th><th>Reference Item</th></tr>
<tr><td>(71)</td><td>Door interlock</td><td>This cuts off the power supply when the door of the NC is opened.</td><td></td></tr>
<tr><td>(72)</td><td>Manual pulse generator</td><td>By the use of a manual pulse generator, fine-adjustment feed is available for a machine.
The manual pulse generator generates 100 pulses per rotation.
The move amount per pulse can be switched three levels.</td><td>IV 4.4</td></tr>
<tr><td>(73)</td><td>PC</td><td>It is available to build a programmable controller in the NC.
PC-MODEL A, B are provided.
<table>
<tr><th colspan="2"></th><th>FANUC PC MODEL A</th><th>FANUC PC MODEL B</th></tr>
<tr><td rowspan="2">Number of input point</td><td>Contact (Max.)</td><td>176 points</td><td>176 points</td></tr>
<tr><td>Voltage (Max.)</td><td>16</td><td>16</td></tr>
<tr><td rowspan="2">Number of output point</td><td>Contact (Max.)</td><td>104</td><td>104</td></tr>
<tr><td>Noncontact (Max.)</td><td>24</td><td>24</td></tr>
<tr><td colspan="2">Number of program step (Max.)</td><td>2000 steps</td><td>5000 steps</td></tr>
</table>
</td><td></td></tr>
<tr><td>(74)</td><td>Remote type position display</td><td>A position display (2 to 4 axes) in which independent display on each axis is provided can be mounted on the machine side. Display lock is made possible by mounting the switch on the machine side.</td><td>IV 6</td></tr>
<tr><td>(75)</td><td>Additional cabinet</td><td>The additional cabinet is used for housing the servo unit for the 4th axis or/and 5th axis.
(a) Capacity:
2 servo units for motor model 0M, 5M, 10M, 20M, or 30M.
(b) Size:
400mm (width) x 700mm (depth) x 1,500mm (height)
(c) Painting:
Front door P-LT-5GY3.5/0.5 (dark gray)
Others P-LT-5Y7/1 (light gray)
(d) Maintenance sides:
Front and back sides
(e) Cable inlet:
Lower parts of both sides and lower part of the rear.</td><td></td></tr>
</table>

III. PROGRAMMING

1. WHAT IS PROGRAMMING

The NC machine tool moves as directed by a program punched on an NC tape. When machining a part using an NC machine tool, the tool path and other machining conditions are programmed. This program is called a part program.

The figure below shows NC tape creation beginning from a part drawing.



1) Determination of machining range by NC and selection of NC machine tools used.
2) Method of loading workpieces on the machine tool and selection of required jigs and tools.
3) Machining sequence (classification of process, tool start point, cutting depth, tool path in rough machining and finishing).
4) Selection of cutting tools and tool holders and determination of mounting location on the machine for cutting tools and tool holders.
5) Cutting conditions (spindle speed, feedrate, coolant, etc.)

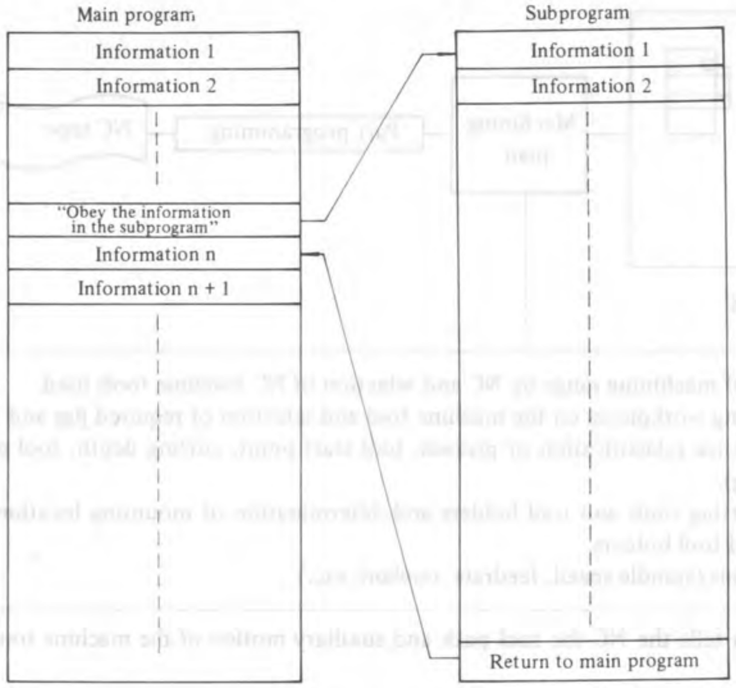
The part program tells the NC the tool path and auxiliary motion of the machine tool. This is written on the process sheet.

The program on this process sheet is punched on paper tape using a paper tape punch. This paper tape is called an NC tape and is input to the NC.

This chapter explains how to write a part programs.

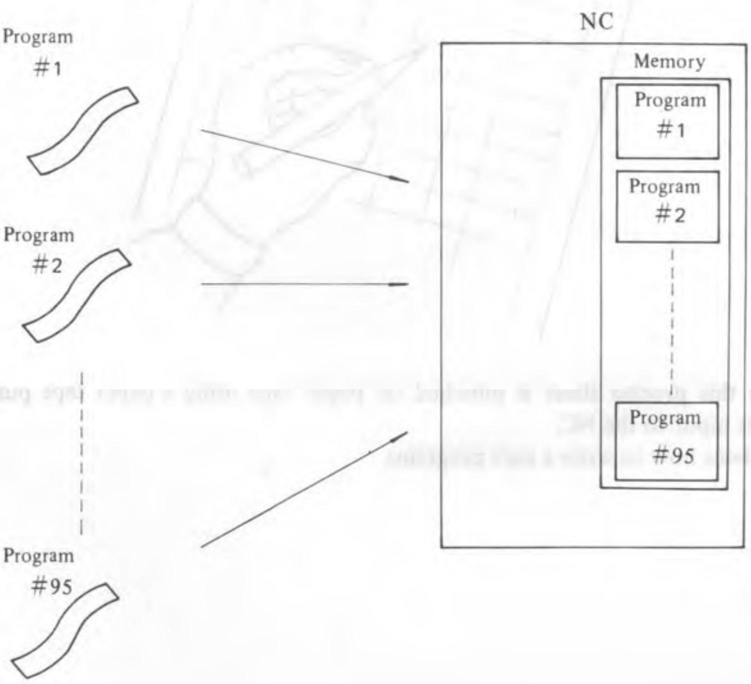
2. PROGRAM COMPOSITION

A program is divided into a main program and subprogram: Normally, the NC operates according to the main program, but when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command indicating to return to the main program is encountered in the subprogram, control is returned to the main program.



A total of 95 main programs and subprograms may be stored in the NC memory, and the NC uses one of these main programs to move the NC machine tool.

(Note 1) Up to 191 programs can be registered by selecting the Registered Programs Addition (option) and Part Program Storage & Edit C (80 meters) ~ F (1280 meters).

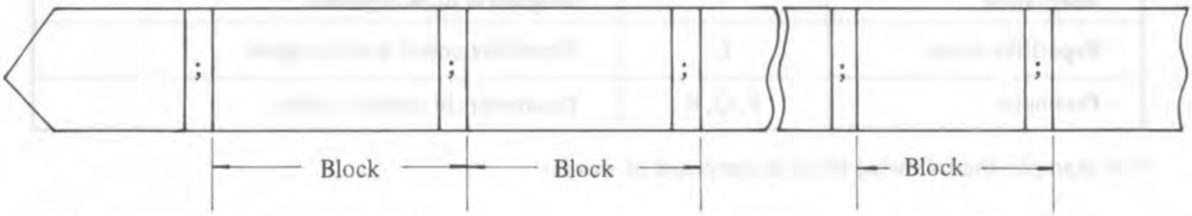


36

Refer to the section on OPERATION for how to store and select one program.

2.1 Block

A program is composed of several commands.
One command is called a block.
One block is separated from another block with an end of block code. This manual uses ; as the end of block code.



(Note 1) There is no limit to the number of characters in a block.

(Note 2) The end of block code is CR in EIA code and LF in ISO code.

2.2 Word

A block is composed of one or more words. A word is composed of an address followed by numbers as is shown below. (An algebric sign (+ or –) may be added before the numerical value.)

X – 1000

Address Numerical value

Word

The address is a letter which indicates the meaning of the numerical value following the address.
Addresses and their meanings in SYSTEM 6M are shown below.
Some addresses may vary in meaning depending on preparatory functions specified in the program.

Function	Address	Meaning
Program number	: (ISO)/O (EIA)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Motion mode (Linear, arc, etc.)
Dimension word	X, Y, Z	Coordinate axis motion command
	A, B, C, U, V, W	Additional axis motion command
	R	Arc radius, corner R
	I, J, K	Coordinate values of arc center, chamfering
Feed function	F	Feedrate, thread lead
Spindle speed function	S	Spindle speed
Tool function	T	Tool number, tool offset number
Miscellaneous function	M	ON/OFF control on the machine tool
	B	Index of table, etc.
Offset number	H, D	Designation of offset number

Function	Address	Meaning
Dwell	P, X	Dwell time
Program number designation	P	Designation of the subprogram number
Sequence number designation	P	Designation of the sequence number where the program is to be repeated.
Repetitive count	L	Repetitive count in subprogram
Parameter	P, Q, R	Parameters in canned cycles

For example, the following block is composed of words:

;	N___	G___	X___	Y___	F___	S___	T___	M___	;
	Sequence number	Preparatory function	Dimension word		Feed function	Spindle function	Tool function	Miscellaneous function	

2.3 Input Format

Each word in a block must be specified in a format as explained below. The input format used by the SYSTEM 6M is called a variable block format since the number of words in a block and the number of characters in a word can be changed. This format is very convenient for programming.

(1) Metric input

$$N04 \cdot G02 \cdot XL+053 \cdot YL+053 \cdot ZL+053 \cdot \alpha L+053 \cdot \beta L+053 \cdot \begin{Bmatrix} RD053 \\ ID053.\ JD053 \cdot KD+053 \end{Bmatrix}$$

$$\cdot F050 \cdot \begin{Bmatrix} D02 \\ H02 \end{Bmatrix} \cdot \begin{Bmatrix} S02 \\ S04 \end{Bmatrix} \cdot \begin{Bmatrix} T02 \\ T04 \end{Bmatrix} \cdot B03 \cdot M02\ ;$$

(2) Inch input

$$N04 \cdot G02 \cdot XL+044 \cdot YL+044 \cdot ZL+044 \cdot \alpha L+053 \cdot \beta L+053 \cdot \begin{Bmatrix} RD044 \\ ID044 \cdot JD044 \cdot KD044 \end{Bmatrix}$$

$$\cdot F032 \cdot \begin{Bmatrix} D02 \\ H02 \end{Bmatrix} \cdot \begin{Bmatrix} S02 \\ S04 \end{Bmatrix} \cdot \begin{Bmatrix} T02 \\ T04 \end{Bmatrix} \cdot B03 \cdot M02\ ;$$

(Note 1) α and β stand for any of A, B, C, U, V or W or additional axis.

(Note 2) The above addresses and numerical values indicate the followings:

X L + 0 4 4

- 4 (last): 4 digits to the right of the decimal point
- 4: 4 digits to the left of the decimal point
- 0: Leading zero suppression
- +: With sign
- L: Absolute or incremental
- X: Address

J D 0 4 4

- 4 (last): 4 digits to the right of the decimal point
- 4: 4 digits to the left of the decimal point
- 0: Leading zero suppression
- D: Incremental with a sign
- J: Address

Program name	Test Program 2	Note	Date '79. 10. 10	Page /
Program number	O (:) 2002		Programmer	

/	N	G	X	Y	Z	A/B/C	R/I	J	K	F	S	T	M	B	H/D	L	P	Q	;
	N20	G92	X100.0	Y200.0	Z300.0														;
	N21	G00	X196.0	Y315.0							S400	T15	M03						;
	N22	G01			Z500.0					F10.0									;

(Note) End of block code (;) is CR in EIA code or LF in ISO code.

For example, a command to move a tool 5.0125 inches along the X axis at the rapid traverse rate is as follows:

G 0 0 X 5 0 1 2 5 ;

- 4 digits to the right of the decimal point
- Although 4 digits are specified to the left of the decimal point, the leading zeros in 0005 are omitted. (Sign + is omitted)
- G0 is also possible due to leading zero suppression. (G00 indicates rapid traverse.)

(Note 3) *When 2 or more words of the same address are specified in a block, the word specified last is effective as a rule.*
No alarm is generated.

Example
G01M03S200M08; *M08 is effective, while M03 is ignored.*

- *On G codes, any G codes specified last from each group is effective.*
On G90 and G91, however, each of them is effected wherever it is specified. (See item 3.8 of OPERATOR'S MANUAL.)

Example
G90X<u>10.0</u>G91Y<u>20.0</u>;
Absolute Incremental

- *If both R and I, J, K are specified for a circular interpolation command, always address R is effective, irrespective of specified sequence.*

(Note 4) *F050 in millimeter input can be changed to F051 by switching parameters.*
See Section 4.3.

(Note 5) *The above format omits addresses P and Q because they have various meanings.*

(Note 6) *See Section 2.4 for decimal point programming.*

(Note 7) *The millimeter input values for the X, Y, Z, A, B, C, I, J, K, Q and R addresses may be multiplied by ten by setting a parameter.*

$$XL + 052 \cdot YL + 052 \cdot ZL + 052 \cdot \alpha L + 052 \cdot \begin{Bmatrix} RD052 \\ ID052 \cdot JD052 \cdot KD052 \end{Bmatrix} \qquad (\alpha = A,\ B \text{ or } C)$$

See Section 3.2.3 (Multiplication of input values by ten).

(Note 8) *See Section 3.2.4.*

2.4 Decimal Point Programming

SYSTEM 6M can input numerical values with a decimal point. However some addresses cannot use a decimal point. A decimal point may be used with mm, inches or second values. The location of decimal point is mm, inch or second.

Z15.0 Z15 millimeters or Z15 inches.
F10.0 10 mm/rev, 10 mm/min., 10 inch/rev or 10 inch/min.
G04X1. Dwell for one second
The following addresses can be used with a decimal point: X,Y,Z,A,B,C,I,J,K,R,Q,F

(Note 1) *In the dwell command, decimal point can be used with address X but not with address P.*
(This is because P is also used for a sequence number.)

(Note 2) *The appropriate G code should be specified before the numerical values are specified in one block.*
G20; (Inch dimension)
X1.0G04; . . . . . . *Because the value X1.0 is not regarded as the number of seconds, but the distance of motion (in inches), X10000G04 is assumed resulting in dwelling for 10 seconds. Indication also changes from 1.0 to 10.0 when G04 is input.*
G04X1.0 . . . . *This is regarded as G04X1000 and dwell is performed for a second.*

(Note 3) *There is a great difference in values with and without the decimal point. Programming differs from methods used in electric calculators, etc.*
G21; (millimeter dimensions)
X1. . . . . *X 1 mm*
X1 . . . . *X 0.001 mm*
G20; (inch dimensions)

40

X1. . . . . X 1 inch
X1 . . . . X 0.0001 inch

(Note 4) *Values with and without a decimal point can be specified together.*
X1000Z23.7;
X10.Y22359;

(Note 5) *Values less than the least input increment are deleted.*
When X1.23456 is specified, X1.234 is assumed in millimeter input and X1.2345 is assumed in inch input.
In incremental dimensions, errors accumulate.
In absolute dimensions, errors are not accumulated but an error deleted exists in absolute programming.
Also, the number of digits must not exceed the maximum number of digits allowed.
X1.23456789 . . This is an error because it exceeds 8 digits.
X1.2345678 . . . This is not an error because it is within 8 digits.

(Note 6) *When a number with a decimal point has been input, the number is converted into an integer of the least input increment.*

Example
X12.34 → X123400 (input in inches)
This converted integer is checked for its number of digits.

Example
X1234567.8 → X12345678000 (input in inch)
An alarm occurs because the converted integer exceeds 8 digits.

2.5 Maximum Programmable Dimensions

The maximum programmable dimensions of each address are listed in Table 2.5. Note that these figures give the maximum numerical limit, not the mechanical limit of the NC machine tool. Under NC control, the tool may traverse up to 100 m (in millimeter input) along the X axis, but actually travelling distance may be limited to 2 m for the specific machine tool. Similarly, NC control may permit a maximum cutting feed of 15 m/min., but the NC machine tool may be limited to 6 m/min. The manual issued by the machine tool manufacturer should be closely consulted, in addition to this manual when programming to ensure that the actual limitations of the specific machine tool are not exceeded.

Table 2.5 Basic Addresses and Command Range (Includes additional options)

Function	Address	Input in mm Output in mm	Input in inches Output in mm	Input in mm Output in inches	Input in inches Output in inches
Program number	:(ISO) O(EIA)	1 to 9999	Same as left	Same as left	Same as left
Sequence number	N	1 to 9999	Same as left	Same as left	Same as left
Preparatory function	G	0 to 99	Same as left	Same as left	Same as left
Dimension word	X,Y,Z, I,J,K, Q,R, A,B,C, U,V,W	±99999.999 mm ±99999.999 deg.	±3937.0078 inches ±99999.999 deg.	±99999.999 mm ±99999.999 deg.	±9999.9999 inches ±99999.999 deg.
Feed per minute	F	1 to 15000 mm/min.	0.01 to 600.00 inches/min.	1 to 15000 mm/min.	0.01 to 600.00 inches/mm
Feed per minute (Feedrate 1/10) (Parameter setting)	F	0.1 to 15000.0 mm/min.	Same as above	0.1 to 15000.0 mm/min.	Same as above
Spindle speed function	S	0 to 30000	Same as left	Same as left	Same as left
Tool function	T	0 to 9999	Same as left	Same as left	Same as left
Miscellaneous function	M	0 to 99	Same as left	Same as left	Same as left
Dwell	X,P	0 to 99999.999 second	Same as left	Same as left	Same as left
Designation of sequence number	P	1 to 9999	Same as left	Same as left	Same as left
Repetition count	L	1 to 9999	Same as left	Same as left	Same as left
Offset number	D,H	0 to 200	Same as left	Same as left	Same as left
2nd auxiliary function	B	0 to 999	Same as left	Same as left	Same as left

2.6 Program Number

The controller can store programs in it's memory. The program number is used to differentiate one program from another.

The program number is identified as follows:

$$\text{O}\underbrace{\square\ \square\ \square\ \square}_{\text{4 digits}}$$

(1 to 9999, zero cannot be used; leading zero suppression.)

A program begins with a program number and ends with M02;, M30; or M99;.

O1111 .... M02;	O2222 .... M30;
#1111 program	#2222 program

M02; or M30; means the end of a main program.
M99; means the end of a subprogram.

O5555 .... M99 ;
#5555 subprogram

(Note 1) In ISO code, colon (:) can be used instead of O.

(Note 2) A block with an optional block skip code such as/M02;,/M30; or /M99; is not regarded as the end of a program.

(Note 3) When the program number is not specified at the start of the program, the first sequence number (N. . .) in that program is regarded as the program number. However, N0 is not used as the program number.

(Note 4) When neither program number nor sequence number are specified, a program number must be specified from the MDI keyboard when the program is registered.
(See Section 5.17 in Chapter IV.)

(Note 5) When a tape contains more than one program, an EOB code is unnecessary before the second and subsequent programs. However, when a preceding program ends with ER (EIA code) or % (ISO code), an EOB code is necessary at the head of a program. This is because of the label skip feature.

(Note 6) A programmed tape operation can be performed even when using a programmed tape without a program number. Program numbers are necessary for subprograms.

(Note 7) In some cases, program numbers 9000-9899 are reserved and cannot be used by users.

(Note 8) When the robot option is used, program numbers 9900-9999 are used for robot data.

(Note 9) There is a parameter NEOP to specify that a program end is not assumed by M02, M30 or M99 but ER (EIA), % (ISO) or the next program number (O).

2.7 Sequence Number

A sequence number can be specified (1 – 9999) following address N at the head of a block. The order of sequence numbers is arbitrary and need not be consecutive. Sequence numbers can be specified in all blocks or only in blocks requiring them.

It is recommended that sequence numbers be specified sequentially at important points such as when a tool is changed and a new tool is used.

(Note 1) Sequence number N0 should not be used for compatibility with other NC equipment.

(Note 2) Because program number 0 is invalid, a sequence number regarded as a program number must not be 0.

42

2.8 Optional Block Skip

When a slash followed by a number /n (n = 1 – 9) is specified at the beginning of a block and the optional block skip switch n on the machine operator panel is set ON, the block with /n corresponding to switch number n is ignored in tape or memory operation.

When the optional block skip switch n is set OFF, the block with /n is valid. That is, the operator can skip a block with /n at his discretion. 1 in /1 may be omitted.

The following range is ignored when the optional block skip switch is ON.

```
; /2N123G01X4 . . . . . . . . . . ; N7856
|<--------- Range ignored --------->|
```

Example:

N100X100 ;
N101/2Z100 ;
N102/2/3X200 ;
N103/3Z200 ;

In the above example, the blocks of

N101 and N102 when switch No. 2 is ON
N102 and N103 when switch No. 3 is ON

are skipped.

(Note 1) *A slash (/) must be specified at the start of a block. If it is placed elsewhere in the block, the information from the slash (/) to the EOB code is ignored, while information before the slash (/) is effective.*

(Note 2) *When the optional block skip switch is ON, TH and TV checks are performed for skipped portions, as with the switch is OFF.*

(Note 3) *The optional block skip is identified when the information is read into the buffer storage from tape or memory. When a block preceded by slash has been read into the buffer, it is not ignored even if the OPTIONAL BLOCK SKIP switch is turned on.*

(Note 4) *This function is also effective during a sequence number search.*

(Note 5) *When storing the program in memory, this function is ineffective. A block with a slash (/) is also stored in the memory regardless of the position of the OPTIONAL BLOCK SKIP switch.*

(Note 6) *When punching out the program from memory, the program is punched out regardless of the position of the OPTIONAL BLOCK SKIP switch.*

(Note 7) *Some of the optional block skip switches (1 to 9) may not be used on some machine tools. Ask the machine tool manufacturer how many switches that can be used.*

(Note 8) *If more than one optional block skip code are specified in a block for the system with additional optional block skip function, 1 in /1 cannot be omitted.*
Please specify /1 under the above condition.

Example:

Error

//3 G00X10.0;

Correct

/1/3 G00X10.0;

3. DIMENSION WORD

A dimension word specifies a tool movement and is composed of the address of the axis to be moved and the value indicating the move direction and amount. The value varies depending on the absolute and incremental programming. (Refer to item 3.8)

<table>
<tr><th colspan="2">Address of dimension word</th><th>Meaning</th></tr>
<tr><td>Basic axes</td><td>X, Y, Z</td><td>Addresses of each axis in cartisian coordinate. Specifies a position of the axis or the distance along the axis.</td></tr>
<tr><td>Additional axes</td><td>A, B, C, U, V, W</td><td>Address of 4th and 5th axis. Specifies an angle of the rotary axis or a position and distance of a linear axis.</td></tr>
<tr><td rowspan="2">Parameters on circular interpolation</td><td>R</td><td>Specifies the radius of an arc.</td></tr>
<tr><td>I, J, K</td><td>Specifies a distance from start point to the center of an arc along X, Y, Z or their parallel axes.</td></tr>
</table>

3.1 Controlled Axis

Of the movement axes in the machine, an axis controlled by this NC is called a controlled axis.

Each controlled axis is called by address of the dimension word to be used in this control.

The number of axes that this NC can control is basically 3 axes and it is increased up to 4 or 5 axes by adding respective options.

An additional axis can use any of addresses A, B, C, U, V or W. It is recommended to use addresses A, B and C for rotary axes and U, V and W for liner axes.

The number of axes that can be specified in the same block is basically 2 and this is increased up to 3 or 4 by adding respective options. Additional axis is controlled singly with only simultaneous 3-axis control option. With the additional axis simultaneous control option, simultaneous 2 or 3-axis control with additional axis is possible. With simultaneous 4-axis option, no additional axis simultaneous control option is required.

<table>
<tr><th rowspan="2">No. of controlled axes</th><th colspan="5">No. of simultaneously controlled axes</th></tr>
<tr><th>Standard</th><th>Additional axis simultaneous control option</th><th>Simultaneous 3 axes option</th><th>Simultaneous 3 axes option + additional axis simultaneous control option</th><th>Simultaneous 4 axes option</th></tr>
<tr><td>3</td><td rowspan="3">2</td><td></td><td rowspan="3">3
(X, Y, Z)</td><td></td><td></td></tr>
<tr><td>4</td><td rowspan="2">2
(Includes additional axis)</td><td rowspan="2">3
(Includes additional axis)</td><td rowspan="2">4</td></tr>
<tr><td>5</td></tr>
</table>

(Note 1) If an additional axis (A, B, C, U, V, W) is specified to the system without additional axis control option, alarm No. 17 will be indicated.

(Note 2) The number of simultaneously controlled axes is always 2 in manual operation.

(Note 3) When the system is provided with 5th axis, the following limitations of functions are supposed to be taken.

(1) Threading and synchronous feed are impossible.

(2) Part program capacity 20 meters is reduced to 15 meters.

(3) S 4-digit analog output option cannot be combined.

(4) Constant surface speed control option cannot be combined.

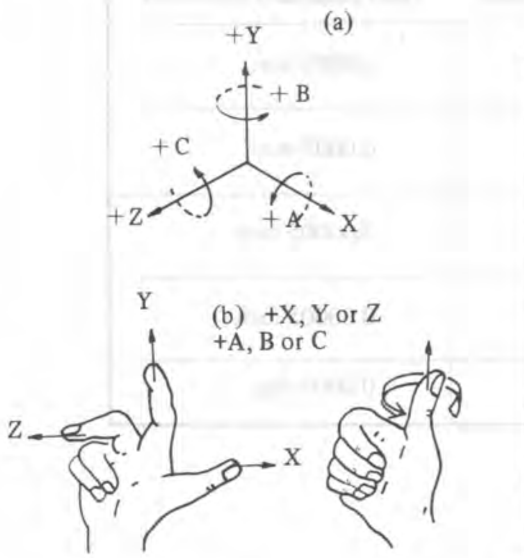
(1) Coordinate axes and symbol of motion

If machine tools provide different correlations between coordinate axes of machine tool and symbols of tool motion, severe confusion will result at programming. Basic concept on this matter is established in EIA RS-267-A or ISO 841.

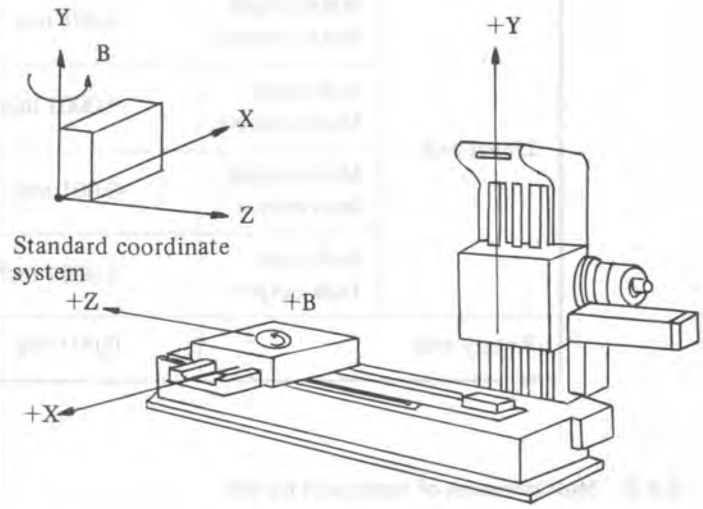
However, care must be exercised on the following points at your programming:

(a) Programs must refer to the standard coordinate system (the right-hand cartesian coordinate system).

(b) At programming, assume that the workpiece stands still but the tool is moved around the work.



The right-hand cartesian coordinate system



Coordinate axes of a horizontal boring mill

3.2 Increment System

The increment system is determined by the following two factors:

3.2.1 Least input increment and least command increment

(1) Least input increment (Input unit)

The minimum unit of tool travel input by a command tape. This is given in mm, inch or deg.

(2) Least command increment (Output unit)

The minimum unit of command input to machine tool, input in mm, inch or deg. Either one of the following combinations is used:

Input/Output		Leat input increment	Least command increment
Linear axis	mm input, mm output	0.001 mm	0.001 mm
	inch input, mm output	0.0001 inch	0.001 mm
	mm input, inch output	0.001 mm	0.0001 inch
	inch input, inch output	0.0001 inch	0.0001 inch
Rotation axis		0.001 deg.	0.001 deg.

(Note) The increment system of rotation axis is not changed by inch/metric conversion.

Whether the least command increment is 0.001 mm or 0.0001 inch is determined by the individual machine tool and must be selected by setting a parameter (SCW) in advance.

Selection of least input increment between 0.001 mm and 0.0001 inch can be made by G codes or the setting parameter input via MDI panel.

G20 . . . . Least input increment 0.0001 inch
G21 . . . . Least input increment 0.001 mm

The selection of G20 or G21 when turning power on is the same as that when the power supply was turned off.

3.2.2 Interpolation unit 1/2

Interpolation unit is halved by parameter. This function is to perform double-precision calculation of interpolation as compared with the standard system. No input units change. Output units are halved.

		Least input increment	Least command increment
Linear axis	Metric input Metric output	0.001 mm	0.0005 mm
	Inch input Metric output	0.0001 inch	0.0005 mm
	Metric input Inch output	0.001 mm	0.00005 inch
	Inch input Inch output	0.0001 inch	0.00005 inch
Rotary axis		0.001 deg	0.0005 deg

3.2.3 Multiplication of input unit by ten

The least input increment in millimeters can be changed to 0.01 mm by a parameter (MIC) setting. The least input increment in inch does not change.

	Address	Leat input increment	
		mm input	inch input
Dimension word	X,Y,Z,Q,R,I,J,K,U,V,W	0.01 mm	0.0001 inch
	A, B, C	0.01 deg.	0.01 deg.
Dwell	X	0.01 second	0.001 second
	P	0.01 second	0.001 second

The following also do not change:

(a) Input other than the above
(b) Unit of display
(c) Maximum command range
(d) Unit of step feed and handle feed
(e) Input of offset value
(f) Others

(Note 1) In the subsequence explanations of this manual, input unit are either 0.0001 inch or 0.001 mm.
(Note 2) Unit of display can be made 0.01 mm or 0.01 deg. unit by a parameter setting (MDL).

3.2.4 Input unit 1/10

The least input increment and the feedrate unit are reduced to one-tenth.

Basic Addresses and Range of Command Value (includes additional option)

Function	Address	mm input mm output	inch input mm output	mm input inch output	inch input inch output
Dimension word	X, Y, Z, I, J, K, Q, R, A, B, C, U, V, W	±9999.9999 mm ±9999.9999 deg	±393.70078 inches ±9999.9999 deg	±9999.9999 mm ±9999.9999 deg	±999.99999 inches ±9999.9999 deg
Feed per minute	F	0.1 ~ 12000.0 mm/min	0.001 ~ 470.000 inches/min	0.1 ~ 12000.0 mm/min	0.001 ~ 470.000 inches/min
Feed per minute (Feedrate 1/10) (Parameter setting)	F	0.01 ~ 12000.00 mm/min	Same as above.	0.01 ~ 12000.00 mm/min	Same as above.
Syn-chronous feed	F	0.001 ~ 99.999 mm/rev	0.00001 ~ 9.99999 inches/rev	0.001 ~ 99.999 mm/rev	0.00001 ~ 9.99999 inches/rev
Syn-chronous feed (Feedrate 1/10) (Parameter setting)	F	0.0001 ~ 99.9999 mm/rev	Same as above.	0.0001 ~ 99.9999 mm/rev	Same as above.
Dwell	X, P	0 ~ 99999.999 sec.	Same as left.	Same as left.	Same as left.

(Note) *Max. feedrate may differ from the one shown in this table depending on output unit. Dimensions of other addresses than the above addresses do not change. The following are also reduced to one-tenth.*

(1) Maximum programmable dimensions.

(2) Step feed and handle feed units.

(3) Input offset values.

3.3 Maximum Stroke

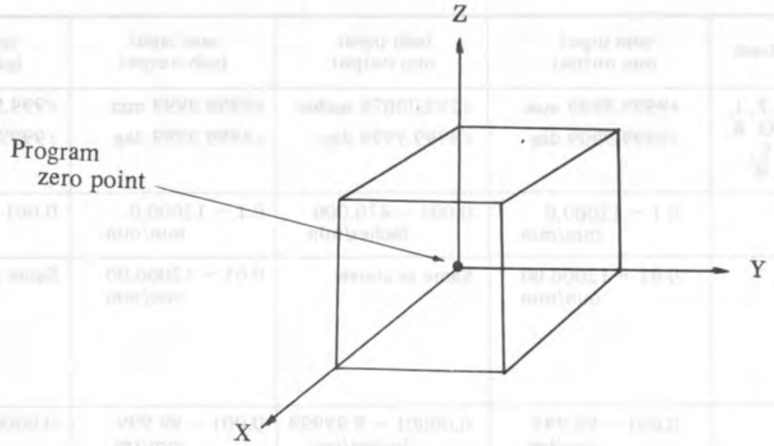
The maximum stroke commandable in this control device is listed in table below:

mm input mm output	inch input mm output	mm input inch output	inch input inch output
±99999.999 mm ±99999.999 deg.	±3937.0078 inches ±99999.999 deg.	±99999.999 mm ±99999.999 deg.	±9999.9999 inches ±99999.999 deg.

(Note 1) Above stroke, of course, varies depending on the machine tool.

3.4 Program Zero Point and Coordinate System

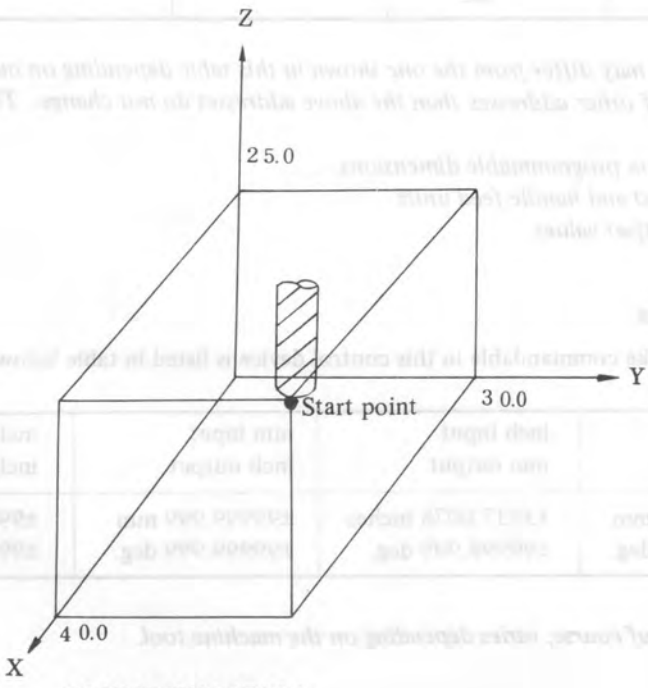
When programming, a program zero point and a coordinate system must be determined. Usually, program zero point is placed at an arbitrary position on a workpiece.



This coordinate system is called the work coordinate system.

3.5 Coordinate System and Start Point

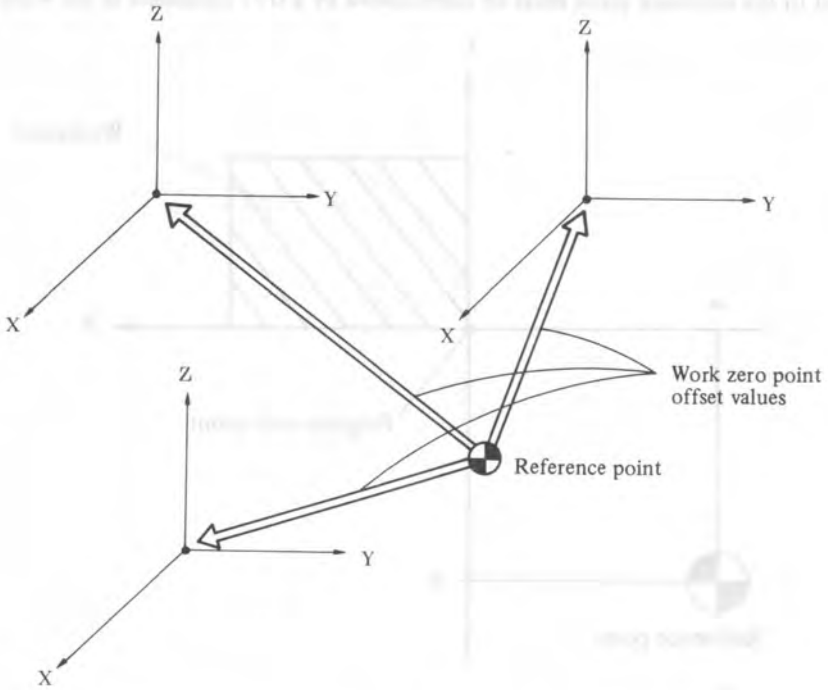
The work coordinate system used when programming must be sent to the NC. A tool moves from the start point and the program also starts from the start point. But the NC must know the coordinate value of a tool at the start point by using a G92 command.



G92X40.0Y30.0Z25.0 ;

3.6 Work Coordinate System

When pallets used differ in loading position, work coordinate systems are required. In this case six coordinate systems can be selected which are set in advance on the machine tool according to G54 to G59. Subsequento programs are executed in the selected coordinate system. The respective coordinate systems are determined by setting distances in each axis from a reference point (a fixed point on the machine tool) discussed in the following section, to their coordinate zero points (work zero point offset values).



In the use of these work coordinate systems, G92 need not set coordinate systems. Simultaneous use of G54 to G59 and G92 displaces coordinate systems by G54 to G59, so usually they are not used simultaneously.

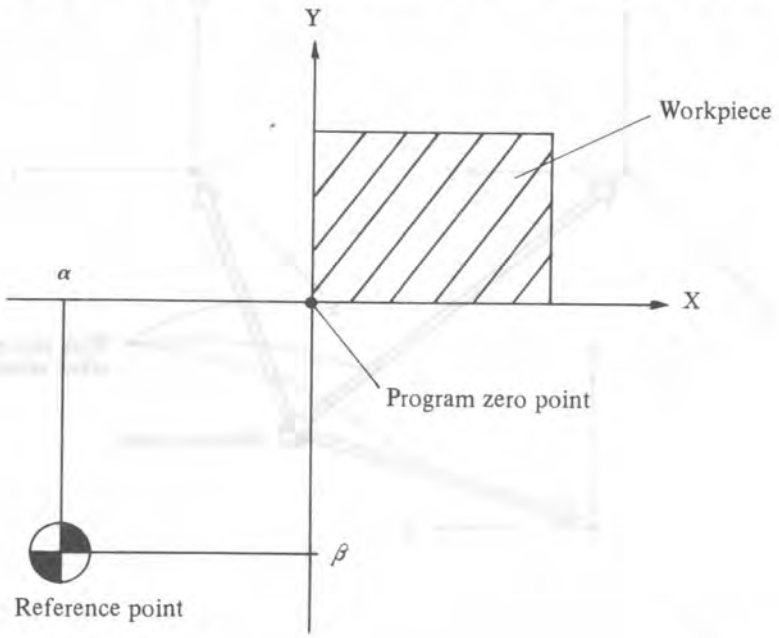
(Note 1) In use of work coordinate system by G54 to G59, first reference point return after turning power on produces a work coordinate system by G54 automatically. Therefore, automatic programming of absolute zero point is required.

49

3.7 Reference Point

The reference point is a fixed position on a machine tool. The function of reference point return will return the tool to the reference point.

Accordingly, a program may not start from a certain position in the work coordinate system but may start from a reference point. In this case, because the reference point is a certain point on the machine tool and the program is made according to the work coordinate system in which the zero point is on the workpiece, the position of the tool returned to the reference point must be commanded by a G92 command in the work coordinate system.



G92X $\underline{\quad\alpha\quad}$ Y $\underline{\quad\beta\quad}$;

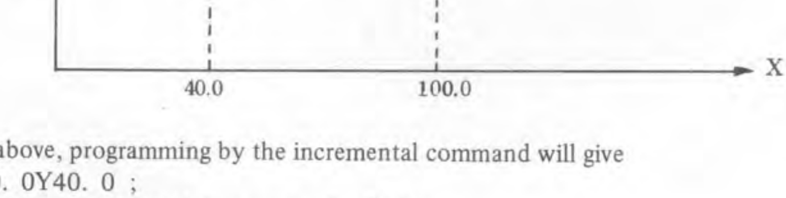
(Note 1) In use of work coordinate system by G54 to G59, G92 command is unnecessary.

3.8 Absolute and Incremental Programming

The distance of tool travel in each axis may be commanded by either an incremental command or an absolute command.

Using incremental commands (G91), travelling distance in a block is directly programmed.

Using absolute command (G90), the end location of the tool within a block is represented by coordinate values in the work coordinate system, and the coordinate values are programmed.



For the figure above, programming by the incremental command will give

G91X-60. 0Y40. 0 ;

While, programming by the absolute command will give

G90X40. 0Y70. 0 ;

The command mode, G90/G91, cannot be altered for each address within a block for the program compatibility with the other NC.

4. FEED FUNCTION

4.1 Rapid Traverse Rate

At rapid traverse, the machine moves at the specified rapid traverse rate of each axis.

Usually, this rapid traverse rate is determined and set (by parameters RPDFX to RPDF4) by the machine tool builder before shipping.

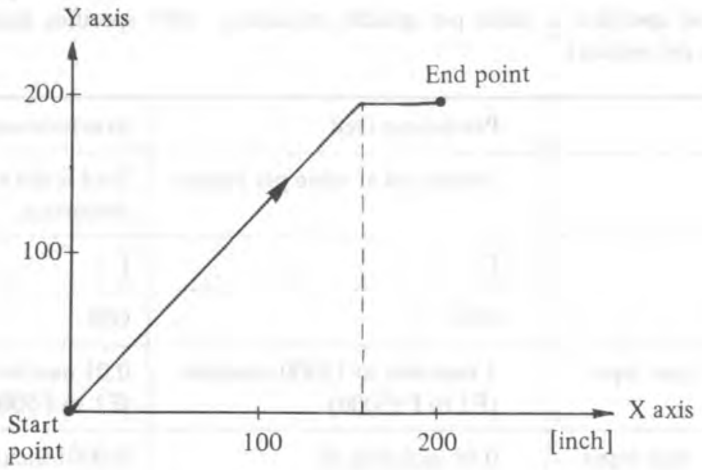
As the machine moves in each axis independently, the times in which it moves between the start and end point of the axis are not equal.

For example, when X and Y axis rapid traverse rates are 500 inch/mm and 800 inch/min respectively and the following command is programmed,

G91X200. 0Y200. 0 ;

the times in which the machine moves between start and end point of X and Y axes are 24 sec. and 15 sec. respectively.

The tool path in above example



Override can be applied to the rapid traverse rate by the switch on the machine operator's panel. (Fo, 25%, 50%, 100%) Fo is specified by parameter setting (RPDFL) and its unit are not percent (%), but mm/min or inch/min.

4.2 Cutting Feed Rate

Command a cutting feed rate in the form of required feed distance per minute. This feed rate may be commanded as follows, by using an F code:

F1 (1 mm/min, 0.01 inch/min)
to
F15000 (15000 mm/min) or F60000 (600.00 inch/min).

This feed rate is clamped to the upper limit value.

The upper limit value is set as the parameter (FEDMX), by the machine tool builder. This feed rate can be overridden by a switch located on the operator's panel, from 0 to 200% (step 10%). The upper limit value also clamps the overridden feed rate. The designation of feed rate by F code is also used for the feed rate of the rotational axis.

Metric input F050
Inch input F032

A decimal point is allowed in deg/min position both metric input and inch input.

(Example)

Metric input	F12	12 deg/min
Inch input	F12	0.12 deg/min
Metric input	F12.0	12 deg/min
Inch input	F12.0	12 deg/min

(Note 1) Except in the course of acceleration or deceleration, accuracy of NC operation for commanded feed rate is retained within ±2% of the commanded rate.

(Note 2) The max. number of digits in an F code is seven. If a value larger than the upper limit of the feed rate is input, the feed rate is clamped to the upper limit feed rate during the move command.*

4.3 Feed Rate Reduction 1/10

By parameter setting (FMIC), the metric feed rate input can be changed to one tenth of the usual increment.

Item	Least input increment	Range
Feed per minute	0.1 mm/min	F1 to F150,000 (0.1 to 15000.0 mm/min)

4.4 Synchronous Feed (Per-Revolution Feed)

The feed rate can be specified in value per spindle revolution. G95 specifies synchronous feed; G96 per-minute feed (feed at value per minute).

		Per-minute feed	Synchronous feed
Meaning		Tool is fed at value per minute.	Tool is fed at value per spindle revolution.
Address		F	F
G code		G94	G95
Range	mm input	1 mm/min to 15000 mm/min (F1 to F15000)	0.01 mm/rev to 500.00 mm/rev (F1 to F50000)
	inch input	0.01 inch/min to 600.00 inch/min (F1 to F60000)	0.0001 inch/rev to 50.0000 inch/rev (F1 to F500000)
Clamped value		Clamp is made at a certain feed rate in common to per-minute feed and synchronous feed. (Feed rate applied with override is clamped)	
Override		0 to 200% (in increments of 10%) of override is effected in both per-minute feed and synchronous feed.	

Clamped value is set in mm/min or in inch/min. Convert synchronous feed rate to mm/min or inch/min by the following equation.

fm = fr x R,

where fm: per-minute feed rate in min/min or inch/min
fr: synchronous feed rate in mm/rev or inch/rev
R: number of spindle revolutions in rpm

(Note 1) G94 and G95 are modal and once specified, are each effective until the other G code appears.

(Note 2) Synchronous feed requires the spindle to be mounted with the position coder.

(Note 3) When the number of revolutions of the position coder is up to 1 rpm, the feed rate becomes irregular. In machining not affected by the irregularity, the number of revolutions up to 1 rpm is usable. The extent of the irregularity cannot be generalized, but it becomes worse according as the revolution becomes slower.

4.5 One-Digit F Code Feed

Specifying a one-digit number (1 to 9) following F produces a feed rate set correspondingly to that number. The feed rate is set in advance as a parameter for each number. F0 produces rapid traverse speed.

Rotating the manual pulse generator with the F 1-digit feed rate change switch on the machine operator's panel ON, increases or decreases the feed rate for the currently selected number.

Increase or decrease of feed rate $\triangle F = \frac{Fmax\ i}{100X}$ per scale of manual pulse generator,

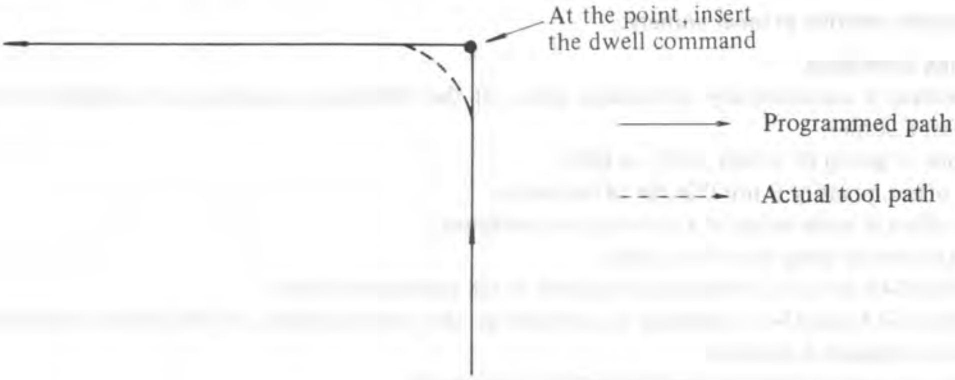
where,
- Fmax 1: feed rate upper limit for F1-F4 (parameter setting)
- Fmax 2: feed rate upper limit for F5-F9 (parameter setting)
- X: any value of 1-127 (parameter setting)

The feed rate set or altered is kept even while the power is off. The current feed rate is displayed on the CRT.

4.6 Automatic Acceleration and Deceleration

In feed start of feed stop, the acceleration or deceleration is applied automatically with a time constant so that the machine tool system is not jarred. Therefore, this need not be considered when programming.

Because of automatic acceleration and deceleration, corners are not cut sharply. The dwell command (G04) must be inserted between the two blocks to cut a sharp corner.



If the dwell command is inserted, the actual tool path matches the programmed path. The faster the feedrate and the larger the acceleration/deceleration time constant, the larger the error at the corner.

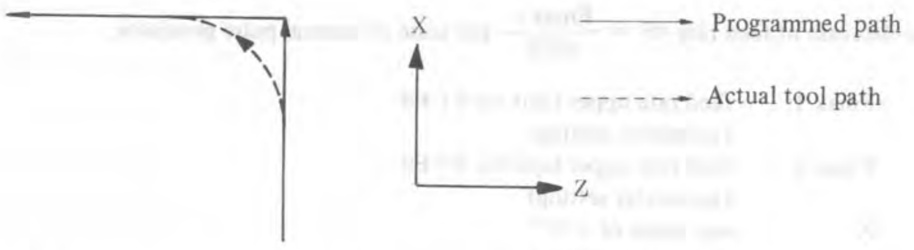
(Note 1) The following chart shows feedrate changes between blocks of information specifying different types of movement.

New block \ Previous block	Positioning	Feed	Not moving
Positioning	X	X	X
Feed	X	O	X
Not moving	X	X	X

X: The next block is executed after commanded rate has decelerated to zero.
O: The next block is executed sequentially so that the feedrate is not changed by very much.

53

(Note 2) The actual tool path does not match the programmed path because acceleration and deceleration are applied along each axis (X and Z) independently and the feedrate for each axis is changed between blocks. For example, if the tool moves along the X axis only in one block and along the Z axis in the next block, the feedrate for the X axis decelerates while motion along the Z axis accelerates and the actual tool path is as follows.



In circular interpolation, the actual arc radius is smaller than that of the programmed arc. (See the appendix.) This error can be minimized by making the acceleration/deceleration time constant of feedrate small.

4.7 Automatic Corner Override

If cutting is made at a programmed feedrate at inner corners and inner circular region during the execution of the cutter compensation, the cutter may be overloaded to rough cut surfaces. This function automatically decelerates the tool movement to lighten the cutter load at these region so as to obtain clean cut surfaces.

4.7.1 Automatic override at inner corners

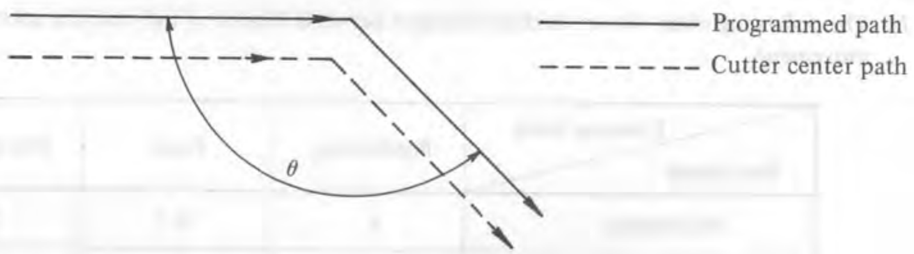
(1) Operating conditions

The feedrate is automatically overridden when all the following conditions are satisfied in both blocks before and after a corner.

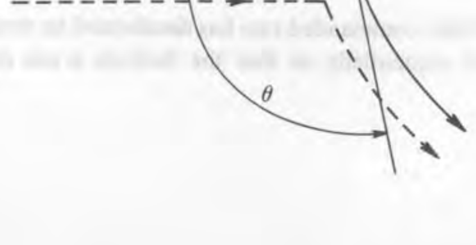
(a) G code of group 01 is G01, G02, or G03.
(b) The offset quantity is not 0 in the offset mode.
(c) The offset is made inside at a corner to be machined.
(d) An axis moves along the offset plane.
(e) Neither G41 nor G42 command is included in the subsequent block.
(f) Neither G41 nor G42 command is included in the previous block, or the block is not started up, if either command is included.
(g) An inner corner is smaller than θ preset by a parameter.

The angle is judged about the programmed path.

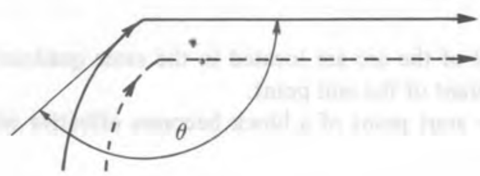
(i) Straight line – straight line



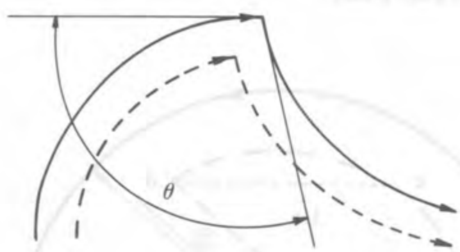
(ii) Straight line – arc



(iii) Arc – straight line



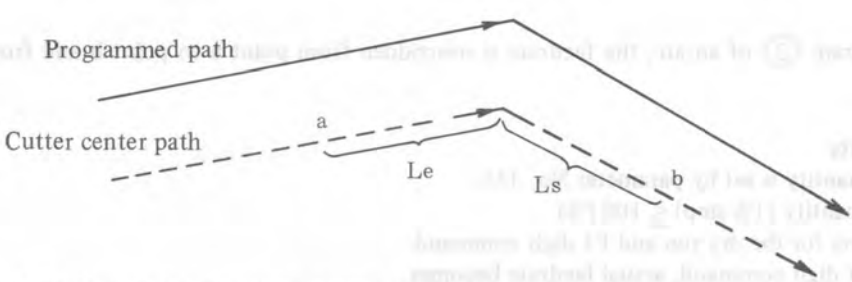
(iv) Arc – arc



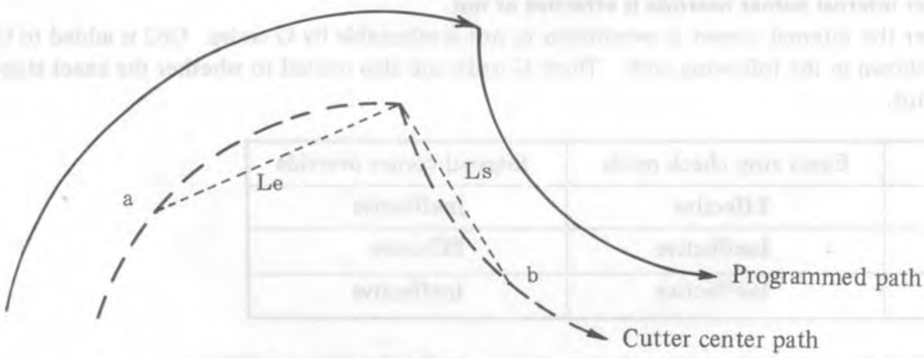
When $\theta \leqq \theta p$, the corner is regarded as inside.
θp value is set by parameter (No. 335) ($1 \leqq \theta p \leqq 179$ unit °)
If θ is almost equal to θp, the decision may include an error within 0.001°.

(2) Motion area

When a corner is judged as inside, the feedrate is overridden from the distance range within Le of the block on this side from the intersection of the corner to the distance range within Ls of the next block from the intersection of the corner. Distances Ls and Le are linear distances from a point on the cutter center path to the intersection of the corner. Le and Ls are set by parameters (No. 355 and 356), respectively.



The feedrate is overridden from point a to point b.



The feedrate is overridden from point a to b.

In case of an arc, this override function to the end point of a block is effective when the following conditions are satisfied.

(1) The distance is within Le.

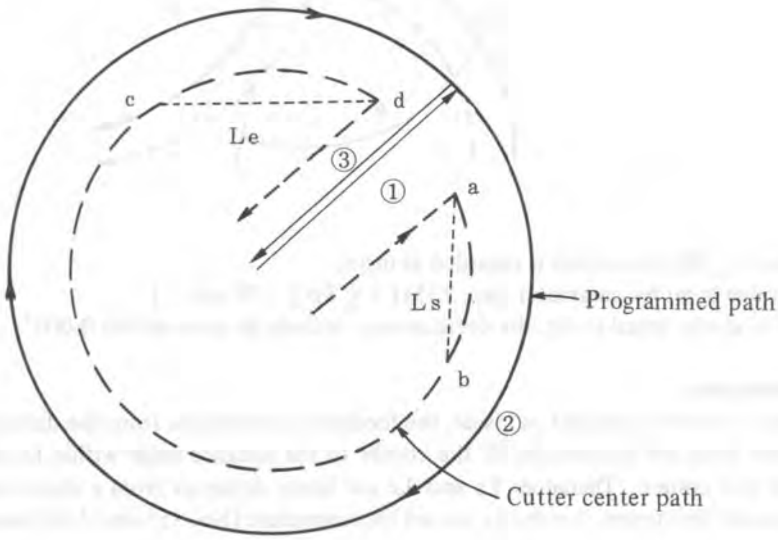
(2) The start point and end point of the arc are located in the same quadrant or the start point is located in a quadrant adjacent to the quadrant of the end point.

The override function to the start point of a block becomes effective when the following conditions are satisfied.

(1) The distance is within Ls.

(2) The end point of the arc is located in the same quadrant as the quadrant of the start point, or in a quadrant adjacent to the quadrant of the start point.

(Example) In case of disc



Regarding program ② of an arc, the feedrate is overridden from point a to point b and from point c to point d.

(3) Override quantity

The override quantity is set by parameter No. 335.

1 ≦ Override quantity (1% step) ≦ 100 (%)

It is also effective for the dry run and F1 digit command.

In case of an F4 digit command, actual feedrate becomes

F x (internal corner override) x (feedrate override)

(4) Whether internal corner override is effective or not

Whether the internal corner is overridden or not is selectable by G codes. G62 is added to G61 and G64 in group 15 as shown in the following table. These G codes are also related to whether the exact stop check mode is effective or not.

	Exact stop check mode	Internal corner override
G61	Effective	Ineffective
G62	Ineffective	Effective
G64	Ineffective	Ineffective

(Note 1) G64 functions when power is turned on or under the clear condition.

(Note 2) Specify G09, if you want to check the exact stop in G62 mode.

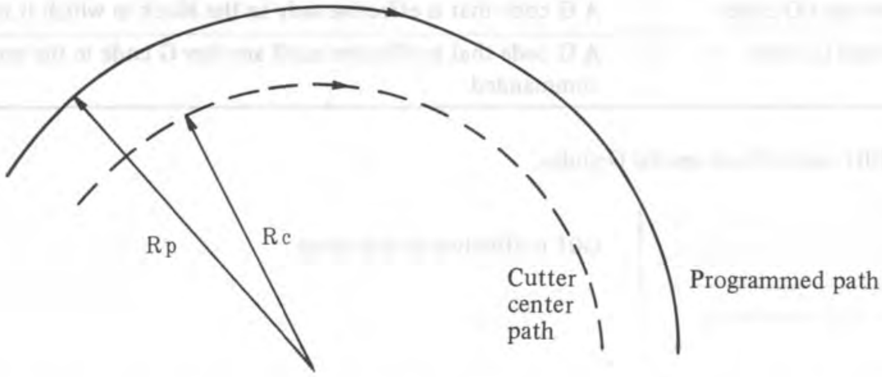
(Note 3) The internal circular cutting feedrate change described in 4.7.2 is always effective without being affected by these G codes.

4.7.2 Internal circular cutting feedrate change

In case of the internal offset circular cutting, the feedrate in the programmed path is set to the specified F by setting actual feedrate to F x Rc/Rp (where, Rc: Cutter center path radius Rp: programmed radius) with reference to the specified feedrate (F).

This change is also effective for dry run and F1 digit command.

(Example 1)



However, if Rc is very small as compared with Rp, Rc/Rp = 0, causing the cutter to stop. Accordingly, the minimum reduction ratio (MDR) is set to set the actual feedrate to F x (MDR) when Rc/Rp $\leqq$ MDR.

MDR is set by parameter No. 333.

$1 \leqq$ MDR (1% step) $\leqq 100$

This is also applicable to F1 digit and dry run.

The reduction ratio of the automatic override at internal corners is not affected by MDR.

(Note 1) *If the automatic override at internal corner overlaps the internal circular cutting, actual feedrate becomes as follows.*

F x Rc/Rp x (override at corner) x (feedrate override)

5. PREPARATORY FUNCTION (G FUNCTION)

A 2-digit number following address G determines the meaning of the commands in the blocks concerned. The G codes are divided into the following two types.

Type	Meaning
One-shot G codes	A G code that is effective only in the block in which it is specified.
Modal G codes	A G code that is effective until another G code in the same group is commanded.

(Example) G01 and G00 are modal G codes.

```
G01 X ____________ ;  }
    Y ____________ ;  }  G01 is effective in this range.
    X ____________ ;  }
G00 Y ____________ ;
```

Table 5.1 List of G codes

G code	Group	Function	Basic or option
G00	01	Positioning (Rapid traverse)	B
G01		Linear interpolation (Feed)	B
G02		Circular interpolation CW	O
G03		Circular interpolation CCW	O
G04	00	Dwell	B
G07		SIN interpolation (imaginary axis specification)	O
G09		Exact stop check	B
G10		Offset value setting, work zero point offset value setting	O
G17	02	XY plane selection	O
G18		ZX plane selection	O
G19		YZ plane selection	O
G20	06	Input in inch	O
G21		Input in mm	O
G22	04	Stored stroke limit ON	O
G23		Stored stroke limit OFF	O
G27	00	Reference point return check	O
G28		Return to reference point	O
G29		Return from reference point	O
G30		Return to 2nd, 3rd and 4th reference point	O
G31		Skip cutting	O
G33	01	Thread cutting	O
G40	07	Cutter compensation cancel	B
G41		Cutter compensation left	O
G42		Cutter compensation right	O
G43	08	Tool length offset + direction	O
G44		Tool length offset – direction	O
G49		Tool length offset cancel	O
G45	00	Tool offset increase	O
G46		Tool offset decrease	O
G47		Tool offset double increase	O
G48		Tool offset double decrease	O
G50	11	Scaling off	O
G51		Scaling on	O
G52	00	Local coordinate system setting	O

G code	Group	Function	Basic or option
G54	14	Work coordinate system 1 select	O
G55		Work coordinate system 2 select	O
G56		Work coordinate system 3 select	O
G57		Work coordinate system 4 select	O
G58		Work coordinate system 5 select	O
G59		Work coordinate system 6 select	O
G60	00	Single direction positioning	O
G61	15	Exact stop check mode	B
G62		Automatic corner override effective	O
G64		Cutting mode	B
G65	00	Custom macro simple call	O
G66	12	Custom macro modal call	O
G67		Custom macro modal call cancellation	O
G68	16	Coordinate system rotation ON	O
G69		Coordinate system rotation OFF	O
G73	09	Peck drilling cycle	O
G74		Counter tapping cycle	O
G76		Fine boring	O
G80		Canned cycle cancel	O
G81		Drilling cycle, spot boring	O
G82		Drilling cycle, counter boring	O
G83		Peck drilling cycle	O
G84		Tapping cycle	O
G85		Boring cycle	O
G86		Boring cycle	O
G87		Back boring cycle	O
G88		Boring cycle	O
G89		Boring cycle	O
G90	03	Absolute programming	B
G91		Incremental programming	B
G92	00	Programming of absolute zero point	B
G94	05	Per minute feed	B
G95		Per revolution feed	O
G96	13	Constant surface speed control	O
G97		Constant surface speed control cancel	O
G98	10	Return to initial point in canned cycle	O
G99		Return to R point in canned cycle	O

B: basic
O: option

60

(Note 1) *The G codes marked with ▼ are initial G codes in each group. That is, these G codes are set when the power is turned on or when the reset button is pressed while the system parameter that specifies to initialize G codes is effective. For G22 and G23, G22 is selected when the power is turned on. After reset, G22 or G23 (the one effective before resetting) is set.*
For G00 and G01, G43, G44 and G49, G90 and G91 or G94 and G95, the value selected for the initial G codes is specified by parameter (G00, G43, G44, G90, G95).
For G20 and G21, the one in effect before the power was cut or the reset button was pressed is selected.

(Note 2) *The G codes in the group 00 are not modal. They are effective only in the block in which they are specified.*

(Note 3) *An alarm occurs when a G code not listed in the above table is specified or when an optional G code not defined in the controller is specified (No. 010). However, G38 and G39 are ignored.*

(Note 4) *A number of G codes can be specified in a block even if they do not belong to the same group. When a number of G codes of the same group are specified, the G code specified last is effective.*

(Note 5) *If any G code in group 01 is specified in a canned cycle mode, the canned cycle is automatically cancelled and the G80 condition is entered. However G codes in group 01 are not affected by any of the canned cycle G codes.*

(Note 6) *G70 and G71 can be used by parameter setting (GSP) instead of G20 and G21 respectively (Special G code).*

(Note 7) *A G code from each group is displayed.*

5.1 Plane Selection

This command selects a plane in which the circular interpolation or the cutter compensation will be provided.

G17 .... XY plane
G18 .... ZX plane
G19 .... YZ plane

The movement command has no relation to the plane selection of G17/G18/G19. For example, when G17Z____; is specified, the Z axis is moved.

5.2 Positioning (G00)

By this code, the tool is positioned for each axis at the point programmed by the address X, Y, Z or A, B, C, U, V, W (additional axis). In absolute command, the coordinate value must be commanded, while in incremental command, the distance from the start point to the end point must be commanded. The tool path in positioning is not always linear.

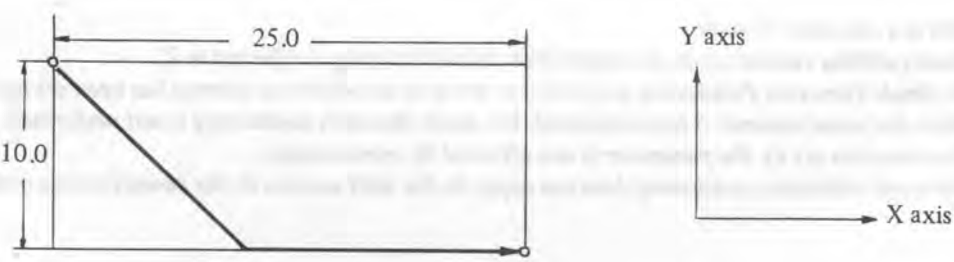
In the same block, 2 axes (2 addresses) can be programmed. (However, for 4th axis, one axis can be programmed.)

G00 specifies positioning.

G00 α____ β____ ;
(α, β = X, Y or Z)

(Example)

Rapid traverse	X axis	9,600 mm/min.
	Y axis	9,600 mm/min.
Program	G00 X25. 0 Y-10. 0 ;	



(Note 1) *The rapid traverse rate in the G00 command is set for each axis independently by the machine tool builder. Accordingly, rapid traverse rate cannot be specified by the program.*

In G00 positioning mode, the tool is accelerated to the predetermined speed at the start, moved at rapid traverse and then the tool is decelerated to the end and the execution proceeds to the next block after confirmation of the in-position (Note 2).

(Note 2) In-position means that the feed motor is within the specified range. (This range is determined by the machine tool builder.)

If simultaneously 3 axes control option is specified, a move command is as follows.

G00 X ________ Y ________ Z ________ ;

In this example, the tool is positioned to the specified position at the specified rapid traverse rate in the X, Y and Z axis simultaneously.

When the additional axis simultaneous control option is specified, instead of the X, Y or Z address the address of the additional axis (A, B, C, U, V, W) can be commanded. The tool can be controlled simultaneously in 4 axes if the option is provided.

(Example)

G00 X500. 0 Y300. 0 Z25. 0 B20. 0 ;

5.3 Single Direction Positioning (G60)

For accurate positioning without backlash, final positioning only from one direction is available.

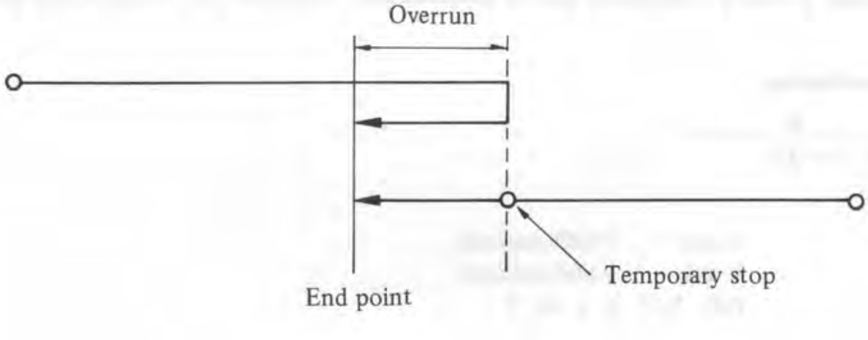
(Direction for final positioning is right to left.)

G60 is used instead of G00 as below.

G60 α ________ β ________ γ ________ δ ________ ;

(α, β, and γ = X, Y, and Z or additional axis A, B, C, U, V, or W where simultaneous 3-axis or 4-axis control and simultaneous 2 or more axis control including an additional axis are options.)

An overrun and a positioning direction are set by the parameter. Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.



(Note 1) G60 is a one-short G code.
(Note 2) During drilling canned cycle, no Single Direction Positioning is effected in Z.
(Note 3) No Single Direction Positioning is effected in the axis for which no overrun has been set by the parameter.
(Note 4) When the move distance 0 is commanded, the single direction positioning is not performed.
(Note 5) The direction set by the parameter is not affected by mirror image.
(Note 6) The single direction positioning does not apply to the shift motion in the canned cycles of G76 and G87.

62

5.4 Linear Interpolation (G01)

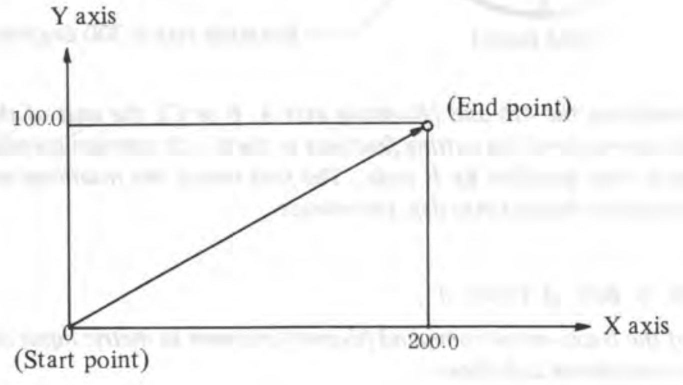
G01 α ________ β ________ F ________ ;

(α , β = X, Y, Z, A, B, C, U, V, W. The 4th axis is controlled simultaneously in one axis.)

This command actuates the linear interpolation mode. The values of α and β define the distance of tool travel which will be conducted in absolute or incremental mode, according to the current status of G90/G91. The feed rate is set to a cutting feed speed commanded by F code and is a modal data.

(Program example)

(G91) G01 X200. 0 Y100. 0 F200. 0 ;



The feed rate commanded by the F code is measured along the tool path. If it is not commanded, the feed rate is regarded as zero.

If simultaneously 3 axes control option is provided, a move command (linear interpolation) is as foolows.

G01 X ________ Y ________ Z ________ F ________ ;

By this command, simultaneously linear interpolation in 3 axes can be performed.

When additional axis simultaneous control option is provided, instead of the X, Y or Z address, the address of the 4th axis (A, B or C) can be commanded. The tool can be controlled simultaneously in 3 axes including 4th axis.

(Example)

G01 X500. 0 Y300. 0 B20. 0 F10. 0 ;

When optional simultaneous 4-axis control is equipped, the following command is allowed:

G01 α ____ β ____ γ ____ δ ____ F ____ ;

where

α , β , γ , δ = X, Y, Z, A, B, C, U, V, or W

(Note 1) The feed rate of each axis direction is as follows.

G01 α $\underline{\quad\alpha\quad}$ β $\underline{\quad\beta\quad}$ F $\underline{\quad f\quad}$;

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \cdot f$

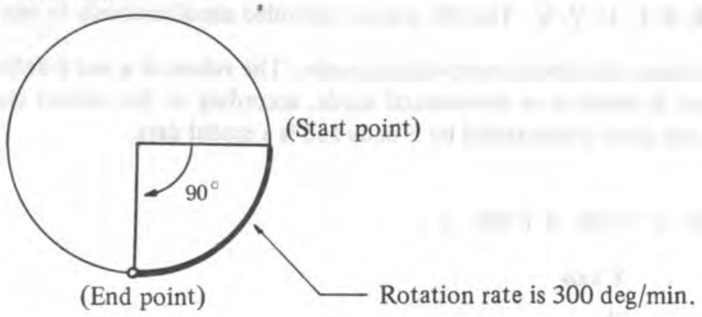
Feed rate of β axis direction : $F\beta = \frac{\beta}{L} \cdot f$

$L = \sqrt{\alpha^2 + \beta^2}$

63

(Note 2) *The feed rate of the rotational axis is commanded by deg/min (metric input: F050, inch input: F032).*

G91 G01 B-90. 0 F300. ;



(Note 3) *In interpolation including the 4th axis (Rotating axis A, B or C), the unit of the cutting feed (degree) is changed into inch (or mm) and the cutting feed rate in the α – β cartesian coordinate system is controlled so that it is equal to that specified by F code. The feed rate of the rotational axis is calculated as shown in (Note 1) and its unit is changed into deg. per minute.*

(Example)

G91 G01 X20. 0 B40. 0 F300. 0 ;

Change the unit of the B-axis move command (degree) into mm in metric input or inch in inch input. The cutting time is calculated as follows.

$$\frac{\sqrt{20^2 + 40^2}}{300} = 0.14907 \text{ (min)}$$

The feed rate for the B axis is

$$\frac{40}{0.14907} = 268.3 \text{ deg/min}$$

(Note 4) *In simultaneously 3 or 4 axes control, the feed rate is calculated the same way as in 2 axes control.*

(Note 5) *In inch input and metric output, upper limit of feed rate along the rotational axis become about 6000 deg./min. Even if the feedrate larger than the upper limit is commanded, the feedrate is clamped at the upper limit value.*

5.5 Circular Interpolation (G02, G03)

5.5.1 Circular interpolation without additional axis

The command below will move a tool along a circular arc.

Arc on X-Y plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\_\_\_ \; Y\_\_\_\_ \begin{Bmatrix} R\_\_\_\_ \\ I\_\_\_\_ \; J\_\_\_\_ \end{Bmatrix} F\_\_\_\_ \;;$$

Arc on Z-X plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\_\_\_ \; Z\_\_\_\_ \begin{Bmatrix} R\_\_\_\_ \\ I\_\_\_\_ \; K\_\_\_\_ \end{Bmatrix} F\_\_\_\_ \;;$$

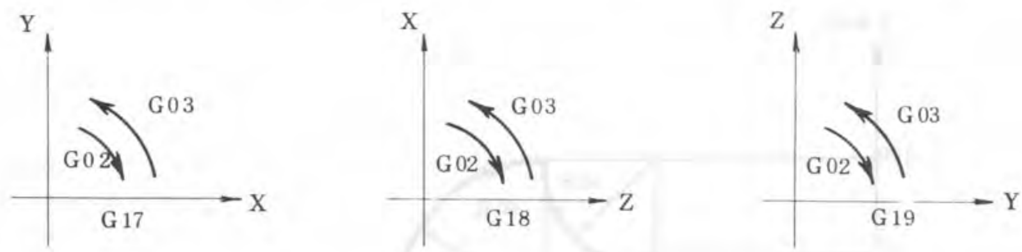
Arc on Y-Z plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_\_\_\_ \; Z\_\_\_\_ \begin{Bmatrix} R\_\_\_\_ \\ J\_\_\_\_ \; K\_\_\_\_ \end{Bmatrix} F\_\_\_\_ \;;$$

64

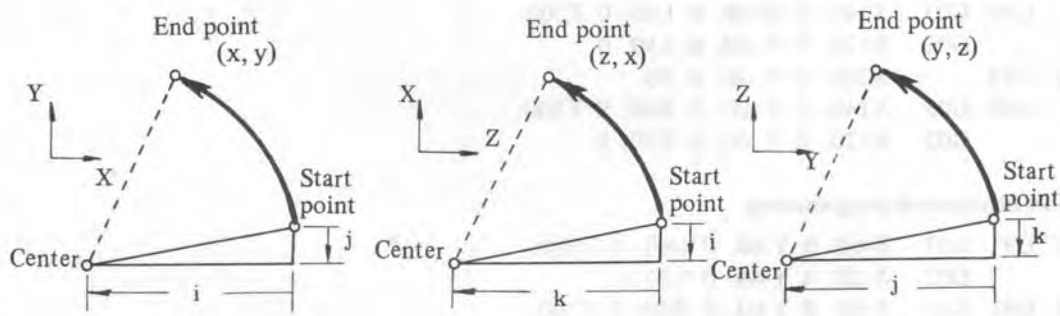
	Data to be given		Command	Meaning
1	Plane selection		G17	Arc on XY plane
			G18	Arc on ZX plane
			G19	Arc on YZ plane
2	Rotation direction		G02	Clockwise direction (CW)
			G03	Counterclockwise direction (CCW)
3	End point position	G90 mode	X, Y or Z	End point position in the work coordinate sytem.
		G91 mode	X, Y or Z	Distance from start point to end point.
4	Distance from start point to center		I, J or K	Distance from start point to center
	Radius of arc center		R	Radius of arc center.

The G17 is selected at the power on as the initial code for the plane selection.
Clockwise direction or counterclockwise direction varies with right or left hand coordinate system.



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 as an Incremental value, independently of G90 and G91.



The circular interpolation can be commanded with address R instead of address I, J or K. The command format is as follows.

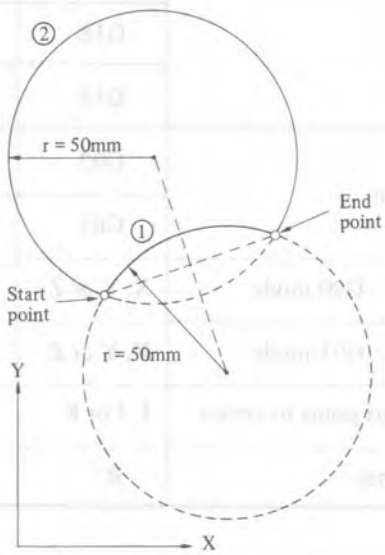
$$\left.\begin{matrix} G02 \\ G03 \end{matrix}\right\} \; X\_\_\_\_ \; Y\_\_\_\_ \; R\_\_\_\_ \; ;$$

In circular interpolation by using address R (radius designation), two types of arc, the arc less than 180°, and the other arc more than 180°, can be calculated, as shown in the figure below. In the arc more than 180°, specify the radius of the arc by negative value, as follows.

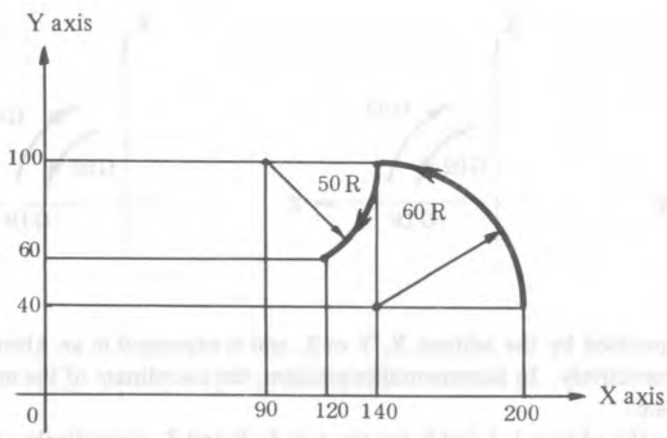
(Command example)

(1) Arc less than 180°
G02 X6. 0 Y2. 0 R5. 0 ;

(2) Arc more than 180°
G02 X6. 0 Y2. 0 R-5. 0 ;



(Program example)



(a) In absolute programming

```
(i)  G92       X200. 0 Y 40. 0 Z0                ;
     G90 G03   X140. 0 Y100. 0 I-60. 0 F300.     ;
         G02   X120. 0 Y 60. 0 I-50. 0           ;
(ii) G92       X200. 0 Y 40. 0 Z0                ;
     G90 G03   X140. 0 Y100. 0 R60. 0 F300.      ;
         G02   X120. 0 Y 60. 0 R50. 0            ;
```

(b) In incremental programming

```
(i)  G91 G03   X-60. 0 Y 60. 0 I-60. 0 F300.     ;
         G02   X-20. 0 Y-40. 0 I-50. 0           ;
(ii) G91 G03   X-60. 0 Y 60. 0 R60. 0 F300.      ;
         G02   X-20. 0 Y-40. 0 R50. 0            ;
```

The tangential feed rate in circular interpolation is equal to feed rate specified by F code. However, circular interpolation including the 4th axis cannot be performed.

(Note 1) In circular interpolation, the I0, J0 or K0 can be omitted.

(Note 2) *By omitting X, Y and Z, when the end point of an arc is equal to the start point, and commanding an arc center by I, J and K, an arc of 360° (a complete circle) is programmed.*

G02 I____ ; (A complete circle) *.

In radius programming, an arc of 0° is programmed. Namely, the cutter does not move.

G02 R____; (No movement)

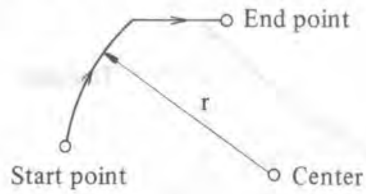
(Note 3) *If an arc of radius 0 is programmed, an alarm (No. 023) is displayed.*

(Note 4) *The error between the specified feed rate and the actual cutter feed rate is ±2% or less. In cutter radius compensation, the actual cutter feed rate is measured on the cutter center path.*

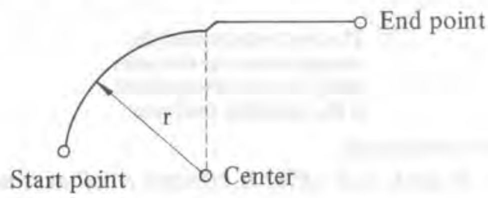
(Note 5) *If the addresses I, J, K and R are specified in the same block, the arc specified by address R is effective and the others are ignored.*

(Note 6) *When the end point is not on the arc, the tool moves as shown in the following figures.*

(i) *After a tool has reached the coordinate value of the end point on one axis, it moves to the coordinate value of the end point on the other axis.*



(ii) *When the arc does not match up with the coordinate value of the end point, the following results.*



5.5.2 Circular interpolation including additional axis

A circular interpolation including an additional axis is possible. Set to parameters which axis (X, Y or Z) is parallel with the additional axis in advance. If the additional axis is not parallel with any X, Y and Z axes, circular interpolation is impossible. Specify a plane selection G code (G17, G18 or G19) for the circular interpolation command. Specifying an axis address with a plane selection G code determines the axes on which the circular interpolation is performed.

(Example)
If additional axes U and W are parallel with X and Y axes respectively,

(i)	G17X__ Y__	XY plane
(ii)	G17U__ Y__	UY plane
(iii)	G17Y__	XY plane
(iv)	G17	XY plane
(v)	G17X__ Y__ U__	Alarm
(vi)	G18X__ W__	XW plane

Addresses I, J and K are also used to specify the circular center the same as circular interpolation not including additional axis. These addresses are employed to the axes which are parallel with X, Y and Z axes respectively.

Circular interpolation with R specification is also available.

5.6 Helical Interpolation (G02, G03)

Specifying, in addition to a circular command, a linear command for another axis moving synchronously with the circular interpolation, enables helical interpolation to be performed. That is, this command moves the tool helically.

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\_ \; Y\_\_ \begin{Bmatrix} R\_\_ \\ I\_\_ \; J\_\_ \end{Bmatrix} Z(4)\_\_ \; F\_\_ \; ;$$

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\_ \; Z\_\_ \begin{Bmatrix} R\_\_ \\ I\_\_ \; K\_\_ \end{Bmatrix} Y(4)\_\_ \; F\_\_ \; ;$$

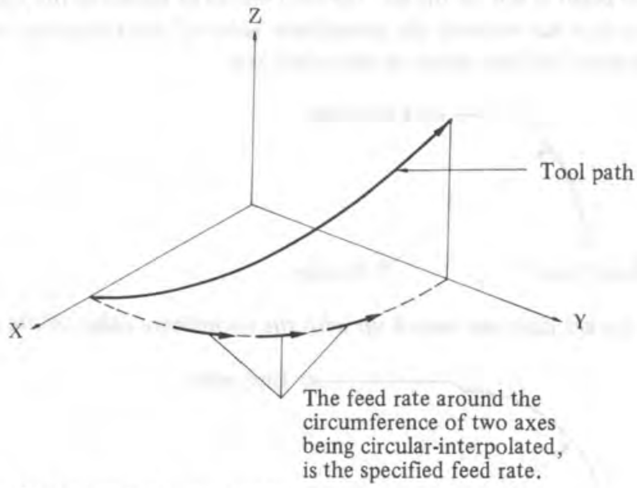
$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} \; Y\_\_\_ \; Z\_\_\_ \begin{Bmatrix} R\_\_\_ \\ J\_\_\_ \; K\_\_\_ \end{Bmatrix} X(4)\_\_\_ \; F\_\_\_ \; ;$$

The commanding method basically is only to add a move command for another axis not contained in the plane of the circular arc, to the circular interpolation command discussed previously. A one-axis command of any value can be added for a circular arc at any angle (within 360°).

An F command specifies a feed rate around a circular arc. Therefore, the linear-axis feed rate is

$$F \times \frac{\text{length of linear-axis}}{\text{length of circular arc}}$$

Determine the feed rate so that the linear-axis feed rate may not exceed any of the various limit values.



(Note 1) *Cutter compensation is effected only for a circular arc.*

(Note 2) *Tool offset cannot be used in practice. If used, tool offset is effected to all axes and I, J, and K for a circular arc as well.*

(Note 3) *Tool length offset is ignored.*

(Note 4) *An additional axis can also be specified as a linear-axis in helical interpolation. In this case, however, the additional-axis simultaneous control option is required (In case of simultaneous 3 axes option). Also this axis must be the one which does not belong to circular plane.*

5.7 SIN Interpolation

The SIN interpolation can be done by specifying one circular command axis as an exclusive interpolation axis (imaginary axis) without moving it in helical cutting command.

The imaginary axis is specified as follows.

G07 α0 ; (α is specified as a imaginary axis)

G07 α1 ; (α is specified as a real axis)

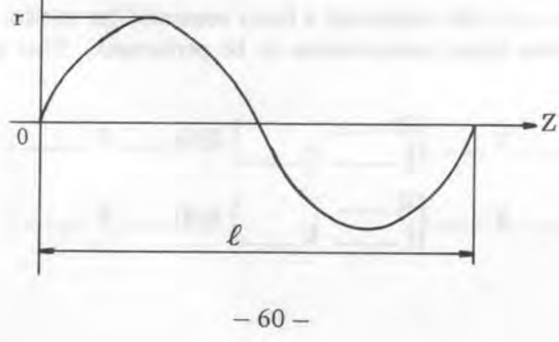
(α = X, Y, Z or additional axes (A, B, C, U, V, W))

The α axis serves as the imaginary axis after G07 α0 command until G07 α1 command is given.

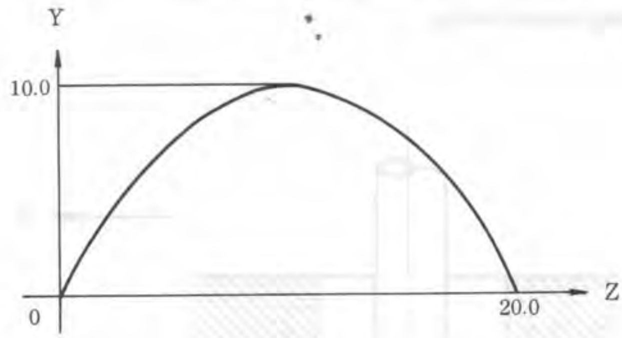
Regarding the one-cycle SIN interpolation on the YZ plane, the X axis serves as the imaginary axis.

$X^2 + Y^2 = r^2$ (r : circular radius)

$Y = r\mathrm{SIN}(\frac{2\pi}{\ell})Z$ (ℓ : one-cycle Z-axis move amount)



Program Example



N001 G07 X0 ;
N002 G91 G17 G03 X–20.0 Y0.0 I–10.0 Z20.0 F100 ;
N003 G01 X10.0 ;
N004 G07 X1 ;

The X axis serves as the imaginary axis during N002 ~ N003 blocks.

In N002 block, the helical cutting command is given in such a way as the Z axis serves as a linear axis. However, since the X axis does not move, the Y axis moves while executing the SIN interpolation to the Z axis.

In N003 block, the machine is set to the dwell condition until the interpolation ends because the X axis does not move.

(Note 1) *The imaginary axis is effective for automatic operation only. It is ineffective for manual operation, and it moves.*

(Note 2) *The interlock, stroke limit, and external deceleration are effective for the imaginary axis, too.*

(Note 3) *The handle interrupt is also effective for the imaginary axis. In other words, the axis moves by the handle interrupt component.*

5.8 Thread Cutting

Threads of a specified lead can be cut.

G33Z__z__F__f__;

where z: screw length (incremental command) or screw end point (absolute command)
f: thread lead

	Least input increment	Range
mm input	0.01 mm	F1 to F50000 (0.01 to 500.00 mm)
inch input	0.0001 inch	F1 to F500000 (0.0001 to 50.0000 inch)

The spindle speed is limited as follows:

$$1 \leq R \leq \frac{\text{maximum feed rate}}{\text{thread lead}}$$ or allowable number of position coder revolutions,

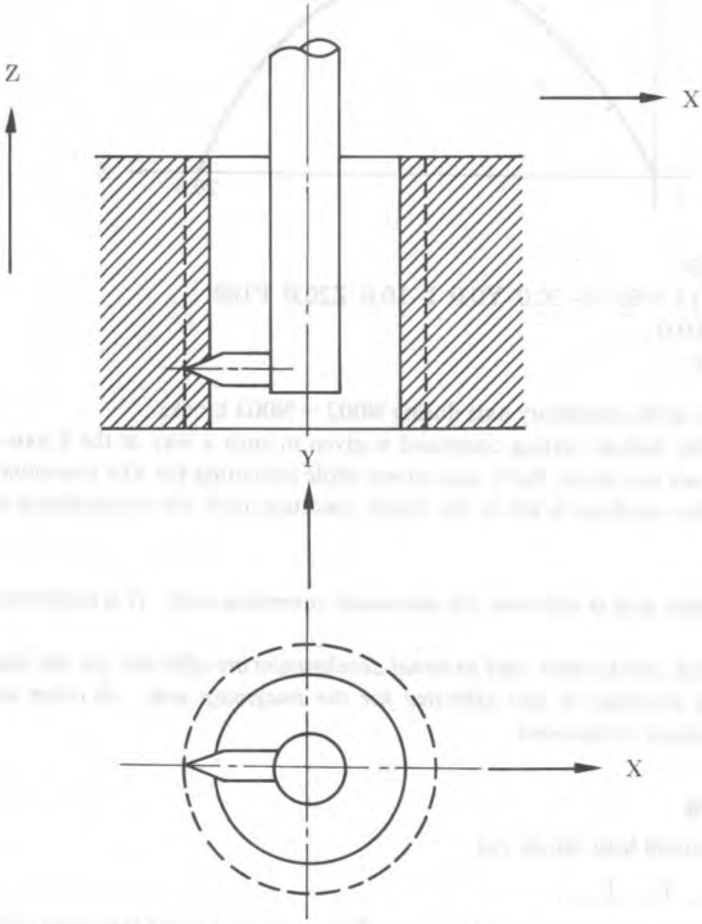
where R: spindle speed rpm
Thread lead : in mm or inch
Maximum feed rate : mm/min or inch/min
maximum command value in per-minute feed or maximum feed rate subject to restrictions by motor or machine tool, whichever is smaller.
Allowable number of position coder revolutions : 4,000 rpm (position coder A)
6,000 rpm (position coder B)

(Note 1) *Spindle speed is continuously read from the position coder mounted on the spindle and it is converted into per-minute cutting feed rates to feed the tool.*

69

(Note 2) Converted cutting feed rates are not applied with override but fixed to 100%.
(Note 3) Converted cutting feed rates are applied with cutting feed rate clamp.
(Note 4) Feed hold is ineffective during thread cutting.

(Example)




```
N20G90G00X100. 0 Y                                  S45 M03 ;
N21                     Z200.                              ;
N22G33                  Z120. 0F5. 0                       ;
N23                                                     M19 ;
N24G00     X105. 0                                         ;
N25                     Z100. 0                         M00 ;
N26        X100. 0                                      M03 ;
N27G04                       X2.0                          ;
N28G33                  Z120. 0F5. 0                       ;
```

Description

N20, N21	The tool is centered with the sub-hole. The main spindle is rotated CW.
N22	The first thread cutting is started. Lead of the thread is specified by address F.
N23	M19 commands the main spindle to stop at a fixed position on its circumference (M19: Spindle Stop at Fixed Position).
N24	Moves the tool in X-axis direction.
N25	Pulls the tool above the top of hole. M00 commands a Program Stop allowing the operator to adjust the tool, for the second thread cutting.
N26	The tool is centered with the sub-hole. The spindle starts rotation.
N27	When the move command in block N26 is short, a Dwell is commanded so that the main spindle speed may reach the rated speed.
N28	The second thread cutting is started.

5.9 Automatic Reference Point Return (G27 to G30)

5.9.1 Reference point return check (G27)

A point fixed on the machining plane is called the reference point and by reference point return function, the cutter is positioned at this point.

This function checks whether the cutter is positioned at the reference point programmed by G27 command.

G27 α______ β______ ;
(α , β : selected from among addresses X, Y, Z and additional axes A, B, C, U, V, W)

By this command, the cutter is positioned at the specified position at rapid traverse.

If the cutter is at the reference point, a reference point return lamp corresponding to the controlled axis is lit.

After returning to the reference point, the control will continue on the next block unless an M00 or M01 is inserted in the block. The optional block skip can be used if the reference point return is not needed every cycle.

If the simultaneously 3 axes control option is provided, the G27 command is as follows.

G27 α______ β______ γ______ ;
(Addresses α, β and γ is selected among X, Y, Z and additional axes A, B, C, U, V, W. However, without the simultaneously 3 axes control option, the additional axis can be controlled by itself.)

When optional simultaneous 4-axis control is equipped, the following command is allowed:

G27 α ___ β ___ γ ___ δ ___ ;

where

α , β , γ , δ = X, Y, Z, A, B, C, U, V, or W

(Note 1) In cutter compensation mode, the point where the cutter is reached is translated by the cutter radius. In this case, the cutter position is not equal to the reference point and the reference point return lamp is not lit.
Usually, G27 is commanded in the compensation cancel mode.

(Note 2) In inch machine systems with metric input, even if the programmed position of the cutter is shifted from the reference point by 1μ, the lamp will light.
This is because the least input increment is smaller than the least command increment of the machine system.

5.9.2 Automatic return to reference point (G28)

G28 α______ β______ ;
(Addresses α and β are selected among X, Y, Z and additional axes A, B, C, U, V, W. However, without the additional axis simultaneous control option, the additional axis can only be controlled by itself.)

By this command, the specified axes can be positioned to the reference point automatically. This is a move command with addresses α and β . It is specified in absolute/incremental value by the status of G90/G91.

The end point of this command is called the "Intermediate point", and the coordinate values commanded by this command are stored in the NC.

Movement in the G28 block is as follows:

All commanded axes are positioned to the intermediate point at the rapid traverse rate, and then to the reference point from the intermediate point. If the machine lock is not on, the reference point return lamp will light.

The positioning to the intermediate point and to the reference point is equivalent to positioning by G00.

When simultaneously 3 axes control is provided, the command is as follows.

G28 α______ β______ γ______ ;
(Addresses α , β and γ are selected among X, Y, Z additional axes A, B, C, U, V, W)

When optional simultaneous 4-axis control is equipped, the following command is allowed:

G28 α ___ β ___ γ ___ δ ___ ;

where

α , β , γ , δ = X, Y, Z, A, B, C, U, V, or W

In general, this command is used for automatic tool change (ATC). Therefore, in principle, it is advisable, for safety, to cancel the cutter radius compensation and tool offset before executing this command.

(Note 1) *As the coordinate value of the intermediate point, the coordinate value of the move command used in G28 is memorized. In other words, for axes not commanded by the G28 block, the coordinate value of the previous G28 is given as the coordinate value of the intermediate point for the axis.*

(Example)

N1 G90 X100.0 Y200.0 Z300.0 ;	
N2 G28 X400.0 Y500.0 ;	*Intermediate point (400.0, 500.0)*
N3 G28 Z600.0 ;	*Intermediate point (400.0, 500.0, 600.0)*

(Note 2) *When the G28 command is specified when manual return to the reference point has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference point. In this case, the direction from the intermediate point is equal to that for return to reference point selected by parameter setting.*

(Note 3) *Tool movement direction from intermediate direction to reference point is the one set by a parameter when a G28 command is specified for a rotary axis. In this case, movement amount is within 360°.*

5.9.3 Automatic return from reference point G29

G29 α ________ β ________ ;

(Addresses α and β are selected from among X, Y, Z and additional axes A, B, C, U, V, W. If the additional axis simultaneous control option is not provided the additional axis cannot be controlled simultaneously with one of the other three axes).

By this command, the tool is positioned at the commanded point via an intermediate point. In general, this command is used immediately following a G28 command.

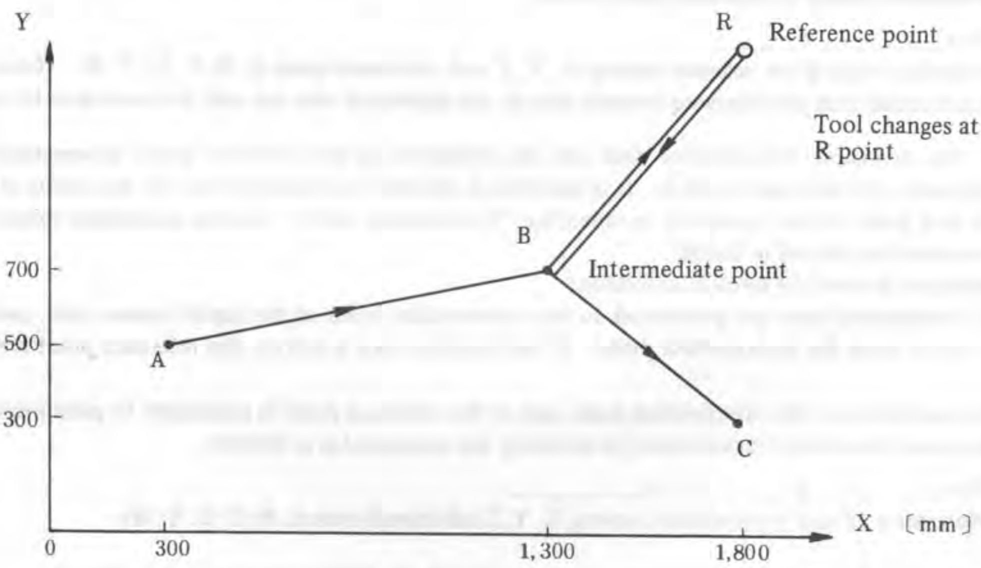
This is a move command with addresses α and β, and it is specified in absolute/incremental value depending on the status of G90/G91 at the time.

At an incremental command, an incremental distance from the intermediate point must be commanded.

With a G29 block, all commanded axes are moved to the intermediate point defined by the earlier G28 command, and then to the commanded point, at the rapid traverse speed.

The positioning operation to intermediate or commanded point is quite equivalent to that for positioning by G00.

Example of applications of G28 and G29



(G91 mode)

G28 X1000. 0 Y200. 0 ;	(Program of A to B)
M06 ;	
G29 X 500. 0 Y-400. 0 ;	Program of B to C)

92

As shown in the example, the programmer is not required to calculate the actual travelling distance from intermediate point to reference point.

If simultaneously 3 axes control option is installed, a G29 command is as follows:

G29 α________ β________ γ________ ;

(Addresses α, β and γ are selected from among X, Y, Z and additional axes A, B, C, U, V, W. If the additional axis simultaneous control option is not specified, the additional axis cannot be controlled simultaneously with one of the other three axes.)

When optional simultaneous 4-axis control is equipped, the following command is allowed:

G29 α____ β____ γ____ δ____ ;

where

$\alpha, \beta, \gamma, \delta$ = X, Y, Z, A, B, C, U, V, or W

(Note) *When the position of a work coordinate system is changed after the tool comes to the reference point through the intermediate point by G28/G30, intermediate point also shifts to a new coordinate system. Thereafter, when G29 is commanded, the tool is positioned to the specified position through the intermediate point which has shifted to the new coordinate system.*

5.9.4 2nd, 3rd and 4th reference point return (G30)

The command below moves the commanded axes to the 2nd, 3rd or 4th reference point.

$$G30 \left\{ \begin{matrix} P2 \\ P3 \\ P4 \end{matrix} \right\} \alpha\_\_\_\_ \beta\_\_\_\_ ;$$ (P2 can be omitted)

P2 : 2nd reference point
P3 : 3rd reference point
P4 : 4th reference point

The 2nd, 3rd and 4th reference points are determined by parameter setting as a specific distance from the 1st reference point as set by field adjustment. This function is the same as reference point return G28 except that the tool does not return to the first reference point but to a 2nd, 3rd or 4th point. A G29 command following a G30 command positions the tool to the commanded position via the intermediate point which was established by the G30 command. This motion is the same as that by a G29 following a G28 command.

A G30 command is generally used when the automatic tool change (ATC) position differs from the reference point.

If simultaneously 3 axes control option is provided, a G30 command is as follows.

$$G30 \left\{ \begin{matrix} P2 \\ P3 \\ P4 \end{matrix} \right\} \alpha\_\_\_\_ \beta\_\_\_\_ \gamma\_\_\_\_ ;$$

(Addresses α, β and γ are selected from among X, Y, Z and additional axes A, B, C, U, V, W. If the additional axis simultaneous control option is not specified, the 4th axis cannot be controlled simultaneously with one of the other three axes.)

When optional simultaneous 4-axis control is equipped, the following command is allowed:

$$G30 \left\{ \begin{matrix} P2 \\ P3 \\ P4 \end{matrix} \right\} \alpha\_\_\_\_ \beta\_\_\_\_ \gamma\_\_\_\_ \delta\_\_\_\_ ;$$

where

$\alpha, \beta, \gamma, \delta$ = X, Y, Z, A, B, C, U, V, or W

(Note 1) *Manual reference point return or automatic reference point return (G28) must be performed before using the G30 command.*

5.10 Dwell (G04)

G04 X $\underset{}{\overset{(t)}{\_\_\_\_\_\_\_\_}}$; or

G04 P $\underset{}{\overset{(t)}{\_\_\_\_\_\_\_\_}}$;

73

Any one of these commands may be used for dwell. Upon completion of the previous block (t) msec. time elapses before beginning the next block.

The maximum command time is 99999.999 seconds. Error of time (t) is approximately within 16 msec.

(Example) A dwell for 2.5 seconds.

G04 X2.5 ; or G04 P2500 ;

(Note 1) Address P cannot use decimal point programming.
(Note 2) Dwell starts in the following two ways.
Parameter (CINP) setting decides which is effective.
(1) After the speed of the earlier block has become zero.
(2) After the tool has arrived at the commanded value. (After in-position is checked)

5.11 Exact Stop Check (G09)

A block containing a G09 decelerates the feed rate to 0 at its end, recognizes an in-position state (Note 2), then proceeds to the next block. This function is used to develop a sharp edge at a corner, etc. G09 is effective only in the block in which it is specified.

(Note 1) In-position check is automatically effected without G09 in the positioning mode (G00, G60).
(Note 2) In-position means that the feed motor has reached within the width in which the specified end point is located.

5.12 Exact Stop Check Mode (G61) and Cutting Mode (G64)

(1) Exact stop check mode (G61)

G61, in subsequent move commands until G64 is commanded, decelerates the feed rate to 0 at the end of each block, recognizes an in-position state, then proceeds to the next block.

(2) Cutting Mode (G64)

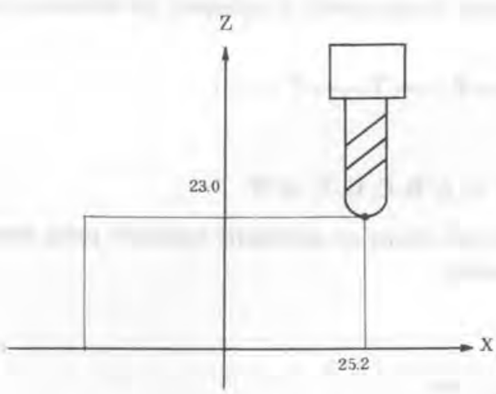
G64, in subsequent move commands until G61 is commanded, does not decelerate as much as possible the feed rate at the end of each block but shifts the block to the next at once. Even in the G64 mode, however, the feed rate is decelerated to 0 and in-position check is performed in the positioning mode (G00 or G60), in a block in which Exact Stop Check (G09) is specified, in a block whose next block contains no move command, etc.

5.13 Programming of Absolute Zero Point (G92)

When it is desired to move a tool to a certain position by absolute command, the coordinate system must be set in advance. The coordinate system is established by the following command.

G92 X $\underline{\quad (x) \quad}$ Y $\underline{\quad (y) \quad}$ Z $\underline{\quad (z) \quad}$ γ $\underline{\quad (\gamma) \quad}$ δ $\underline{\quad (\delta) \quad}$;
(γ , δ = A, B, C, U, V, W)

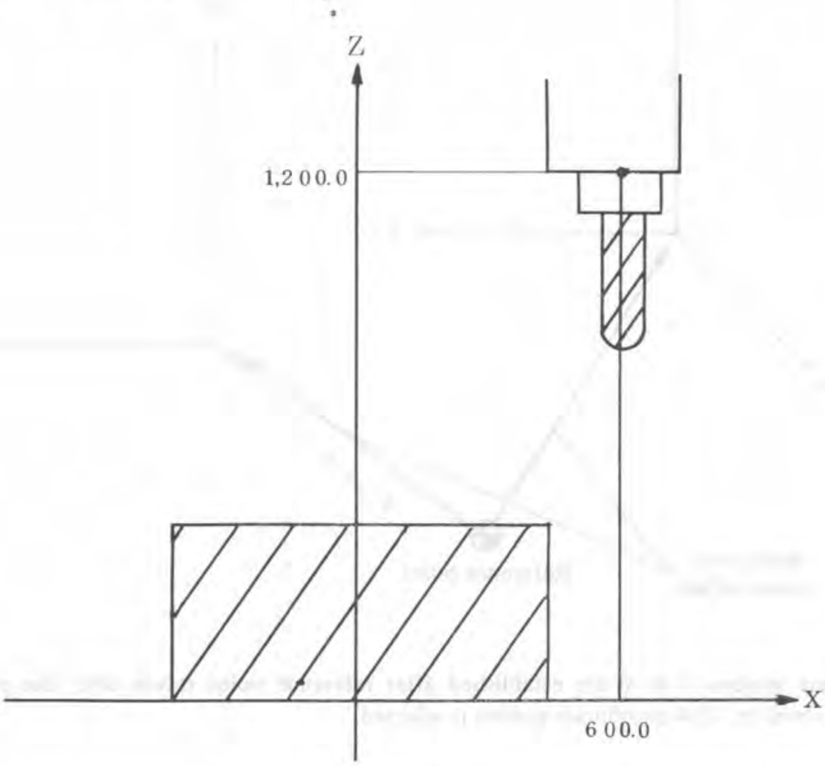
This command creates a coordinate system in which the origin is set at a specific distance from the tool position. This is called the work coordinate system. Once created, any subsequent absolute commands refer to the coordinate value in the work coordinate system.



G92 X25. 2 Z23. 0 ;

94

As shown in the above program, G92 is used to insure that the tool tip coincide with the program start point, and should be used at the start of a program.



G92 X600. 0 Z1200. 0 ;

As shown above, a G92 should be at the start of the program to make sure the tool point and the program start point coincide. If an absolute command is performed, the standard point is positioned at the specified point. In order to position a tool tip at the specified point, the distance from tool tip to the standard point must be compendated for by tool length compensation.

(Note 1) *When the coordinate system is set with a G92 command during the offset mode, the coordinate system in which coordinate value of a tool excluding the offset value, is the specified position, is established.*

(Note 2) *The cutter radius compensation mode is temporarily cancelled by a G92 command.*

5.14 Work Coordinate System

No coordinate systems are determined by G92, but six coordinate systems proper to the machine tool are set in advance, permitting selecting any of them by G54 to G59.

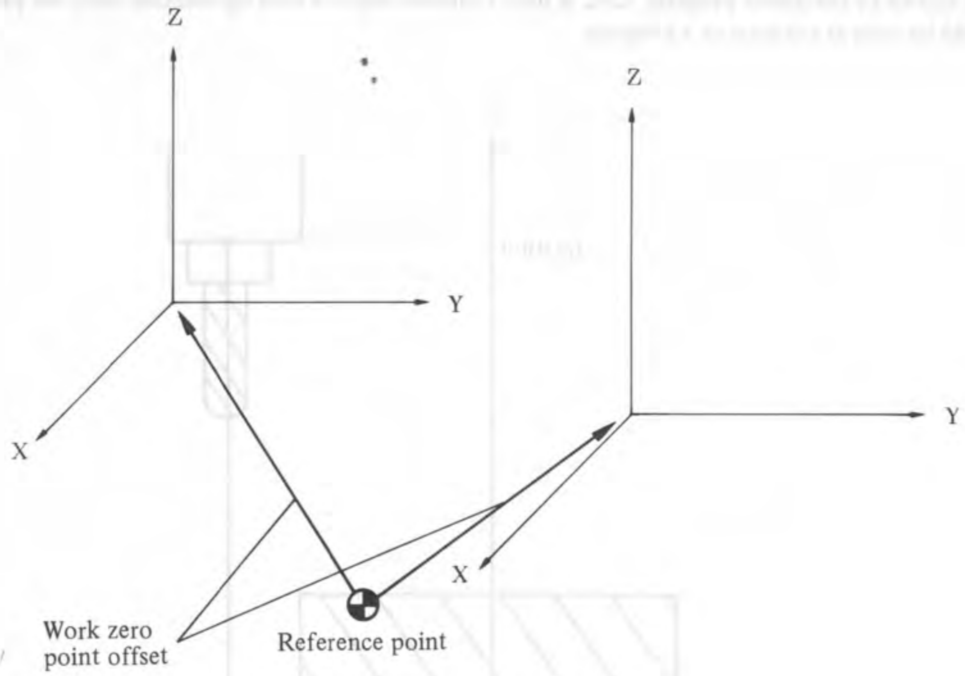
G54 .... Work coordinate system 1
G55 .... Work coordinate system 2
G56 .... Work coordinate system 3
G57 .... Work coordinate system 4
G58 .... Work coordinate system 5
G59 .... Work coordinate system 6

The six coordinate systems are determined by setting distances (work zero offset values) in each axis from the 1st reference point to their respective zero points.

Example: G55G00X100.0 Z20.0;
X 15.5Z25.5;

(Example)

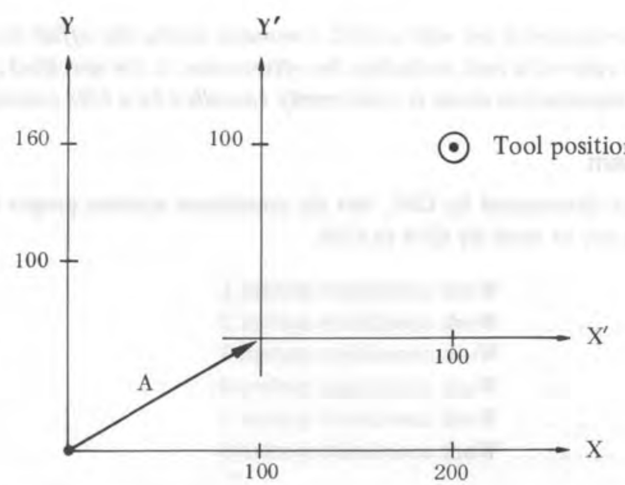
In the above example, positioning is made to positions (X = 100.0, Z = 20.0) and (X = 15.5, Z=25.5) in work coordinate system 2.



Work coordinate systems 1 to 6 are established after reference point return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

(Note 1) *External data input function (option) compensates work zero point offset values in each axis in the range of 0 to ±0.7999mm or of 0 to ±0.7999 in. Refer to the machine tool builder's manual as to whether this function is available or not.*

(Note 2) *In the use of G54 to G59, no coordinate system need be set by G92. Setting coordinate system by G92 shifts coordinate systems by G54 to G59. Do not mix G54 to G59 and G92, except particularly when coordinate systems by G54 to G59 are to be shifted.*



When the tool is positioned at (200, 160) in a G54 state, G92X100Y100; provides work coordinate system 1 (X′ – Y′) displaced by vector A. At the same time all the other work coordinate systems are also displaced by vector A.

(Note 3) *Set proper values to parameters 309APX to AP4 even when optional automatic coordinate system setting function is not equipped.*

96

5.15 Shift of Work Coordinate System by Programmed Command (G10)

When work coordinate systems are insufficient in spite of the availability of six ones by G54 to G59, or when work coordinate systems are to be moved for each program, a programmed command shifts them.

G10L2PpX____Y____Z____ γ ____ δ ____ ;

where p = 1-6 : correspond to work coordinate systems 1-6.
X, Y, Z, γ, δ (γ, δ = A, B, C, U, V, W) : work zero point offset values in each axis.

(Note 1) Putting p = 0 changes external work zero point offset values.

5.16 Automatic Coordinate System Setting

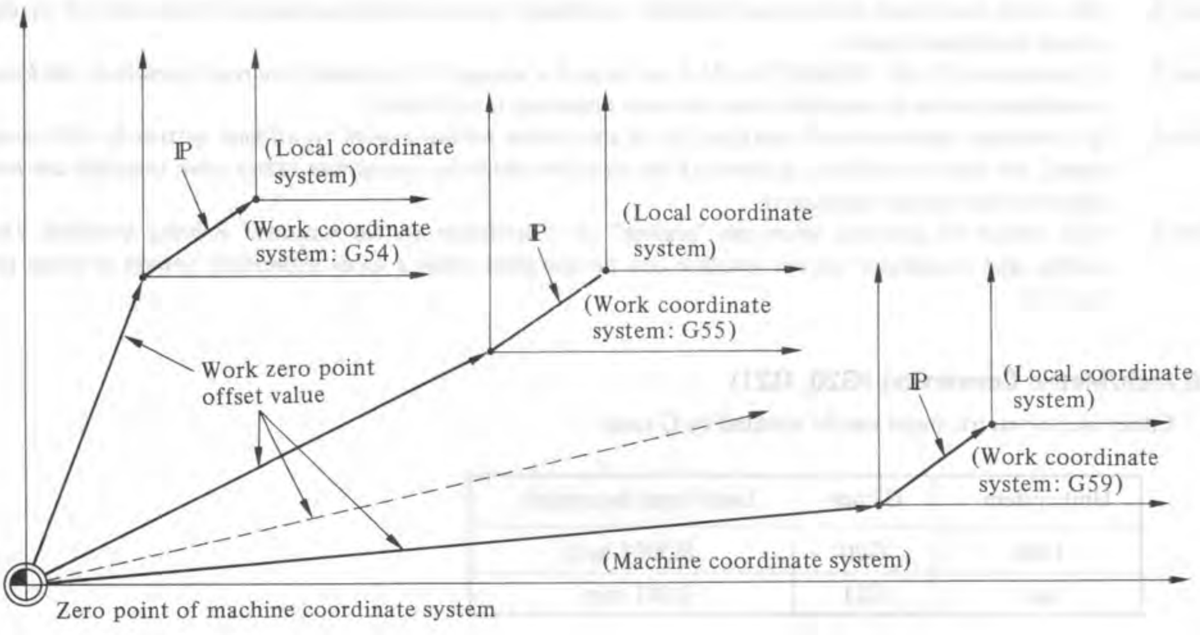
A coordinate system is established at the first reference point return time after the power is turned on, according to the preset parameter (No. 375 PPRTMX to No. 441 PPRT15). That is, the same effect as coordinate system setting using G92 at a reference point is automatically provided.

(Note 1) If work coordinate system setting is used, always set zero to parameter Nos. 375 to 378, 440 or to parameter Nos. 379 to 382, 441. If a value other than 0 is set, work coordinate system 1 to 6 will shift.

5.17 Local Coordinate System Setting (G52)

Child coordinate systems, or, local coordinate systems are settable in all work coordinate systems (G54~G59) by specifying G52 IP____; . (IP____ = X____Y____Z____ γ ____ δ ____;)

The zero points of respective local coordinate systems are equal to the IP positions in respective work coordinate systems.

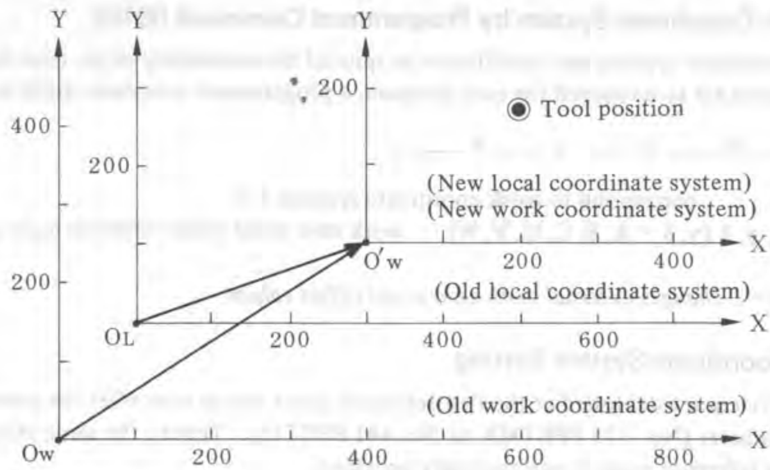


When a local coordinate system is set, the move commands specified in the absolute mode (G90), which are subsequently specified, are the coordinate values in the local coordinate system.

The local coordinate system can be changed by specifying the zero point of new local coordinate system in the work coordinate system together with G52.

For specifying the coordinate value in the work coordinate system after cancelling the local coordinate system, set the zero point of the local coordinate system to coincide with the zero point of the work coordinate system by specifying G52 IP 0;.

When a work coordinate system is set by G92 IP____; , the local coordinate system is cancelled in the same way as G52 IP 0 is specified.



If G92X200.0Y150.0; is specified when the zero point of the local coordinate system is positioned at (100, 150) in the work coordinate system and the tool is positioned at (500, 250) in the local coordinate system, zero point Ow of the work coordinate system and zero point O_L of the local coordinate system shift to the O'w position.

(Note 1) *If an axis returns to the reference point by manual reference point return, the zero point in the axis of the local coordinate system coincides with the zero point of the work coordinate system.*

G52α0;

α ; Axis which returns to the reference point.

By setting parameter (CLCL: PRM413/bit 7), the local coordinate system is not cancelled even when an axis returns to the reference point by the manual reference point return.

(Note 2) *The work coordinate system and machine coordinate system remain unchanged irrespective of setting a local coordinate system.*

(Note 3) *If parameter (CLER: PRM007/bit 3) is set in such a way as NC is cleared by a reset operation, the local coordinate system is cancelled when the reset operation is performed.*

(Note 4) *If coordinate values are not specified for all axes when setting a work coordinate system by G92 command, the local coordinate systems of the axes for which no coordinate values were specified are not cancelled but remain unchanged.*

(Note 5) *G52 cannot be specified when the "scaling" or "coordinate system rotation" is being specified. The scaling and coordinate system rotation can be specified when a local coordinate system is being set by G52.*

5.18 Inch/Metric Conversion (G20, G21)

Either inch or metric input can be selected by G code.

Unit system	G code	Least input increment
Inch	G20	0.0001 inch
mm	G21	0.001 mm

These G codes must be the only information commanded in a block and must be commanded before the work coordinate system setting block (G92) at the start of the program.

N10 G20 ;
N20 G92 X____ y____;

The following conditions change depending on these G codes.

(1) Feed rate command by F code
(2) Display of positions
(3) Offset value
(4) Unit of scale on the manual pulse generator
(5) Move distance in incremental feed
(6) Some parameters

78

(Note 1) *When the power is turned on, the NC status (of G20/G21) is the same as before the power was turned off.*

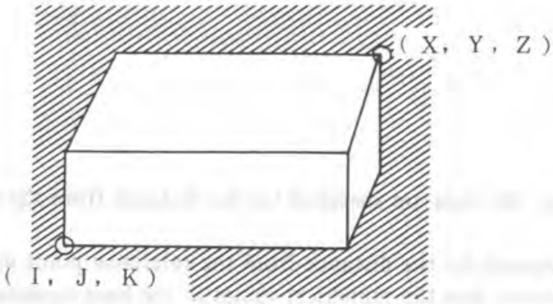
(Note 2) *G20 and G21 must not be changed during a program.*

(Note 3) *When the unit system of a machine and that of the program are different, the maximum switching error is half of the least command increment.*
The error is not accumulated.

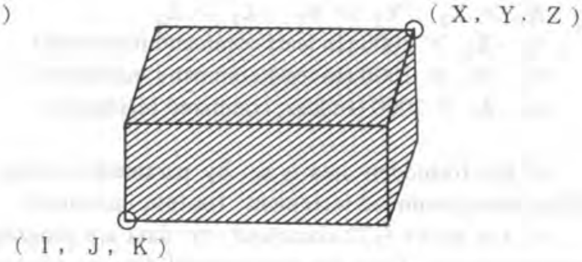
5.19 Stored Stroke Limit (G22, G23)

The movable range of a tool can be restricted in the following two ways.

(Tools cannot enter into the shaded areas)



Forbidden area on the outside



Forbidden area on the inside

Stored stroke limit 1:

Parameters set boundary. Outside the area of the limits set is a forbidden area. This is not normally changed, once the machine tool builder sets this area. Usually this area is set at the maximum stroke. This is equivalent to a conventional software limit.

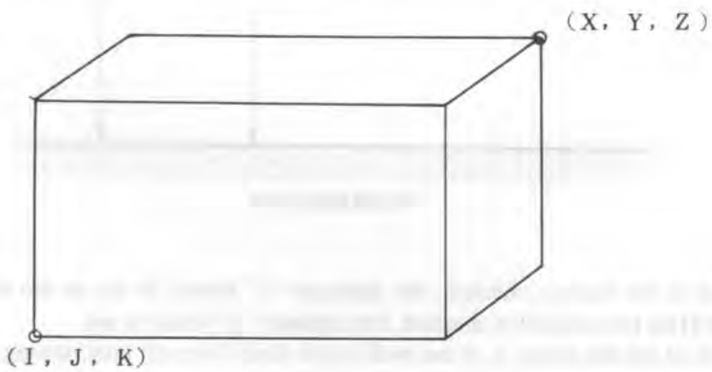
Stored stroke limit 2:

Parameters or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter (RWL) selects either inside or outside as the forbidden area.

A G22 command forbids a tool to enter the forbidden area and a G23 command permits a tool to enter the forbidden area.

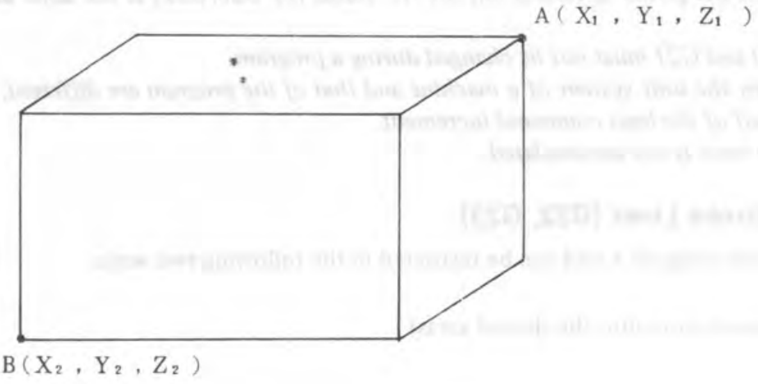
The command below creates or changes the forbidden area.

G22 X____Y____Z____I____J____K____;



$X > I, Y > J, Z > K$
$X - I > 2000$ (In least command increment)
$Y - J > 2000$ (In least command increment)
$Z - K > 2000$ (In least command increment)

In parameter setting, points A and B in the figure below must be set.

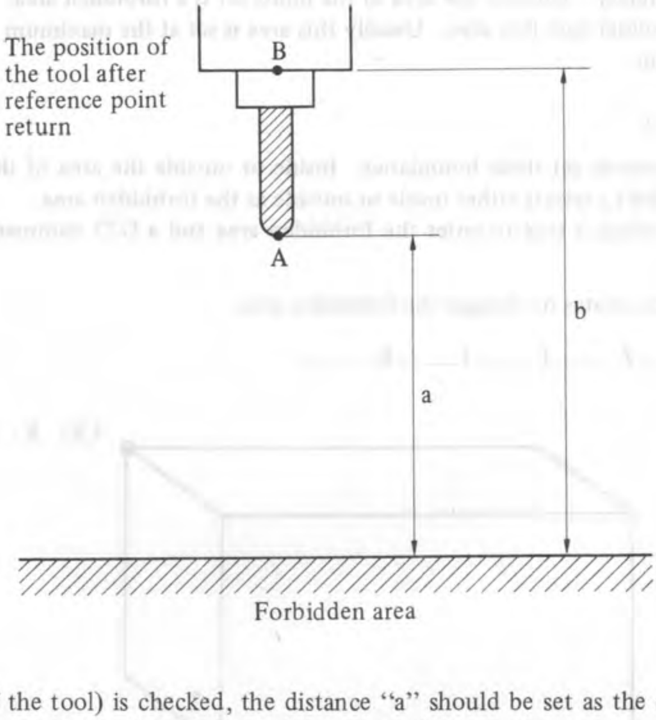


$X_1 > X_2$, $Y_1 > Y_2$, $Z_1 > Z_2$
$X_1 - X_2 > 2000$ (In least command increment)
$Y_1 - Y_2 > 2000$ (In least command increment)
$Z_1 - Z_2 > 2000$ (In least command increment)

If the forbidden area is set by parameter setting, the data are specified by the distance from the reference point in least command increment. (output increment)

If it is set by G22 command, the data are programmed by the distance from the reference point in the least input increment. Then the programmed data are converted into the numerical values in the least command increment. And the values are set as the parameters.

Confirm the checking position (the top of the tool or the tool chuck) before programming by this stored stroke limit function.

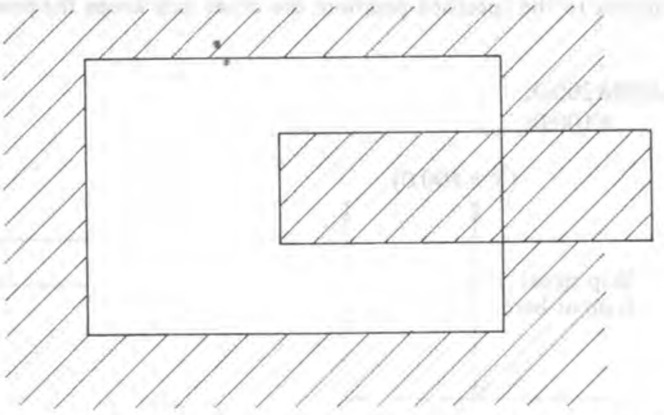


If point A (The top of the tool) is checked, the distance "a" should be set as the data for the stored stroke limit function. If point B (The tool chuck) is checked, the distance "b" must be set.

When checking the tool tip like point A, if the tool length varies for each tool, setting of the forbidden area for the longest tool requires no re-setting and results in safe operation.

80

Area can be set in piles.



(Note 1) *Each limit becomes effective after the power was turned on and manual reference point return or automatic reference point return by G28 has been performed.*

(Note 2) *After turning on power and performing the reference point return, if the reference point is in the forbidden area of each limit, an alarm is generaged immediately. (Only in G22 mode for stored stroke limit 2).*
When G23 is switched to G22 while the tool is in the forbidden area, an alarm will be generated in the next block.

(Note 3) *When the tool has become unmovable in the forbidden area under the condition of (Note 2), push the emergency stop button to release the forbidden condition and move the tool out of the forbidden area in the G23 mode. Then, perform the reference point return again after correcting the setting if the setting was wrong.*

(Note 4) *Because an axis not equipped with reference point return function has no forbidden area, the alarm associated with that axis forbidden area does not exist.*

(Note 5) *In setting a forbidden area, if the two points to be set are the same, the area is as follows.*
When the forbidden area is the outside of the specified area, all areas are movable areas.
When the forbidden area is the inside of the specified area, in G22 mode, all areas are forbidden areas.

(Note 6) *Unnecessary limits should be set beyond the machine stroke.*

(Note 7) *When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came into the forbidden area.*

(Note 8) *In setting the area, even if you mistake the order of the coordinate value of the two points, a rectangular with the two points being the apexes will be set as the area.*

(Note 9) *Each of G22 ——— ; and G23 ; should be commanded singly in a block.*

(Note 10) *The stored stroke limit function is not available for additional axis.*

5.20 Skip Function (G31)

A block with a G31 code commands linear interpolation as in the case of G01. Input of the skip signal during execution of this command interrupts the rest of the block and executes the next.

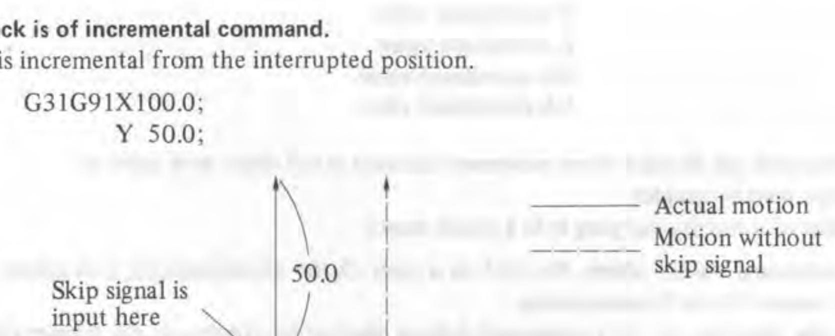
G31 is a one-shot command and effective only to the block concerned.

The motion after input of the skip signal depends on whether the next block is of incremental or absolute command.

(1) The next block is of incremental command.

The motion is incremental from the interrupted position.

(Example) G31G91X100.0;
Y 50.0;

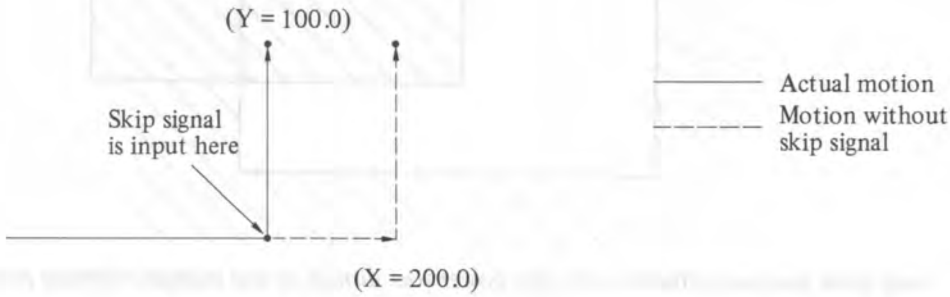


81

(2) The next block is of absolute command only with one axis specified.

The specified axis moves to the specified position; the other axis keeps the position when the skip signal was input.

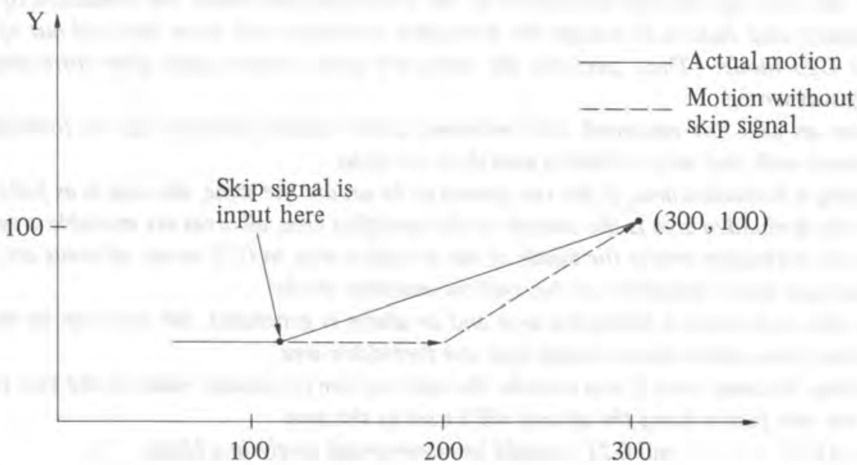
(Example) G31G90X200.0;
Y100.0;



(3) The next block is of absolute command with two axes specified.

The tool moves to the specified position regardless of the skip signal input.

(Example) G31G90X200.0;
X300.0Y100.0;



Parameter No. 306 (SKPF) specifies the feed rate of G31 block in the following two ways.

(a) Feed rate specified by F (which may be specified before or in G31 block)

(b) Feed rate set in parameter No. 342

The custom macro can use the coordinate values when the skip signal turned on, because they are stored in the system variables #5061 to #5065 of the custom macro.

#5061 .... X coordinate value
#5062 .... Y coordinate value
#5063 .... Z coordinate value
#5064 .... 4th coordinate value
#5065 .... 5th coordinate value

The skip function can be used when movement amount is not clear, as in cases of:

(a) Standard-size feed by grinder

(b) Measurement of a tool by carrying it to a touch sensor

(Note 1) G31 command causes alarm No. 035 in a state Cutter Compensation is in effect. Before G31 command, cancel Cutter Compensation.

(Note 2) When the feed rate by G31 command follows that set by parameter, the former follows the latter also when Dry Run is in effect.

(Note 3) When the feed rate by G31 command follows that set by parameter, no automatic acceleration/deceleration is effected, in order to raise accuracy of automatic measurement in use of the skip function.

6. COMPENSATION FUNCTION

6.1 Tool Length Compensation (G43, G44, G49)

By the command of G43
G44) Z______ H______ ;

or

G43
G44) H______ ;

the position of the terminal point of the movement command in the Z axis can be shifted + or – by the value set in the offset memory. This function can be used by setting the difference between the tool length assumed during programming and the actual tool length of the tool used when the work is performed into the offset memory. It is now possible to apply compensation without changing the program. Assign the offset direction with a G43 or G44, and design the offset amount that has been set in the offset memory with an H code.

(1) Direction of offset

G43 } + side offset
G44 } –side offset

In any case of absolute or incremental commands, the offset amount that has been set into the offset memory assigned by H code is in G43, added to, and in G44, subtracted from the coordinate value of the terminal point of the Z axis movement command. The coordinate value after the calculation becomes the terminal point.

When the movement command for the Z axis was omitted, it is taken in the same way as;

G43
G44) G91Z0H ______ ;

and it moves by the offset amount in the + direction in G43 and in the – direction in G44.

G43 and G44 being modal codes, are effective until another G code in the same group is programmed. Either G43 or G44 can be in effect when the power is turned on depending on a parameter setting.

(2) Assignment of offset amount

Assign offset number by H code. The offset amount that has been set in the offset memory is added to or subtracted from the programmed command value for the Z axis. The offset number can be assigned by H00 to H200. However, there are 200 offsets to be used jointly with the D codes used for cutter compensation (32 offsets are standard).

The offset amount may be set in the offset memory through the MDI & CRT unit or the tape reader. The range of values that can be set as the offset amount is as follows;

	mm input	inch input
Offset amount	0 to ±999.999 mm	0 to ±99.9999 inch

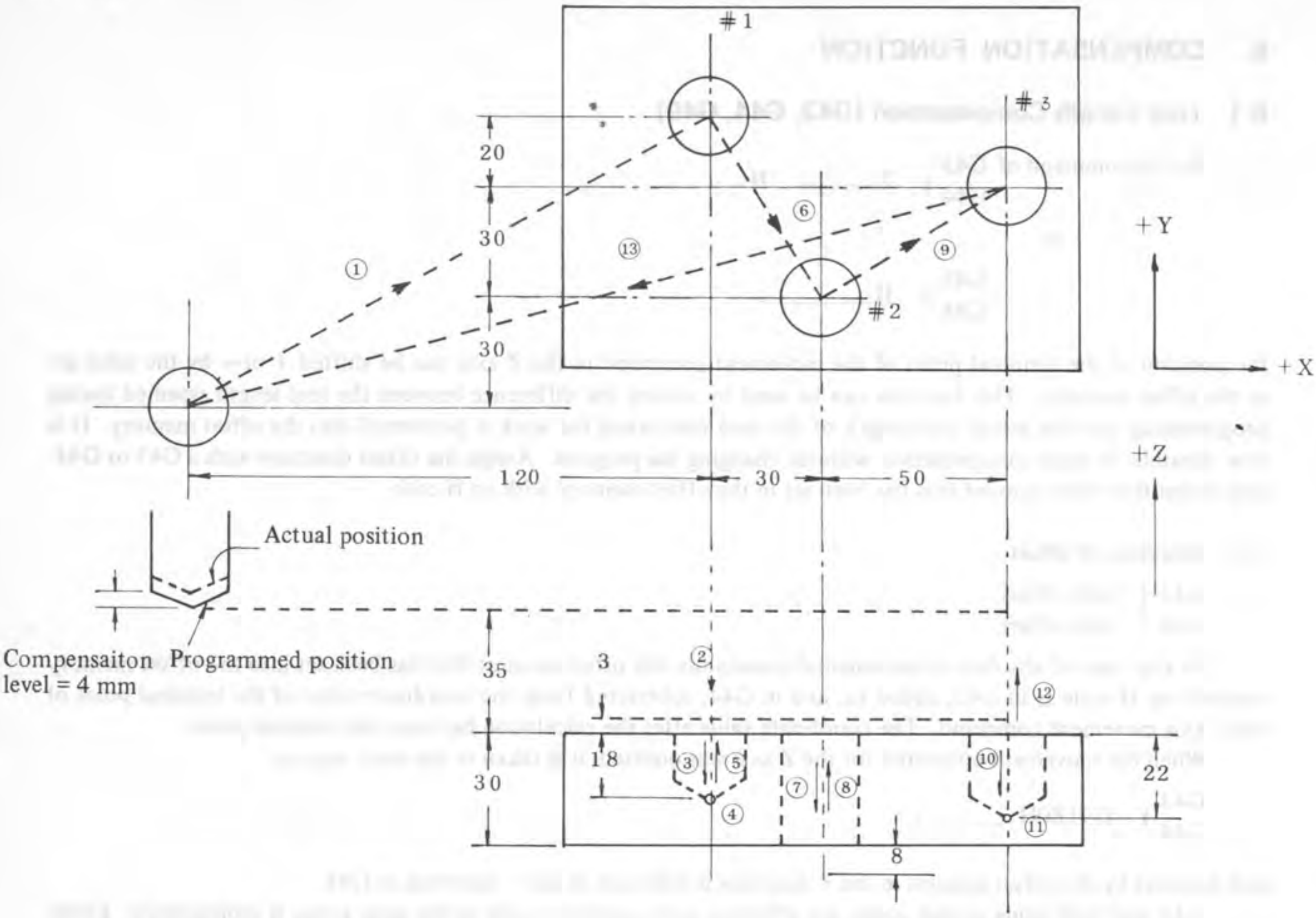
The offset value corresponds to offset No. 00 or H00 always means 0. It is impossible to set H00 to any other offset value.

(3) Cancelling tool length compensation

To cancel the offset, command a G49 or assign offset H00. When H00 or G49 is commanded, the cancelling action is taken immediately.

(4) Example of tool length compensation

(No. 1, 2 and 3 borings)



H01 = -4.0 (Offset amount)

N1G91	G00	X120.0	Y80.0	;	....	①
N2	G43		Z-32.0		H01 ;	②
N3	G01		Z-21.0		F1000 ;	③
N4	G04				P2000 ;	④
N5	G00		Z21.0	;	....	⑤
N6		X30.0	Y-50.0	;	....	⑥
N7	G01		Z-41.0	;	....	⑦
N8	G00		Z41.0	;	....	⑧
N9		X50.0	Y30.0	;	....	⑨
N10	G01		Z-25.0	;	....	⑩
N11	G04				P2000 ;	⑪
N12	G00		Z57.0	;	H00 ;	⑫
N13		X-200.0	Y-60.0	;	....	⑬

N14M02 ;

(Note 1) *When the offset amount is changed due to a change of the offset number, and offset value changes to the new offset amount, the new offset amount is not added to the old offset amount.*

H01 .... offset amount 20.0
H02 .... offset amount 30.0
G90G43Z100.0H01; ..... Z will go to 120.0
G90G43Z100.0H02; ..... Z will go to 130.0

(Note 2) *For tool length compensation, D codes cannot be used.*

It is possible to apply too length compensation to other than Z axis.
Specify an axis, to which tool length compensation is applied, in a block with G43 or G44.

$\left.\begin{array}{l}\text{G43}\\ \text{G44}\end{array}\right\}$ α ___ H ___ ; (α : An axis)

An alarm will be generated by the following command, because the tool length compensation is applied to one axis.

84

G43Z___H___;
G43X___H___; (Alarm)

To change the axis, cancel the tool length compensation once.

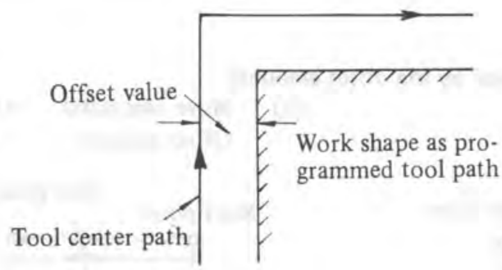
6.2 Tool Offset (G45 to G48)

By specifying G45 to G48, the movement distance of the axis specified on NC tape, etc. can be expanded or reduced by the value set in the offset memory. Table 6.2 shows the G codes and their functions.

Table 6.2 Tool offset and G codes

G code	Function
G45	Tool offset expansion
G46	Tool offset reduction
G47	Tool offset double expansion
G48	Tool offset double reduction

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 or H code, remain unchanged until other offset amounts are selected. The usable code D/H can be set by parameter setting (OFSD).



The offset amount ranges as follows:

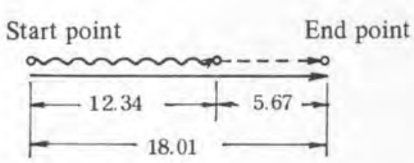
	Input in metric	Input in inch
Offset amount	0 to ±999.999 mm	0 to ±99.9999 inch
	0 to ±999.999 deg	0 to ±999.999 deg

This tool offset function is effective for the additional axis (4th axis).
When the offset number is 00 (H00 or D00), the offset amount is always zero.
Expansion and reduction are made in the axis direction in which the tool is moved.
Also in absolute programming, expansion and reduction are made in the moving direction.

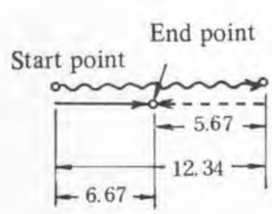
(1) G45 command (expansion by the offset amount)

Move command value
Offset amount
Actual movement amount

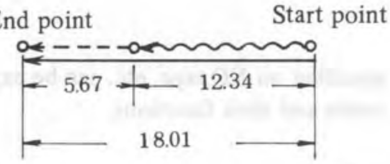
(a) Move command +12.34
Offset amount +5.67



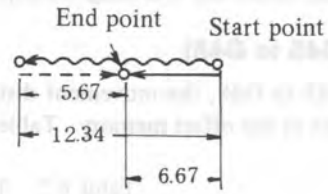
(b) Move command +12.34
Offset amount –5.67



(c) Move command –12.34
Offset amount +5.67



(d) Move command –12.34
Offset amount –5.67

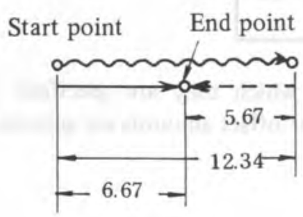


(2) G46 command (reduction by the offset amount)

The actual motion becomes equivalent if the sign of the offset is reversed in G45 command.

(a) Move command +12.34
Offset amount +5.67

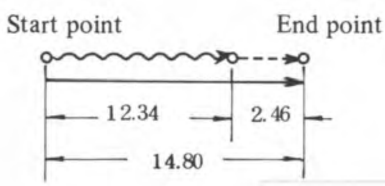
(b) to (d) are omitted.



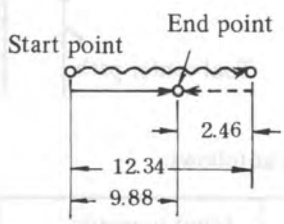
(same as in (1) - (b))

(3) G47 command (double expansion by the offset amount)

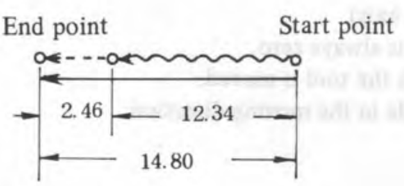
(a) Move command +12.34
Offset amount +1.23



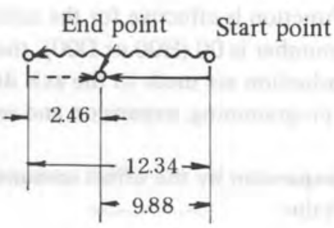
(b) Move command +12.34
Offset amount –1.23



(c) Move command –12.34
Offset amount +1.23



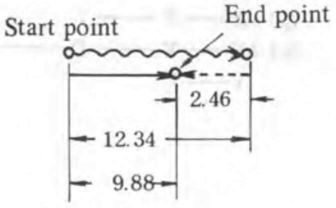
(d) Move command –12.34
Offset amount –1.23



(4) G48 command (double reduction by the offset amount)

The actual motion becomes equivalent if the sign of the offset is reversed in G47 command.

(a) Move command +12.34
Offset amount +1.23



(b) to (d) are omitted.

(same as in (3) - (b))

In incremental mode, if the movement amount is specified as zero, the tool moves only by the offset amount. In absolute mode, if the movement amount is specified as zero, the tool does not move.

Offset amount +12.34 (Offset No. 01)

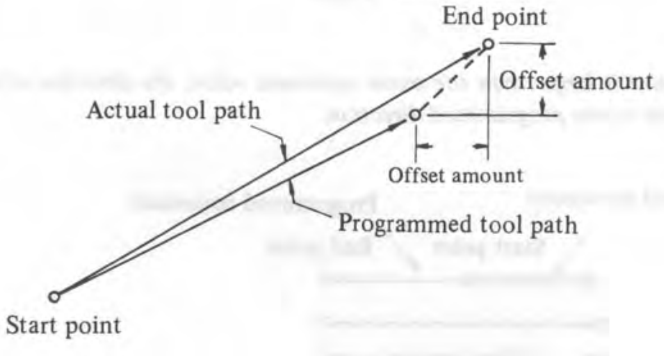
Programming command	G91G45X0D01;	G91G46X0D01;	G91G45X-0D01;	G91G46X-0D01;
Equivalent command	X12.34;	X-12.34;	X-12.34;	X12.34;

Notes on tool offset:

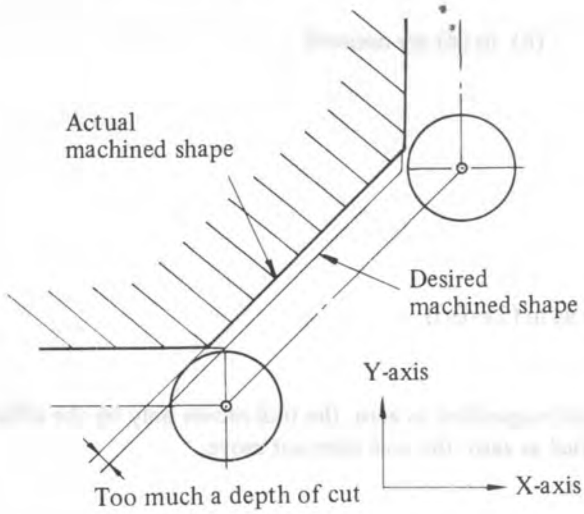
(Note 1) *If any one of G45 to G48 is specified for the simultaneously 2 axes control, tool offset is effected for both axes.*

In case of G45

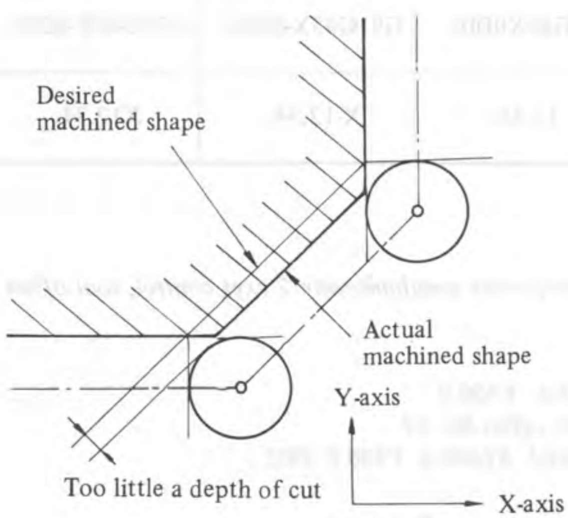
Move command	*X1000.0, Y500.0*
Offset amount	*+200.0, offset No. 02*
Programmed command	*G45 G01 X1000.0 Y500.0 D02 ;*



(Note 2) *In taper cutting, if usual tool offset is performed, too much or little a depth of cut is generated.*

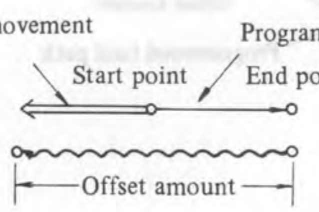



```
G01X—F—;
G47X—Y—D—;
Y—;
```




```
G01G45X—F—D—;
X—Y—;
G45Y—;
```

(Note 3) *When the offset amount is larger than the move command value, the direction of the actual tool movement becomes opposite to the programmed direction.*



(Example)

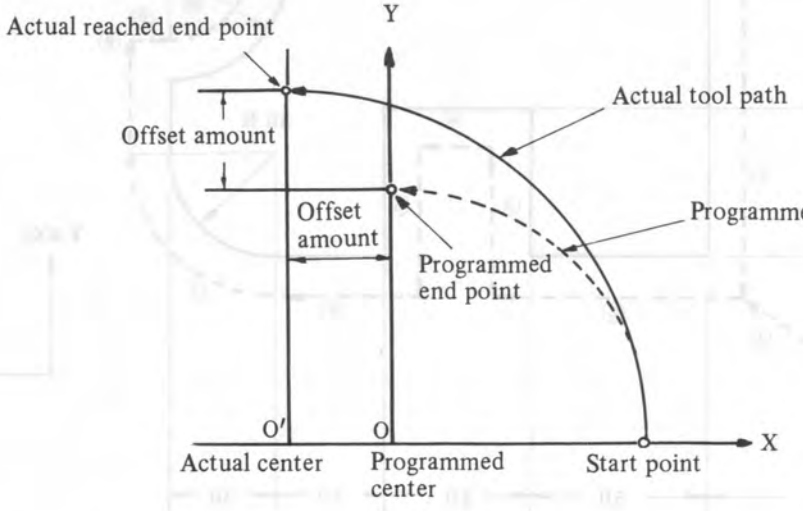
G46X 2.50 ;	
(Incremental command)	*Equivalent command*
Offset amount +3.70	*X-1.20 ;*

(Note 4) *Tool offset can be effected by G45 to G48 commands for circular interpolation (G02, G03) in the case of one-quarter and three-quarter arc command. That is, cutter compensation is possible only in the case of one-quarter and three-quarter arc command.*

98

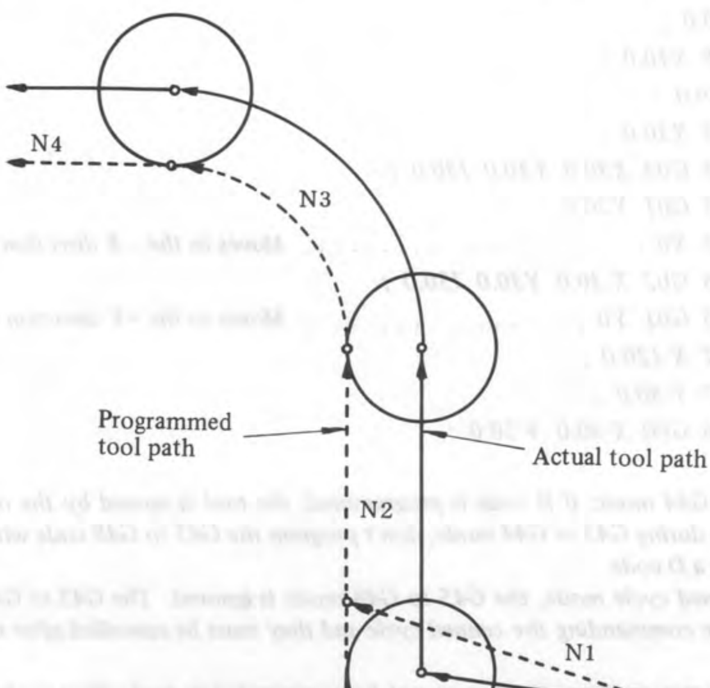
(Example 6.2.1)   Offset amount +20.0, offset number 01

Programmed command
(G91 mode)
G45 G03 X-70.0 Y70.0 I-70.0 D01 ;

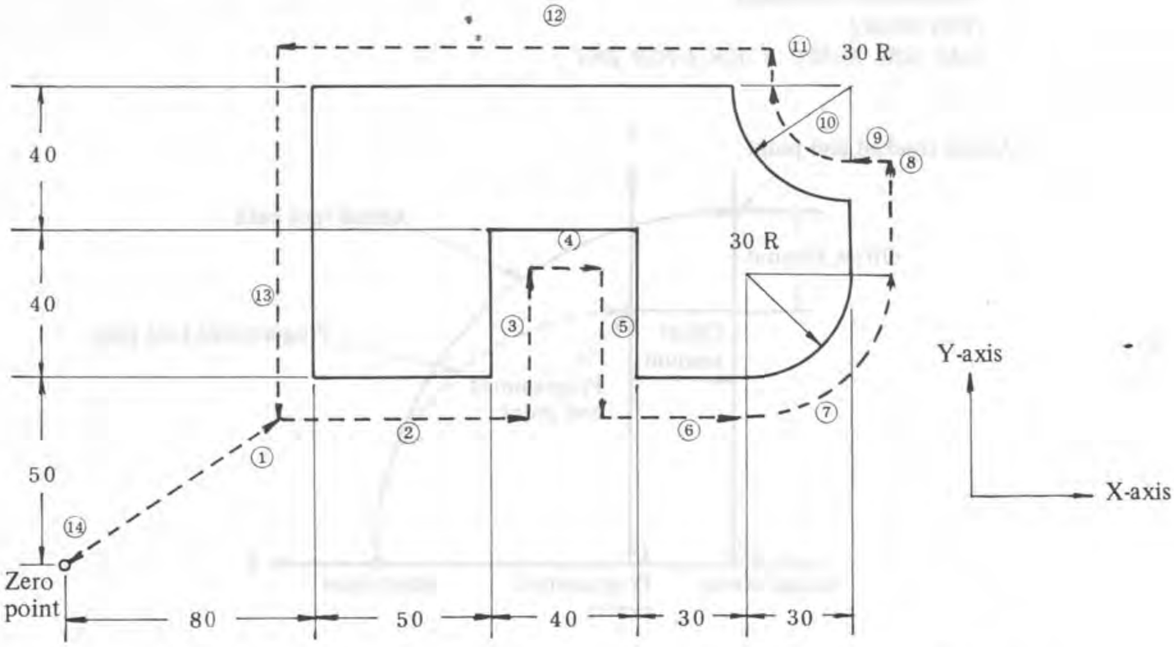


(Example 6.2.2)   Cutter compensation in circular interpolation

N1 G46 G00 X_____ Y_____ D_____ ;
N2 G45 G01 Y_____ F_____ ;
N3 G45 G03 X_____ Y_____ I _____ ;
N4 G01 X___ ;



(Example 6.2.3) Program example using tool offset



Unit: mm

Tool diameter	:	*20 mm dia.*
Offset number	:	*01*
Offset amount	:	*+10.0 mm*

① *G91 G46 G00 X80.0 Y50.0 D01 ;*
② *G47 G01 X50.0 F120 ;*
③ *Y40.0 ;*
④ *G48 X40.0 ;*
⑤ *Y-40.0 ;*
⑥ *G45 X30.0 ;*
⑦ *G45 G03 X30.0 Y30.0 J30.0 ;*
⑧ *G45 G01 Y20.0 ;*
⑨ *G46 X0 ;* .... *Moves to the –X direction by the offset amount.*
⑩ *G46 G02 X-30.0 Y30.0 J30.0 ;*
⑪ *G45 G01 Y0 ;* .... *Moves to the +Y direction by the offset amount.*
⑫ *G47 X-120.0 ;*
⑬ *G47 Y-80.0 ;*
⑭ *G46 G00 X-80.0 Y-50.0 ;*

(Note 5) *During G43 or G44 mode, if H code is programmed, the tool is moved by the offset amount in the Z axis. Therefore during G43 or G44 mode, don't program the G45 to G48 code with a H code. In this case, use a D code.*

(Note 6) *During the canned cycle mode, the G45 to G48 mode is ignored. The G45 to G48 modes must be programmed before commanding the canned cycle and they must be cancelled after terminating the canned cycle.*

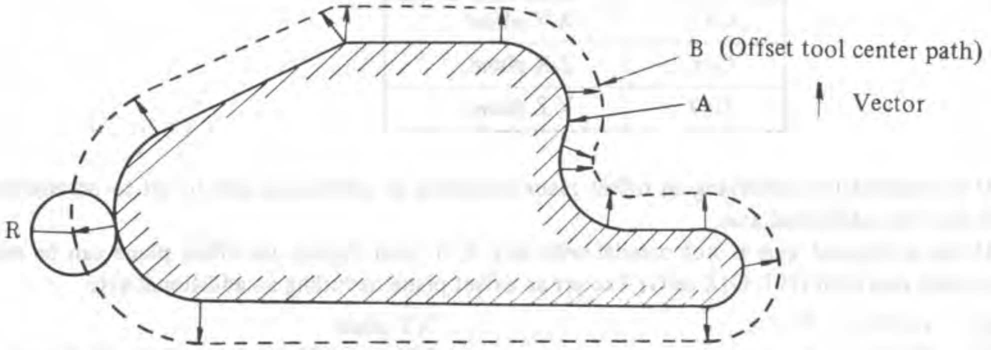
(Note 7) *G41 or G42 (cutter compensation) must not be commanded in tool offset mode (G45 to G48). In the above case, P/S alarm (alarm number 36) occurs.*

6.3 Cutter Compensation (G40 to G42)

6.3.1 Cutter compensation function

In the figure below, in order to cut a workpiece indicated as A with an R-radius tool, the path for the center of the tool must be the B which is separated R distance from the A. The tool being separated some distance like this is called offset. By the cutter compensation function, the tool path being separated some distance (namely, offset) is computed.

Therefore, the workpiece shape is programmed with the cutter compensation mode by a programmer, and in machining, if the cutter radius (offset amount) is measured and set to the NC, the tool path is offset (path B) regardless of the program.



Offset and Vector

Cutter compensation has two types (B and C). In this section, only type C is explained. The difference between B and C is as follows.

In the cutter compensation B mode, the inside offset at the acute corner of 90 deg. or less cannot be performed. In this case, the proper corner arc must be inserted in programming.

6.3.2 Offset amount (D code)

The offset amount can be set in the offset memory up to 200 offsets (in standard 32 offsets). (However, they are 200 offsets in total for tool length compensation and tool offset.) The offset amount depends on the two digit number following the D code commanded on the program, and has been set via the MDI & DPL or the tape reader.

The range of values in which the offset amount can be set is as follows;

	mm input	inch input
Offset amount	0 to ±999.999 mm	0 to ±99.9999 inch

The offset amount corresponding to offset No. 00 or D00 always means 0.
It is impossible to set D00 to any other offset amount.

6.3.3 Offset vector

The offset vector is the two dimensional vector that is equal to the offset amount assigned by D code. It is calculated inside the control unit, and its direction is up-dated in accordance with the progress of the tool in each block. This offset vector (hereinafter called vector) is produced inside the control unit in order to find out how much the tool motion should be offset, and is used to compute a path offset from the programmed path by the tool radius. The offset vector is deleted by reset.

This vector always follows the tool as its progress, and it is very important, when making a program, to understand the state of the vector. Read the following description carefully to understand how the vector is generated.

91

6.3.4 Plane selection and vector

Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection.) This plane is called the offset plane. For instance, when the XY plane has been selected, the offset calculations are carried out using (X, Y) or (I, J) in the program tape and the vector is computed. The coordinate values of an axis not in the offset plane are not subject to the offset, and the command values for that axis on the program tape are used as they are.

In simultaneous 3 axes control, the tool path projected on the offset plane is compensated.

The offset plane must be changed during the offset cancel mode. If it is performed during the offset mode, an alarm (No. 37) is displayed and the machine is stopped.

G code	Offset plane
G17	X-Y plane
G18	Z-X plane
G19	Y-Z plane

It is required for specifying an offset plane including an additional axis to set to parameters which axis is parallel with the additional axis.

If the additional axis is not parallel with any X, Y, and Z axes, no offset plane can be created. Specify an additional axis with G17, G18 or G19 to set an offset plane including an additional axis.

(i) G17X___Y___; .... XY plane
(ii) G17U___Y___; .... UY plane (U axis is parallel with X axis)
(iii) G17Y___; .... XY plane
(iv) G17; .... XY plane
(v) G17X___Y___U___; .... Alarm
(vi) G18X___W___; .... XW plane (W axis is parallel with Y axis)

6.3.5 G40, G41 and G42

Using G40, G41 and G42, the deletion and generation of cutter radius compensation vectors is commanded. They are commanded simultaneously with G00, G01, G02 or G03 to define a mode which determines the amount and direction of offset vectors, and the direction of tool motion.

G code	Function
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right

A G41 or G42 command causes the equipment to enter the offset mode, and a G40 command causes the equipment to enter the cancel mode.

The offset procedure, for example, is explained in the figure below.

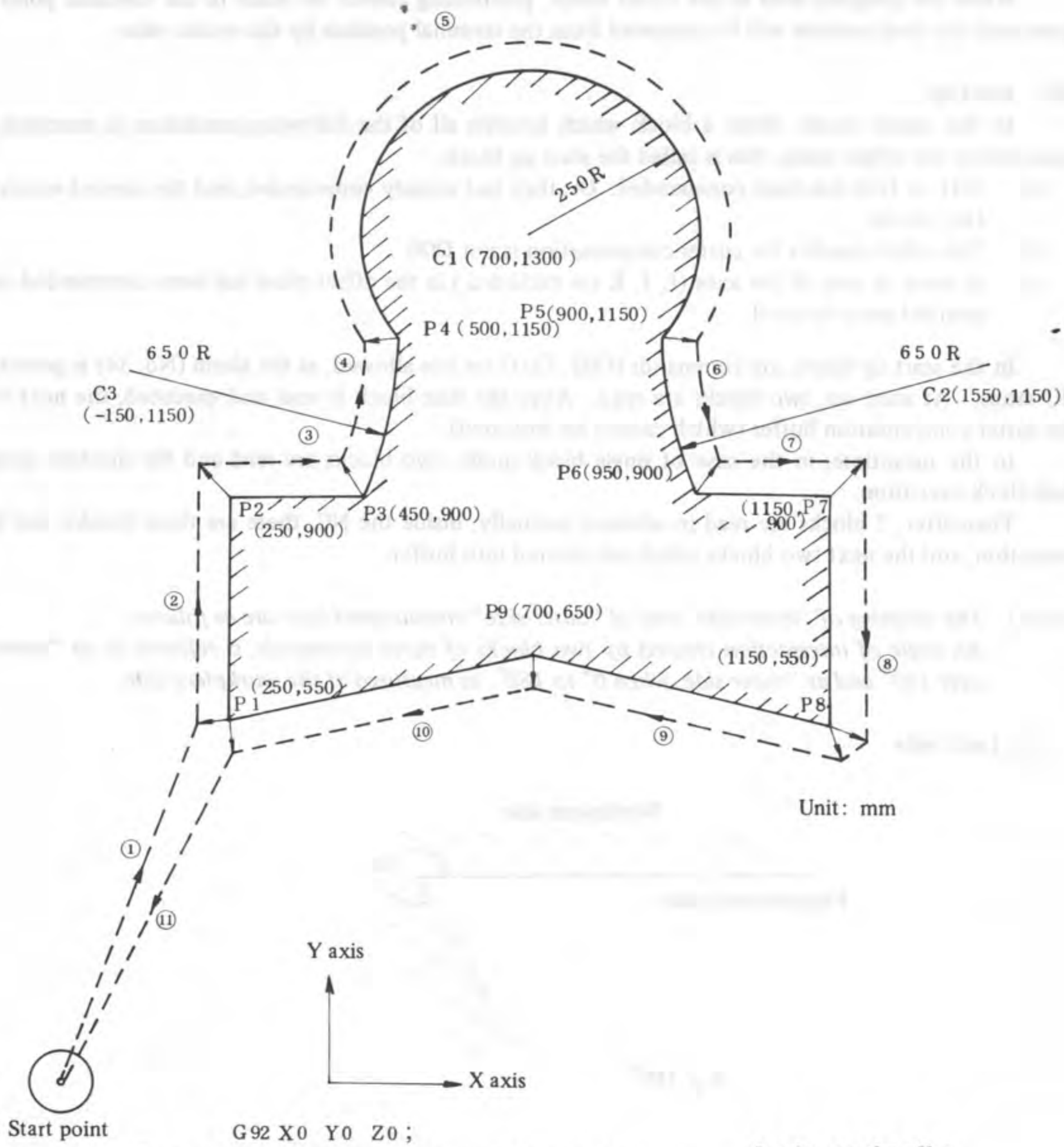
The block ① is called the start-up. In this block, the offset cancel mode is changed to the offset mode (G41). At the end point of this block, the tool center is translated by the cutter radius in the direction being vertical to the next block path (from P1 to P2). The cutter compensation value is specified by D07, namely the offset number is 7, and G41 means the cutter compensation left.

After being start-up, when the workpiece shape is programmed as P1 → P2 ..... → P8 → P9 → P1, the cutter compensation is performed automatically.

In the block ⑪, the cutter returns to the start point by commanding G40 (offset cancel). At the end point of the block ⑩, the cutter center is translated vertically to the programmed path (from P9 to P1).

At the end of the program, the G40 (offset cancel) must be command.

Example of program of cutter compensation C




```
            G92 X0  Y0  Z0 ;
①  N1   G90 G17 G00 G41  D07 X250.0 Y550.0 ;
②  N2   G01  Y900.0 F150 ;
③  N3   X450.0 ;
④  N4   G03 X500.0  Y1150.0  I-600.0  J250.0 ;
⑤  N5   G02 X900.0  I200.0  J150.0 ;
⑥  N6   G03 X950.0  Y900.0  I650.0  J0 ;
⑦  N7   G01 X1150.0 ;
⑧  N8   Y550.0 ;
⑨  N9   X700.0  Y650.0 ;
⑩  N10  X250.0  Y550.0 ;
⑪  N11  G00 G40 X0 Y0 ;
```

In advance, the offset amount must be set with the offset number 07.

6.3.6 Details of cutter compensation C

In this item, the details of the cutter compensation C is explained.

(1) Cancel Mode

At the beginning when power is applied, after reset has been performed or when the program has been terminated by execution of M02 and M30, the control is in the cutter compensation cancel mode.

93

In the cancel mode, the vector is always 0, and the tool center path coincides with the programmed path. The cancel mode (G40) should be programmed before the end of a program.

When the program ends in the offset mode, positioning cannot be made to the terminal point of the program, and the tool position will be separated from the terminal position by the vector value.

(2) Start up

In the cancel mode, when a block which satisfies all of the following conditions is executed, the equipment enters the offset mode, this is called the start up block.

(a) G41 or G42 has been commanded. Or, they had already commanded, and the control enters the G41 or G42 mode.

(b) The offset number for cutter compensation is not D00.

(c) A move in any of the axes (I, J, K are excluded.) in the offset plane has been commanded, and its commanded move is not 0.

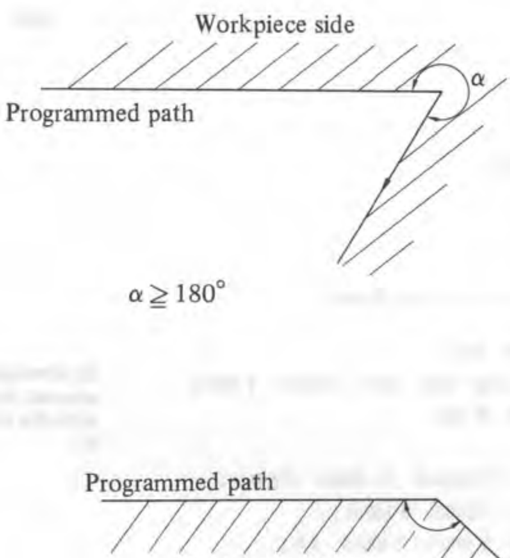
In the start up block, arc commands (G02, G03) are not allowed, as the alarm (No. 34) is generated and the NC stops. At start up, two blocks are read. After the first block is read and executed, the next block enters the cutter compensation buffer (which cannot be indicated).

In the meantime, in the case of single block mode, two blocks are read and the machine stops after the first block execution.

Thereafter, 2 blocks are read in advance normally, inside the NC, there are three blocks, the block under execution, and the next two blocks which are entered into buffer.

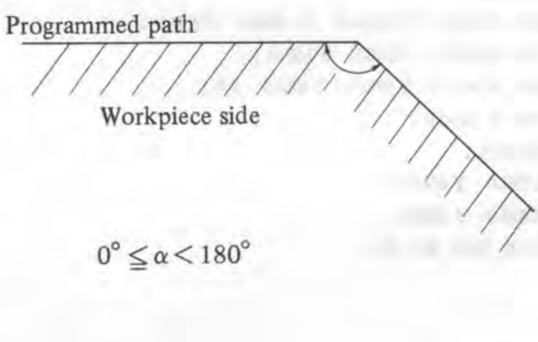
(Note) The meaning of "inner-side" and of "outer-side" encountered later are as follows:
An angle of intersection created by two blocks of move commands, is referred to as "inner-side, when over 180° and as "outer-side, when 0° to 180°, as measured at the workpiece side.

① Inner side



$\alpha \geqq 180°$

② Outer side

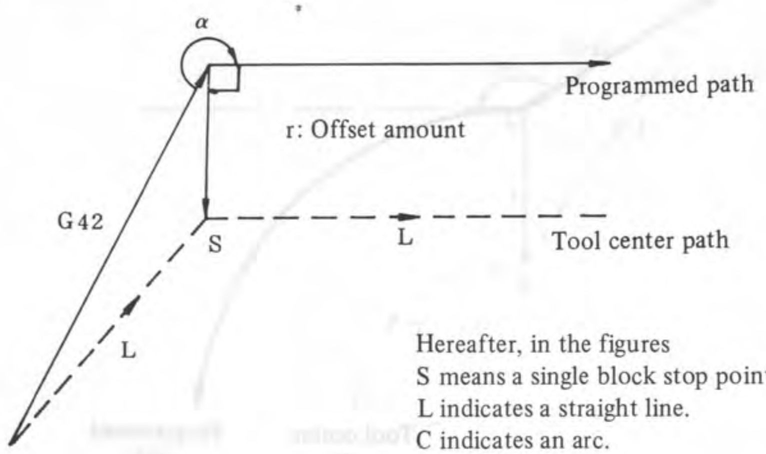


$0° \leqq \alpha < 180°$

94

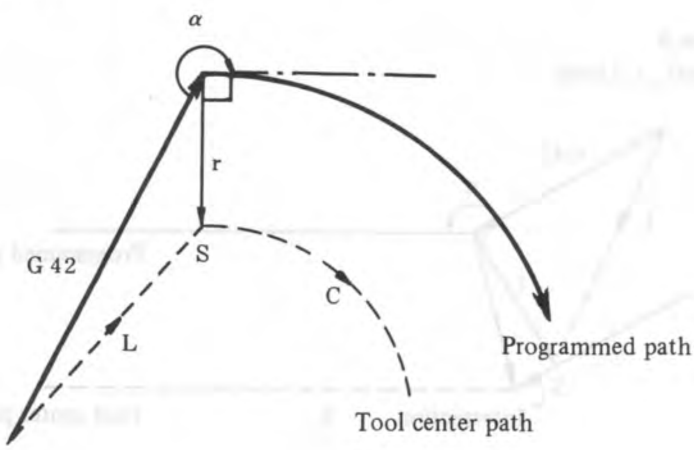
(i) When going around an inside corner (180° $\leqq \alpha$)

Linear → Linear



Hereafter, in the figures
S means a single block stop point.
L indicates a straight line.
C indicates an arc.

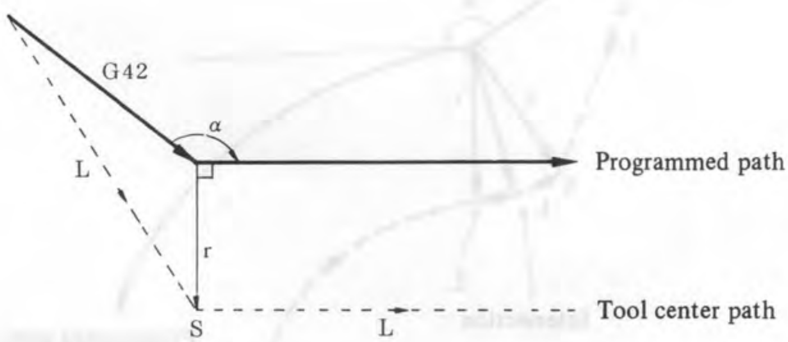
Linear → Circular



(ii) When the tool goes around the outside of a corner at an obtuse angle (90° $\leqq \alpha \leqq$ 180°)
Tool path in start up or cancel has two types A and B, and they are selected by parameter setting (SUPM).

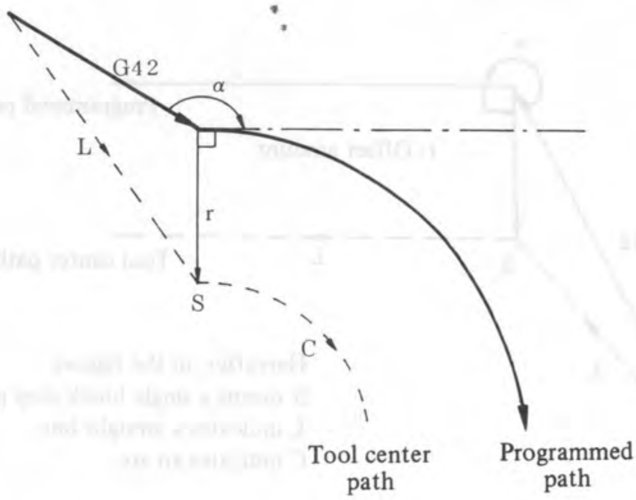
(a) Type A

Linear → Linear

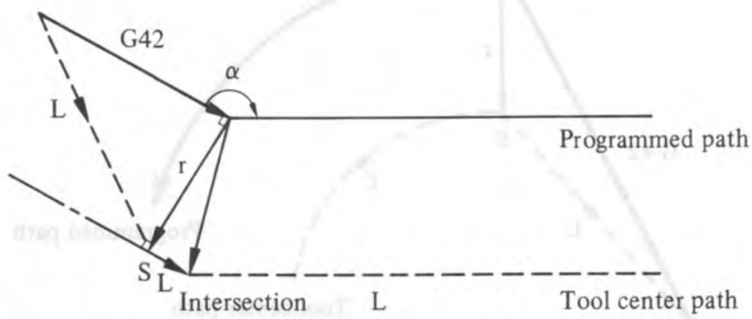


95

Linear → Circular

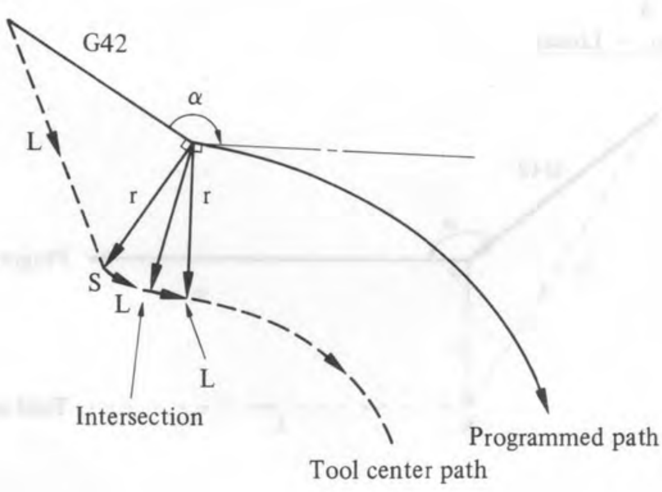


(b) Type B
Linear → Linear



The intersection is the point where two offset paths calculated by continuous two blocks are intersected.

Linear → Circular

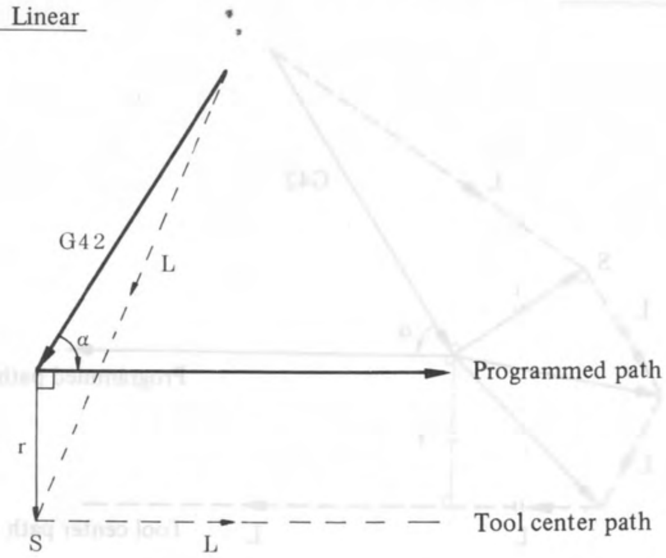


The intersection in the above figure is the intersecting point between offset paths by r distance of two blocks.

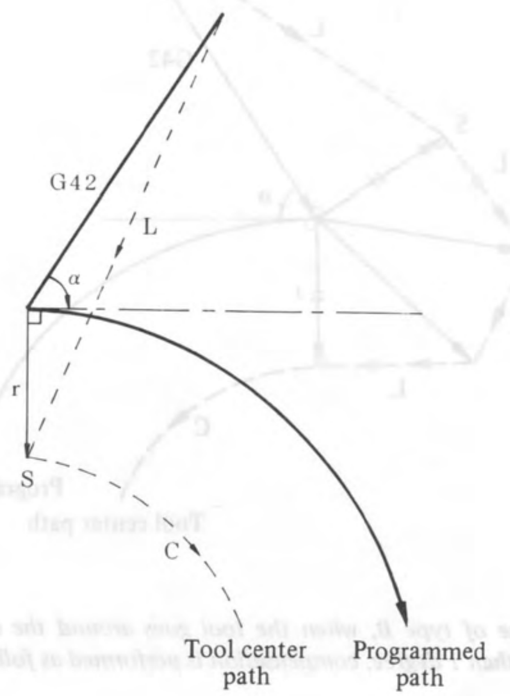
(iii) When going around the outside of an acute angle ($\alpha < 90°$)

(a) Type A

Linear → Linear



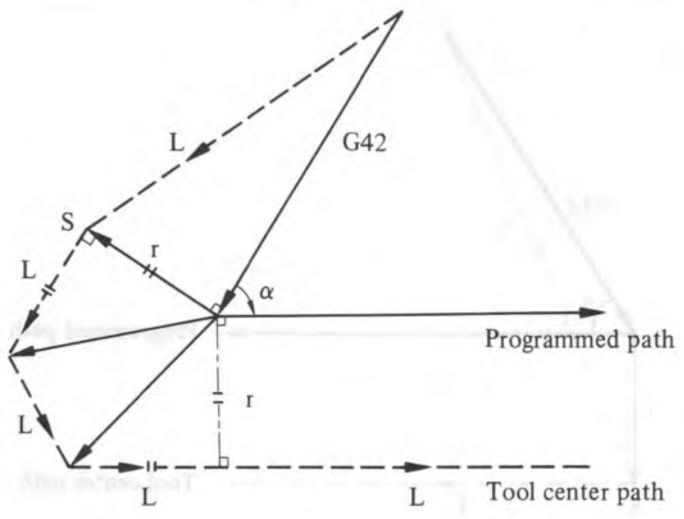
Linear → Circular



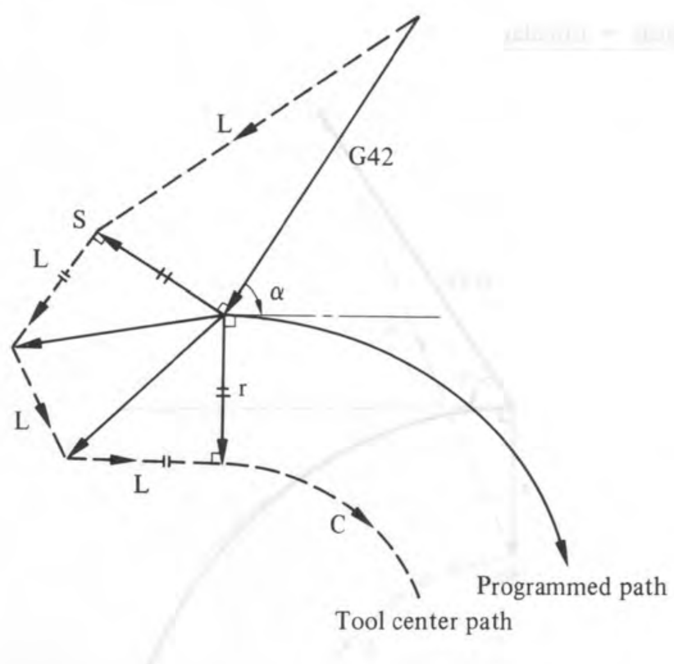
97

(b) Type B

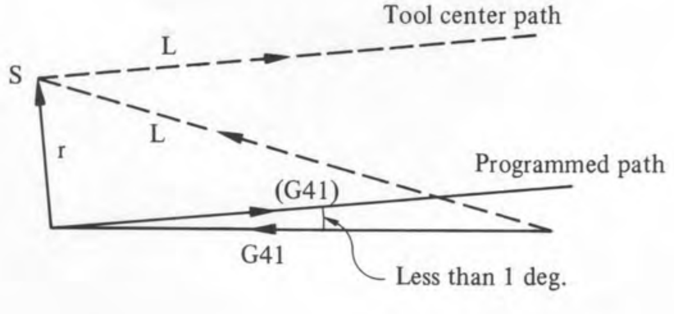
Linear → Linear



Linear → Circular



(Note) In the case of type B, when the tool goes around the outside linear → linear at an acute angle less than 1 degree, compensation is performed as follows.



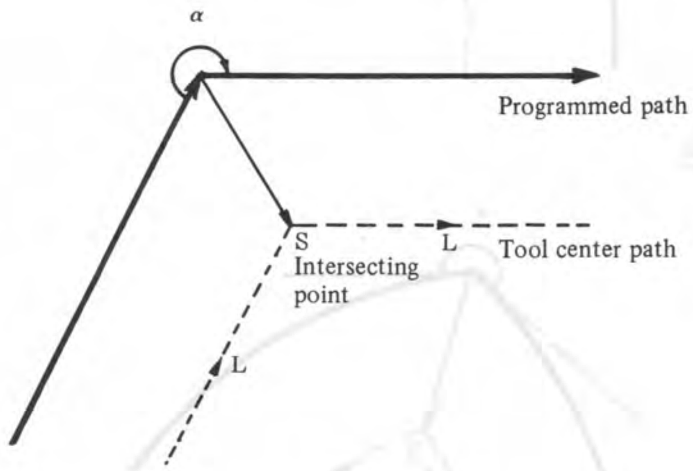
(3) In offset mode

In the offset mode, offset is performed correctly, if non-positioning commands such as auxiliary functions or dwell would not be programmed in two or more successive blocks. Otherwise overcutting or undercutting will occur. During the offset mode, offset plane change-over should not be carried out. If it does, the alarm (No. 37) is generated and the NC stops.

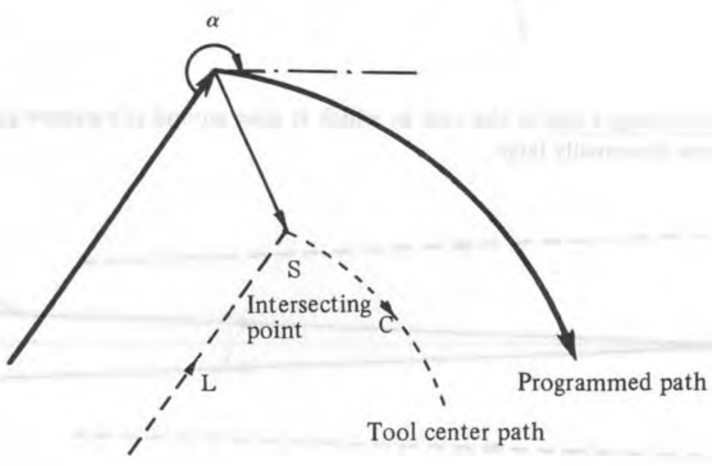
(a) When going around the inside ($\alpha \geqq 180°$)

When cutting an inside corner at an angle greater than 270°, or when going around an inside corner at an acute angle, machining is not performed and an alarm may be generated in the cutter compensation B. However, in the cutter compensation C, the machining can be carried out correctly as follows:

Linear → Linear

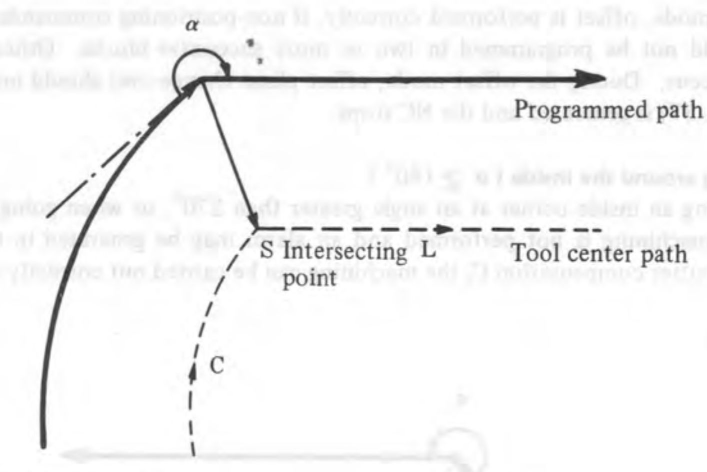


Linear → Circular

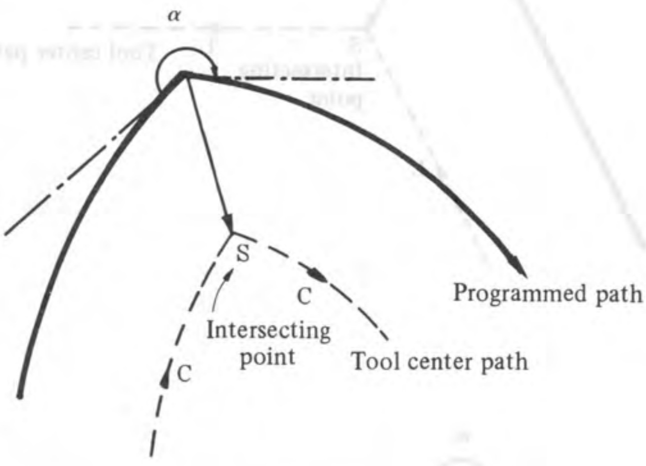


99

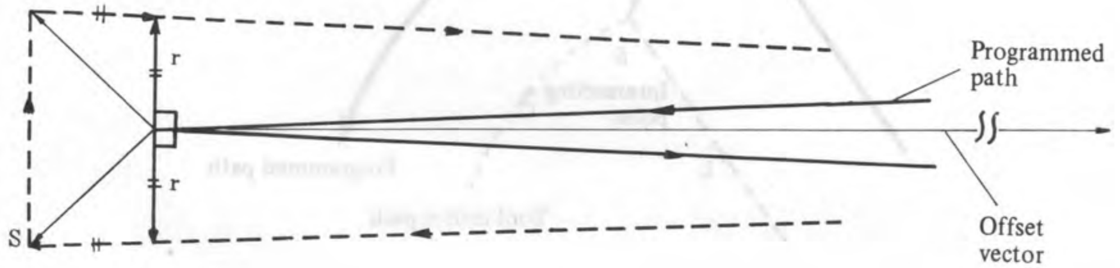
Circular → Linear



Circular → Circular



Straight line to straight line in the case in which it goes around at a narrow angle (less than 1°) and the offset vector becames abnormally large.

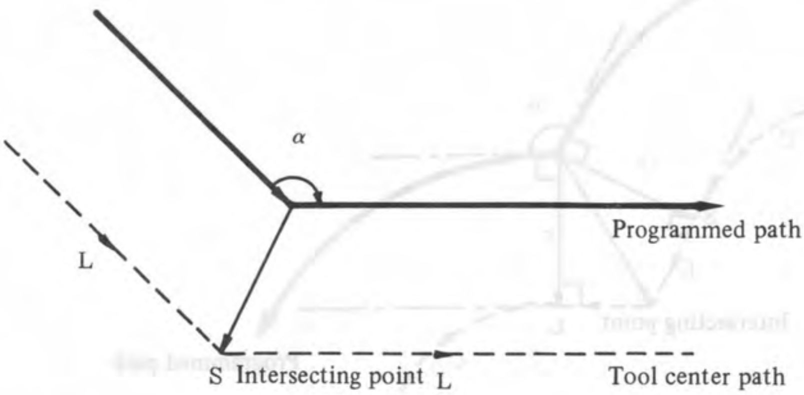


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

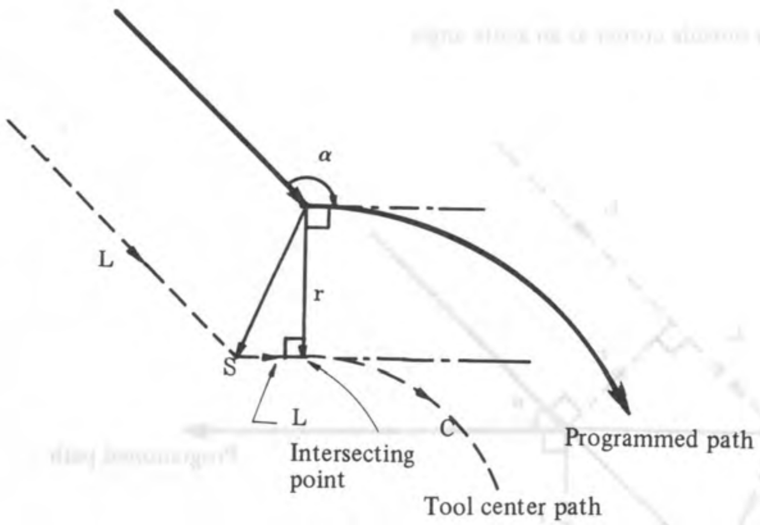
100

(b) When going around an outside corner at an obtuse angle ($90 \leqq \alpha < 180°$)

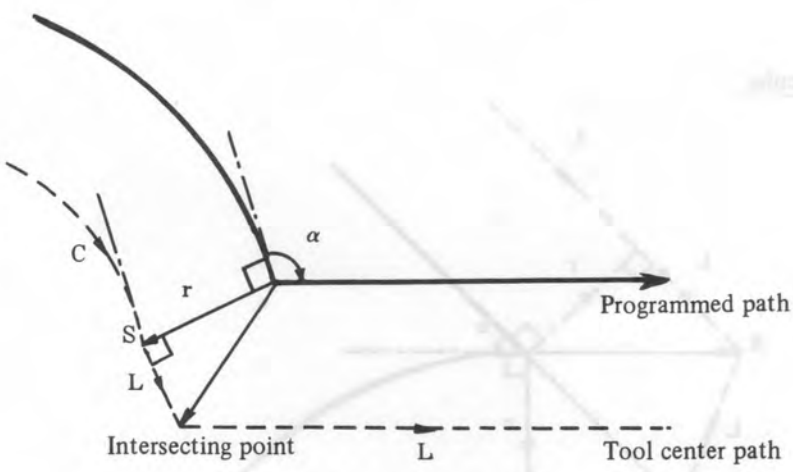
Linear → Linear



Linear → Circular

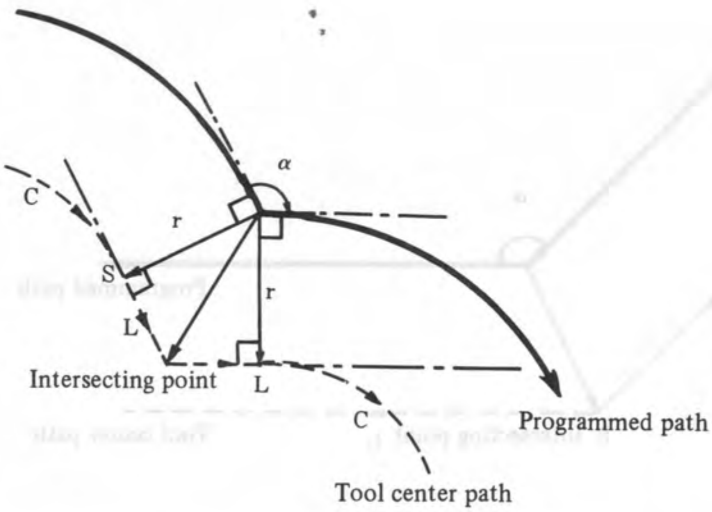


Circular → Linear



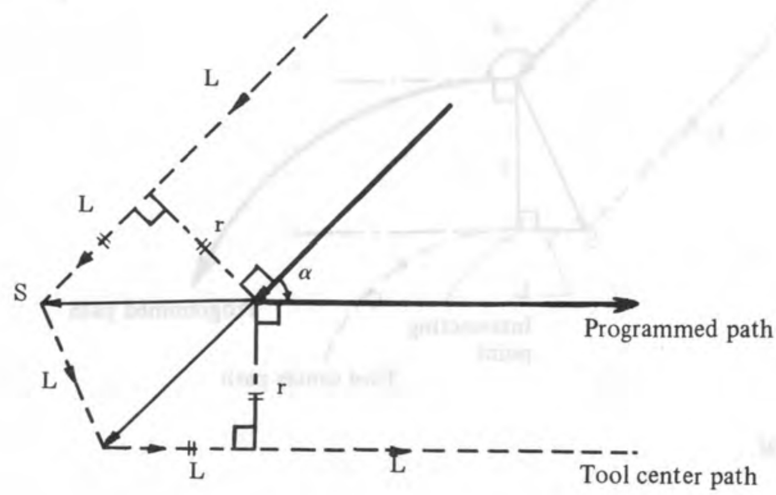
101

Circular → Circular

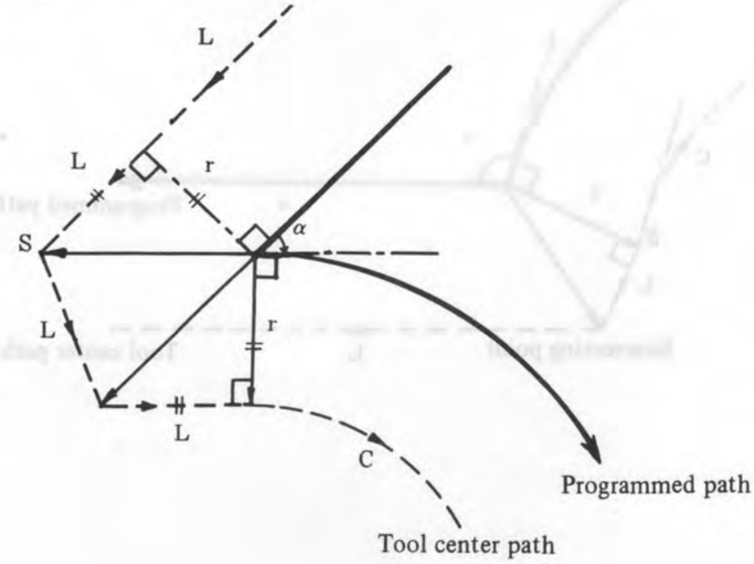


(c) When going an outside corner at an acute angle

Linear → Linear

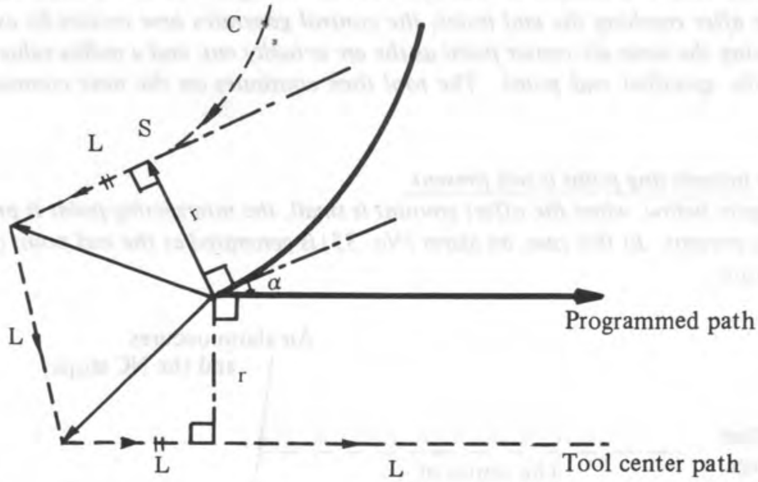


Linear → Circular

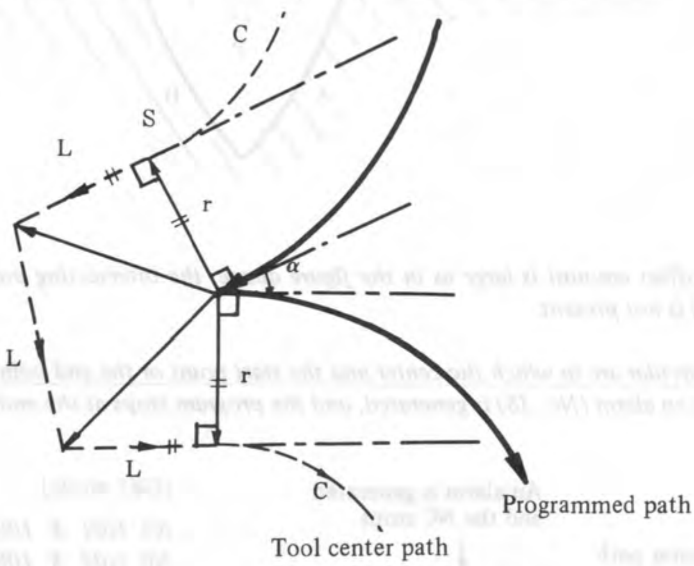


102

Circular → Linear

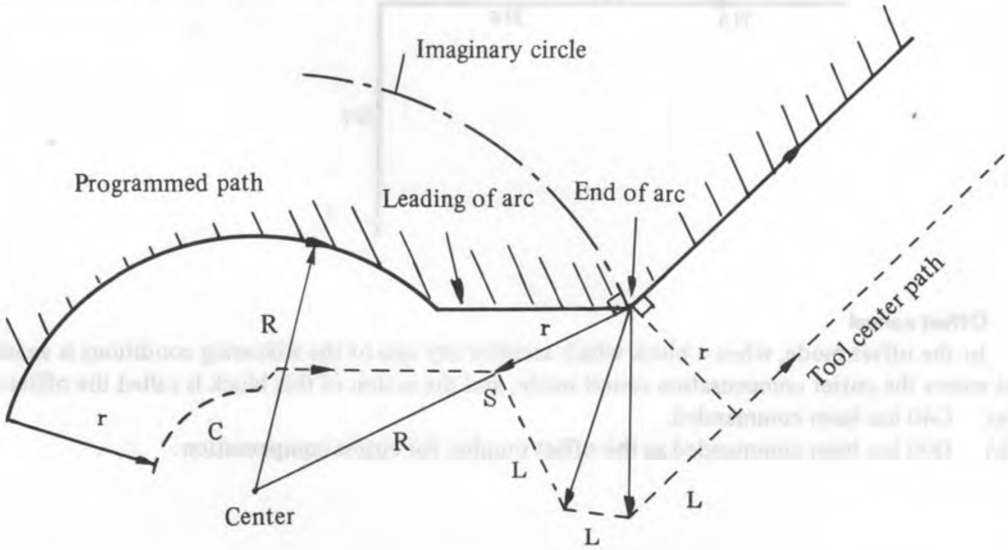


Circular → Circular



(Note 1) *Exceptional case*

When the end point of an arc is not on the arc.

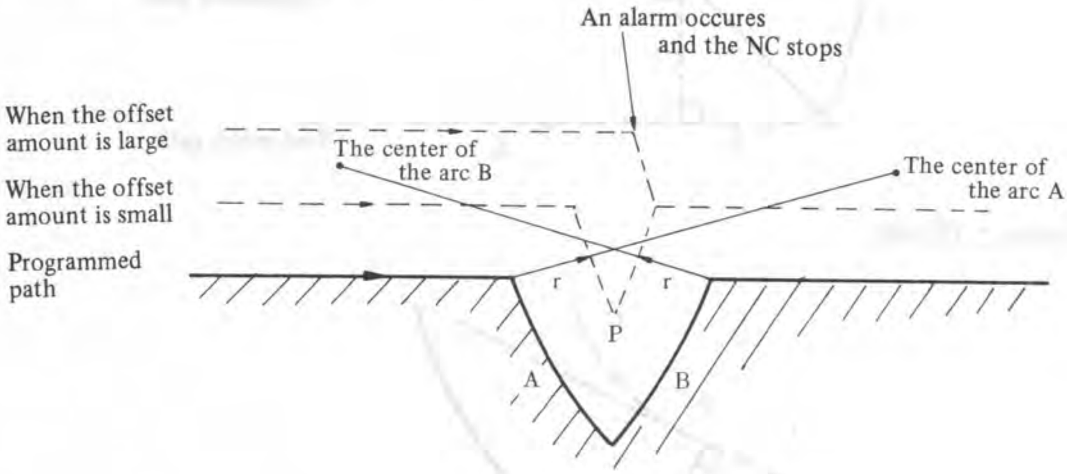


103

When the end point specified in the program is not on the arc formed by the specified radius as it is swung around the center, a straight lead-in from the arc being cut to the specified arc end point is formed as is shown above after reaching the end point, the control generates new vectors by assuming that the tool had cut the arc having the same arc center point as the arc actually cut, and a radius value of the distance from the arc center to the specified end point. The tool then continues on the next command by following normal procedures.

When an inside intersecting point is not present

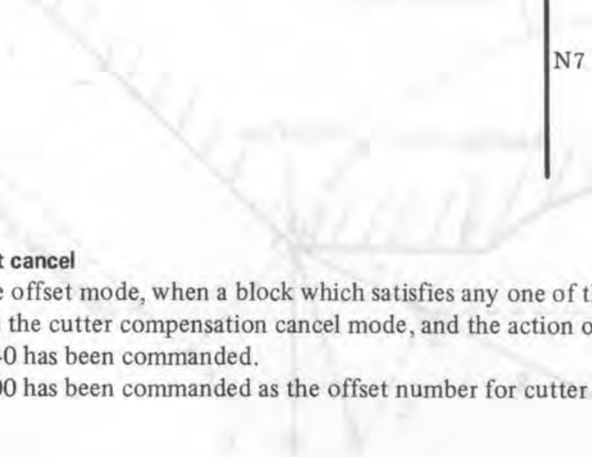
In the figure below, when the offset amount is small, the intersecting point is present. However, if large, it is not always present. In this case, an alarm (No. 33) is generated at the end point of the previous block and the program stops.



When the offset amount is large as in the figure above, the intersecting point as that in the small offset amount (point P) is not present.

In the case of a circular arc in which the center and the start point or the end point coincide

In this case an alarm (No. 38) is generated, and the program stops at the end point of the previous block.



(G41 mode)

N5 G01 X 1000 ;
N6 G02 X 1000 I0 J0 ;
N7 G03 Y-1000 J-1000 ;

(4) Offset cancel

In the offset mode, when a block which satisfies any one of the following conditions is executed, the equipment enters the cutter compensation cancel mode, and the action of this block is called the offset cancel.

(a) G40 has been commanded.

(b) D00 has been commanded as the offset number for cutter compensation.

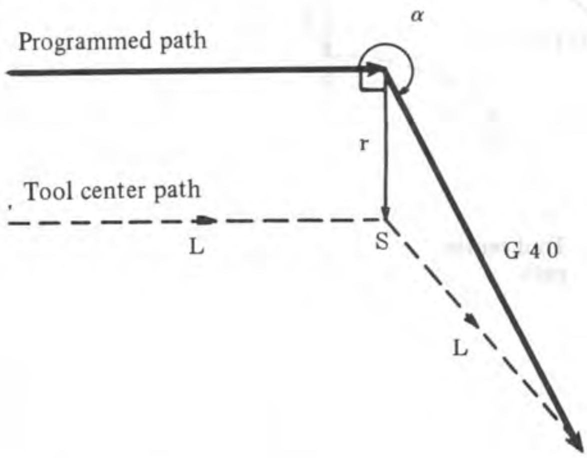
When performing offset cancel, circular arc commands (G02 and G03) are not available. If a circular arc is commanded, an alarm (No. 34) is generated and the NC stops.

In the offset cancel, the control executes the instructions in that block and the block in the cutter compensation buffer. In the meantime, in the case of a single block mode, after reading one block, the control executes it and stops. By pushing the cycle start button once more, one block is executed without read of the next block.

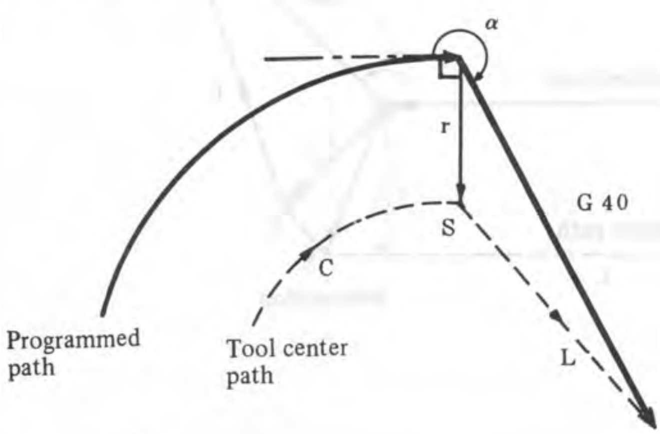
Then the control is in the cancel mode, and normally, the block to be executed next will be stored in the buffer register and the next block is not read into the buffer for cutter compensation.

(a) When going around an inside corner ($\alpha \geqq 180°$)

Linear → Linear



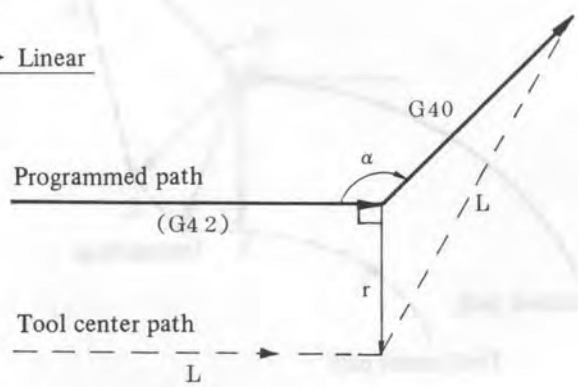
Circular → Linear



(b) When going around an outside corner at an obtuse angle ($90° \leqq \alpha < 180°$)

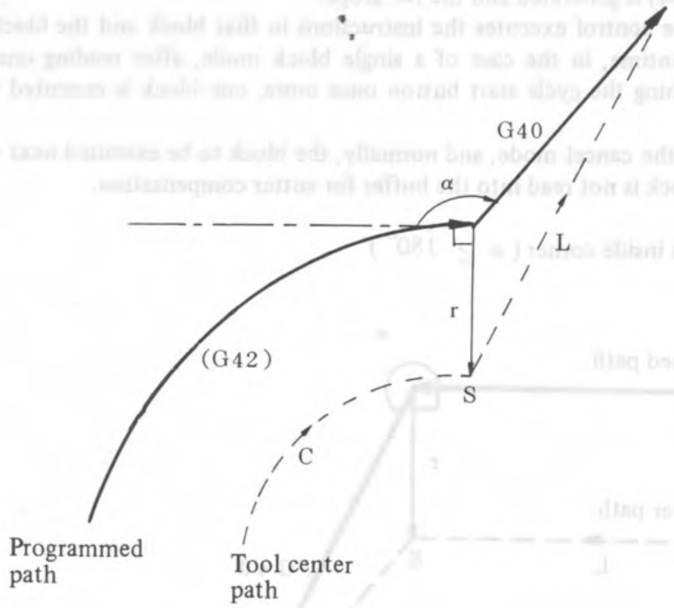
(i) Type A

Linear → Linear



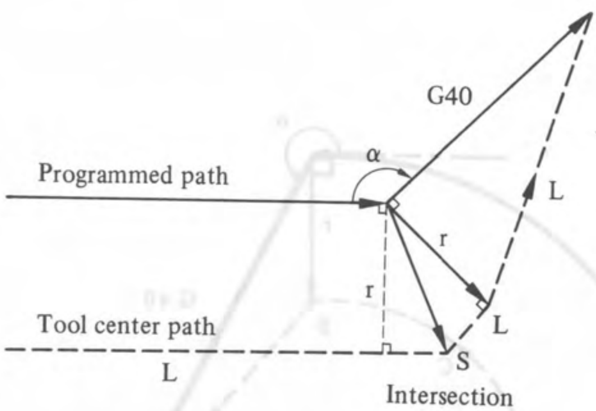
105

Circular → Linear

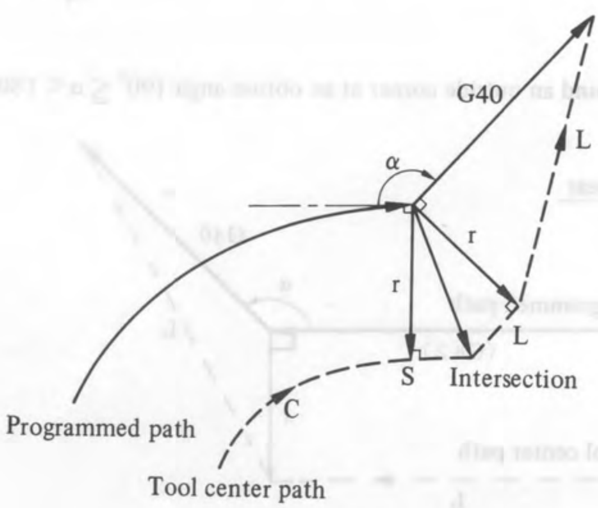


(ii) Type B

Linear → Linear



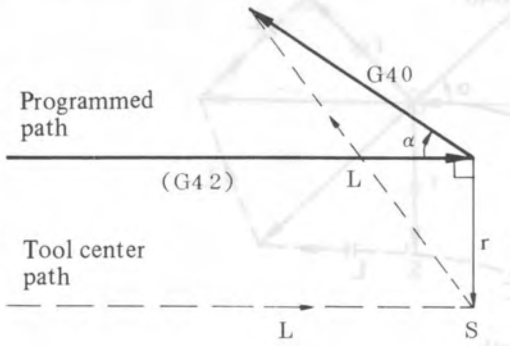
Circular → Linear



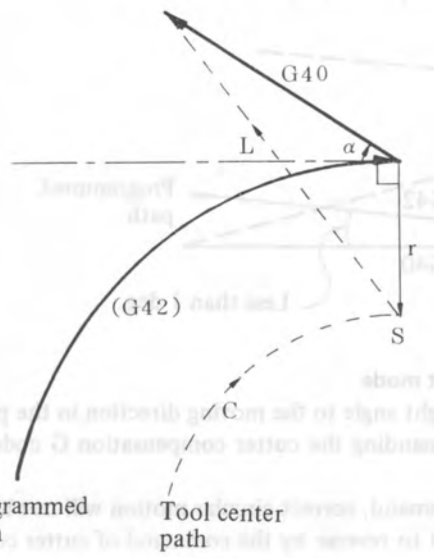
(c) When going around an outside corner at an acute angle ($\alpha < 90^\circ$)

(i) Type A

Linear → Linear

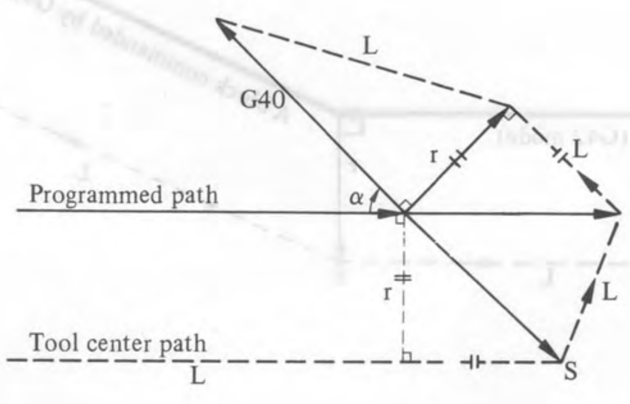


Circular → Linear



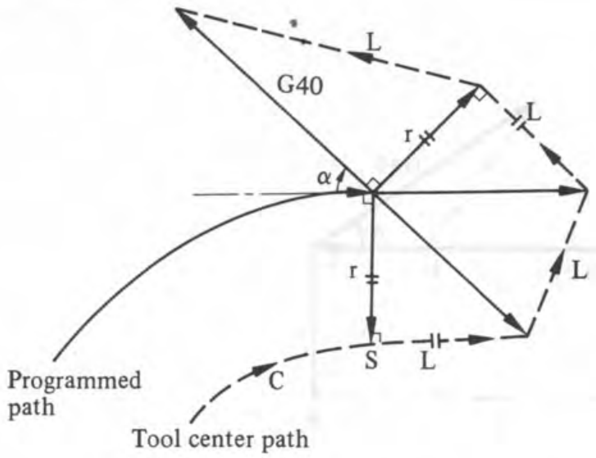
(ii) Type B

Linear → Linear

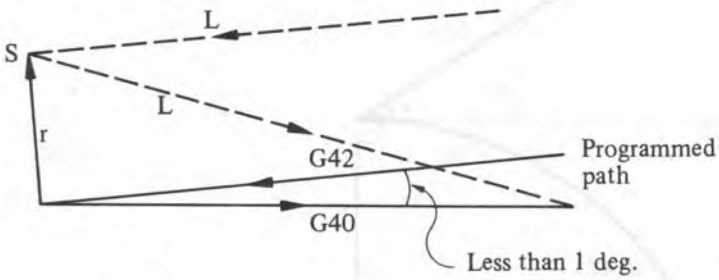


107

Circular → Linear



(Note) In the case of type B, when the tool goes around the outside linear → linear at an acute angle less than 1 degree, compensation is performed as follows.



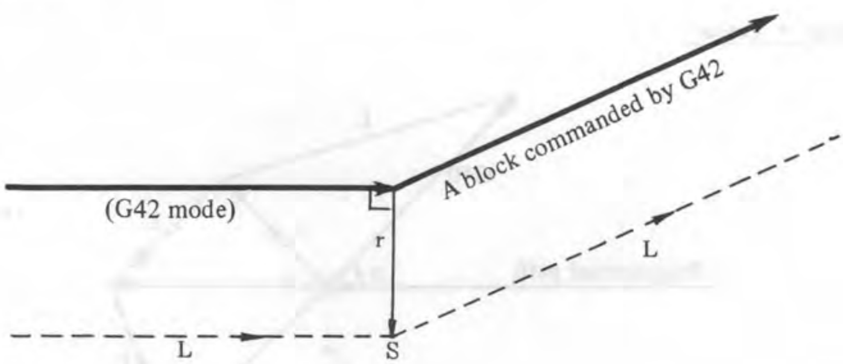
(5) Cutter compensation G code in the offset mode

The offset vector can be set to form a right angle to the moving direction in the previous block, irrespective of machining on inner or outer wall, by commanding the cutter compensation G code (G41, G42) in the offset mode, independently.

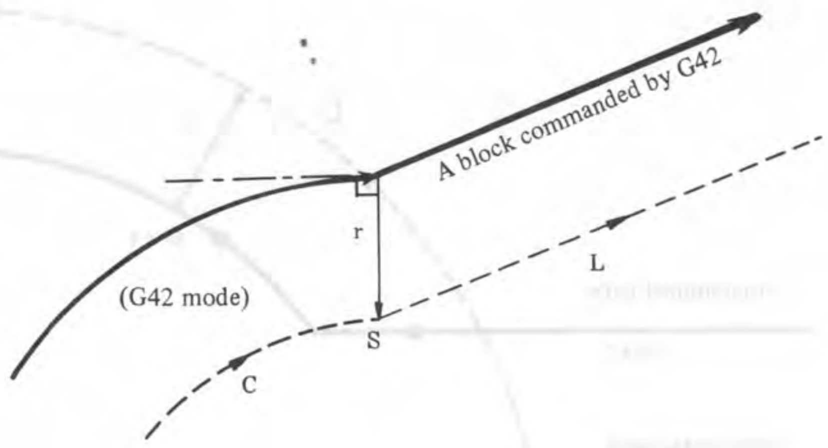
If this code is contained in a circular command, correct circular motion will not be obtained.

When the direction of offset is expected to reverse by the command of cutter compensation G code (G41, G42), refer to Note 2. "Change of offset directions in offset mode."

Linear → Linear



Circular → Linear



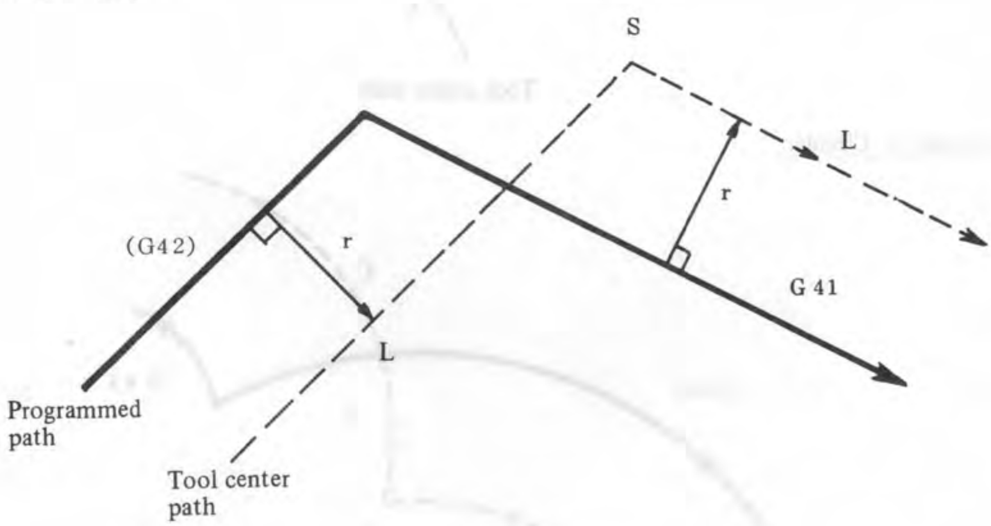
(Note 2) Change of offset direction while in the offset mode

The offset direction is decided by G codes (G41 and G42) for cutter compensation and the sign of offset amount is as follows.

G code \ *Single of offset amount*	*+*	*–*
G41	*Left side offset*	*Right side offset*
G42	*Right side offset*	*Left side offset*

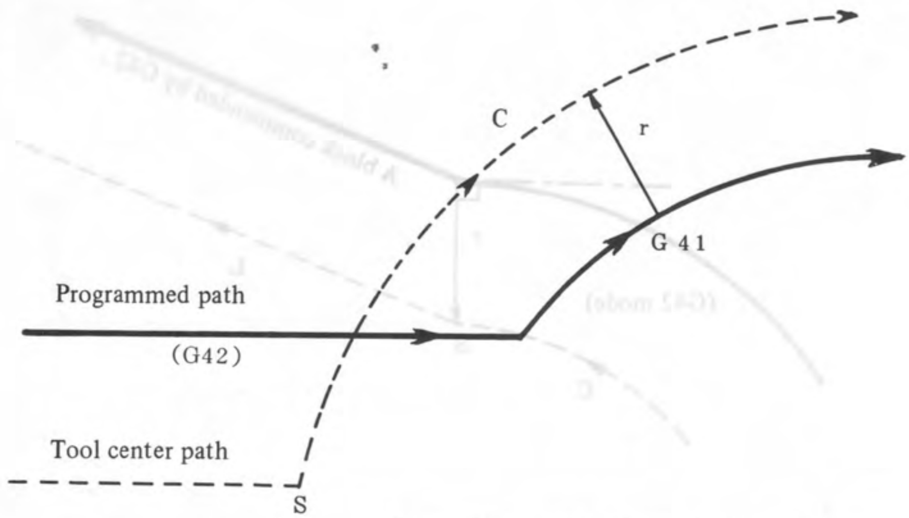
In a special case, the offset direction may be changed in the offset mode. However, the change is not available in the start up block and block following it. When the offset direction is changed, the conception of inside and outside disappears to become common for all cases. What will be described hereafter is the case when the offset direction is changed by G41 and G42. The offset amount in the following example is assumed to be positive.

Linear → Linear

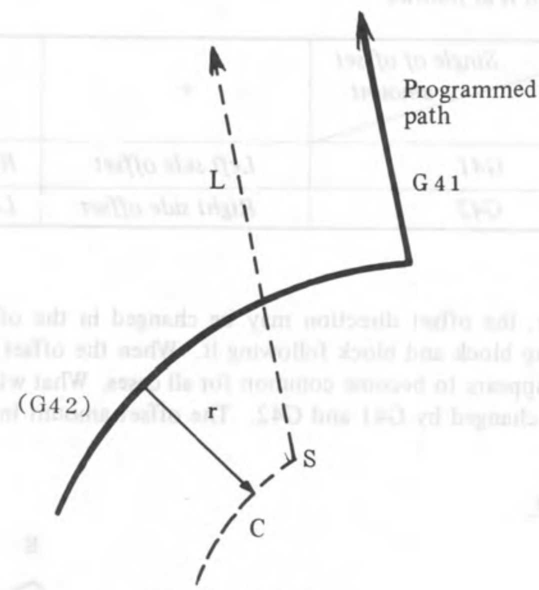


109

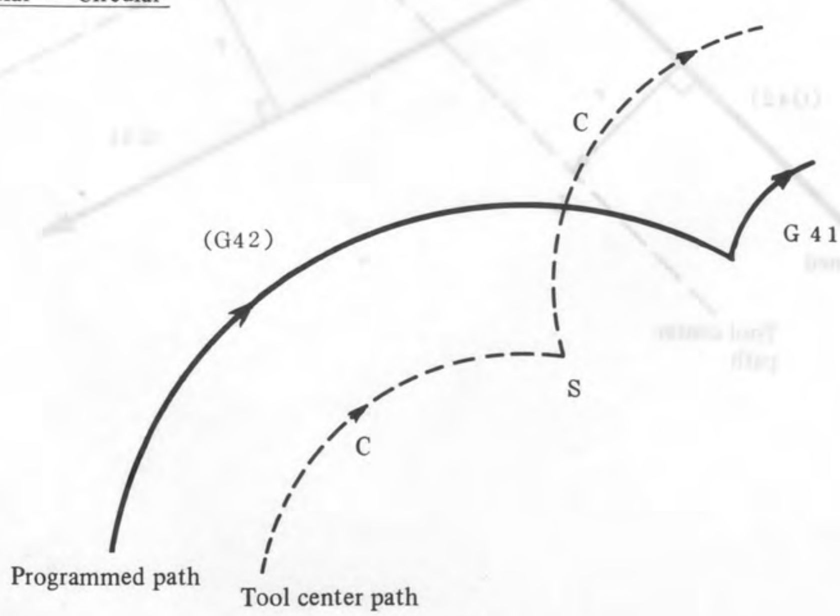
Linear → Circular



Circular → Linear



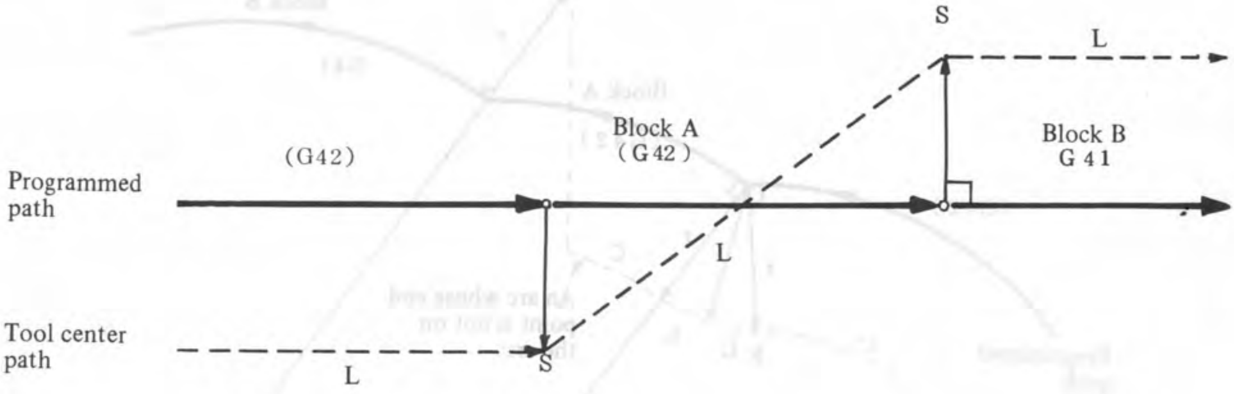
Circular → Circular

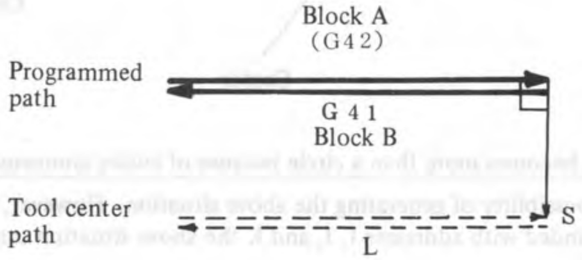


When an intersection is not required if offset is normally performed.

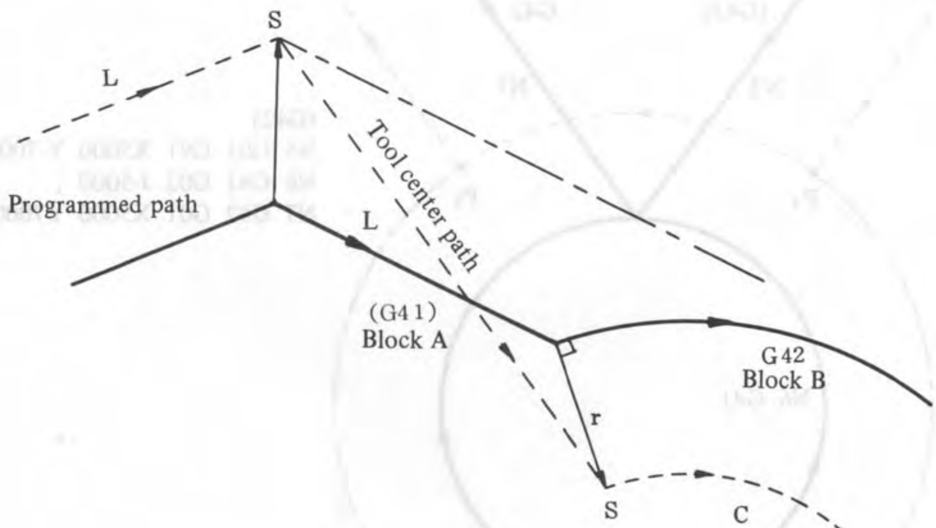
When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.

(a) Linear → Linear

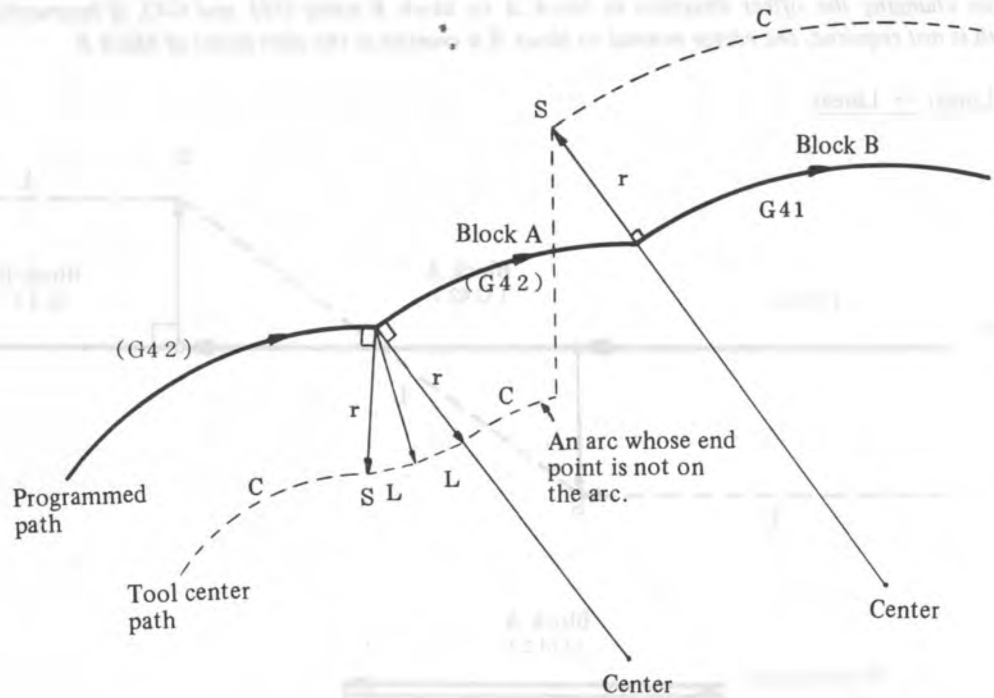




(b) Linear → Circular

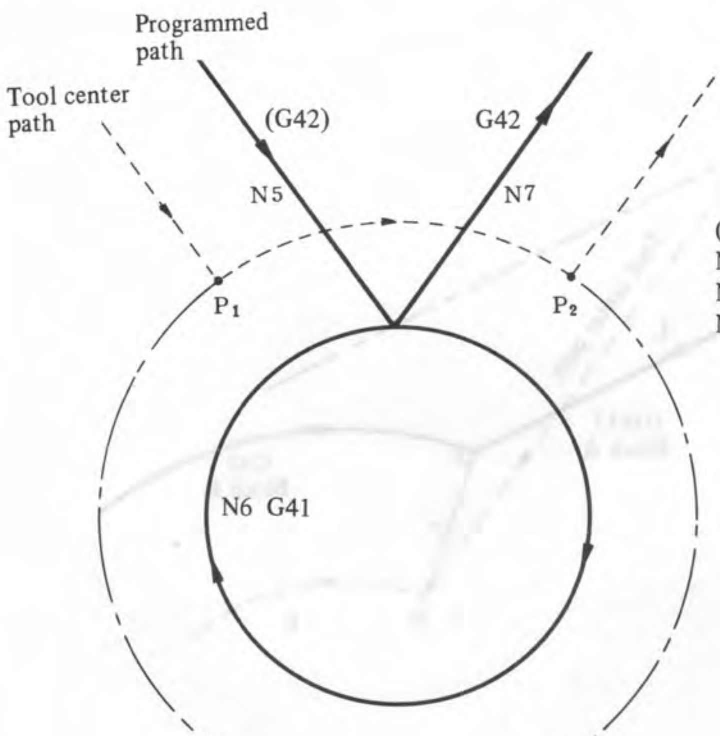


(c) Circular → Circular



When the length of tool center path becomes more than a circle because of cutter compensation

Normally there is almost no possibility of generating the above situation. However, when G41 and G42 are changed, or when a G40 was commanded with addresses I, J, and K the above situation can occur.



(G42)
N5 G01 G91 X5000 Y-7000 ;
N6 G41 G02 J-5000 ;
N7 G42 G01 X5000 Y7000 ;

112

In the above case, the tool center path does not go around a circle but moves only from point P1 to P2 along an arc.

According to under some circumstance, an alarm may be generated by the interference check described below. If it is desired to move a tool around a circle, a circle must be commanded with partitions.

(6) Temporary offset cancel

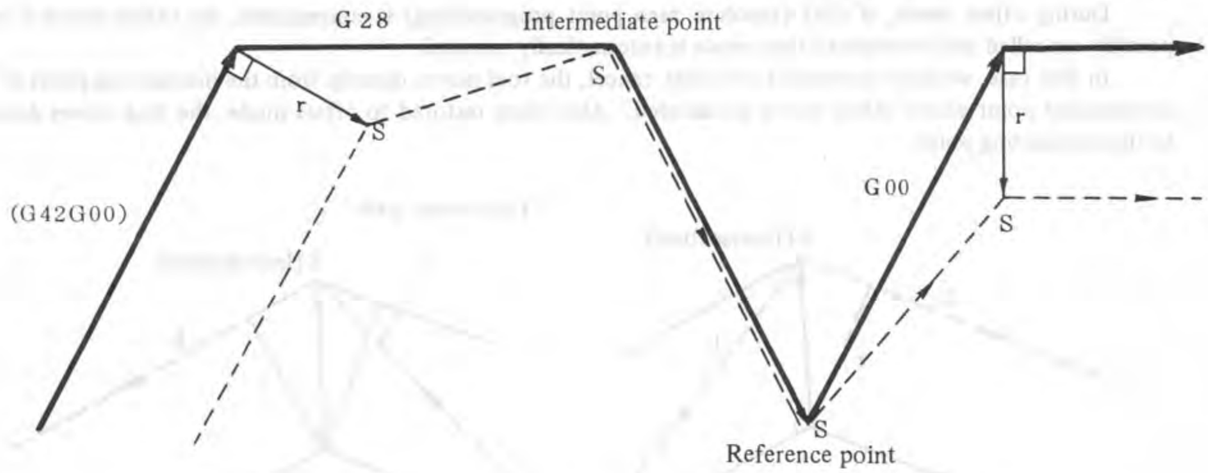
If the commands below are specified in the offset mode, a temporary offset cancel is actuated and thereafter the system will restore the offset mode automatically.

Refer to 6.3.6 (4) Offset cancel and 6.3.6 (2) Start-up, for detail of these operations.

(a) G28 automatic return to reference point

If G28 is commanded in the offset mode, the offset will be cancelled at the intermediate point, and the offset mode will be automatically restored after the reference point is reached.

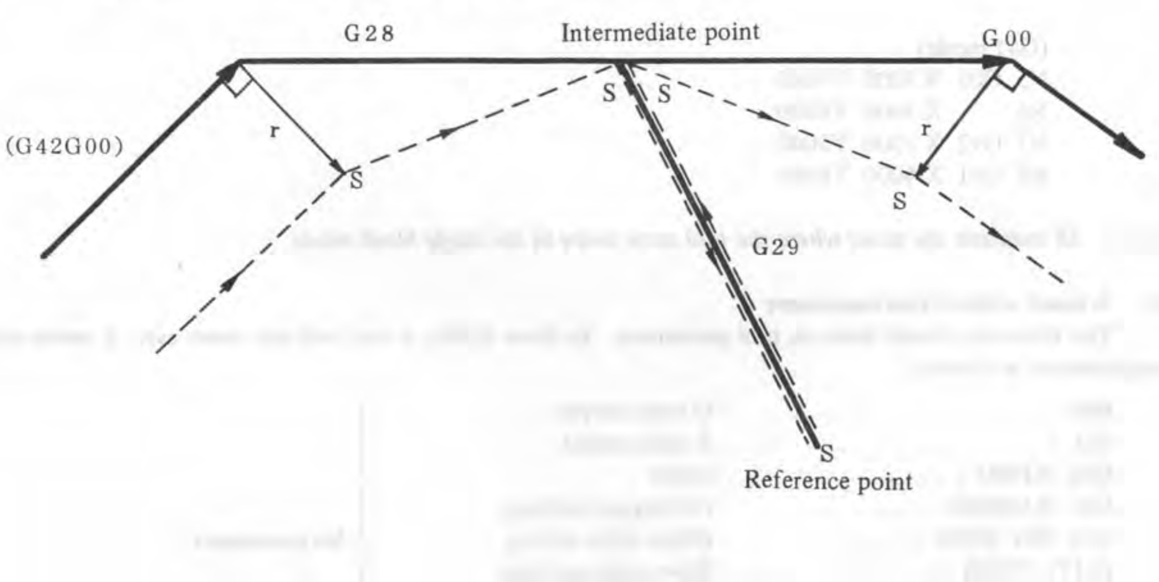
If the offset vector remains at the reference point return in such a case as the current position being the intermediate point, the NC makes it to zero for each axis of which reference point return was completed.



(b) G29 automatic return from reference point

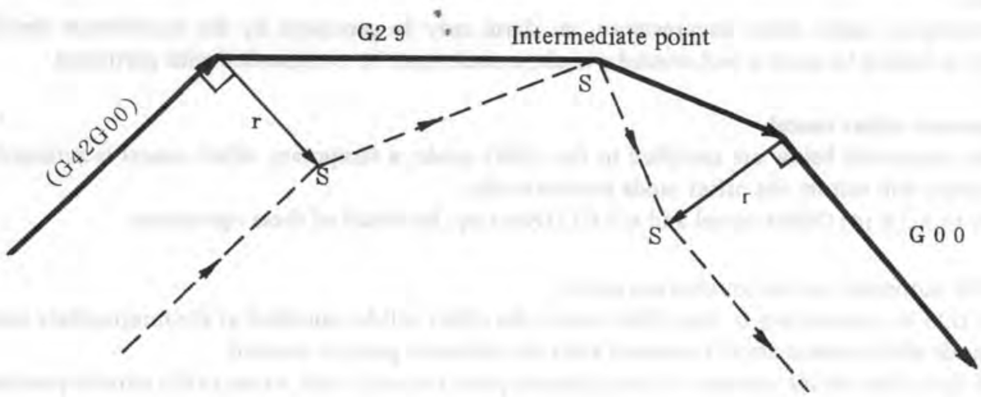
If G29 is commanded in the offset mode, the offset will be cancelled at the intermediate point, and the offset mode will be restored automatically from the subsequent block.

<u>When commanded immediately after G28</u>



113

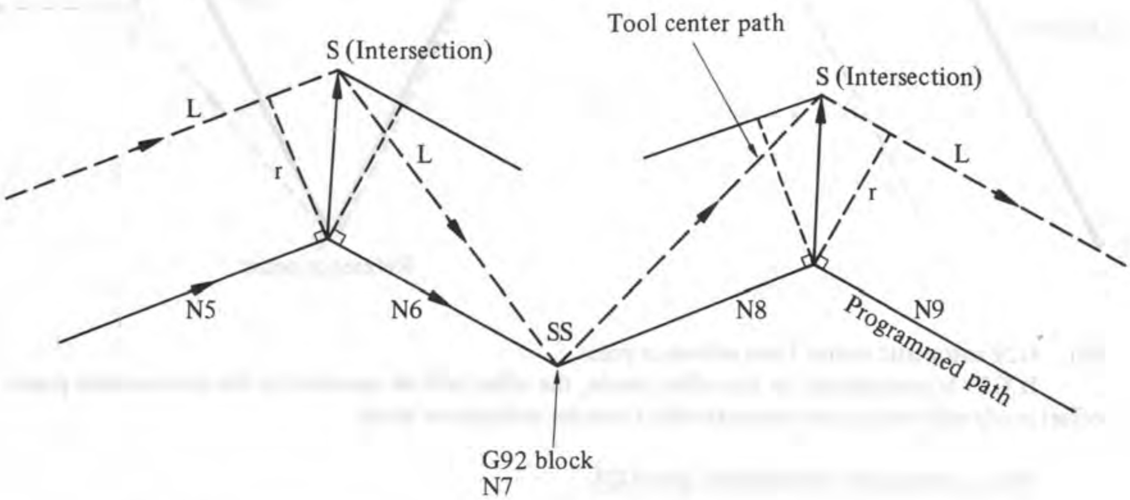
When G29 command is not immediately after G28



(7) Command for temporary cancel of offset vector

During offset mode, if G92 (absolute zero point programming) is commanded, the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.

In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled. Also when restored to offset mode, the tool moves directly to the intersecting point.



(G41 mode)
N5 G01 X 3000 Y7000 ;
N6 X-3000 Y6000 ;
N7 G92 X 1000 Y2000 ;
N8 G01 X 4000 Y8000 ;

(Note) SS indicates the point where the tool stops twice in the single block mode.

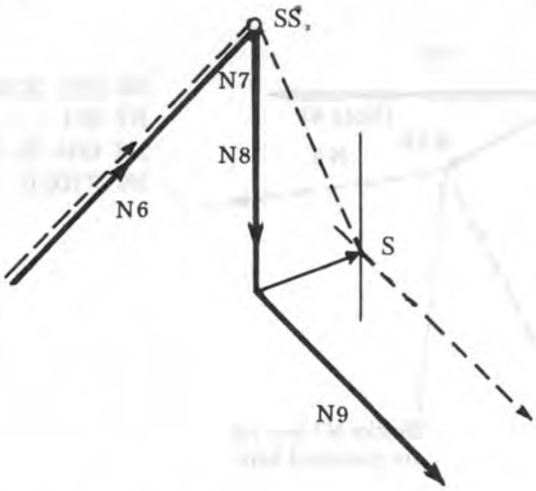
(8) A block without tool movement

The following blocks have no tool movement. In these blocks, a tool will not move even if cutter radius compensation is effected.

M05 ;	M code output	No movement
S21 ;	S code output	
G04 X1000 ;	Dwell	
G22 X100000 ;	Cutting area setting	
G10 P01 X100 ;	Offset value setting	
(G17) Z2000 ;	Move command not included in the offset plane.	
G90 ;	G code only	
G91 X0 ;	Move distance is zero.	

(a) When commanded at start-up

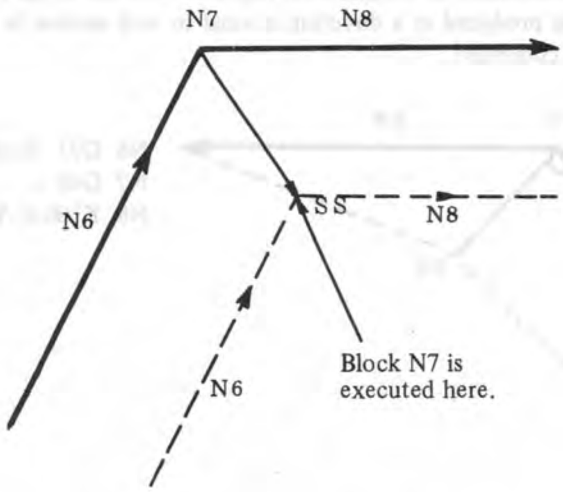
If a block without tool movement is commanded at start-up, the offset vector is not produced.




```
G40 G91 ..........
 .
 .
 .
N6 X1000.0 Y1000.0 ;
N7 G41 X0 ;
N8 Y-1000.0 ;
N9 X1000.0 Y-1000.0 ;
```

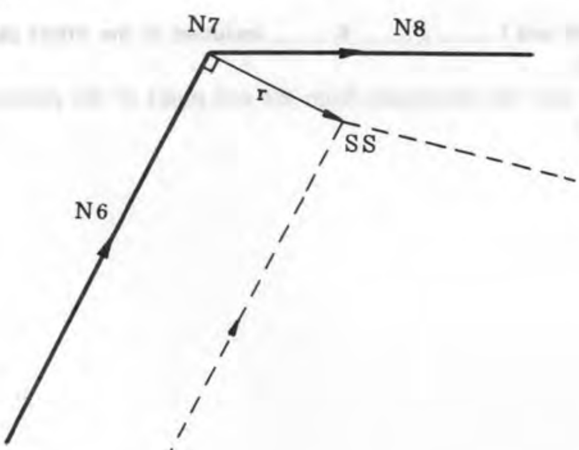
(b) When commanded in offset mode

When a single block without tool movement is commanded in the offset mode, the vector and tool center path are the same as those when the block is not commanded. (Refer to item 6.3.6 (3) Offset mode) This block is executed at the single block stop point.



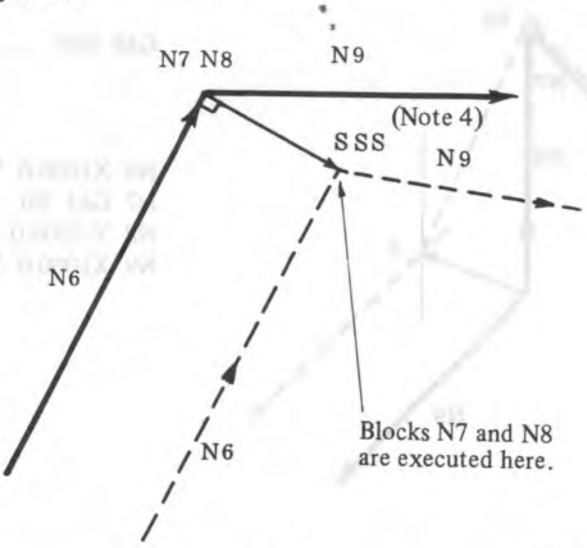

```
N6 G91 X100.0 Y200.0 ;
N7 G04 X100.0 ;
N8 X100.0 ;
```

However, when the move distance is zero, even if the block is commanded singly, tool motion becomes the same as that when more than one block of without tool movement are commanded, which will be described subsequently.




```
N6 G91 X1000 Y2000 ;
N7 X0 ;
N8 X1000 ;
```

Two blocks without tool movement should not be commanded consecutively. If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier block, so overcutting may result.

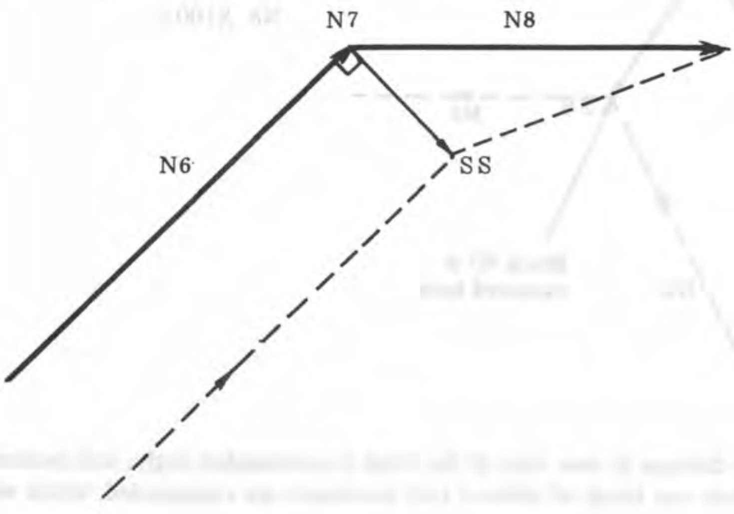


N6 G91 X100.0 Y200.0 ;
N7 S21 ;
N8 G04 X1.0 ;
N9 X100.0 ;

(Note 4)   SSS means that tool stops three times by single block operation.

(c) When commanded together with offset cancel

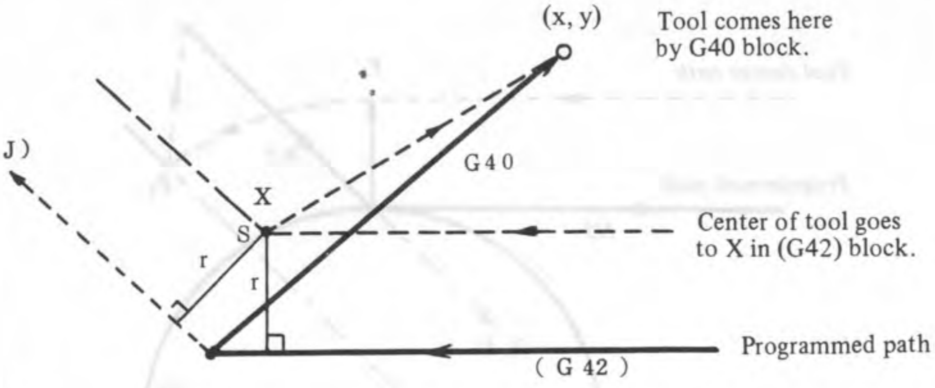
When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a direction normal to tool motion in the earlier block, the vector is cancelled in the next move command.



N6 G91 X100.0 Y100.0 ;
N7 G40 ;
N8 X100.0 Y0 ;

(9) When a block contains a G40 and I ____ J ____ K ____ included in the offset plane and the mode of the previous block is G41 or G42

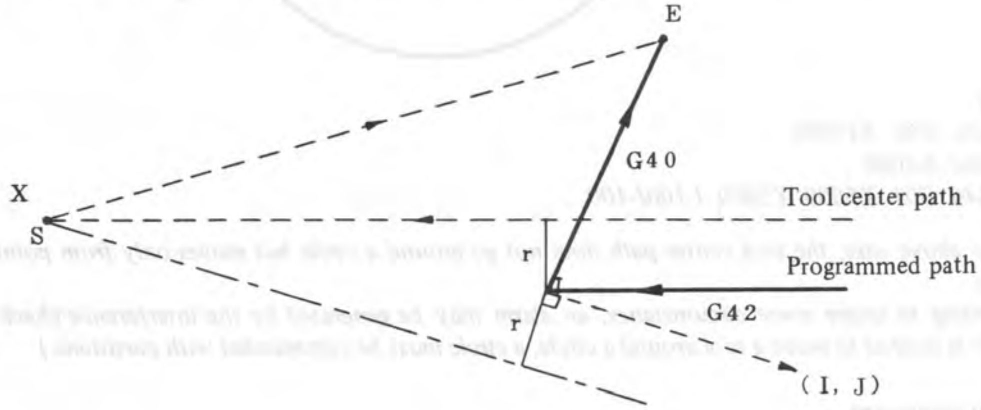
In this case the NC assumes that the movement from the end point of the previous block in the direction of (I, J or K) has been commanded.



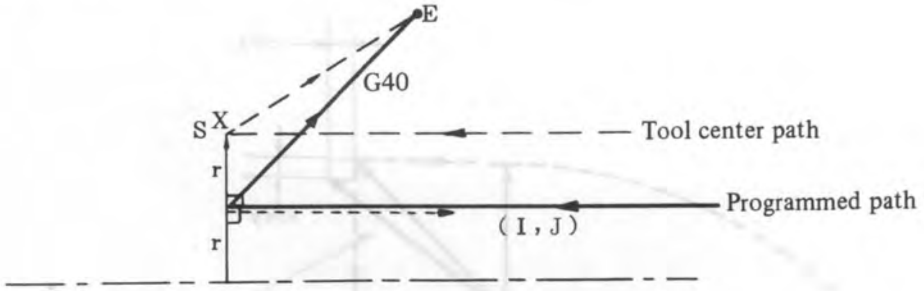
(G42 mode)

G40X ___x___ Y ___y___ I ______ J ______;

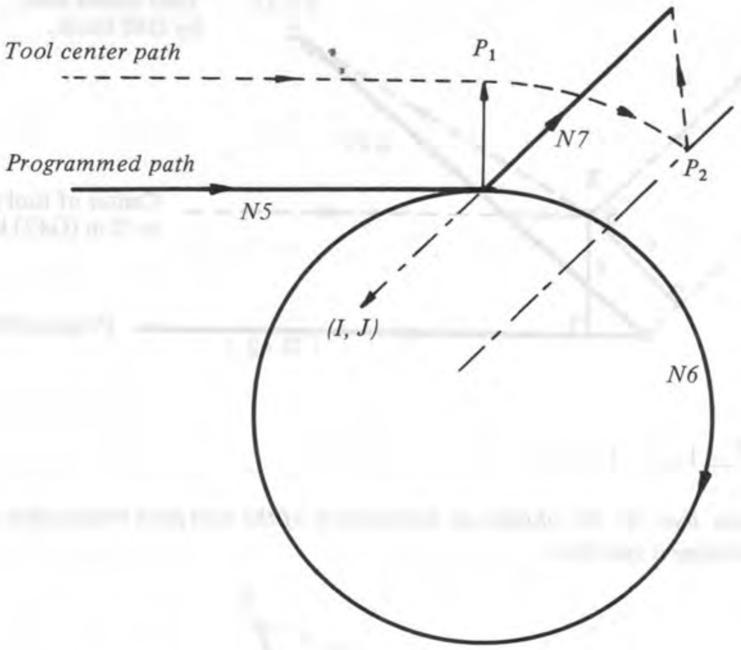
(Note 5) In this case, note that the NC obtains an intersection of the tool path irrespective of whether inner or outer wall machining is specified.



(Note 6) When an intersection is not obtainable, the tool comes to the position normal to the previous block at the end of the previous block.



(Note 7) When the length of the tool center path becomes more than a circle.



(G41)
N5 G01 G91 X10000 ;
N6 G02 J-6000 ;
N7 G40 G01 X5000 Y5000 I-100J-100 ;

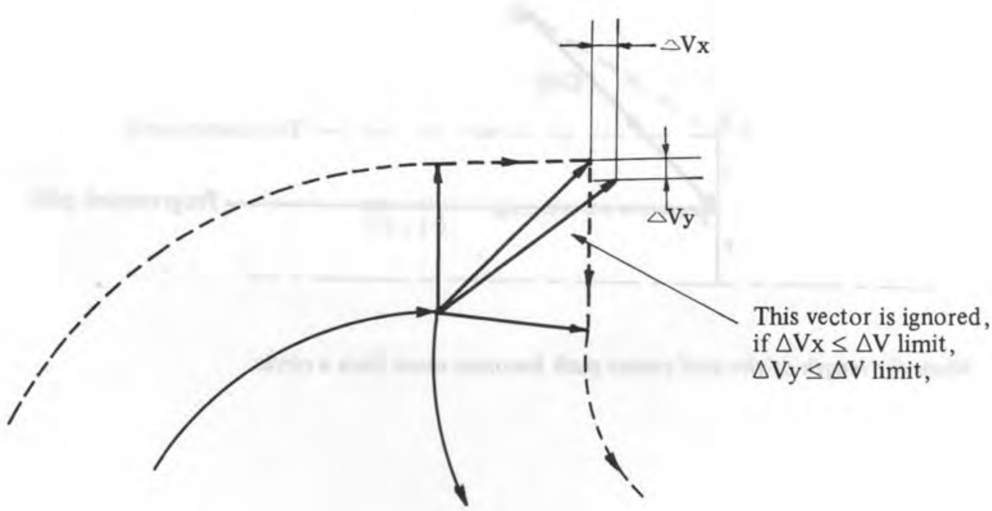
In the above case, the tool center path does not go around a circle but moves only from point P1 to P2 along an arc.

According to under some circumstance, an alarm may be generated by the interference check described below. (If it is desired to move a tool around a circle, a circle must be commanded with partitions.)

(10) Corner movement

When two or more vectors are produced at the end of a block, the tool moves linearly from one vector to another.

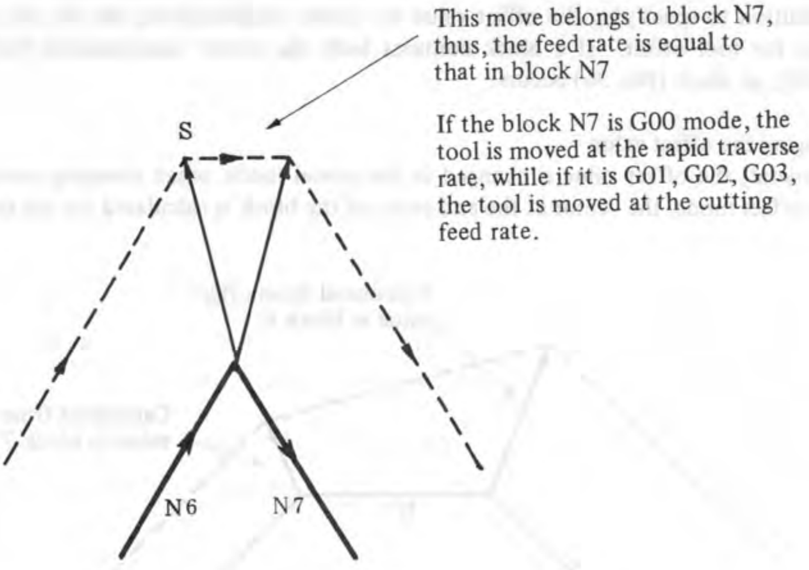
If these vectors almost coincide with each other, the corner movement isn't performed and the latter vector is ignored.



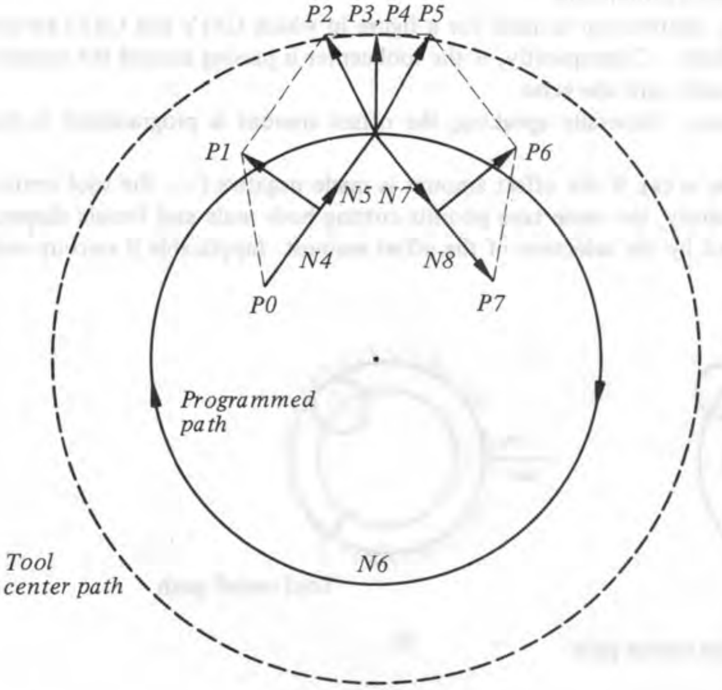
118

If $\Delta Vx < \Delta V_{limit}$ and $\Delta Vy < \Delta V_{limit}$, the latter vector is ignored. The ΔV_{limit} is set in advance by parameter (CRCDL).

If these vectors do not coincide, a move is generated to turn around the corner. This move belongs to the latter block.



(Note 8) However, if the path of the next block is semicircular or more, the above function is not performed. The reason for this is as follows:



N4 G41 G91 G01 X1500 Y2000 ;
N5 X1500 Y2000 ;
N6 G02 J-6000 ;
N7 G01 X1500 Y-2000 ;
N8 G40 X1500 Y-2000 ;

If the vector is not ignored, the tool path is as follows:

P0 → P1 → P2 → P3 → (Circle) → P4 → P5 → P6 → P7

But if the distance between P2 and P4 is negligible, the point P3 is ignored. Therefore, the tool path is as follows:

P0 → P1 → P2 → P4 → P5 → P6 → P7

Namely, circle cutting by the block N6 is ignored.

(11) General precatuions on offset

(a) Offset value command

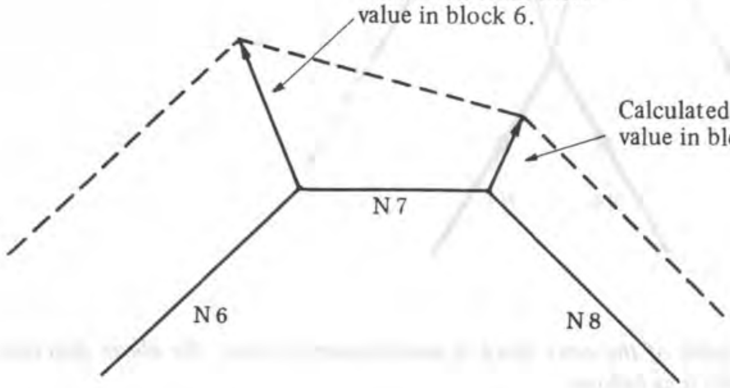
The offset value is commanded by a D code which specifies an offset value number.

Once commanded, a D code remains valid until another D code is commanded or cleared.

In addition to specifying the offset value for cutter compensation, the D code is also used to command offset value for tool offset. If a block contains both the cutter compensation (G41/G42) and tool offset (G45 to G48), an alarm (No. 36) occurs.

(b) Changing the offset value

In general, the offset value is changed in the cancel mode, when changing tools. If the offset value is changed in offset mode, the vector at the end point of the block is calculated for the new offset value.

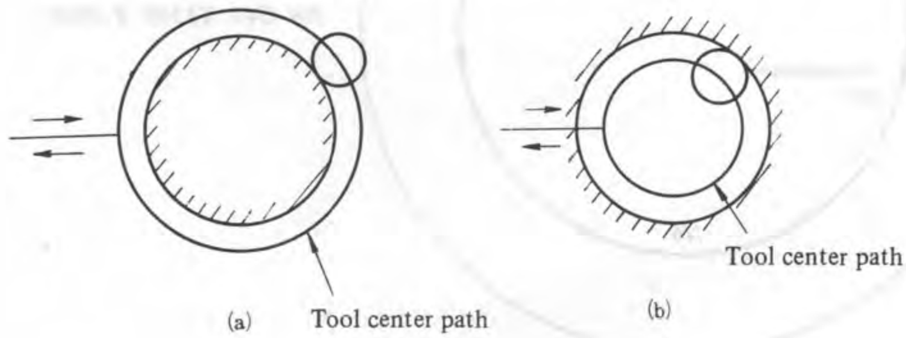


(c) Positive/negative offset amount and tool center path

If the offset amount is negative (–), distribution is made for a figure in which G41's and G42's are all replaced with each other on the process sheet. Consequently, if the tool center is passing around the outside of the workpiece, it will pass around the inside, and vise versa.

The figure below shows one example. Generally speaking, the offset amount is programmed to be positive (+).

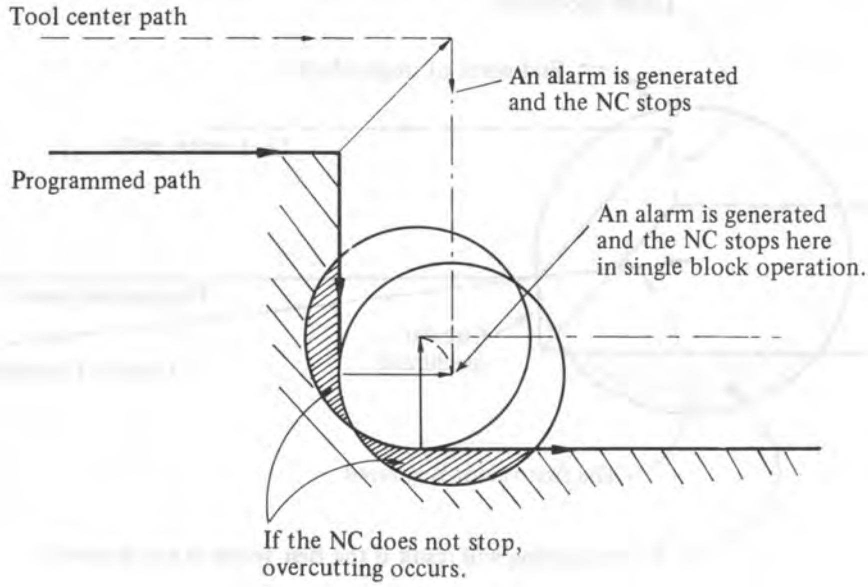
When a tool path is programmed as in (a), if the offset amount is made negative (–), the tool center moves as in (b), and vise versa. Consequently, the same tape permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the offset amount. (applicable if start-up and cancel is A type).



(d) Overcutting by cutter compensation

(i) Machining an inside corner at a radius smaller than the cutter radius

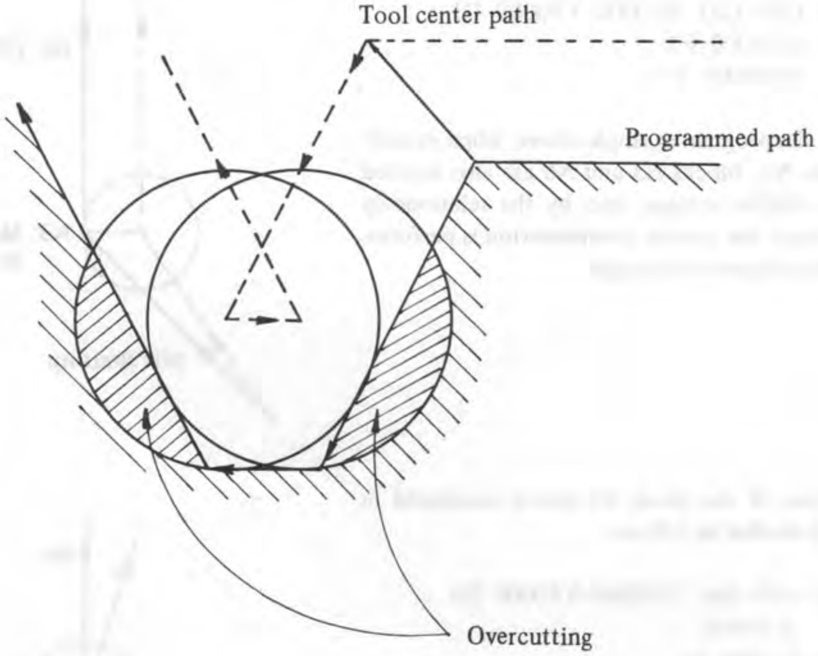
When the radius of a corner is smaller than the cutter radius, because the inner offsetting of the cutter will result in overcuttings, an alarm (No. 41) is generated and the NC stops at the start of the block. In single block operation, the overcutting is generated because the tool is stopped after the block execution. The tool movement described above is the same as in an alarm (No. 41) described below.



(ii) Machining a groove smaller than the tool diameter

Since the cutter compensation forces the path of the center of the tool to move in the reverse of the programmed direction, overcutting will result.

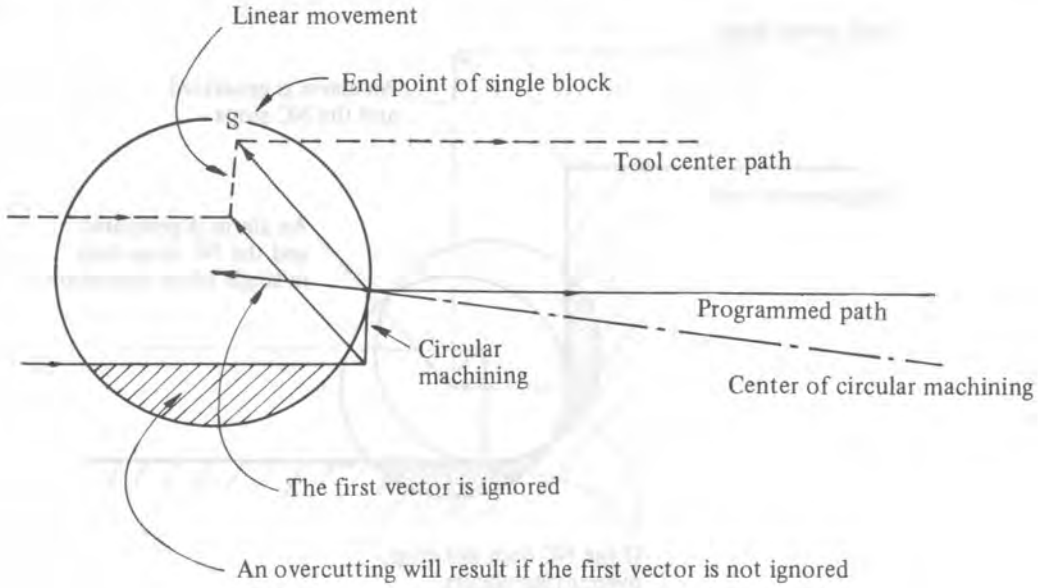
In this case an alarm (No. 41) is generated and the NC stops.



121

(iii) When machining a step smaller than the tool radius

When machining of the step is commanded by circular machining in the case of a program containing a step smaller than the tool radius, the path of the center of tool with the ordinary offset 6.3.6 (3) becomes reverse to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The single block operation is stopped at this point. If the machining is not in the single block mode, the cycle operation is continued.



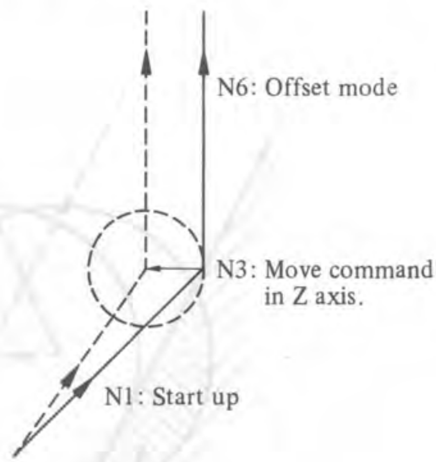
(iv) The start up and the movement in Z axis on the cutter compensation C

It is usually used such a method that the tool is moved along the Z axis after the cutter compensation is effected at some distance from the workpiece at the start of the machining.

In the case above, if it is desired to divide the motion along the Z axis into rapid traverse and cutting feed, follow the procedure below.

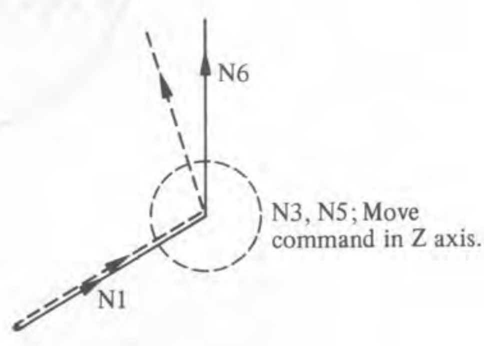
```
N1 G91 G00 G41 X50000 Y50000 D1 ;
N3 G01 Z-30000 F1 ;
N6      Y100000 F2 ;
```

In the program example above, when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the figure to the right.



Then, if the block N3 (move command in Z axis) is divided as follows:

```
N1 G91 G00 G41 X50000 Y50000 D1 ;
N3      Z-25000 ;
N5 G01 Z-5000 F1 ;
N6      Y100000 F2 ;
```



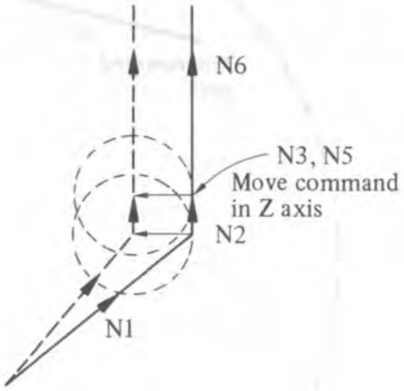
As there are two move command blocks not included in the selected plane and the block N6 cannot be entered into the buffer storage, the tool center path is calculated by the information of N1 in the figure above. That is, in the system 6M, the offset vector is not calculated in start up and the overcutting may result.

The above example should be modified as follow:

The move command in the same direction as that of the move command after the motion in Z axis should be programmed.

```
N1 G91 G00 G41 X50000 Y40000 D1 ;
N2         Y10000 ;
N3         Z-25000 ;
N5 G01 Z-5000 F1 ;
N6         Y100000 F2 ;
```

(The direction of the move command N2 is same as that of N6.)



When executing the block N1, the blocks N2 and N3 are entered into the buffer storage and the correct compensation is performed by the relationship between N1 and N2.

(Note 9) *Interference check*

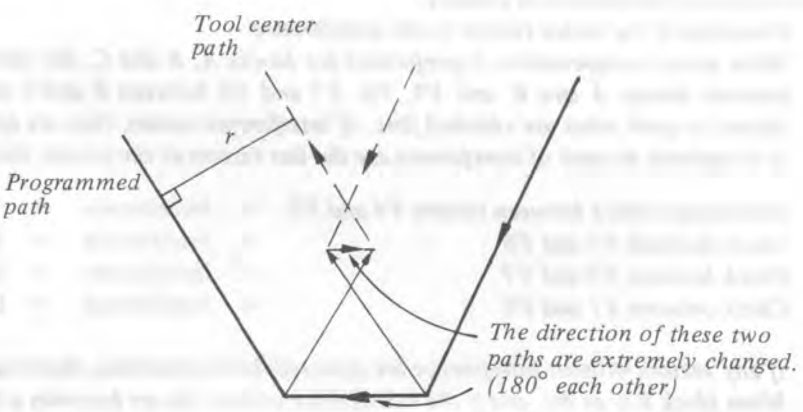
Tool overcutting is called "interference". The function that checks for tool overcutting in advance is the interference check. However, all interferences are not checked by this function. The interference check is performed even if overcutting does not occur.

(1) Reference condition for interference

(a) The direction of the tool center path in cutter compensation is different from that of the programmed path. (from 90 degrees to 270 degrees between these paths).

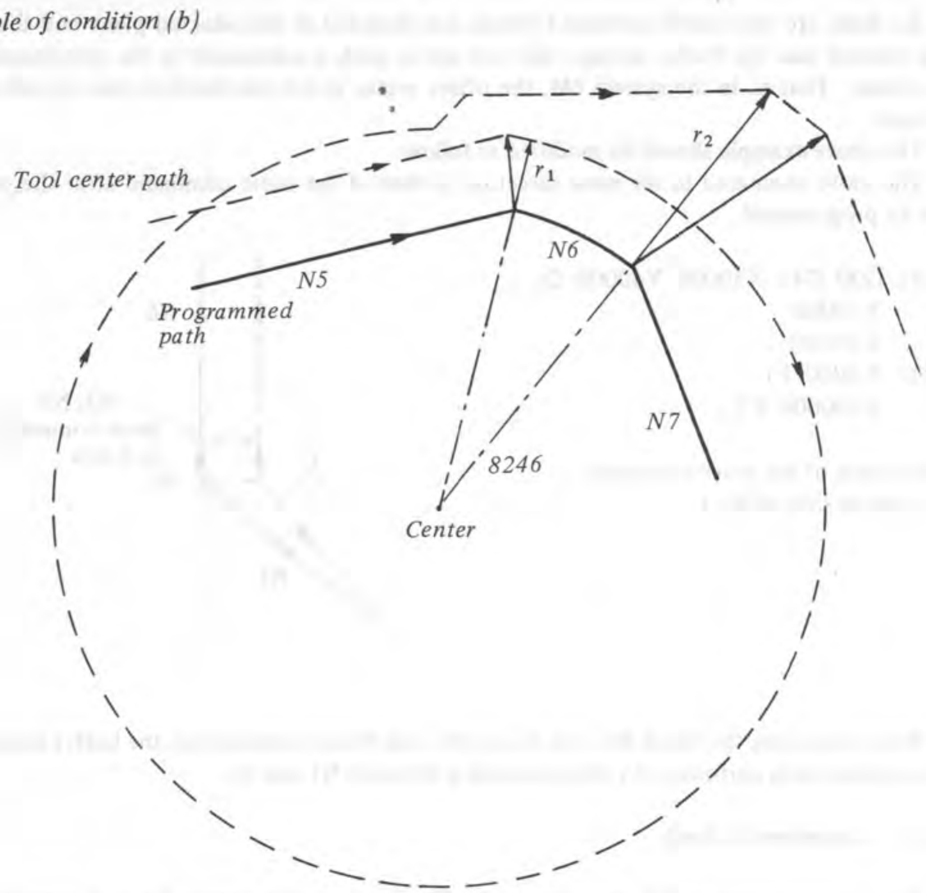
(b) In addition to the above condition (a), the angle between the start point and end point of tool center path is extremely different from that of the programmed path in circular machining. (more than 180 degrees).

Example of condition (a)



123

Example of condition (b)



(G41)
N5 G01 G91 X8000 Y2000 D01 ;
N6 G02 X3200 Y-1600 I-2000 J-8000 D02 ;
N7 G01 X2000 Y-5000 ;

(Offset value corresponding to D01 : r_1 = 2000)
(Offset value corresponding to D02 : r_2 = 6000)

In the above example, the arc in block N6 is placed in the one quadrant. But after cutter compensation, the arc is placed in the four quadrants.

(2) Correction of interference in advance

(a) Removing of the vector related to the interference

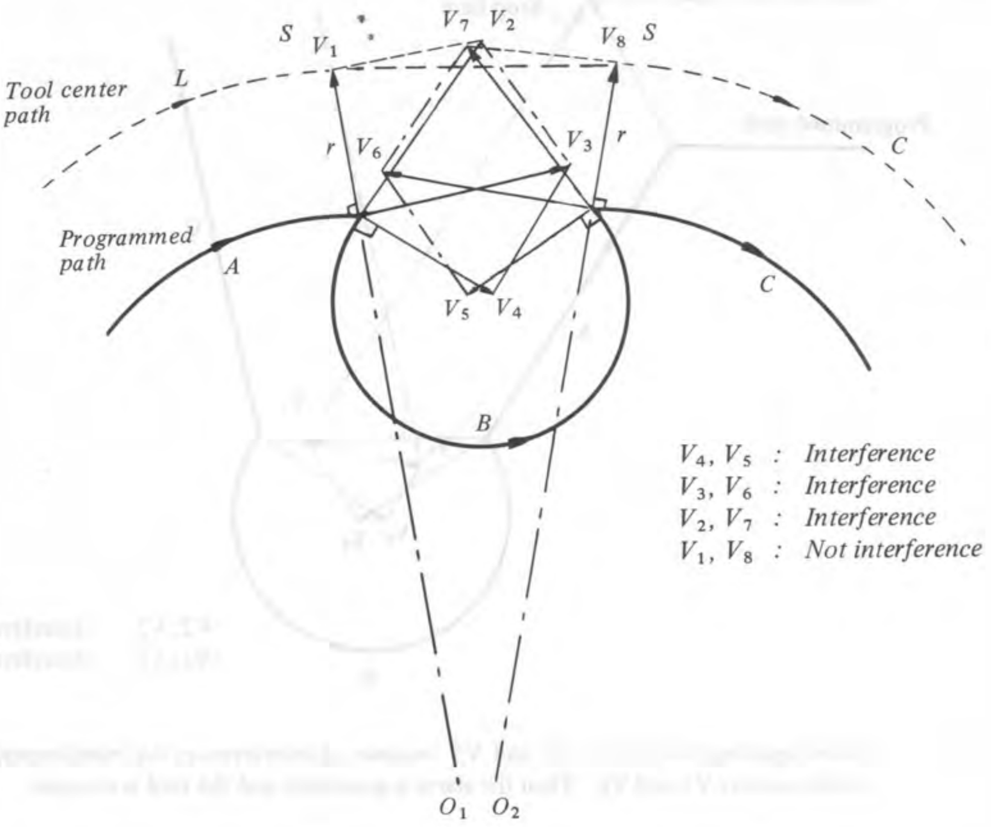
When cutter compensation is performed for blocks A, B and C, the vectors V1, V2, V3 and V4 between blocks A and B, and V5, V6, V7 and V8 between B and C are produced, the vectors closest to each other are checked first. If interference occurs, they are ignored. But if the vectors to be ignored because of interference are the last vectors at the corner, they cannot be ignored.

Interference check between vectors V4 and V5	→	*Interference*	→	*V4 and V5 are ignored*
Check between V3 and V6	→	*Interference*	→	*Ignored*
Check between V2 and V7	→	*Interference*	→	*Ignored*
Check between V1 and V8	→	*Interference*	→	*Cannot be ignored*

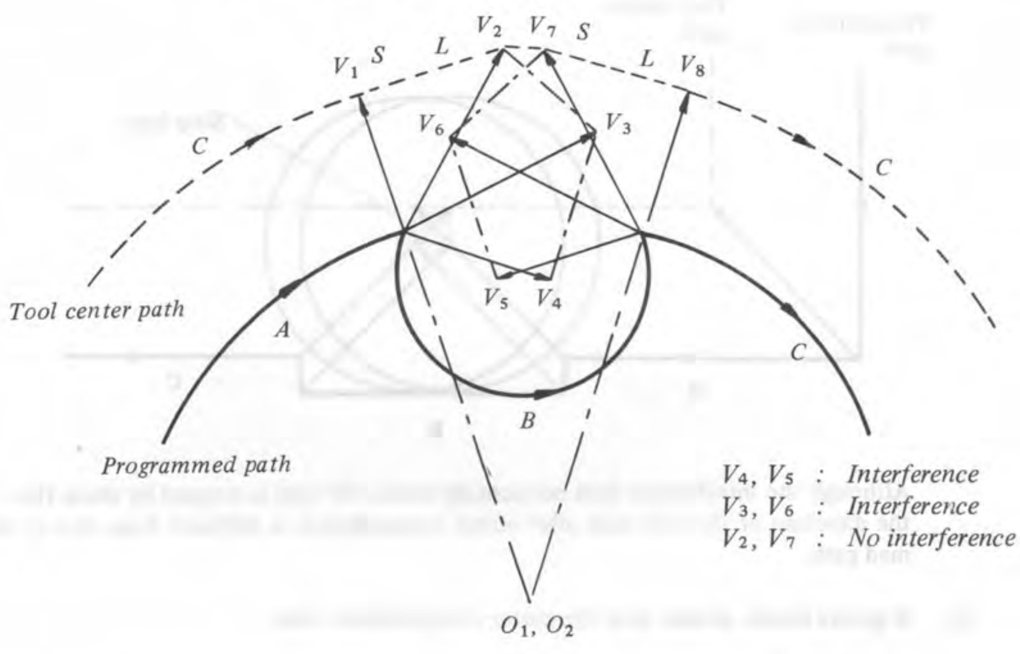
If any vectors without interference are detected during checking, checking is terminated.
When block B is an arc, and if the interference occurs, the arc becomes a linear movement.

124

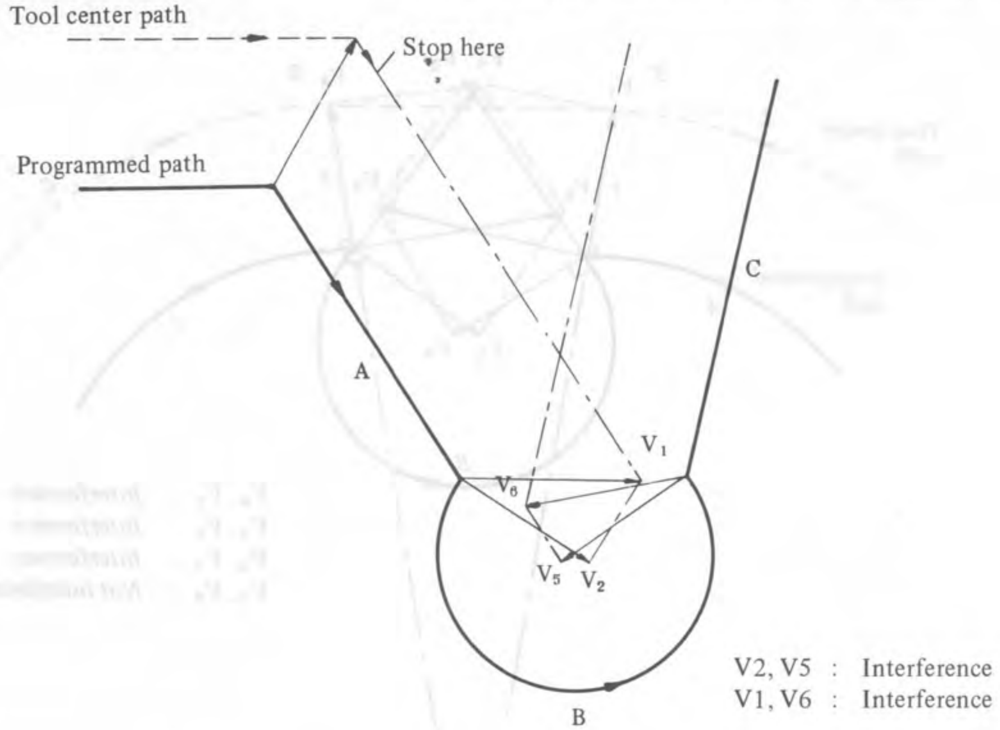
(Example 1) *The tool moves linearly from V1 to V8.*



(Example 2) *The tool moves linearly as follows:*
Tool path: $V_1 \rightarrow V_2 \rightarrow V_7 \rightarrow V_8$



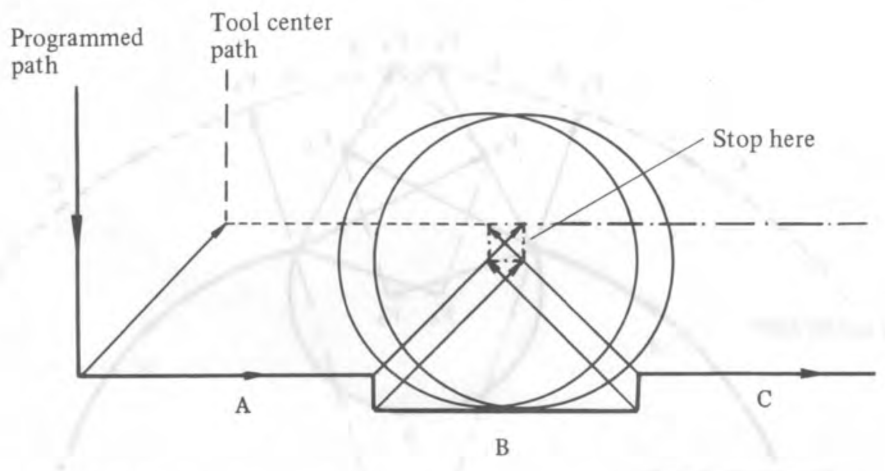
(b) *If the interference occurs after correction (a) the tool is stopped by an alarm.*
If the interference occurs to the last vectors after correction (a) or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the alarm (No. 41) is generated and the tool is stopped just after execution of the previous block.



After ignoring the vector V2 and V5 because of interference, the interference also occurs between vectors V1 and V6. Then the alarm is generated and the tool is stopped.

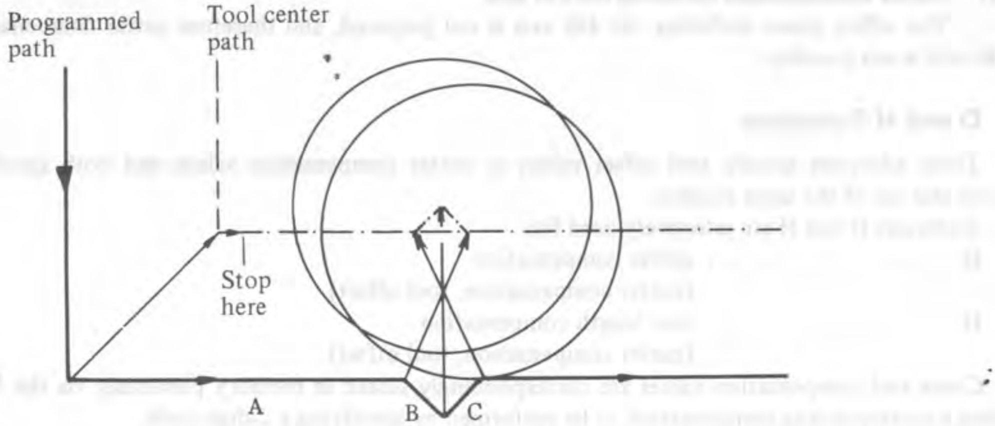
(3) Checking is performed even though no interference actually occurs
There are many examples as follows:

(a) The hollow depth smaller than the cutter compensation value



Although the interference does not actually occur, the tool is stopped by alarm (No. 41) because the direction of the tool path after cutter compensation is different from that of the programmed path.

(b) A groove depth, smaller than the cutter compensation value



The direction of the tool path is different from that of the programmed path the same as example (a).

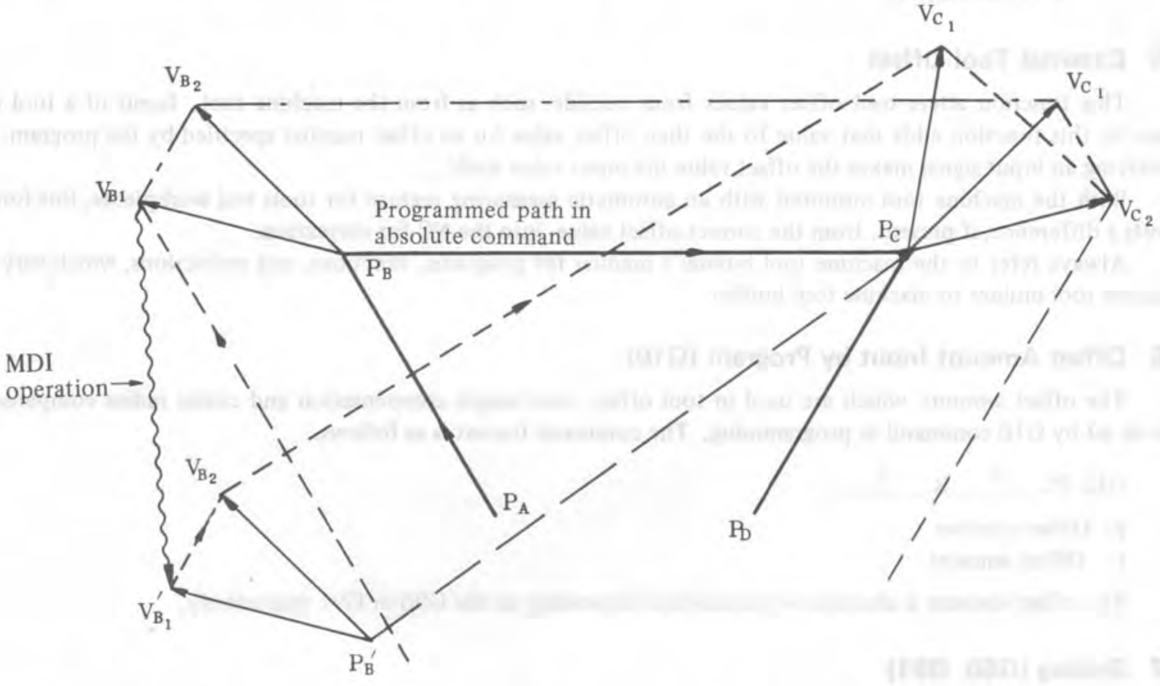
(12) Input command from MDI

Compensation is not performed for commands input from the MDI.

However, when automatic operation using the NC tape composed of absolute commands is temporarily stopped by the single block function, MDI operation is performed, then automatic operation starts again, the tool path is as follows:

In this case, the vectors at the start point of the next block are translated and the other vectors are produced by the next two blocks.

Therefore, from point Pc, compensation is accurately performed.



When points P_A, P_B and P_C are programmed in an absolute command, the tool is stopped by the single block function after executing the block from P_A to P_B and the tool is moved by MDI operation. Vectors V_{B1} and V_{B2} are translated to V'_{B1}, and V'_{B2} and offset vectors are recalculated for the vectors V_{C1} and V_{C2} between block P_B-P_C and P_C-P_D.

However, since vector V'_{B2} is not calculated again, compensation is accurately performed from point P_C.

(13) Manual operation

For manual operation during the cutter compensation, refer to item 4.4.4 of Section IV.

(14) Cutter compensation including the 4th axis

The offset plane including the 4th axis is not prepared, and therefore cutter compensation including the 4th axis is not possible.

6.4 D and H Functions

These addresses specify tool offset values or cutter compensation values and both specify the same offset value by the use of the same number.

Addresses D and H are selectively used for:

D .... cutter compensation
(cutter compensation, tool offset)

H .... tool length compensation
(cutter compensation, tool offset)

Codes and compensation values are correspondingly stored in memory previously via the MDI & DPL panel, enabling a corresponding compensation to be performed by specifying a 2-digit code.

The compensation value by H00 and D00 is always 0. When the power has been turned on, H00 and D00 are assumed.

Standard Tool Offset numbers are 01 to 32, 01 to 64, 01 to 99 or 01 to 200 when Tool Offset Number Addition A, B or C (option) are added respectively.

Commanding a value out of the range causes an alarm (No. 30).

(Note 1) Cutter compensation (G40, G41, G42) always uses D code. Tool length compensation (G43, G44, G49) always uses H code. The parameter (OFSD) specifies that tool offset (G45, G46, G47, G48) is to use D or H code.

(Note 2) Use of Tool Offset Number Addition B requires any of Part Program Storage & Edit B (40 meters) to F (1280 meters).

(Note 3) Use of Tool Offset Number Addition C requires any of Part Program Storage & Edit C (80 meters) to F (1280 meters).

6.5 External Tool Offset

This function alters tool offset values from outside, such as from the machine tool. Input of a tool offset value by this function adds that value to the then offset value for an offset number specified by the program. Also specifying an input signal makes the offset value the input value itself.

With the machine tool mounted with an automatic measuring feature for tools and workpieces, this function inputs a difference, if present, from the correct offset value, into the NC for correction.

Always refer to the machine tool builder's manual for programs, functions, and restrictions, which vary from machine tool builder to machine tool builder.

6.6 Offset Amount Input by Program (G10)

The offset amount, which are used in tool offset, tool length compensation and cutter radius compensation, can be set by G10 command in programming. The command format is as follows.

G10 P__p__ R__r__ ;

p: Offset number

r: Offset amount

The offset amount is absolute or incremental depending on the G90 or G91 respectively.

6.7 Scaling (G50, G51)

Scaling is commanded to figures specified by machining programs. For specifying this scaling, effectuate it by setting parameter (No. 64) first.

G51 I ___ J ___ K ___ P ___ ;

I, J, K : Scaling center X, Y, Z coordinate values

P : Scale factor (Least input increment : 0.001)

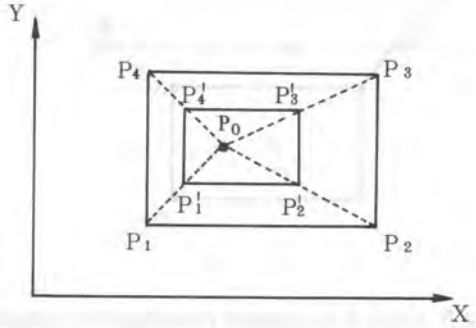
By this command, the subsequent move command is scaled by the scale factor specified by P, starting with the point specified by I, J, K as the scaling factor.

This scaling mode is cancelled by G50.

G50 : Scaling mode cancel command
G51 : Scaling mode command

The scale factor can be specified within the following range:

0.001 ~ 99.999 times (P1 ~ P99999)

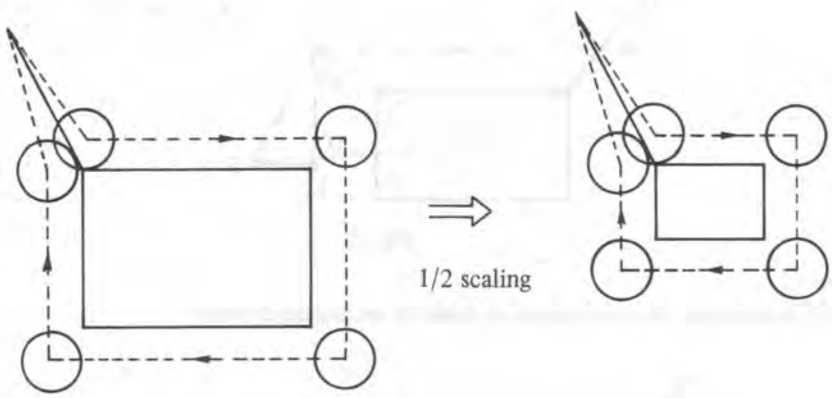


P1 ~ P4 : Figures of machining program
P1'~P4' : Figures after scaling
Po : Scaling center

If P is not specified, the scaling factor preset by MDI & CRT is applicable.

If I, J, K are omitted, the G51 command point serves as the scaling center.

This scaling is not applicable to the offset quantities, such as cutter compensation amount, tool length compensation amount, tool offset amount, and others.



(Note 1) *Specify G51 in G40 mode in an independent block. G50 can be specified in the offset mode. Cancel G51 by G50 without fail after scaling.*

(Note 2) *The position display represents the coordinate value after scaling.*

(Note 3) *If a setting value is employed as a scale factor without specifying P, the setting value at the G51 command time is employed as the scale factor, and a change of this value by another command if any, is not effective.*

(Note 4) *Whether the scaling function is effective or not can be set by a parameter for each axis. The scaling function always becomes effective for the circular radius command R in the G51 mode, irrespective of these parameters.*
Scaling is ineffective for additional axes at all times.

(Note 5) *Scaling is not effective for manual operation, but effective for the tape, memory, or MDI operation.*

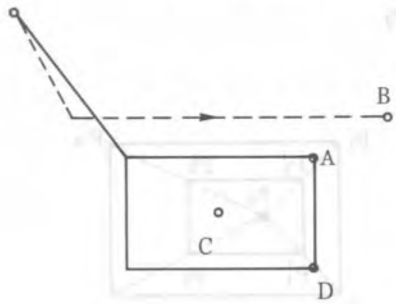
(Note 6) *Scaling is not applicable to the following movement in case of the canned cycle Z-axis movement.*
** Cutting depth Q and return relief of deep hole cycle (G83, G73).*
** X, Y shift quantity in fine boring (G76) and back boring (G87).*

(Note 7) *Specify G27, G28, G29, G30, and G92 in G50 mode without fail.*

(Note 8) *If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may become zero. In such a case, the block is regarded as no movement block, and therefore, it may affect the cutter movement by cutter compensation C. (See 6.3.6 (8))*

(Note 9) *Reset*

(a) The programmed coordinates turn to the present coordinates value, or scaled coordinates. Accordingly, the movement after reset differs according to whether the command is incremental or absolute.



If reset is made at point B, point A in program coordinates is regarded as being present at point B. When the move command to point D is executed, the following movement occurs according to whether the command is incremental or absolute.

**Incremental*

If the movement from point A to point D is incremental, D' becomes a target point on the programmed path, and the scaled D' point becomes point E, which moves the tool to point Ey because of the move command for the Y axis only.

AD = A' D'

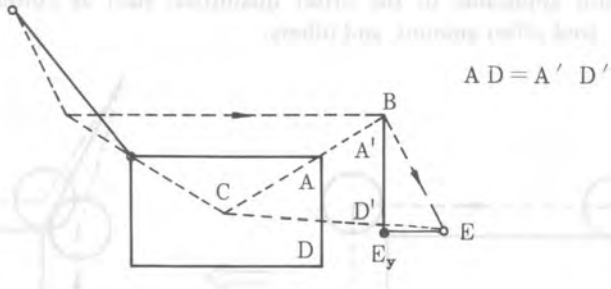


Fig. A

**Absolute*

If point D is absolute, the tool moves to point E, or, scaled D point.

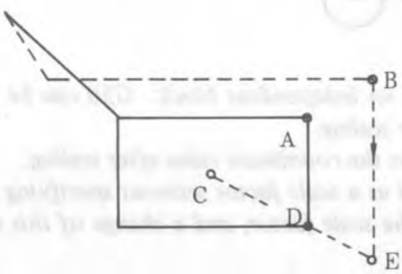


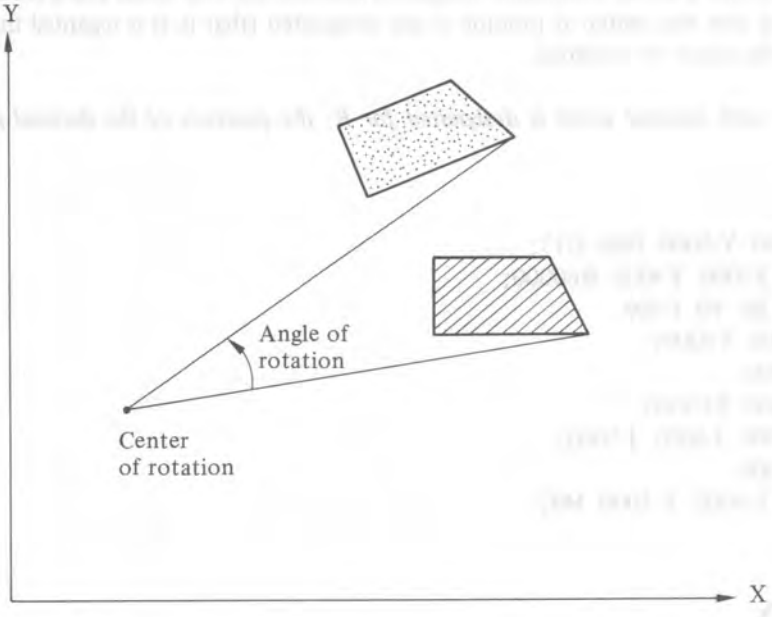
Fig. B

(b) When parameter CLER specifies clear state by resetting operation.
The G51 mode turns to G50, and the cutter moves to point D', if the move command is incremental (Fig. A) or the cutter moves to point D, if the move command is absolute (Fig. B).

6.8 Coordinate System Rotation (G68, G69)

A programmed shape can be rotated. By using this function it becomes possible, for example, to modify a program using a rotation command when a workpiece has been placed with some angle rotated from the programmed position on the machine.

Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a sub-program of the shape and calling it after rotation.



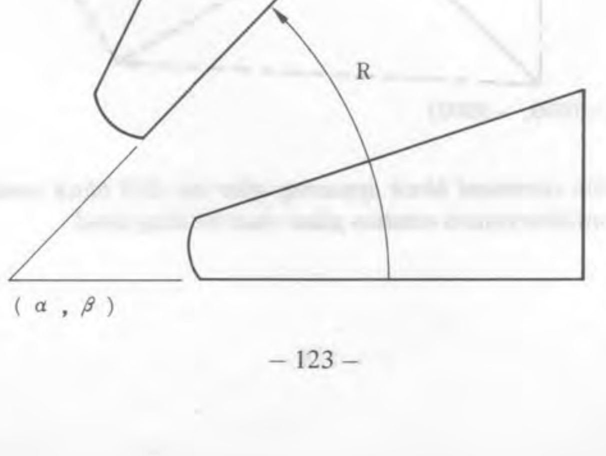
6.8.1. Command format

$$\left\{\begin{matrix} G17 \\ G18 \\ G19 \end{matrix}\right\} \text{G68} \quad \alpha\_\_\_ \beta\_\_\_ \text{R}\_\_\_ ;$$

α, β: Designate two axes among X, Y and Z that correspond to G17, G18 and G19. (G90/G91 mode is effective.)

R : Angle of rotation ("+" is the CCW direction. Command with an absolute value. An incremental value can also be command by the setting of parameter ROTR described later.)

After this command is specified, the subsequent commands are rotated by the angle specified with R around the point designated with α and β. Command the angle of rotation within the range of $0 \leqq R \leqq 360{,}000$ in 0.001 deg. increment.



The rotation plane depends on the plane (G17, G18, G19) selected when G68 is designated. G17, G18 or G19 is not required to be designated in the same block as G68.

When α and β are omitted, the position where G68 is commanded is set as the center of rotation.

When R is omitted, the value set to parameter No. 716 (setting input enable) is regarded as the angle.

The coordinate system rotation is cancelled by G69;

G69 may be designated in the same block as the other commands.

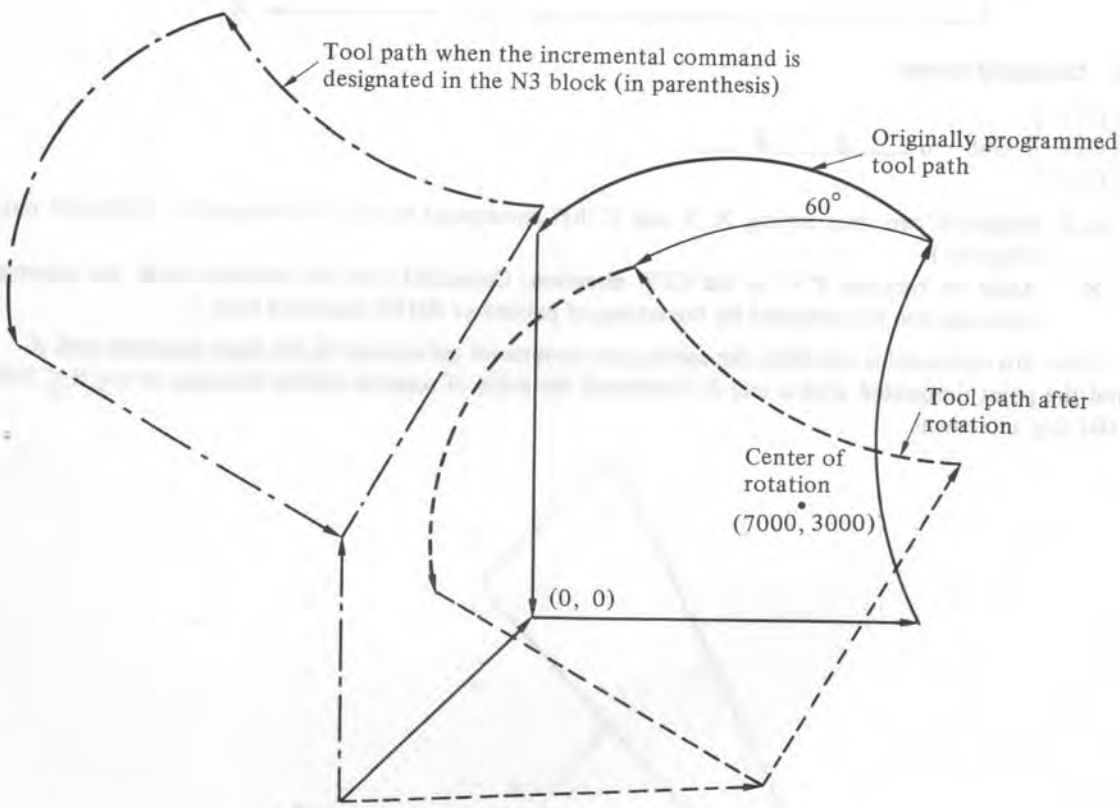
Tool offset such as cutter compensation, tool length compensation, or tool offset is performed after the coordinate system is rotated for the command program.

As for the incremental position commands designated between the G68 block and a block with an absolute command, it is regarded that the center of rotation is not designated (that is, it is regarded that the point where G68 was designated is the center of rotation).

(Note 1) When a value with decimal point is designated for R, the position of the decimal point becomes the unit of angle.

[Example 1]

```
N1  G92 X-5000 Y-5000 G69 G17;
N2  G68 G90 X7000 Y3000 R60000;
N3  G90 G01 X0 Y0 F200;
    (G91 X5000 Y5000)
N4  G91 X10000;
N5  G02 Y10000 R10000;
N6  G03 X-10000 I-5000 J-5000;
N7  G01 Y-10000;
N8  G69 G90 X-5000 Y-5000 M02;
```

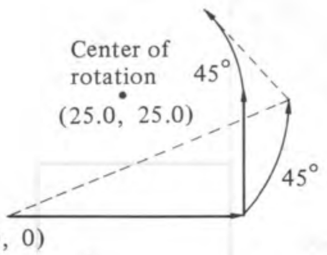


(Note 2) When the first position command block appearing after the G68 block contains an absolute command, both axes on the coordinate system rotation plane must be designated.

(Example)

[Not coorect] G92 X0 Y0 G17 G69;
G68 X25. Y25. R45;
G90 X50.;
Y25.;

[Correct] G92 X0 Y0 G17 G69;
G68 X25. Y25. R45.;
G90 X50. Y0;
Y25.;

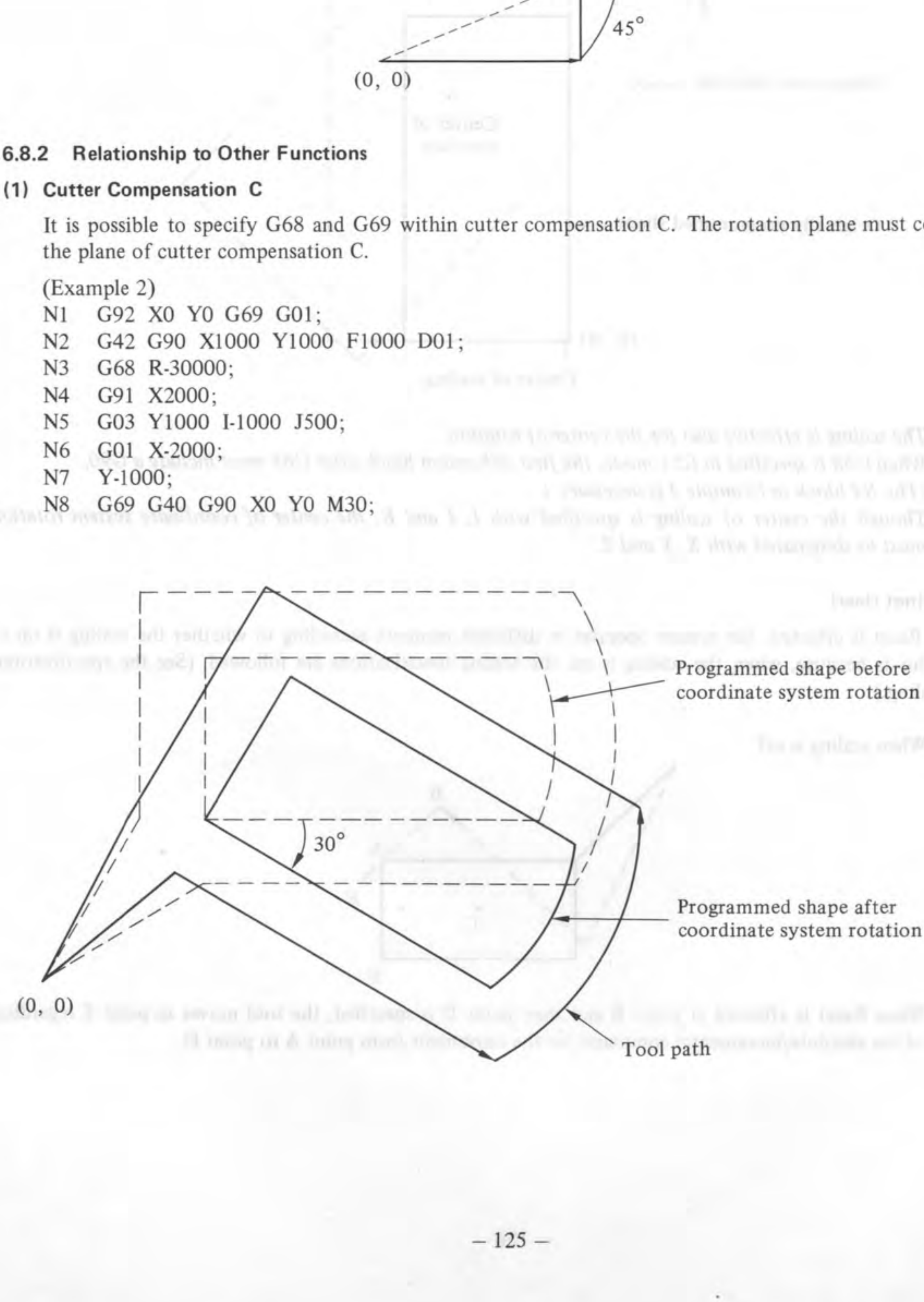


6.8.2 Relationship to Other Functions

(1) Cutter Compensation C

It is possible to specify G68 and G69 within cutter compensation C. The rotation plane must coincide with the plane of cutter compensation C.

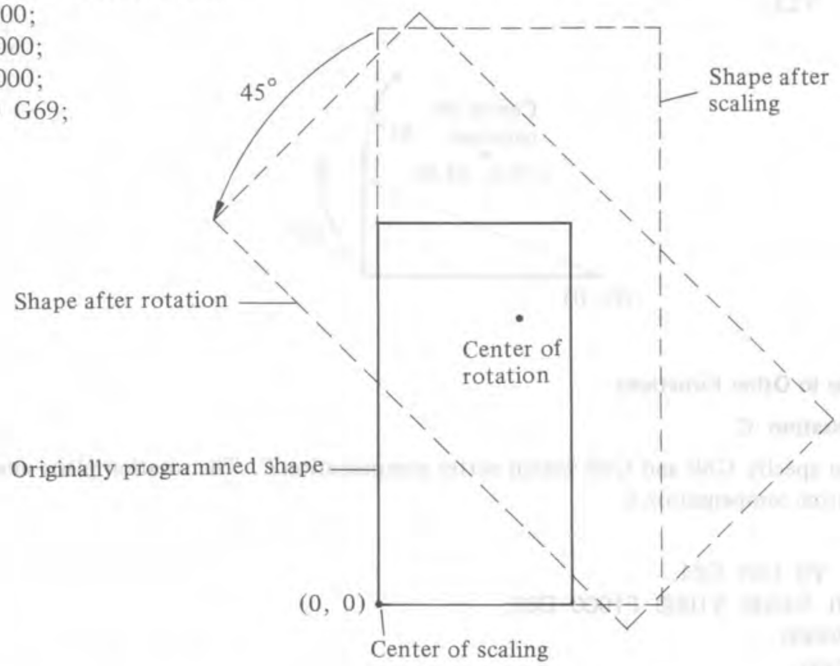
(Example 2)
N1 G92 X0 Y0 G69 G01;
N2 G42 G90 X1000 Y1000 F1000 D01;
N3 G68 R-30000;
N4 G91 X2000;
N5 G03 Y1000 I-1000 J500;
N6 G01 X-2000;
N7 Y-1000;
N8 G69 G40 G90 X0 Y0 M30;



(2) Scaling

When both scaling and coordinate system rotation are specified together, the coordinate system rotation is effected after the scaling.

(Example 3)
N1 G92 X0 Y0 Z0 G69 G50;
N2 G51 I0 J0 P1500;
N3 G68 X500 Y1000 R45000;
N5 G01 G91 X1000 F200;
N6 Y2000;
N7 X-1000;
N8 Y-2000;
N9 G50 G69;

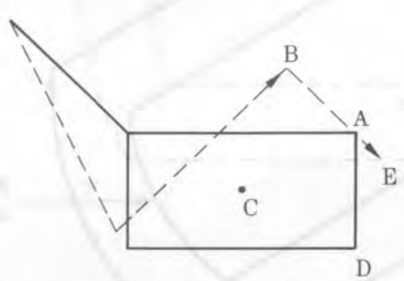


(Note 1) The scaling is effective also for the center of rotation.
(Note 2) When G68 is specified in G51 mode, the first dimension block after G68 must include a G90. (The N4 block in Example 3 is necessary.)
(Note 3) Though the center of scaling is specified with I, J and K, the center of coordinate system rotation must be designated with X, Y and Z.

(3) Reset (not clear)

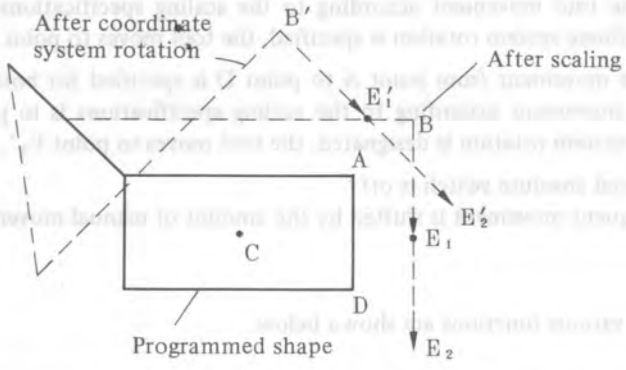
When Reset is effected, the system operates in different manners according to whether the scaling is on or off. This is because, when the scaling is on, the scaling specifications are followed. (See the specifications for scaling.)

(a) When scaling is off



When Reset is effected at point B and then point D is specified, the tool moves to point E regardless of the absolute/incremental command for the movement from point A to point D.

(b) During scaling



When Reset is effected at point B′ and the point D is specified:

(i) When the command of movement from point A to point D is incremental, the tool movement according to the scaling specifications is to point E_1. Therefore, when the coordinate system rotation is specified, the tool moves to point E_1′.

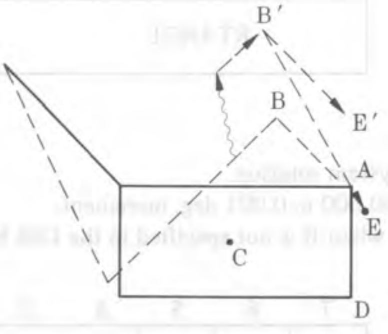
(ii) When the command of movement from point A to point D is absolute, the tool movement according to the scaling specifications is to point E_2. Therefore, when the coordinate system rotation is designated, the tool moves to point E_2′.

(4) Manual intervention

When manual intervention is performed, the system operates in different manners according to whether the scaling is on or off.

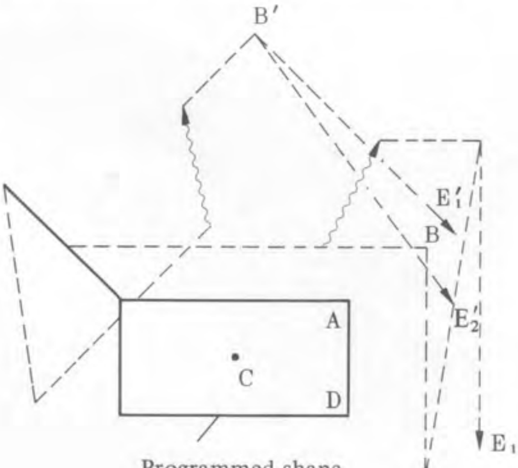
(a) When manual absolute switch is on

(i) When scaling is off



When the command of point D is absolute and both X and Y axes are specified, the tool moves to point E. When the command of point D is incremental, the tool moves to point E′.

(ii) When scaling is on



① When the movement from point A to point D is specified only for the Y axis in the incremental mode, the tool movement according to the scaling specifications is to point E_1. Therefore, when the coordinate system rotation is specified, the tool moves to point E_1'.

② When the movement from point A to point D is specified for both X and Y axes in absolute mode, the tool movement according to the scaling specifications is to point E_2. Therefore, when the coordinate system rotation is designated, the tool moves to point E_2'.

(b) When manual absolute switch is off
The subsequent movement is shifted by the amount of manual movement.

(5) Others

Notes related to various functions are shown below.

(Note 1) The position display indicates the coordinate value after the coordinate system rotation.
(Note 2) When an arc is specified during coordinate system rotation, the rotation plane must coincide with the arc plane.
(Note 3) The coordinate system rotation is invalid for manual operations.
(Note 4) The coordinate system rotation cannot be performed for the plane including the Z axis during a canned cycle.
(Note 5) The coordinate system rotation is not applied to shift amounts along the X and Y axes in G76 and G87.
(Note 6) Be sure to specify G27, G28, G29, G30 and G92 in G69 mode.
(Note 7) Perform clear of the coordinate value using the ORIGIN button in G69 mode.
(Note 8) The coordinate system rotation is not applied to the 4th and 5th axes.
(Note 9) G68 must not be specified when tool offset of G45 to G48 is being executed.
(Note 10) G31 (skip function) must not be commanded during G68 mode.

6.8.3 Parameter

	7	1	6	RTANGL

(This can be set by Setting.)

RTANGL Angle of coordinate system rotation.
Setting value: 0 to 360,000 in 0.001 deg. increment.
This value is effective when R is not specified in the G68 block.

				7	6	5	4	3	2	1	0
	6	3	3	ROTR							

ROTR = 0: R for angle designation is an absolute value.
= 1: Whether R for angle designation is absolute or incremental depends on G90/G91 mode.

136

6.8.4 Repetitive Commands

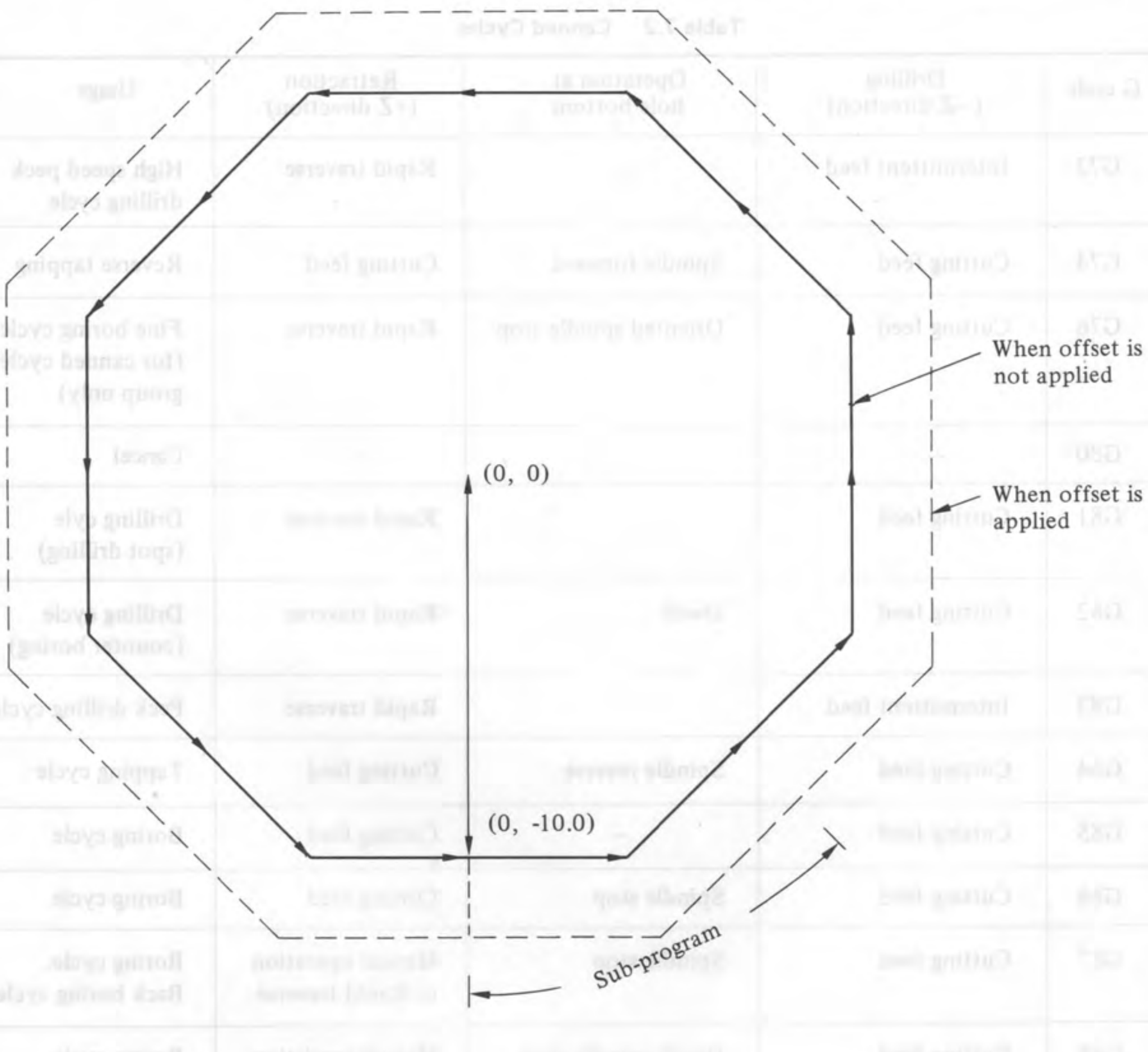
It is possible to store one program as a sub-program and recall the sub-program by changing the angle.

(Example 4) Setting the parameter ROTR to 1

```
G92 X0 Y0 G69 G17;
G01 F200 D01;
M98 P2100;
M98 P2200 L7;
G00 G90 X0 Y0 M30;

: 2200 G68 X0 Y0 G91 R45.0;
  G90 M98 P2100;
  M99;

: 2100 G90 G01 G42 X0 Y-10.0;
  X4.142 Y-10.0;
  X7.071 Y-7.071;
  G40;
  M99;
```



7. MACHINING CYCLE FUNCTION

7.1 Remote Operation Function (for 2-axis Control) (G80, G81)

G81X______Y______L______ ;

In the case of 2-axis control, drilling, clamping, punching, or the like can be performed for each block of tape command.

G81 permits, each time the operation of a block of tape command is completed, the transmitting of the remote operation function signal (EF), thereby performing drilling etc. The G81 function, once specified, remains until cancelled by the G80 (cancel) command. The G81 mode is reset or not, by parameter setting (CLER). The NC becomes cancel state (G80 mode) after turning power off. When power is turned on, the control comes on in G80. The repetition count is designated by the address L. When the word X, Y or L has not been commanded EF is not transmitted, even in G81 mode. G81 can be used for the remote operation function or for the canned cycle mentioned below by parameter setting (MCF).

7.2 Canned Cycles A, B (for 3-axis Control) (G73, G74, G76, G80 to G89)

The canned cycle is used for easy programming. Because cutting motion programmed by two or more blocks can be commanded by one block with G code. Optional canned cycles have two types (option A and B). With option A, G80, G81, G82, G84, G85, G86, and G89 can be used, while with option B, all canned cycles indicated in the table 7.2.

The following canned cycles are available.

Table 7.2 Canned Cycles

G code	Drilling (–Z direction)	Operation at hole bottom	Retraction (+Z direction)	Usage
G73	Intermittent feed	–	Rapid traverse	High speed peck drilling cycle
G74	Cutting feed	Spindle forward	Cutting feed	Reverse tapping
G76	Cutting feed	Oriented spindle stop	Rapid traverse	Fine boring cycle (for canned cycle II group only)
G80	–	–	–	Cancel
G81	Cutting feed	–	Rapid traverse	Drilling cyle (spot drilling)
G82	Cutting feed	Dwell	Rapid traverse	Drilling cycle (counter boring)
G83	Intermittent feed	–	Rapid traverse	Peck drilling cycle
G84	Cutting feed	Spindle reverse	Cutting feed	Tapping cycle
G85	Cutting feed	–	Cutting feed	Boring cycle
G86	Cutting feed	Spindle stop	Cutting feed	Boring cycle
G87	Cutting feed	Spindle stop	Manual operation or Rapid traverse	Boring cycle, Back boring cycle
G88	Cutting feed	Dwell, spindle stop	Manual operation or Rapid traverse	Boring cycle
G89	Cutting feed	Dwell	Cutting feed	Boring cycle

(Note 1) *Whether the output signals from the NC (SRV, SSP) (canned cycle I) or M codes (canned cycle II) are used for spindle CCW rotation and spindle stop is set by a parameter (FIX 2).*

(Note 2) *Different operations are performed for canned cycle I and II in G87 mode.*

The canned cycles generally comprise a sequence of the following six operations:

Operation
1. Positioning on X and Y axes
2. Rapid traverse to point R
3. Drilling
4. Operation at the hole bottom
5. Return to R point level
6. Return to initial point level

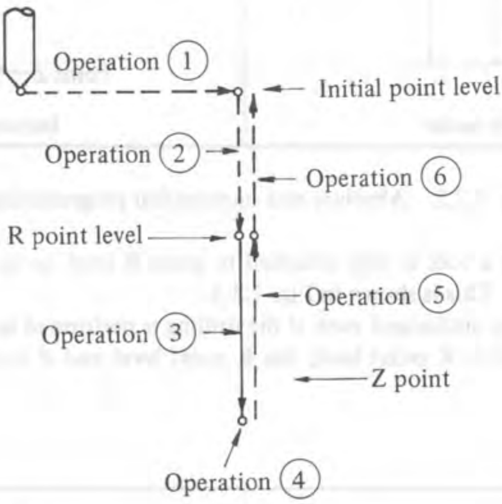


Fig. 7.2.1 Canned cycle operation

Positioning is performed on the XY plane and drilling is performed on the Z axis. Positioning and drilling cannot be performed on any other than this plane and axis combination, this is indifferent to the plane selection G command.

These canned cycle operations, to be more strict, consist of these three modes, each of which are specified in a particular modal G code, as shown below.

(1) Data format
- G90 Absolute
- G91 Incremental

(2) Return point level
- G98 Initial point level
- G99 R point level

(3) Drilling mode
- G73
- G74
- G76
- G80 See Table 7.2.
- G81
- .
- .
- .
- G89

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

139

(a) Data is given as shown in Fig. 7.2.2, depending on whether G90 or G91 mode.

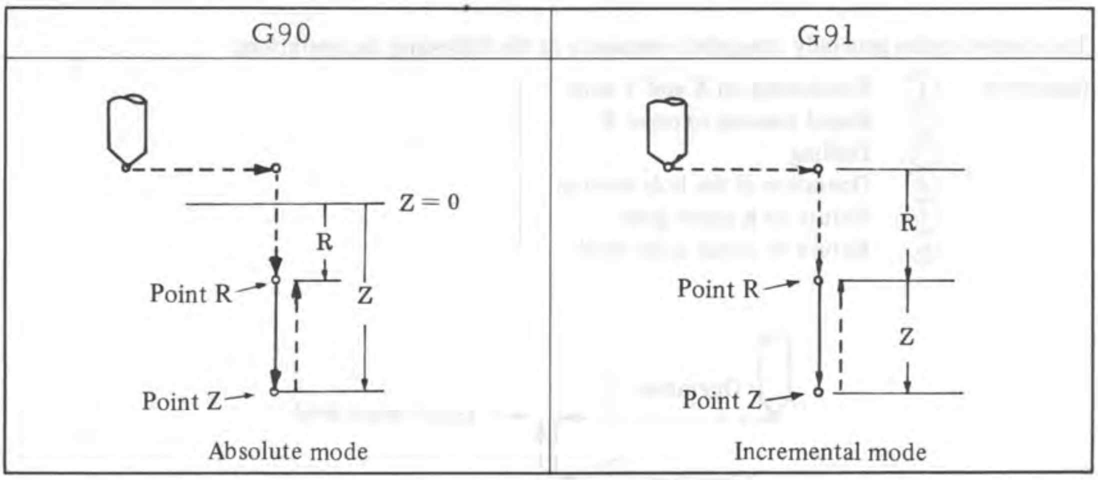


Fig. 7.2.2 Absolute and incremental programming

(b) By return operation, whether a tool is only returned to point R level, or up to the initial point level, depends on if G98 or G99 is specified. This is shown in Fig. 7.2.3.
The initial point level remains unchanged even if the drilling is performed in G99 mode. The start position is, if the previous cycle returned to R point level, the R point level and if it returned to the initial point level, the initial point level.

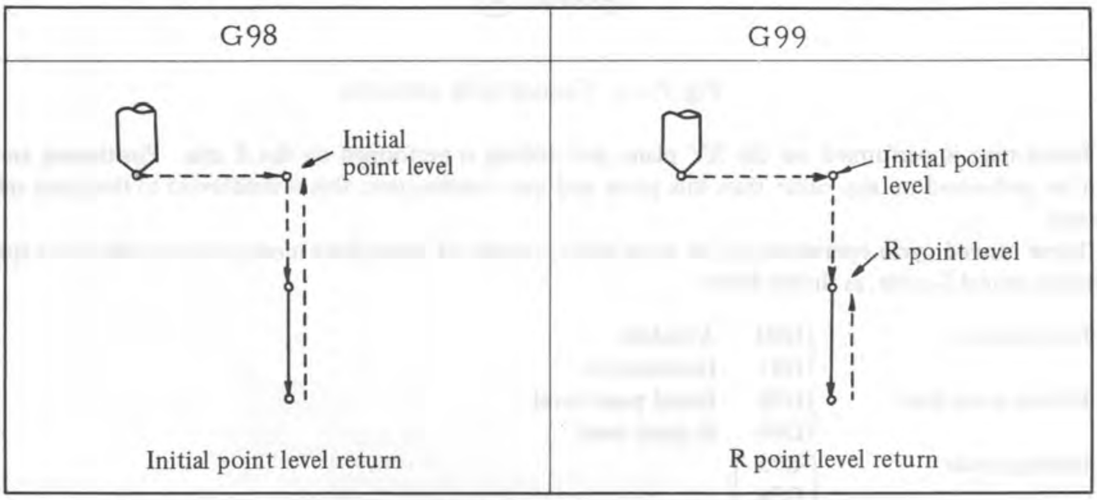


Fig. 7.2.3 Initial point level and R point level

(c) Data related to drilling operation specified subsequent to these G73, G74, G76, G81 to G89, G98 and G99 constitute one block. This command permits the data to be stored in the control unit as a modal value. The data related to drilling operation is commanded as follows:

$$\underbrace{G\square\square}_{\text{Hole machining mode}}\ \underbrace{X\text{——}Y\text{——}}_{\text{Hole position data}}\ \underbrace{Z\text{——}R\text{——}Q\text{——}P\text{——}F\text{——}}_{\text{Hole machining data}}\ \underbrace{L\text{——}}_{\text{Repitition count}};$$

140

Hole machining mode	:	G □□ (Refer to Table 7.2)
Hole position data X, Y	:	Specifies the hole position by an absolute or incremental value. For the tool path and the feed rate they are selected whether they follow a 01 group's G code currently available (G02 and G03 are regarded as G01) or they are always positioned at a rapid traverse rate by a parameter setting (FCUT).
Hole machining data Z	:	Specifies the distance from point R to the hole bottom in increment or the coordinate values of the hole bottom in absolute. Feed rate in operation ③ of Fig. 7.2.1 depends on the F code. Feed rate in operation ⑤ depends on the drilling mode (G73, G74, G76, G81 to G89).
R	:	R indicates a distance from the initial point to point R in increment, or the coordinate values of R point position in absolute. Feed rate in operation ② and ⑥ is rapid traverse.
Q	:	Specifies a depth of cut at each time in G73 or G83 mode, and shift value in G76 or G87 (canned cycle II) mode (this is always in incremental value).
P	:	Specifies a dwell time in the hole bottom position. Relation between actual dwell time and command value accords with a G04 command.
F	:	Specifies a feed rate in cutting feed.
Repetition count L	:	Specifies the repetition count of the canned cycle. When L is omitted, it is regarded as 1. If L = 0 is specified, hole machining data are memorized but no machining is done.

Because the hole machining mode (G□□), if once specified, does not change until other hole machining mode is specified or until the canned cycle cancelling G code is specified, it is not required to specify the hole machining mode in each block to execute the same hole machining. The canned cycle cancelling G codes are G80 and G codes in the 01 group.
The hole machining data, once specified, remains unchanged until the data is altered or the canned cycle is cancelled. Accordingly, specify all the required hole machining data when starting the hole machining and specify the altered data, if necessary, during canned cycle.
Specify the repetition count L when it is necessary. The data L is effective in the block in which it was specified. When the control unit is reset during canned cycle, the hole machining mode and the hole machining data remain unchanged, but the hole position data and the repetition count are cleared.

The example for holding and cancelling of data is shown as follows.

① G00 X____ M03 ;

② G81 X____ Y____ Z____ R____ F____ L____ ;
Initial data of canned cycle parameter "Z, R and F" must be specified. Drilling operation by G81 is performed "L" times.

③ Y____ ;
As drilling mode and drilling data are same as in program ②, G81, Z, R and F can be neglected. After the positioning motion specified by the address Y, the drilling is executed by G81 mode.

④ G82 X____ P____ L____ ;
After the positioning motion specified by the address X, the drilling is executed by G82 mode "L" times. At this time, the drilling data used are as follows.
Z, R and F . . . . specified in the block ②
P . . . . specified in the block ④

⑤ G80 X____ Y____ M05 ;
The drilling motion is not performed. All drilling data except F is cancelled.

⑥ G85 X____ Z____ R____ P____ ;
The addresses Z and R must be specified because these data have been cancelled in block ⑤. The F code is the same as that in block ②, and is omitted. The address P is not required in this block but is stored.

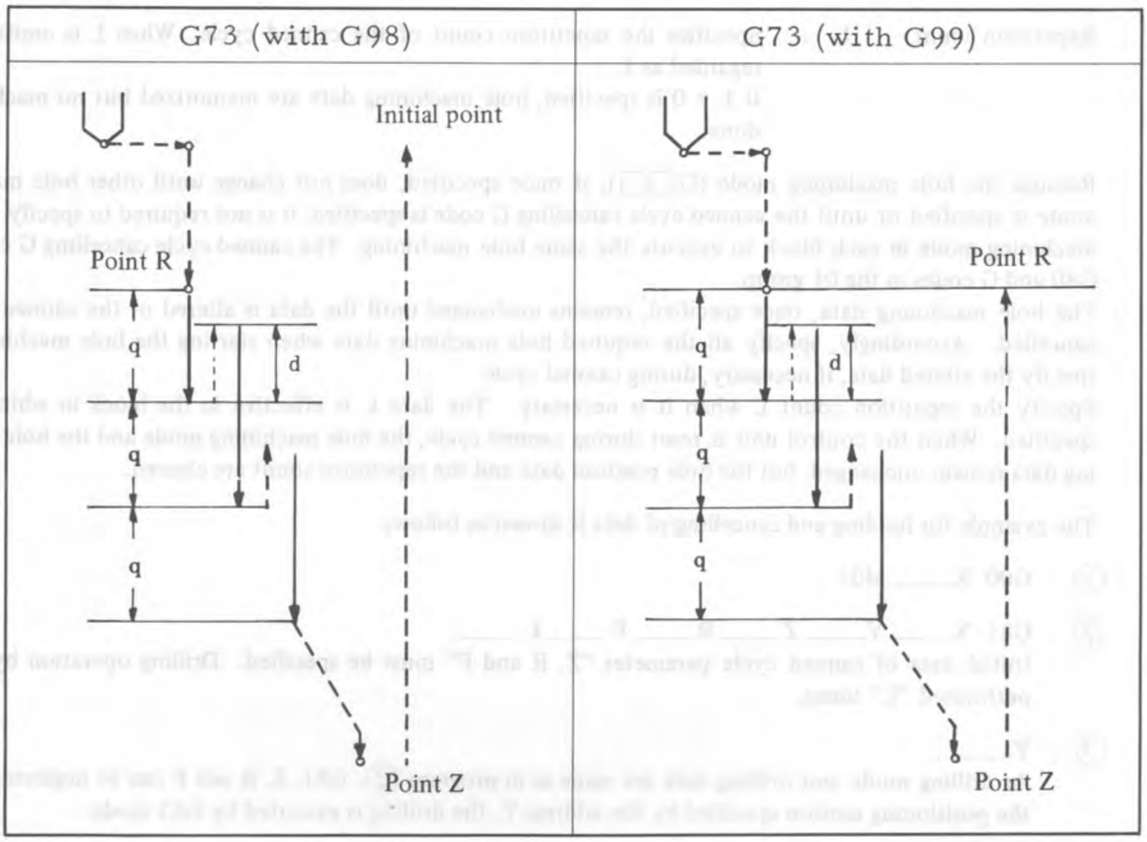
⑦ X____Z____;
After the positioning motion specified by the address X, the drilling is executed. At this time, the drilling data used are as follows.
Z .... specified in block ⑦
R .... specified in block ⑥
F .... specified in block ②

⑧ G89 X____Y____;
After the positioning motion specified by the addresses X and Y, the drilling is executed in G89 mode.
At this time, the drilling data used are as follows.
Z .... specified in block ⑦
R, P .... specified in block ⑥
F .... specified in block ②

⑨ G01 X____Y____;
The hole machining mode and the hole machining data are cancelled.

Each machining mode is descrbied in the followings.

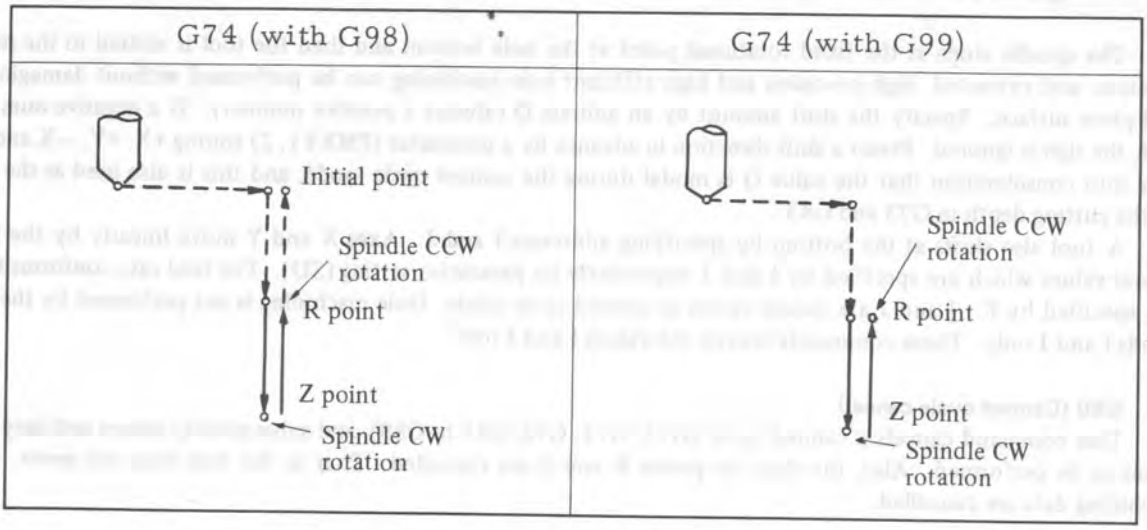
(1) G73 (High speed peck drilling cycle)



The "d" value is set by a parameter setting (CYCR).

Highly efficient hole machining can be performed, making removal of chips easier and keeping relief motions to a minimum. The relief motion is effected at the rapid traverse rate.

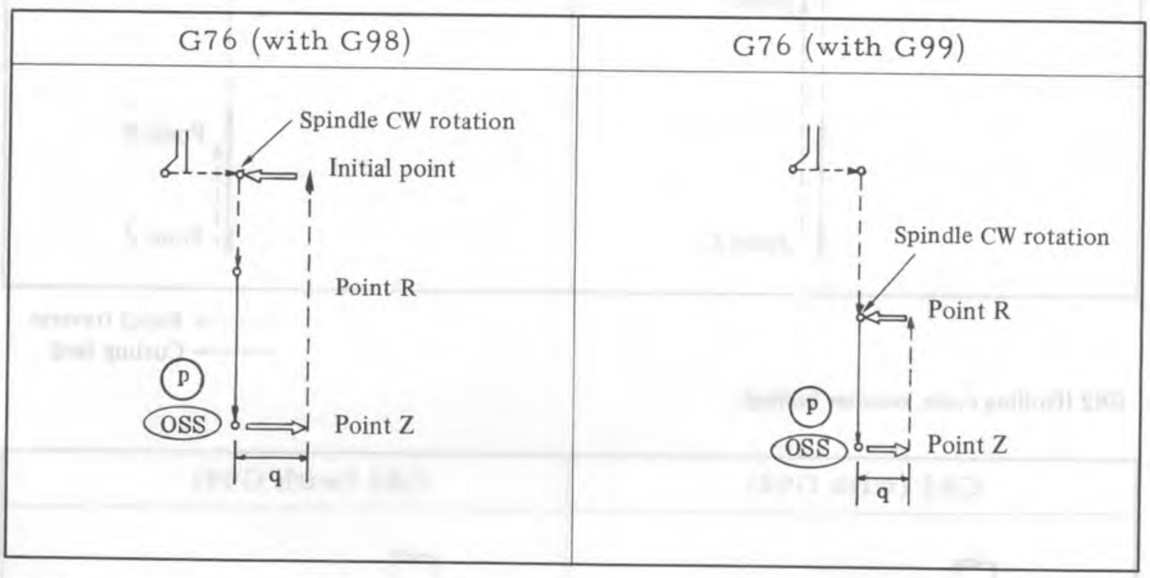
(2) G74 (Left-handed tapping cycle)

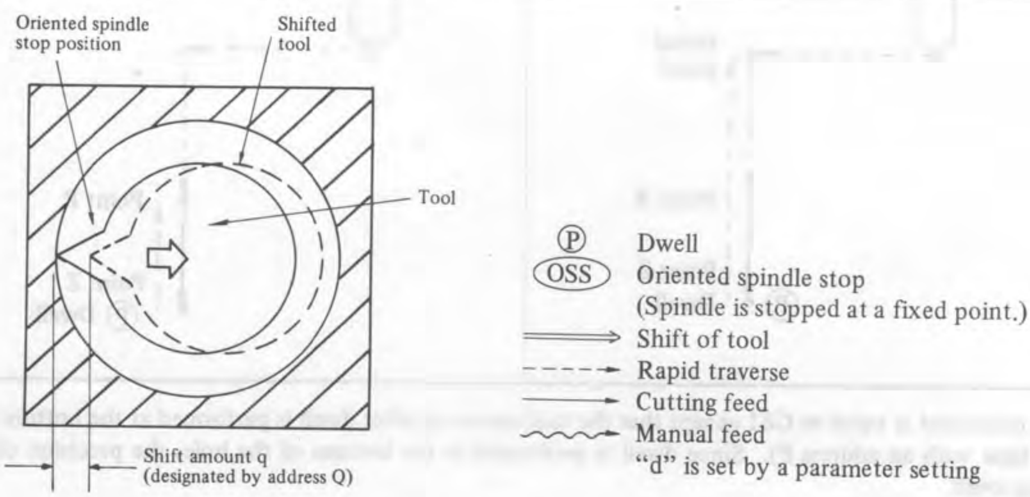


This command specifies the spindle to rotate clockwise at the hole bottom, and then a left-handed tapping cycle to be performed.

(Note) During tapping by G74, feed rate override is ignored and the motion doesn't stop until completion of the cycle even if the feed hold is effected.

(3) G76 (Fine boring cycle)





(Note 1) G76 can be used only when the parameter (FIX2) is set to output M codes as the output signals for spindle CCW rotation, spindle CW rotation and oriented spindle stop.

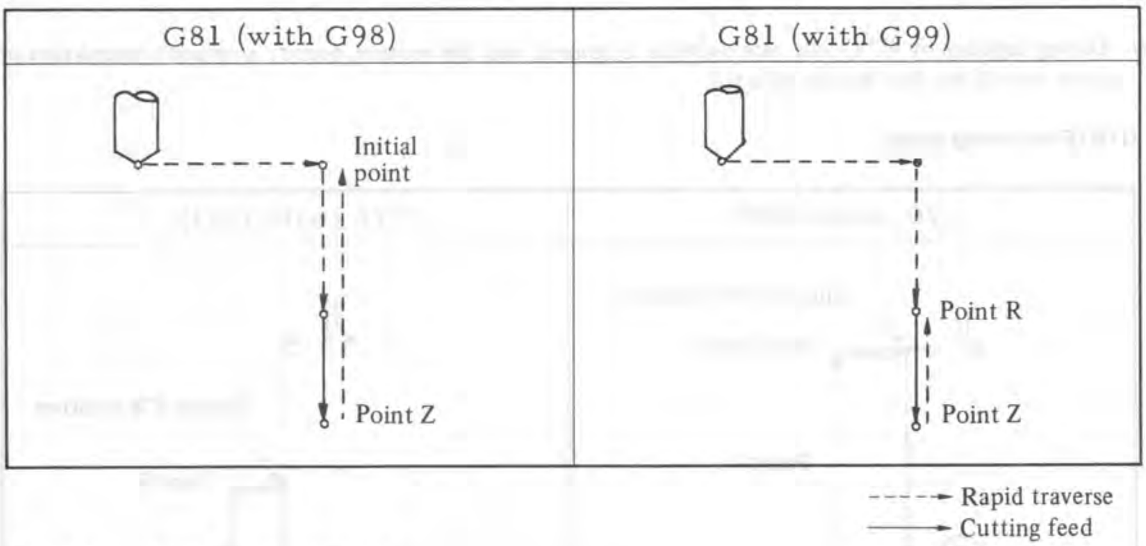
The spindle stops at the fixed rotational point at the hole bottom and then the tool is shifted to the reverse direction and extracted, high precision and high efficient hole machining can be performed without damaging the workpiece surface. Specify the shift amount by an address Q (always a positive number). If a negative number is used, the sign is ignored. Preset a shift direction in advance by a parameter (PMXY1, 2) among +X, +Y, –X and –Y. Take into consideration that the value Q is modal during the canned cycle mode, and this is also used as the value for the cutting depth in G73 and G83.

A tool also shifts at the bottom by specifying addresses I and J. Axes X and Y move linearly by the incremental values which are specified by I and J respectively by parameter setting (SIJ). The feed rate conforms to the rate specified by F. I and J are modal values in canned cycle mode. Hole machining is not performed by the commands I and J only. These commands rewrite the values I and J only.

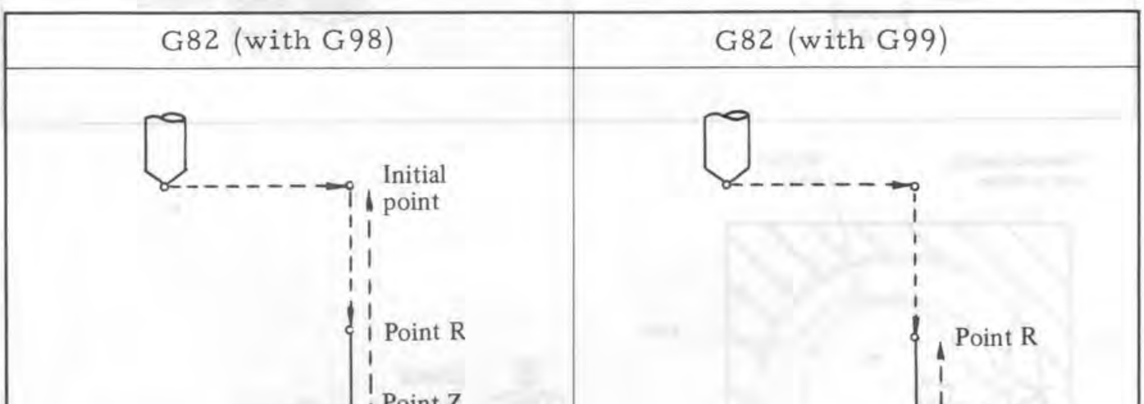
(4) G80 (Canned cycle cancel)

This command cancels a canned cycle (G73, G74, G76, G81 to G89), and subsequently causes ordinary operations to be performed. Also, the data for points R and Z are cancelled. That is, the tool does not move. Other machining data are cancelled.

(5) G81 (Drilling cycle, spot drilling)

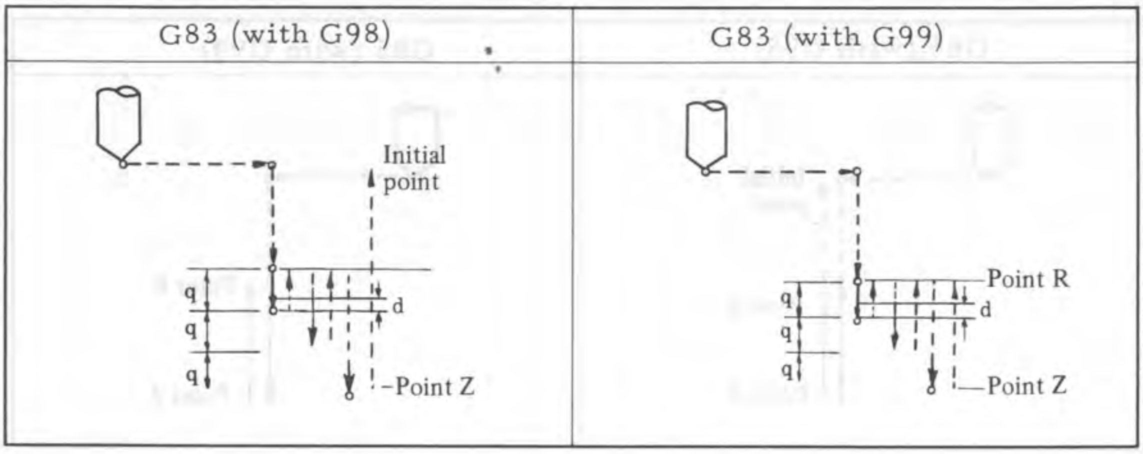


(6) G82 (Drilling cycle, counter boring)



This command is equal to G81 except that the tool moves up after dwell is performed at the bottom (specifies the dwell time with an address P). Since dwell is performed at the bottom of the hole, the precision of the hole depth is improved.

(7) G83 (Peck drilling cycle)

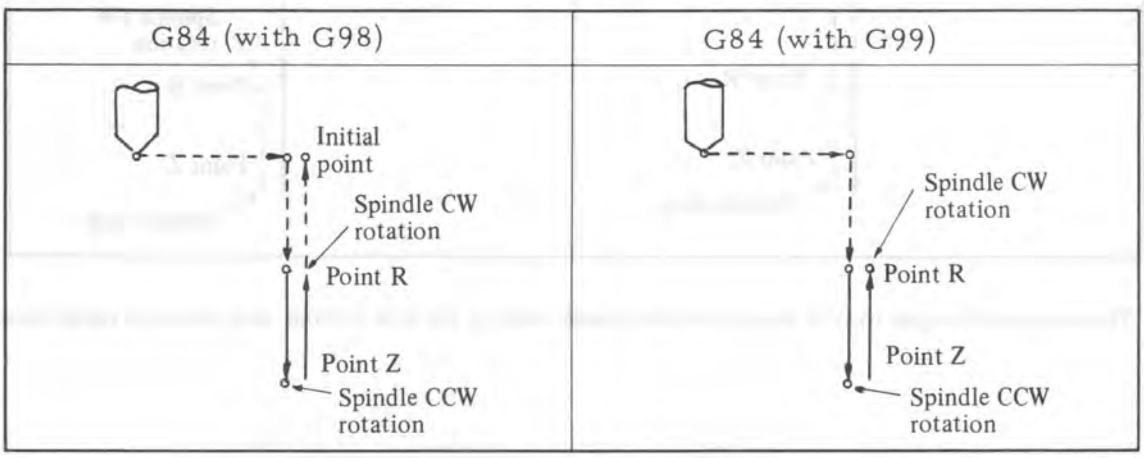


In this case, command as follows.

G83 X____Y____Z____Q____R____F____;

Q represents the depth of cut at each time, and is always commanded as an incremental value. Rapid traverse changes to cutting feed at the distance "d" inches (or mm) above the position where the immediately preceeding cut ended. Specify the value Q with a positive number. Even if it is specified with a negative number, the sign is ignored. The distance "d" is set by parameter (CYCD).

(8) G84 (Tapping cycle)

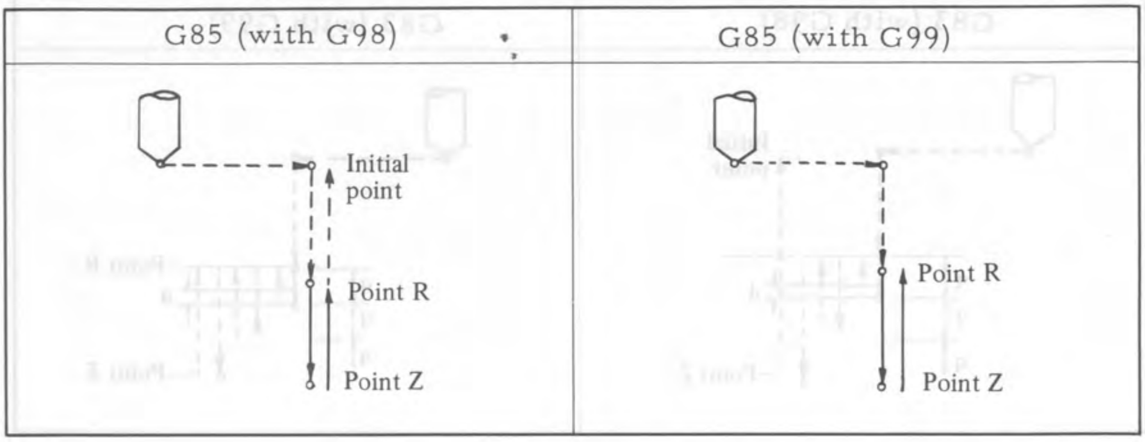


- - - - → Rapid traverse
———→ Cutting feed

This command specifies that the spindle rotates in the counterclockwise direction at the hole bottom, and a tapping cycle is performed.

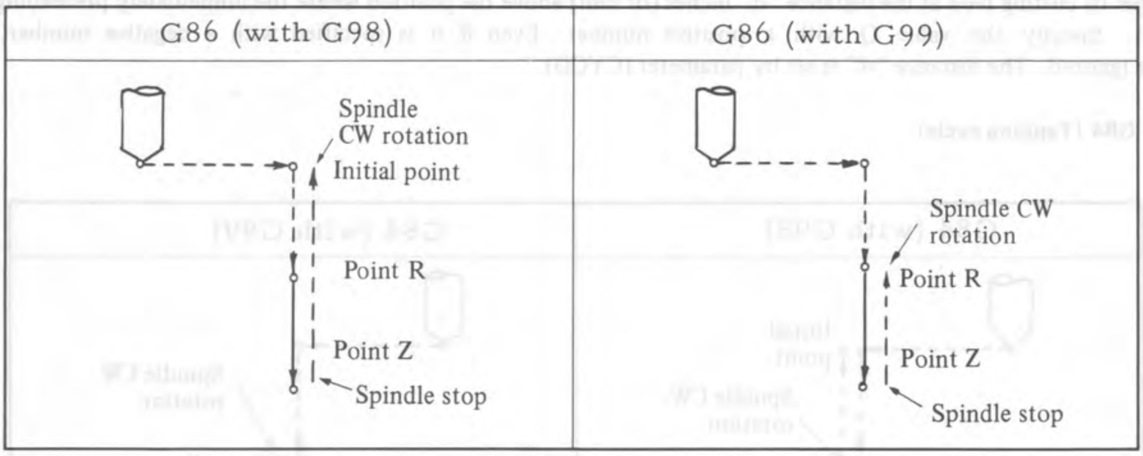
(Note) During tapping by G84, feed rate override is ignored and the motion doesn't stop until completion of the cycle even if the feed hold is effected.

(9) G85 (Boring cycle)



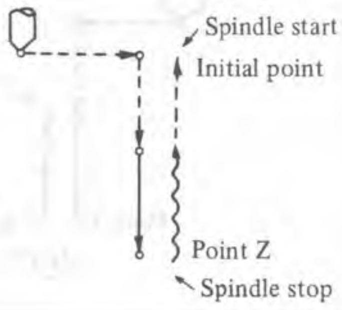
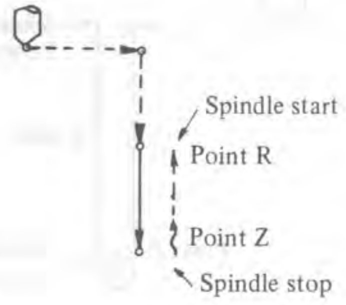
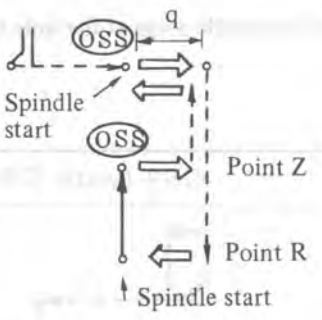
This command is equal to G84 except that the spindle does not reverse at the hole bottom.

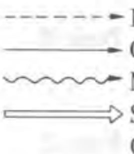
(10) G86 (Boring cycle)



This command is equal to G81 except that the spindle stops at the hole bottom, and returns in rapid traverse.

(11) G87 (Boring cycle/back boring cycle)

	G87 (with G98)	G87 (with G99)
Canned cycle I		
Canned cycle II		Not used.



Canned cycle I (Boring cycle)

When the tool reaches the hole bottom and the spindle stops, the control goes into the feed hold condition, and the tool can be moved in a manual mode while in this condition. Any manual operation can be performed, but the tool should be drawn out of the hole for safety.

To restart machining, turn the mode to tape or memory and push the START button. Then the tool returns to the initial level or R point level according to the command G98 or G99 respectively. And the tool starts rotation in the clockwise direction, and the next NC command will be executed.

Canned cycle II (Back boring cycle)

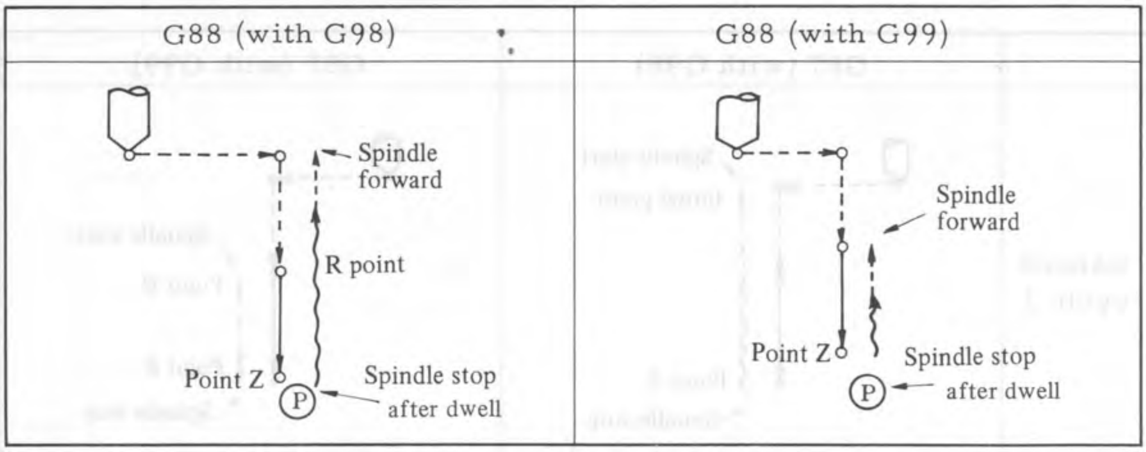
After positioning of X and Y axes, the spindle stops at the fixed rotational point, the tool is shifted in the direction opposite to the tool tip direction, and positioned at the hole bottom (point R). At this point, the tool is returned by the shifted amount, spindle rotates in the clockwise direction and the boring is performed to the point Z. After the spindle stops here at the fixed position again and the tool is released by the shifted amount, the tool is moved up from the hole. Then, the tool is turned to the initial point, shifted by the shifted amount, and returned to the initial position in X and Y axes. Here, the spindle rotates in the clockwise direction and the next block is executed.

For details of the shift amount and direction, refer to the explanation of the canned cycle G76.

(Note) Canned cycle I : it is set as a parameter (FIX2), that the signals SRV and SSP are used as the output signals for spindle CCW rotation and spindle stop respectively.

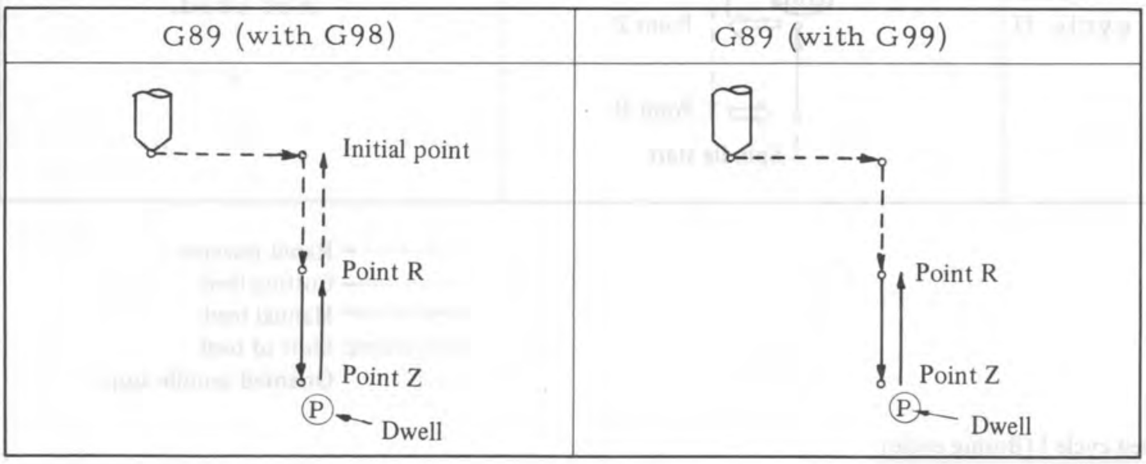
Canned cycle II : it is set as a parameter (FIX2), that M codes are output as the output signals for spindle CCW rotation, spindle stop and oriented spindle stop.

(12) G88 (Boring cycle)



This command is equal to G87 (canned cycle I) except the spindle stops at the hole bottom after dwell.

(13) G89 (Boring cycle)

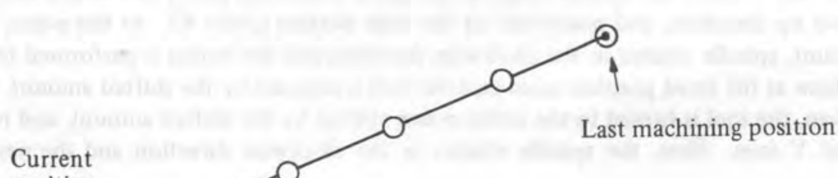


This command is equal to G85 except that dwell is made at the hole bottom.

7.2.1 Canned cycle repetition

If drilling at equal intervals is to be repeated in the same cycle, specify the repetition count with an address L. The maximum value of L is 9999. L is only effective in the block in which it is programmed.

Example 7.2.1



G81X___ Y___ Z___ R___ L5F___ ;

146

The command

X____ Y____ specifies the first machining position in incremental values (in G91). If this command is specified in absolute values (in the case of G90), drilling is repeated in the same position.

The automatic acceleration/deceleration time constant within the canned cycle can be automatically switched to the rapid traverse or cutting feed time constant according to the feed at each movement. At the terminal point of one movement, after deceleration is completed, the operation shifts to the next sequence.

However, when rapid traverse is used to return from the bottom of the hole to the R point level (for example G81) in G98 (return to the initial level), the return is made directly to the initial level at rapid traverse.

Notes on canned cycle:

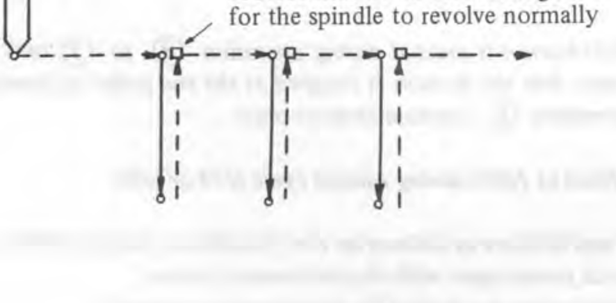
(Note 1) Canned cycle commands must be specified in Spindle-on condition.

```
ER ;
M03 ;                                  Spindle CW
.
.
.
G□□          ..... ;                   Correct
.
.
.
M05 ;                                  Spindle stop
.
.
.
G□□          ..... ;                   Incorrect
                                       (M03 or M04 must have been specified before
                                        this block.)
```

(Note 2) In the drilling mode, if one of the position data in X, Y, Z and 4th axes is specified in the block, the drilling operation can be performed. However, when they are not specified, it is not performed. When the dwell command (G04 X____ ;) is programmed, the drilling operation is not performed. If address X is specified as a dwell command (G04X____;), the drilling operation is not performed.

```
G00 X____;
G81 X____ Y____ Z____ R____ F____ P____ L____;
;                      (No drilling)
F____;                 (No drilling, F code is updated)
M____;                 (No drilling, M function is performed)
G04 P____;             (No drilling, drilling data P is not updated by this command)
```

(Note 3) In a canned cycle which controls spindle motion, for example G74, G84, G86, it may sometimes occurs, because drilling is performed continuously under such conditions that the interval between two holes is short and the distance from initial point level to R point level is short, that the aimed rotation speed of the spindle is not attained before the next drilling begins. In such case, some period of time should be inserted between two drilling operations by specifying a G04 (dwell). Accordingly, program as follows without specifying L.



G00 M____ ;
G86 X____ Y____ Z____ R____ F____ ;
G04 P____ ; (Dwell is performed, and no drilling)
X____ Y____ ;
G04 P____ ; (Dwell is performed, and no drilling)
X____ Y____ ;
G04 P____ ; (Dwell is performed, and no drilling)
:
:
.

This note may not be necessary depending on the machine tool. Refer to the manual issued by the machine tool builder.

(Note 4) *A canned cycle is cancelled by G00, G01, G02, or G03. When G00, G01, G02 or G03 is designated in the canned cycle mode, the following motions are performed. (The mark # represents 0, 1, 2 or 3)*

(G0 # : G00, G01, G02, or G03)
(G □□ : Canned cycle)

G0 # G □□ X____ Y____ Z____ R____ Q____ P____ F____ L____ ;
(Canned cycle is performed.)

G □□ G0 # X____ Y____ R____ Q____ P____ F____ L____ ;
(The tool is moved along X and Y axes according to group 01 G code, R, P and L values are neglected and F code is stored.)

G □□ G0 # X____ Y____ Z____ R____ Q____ P____ F____ L____ ;
(If optional 3-axis simultaneous control is not equipped, this program will cause to generate an alarm.)

(Note 5) *When both M code and canned cycle are specified in the same block, the M code and the signal MF are sent out at the first positioning (operation ①) and when the signal FIN is received at the end of a cycle, the next drilling is performed. When the cycle has the command for repetitive operation (L), the signal MF is setn out at the first cycle only.*

(Note 6) *In the canned cycle mode, the tool offset commands (G45 to G48) are ignored.*

(Note 7) *If a tool length compensation (G43 or G44) is commanded in the canned cycle mode, the tool length compensation is effected at the time of positioning at R point (at operation ② in Fig. 7.2.1).*

(Note 8) *Operator's notes.*

(a) *Reset*

Take notice that even if the reset state is entered via the RESET button or a emergency stop, generally, the drilling mode and the drilling data remains unchanged. The drilling mode and the drilling data can be cancelled by the parameter (CLER).

(b) *Single block*

When canned cycles are performed in Single Block mode, the motion is stopped at the end point of an operation (①, ② and ⑥ in Fig. 7.2.1.)

Three cycle starts are necessary for a canned cycle in this mode.

Feed Hold lamp lights at the end point of operation ① and ②. At the end of operation ⑥, if repetition cycles remain to be performed in the block, the motion stops in the feed hold state, and it stops in the stop state in other cases.

(c) *Feed hold*

When the FEED HOLD button is pressed during operation ③ to ⑤ in G74 or G84 mode, Feed Hold Lamp is lit at once, but the motion is stopped at the end point of operation ⑥ . If the feed hold is effected during motion ⑥ , motion stops at once.

(d) *Feed rate override*

Feed rate override is fixed at 100% during canned cycle G74 or G84.

(e) *Manual absolute*

G87 (canned cycle I) and G88 are as follows by the "MANUAL ABSOLUTE" switch.

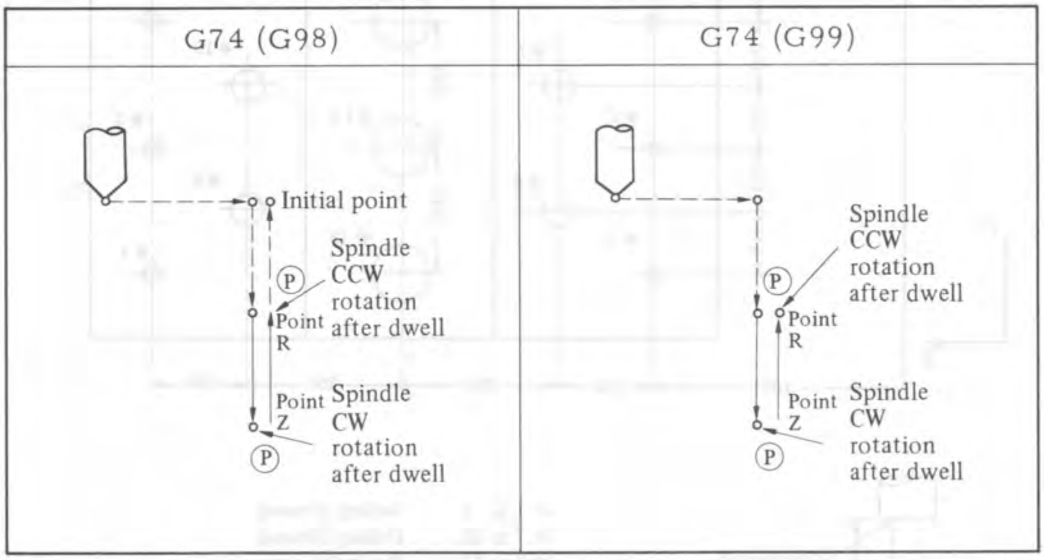
ON : R and initial points agree with the programmed value.
OFF : R and initial points are shifted by the manual movement.

(f) *ORIGIN button*
Don't push the ORIGIN button during canned cycle mode.

(Note 9) *The canned cycle G74 and G84 can be changed as follows, by parameter setting (FXCD). By this parameter setting, the dwell can be performed before spindle CW and CCW rotation. It is necessary when the special tapper is used. That is, thread cutting is performed by the forward/backward movement of tap by rotational force during dwell, without Z axis movement.*

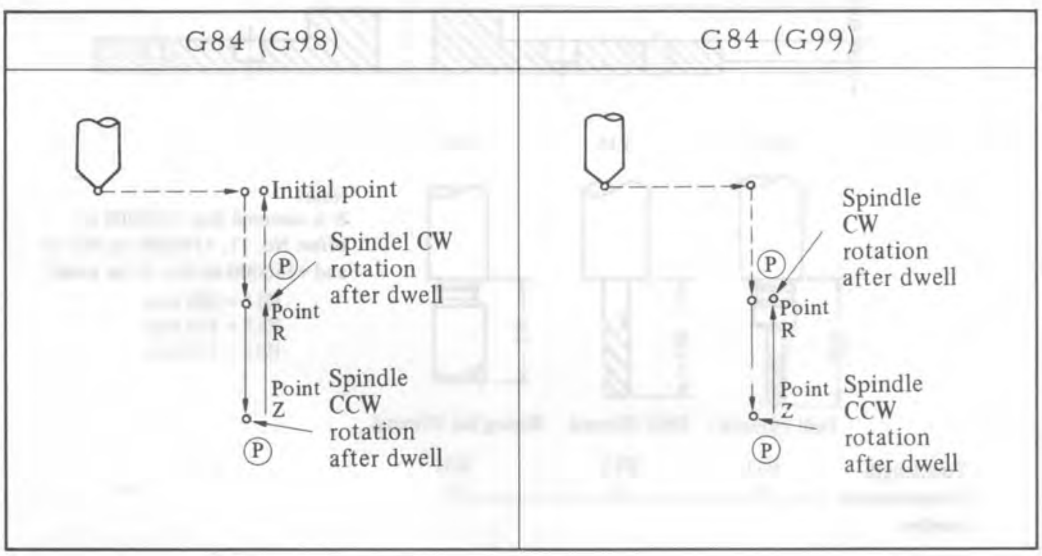
(a) *G74 (Left-handed tapping cycle)*

G74 X____ Y____ Z____ R____ P____ F____ ;

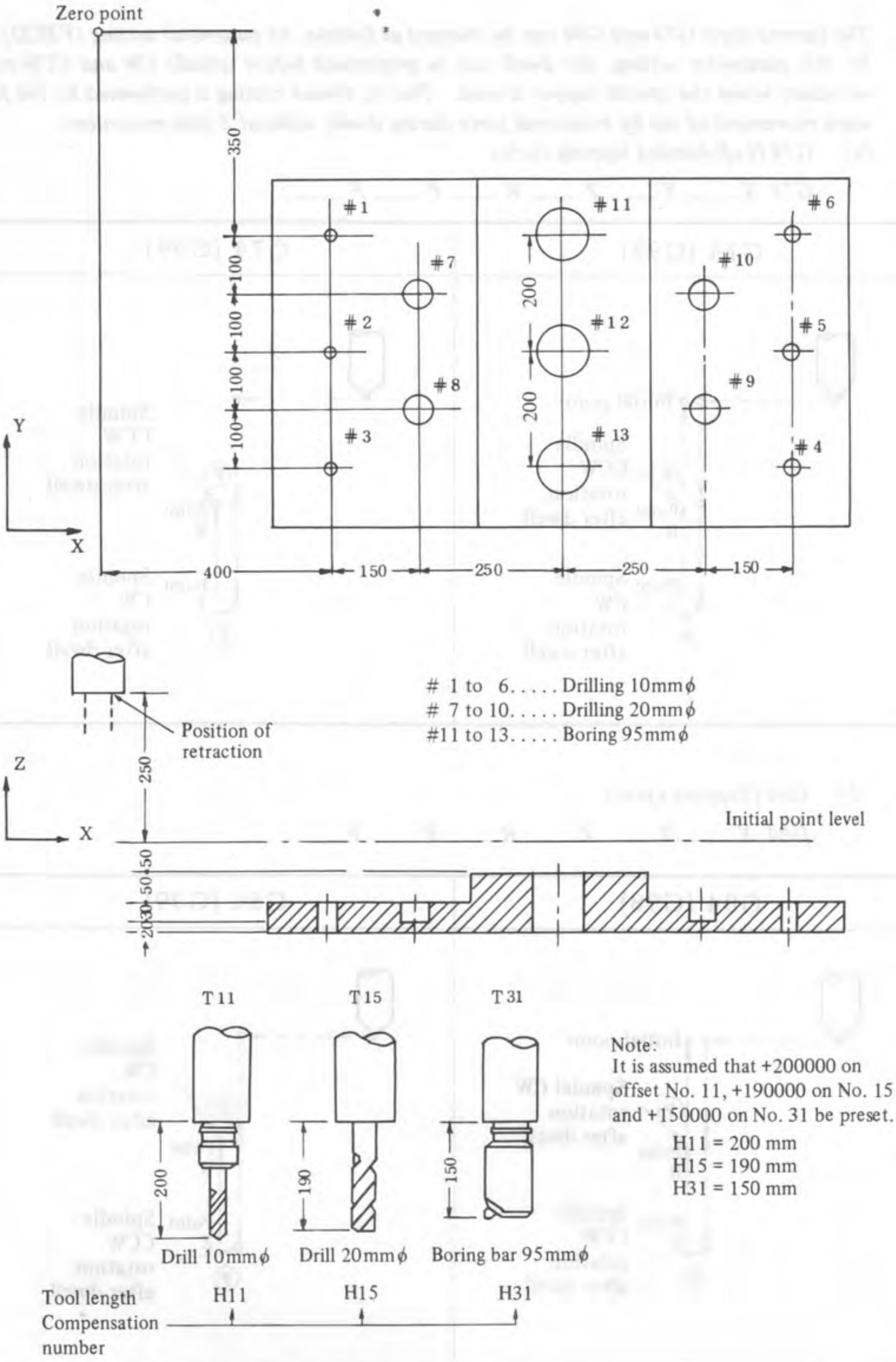


(b) *G84 (Tapping cycle)*

G84 X____ Y____ Z____ R____ P____ F____ ;



Example 7.2.2 Programming using tool length compensation and canned cycle



1 to 6. . . . . Drilling 10mmφ
7 to 10. . . . . Drilling 20mmφ
#11 to 13. . . . . Boring 95mmφ

Note:
It is assumed that +200000 on offset No. 11, +190000 on No. 15 and +150000 on No. 31 be preset.
H11 = 200 mm
H15 = 190 mm
H31 = 150 mm

Programming example

N001 G92 X0 Y0 Z0 ;	Programming of absolute zero point
N002 G90 G00 Z250.0 T11M06;	Tool change
N003 G43 Z0H11;	Initial level & tool length compensation
N004 S30 M03;	Spindle start
N005 G99 G81 X400.0 Y-350.0 Z-153.0 R-97.0 F120;	Drilling #1 hole after positioning
N006 Y-550.0;	Drilling #2 hole after positioning & return to R point
N007 G98 Y-750.0;	Drilling #3 hole after positioning & return to initial point
N008 G99 X1200.0;	Drilling #4 hole after positioning & return to R point
N009 Y-550.0;	Drilling #5 hole after positioning & return ro R point
N010 G98 Y-350.0;	Drilling #6 hole after positioning & return to initial point
N011 G00 X0 Y0 M05;	Reference point return & spindle stop
N012 G49 Z250.0 T15M06;	Tool length compensation cancel & tool change
N013 G43 Z0H15;	Initial level & tool length compensation
N014 S20 M03;	Spindle start
N015 G99 G82 X550.0 Y-450.0 Z-130.0 R-97.0 P300 F70;	Drilling #7 hole after positioning & return to R point
N016 G98 Y-650.0;	Drilling #8 hole after positioning & return to initial point
N017 G99 X1050.0;	Drilling #9 hole after positioning & return to R point
N018 G98 Y-450.0;	Drilling #10 hole after positioning & return to initial point
N019 G00 X0 Y0 M05;	Reference point return & spindle stop
N020 G49 Z250.0 T31M06;	Tool length compensation cancel & tool change
N021 G43 Z0H31;	Initial level & tool length compensation
N022 S10 M03;	Spindel start
N023 G85 G99 X800.0 Y-350.0 Z-153.0 R-47.0 F50;	Drilling #11 hole after positioning
N024 G91Y-200.0 L2;	Drilling #12 and #13 holes after positioning & return to R point
N025 G00 G90 X0 Y0 Z0 M05;	Reference point return & spindle stop
N026 G49 G91 Z0;	Tool length compensation cancel
M02;	Program stop

(Note) In G98/G99 mode, when the repetitive count (L) is programmed, the tool is returned to initial point level/R point level respectively from the first drilling.

7.3 Initial Point Level and R Point Level in Canned Cycle (G98, G99)

G98 and G99 specify a return point level in canned cycle to be the initial point level and the R point level, repectively, as shown in Fig. 7.3. The start point is the same as the end point of the preceding canned cycle. As usual, G99 is used for the first drilling and G98 is used for the last drilling. When the repetitive count (L) is programmed, the tool is returned to the initial level with G98 from the first drilling.

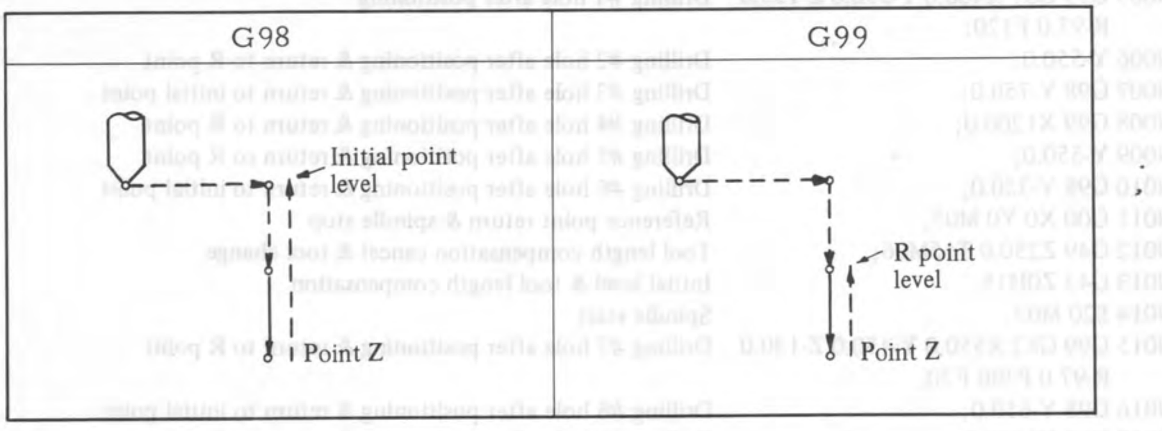


Fig. 7.3 Initial point level and R point level

8. SPINDLE SPEED FUNCTION (S FUNCTION), TOOL FUNCTION (T FUNCTION) MISCELLANEOUS FUNCTION (M FUNCTION), 2ND MISCELLANEOUS FUNCTION

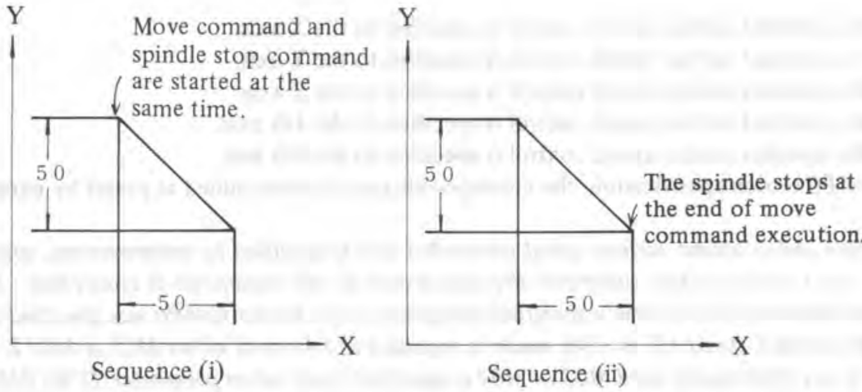
By specifying a numerical value following address S, T, M or B, a BCD signal and a strobe signal are transmitted to the NC machine tool. This is mainly used to control the ON/OFF function of the machine.

S codes are used for spindle control, T codes are used for tool changes, and M codes are used to control ON/OFF of the various functions of the machine tool and B codes are used for indexing of the table. Refer to the machine tool builder's manual for application of these addresses and codes to functions.

When a move command and S, T, M or B codes are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and S, T, M or B function commands.
(ii) Executing S, T, M and B function commands upon completion of move command execution.

(Example) N1 G01 G91X50.0Y-50.0 M05; (Spindle stop)



Sequence (i) Sequence (ii)

The selection of either sequence depends on the machine tool builder's specifications. In certain cases, these sequences may be provided together in an NC machine. Refer to the manual issued by the machine tool builder for details.

8.1 Spindle Speed Function (S Function)

8.1.1 S 2-digit

The spindle speed is controlled by address S and the following 2-digit number. Refer to the manual issued by the machine tool builder for details.

(Note 1) When a 4-digit S code is specified in the 2-digit specification, the lower 2 digits are used.

8.1.2 S 4-digit

The spindle speed (rpm) is directly specified by address S and the following 4-digit number.
The unit of spindle speed varies with the machine tool builder.

8.2 Constant Surface Speed Control

The constant surface speed control is provided to calculate the spindle speed so that the surface speed is always kept as specified with a tool position change and then supply a voltage corresponding to the calculated spindle speed to the spindle control so as to rotate the spindle at correct surface speed when the surface speed (relative speed of tool and a workpiece) is specified after S.

The surface speed unit is as shown below.

Input unit	Surface speed unit
Millimeter	m/min
Inch	feet/min

This surface speed unit may change according to machine tool builders.

8.2.1 Specification method

For the constant surface speed control, specify the following G codes.

G code	Meanings	Unit
G96	Constant surface speed control ON	m/min feed/min
G97	Constant surface speed control OFF	rpm

It is necessary for executing the constant surface speed control to set the work coordinate system so that the coordinate value at the center of the rotary axis (axis to which the constant surface speed control applies) becomes zero.

The constant surface speed controlled axis is selectable by a programmed command.

$$G96P \left\{ \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \end{array} \right\} \underline{\qquad\qquad} ;$$

P1 . . . . . The constant surface speed control is specified to the X axis.
P2 . . . . . The constant surface speed control is specified to the Y axis.
P3 . . . . . The constant surface speed control is specified to the Z axis.
P4 . . . . . The constant surface speed control is specified to the 4th axis.
P5 . . . . . The constant surface speed control is specified to the 5th axis.

In case of P0 or non-specification, the corresponding axis is determined as preset by parameter No. 315.

(Note 1) *When the constant surface speed controlled axis is specified by programming, specify Pα (α = 1, 2, 3, 4, or 5) without fail, otherwise the axis preset by the parameter is controlled. Specify Pα without fail whenever G96 is newly specified, irrespective of whether G96Pα was specified before or not.*

(Note 2) *The surface speed (S) in G96 mode is regarded as S=0 until either M03 or M04 is specified. Namely, S is not effectuated until M03 or M04 is specified (only when parameter TCW (010-bit 7) = 1).*

8.2.2 Spindle override

The specified surface speed or spindle speed can be overridden by 50, 60, 70, 80, 90, 100, 110, or 120% according to signals from the machine tool side.

8.2.3 Clamp of maximum spindle speed

The maximum spindle speed can be specified by rpm by a numeric value following G92S during the constant surface speed control.

G92S ____ ;

If the spindle speed becomes higher than the programmed value during the constant surface speed control, it is clamped to the maximum spindle speed.

8.2.4 Rapid traverse (G00)

In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but it is made by calculating the surface speed based on the position at the end point of the block from the beginning, on condition that cutting is not executed at rapid traverse.

Cautions:

(Note 1) *The constant surface speed control is turned off (G97) when power is turned on.*
(Note 2) *The spindle override is effective on condition that parameter SOV (No. 010 bit 5) is preset to 1.*
(Note 3) *The maximum spindle speed is not being preset (or, not being clamped) when power is turned on.*
(Note 4) *The spindle speed is clamped to the maximum speed in the G96 mode only. It is not clamped in G97 mode. However, the spindle motor is clamped by parameter No. 136 in G97 mode.*
(Note 5) *G92S0; means that the spindle speed is clamped to 0 rpm.*
(Note 6) *The S value specified in G96 mode is stored even after G96 mode has turned to G97 mode and recovered when G97 mode has returned to G96 mode.*

G96S50 : (50 m/min or 50 feet/min)
G97S1000 : (1000 rpm)
G96X3000 : (50 m/min or 50 feet/min)

156

(Note 7) *If the tool length compensation (G43 or G44) is applied in advance, the constant surface speed calculation is made based on the programmed coordinate value. However, if the tool offset (G45~G48) is applied in advance, the present value is employed.*

(Note 8) *With machine lock, the constant surface speed is calculated according to a change of the coordinate values of the axis to which the constant surface speed control is applied, even if the machine tool does not move.*

(Note 9) *The constant surface speed control is also effective during threading. Accordingly, it is recommended to invalidate the constant surface speed control with G97 command before starting the face threading and taper threading, because the response problem in the servo system may be left out of consideration when the spindle speed changes.*

(Note 10) *The constant surface speed control mode (G96) is allowable in G94 (feed per minute).*

(Note 11) *If S (rpm) is not specified in advance in G97 block when G96 mode has turned to G97 mode, the last spindle speed in G96 mode is employed as S in G97 mode.*

N111G97S800; 800rpm

⋮

N222G96S100; 100m/min

⋮

N333G97; Xrpm

X shows spindle speed Xrpm in the block before N333. In other words, the spindle speed remains unchanged when the mode has changed from G96 to G97.
The S value specified last in G96 mode becomes effective when G97 turns to G96.
S = 0 m/min (feed/min), if S is not specified.

8.3 Tool Function (T Function)

The tool function is commanded by 2 or 4 digits code following address T. The relationship between T codes and tools are specified by machine tool builders.

8.4 Miscellaneous Function (M Function)

When a 2-digit figure is specified following address M, a 2-digit BCD code signal and a strobe signal are transmitted. These signals are used for ON/OFF control of a machine function. One M code can be specified in one block. When two or more M codes are specified, only the last one is effective. Selection of M codes for functions varies with the machine tool builder.

The following M codes indicate special meaning.

(1) M02, M30: End of program

(i) This indicates the end of the main program and is necessary for registration of NC commands from tape to memory.

(ii) Cycle operation is stopped and the NC unit is reset. (This differs with the machine tool builder.)

(iii) Only M30
The NC tape is rewound to the start of the program in both memory and tape operation. However, when using a tape reader without reels, the tape is not rewound. When using a tape reader with reels, the tape returns to the ER (%) code at the start of the tape even if several programs exist. (This differs with the machine tool builder. Some machines indicate tape rewind with M02.)

(2) M00: Program stop

Cycle operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The cycle operation can be restarted by specifying an NC start. (This differs with the machine tool builder.)

(3) M01: Optional stop

Similarly to M00, cycle operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine control panel has been pressed.

(4) M98: Calling of subprogram

This code is used to enter a subprogram. See section 9 for details.

(5) M99: End of subprogram

This code indicates the end of a subprogram. Executing M99 returns control to the main program. See section 9 for details.

(Note 1) If there is a block following M00, M01, M02, or M30, it is not read into the buffer storage. Similarly, two M codes which cause the block following them to be ignored can be used by setting a parameter. Refer to the manual issued by the machine tool builder for these M codes.

(Note 2) When executing M98 or M99, a code signal and a strobe signal are not transmitted.

(Note 3) M codes except for M98 and M99 are processed by the machine tool. Nothing is processed only by the NC unit. Refer to the manual issued by the machine tool builder.

8.5 2nd Miscellaneous Function (B Function)

Indexing of the table is performed by address B and a following 3-digit number. The relationship between B codes and the corresponding indexing differs according to the machine tool builders.

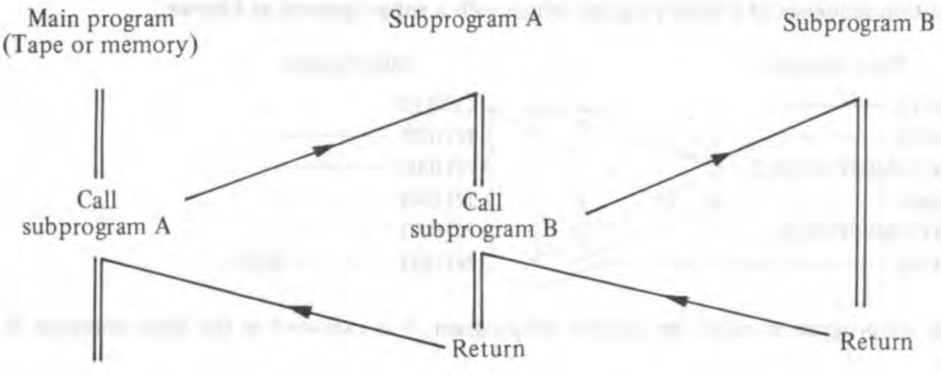
154

9. SUBPROGRAMS

When a program contains certain fixed sequences or frequently repeated patterns, these sequences or patterns may be entered into memory as a subprogram to simplify programming.

The subprogram can be called in either tape command mode or memory command mode. A subprogram can call another subprogram.

When the main program calls a subprogram, it is regarded as a one loop subprogram call. However, a two loop subprogram call can be executed as shown below.



A call command can be used to call a subprogram repeatedly. A call command can specify up to 9999 repeations of a subprogram.

9.1 Preparation of Subprogram

A subprogram is prepared in the following format:

```
O (:) XXXX ;
.................................;
.................................;
             :
.................................;
M99 ;
```

At the top of a subprogram, a subprogram No. identifying the subprogram is specified after 'O' (EIA) or ':' (ISO). Subprogram end command 'M99' need not be specified in a block by itself.

(Example)

X .... M99 ;

For how to enter subprograms into memory, see Chapter IV Sections 5.17 to 5.19.

(Note 1) For compatibility of the NC tape with other devices, 'Nxxxx' is also used as a subprogram No. instead of O (:).
A sequence number under N is registered as a subprogram number.

9.2 Subprogram Execution

A subprogram is executed when called by the main program or another subprogram.
A subprogram call has the following format:

```
M98P ...........L...........;
     ↑          ↑
     |          └──────────── Number of times the subprogram is to be repeated.
     └─────────────────────── Subprogram No.
```

When 'L' is omitted, the subprogram is repeated once.

(Example)   M98P 1002L5;

This command is read "Call the subprogram (subprogram number 1002) five times."
The subprogram call command (M98P____L____) and a move command can be specified in the same block.

(Example) X1000 M98P 1200;

In this example, the subprogram (subprogram number 1200) is called after completing movement in the X-axis direction.

(Example)

The execution sequence of a main program which calls a subprogram is as follows:

```
Main program                      Subprogram
N0010 ————— ;                     O1010;              ;
N0020 ————— ;                     N1020 ————— ;
N0030M98P1010L2;                  N1030 ————— ;
N0040 ————— ;                     N1040 ————— ;
N0050M98P1010;                    N1050 ————— ;
N0060 ————— ;                     N1060 ————— M99;
```

When the subprogram is called by another subprogram, it is executed in the same sequence as shown in the above example.

(Note 1) M98 and M99 signals are not issued to the machine tool.

(Note 2) If the subprogram number specified by address P can not be found, alarm No. 78 is generated.

(Note 3) The subprogram call command 'M98P____;' cannot be input from the MID. In this case, prepare the main program in EDIT mode as follows:

O xxxx ;
M98P xxxx ;
M02 ;

Then execute it in the memory mode.

(Note 4) A single block stop is not effective in blocks containing M98P____; M99; but is effective when the block contains addresses other than O, N, L and P.

9.3 Notes on Subprogram Control

The following special specifications can be used to control subprogram execution.

9.3.1

When a sequence number is specified in address P of the last block of a subprogram, control does not return to the block after the block in which the subprogram was called, but rather to the block with the sequence number specified in address P. However, this is only effective in memory operation.

The processing time for return to the specified block is considerably longer than that for normal return.

```
Main program                      Subprogram
N0010 ————— ;                     O1010 ; ————— ;
N0020 ————— ;                     N1020 ————— ;
N0030 ————— ;                     N1030 ————— ;
N0040M98P1010 ;                   N1040 ————— ;
N0050 ————— ;                     N1050 ————— ;
N0060 ————— ;                     N1060 ————— ;
N0070 ————— ;                     N1070M99P0070 ;
```


160

9.3.2

If the M99 command is executed in the main program, control returns to the start of the main program.

For example, a '/M99;' block is inserted in the proper position of the main program and the optional block skip is off, M99 is executed, and control returns to the start of the main program and the program is executed again.

If the optional block skip is turned on, '/M99;' is omitted and control is passed to the next block.

If '/M99Pn;' has been inserted, control does not return to the start, but returns to the block whose sequence number is 'n'. The processing time for return to sequence number n is longer than return to the start.

```
                          N0010 ———————— ;
                          N0020 ———————— .
                       ┌— N0030 ———————— ;
Optional block         │  N0040 ———————— ;
skip OFF               │  N0050 ———————— ;
                       │  N0060 ———————— ;  ┐  Optional block
                       └—/N0070 M99P0030 ;  │  skip ON
                          N0080 ———————— ;  ┘
                          N0090 M02;     ;
```

9.3.3

It is possible to execute a subprogram from the start by specifying a search for it from the MDI keyboard as well as the main program. (See section IV 5.14 for search)

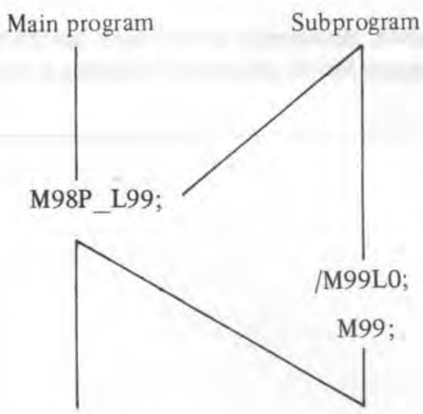
In this case, if the M99 command is executed, control returns to the top of the subprogram and execution is repeated. If 'M99Pn;' is executed, control returns to the block whose sequence number is 'n' and execution is repeated.

In the above operation, if you want to stop execution, insert '/M02;' or the '/M30;' at the appropriate position. When the optional block skip switch is turned off, the above command is executed and the program is terminated. In the example below, execution will continue while the optional block skip is on and will stop when the optional block skip is turned off.

```
   N1040 ————————;
┌→ N1050 ————————;
│  N1060 ————————;
│ /N1070 M02;    ;
└— N1080M99P1050 ;
```

9.3.4 M99Lα;

Execution of the above command changes repetitive count L in the subprogram calling block to α repetitions.



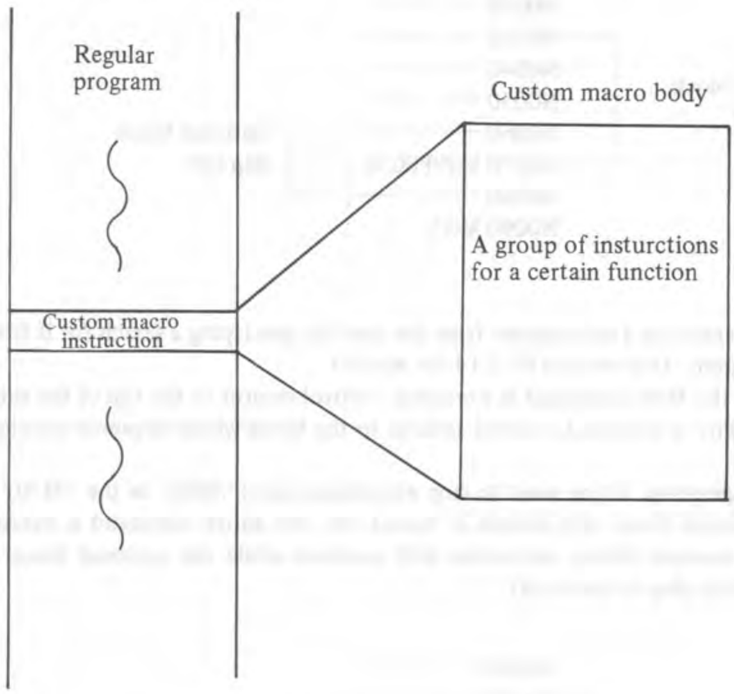
If the optional block skip is turned off in this program, the repetitive count becomes zero and control is passed to the main program when the subprogram end command (M99) is executed.

10. CUSTOM MACRO A AND B

10.1 General

Custom macros A and B are basically the same with the functional differences described in Section 10.10 (9).

A function covering a group of instructions is stored in memory the same as a subprogram. The stored function is represented by one instruction, so that only the representative instruction need be specified to execute the function. This group of registered instructions is called a "custom macro body" and the representative instruction is called a "custom macro instruction". The custom macro body may simply be called a macro. And the custom macro instruction may be called a macro call command.

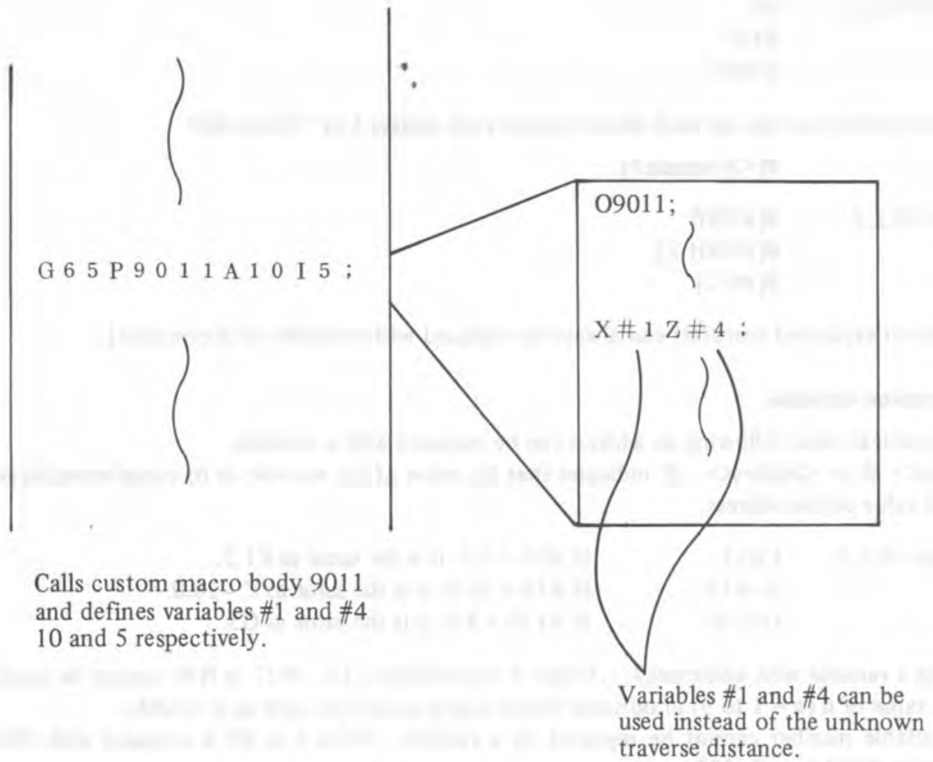


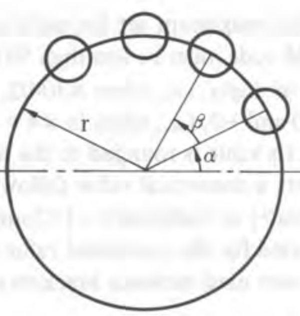
Programmers need only remember representative macro instructions without having to remember all the instructions in a custom macro body.

The three most significant points on custom macros are that variables can be used in the custom macro body, operations can be performed on variables and actual values can be assigned to the variables in custom macro instructions.

> **Request:**
>
> Machine tools builders are requested to attach your custom macro program tape or program list to the NC unit without fail.
>
> If it is necessary to replace PCB due to a failure, FANUC servicemen or end users operators in charge of maintenance should know the contents of custom macro for the purpose of repairing a trouble immediately.

162





Bolt hole circles as shown in the above figure can be made easily.

Once a custom macro body for the bolt hole circle is programmed and registered, NC can operate as if it has the bolt hole circle cutting function.

Programmers can use the bolt hole circle function by using the following command only:

G65P __p__ R __r__ A __α__ B __β__ K __k__ ;

p : Macro number of bolt hole circle
r : Radius
α : Start angle
β : Angle between circles
k : Number of circles

This enables the user to improve the NC performance by themselves. Custom macro bodies may be provided by machine tool builder but they can be created by users, too.

10.2 Variables

Variables can be used in the macro instead of numerical data. The user can assign any value (within allowed range) to them. Using variable allows custom macros to become much more flexible than the conventional subroutines.

Several variables can be used and each variable is identified by a variable number.

10.2.1 Variable expressions

A variable is composed of the code # and a number as shown in Ex. 10.2.1.

#i (i = 1, 2, 3, 4 . . . .)

Example 10.2.1: #5
#109
#1005

Following format can also be used where numbers are replaced by <Formula>

#[<Formula>]

Example 10.2.2: #[#100]
#[#1001-1]
#[#6/2]

Variable #i explained hereafter can always be replaced with variable #[<Formula>].

10.2.2 Quotation variables

The numerical value following an address can be replaced with a variable.
<address> #i or <address> –#i indicates that the value of the variable or its complement is substituted for the command value of the address.

Example 10.2.3:	F#33	If #33 = 1.5, it is the same as F1.5.
	Z–#18	If #18 = 20.0, it is the same as Z –20.0.
	G#130	If #130 = 3.0, it is the same as G3.

(1) Using a variable with addresses /, :, O and N is prohibited, i.e., :#27 or N#1 cannot be used.
The value of n (n = 1 to 9) in optional block skip/n cannot be used as a variable.
(2) A variable number cannot be replaced by a variable. When 5 in #5 is replaced with #30, it does not become ##30 but #[#30].
(3) The value of a variable cannot exceed the maximum set for each address. For example, when #140 = 120, M#140 exceeds the maximum (an M code must be less than 99).
(4) Identification is not capable by number of digits, i.e., when #30-02, F#30 is regarded as F2.
(5) Identification is not capable between –0 and +0, i.e., when in #4 = –0, X#4 is regarded as X0.
(6) When a variable is used for address data, its value is rounded to the nearest.
(7) By using <Formula> as explained later, a numerical value following the address can be also replaced with <Formula>. <address> [<Formula>] or <address> – [<formula>] indicates that the value of the <formula> or its complement is substituted for the command value of the address.
Note that a constant with no decimal point used between brackets is assumed to have a decimal point at its end.

Example 10.2.4: X[#24+#18*COS[#1]]
Z–[#18+#26]

10.2.3 Undefined variables

The value of a variable which has not yet been defined is called <vacant>. Variable #0 is used as a variable that is always <vacant>.

An undefined variable has the following nature:

(1) Quotation

When an undefined variable is quoted, the address itself is also ignored.

When #1 = <vacant>	When #1 = 0
G90X100Z#1 ↓ G90X100	G90X100Z#1 ↓ G90X100Z0

(2) Operation

<vacant> is the same as 0 except when replaced by <vacant>.

When #1 = <vacant>	When #1 = 0
#2=#1 ↓ #2=<vacant>	#2=#1 ↓ #2=0
#2=#1*5 ↓ #2=0	#2=#1*5 ↓ #2=0
#2=#1+#1 ↓ #2=0	#2=#1+#1 ↓ #2=0

(3) Conditional expressions

<vacant> differs from 0 only for EQ and NE.

When #1 = <vacant>	When #1 = 0
#1 EQ #0 ↓ Established	#1 EQ #0 ↓ Not established
#1 NE 0 ↓ Established	#1 NE 0 ↓ Not established
#1 GE #0 ↓ Established	#1 GE #0 ↓ Established
#1 GT 0 ↓ Not established	#1 GT 0 ↓ Not established

10.2.4 Display and setting variable values

Variable values can be displayed on the CRT screen. Variable values can also be set in the MDI mode. See Chapter IV Section 5.8.2.

10.3 Types of Variables

Variables are classified into local variables, common variables and system variables, depending on the variable number. Usage and properties are different for each type.

10.3.1 Local variable #1 to #33

The local variable is a variable locally used in the macro. That is, a local variable #i used in the macro called at one point in time, is different from #i used in the macro (regardless of whether it is the same macro or different one) called at another point in time. Accordingly, when macro B is called from macro A, as in a multiplex call, a local variable used in macro A is never destroyed by being used in macro B.

A local variable is used for argument transfer. For information on the correspondence to the argument address, see section 10.7. A local variable without a transferred argument is vacant in its initial status and can be freely used by the user.

10.3.2 Common variable #100 to #149, #500 to #509

Compared with a local variable used locally in the macro, the common variable is in common throughout the main program, each subprogram called from the main program and each macro. That is, #i used in a certain macro is the same as #i used in the other macro. Accordingly the calculated value of a common variable #i in a certain macro can be used in an other macro.

The use of common variables is not specially determined in the system. They can be freely used by the user.

Common variables #100 to 149 are cleared when the power goes off; however, variables #500 to #509 cannot be cleared by turning off the power.

10.3.3 System variables (For custom macro B)

Use of a system variable is fixed in the system.

(1) Interface signals #1000 to #1015 and #1032, #1100 to #1115 and #1132

[Input signal]

The status of the interface input signal is determined by reading the values of system variables #1000 to #1032.

System variable	Interface input signal
#1000	2^0 UI 0
#1001	2^1 UI 1
#1002	2^2 UI 2
#1003	2^3 UI 3
#1004	2^4 UI 4
#1005	2^5 UI 5
#1006	2^6 UI 6
#1007	2^7 UI 7
#1008	2^8 UI 8
#1009	2^9 UI 9
#1010	2^{10} UI 10
#1011	2^{11} UI 11
#1012	2^{12} UI 12
#1013	2^{13} UI 13
#1014	2^{14} UI 14
#1015	2^{15} UI 15

Value of variable	Input signal
1	Contact closed
0	Contact open

Since the variable value read is 1.0 or 0.0 regardless of the unit system, the unit system must be considered in preparing a macro.

Reading system variable #1032 is used to read all input signals at one time.

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

System variables #1000 to #1032 cannot be used as a left side term in a calculation command.

166

[Output signal]

Interface output signals can be issued by assigning values to system variables #1100 to #1132.

System variable	Interface output signal
#1100	2^0 UO0
#1101	2^1 UO1
#1102	2^2 UO2
#1103	2^3 UO3
#1104	2^4 UO4
#1105	2^5 UO5
#1106	2^6 UO6
#1107	2^7 UO7
#1108	2^8 UO8
#1109	2^9 UO9
#1110	2^{10} UO10
#1111	2^{11} UO11
#1112	2^{12} UO12
#1113	2^{13} UO13
#1114	2^{14} UO14
#1115	2^{15} UO15

Value of variable	Output signal
1	Contact closed
0	Contact open

All output signals can be issued at one time by assigning a value to system variable #1132.

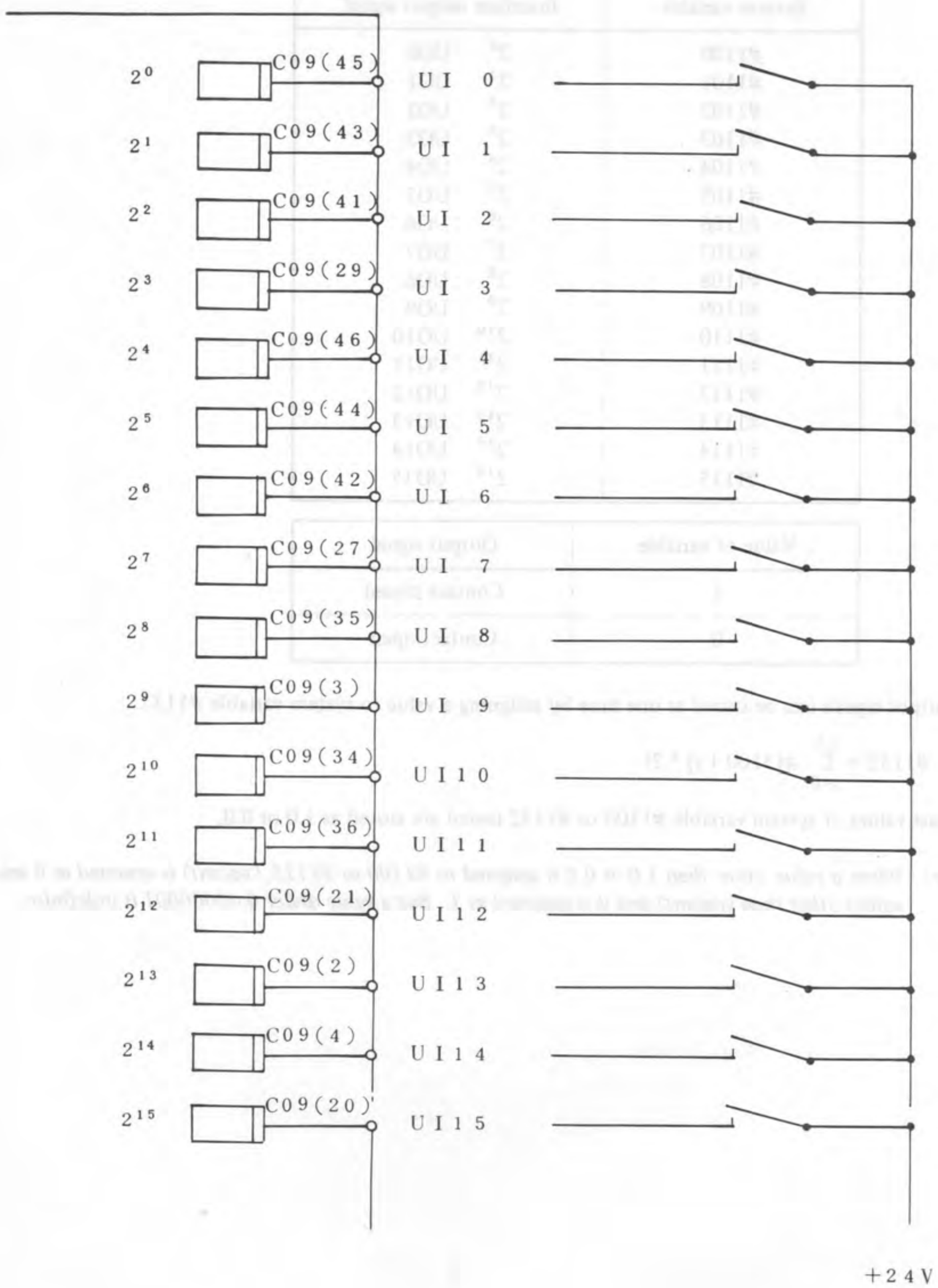
$$\#1132 = \sum_{i=0}^{15} \#[1100 + i] * 2^i$$

The last values of system variable #1100 to #1132 issued are stored as 1.0 or 0.0.

(Note) *When a value other than 1.0 or 0.0 is assigned to #1100 to #1115, ⟨vacant⟩ is assumed as 0 and values other than ⟨vacant⟩ and 0 is assumed as 1. But a value under 0.00000001 is indefinite.*

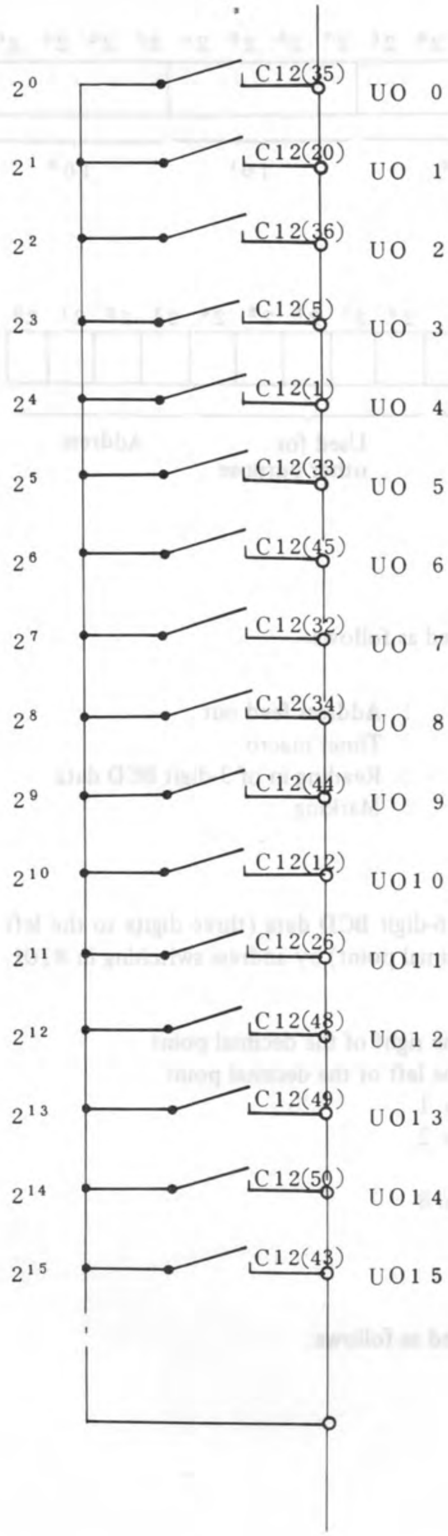
167

(Note 1) The following diagram shows the connection of input signals for a custom macro.



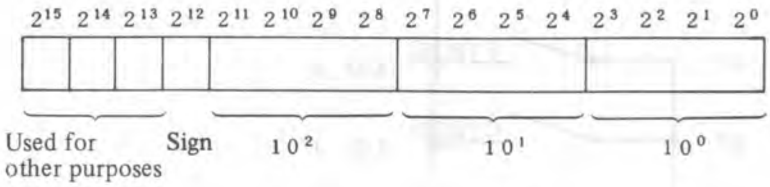
168

(Note 2) The following diagram shows the connection of input signals for a custom macro.

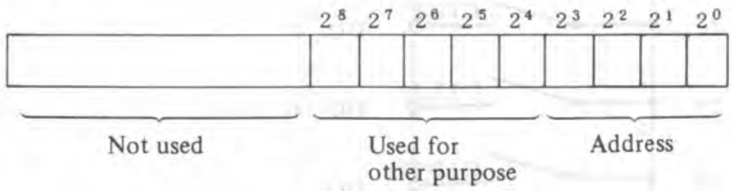


[Example 10.3.1] ① Macro used to read three digits of BCD data with a sign by address switching.

Structure of DI



Structure of DO



Macro call command

G65 P9100 D (address) ;

The custom macro body is specified as follows:

```
O 9100 ;
#1132=#1132 AND 496  OR#7 ;      : Address feed out
G65 P9101 T60 ;                  : Timer macro
#100=BIN[#1032 AND 4095] ;       : Reading in of 3-digit BCD data
LF [#1012EQ 0] GOTO 9100 ;       : Marking
#100=-#100 ;
N9100 M99 ;
```

② Read eight types of signed 6-digit BCD data (three digits to the left of the decimal point + three digits to the right of the decimal point) by address switching in #101.

Machine tool composition

When DO 2^0 = 0, three digits to the right of the decimal point
= 1, three digits to the left of the decimal point
When DO 2^3 to 2^1 = 000, data No. 1
001, data No. 2
⋮
111, data No. 8

Macro call command

G65 P9101 D (data number) ;

The custom macro body is specified as follows:

```
O 9101 ;
G65 P9100 D [#7*2+1] ;
#101=#100 ;
G65 P9100 D [#7*2] ;
#101=#101+#100/1000 ;
M99 ;
```

(2) Tool offset amount #2000 to #2200, work offset amount #2500 to #2906

The tool offset amount and shift amount can be determined by reading the value of system variable #2001 to #2200 for the tool offset and the value of system variable #2500 and #2906 for the shift amount of the work coordinate system. The offset and shift values can also be modified by assigning a value to system variable #i.

Tool offset number	Tool offset value
1	# 2001
2	# 2002
3	# 2003
:	:
199	# 2199
200	# 2200

#2000 can be read but it is always 0.

Axis	Work offset No.	Work offset amount
X	External work offset	#2500
	G54	#2501
	|	|
	G59	#2506
Y	External work offset	#2600
	G54	#2601
	|	|
	G59	#2606
Z	External work offset	#2700
	G54	#2701
	|	|
	G59	#2706
4th	External work offset	#2800
	G54	#2801
	|	|
	G59	#2806
5th	External work offset	#2900
	G54	#2901
	|	|
	G59	#2906

[Example 10.3.2] #30=#2005

Tool offset amount of offset number 5 is assigned to variable #30.
When the offset amount is 1.500 mm (0.1500 inch), the value of #30 becomes 1.5 (0.15).

#2210=#8

The compensation amount for offset number 10 is made equal to the value of variable #8.

(3) Alarm #3000

When detecting an error in the macro, an alarm can be generated. When an alarm number is specified in system variable #3000, the alarm lamp is turned on and the alarm status is entered after the preceeding block is processed.

#3000 = n (ALARM MESSAGE) ;

A number not previously used should be selected as the alarm number set in the custom macro.(n<200)
An alarm message of less than 26 characters can be specified in the section between control-out and control-in.

(4) Clock #3001, #3002

The clock time can be determined by reading the values of system variables #3001, #3002. The time can be preset by assigning a value to the system variable.

Kind	System variable	Unit time	At the time of power-on	Counting
Clock 1	#3001	1 msec.	Reset to 0	Always
Clock 2	#3002	1 hour	Same as that at the time of power-off.	While STL signal is on.

The accuracy of each clock is within 16 msec. Clock 1 is set again to zero at 65536 msec. Clock 2 continues to increase unless it is preset.

When exceeding the maximum value of 9544 hours, the time cannot be correctly determined.

[Example 10.3.3] Timer

Macro call command

G65 P9101 T (Awaiting time) msec;

This macro may be specified as follows:

```
O 9101;
#3001 = 0;                    : Initial setting
WHILE [#3001 LE #20] DO1; :  Wait for the prescribed length of time
END1;
M99;
```

(5) Suppression of single block stop and wait for the auxiliary function end signal

When the following values are assigned to system variable #3003, the single block stop function is suppressed and execution advances from one block to the next without awaiting the end signal (FIN) of auxiliary functions (S, T, M and B). When the end signal is not awaited, the distribution end signal (DEN) is not transmitted. Be careful not to specify a subsequent auxiliary function without awaiting the end signal.

#3003	Single block stop	Auxiliary function end signal
0	Not suppressed	Awaited
1	Suppressed	Awaited
2	Not suppressed	Not awaited
3	Suppressed	Not awaited

[Example 10.3.4] Drill cycle (For incremental programming)
(Equivalent to G81)

Macro call command

G65 P9081 L (Repetition time) R (R point) Z (Z-point);

The custom macro body is specified as follows:

```
O9081 ;
#3003=1 ;
G00Z #18;
G01Z #23;                                   Single block stop is not executed. #18 corresponds
G00Z–[ROUND[#18] + ROUND [#23] ];           to R and #23 corresponds to Z.
#3003=0;
M99 ;
```

(6) Feed hold, feedrate override, and exact stop check suppression specified in #3004

When the following values are assigned to system variable #3004 feed hold and feedrate override are suppressed for subsequent blocks and the exact stop check is not performed. Pressing the feed hold button during execution of a block for which feed hold has been suppressed:

① Executes feed hold at the start of the first block outside the suppression range, when the button is kept pushed.

② Lights the feed hold lamp and does not execute feed hold at this point but at the end of the first block outside the suppression range, when the button is pushed and released.

#3004	Feed hold	Feedrate override	Exact stop check
0	O	O	O
1	X	O	O
2	O	X	O
3	X	X	O
4	O	O	X
5	X	O	X
6	O	X	X
7	X	X	X

O: Effective, X: Suppressed

(Example 10.3.5) Tapping cycle (For incremental programming)
(Equivalent to G84)

Macro call command

G65 P9084 L (Repetition times) R (R point) Z (Z point);

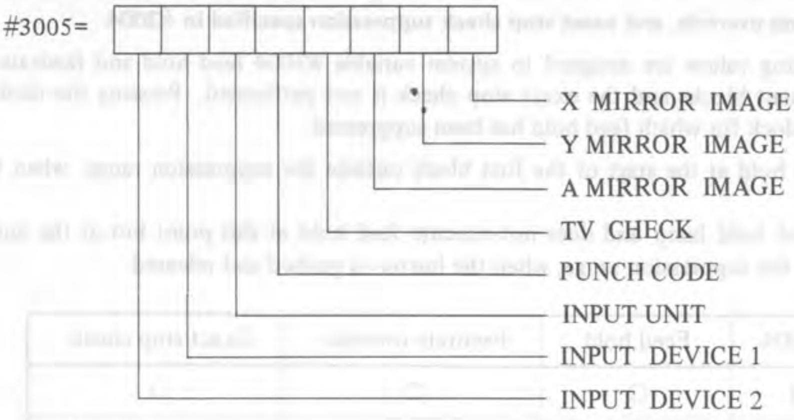
Custom macro body is made as follows.

```
O9084;
#3003=1;       : Suppression of single block stop
G00Z #18;
#3004=7;
G01Z #26;      } Feed hold, feedrate override and exact stop check are ineffective
M05;
M04;
Z-#26;
#3004=0;
M05;
M03;
G00 Z-#18;
#3003=0;
M99;
```

(Note) "M05" can be omitted for some machine tools.

(7) Variable corresponding to setting data #3005

Setting data is set by assigning a value to system variable #3005. When a value of system variable #3005 is expressed with a binary number, each digit corresponds to each setting data.



[Example]

If a command of #3005 = 55; is executed, setting data is set as follows:

X MIRROR IMAGE = 1
Y MIRROR IMAGE = 1
A MIRROR IMAGE = 1
TV CHECK = 0
PUNCH CODE = 1
INPUT UNIT = 1
INPUT DEVICE 1 = 0
INPUT DEVICE 2 = 0

(8) Modal information #4001 to #4120

Present values of modal commands can be determined by reading the values of system variables #4001 to #4120. The unit specified is effective.

System variable	Modal information
#4001	G code (group 01)
:	:
#4021	G code (group 21)
#4102	B code
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Sequence number
#4115	Program number
#4119	S code
#4120	T code

(Example 10.3.6) Combined use of incremental/absolute programming

Boring cycle (Equivalent to G86)

Macro call command

G65 P9086 L (Repetition times) R (R point) Z (Z point);

Custom macro body must be made as follows.

```
O9086;
#1=#4003;          : 03 group G code stored.
#3003=1;           : Suppression of single block stop.
G00 G91 Z #18;
G01 Z #26;
M05;
G00 Z- [#18 + #26] ;
```

M03;
#3003=0;
G#1 M99; : Restoration of 03 group G code.

System variable #4001 to #4120 can not be used as a lefthand-side term of the arithmetic command.

(9) Positional information #5001 to #5105

Positional information can be determined by reading system variables #5001 to #5015. The unit of positional information is mm or inches depending on the input system.

System variables #5001 to #5105 cannot be used as the left-hand-side term of an arithmetic command.

System variable	Positional information	Reading in during movement
# 5001 # 5002 # 5003 # 5004 # 5005	X axis block end coordinate (ABSIO) Y axis block end coordinate (ABSIO) Z axis block end coordinate (ABSIO) 4th axis block end coordinate (ABSIO) 5th axis block end coordinate	Possible
# 5021 # 5022 # 5023 # 5024 # 5025	X axis present coordinate (ABSMT) Y axis present coordinate (ABSMT) Z axis present coordinate (ABSMT) 4th axis present coordinate (ABSMT) 5th axis present coordinate (ABSMT)	Impossible
# 5041 # 5042 # 5043 # 5044 # 5045	X axis present coordinate (ABSOT) Y axis present coordinate (ABSOT) Z axis present coordinate (ABSOT) 4th axis present coordinate (ABSOT) 5th axis present coordinate (ABSOT)	Impossible
# 5061 # 5062 # 5063 # 5064 # 5065	X axis skip signal position (ABSKP) Y axis skip signal position (ABSKP) Z axis skip signal position (ABSKP) 4th axis skip signal position (ABSKP) 5th axis skip signal position (ABSKP)	Possible
# 5083	Tool length offset value	Impossible
# 5101 # 5102 # 5103 # 5104 # 5105	X axis servo position deviation Y axis servo position deviation Z axis servo position deviation 4th axis servo position deviation 5th axis servo position deviation	Impossible

Abbreviation	ABSIO	ABSMT	ABSOT	ABSKP
Meaning	End point coordinate of the preceding block	Command present coordinate (Equal to POS. MACHINE display)	Command present coordinate (Equal to POS. MACHINE display)	Position where skip signal has turned on in the G31 block
Coordinate system	Work coordinate system	Machine coordinate system	Work coordinate system	Work coordinate system
Tool offset	Not considered	Considered	Considered	Considered
Tool length offset Cutter compensation	Tool top position	Tool standard point	Tool standard point	Tool standard point

(Note) The tool length offset amount is not the amount in effect just before the block is executed; it is the amount for the current block. The skip signal position is the end point of the block if no skip signal is input in block G31.

[Example 10.3.6]

The tool moves to a point appropriate to the machine tool (xp, yp, zp distant from a reference point) through a programmed intermediate point and after processing a sequence of operations, returns to the original point.

Macro call command

G65 P9300 X (intermediate point) Y (intermediate point) Z (intermediate point);

The custom macro body is specified as follows:

```
O9300;
#1=#5001;
#2=#5002;
#3=#5003;
G00 Z#26;
X#24 Y#25 ;
G04;   (Move is interrupted to read #5021 to #5023)
G91 X[xp–#5021] Y [yp–#5022] Z [zp–#5023];
 .
 .
 :              (processing)
 .

X#24 Y#25 Z#26;
X#1 Y#2;
Z#3;
M99;
```

(10) Display and setting of variable names

A name of up to 8 characters can be assigned to variables #500 to #511 using the following command.

SETVN n [$\alpha_1 \alpha_2$ . . . . . α_8, β_1 β_2 . . . . . β_8, ...] ;

n is leading variable number of variables to which names are assigned. $\alpha_1 \alpha_2$ . . . . . α_8 is a name of variable number n, $\beta_1 \beta_2$ . . . . . β_8 is a name of variable number n + 1, and so on.

Discriminate character strings with " , ".

All the codes usable in the significant information zone can be used for characters except for the following:

Control in, Control out, [,], EOB, EOR, :

Variable names are not cleared by power off.

*(Note) This function may not be used depending on a system.
In this case, variables #510 and #511 cannot be used.*

(Example)

SETVN 500 [ABCDEFGH, COUNTER, POINTER] ;

```
MACRO    VAL:          06: O1234      N3456

No.          Name              Data
0500     ABCDEFGH          – 1234. 5678
0501     COUNTER             00020. 000
0502     POINTER
0503     1ST               – 0000. 4025
0504     2ND                 00004. 500
0505                         124000. 00
0506
0507
0508     START
0509
0510     TOOL – PT           000045. 00
0511
P                               LSK
```


176

10.4 Arithmetic Commands

A variety of arithmetic operations can be performed on variables. An arithmetic command must be specified the same as in general arithmetic expressions.

#i = <Formula>

<Formula>, the right-hand-side of an arithmetic command is a combination of constants, variables, functions and operators. A constant can be used instead of #i and #k. A constant without a decimal point used in <Formula> is considered to have a decimal point at the end.

10.4.1 Definition and substitution of variables

#i=#j	Definition, substitution

10.4.2 Addition arithmetic

#i=#j+#k	Sum
#i=#j–#k	Subtraction
#i=#j OR #k	Logical sum (at every bit of 32 bits)
#i=#j XOR #k	Exclusive OR (at every bit of 32 bits)

10.4.3 Multiplication arithmetic (Custom macro B option)

#i=#j*#k	Product
#i=#j/#k	Quotient
#i=#j AND #k	Logical product (at every bit of 32 bits)

10.4.4 Functions (Custom macro B option)

#i=SIN [#j]	Sine (degree unit)
#i=COS [#j]	Cosine (degree unit)
#i=TAN [#j]	Tangent (degree unit)
#i=ATAN [#j]/[#k]	Arctangent (degree unit)
#i=SQRT [#j]	Square root
#i=ABS [#j]	Absolute value
#i=BIN [#j]	Conversion from BCD to BIN
#i=BCD [#j]	Conversion from BIN to BCD
#i=ROUND [#j]	Rounding off
#i=FIX [#j]	Discard fractions less than 1
#i=FUP [#j]	Add 1 for fractions less than 1

(Note) How to use function ROUND

(1) If function ROUND is employed in an arithmetic operation command or in an IF or WHILE conditional expression, the figure in function ROUND is rounded (counting fractions of 5 and over as a unit and disregarding the rest) as ordinary data with a decimal point.

(Example)

#1 = ROUND [1.2345];

#1 becomes 1.0.

IF [#1 LE ROUND [#2]] GOTO 10;

ROUND [#2] is 4.0 if #2 is 3.567.

(2) If function ROUND is employed in a command to an address, it is rounded to the least input increment of the address.

(Example)

G01 X [ROUND [#1]];

If #1 is 1.4567 and the least input increment of X is 0.001, this block becomes G01 X 1.457.

In this example, this command is the same as G01 X #1; command.

Function ROUND in an address command is mainly used in the following case.

(Example)

[Program to move incrementally by #1 and #2 only and then return to the starting point]

N1 #1 = 1.2345;

N2 #2 = 2.3456;

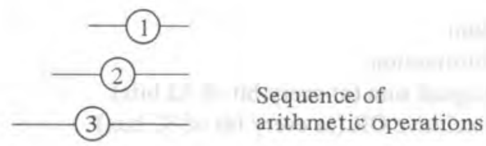
N3 G01 X #1 F100; : *X moves 1.235.*
N4 X #2; : *X moves 2.346.*
N5 X –[#1 + #2] ; : *X moves –3.58, since #1 + #2 is 3.5801.*

Since 1.235 + 2.346 = 3.581, the program does not return to the starting point by N5.
Assume N5 X –[ROUND [#1] + ROUND [#2]] ;
It becomes equal to N5 X –[1.235 + 2.346] ; and the program returns to the starting point.

10.4.5 Combination of arithmetic operations

The above arithmetic operations and functions can be combined. Priority in an arithmetic operation is in the order of function, multiplication arithmetic then addition arithmetic.

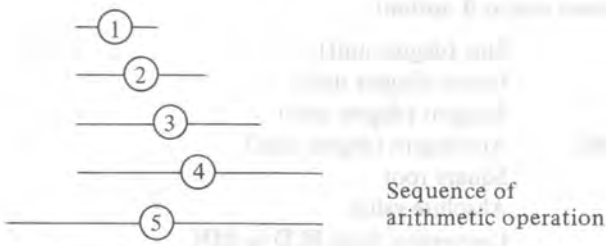
[Example 10.4.1] #i=#j+#k*SIN [#ℓ]



10.4.6 Modification of arithmetic sequence using [ ]

A portion to be assigned priority in an arithmetic sequence can be enclosed in [ ].
[ ] can be nested up to five times (including [ ] used in functions).

[Example 10.4.2] #i=SIN[[[#j+#k]*#ℓ+#m]*#n] (Three fold nesting)



10.4.7 Precision

Always consider the precision of a custom macro function used for preparation of programs.

(1) Data format

Numeric data handled by a custom macro is in floating decimal point format as follows:

$M*2^E$

where M: 1-bit sign + 31-bit binary data
E: 1-bit sign + 7-bit binary data

(2) Operational precision

An operation executed once generates the following error. These errors are accumulated with each repeated operation.

Operation format	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error $\left\|\frac{\epsilon}{a}\right\|$
a = b/c	4.66×10^{-10}	1.86×10^{-9}	
$a = \sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a = b + c a = b – c	2.33×10^{-10}	5.32×10^{-10}	min. $(\left\|\frac{\epsilon}{b}\right\|, \left\|\frac{\epsilon}{c}\right\|)$
a = SIN b a = COS b	5.0×10^{-9}	1.0×10^{-8}	Absolute error $\left\|\epsilon\right\|$ degrees
a = ATAN b/c	1.8×10^{-6}	3.6×10^{-6}	

(Note) Function TAN performs SIN/COS.

10.4.8 Notes on decreased precision

(1) Addition and subtraction

Note that when absolute values are used subtractively in addition or subtraction, the relative error cannot be held under 10^{-8}. For example, suppose that the real values of #1 and #2 are as follows.

#1 = 9876543210123.456
#2 = 9876543277777.777

Performing operation #2 – #1 does not produce.

#2 – #1 = 67654.321,

since the custom macro has a precision of only eight decimal digits, the values of #1 and #2 have a precision as low as approximately

#1 = 9876543200000.000
#2 = 9876543300000.000, respectively. (Strictly, the internal values differ somewhat from the above values because they are binary numbers.)
Consequently,

#2 – #1 = 100000.000

which generates a large error.

(2) Logical operation

EQ, NE, GT, LT, GE, and LE are basically the same as addition and subtraction. Therefore, be careful of errors. To determine whether or not #1 and #2 are equal in the above example, for example,

IF [#1 EQ #2]

is not always evaluated correctly. When the error is evaluated as in

IF [ABS [#1 – #2] LT 50000]

and the difference between #1 and #2 falls within the range error, both values must be considered equal.

(3) Trigonometric functions

Absolute errors occur in trigonometric functions, but since they are not under 10^{-8} be careful in integration or division after using a trigonometric functions.

10.5 Control Command

The program flow can be controlled by using the following commands.

10.5.1 Divergence (GOTO)

IF [<Conditional expression>] GOTO n

When <conditional expression> is satisfied, the next operation is executed in the block with sequence number n in the same program. Sequence number n can be replaced by a variable or <Formula>.

When the condition is not satisfied, control proceeds to the next block.

IF [<conditional expression>] can also be omitted, and when it is, control passes to block n unconditionally.

The following expressions can be used for <conditional expression>.

#j EQ #k =
#j NE #k ≠
#j GT #k >
#j LT #k <
#j GE #k ≧
#j LE #k ≦

<Formula> can be used instead of #j and #k. And a variable or <Formula> can be used instead of n.

(Note 1) *In the block with sequence number "n" which will be executed after a GOTO n command, the sequence number must be at the top of the block.*

(Note 2) *When GOTO n is executed, execution time increases as the distance from the GOTO n block to the Nn block, viewed in the forward direction, increases.*

```
O  ××××
Nn③
 .
Nn④  GOTOn ;

Nn①

Nn②
%
```

In the above diagram, the execution time increases in the order of ①, ②, ③ and ④. Therefore, it is efficient that GOTO n to be executed frequently be programmed nearer to the Nn block, viewed in the forward direction.
When an alarm should be indicated depending on the contents of a variable, it is recommended that the flow in the no-alarm case be programmed short and the flow in the alarm case long.

(Example) *Indicating alarm No. 510 when #1 ≧ 10.*

```
                 .
                 .
        IF [#1 GE10 ]  GOTO 150 ;
              When no alarm
When    M 99 ;
alarm            .
                 .
        N150  #3000 = 150 ;
        M 99 ;
```

Above program can also be stored in memory if a pameter NEOP (No. 306) is set so as not to regard M99 as program end.

(Note 3) *An alarm may be produced in the following cases during execution of GOTO command.*

① *When macro operation cannot be executed correctly in an address.*
If GOTO statement is executed when #1 = – 1, alarm No. 119 may be generated in the following block:
X [SQRT [#1]] ;

② *When a conditional statement specified in WHILE cannot be executed correctly.*
If GOTO statement is executed when #1 = 0, alarm No. 112 may be generated in the following block:
WHILE [10/#1 GE 2] DO 1 ;

In these cases, modify programs as follows:

① *#2 = SQRT [#1] ;*
X# 2 ;

② *#2 = 10/#1 ;*
WHILE [#2 GE 2] DO 1 ;
.
.
#2 = 10/#1 ;
END 1 ;
No alarm may be generated by GOTO.

10.5.2 Iteration (Custom macro B option)

WHILE [<conditional expression>] DO m (m = 1, 2, 3)
END m

While < conditional expression> is satisfied, blocks DO m to END m are repeatedly executed. That is, the condition of DO m block is examined and when the <conditional expression> is satisfied control passes to the next block. But when it is not satisfied, the block after END m is executed.

WHILE [<conditional expression>], can also be omitted as with IF, and if omitted, blocks from DOm to ENDm are executed eternally.

WHILE [<conditional expression>] DO m and END m must be used as a pair, and identification number m identifies this pair.

Example 10.5.1

```
#120 = 1 ;
N1 WHILE [#120 LE 10] DO 1 ;
    N2 WHILE [#30 EQ 1] DO2 ;        <---- Repeated ten times
        N3END 2 ;                    <---- Repeated while variable #30 = 1
#120 = #120 + 1 ;
N4END 1 ;
```

(Note 1) Notes on REPEAT programming.

① *DO m must be specified before END m.*

```
.
.
END 1 ;
.
.        (No)
.
DO 1 ;
```

② *DO m and END m must correspond to each other in one program.*

```
.
.
DO 1 ;
.
.
DO 1 ;      (No)
.
.
END 1 ;
.
.
.
.
DO 1 ;
.
.
END 1 ;     (No)
.
.
END 1 ;
```

③ *The same identification number can be used many times.*

```
        .
        .
        .
    DO 1 ;
        .
        .
        .
    END 1 ;
        .
        .              (Yes)
        .
    DO 1 ;
        .
        .
        .
    END 1 ;
        .
        .
        .
```

④ *DO statement can be nested up to three times.*

```
        .
        .
        .
    DO 1 ;
        .
        .
        .
    DO 2 ;
        .
        .
        .
    DO 3 ;
        .
        .              (Yes)
        .
    END 3 ;
        .
        .
        .
    END 2 ;
        .
        .
        .
    END 1 ;
        .
        .
        .
```

⑤ *DO ranges cannot be intersected.*

```
        .
        .
        .
    DO 1 ;
        .
        .
        .
    DO 2 ;
        .
        .              (No)
        .
    END 1 ;
        .
        .
        .
    END 2 ;
        .
        .
        .
```

⑥ *A branch can be made from inside to outside a DO range.*

```
      .
      .
      .
DO 1 ;
      .
      .
      .
GOTO 9000 ;
      .
      .            (Yes)
      .
END 1 ;
      .
      .
      .
N9000 . . . . . ;
      .
      .
      .
```

⑦ *A branch cannot be made from outside to inside a DO range.*

```
      .
      .
      .
GOTO 9000 ;
      .
      .
      .
DO 1 ;             (No)
      .
      .
      .
N9000 . . . . . ;
      .
      .
      .
END 1 ;
      .
      .
      .
DO 1 ;
      .
      .
      .
N9000 . . . . . ;
      .
      .            (No)
      .
END 1 ;
      .
      .
      .
GOTO 9000 ;
      .
      .
      .
```

⑧ *Custom macro bodies or subprograms can be called from inside a DO range. DO statements can be nested up to three times more in the custom macro body or in the subprogram.*

```
      .
      .
      .
DO 1 ;
      .
      .
      .
G65 . . . . . ;    (Yes)
      .
      .
      .
```


183

G66 . . . . . ; *(Yes)*
.
.
G67 ; *(Yes)*
.
.
END 1 ;
.
.
.
.
DO 1 ;
.
.
M98 . . . . . ; *(Yes)*
.
.
END 1 ;
.
.

(Note 2) *If a program may be created by using either a divergence or an iteration, execution time of the program may become shorter if it is created by using an iteration command.*

[Example 10.5.3] A program to creat a loop of processing in which system waits for a signal (#1000) to become 1.

Program ①
.
.
N10 IF [#1000 EQ 0] GOTO 10;
.
.

Program ②
.
.
WHILE [#1000 EQ 0] D01;
END1

Execution time of program ② is shorter than that of program ②.

10.6 Creation and Registration of Custom Macro Body

10.6.1 Creation of custom macro body

The format of a custom macro body is the same as that of a subprogram as shown below.

O (Program number) ;

Commands

M99 ;

184

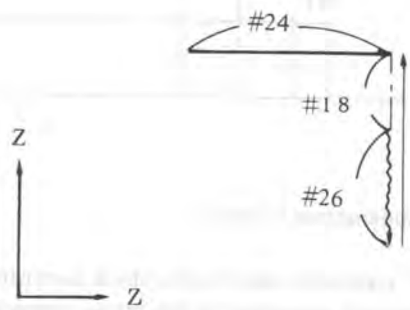
Program numbers are determined as follows:

(1)	O1 ~ O7999	for programs that can be freely registered, cancelled and edited.
(2)	O8000 ~ O8999	for programs that cannot be registered, cancelled, or edited without setting.
(3)	O9000 ~ O9019	for special call-type macros
(4)	O9020 ~ O9899	for programs that cannot be registered, cancelled or edited without setting a parameter
(5)	O9900 ~ O9999	for robot operating programs

Dummy arguments (variables used to accept data from a macro call command) are fixed, i.e. the address which indicates a parameter in a macro call command and a variable in the custom macro body correspond to each other.

[Example 10.6.1]

```
O 9081;
G00X#24;
Z#18;
G01Z#26;
G00Z-[ROUND [#18]
      + ROUND [#26] ];
M99 ;
```



10.6.2 Registration of custom macro body

A custom macro body is a sort of subprogram, and it is registered and edited in the same way as a subprogram.

The memory used to register custom macro bodies is included in the storage capacity of NC.

10.6.3 Macro and NC statements

The following blocks are called macro statements.

(a) Operation command (block including =)
(b) Control command (block including GOTO, DO or END)
(c) Macro call command (block including G65, G66, G67 or G codes calling macro)

Blocks other than macro statements are sometimes referred to as NC statements.

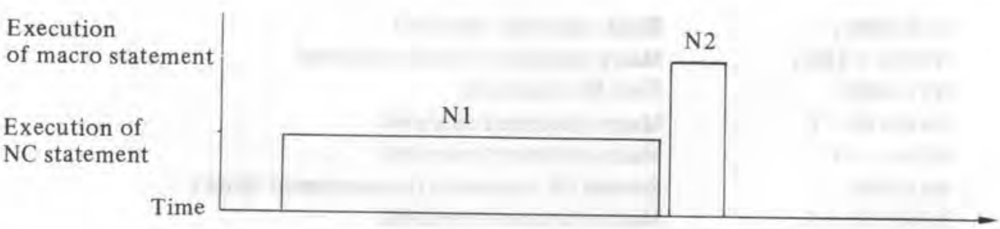
The macro statement differs from the NC statement in the following points:

(a) In the normal single block mode, the single block stop does not occur. (See item (3), Chapter 8)
(b) The macro statement is not regarded as the no movement block in the cutter compensation C.
(c) The time of execution differs.
(c) is described below in further detail.

(i) The macro statement existing next to the block which does not buffer the next one (block of non-buffering M code, or G31 block) is executed after that block is executed.

[Example 10.6.3.1]

N1X1000M00 ;	Block currently executed
N2#1100 = 1 ;	Macro statement



185

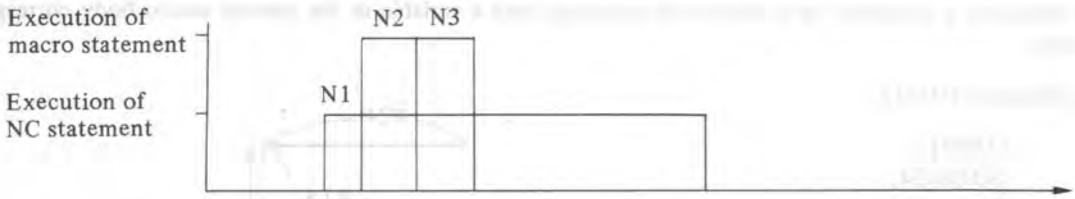
(ii) Macro statement existing next to the block buffering the next one.

- When not in cutter compensation C mode

Immediately when the current block starts to be executed, the next macro statement is executed. The macro statement up to the next NC statement is executed.

[Example 10.6.3.2]

N1G01X1000 ;	Block currently executed
N2#1100 = 1 ;	Macro statement executed
N3#1 = 10 ;	Macro statement executed
N4X2000 ;	Next NC statement



- When in cutter compensation C mode

(2–1)

When the first NC statement next to the block currently executed is not the no-movement block (block containing no movement command in the cutter compensation plane).

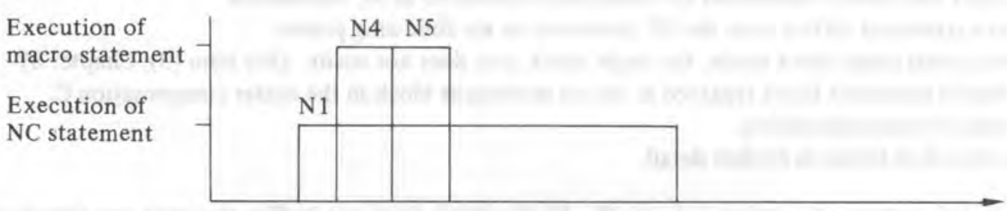
(2-1-1)

When the second NC statement is not the no-movement block,

The macro statement after the first NC statement next to the block currently executed is executed.

[Example 10.6.3.3]

N1X1000 ;	Block currently executed
N2#10=100 ;	Macro statement already executed
N3Y1000 ;	First NC statement
N4#1100 = 1 ;	Macro statement executed
N5#1 = 10 ;	Macro statement executed
N6X–1000 ;	Second NC statement

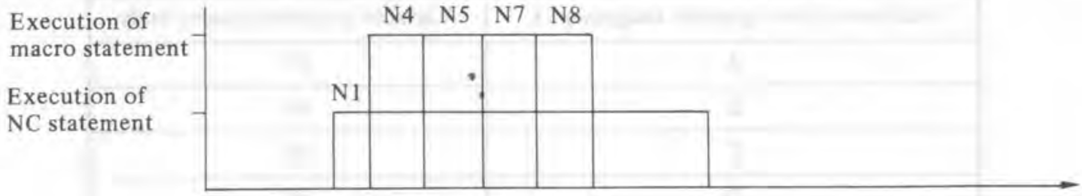


(2-1-2)

When the second NC statement after the block currently executed is the no-movement block, macro statements up to that next to the second NC statement (i.e. the no-movement block) after the block currently executed are executed.

[Example 10.6.3.4]

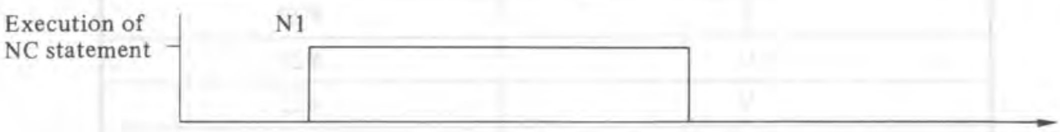
N1X1000 ;	Block currently executed
N2#10 = 100 ;	Macro statement already executed
N3Y1000 ;	First NC statement
N4#1100 = 1 ;	Macro statement executed
N5#1 = 10 ;	Macro statement executed
N6Z1000 ;	Second NC statement (no-movement block)
N7#1101 = 1 ;	Macro statement executed
N8#2 = 20 ;	Macro statement executed
N9X–1000 ;	Third NC statement



(2–2)
When the first NC statement after the block currently executed is the no-movement block, the macro statement is not executed.

[Example 10.6, 3.5]

N1Y1000 ;	Block currently executed
N2#1100 = 1 ;	Macro statement already executed
N3#1 = 10 ;	Macro statement already executed
N4Z1000 ;	First NC statement (no-movement block)
N5#1101 = 1 ;	Macro statement already executed
N6#2 = 20 ;	Macro statement already executed
N7X–1000 ;	Second NC statement



10.7 Macro Calls

A macro can be called simply from a single block, or modally from each block in the call mode.

10.7.1 Simple calls

When the following command is executed, the custom macro body identified by P (program number) is called.

G65 P (program number) L (iteration times) <argument assignment>;

When it is necessary to transfer arguments to a custom macro body, the argument is specified by <argument assignment>. The following two types of <argument assignment> can be specified. The argument mentioned here is the actual numerical value assigned to a variable.

(Note) G65 must be specified before arguments in the G65 block. Negative sign and decimal point can be used regardless of addresses in <argument assignment>.

(1) Argument assignment I

A____ B____ C____ D____ . . . . . Z____

An argument can be assigned for all address except G, L, N, O, and P. Assignment need not be made in alphabetical order. Specification is made according to word address format. Addresses not required may be omitted.

However, when I, J, and K are used, assignment must be made in the alphabetical order.

B____ A____ D____ . . . . . I____ K____ . . . . . valid

B____ A____ D____ . . . . . J____ I____ . . . . . invalid

Addresses assigned in argument assignment I and the number of the variable in custom macro body correspond as follows:

187

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

(2) Argument assignment II

A____ B____ C____ I____ J____ K____ I____ J____ K____ . . . .

In addition to the fact that arguments can be assigned in addresses A, B, and C, a maximum of ten sets of arguments can be set for addresses I, J and K. When several numbers are assigned in the same address, they must be assigned in the determined sequence.

Addresses not required can be omitted.

Addresses assigned in argument assignment II and the number of the variable used in the macro correspond as follows:

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

Suffixes 1 to 10 of I, J and K indicate the sequence of the assigned set.

(3) Coexistence of argument assignment I and II

No alarm is generated even if arguments of both assignment I and II are specified in the same block with a G65 command.

If an argument of type I and an argument of type II are specified to the same variable, the argument specified later is effective.

(Example)

G65A1.0B2.0I-3.0I4.0D5.0P1000;

<Variable>

#1: 1.0

#2: 2.0

#3:

#4: –3.0

#5:

#6:

#7: ~~4.0~~ → 5.0

In this example, even if arguments I4.0 and D5.0 are specified to variable #7, the latter is effective.

(Example 10. 7. 1) Reference point setting

Before commanding drill work pattern, a reference point of the pattern must be set.

X_0 X-coordinate value of pattern reference point
Y_0 Y-coordinate value of pattern reference point

Macro call command
G65 P9200 Xx Yy;

The following variables are used.

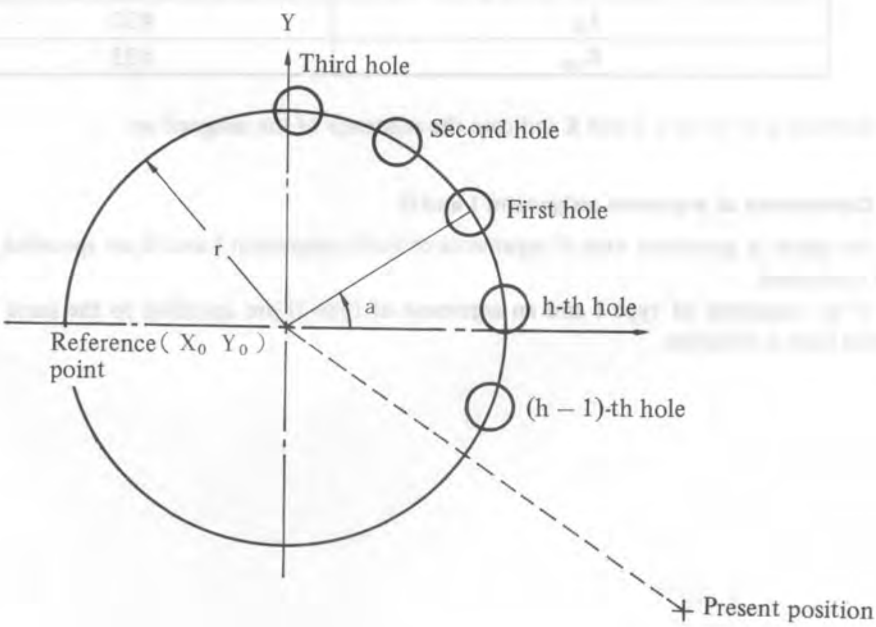
#100	Hole-number counter
#101	X-coordinate value of reference point used in macro of the pattern
#102	Y-coordinate value of reference point used in macro of the pattern
#24	X-coordinate value assigned by macro call command for reference point setting
#25	Y-coordinate value assigned by macro call command for reference point setting

The custom macro body must be created as follows.

```
O 9200;
#101=#24;        :   Reference point is informed to the macro.
#102=#25;
#100=0;          :   Hole-number counter is reset.
M99;
```

(Example 10. 7. 2) Bolt hole circle

With a point, set by the macro for reference point setting as the center of a circle, h holes equally spaced on the circle are drilled. The 1-st hole is on the line of angle a (see Fig.)



190

X_0 Y_0	Coordinate value of the bolt hole circle reference point
R	Radius
A	Start angle
H	Quantity

Macro call command: G65P9207RrAaHh ;

However, in case of h<0, a work is processed clockwise with the number being -h.

The following variables must be used:

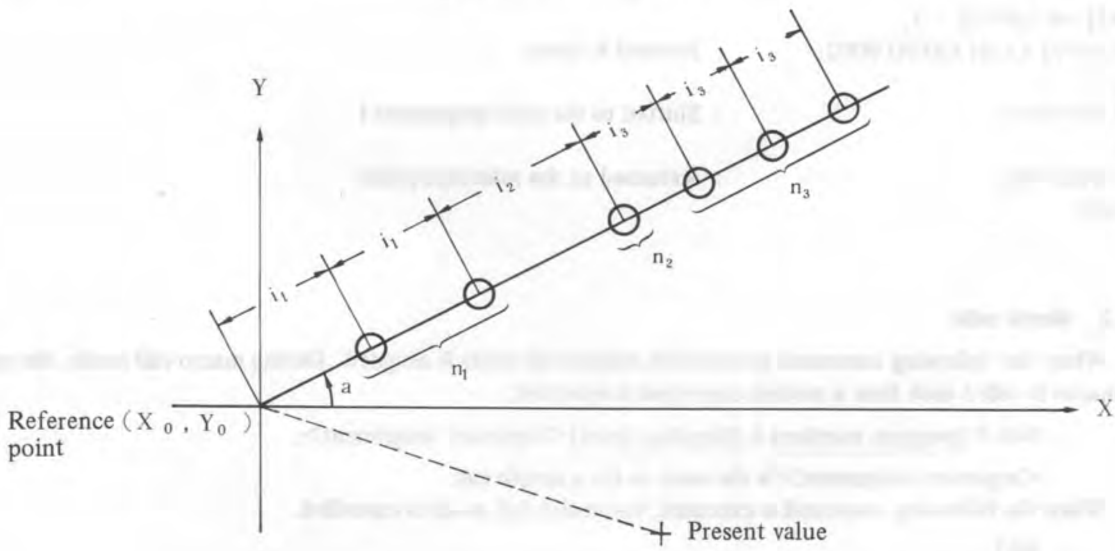
#100	Hole-number counter
#101	Coordinate value of reference point X
#102	Coordinate value of reference point Y
# 18	Radius r
# 1	Start angle a
# 11	Number of holes h
# 30	Storage of coordinate value of reference point X
# 31	Storage of coordinate value of reference point Y
# 32	Counter to show that i-th hole is under work
# 33	Degree of angle of i-th hole

The custom macro body is created as follows. (In case of absolute programming)

```
O 9207;
#30=#101;                              : Reference point stored
#31=#102;
#32=1;
WHILE [#32 LE ABS [#11] ] DO 1;        : Iterated by number of holes
#33=#1+360* [#32–1]/#11;
#101=#30+#18*COS [#33];                : Hole position
#102=#31+#18*SIN [#33];
X#101  Y  #102;
#100=#100+1;                           : 1 is added to the hole number-counter
#32=#32+1;
END 1;
#101=#30;                              : Return to the reference point
#102=#31;
M99;
```

(Example 10. 7. 3) Unequal spacing line at angle

With a point set by the macro of reference point setting as the reference point, holes arranged with an unequal spacing (i_1, i_2 . . . . .) in the direction of angle (a) with respect to X-axis are drilled.



X_0, Y_0	Reference point coordinate value
A	Degree of angle
I	Spacing of holes
K	Number of holes located with equal spacing (Must be assigned with a decimal point)

Macro call command

G65 P9203 A_a I_{i_1}, K_{n1}, I_{i2}, K_{n2} . . . . . . ;

However, in case of n=1, Kn can be defaulted.

The following variables are used

#100	Hole-number counter
#101	Coordinate value of reference point X
#102	Coordinate value of reference point Y
# 1	Angle a
# 4	First spaching i_1
# 6	Number of holes in the first group n_1
# 7	Second spacing i_2
# 9	Number of holes of the second group n_2
•	
•	
•	
# 2	Storage of coordinate value of reference point X
# 3	Storage of coordinate value of reference point Y
# 5	Counter for taking out holes spacing I_i
# 8	Distance from the reference point to the present hole

The macro is created as follows. (In case of absolute programming)

```
O 9203;
#2=#101;                              : Reference point stored
#2=#102;
#5=4;
#8=0;
WHILE [#5 LE31] DO 1;                 : Hole spacing assignment I is limited to ten
IF [# [#5] EQ0] GOTO 9001;            : Completed if assignment I is 0
DO 2;
#8=#8+#[#5] ;
#101=#2+#8*COS [#1] ;                 : Hole position
#102=#3+#8*SIN [#1] ;
X#101Y#102;
#100=#100+1;                          : +1 is added to hole-number-counter
# [#5+2] =# [#5+2] – 1;
IF [# [#5+2] LE 0] GOTO 9002;         : Iterated K-times
END2;
N9002 #5=#5+3;                        : Shifted to the next assignment I
END1;
N9001 #101=#2;                        : Returned to the reference point
#102=#3;
M99;
```

10.7.2 Modal calls

When the following command is executed, macro call mode is assigned. During macro call mode, the specified macro is called each time a motion command is executed.

G66 P (program number) L (iterating times) <argument assignment>;

<argument assignment> is the same as for a simple call.

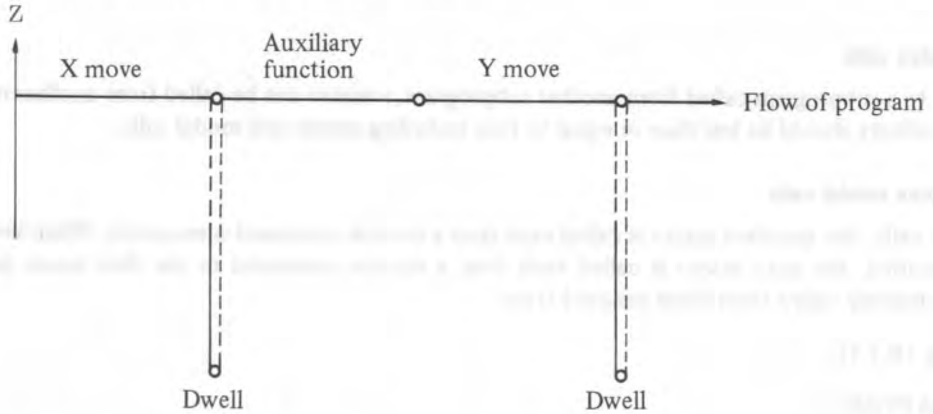
When the following command is executed, the macro call mode is cancelled.

G67 ;

(Note) In the G66 block, G66 must be specified before all arguments. Negative sign and decimal point can be used regardless of addresses in <argument assignment>.

(Example 10. 7. 4) Drilling Cycle

At each positioning point, drilling cycle is operated.



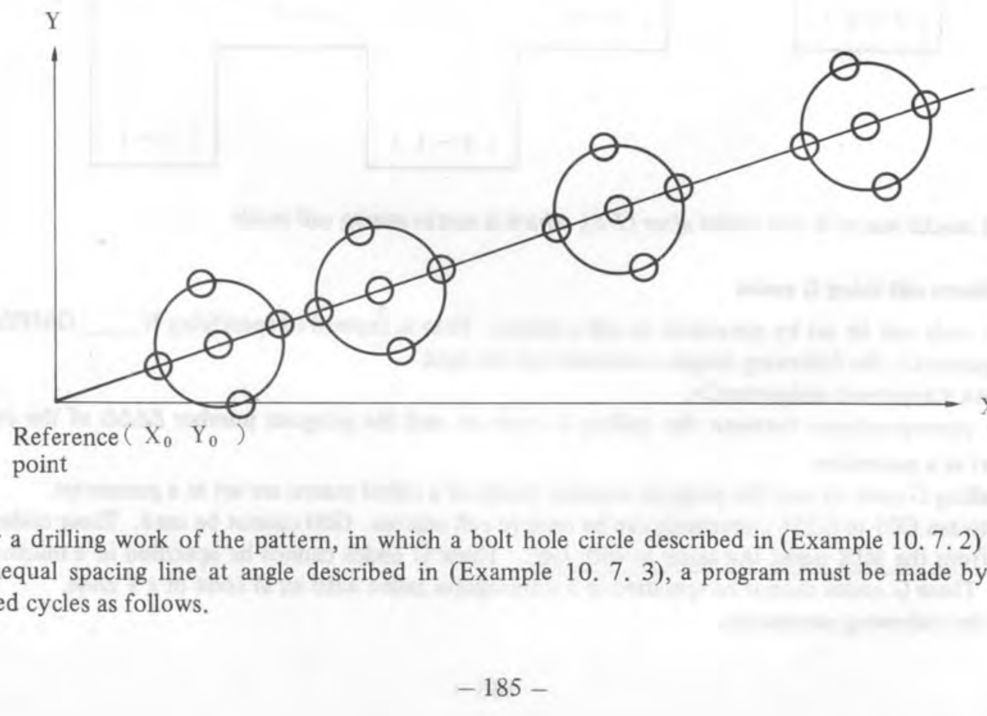
G66 P9082 R (R point) X (Z point) X (Dwell);

```
X     ;  )
M     ;  |
Y     ;  |  Drilling cycle is performed at each end of motion
  .      }  block in this region.
  .      |
  .      |
  .      |
G67;     )
```

Macro is as follows. (In case of incremental programing)

```
O 9082;
G00 Z #18;
G01 Z #26;
G04 X #24;
G00 Z– [ROUND [#18] + ROUND [#26] ] ;
M99;
```

(Example 10. 7. 5) Combined pattern



For a drilling work of the pattern, in which a bolt hole circle described in (Example 10. 7. 2) is overlapped on an unequal spacing line at angle described in (Example 10. 7. 3), a program must be made by using macro and canned cycles as follows.

G81 . . . . ;
G65 P9200X(Reference point coordinate value)Y(Reference point coordinate value);
G66 P9207R(Radius)A(Start angle)K(Quantity);
G65 P9203A(Degree of angle)I(Space)K(Quantity)I(Space);
G67;

10.7.3 Multiplex calls

Similarly to a subprogram called from another subprogram, a macro can be called from another macro. The multiplicity should be less than or equal to four including simple and modal calls.

10.7.4 Multiplex modal calls

In modal calls, the specified macro is called each time a motion command is executed. When several modal macros are specified, the next macro is called each time a motion command in the first macro is executed. Macros are successively called from those assigned later.

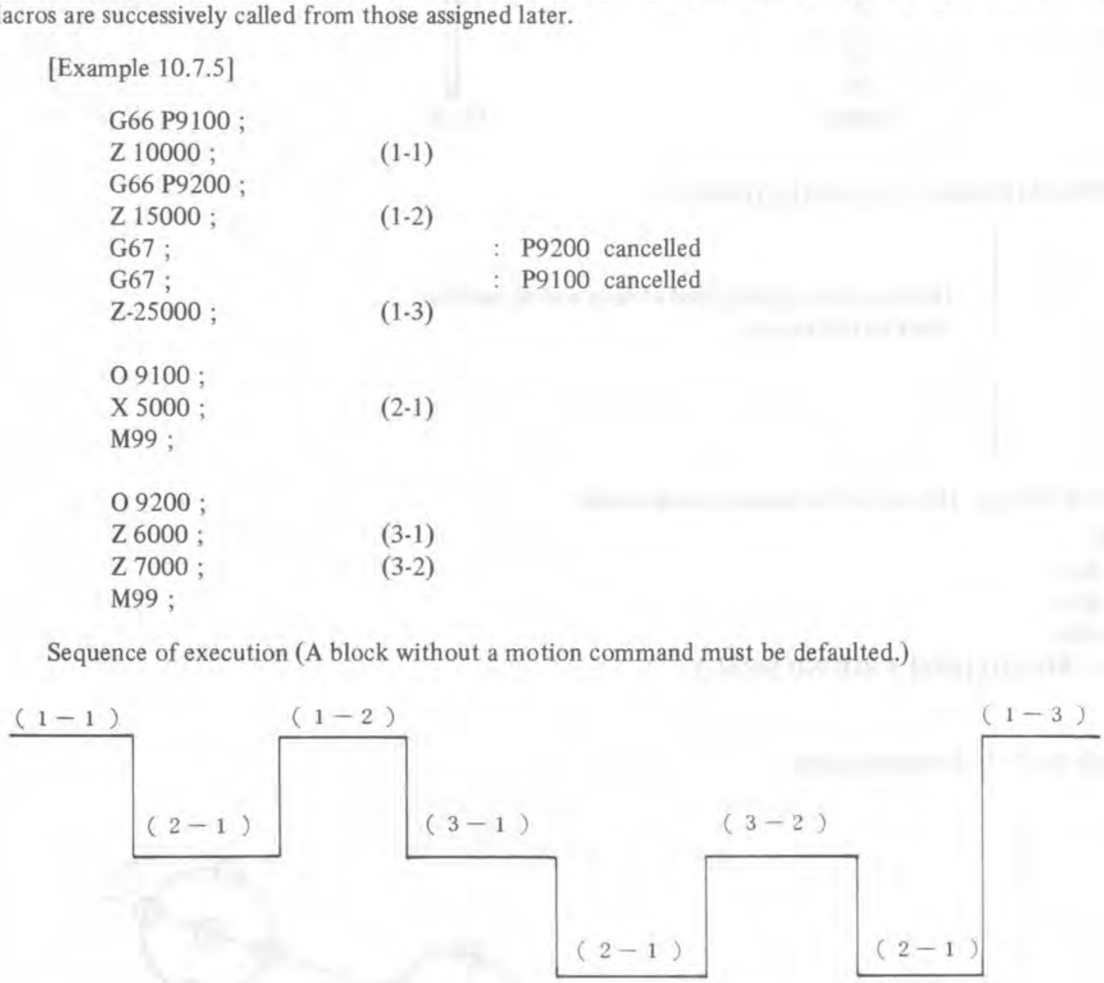
[Example 10.7.5]

```
G66 P9100 ;
Z 10000 ;          (1-1)
G66 P9200 ;
Z 15000 ;          (1-2)
G67 ;                          : P9200 cancelled
G67 ;                          : P9100 cancelled
Z-25000 ;          (1-3)

O 9100 ;
X 5000 ;           (2-1)
M99 ;

O 9200 ;
Z 6000 ;           (3-1)
Z 7000 ;           (3-2)
M99 ;
```

Sequence of execution (A block without a motion command must be defaulted.)



(Note) A modal macro is not called after (1-3), which is not in macro call mode.

10.7.5 Macro call using G codes

A G code can be set by parameter to call a macro. That is, instead of specifying N_____G65PΔΔΔΔ<argument assignment>; the following simple command can be used.
N_____Gxx <argument assignment>;
The correspondence between the calling G code xx and the program number ΔΔΔΔ of the called macro must be set as a parameter.

A calling G code xx and the program number ΔΔΔΔ of a called macro are set in a parameter.

Up to ten G01 to G255 commands can be used to call macros. G00 cannot be used. These codes cannot be specified from the MDI panel the same as with G65. These G codes cannot be specified in a macro called with a G code. These G codes cannot be specified in a subprogram called with an M code or a T code.

Set the following parameters.

194

0	3	2	3	G code used to call macro : 9010
0	3	2	4	G code used to call macro : 9011
⋮				⋮
0	3	3	2	G code used to call macro : 9019

(Example 10. 7. 6) CW circle cutting by G12

G12 I radius D offset number;

(1) Set the following parameter

G code which calls custom macro body : 9010 = 12

(2) Register the following custom macro body

```
O9010;
#1–ABS [#4] – # [2000+#7] ;
IF [#1 LE 0] GOTO 1;
#2=#1/2;
#3003=1;
G01 X[#1–ROUND [#2] ] Y #2;
G17 G02 X #2 Y–#2 R–#2;
I–#1;
X–#2Y–#2 R#2;
G01 X [#1–ROUND [ #2] ] Y#2;
#3003=0;
N1 M99;
```

10.7.6 Subprogram call with M code

An M code can be set by parameter to call a subprogram. That is, instead of N____ G____ X____ Y____ . . . . . M98P ΔΔΔΔ ;

the following simple command can be specified.

N____ G____ X____ Y____ . . . . . Mxx ;

As for M98, the subprogram is displayed on the COMND page, but MF and M codes are not transmitted.

The correspondence between the calling M code xx and the program number ΔΔΔΔ of the called subprogram must be set as a parameter.

Up to three M03 to M97 commands can be used for a macro call except for M30, MBUF1 (parameter number 35) and MBUF2 (parameter number 36).

This command can be specified from MDI keyboard, but no arguments can be entered. When these M codes are specified in a macro called with a G code or in a subprogram called with an M code or a T code, the subprogram is not called, but these M codes are treated as ordinary M codes.

Set the following parameters.

0	3	2	0	M code used to call subprogram : 9001
0	3	2	1	M code used to call subprogram : 9002
0	3	2	2	M code used to call subprogram : 9003

(Example 10.7.7) ATC canned cycle by M06

(1) Set the following parameter

M code which calls subprogram: 9001 = 06

(2) Register the following custom macro body

```
O9001;
#1=#4001;
#3=#4003;
G28G91Z0 M20;
G28 Y0;
M21;
G00 Z 10000;
M22;
G28 Z0;
M23;
G#1 G#3 M99;
```

10.7.7 Macro call using M codes

An M code can be set by parameter to call a macro. That is, instead of specifying N___G65P△△△△<argument assignment >; the following simple command can be used.

N___ Mxx <argument assignment>;

The correspondence between the calling M code xx and the program number △△△△ of the called macro must be set as a parameter.

Up to ten M06 to M255 commands can be used to call macros, except for existing M codes. These codes cannot be specified from the MDI panel the same as with G65. These M codes cannot be specified in a macro or a subprogram called with G, M or T codes.

Set the following parameters.

0	0	4	3	M code used to call macro : 9020
≀				≀
0	0	5	2	M code used to call macro : 9029

10.7.8 Subprogram call with T codes

A T code can be set by parameter to call a subprogram.

N_____G_____X_____Z_____.....T t ; is equivalent to the following two blocks.

#149=t;

N_____G_____X_____Z_____.....M98 P9000 ;

T code t is stored as an argument in common variable #149. The T code is displayed on the COMND page, but TF and T codes are not transmitted.

This command can be specified from MDI keyboard. But it cannot be specified in the same block which includes an M code to call a subprogram.

When these T codes are specified in a macro called with a G code or in a subprogram called with an M code or a T code, the subprogram is not called, but these T codes are treated as ordinary T codes.

Set the following parameter.

0	3	0	6							TMCR	

196

10.7.9 Location of decimal point in an argument

Arguments are generally specified with a decimal point. If the decimal point is not specified, the location of the decimal point is assumed as follows:

Address	mm input	inch input
A, C	3 (2) ⟨4⟩	3
B (without B 3-digit option)	3 (2) ⟨4⟩	3
B (with B 3-digit option)	0	0
D, H	0	0
E, F (in G94 mode)	0 (1) ⟨2⟩	2 ⟨3⟩
E, F (in G95 mode)	2 (3) ⟨4⟩	4 ⟨5⟩
I, J, K	3 (2) ⟨4⟩	4 ⟨5⟩
M, S, T	0	0
Q, R	3 (2) ⟨4⟩	4 ⟨5⟩
U, V, W	3 (2) ⟨4⟩	4 ⟨5⟩
X, Y, Z	3 (2) ⟨4⟩	4 ⟨5⟩

The values in the table above are the location of the decimal point counted from the least significant digit.

The values in parentheses are the number of digits to the right of the decimal point when parameter FMIC = 1 for addresses E and F; when parameter MIC = 1 for the other addresses.

The values in bracket are the number of digits to the right of the decimal point when parameter DIC = 1.

10.7.10 Difference between M98 (subprogram call) and G65 (custom macro body call)

(1) G65 can include arguments; M98 cannot.

(2) M98 is used to branch to a subprogram after executing a command other than M, P or L in the block; G65 is used to branch only.

(3) When a M98 block includes an address other than O, N, P and L, execution of the block stops as single block stop, a G65 block does not.

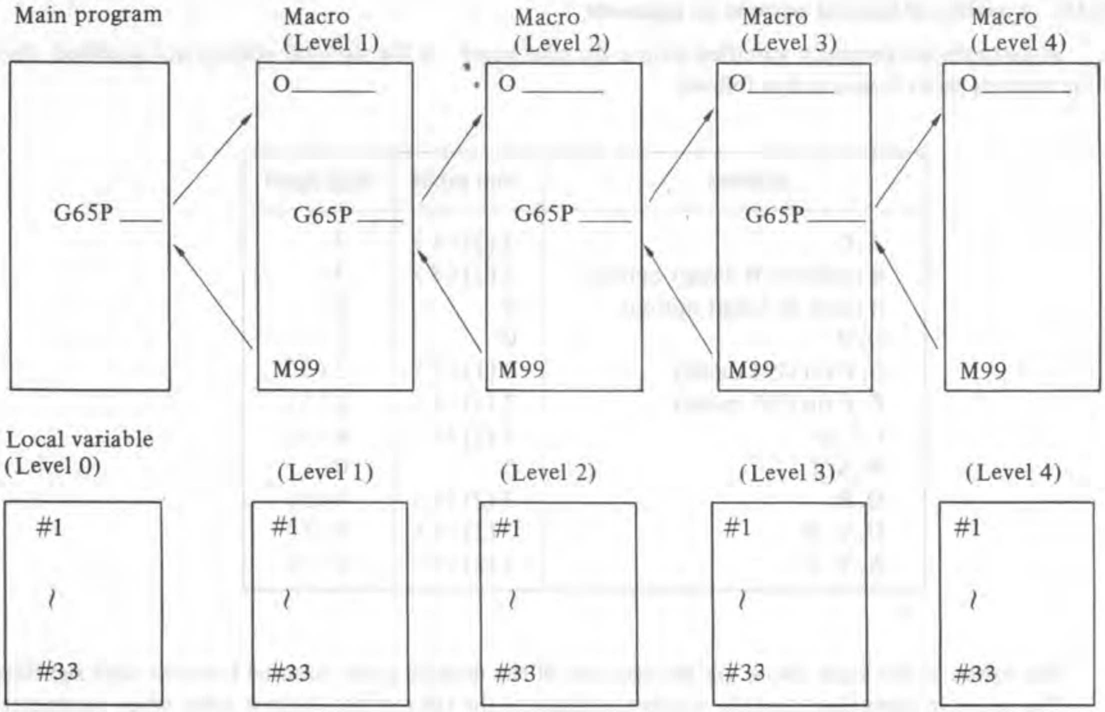
(4) G65 changes the level of local variable; M98 does not. That is, #1 specified before G65 is one thing and #1 in the calling custom macro body is another. #1 specified before M98 is the same as #1 in the calling subprogram.

(5) Up to four G65 calls, including G66, can be made in addition, M98 calls can be made (with custom macro A or B option) up to four calls.

(6) When inserting an operation from MDI during automatic operation, up to four M98 calls can be made in TAPE mode or in MEMORY mode, and separately up to four calls in MDI mode; up to four G65 calls can be made in all modes in common.

10.7.11 Custom macro level and local variable

When a macro is called with G65, G66 or a G code or an M code which calls a macro, the level of the macro increases by one. As a result, the level of the local variable also increases by one. Namely, the relationship between the macro call and local variable is as follows.



(1) The main program is provided with #1 to #33 local variables (level 0).

(2) When the macro (level 1) is called with G65, etc., the local variable (level 0) of the main program is stored, and #1 to #33 local variables (level 1) for the macro (level 1) are prepared. Argument transfer is possible for this purpose. (This also applies to (3) below.)

(3) The local variables (level 1, 2, 3) are stored each time the macros (level 2, 3, 4) are called, and new local variables (level 2, 3, 4) are prepared.

(4) When the operation returns from each macro with M99, the local variables (level 0, 1, 2, 3) stored in (2) and (3) are set in the same conditions as when they were stored.

10.8 Relation to Other Functions

(1) MDI operation

The macro call command, arithmetic command and control command can not be specified in MDI mode.

MDI commands (except those relating to the macro) can be entered midway during execution of the macro even with a single block stop.

A macro is not called even in macro call mode (G66) if a move command is input from the MDI.

(2) Sequence number search

Sequence numbers in the custom macro body cannot be searched for.

(3) Single block

A block other than one including a macro call command, arithmetic command or control command can be executed as a single block stop even when in the macro.

A block including a macro call command (G65, G66, G67), arithmetic command and control command is not stopped even by single block operation.

However, the following settings or parameter setting executes a single block stop for commands other than macro call commands. This function is used for testing a custom macro body.

0	3	1	8

	MCS9						

0	3	1	9

	MCS8						MCS7

When MCS7 = 1, a macro statement in programs numbered O1 to 7999 and O9900 to 9999 executes a single block stop.

When MCS8 = 1, a macro statement in programs numbered O8000 to 8999 executes a single block stop.
When MCS9 = 1, a macro statement in programs numbered O9000 to 9089 executes a single block stop.

But a block in which a macro statement has executed a single block stop, is assumed to have no movement in the cutter compensation mode. Sometimes incorrect compensation is performed. (Strictly, the case is handled the same as for a movement amount of 0 although a move has been specified.)

(4) Optional block skip

/, appearing in the middle of <expression> (right-hand-side of an arithmetic formula, or in [ ]), is treated as division, and not an optional block skip.

(5) Operation in EDIT mode

The following settings can be used to protect registered custom macro bodies and subprograms from accidental destruction.

0	3	1	8

PRG9							

0	3	1	9

PRG8							

(Setting operation is effective)

Setting PRG8 and PRG9 to 1 specifies that custom macro bodies or subprograms with program numbers 8000 to 8999 and 9000 to 9899 cannot be registered, cancelled, and edited, respectively.

However, this does not affect the deletion of all programs by turning on the power or the punching of a program.

(6) PRGRM page display in other than EDIT mode

Generally, when a custom macro body or a subprogram is called, the called program is displayed. The following setting is used to keep displaying the previous program.

0	3	1	8

		MPD9					

0	3	1	9

		MPD8					

Setting MPD8 and MPD9 to 1 specifies that custom macro bodies or subprograms with program numbers 8000 to 8999 and 9000 to 9899 cannot be displayed on the PROGRAM page in other than EDIT mode, respectively.

(7) Reset

When a cleared status is produced by a reset, all local variables and common variables #100 to #149 become <vacant>.

Performing a reset in other than MDI mode clears the called status and the DO status of a custom macro body or of a subprogram, returning control to the main program. Performing a reset in MDI mode clears only the program called in MDI mode.

(8) Macro and NC statements

The following blocks are called macro statements.

① Operation command (block including =)
② Control command (block including GOTO, DO or END)
③ Macro call command (block including G65, G66, G67 or G codes calling macro)

Blocks other than macro statements are sometimes referred to as NC statements.

(9) MDI operation during automatic operation

If a macro is called while inserting an operation from MDI during automatic operation the degrees of multi-macro calling and of multiple DO statements are counted as continuing from the automatic operation, the former and the latter being up to four degrees and up to three degrees in total respectively. Up to four subprogram calls can be made in MDI mode.

(10) Display in PROGRAM RESTART page

The M and T codes used for subprogram calling are not displayed like M98.

(11) Feed hold

Macro statement execution stops by turning on feed hold. (It also stops by resetting or alarm generation).

10.9 Special Codes and Words Used in Custom Macro

The following code can be used in the program of the custom macro in addition to codes used in conventional programs.

(1) ISO

Meaning	8	7	6	5	4		3	2	1	Character
[	O	O		O	O	°		O	O	[
]	O	O		O	O	°	O		O	]
#	O		O			°		O	O	#
*	O		O		O	°		O		*
=	O		O	O	O	°	O		O	=
O	O	O			O	°	O	O	O	O
+			O		O	°		O	O	+

(2) EIA

Meaning	8	7	6	5	4		3	2	1	Character
[				O	O	°	O			
]		O			O	°	O			
#			Parameter			°				
*					O	°	O	O		&
=			O	O	O			O	O	,
+		O	O	O						+

O, the same code O as in the program number must be used. The hole pattern for # in EIA code must be set as a parameter. However, the character with no punched hole cannot be used. Note that alphabetic characters can be used, but when used as #, they are not used in their proper sense.

Parameter number

0	3	1	7

Sprocket hole

Special words used in custom macro A are:

OR, XOR, IF, GOTO, EQ, NE, GT, LT, GE, and LE.

Special words used in custom macro B, in addition to those above, are:

AND, SIN, COS, TAN, ATAN, SQRT, ABS, BIN, BCD, ROUND, FIX, FUP, WHILE, DO, and END.

10.10 Limitations

(1) Usable variables

#0, #1 to #33, #100 to #149, #500 to #509, system variables.

(2) Valid variable values

Maximum value $\pm 10^{47}$

Minimum value $\pm 10^{-29}$

(3) Constant value valid in <Expression>

Maximum value ±99999999

Minimum value ±0.0000001

Decimal point can be used.

(4) Arithmetic precision

Eight-digit decimal number.

(5) Macro nesting

Maximum of four.

(6) Iteration identification number
1 to 3

(7) Nesting of [ ]
Maximum of five.

(8) Subprogram nesting
Maximum of four.

(9) Functions available from custom macro A
Custom macro B can perform everything described above, but custom macro A can perform only the following:

(i) Can use variables other than system variables.
(ii) Can operate using the following variables: +, –, OR, XOR
(iii) Can use IF [<conditional expression>] GOTO n.
(iv) Can use simple and modal calls.

10.11 P/S Alarm Explanation

The following alarms supplement the alarms listed in Appendix 7.

① Alarm number 004
An appropriate address was not found at the proper position.

Example

X1*1

X1, a word, should be followed by an address but was followed by *, generating alarm 004.

② Alarm number 114
A format other than <Formula> is invalid. This alarm occurs under one of the following conditions:

(a) The character following an address is not a numeral, ., –, #, [, and +.

Example: XF1000 ;
XSIN[10] ;

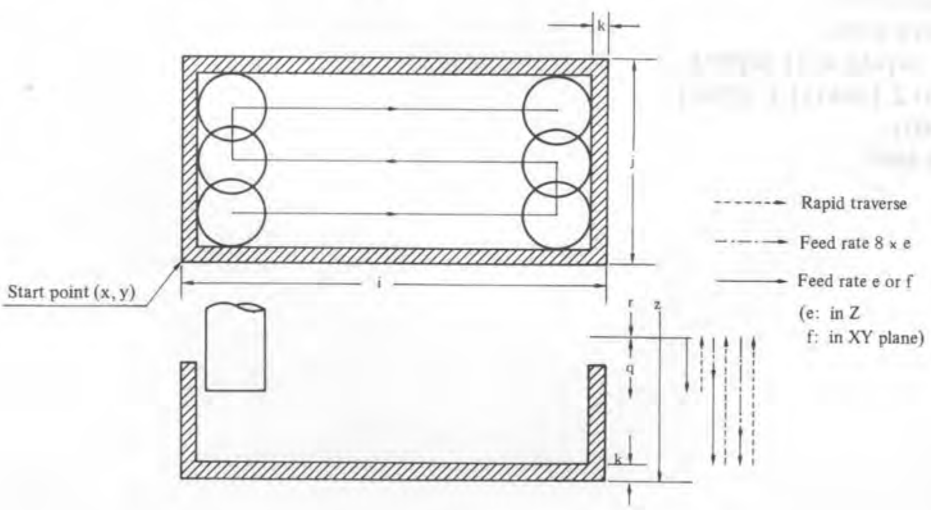
(b) The form is not: IF (or WHILE) [<formula> ΔΔ <formula>]

Example: [IF #1 EQ #2 GOTO 10 ;
WHILE [#1 SIN #2] DO 1 ;

10.12 Examples of Custom Macro

10.12.1 Pocket Machining

In this example a custom macro performs a pocket milling cycle machining an area as in the figure below. Area machining is performed at a certain depth in Z and further with a deeper cut-in.



201

(1) Custom macro call statement

G65 P9802 X x Y y Z z R r Q q I i J j K k T t D d F f E e;

where x, y:	absolute coordinate values of start point (botton left of pocket) in X, Y
z, r:	absolute values of points Z and R (see the figure)
q:	one-cut depth (positive number)
i, j:	length of area in X and Y (positive number) (see the figure) (more efficient machining when i > j)
k:	finishing allowance
t:	take cut width so that machining width may not exceed cutter diameter x t %.
d:	cutter compensation number (01- 99)
f:	feed rate in XY plane
e:	feed rate at cut-in time. Feed is made at 8 x e up to 1 mm before cut-in.

(2) Custom macro body

```
O 9802 ;
#27=# [2000+#7] ;
#28=#6+#27 ;
#29=#5–2*#28 ;
#30=2*#27*#23/100 ;
#31=FUP [#29/#30] ;
#32=#29/#31 ;
#10=#24+#28 ;
#11=#25+#28 ;
#12=#24+#4–#28 ;
#13=#26+#6 ;
G00X#10Y#11 ;
Z#18 ;
#14=#18 ;
DO1 ;
#14=#14–#17 ;
IF [#14GE#13] GOTO1 ;
#14=#13 ;
N1 G01 Z#14 F#8 ;
X#12 F#9 ;
#15=1 ;
WHILE [#15 LE #31] DO2 ;
Y [#11 + #15*#32] ;
IF [#15 AND 1 EQ0] GOTO2 ;
X#10 ;
GOTO3 ;
N2X#12 ;
N3 #15=#15+1 ;
END2 ;
G00 Z#18 ;
X#10 Y#11 ;
IF [#14LE#13] GOTO4 ;
G01 Z [#14+1] F [8*#8] ;
END1 ;
N4 M99 ;
```

10.13 External Output Commands

The following macro commands can be executed in addition to the standard custom macro commands. (These commands are called external output commands.)

(a) BPRNT
(b) DPRNT
(c) POPEN
(d) PCLOS

These commands are provided to output variable values and characters through the RS232C interface. Specify these commands according to the following procedure.

(1) Open command: POPEN

The connection processing to external I/O units is made before executing a series of data output commands.

(2) Data output command: BPRNT or DPRNT

Necessary data output commands are executed.

(3) Close command: PCLOS

This command is specified after all data output commands have been completed to disconnect external I/O units.

10.13.1 Open command POPEN

POPEN:

This command is provided to connect to external I/O units, and it is specified before sending a series of data output commands. The DC2 control code is output from NC.

10.13.2 Data output commands BPRNT, DPRNT

(1) BPRNT

```
[ a  #b  [ c ] . . . . . . . . . . ]
  |   |    |
  |   |    └──────────── Number of effective digits below decimal point
  |   └──────────── Variable
  └──────────── Character
```

Characters are output and variable values are binary-output when BPRNT is given.

(a) The specified characters are output by ISO codes.
The following characters are commandable.
* Alphabetic characters (A ~ Z)
* Numeric characters
* Special characters (* , / , + , –)
Asterisk (*) is output by the space code.

(b) Since all variables are being stored with a decimal point, the number of effective digits below decimal point must be specified by parenthesizing it just after specifying variables.
Variable values are treated as 2-word (32-bit) data by taking the number of digits below decimal point into account, and they are output as binary data, starting with high-order bytes.

(c) EOB code is output by ISO code after command data output.

(d) No "vacant" variable is outputtable (otherwise P/S alarm No. 114 occurs).

(2) DPRNT

```
[ a  #b  [ c d ] . . . . . . . . . . ]
  |   |    | |
  |   |    | └──────────── Number of digits below decimal point
  |   |    └──────────── Number of digits above decimal point
  |   └──────────── Variable
  └──────────── Character
```

When DPRNT is specified, characters and numeric characters every variable value digit are output by ISO codes.

(a) Same as in (a), (c), (d) of BPRNT command

(b) For outputting variable values, specify the variable number following #, and then, specify the number of digits above decimal point and the number of digits below decimal point by parenthesizing these values.

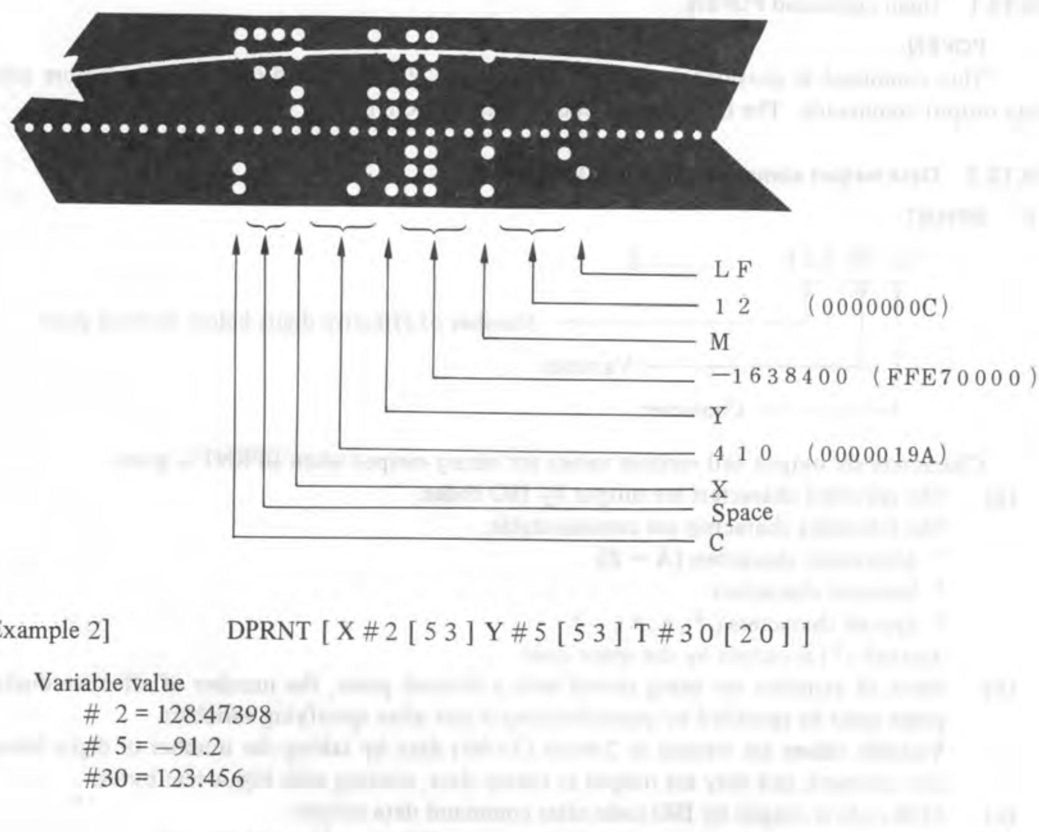
Variable values are output by the specified number of digits every digit starting with higher significant digit by ISO codes. The decimal point is also output by ISO code.

A variable value is regarded as being composed of max. 8-digit numeric value. If the higher significant digits are 0, no code is output when parameter No. 315 PRT=1 and the space code is output when PRT=0.

Whenever the number of digits below decimal point is other than 0, the numeric value below decimal point is output. When the number of digits below decimal point is 0, the decimal point is not output. For the plus (+) code in case of positive sign, the space code is output when parameter No. 315 PRT=0, but no code is output when PRT=1.

[Example 1]　　　BPRNT [C * * X # 100 [3] Y # 1 0 1 [3] M # 1 0 [0]]

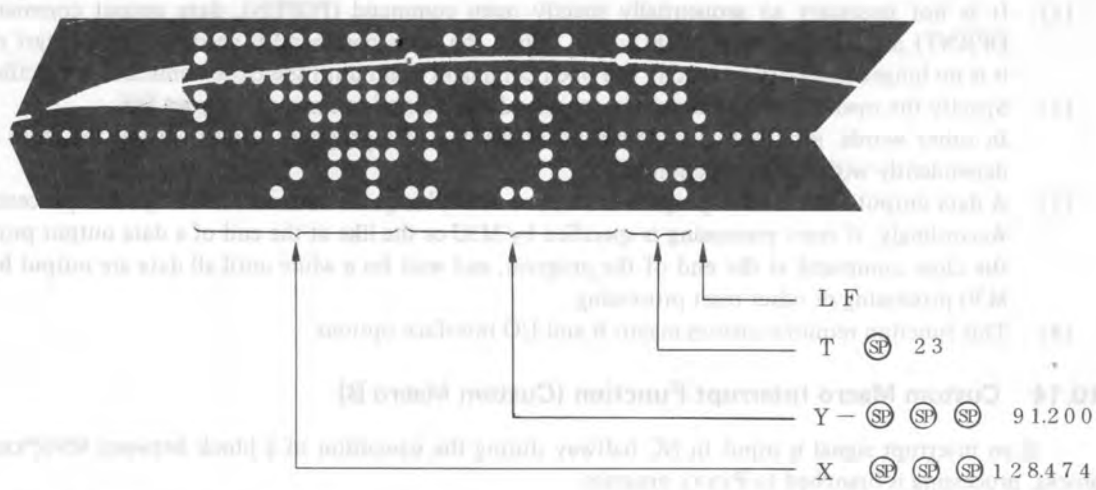
Variable value
#100 = 0.40956
#101 = –1638.4
#10 = 12.34



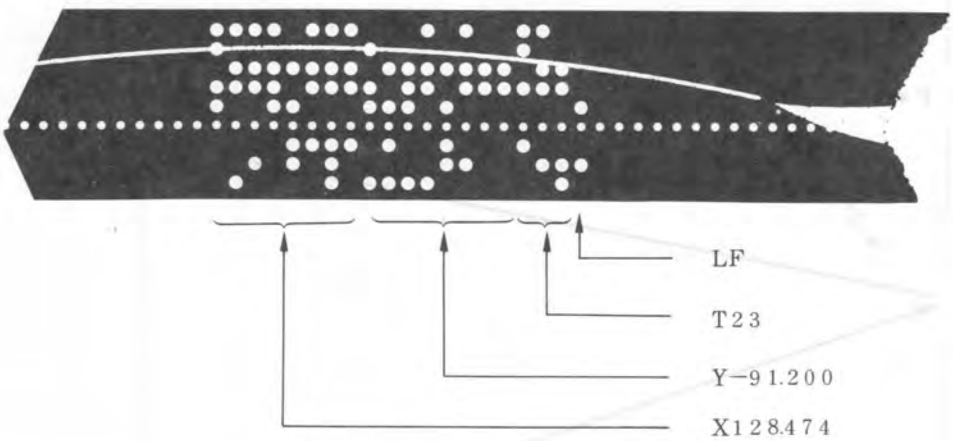
[Example 2]　　　DPRNT [X # 2 [5 3] Y # 5 [5 3] T # 3 0 [2 0]]

Variable value
2 = 128.47398
5 = –91.2
#30 = 123.456

① Parameter PRT = 0



② Parameter PRT = 1



10.13.3 Close command PCLOS

PCLOS:

This command is specified when all data output commands were completed to release the processing connection to external I/O units. DC4 control code is output from NC.

10.13.4 Setting required for using this function

(1) Set parameter No. 341 so that the output unit RS232C is employed for punch-out. However, the output to FANUC cassette is not allowable in this case.

(2) Set various data of RS232C (baud rate, etc.) to one of parameter No. 310 to 313 according to the number of the output units preset to the above parameter No. 341.

(3) Set output codes to ISO codes.

(4) Set parameter No. 315 to determine whether leading zeroes are spaced or not when data are output by DPRNT command.

0	3	1	5

PRT							
7	6	5	4	3	2	1	0

Leading zeroes are treated by PRT DPRNT command as follows during data output.

0 : Spaced 1 : No output

10.13.5 Cautions

(1) It is not necessary to sequentially specify open command (POPEN), data output commands (BPRNT, DPRNT) and close command (PCLOS). When the open command is specified at the start of a program, it is no longer necessary to specify the open command again until the close command is specified next.

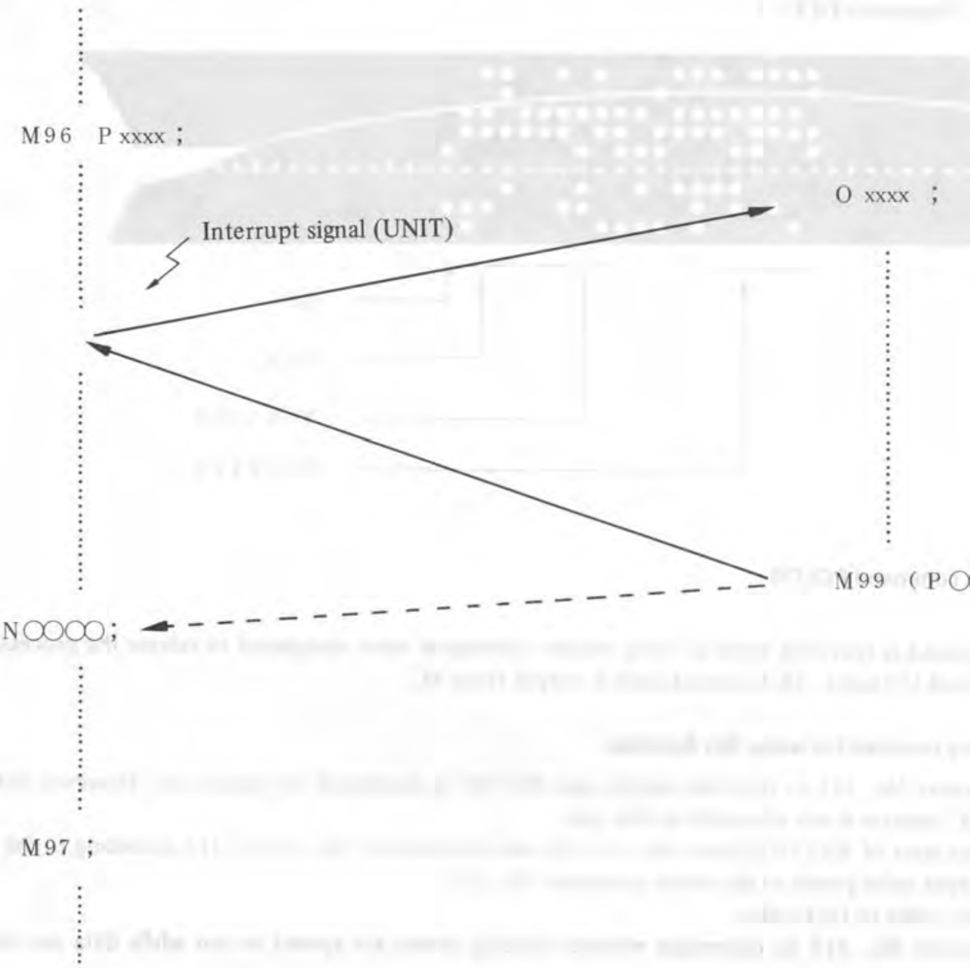
(2) Specify the open command and close command as a pair of commands without fail.
In other words, send the close command at the end of a program. Don't specify the close command independently without open command.

(3) A data output command in progress is stopped and subsequent data are erased by reset processing.
Accordingly, if reset processing is specified by M30 or the like at the end of a data output program, specify the close command at the end of the program, and wait for a while until all data are output before starting M30 processing or other reset processing.

(4) This function requires custom macro B and I/O interface options.

10.14 Custom Macro Interrupt Function (Custom Macro B)

If an interrupt signal is input to NC halfway during the execution of a block between M96Pxxxx; and M97; blocks, processing is branched to Pxxxx program.



Specify M99; for returning from custom macro interrupt to the original program. The sequence number in the program to be returned can also be specified by address P.

(Note 1) For details of custom macro function, see appendix 12.

(Note 2) For using this function, refer to the operator's manual published by the machine tool builder, without fail.

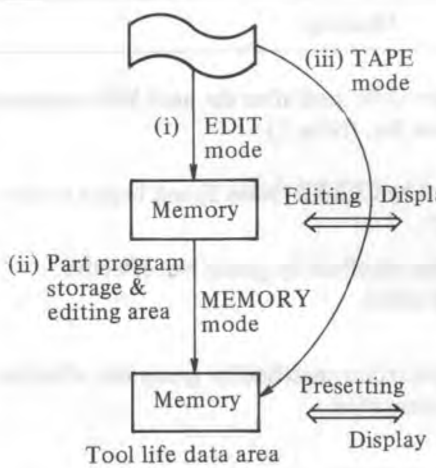
11. TOOL LIFE MANAGEMENT

Tools are classified into various groups, with the tool life for each group (time or frequency of use) being specified. The function of accumulating the tool life of each group in use and selecting and using the next tool previously sequenced in the same group, is called the tool life management function.

11.1 Setting The Tool Groups

Tools used sequentially in each group and their respective tool life are previously set by the following format tape in the NC equipment.

Tape format	Meaning
0□□□□	Program number
G10L3 ;	Start of setting tool group
P□□□L△△△△	P followed by group No. (1 ~ 128) L followed by tool life (1 ~ 9999) (Note 1)
T△△△△H○○D□□	(1)
T△△△△H○○D□□	(2)
⋮	T followed by tool No. H followed by tool length offset No. D followed by cutter compensation No. Tools are selected in the sequence of (1) through (2) to (N).
T△△△△H○○D□□ ;	(N)
P□□□L△△△△ ; T△△△△H○○D□□ ; ⋮	Data for the next group
G 1 1 ;	End of setting tool groups
M02 (M30) ;	End of program



The setting operation is as follows.

(i) Insert the above tape into the tape reader and press the READ button in the EDIT mode, as with an ordinary NC tape.
The program will be registered in the part program memory, and made ready for display and editing.

(ii) Perform a cycle start operation in the MEMORY mode to run the program. The data will be stored in the tool life data area of the memory, and at the same time, the already existing tool life data of all groups will be canceled and the life counters also cleared. Data stored once is not erased by turning the power off.

(iii) Executing a cycle start operation in the TAPE mode instead of the operation of (i), stores the program contents directly from the tape onto the tool life data area. In this case, however, display and editing cannot be done as in (i).

(Note 1) Whether tool life is to be indicated by time (minutes) or by frequency, is set by the parameter (309-LCTM).

	Number of groups	Number of tools
①	16	16
②	32	8
③	64	4
④	128	2

(Note 2) *The number of groups and the number of tools per group that can be registered are combined by setting in one of four ways as follows (309-GST1, GST2).*
In any combination, up to 256 tools can be registered. Combination ① is selected for up to 16 groups, each with up to 16 tools; combination ② for up to 32 groups, each with up to 8 tools; and the like. To change a combination, change the parameter, then turn the power off and on.

(Note 3) *Codes H and D, when in disuse, can be omitted.*
(Note 4) *The same tool No. can appear at any frequency and at any place in the set data.*

The following is a concrete example of the tape format.

```
O0001 ;
G10L3 ;
P001L0150 ;          ┐
T0011H02D13 ;        │
T0132H05D08 ;        │ Data of group 1
T0068H14D16 ;        ┘
P002L1400 ;          ┐
T0061H15D07 ;        │
T0241H25D04 ;        │ Data of group 2
T0134H17D03 ;        │
T0074H08D21 ;        ┘
P003L0700 ;          ┐
T0012H14D08 ;        │ Data of group 3
T0202H22D02 ;        ┘
G11 ;
M02 ;
```

(Note 5) *Group Nos. specified by P need not be consecutive. All registerable groups need not be set, either.*

11.2 Specifying Tool Groups, etc. in a Machining Program

In a machining program, tool groups, etc. are specified by using the T codes as follows.

Tape format	Meaning
- - - - -	
T▽▽▽▽	Group No. of tools which are to be used after the next M06 command + tool life management ignore No. (Note 1)
- - - - -	
M06T□□□□ ;	Terminates the tool specified by □□□□ (Note 2) and begins to use the tools specified by ▽▽▽▽ .
H□□ ;	99 : Makes the tool offset specified by group No. effective. 00 : Cancels tool length offset.
- - - - -	
D○○	99 : Makes cutter compensation specified by group No. effective. 00 : Cancels cutter compensation.
- - - - -	
T△△△△ ;	Uses a tool specified by △△△△ after the next M06 command.
- - - - -	
M06T▽▽▽▽;	Terminates the tools of ▽▽▽▽ and begins to use the tool of △△△△.
- - - - -	
M02 (M30) ;	End of the machining program.

(Note 1) *From T000 to T△△△△, stipulated by the tool life management ignore No. △△△△, are handled as ordinary T code commands, and no tool life management is performed. When the T code of △△△△ plus the group No. is designated, tool life management is executed for related group. The value of the tool life management ignore No. is set by a parameter.*

When the value is 100, for example, from T0000 to T0100 are output as ordinary T codes, and when the T0101 is designated, a T code of a tool which has not reached its life end among the tools of group 1 is output.

(Note 2) *The above tape format uses the tool return No. command method, which requires a tool return command at the time of tool change. In the non-tool return No. command method, which requires no tool return command, omit the T code following M06. In this case also, the same tool change as the above is performed.*

The following is a concrete example of the tape format when the tool life management ignore No. is 100 in the tool return No. command method.

Tape format	Meaning
- - - - -	
T0101 ;	Uses tools of group 1 after the next M06 command.
- - - - -	
M06T0003 ;	Terminates the tool of 0003 used so far, and begins to use tools of group 1.
- - - - -	
G43H99 ;	Uses the tool length offset No. specified under group 1.
- - - - -	
G41D99 ;	Uses the cutter compensation No. specified under group 1.
- - - - -	
D00 ;	Cancels the cutter compensation.
- - - - -	
H00 ;	Cancels the tool length offset.
- - - - -	
T0005 ;	Uses the tool of 0005 after the next M06 command.
- - - - -	
M06T0101 ;	Terminates the tools of group 1, and begins to use the tool of 0005.

11.3 Execution of Tool Life Management

11.3.1 Counting tool life

(1) When tool life is specified by time (minutes)

In this case, T△△△△ (△△△△ = tool life management ignore No. + tool group No.) and successively M06 are commanded, and M06 is commanded again in the machining program. This time, during which tools are actually used in the cutting mode, is counted at certain intervals (four seconds). The time for single block stop, feed hold, rapid traverse, dwell, etc., is disregarded. The maximum settable life value is 4300 minutes.

(2) When tool life is specified by frequency

The counter for groups of tools that were used is increased by one, every process from the time that a cycle start operation is performed until the time that M02 or M30 is commanded and the NC is reset. Even if the command for the same group is given a number of times in one process, the counter increase stays at one. The maximum life value is 9999. Life is counted for each group, and the contents of the counter are not erased by turning off the power.

(Note) *When having executed M02 or M30 with life specified by the frequency of use, input the External Reset (ERS) or Reset & Rewind (RRW) signal to the NC.*

11.3.2 Tool change signal and tool change reset signal

Tools are selected one after another in a predetermined sequence upon the end of each life. When the last tool has come to its life end in a tool group, the Tool Change signal is output. Changing that tool, which is displayed on the CRT screen, then specifying the group No. concerned and inputting the Tool Change Reset signal or operating the MDI panel (see 11.4.3), clears all data of the group, such as the life counter, *, @, etc. (see 11.4.2). Effecting tool change reset for all groups at life end releases the Tool Change signal automatically. Specifying this group after resumption of machining begins selection again with the first tool.

(Note) With life specified by time, the Tool Change signal is output even during machining when the life end is reached. Machining continues until the end of the program. With life specified by frequency, the Tool Change signal is output when M02 or M30 resets the NC after life end has been reached.

11.3.3 New tool select signal

When a new tool is about to be selected in a group, the New Tool Select signal is output simultaneously with the output of the T code of the tool. When a new tool is selected, this signal can be utilized for automatic measurement of compensation amounts of the tool, etc.

11.3.4 Tool skip signal

A tool before its life end can forcibly be changed in one of the following ways.

(i) Specifying the group concerned and inputting the Tool Skip signal selects the next tool in the group with the next T code command.

(ii) Specifying no group No. and inputting the Tool Skip signal assumes that the tool being selected has been specified. The other is the same as in (i).

Whether (i) or (ii) is to be taken is set by parameter. In either way, life is counted from 0. When, however, the Tool Skip signal is input to the last tool, the Tool Change signal is output.

(Note) Input neither Tool Change Reset signal nor the Tool Skip signal when either STL or SPL lamp is on or when both STL and SPL lamps are on.

11.4 Display and Input of Tool Data

11.4.1 Display and modification of tool group data

Tool group data in the part program storage & editing area can be displayed and modified as with ordinary program editing. Be sure to execute the modified program, as was discussed in Section 11.1. Otherwise, it will not be stored in the tool life data area.

11.4.2 Display of tool life data during program execution

Pressing the DGNOS button twice in any mode displays the first page of the tool life data in the following format on the CRT screen.

```
Group No.            Life
TOOL LIFE DATA 01 :   O0001 N0012
               SELECTED GRP 001      No. of group being commanded
GRP 001:  LIFE 9800  COUNT 6501      Life counter
    *0034 *0078 *0012 *0056          Tool Nos. of group 1
    *0090 @0076  0032  0098            *: tool at life end
     0054  0010  0099  0087            @: tool being used
     0077  0066  0055  0024            #: tool skipped
GRP 002:  LIFE 0500  COUNT 0112
    *0011 *0022 *0201 *0144
    *0155 *0066 *0176 *0188          Tool Nos. of group 2
    #0019  0234  0007  0112
     0156  0090  0016  0232
GRP TO BE CHANGED:
     003 006 012 013 014 ---->       Group Nos. of tools to be
N                   LSK  INC         changed.
```

One page displays data on two groups. Pressing the PAGE [↓] button successively displays data on the following groups. Up to five group Nos., for which the Tool Change signal is being issued, are displayed at the bottom of each page. An arrow shown in the figure is displayed for six or more groups, if any. To know data on a particular group, select address N, input the group No., and press the [INPUT] button. Pressing the CURSOR [↓] button moves the cursor to GRP of the next group and displays its data.

11.4.3 Presetting of tool life counters

To change a life counter, select the MDI mode then:

(i) Key in P□□□□ and press the [INPUT] button.
The counter of the group at the current cursor position is preset to □□□□. Other data on this group remains unchanged.

(ii) Key in P-9999 and press the [INPUT] button.
All data executed so far, including *, of the group at the current cursor position is cleared, providing the same effect as tool reset for the group (see Section 11.3.2).

11.5 Notes

The part program storage & editing area is reduced by 6.2 meters of tape equivalent from the end, which is used as the tool life data area. When data is registered in the part program storage & editing area in the EDIT mode as discussed in Section 11.1, it is further occupied by so much area. When data is set directly into the tool life data area in the TAPE mode, the part program storage & editing area is not additionally occupied, but no data can be displayed and edited.

12. INDEX TABLE INDEXING FUNCTION

Index table of the machining center can be indexed using the fourth axis (axis B, for example) of the FANUC SYSTEM 6M-MODEL B (FS6M-B).

Only the indexing angle need be specified with address B, and no M-code need be specified to clamp or unclamp the table, resulting in simplified programming.

12.1 Command Entry

12.1.1 Input unit

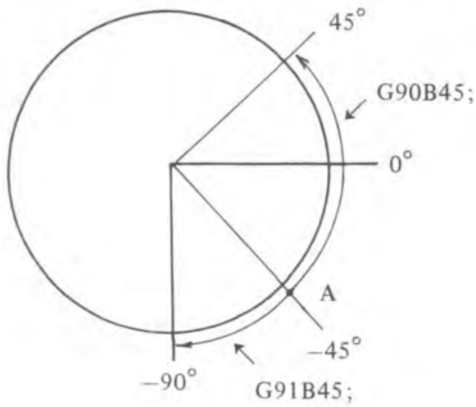
When no decimal points are used; B1 1° or 0.001° (Parameter setting)
When decimal points are used; B1. 1°

(Note) When decimal points are used, a PS alarm occurs if the values lower than the decimal points are specified (No. 180). That is values less than 1° cannot be specified.

12.1.2 Absolute/incremental commands

Absolute and incremental commands can be specified using G90 and G91, respectively.
Absolute command G90 B45; Indexes to the 45° position.
Incremental command G91 B-45; Turns 45° in the minus derection, then indexes the position.

When assuming point A as the current position, the commands described above work as shown in the figure.



12.1.3 Number of simultaneously controlled axes

Axis B must be specified as only one axis. If axes X, Y, and/or Z are specified in the same block, a PS alarm occurs (No. 181).

12.2 Least Command Increment

0.001°/pulse

12.3 Feed Rate

Independent of the group 01 G code status (G00, G01, G02, or G03), the feed rate of axis B is always rapid rate. When axis B is specified in the G01, G02, or G03 mode, these modes are valid for other axes in the subsequent blocks. There is no need to respecify the G01, G02, or G03.

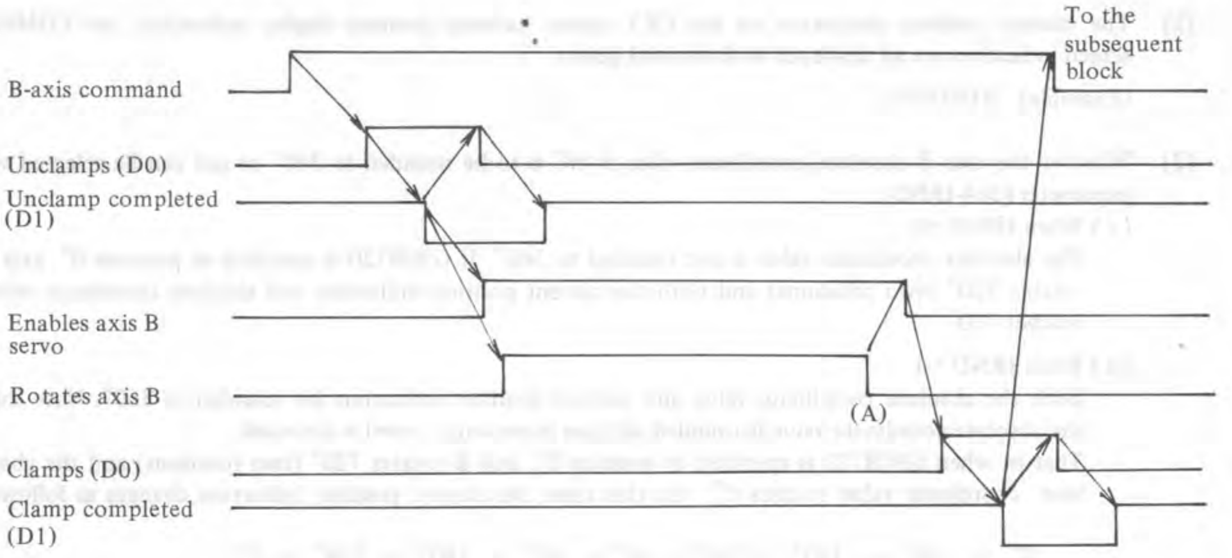
G01X10.F5; Axis X moves at cutting feed rate.
B45 ; Axis B moves at rapid rate.
X20 ; Axis X moves at cutting feed rate (G01 is valid).

Dry run is not valid.

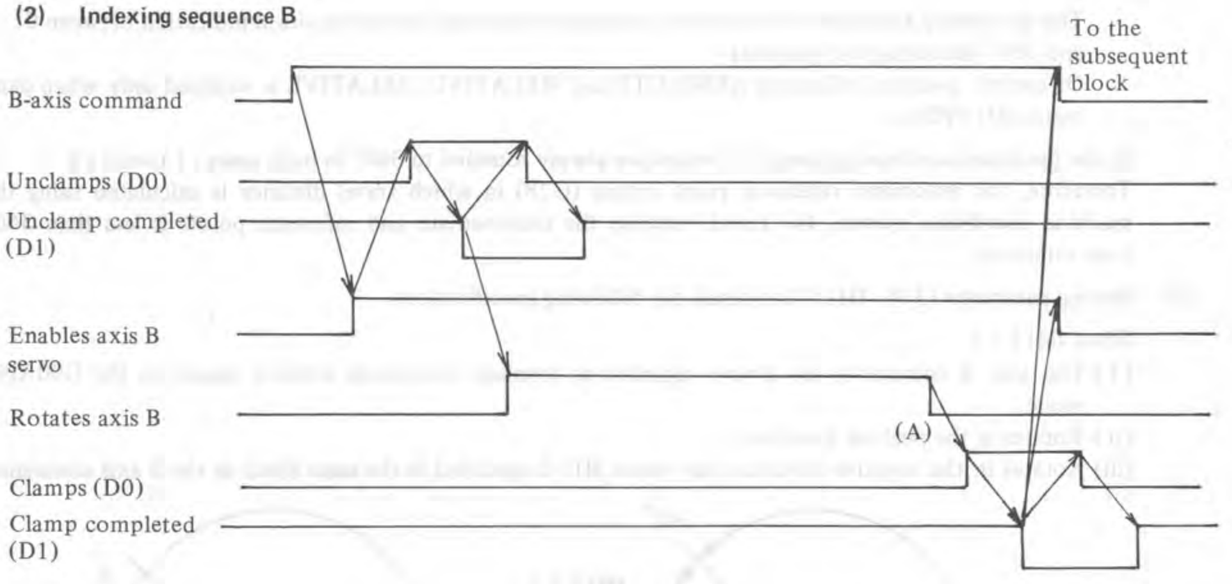
12.4 Clamp and Unclamping the Index Table

Before and after traverse of axis B, the index table is automatically unclamped and clamped, respectively.

(1) Indexing sequence A



(2) Indexing sequence B



Select sequence A or B described above by setting the parameter (IDXB-No. 315). In-position check is always done at point (A).

(Note 1) When a reset is performed in the clamp completion wait or unclamp completion wait status, the clamp or unclamp signal is cleared and NC quits the completion wait status to enter the reset status.

(Note 2) Even if the system is reset in the clamp or unclamp status, no change occurs except that the clamp or unclamp signal is cleared. That is, the unclamp or clamp sequence cannot be performed automatically through reset.

(Note 3) The clamp or unclamp completetion wait status is indicated in the diagnostic status display (DGN 701-BCNT).

12.5 JOG/STEP/HANDLE

Operating axis B in the JOG/STEP/HANDLE mode is invalid, except during manual reference point return in the JOG mode. When the axis selection signal is set to 0 during manual reference point return, operation stops immediately and the clamp command is not executed. To avoide these circumstances, design the maching-side sequence so that the axis selection signal is not set to 0 until the reference point return is completed.

12.6 Other Notes

(1) The current position indication on the CRT screen, external position display indication, and COMND screen indication are all displayed with decimal points.

(Example) B180.000

(2) Whether the axis B absolute coordinate value in NC is to be rounded to 360° or not can be selected via papameter (314-IRND).

(i) When IRND = 0

The absolute coordinate value is not rounded to 360°. If G90B720 is specified at position 0°, axis B rotates 720° (two rotations) and both the current position indication and absolute coordinate value reaches 720°.

(ii) When IRND = 1

Both the absolute coordinate value and current position indication are rounded to 360°. Note that the absolute coordinate value is rounded after an incremental travel is obtained.

That is, when G90B720 is specified at position 0°, axis B rotates 720° (two rotations) and the absolute coordinate value reaches 0°. At this time, the current position indication changes as follows:

0° → 90° → 180° → 270° → 0° → 90° → 180° → 270° → 0°

This processing keeps both the absolute coordinate value and current position indication between 0° and 359° (including the margins).

Of current position indication ABSOLUTE and RELATIVE, RELATIVE is rounded only when parameter 007 PPD is 1.

In the machine coordinate system, the values are alwasy rounded to 360° in both cases (i) and (ii).

Therefore, for automatic reference point return (G28) in which travel distance is calculated using the machine coordinate system, the travel between the intermediate and reference points is less than 360° (one rotation).

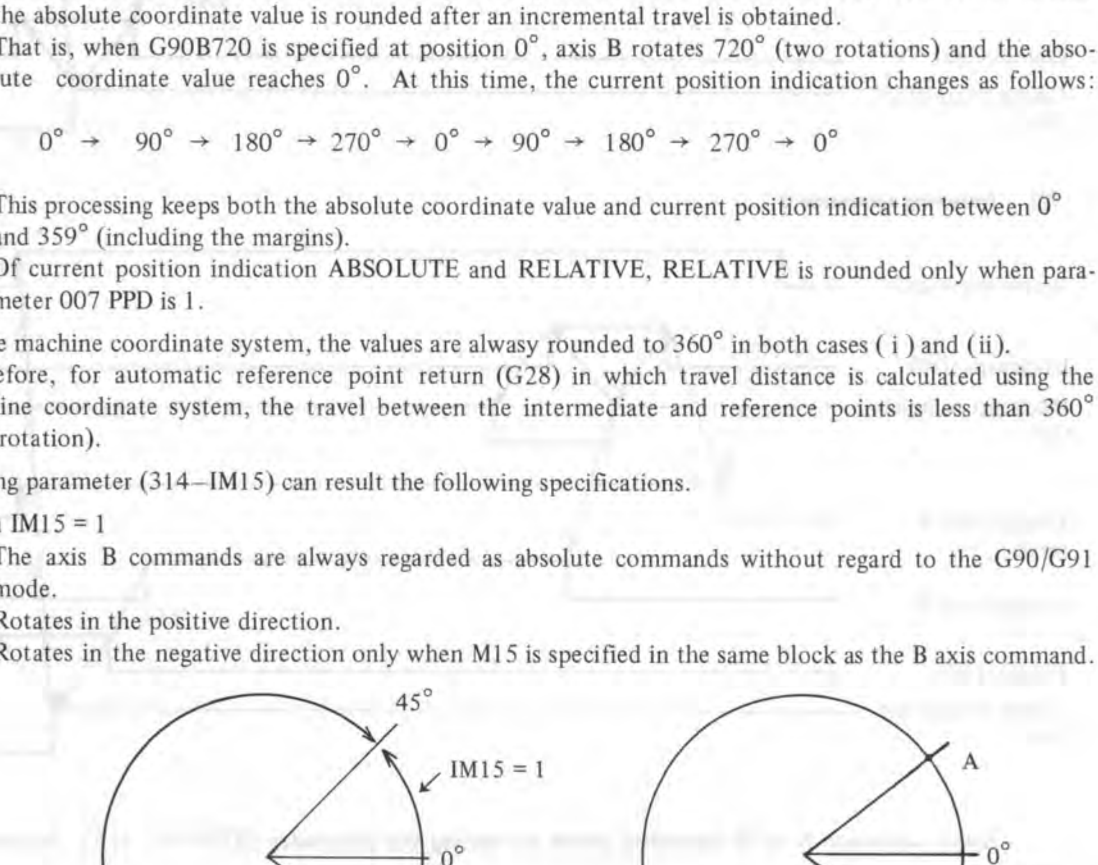
(3) Setting parameter (314–IM15) can result the following specifications.

When IM15 = 1

(i) The axis B commands are always regarded as absolute commands without regard to the G90/G91 mode.

(ii) Rotates in the positive direction.

(iii) Rotates in the negative direction only when M15 is specified in the same block as the B axis command.



B45; B315M15;

(Note) Although M15 is processed in NC, both the MF and M code are transmitted to the machine side. Therefore, return FIN to NC.

(4) Feed hold/reset/emergency stop are all valid while axis B is moving. Not to stop at a mid-way position, the machine side is required to take a necessary measure.

(5) When this option is provided, the 4th axis servo-off signal (*SVF4) is invalid.

(6) For the specifications, parameters and inter-machine connections not mentioned here, use those for the standard additional axis.

13. PROGRAM TAPE COMPOSITION

The program written on a process sheet is punched on paper tape. Various devices for punching paper tape are on the market. Any of them will do so long as the paper tape is punched as shown in Section 13.1.

The SYSTEM P-G is sold as a paper tape punch device.

(Note) *The SYSTEM P-G also edits programs using a graphic display. It can also be used as an automatic programming system.*

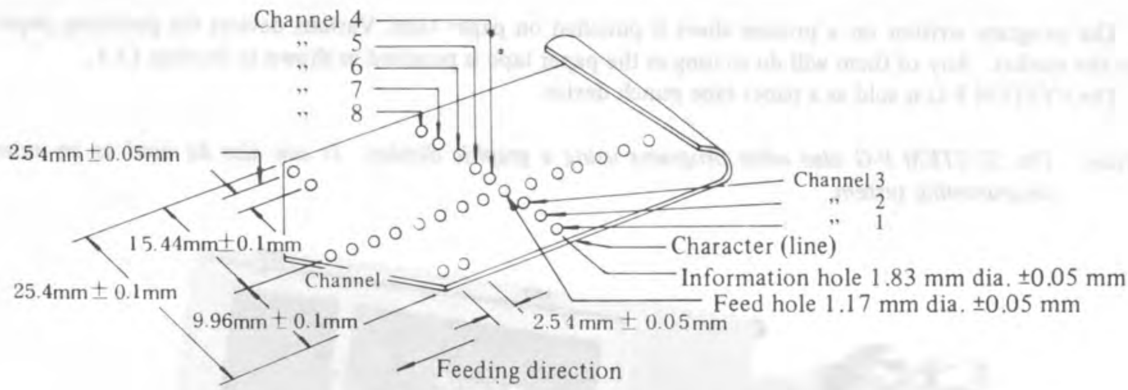
13.1 Paper Tape Used for NC Control

A program containing NC machine tool commands is ordinarily punched on paper tape and read into the NC system through a Tape Reader. An 8-unit paper tape 1 inch in width as normally used for NC control, and dimensions of the tape, such as width, thickness, hole-position, hole-diameter and so on, are determined by various standards including EIA, ISO or JIS. The following table shows the specification of paper tape that can be used in this control.

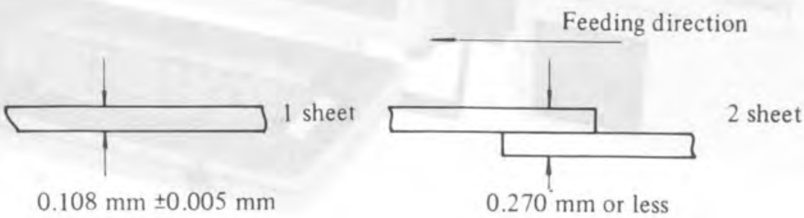
Table 13.1 Specification for Paper Tape Used by NC

	Item		Tape reader without reels	Tape reader with reels
1	Kind of tape		8-channel paper tape (Mylar tape or so on cannot be used)	Same as left
2	Percentage of light transmission $= (\frac{\text{Transmission light}}{\text{Incident light}} \times 100\%)$		40% or less	0.1% or less
3	Colour of tape		Any colour can be used such as black, grey, blue, pink, white, etc., if percentage of light transmission is 40% or less.	Black only
4	Standard	Associated with material of paper tape	JIS C 6243, EIA RS-227-A or ISO 1729 (Provided that percentage of light transmission conforms to Item (2).)	Same as left
		Associated with dimensions and locations of holes in paper tape	JIS C 6246, EIA RS-227-A or ISO 1154	Same as left

Example of EIA standard RS-227 is shown below:



(a) Dimensions of the punched tape



(b) Thickness of tape

Standard dimensions of punched tape (EIA, RS-227)

13.2 Tape Code

A punched line on paper tape represents a numeral, character or sign, and the line is called a character. A character is composed of 8-hole positions which are called the code. The following two tape codes can be used in the SYSTEM 6M : However, ISO code is an option.

(i) EIA code EIA RS-244-B
(ii) ISO code ISO 840

(1) EIA/ISO determination

The code EIA or ISO, used on a punched tape is determined automatically. When reading a paper tape in Label Skip mode (Note 1), if the first End of Block code is read as CR in EIA code, subsequent codes are read written in EIA code; while, if the first End of Block code is read as LF in ISO code, subsequent codes are read as ISO code. EIA and ISO codes cannot be mixed on a paper tape.

(Note 1) After power-on or reset, all codes read before the first EOB code are ignored. This status is called Label Skip mode.

(2) Parity check (TH check)

An alarm (TH alarm) will be generated, if the following conditions are not satisfied by the number of code-holes read in a significant information section (a block following the first CR or LF, excluding the comment section):

(i) EIA code — An odd numbered code-holes. (All punches are excluded.)
(ii) ISO code — An even numbered code-holes.

When the TH alarm is sensed, reading of the tape is stopped immediately (Note 2) and alarm No. 001 is displayed on the CRT screen. This keeps the NC machine tool from machining erroneously due to incorrect punches or a tape-read error.

(Note 2) The tape reader does not stop immediately when a TH alarm is generated. Because of the read-in buffer, the tape reader stops after reading one block or when the read-in buffer is filled. The position and the character where the TH alarm occurred are displayed in DIAGNOSE 710 and 711.

(3) Vertical check (TV check)

A parity check is performed in the vertical direction of a block, and an alarm (TV alarm) is generated if the block contains an odd number of characters (from the character following an EOB to the next EOB).

When the TV alarm is sensed, reading of the tape is stopped immediately, and alarm No. 002 is displayed on the CRT screen. This keeps the NC machine tool from machining erroneously due to incorrect punches or a tape-read error.

This check function can be specified as Valid/Invalid by the SETTING operation. When using this function, punch an ignored code such as the space code so that the number of characters is even.

13.3 Codes Used in Programming

Codes shown in the table below can be used for programming in EIA/ISO code. Codes not included in this table will be ignored when read from a paper tape, but invalid codes will be counted in the parity check which can result in a TH alarm.

Table 13.3 Codes used for programming

ISO code										EIA code											Meaning
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		
0			○	○		∘				0			○			∘					Numeral 0
1	○		○	○		∘			○	1						∘			○		Numeral 1
2	○		○	○		∘		○		2						∘		○			Numeral 2
3			○	○		∘		○	○	3				○		∘		○	○		Numeral 3
4	○		○	○		∘	○			4						∘	○				Numeral 4
5			○	○		∘	○		○	5				○		∘	○		○		Numeral 5
6			○	○		∘	○	○		6				○		∘	○	○			Numeral 6
7	○		○	○		∘	○	○	○	7						∘	○	○	○		Numeral 7
8	○		○	○	○	∘				8					○	∘					Numeral 8
9			○	○	○	∘			○	9				○	○	∘			○		Numeral 9
A		○				∘			○	a		○	○			∘			○		Address A
B		○				∘		○		b		○	○			∘		○		?	Address B
C	○	○				∘		○	○	c		○	○	○		∘		○	○	?	Address C
D		○				∘	○			d		○	○			∘	○				Address D
E	○	○				∘	○		○	e		○	○	○		∘	○		○		Address E
F	○	○				∘	○	○		f		○	○	○		∘	○	○			Address F
G		○				∘	○	○	○	g		○	○			∘	○	○	○		Address G
H		○			○	∘				h		○	○		○	∘				?	Address H
I	○	○			○	∘			○	i		○	○	○	○	∘			○		Address I
J	○	○			○	∘		○		j		○		○		∘			○	?	Address J
K		○			○	∘		○	○	k		○		○		∘		○			Address K
L	○	○			○	∘	○			l		○				∘		○	○		Address L
M		○			○	∘	○		○	m		○		○		∘	○				Address M
N		○			○	∘	○	○		n		○				∘	○		○		Address N
O	○	○			○	∘	○	○	○	o		○				∘	○	○			Address O
P		○		○		∘				p		○		○		∘	○	○	○		Address P
Q	○	○		○		∘			○	q		○		○	○	∘					Address Q
R	○	○		○		∘		○		r		○			○	∘			○		Address R
S		○		○		∘		○	○	s			○	○		∘		○			Address S
T	○	○		○		∘	○			t			○			∘		○	○		Address T
U		○		○		∘	○		○	u			○	○		∘	○				Address U
V		○		○		∘	○	○		v			○			∘	○		○	?	Address V
W	○	○		○		∘	○	○	○	w			○			∘	○	○			Address W
X	○	○		○	○	∘				x			○	○		∘	○	○	○		Address X
Y		○		○	○	∘			○	y			○	○	○	∘				?	Address Y
Z		○		○	○	∘		○		z			○		○	∘			○		Address Z
DEL	○	○	○	○	○	∘	○	○	○	Del		○	○	○	○	∘	○	○	○	*	Delete (cancel an error punch).
NUL						∘				Blank						∘				*	Not punched. Can not be used in significant section in EIA code.
BS	○				○	∘				BS			○		○	∘		○		*	Back space
HT					○	∘			○	Tab			○	○	○	∘	○	○		*	Tabulator
LF or NL					○	∘		○		CR or EOB	○					∘					End of block
CR	○				○	∘	○		○											*	Carriage return
SP	○		○			∘				SP				○		∘				*	Space
%	○		○			∘	○		○	ER					○	∘		○	○		Absolute rewind stop
(			○		○	∘				(2–4–5)				○	○	∘		○			Control out (a comment is started)
)	○		○		○	∘			○	(2–4–7)		○			○	∘		○			Control in (the end of a comment)
+			○		○	∘		○	○	+		○	○	○		∘				*	Positive sign
–			○		○	∘	○		○	–		○				∘					Negative sign
:			○	○	○	∘		○													Colon
/	○		○		○	∘	○	○	○	/			○	○		∘			○		Optional block skip
.			○		○	∘	○	○		.		○	○		○	∘		○	○		Period (A decimal point)
#	○		○			∘		○	○											*	Sharpe
$			○			∘	○													*	Dollar sign
&	○		○			∘	○	○		&					○	∘	○	○		*	Ampersand
'			○			∘	○	○	○											*	Apostrophe
*	○		○		○	∘		○												*	Asterisk
,	○		○		○	∘	○			,			○	○	○	∘		○	○	*	Comma
;	○		○	○	○	∘		○	○											*	Semicolon
<			○	○	○	∘	○													*	Left angle bracket
=	○		○	○	○	∘	○		○											*	Equal
>	○		○	○	○	∘	○	○												*	Right angle bracket
?			○	○	○	∘	○	○	○											*	Question mark
@	○	○				∘														*	Commercial at mark
"			○			∘		○												*	Quotation
[	○	○		○	○	∘		○	○											*	Left brace
]	○	○		○	○	∘	○		○											*	Right brace

218

(Note 1) The codes with an asterisk are read into tape memory only when specified in the comment section. They are ignored in significant information sections.

(Note 2) The codes with a question mark are read in tape memory only when specified in a comment. They generate an alarm if used in significant information sections.

(Note 3) When a custom macro option is used, the following codes can also be used in significant information sections.
*[,], #, *, =, +, B, C, H, J, V, and Y in ISO*
[,], &, +, a code set by parameter, B, C, H, J, V, and Y in EIA

(Note 4) Codes not included in this table and with correct parity are always ignored.

(Note 5) A code that does not have the correct parity generates the TH alarm but ignored in a comment section.

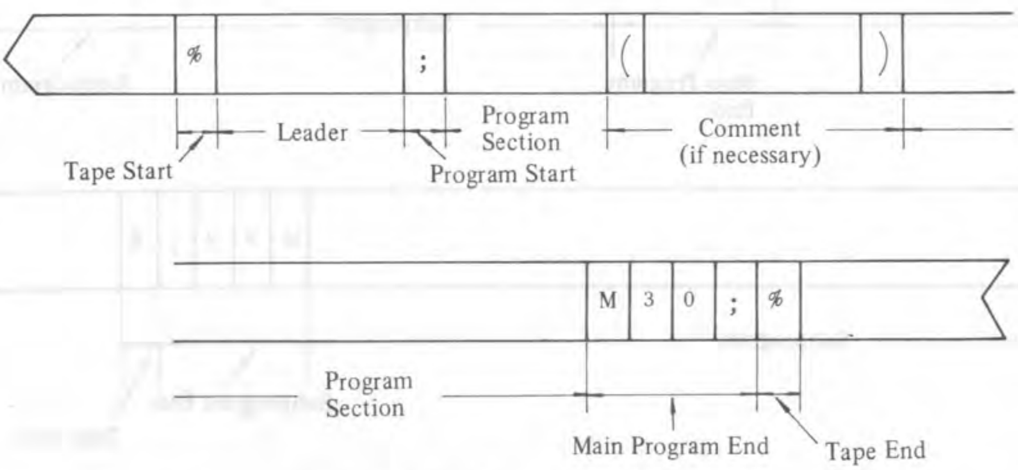
(Note 6) A character with all eight holes punched does not generate TH alarm even in EIA code.

13.4 Program Tape

A program punched on a paper tape is composed of the following sections:

(i) Tape Start
(ii) Leader
(iii) Program Start
(iv) Program Section
(v) Comment
(vi) Program End
(vii) Tape End

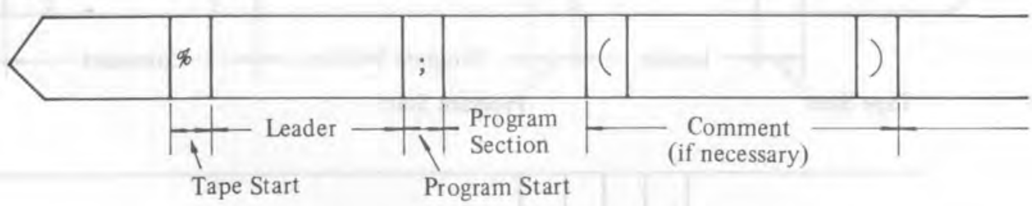
(1) A main program on a tape

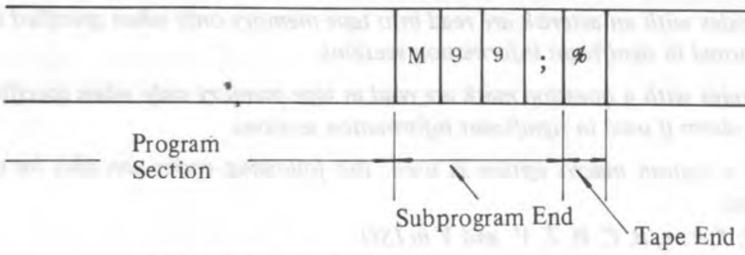


(Note 1) M02 can be used instead of M30 at the end of the main program.

(Note 2) When in ISO code, LF is used for ;, while in EIA code, CR is used.

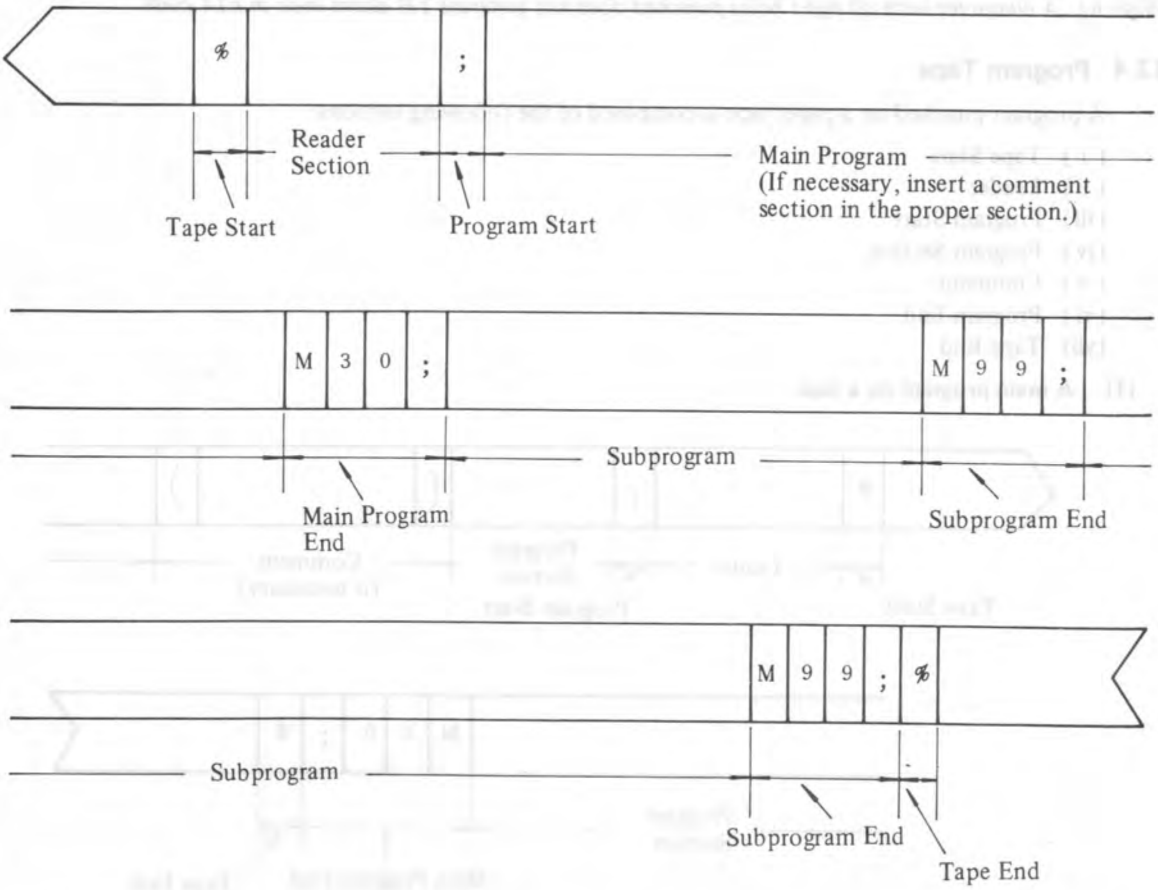
(2) One subprogram on a tape





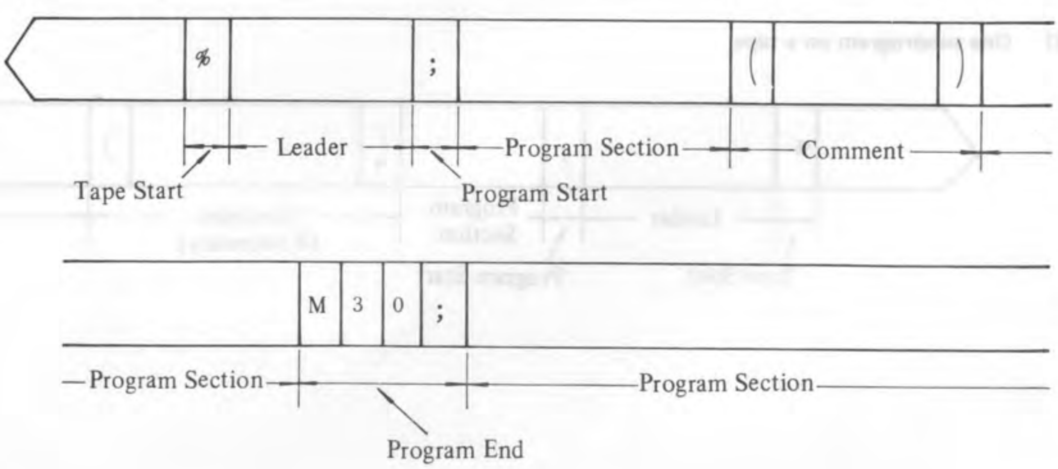
A subprogram must be used after being entered into the tape memory area.

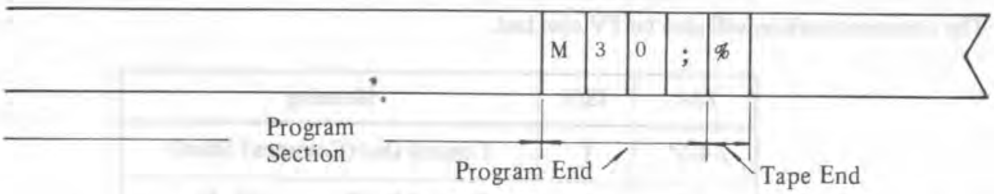
(3) One main program and a number of subprograms on a tape



A program including subprograms on a tape must be used after being entered into the tape memory area.

(4) A number of programs on a tape

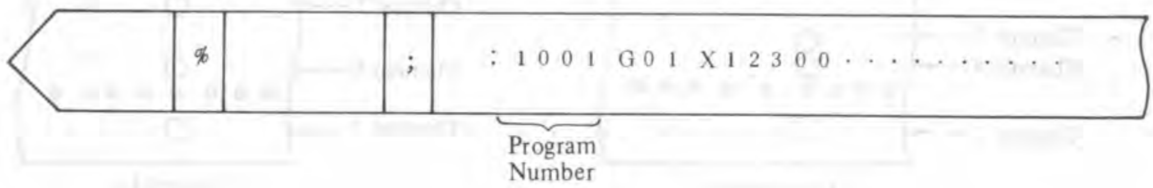




A program number must be put at the first block of each main program section or subprogram section.

A program number is a four-digit number following address O. (When in ISO, : can be also used instead of O)

Program numbers are not necessarily given. See Section 2.6 for details.



13.4.1 Tape start

The Tape Start must be punched at the top of the paper tape by using the following code. This section is necessary to stop rewinding the tape when used with a Tape Reader with Reels. This section may be omitted when the tape is used with a Tape Reader without Reels.

EIA	ISO	Meaning
ER	%	Tape Start (Rewind Stop)

13.4.2 Leader section

Information punched before the first LF (for ISO) or CR (for EIA) on a paper tape is called the Leader Section. Normally, the NC unit is set to the Label Skip condition when turning the power on or resetting and this section of paper tape is loaded on a Tape Reader. The Tape Reader reads but skips this section (by the Label Skip function). Generally, a label for a tape is punched in this section. Since no parity check is performed on the skipped leader section, any combination of codes other than LF and CR may be punched in this section.

The label skip function is used to skip all information encountered from the time the power is turned on or a resetting is performed until reading the first EOB code. (There is no parameter used to delete label skip function.)

13.4.3 Program start

Punching the following code immediately after the end of the Leader (at the top of the first block of a program) indicates the start of a program. This code is necessary to release the Label skip function.

EIA	ISO	Meaning
CR	LF	Program Start

13.4.4 Program section

The part of the paper tape punched between the Program Start and Program End code (except the Comment section explained in Section 13.4.5) is called the Program Section. Information specifying tool travel and machine ON/OFF functions is punched in this section. This section is called the significant section and is different from the non-significant section which includes the Leader and Comment Section. Details on the Program Section are given in Section 13.5.

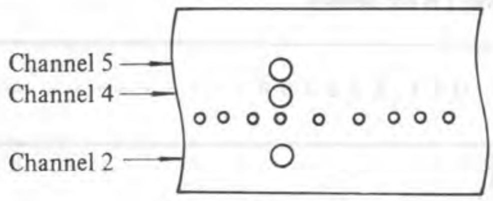
13.4.5 Comment section

Information punched in the section between Control Out and Control In codes is read but skipped and is regarded as a comment. The TH check is not performed on this section, therefore, the leader or comments can be punched by using any combination of punch-holes desired.

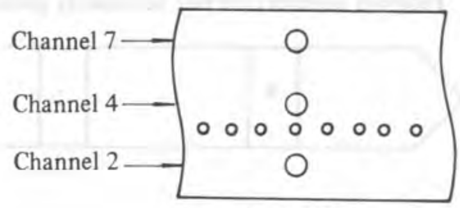
The comment section will also be TV checked.

EIA	ISO	Meaning
2-4-5*	(	Control Out (Comment Start)
2-4-7*	)	Control In (Comment End)

*: Hole combinations in EIA code.



Control Out



Control In

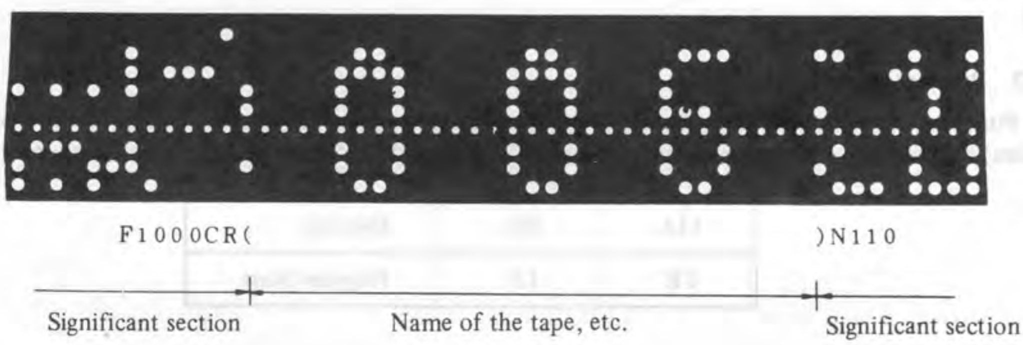
(Example) In ISO code

```
.........                       LF
N1000G00X...........            LF
(MEASURE WORK..........)        LF
N1001G01X..........             LF
.........                       LF
```

When a command tape is read into memory, for memory-controlled operation, the comment section is read into memory. However codes not listed in the Table 13.3 are ignored and are not read into memory. When data in the memory is punched out, information in the Comment Section is also punched out. The Comment Section is displayed on the screen of the CRT. However codes ignored when read-in are neither punched nor displayed on the CRT screen.

During a memory-controlled operation in memory command mode, the comment section in memory is ignored and operation is executed.

(Example) Used as heading



(Note 1) *The rewind stop code (% or ER) cannot be used in the comment section. Reading this code resets the NC.*

(Note 2) *When a comment section is in the middle of a program section and it is long, movement is interrupted for a long time. Therefore, insert comment sections in a place where movement can be interrupted or a place specifying no movement.*

(Note 3) *The length of the comment section is unlimited.*

(Note 4) *When the control out code does not come first and only the control in code is read in, it is ignored.*

(Note 5) *The TV check for the comment section can be made disabled by specifying parameter TVC.*

13.4.6 Program end

The end of a program is indicated by punching the following code at the end of the Program Section.

EIA	ISO	Meaning
M02 CR	M02 LF	Program End
M30 CR	M30 LF	Program End and Rewind
M99 CR	M99 LF	Subprogram End

When the End of Program is sensed during the execution a program, the NC unit terminates the execution of the program and enters a cleared status. In case of M30CR or M30LF, the tape is rewound (in Tape Command mode) or the program is returned to its beginning (in Memory Command mode). When the code is sensed in a subprogram, control is returned to the program which called the subprogram. (Whether or not the tape is rewound by execution of M30 depends on the machine tool.)

13.4.7 Tape end

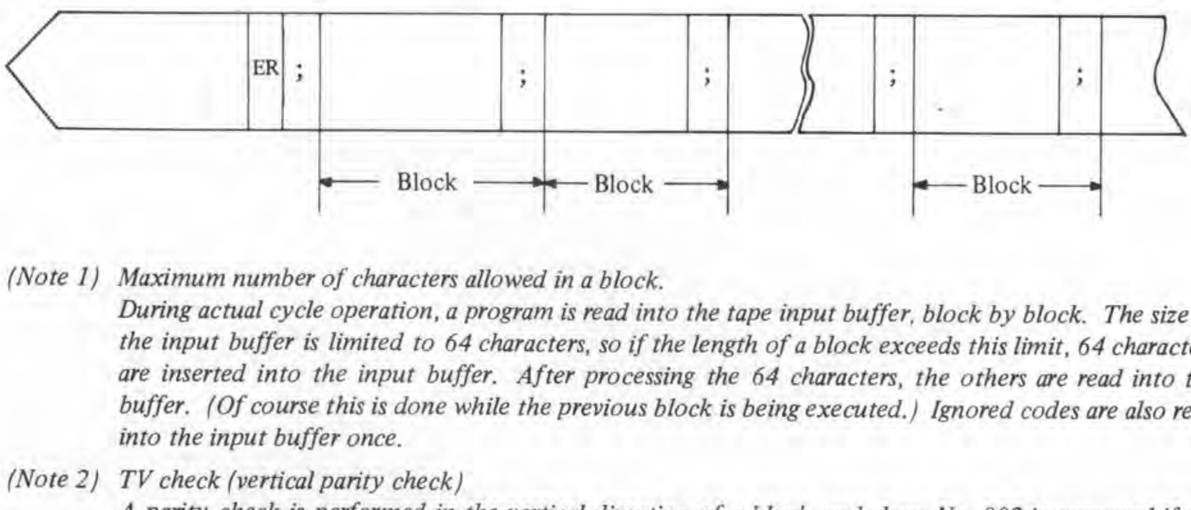
The end of a tape is indicated by punching the code below immediately following the Program End of the last program on a tape.

EIA	ISO	Meaning
ER	%	Tape End

(Note) If M02 and M30 are not at the end of the program section, and ER (EIA) or % (ISO) is about to be executed, the NC enters a reset status.

13.5 Block Composition

A program section is composed of a number of blocks. A block is partitioned into a number of words, and the end of a block is identified by an End of Block Code (LF in ISO code and CR in EIA code).



(Note 1) Maximum number of characters allowed in a block.
During actual cycle operation, a program is read into the tape input buffer, block by block. The size of the input buffer is limited to 64 characters, so if the length of a block exceeds this limit, 64 characters are inserted into the input buffer. After processing the 64 characters, the others are read into the buffer. (Of course this is done while the previous block is being executed.) Ignored codes are also read into the input buffer once.

(Note 2) TV check (vertical parity check)
A parity check is performed in the vertical direction of a block, and alarm No. 002 is generated if the block is found to contain an odd number of characters (between a code immediately follwoing an EOB and the next EOB). The TV check is not performed for a part of tape which is read but skipped by the Label Skip function, but the Command Section between "(" and ")" is checked for the total number of characters in each block. The TV check function can be specified Valid/Invalid by using the MDI unit. (See Section 5.7 in Chapter IV.)

(Note 3) In program examples in this manual, ; is used instead of the End of Block code (LF for ISO and CR for EIA), but the proper EOB code LF (ISO) or CR (EIA) must be used in actual programming.

IV. OPERATION

1. TURNING ON AND OFF POWER

1.1 Turning On Power

(1) Make sure that the front and back doors of the equipment and the door of the tape reader are closed.
(2) Turn on the power according to the manual issued by the machine tool builder.
(3) Make sure that something is displayed on the CRT. Something is displayed in several seconds after the power is turned on.
(4) Make sure that the fan motor is on.

1.2 Turning Off Power

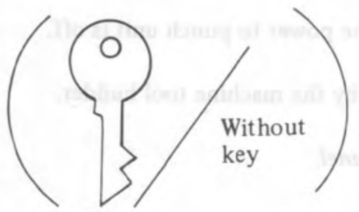
(1) Make sure that the CYCLE START LED on the operator's panel is off.
(2) Make sure that the movable parts on the machine side have stopped.
(3) Set the tape reader switch to RELEASE.
(4) If the punch unit (ASR33/43 or FACIT4070) is provided, insure that the power to punch unit is off.
(5) Push the POWER OFF button for 1 to 2 seconds.
(6) Turn off the power to the machine side according to the manual issued by the machine tool builder.

(Note) When turning on or off the power, don't push the key on the MDI panel.

2. LOCK KEY

A LOCK key can be provided on the operator's panel on the machine tool.
If the LOCK key is not released, some operations cannot be performed.

. . . . In the explanations mentioned in section 5, the marked operations are possible only when the LOCK key is released. However associated data can be displayed on the CRT screen.



. . . . Whether or not the marked operations are possible when the LOCK key is locked depends on the system parameter SETE.

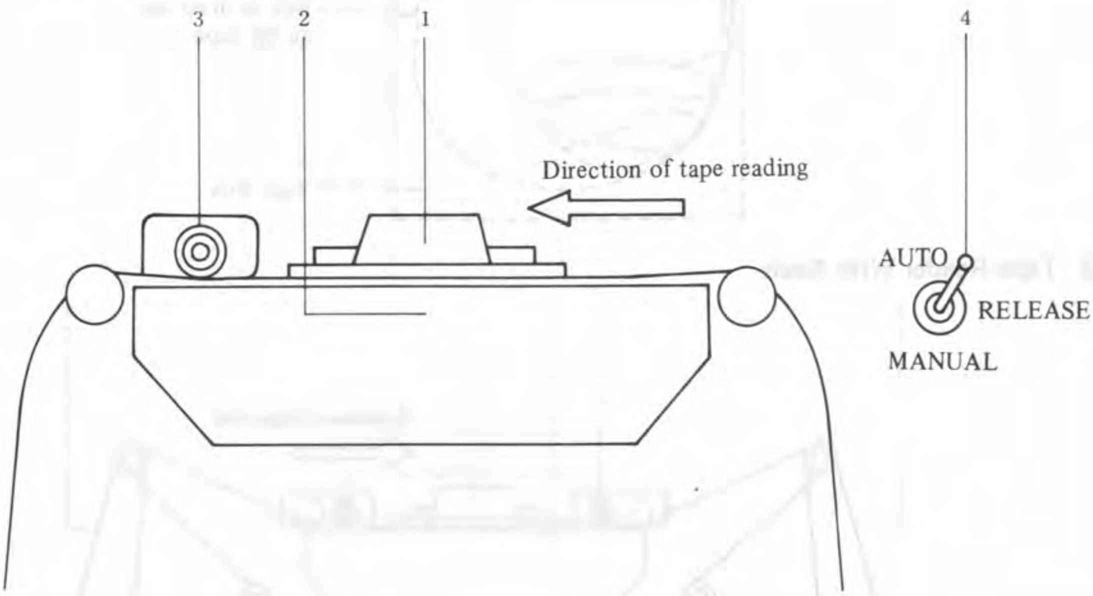
229

3. TAPE READER

The characteristics of the Tape Reader are as follows:

		Tape Reader without Reels	Tape Reader with Reels
Read Speed		250 characters/sec (50 Hz) 300 characters/sec (60 Hz)	
Rewind Speed			500 characters/sec (50 Hz) 600 characters/sec (60 Hz)
Reel Capacity			Diameter 187 mm 150 m (tape length)
Tape box capacity	Free-standing type	Approximately 30 m max.	
	Built-in type 1	Approximately 10 m max.	
	Built-in type 2	Approximately 10 m max.	

3.1 Tape Reader Without Reels



1. Light Sources

An LED (light emitting diode) is mounted for each channel and for the feed hole (9 diodes in total). A built-in Stop Shoe functions to decelerate the tape.

The light source is attracted to the optical reader by a magnet so that the tape will be held in the correct position. This unit can be opened upward, by turning the tape reader control switch to the RELEASE position (this turns off the magnet).

2. Optical Reader

Reads data punched on the tape, through a glass window. Dust or scratches on the glass window can result in reading errors. Keep this window clean.

3. Capstan Roller

Controls the feeding of tape as specified by the Control Unit.

4. Tape Reader Control Switch

A 3-position switch used to control the Tape Reader.

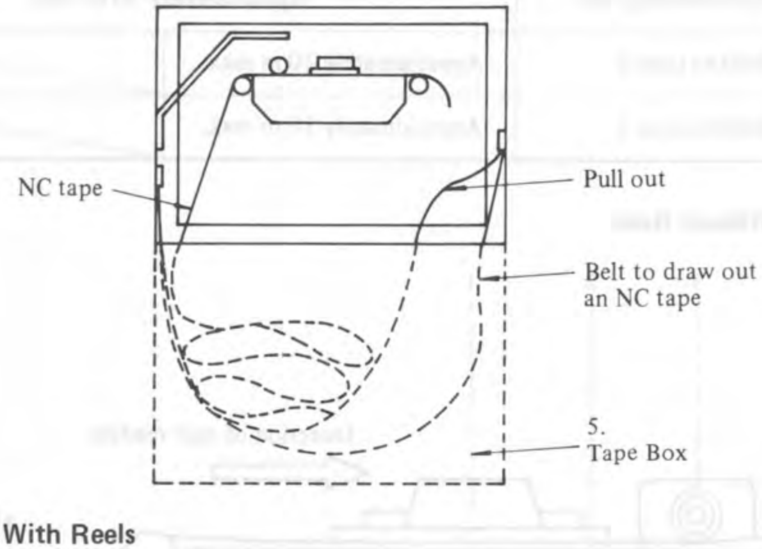
RELEASE . . . . . . The tape is allowed to be free, or used to open the light source. When loading or unloading the tape, this position is selected.

AUTO . . . . The tape is set to fixed position by the Stop Shoe. The feed and stop of the tape is controlled by commands from the Control Unit.
During a tape-controlled operation or data input from tape, the Light Source must be closed and this position must be selected.

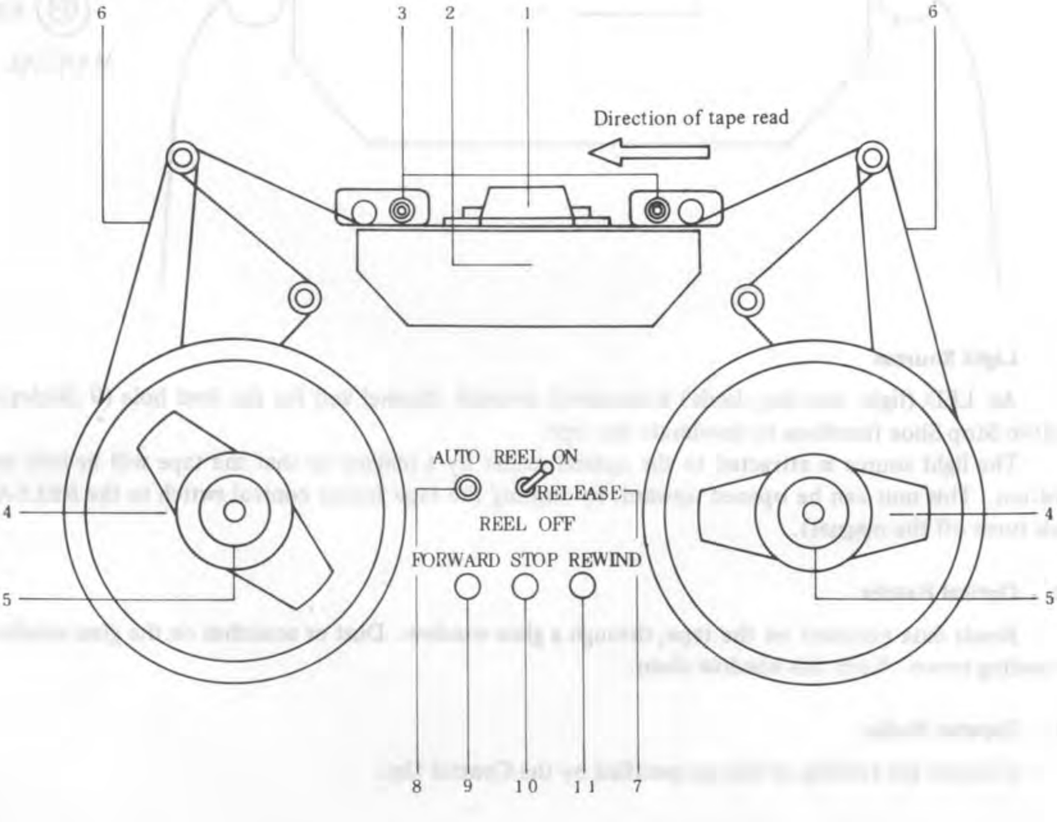
MANUAL . . . . . . The tape can be fed in the forward reading direction. If another position is selected, the tape feed is stopped.

5. Tape Box

A Tape Box is located below the Tape Reader. A belt used to draw out a paper tape is located inside the box. The paper tape can easily be pulled out using this belt.



3.2 Tape Reader With Reels



1. Light Source

An LED (light emitting diode) is mounted for each channel and for the feed hole (9 diodes in total). A built-in Stop Shoe functions to decelerate the tape.

The light source is attracted to the optical reader by a magnet so that tape will be held in the correct position adequately pressurized. The light source can be opened upward by turning the tape reader control switch to the RELEASE position (Turn off the magnet).

2. Optical Reader Unit

Reads data punched on the tape, through a glass window. Dust or scratches on the glass window can result in reading errors. Keep this window clean.

3. Capstan Rollers

Control the feeding of the tape as specified by the Control Unit. One capstan is for Forward Feed; the other, for Reverse Feed.

4. Reel

Used to feed out and take up the NC tape. A 150 m NC tape can be used.

5. Reel boss

A boss used to accept the reel. Provided with pawls to secure the reel. Driven by a reel motor, the boss turns the reel as specified by the Control Unit.

6. Tension Arm

Absorbs the internal delay of the reel motor response at the start/stop of tape motion. This arm also adjusts the feeding of the tape from the feed reel.

7. Tape Reader Control Switch

A 3-position switch used to change the operating mode of the Tape Reader.

RELEASE ...... The tape is allowed to be free. The Light Source unit can be opened. This position must be selected when loading/unloading an NC tape.

REEL ON ...... The tape is secured by the Stop Shoe, and the reel motor starts running. This position is selected to operate the Tape Reader with a reeled tape.

REEL OFF ..... The tape is secured by the Stop Shoe, but the reel motor is not started. This position is selected to operate the Tape Reader without using reels.

When the Tape Reader is operated, or when inputting data from an NC tape, the Light Source must be closed and the switch must be set to the REEL ON or REEL OFF position.

8. AUTO Lamp

Lights while the Tape Reader is automatically operated from the Control Unit. While this lamp is lit, the manual control buttons on the Tape Reader (FORWARD, STOP and REWIND) cannot be used.

9. FORWARD Button

A push button used to feed tape in the reading direction (forward). When the AUTO lamp is OFF and the switch is at REEL ON or REEL OFF, the tape can be fed forward by pushing this button until it is stopped by one of the following conditions:

(i) 10. STOP button is pushed.
(ii) ER in EIA code or % in ISO code is read.
(iii) RESET button on the MDI & DPL panel is pushed.

10. STOP Button

A button used to stop tape feed performed by pressing the FORWARD or REWIND button.

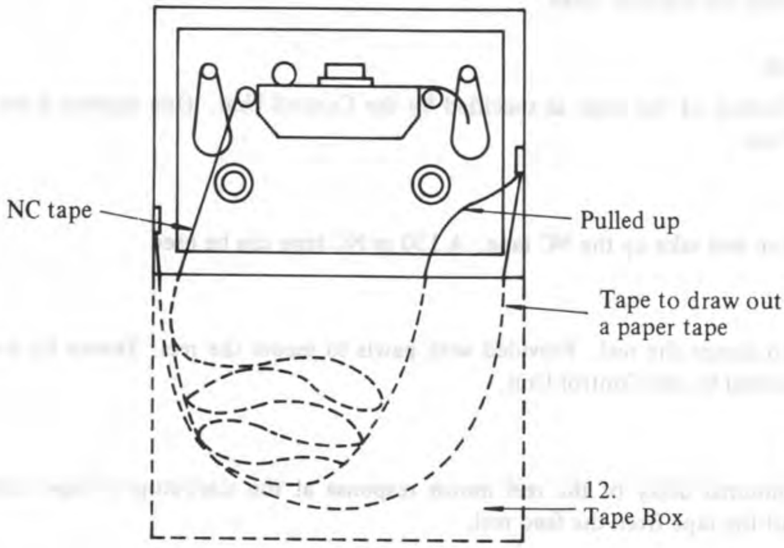
11. REWIND Button

A button used to feed the NC tape in the reverse direction. The NC tape is fed in the reverse direction when this button is pushed while the AUTO lamp is OFF and the switch is set to REEL ON or REEL OFF. Once fed in the reverse direction, the tape is rewound until one of the following condition is sensed:

(i) STOP button is pushed.
(ii) ER in EIA code or % in ISO code is read.*
(iii) RESET button on MDI & DPL panel is pushed.
The NC tape cannot stop at the specified character but will overrun.

12. Tape Box

A Tape Box is located below the Tape Reader. A belt to draw out a paper tape in the box facilitates the removal of an NC tape.

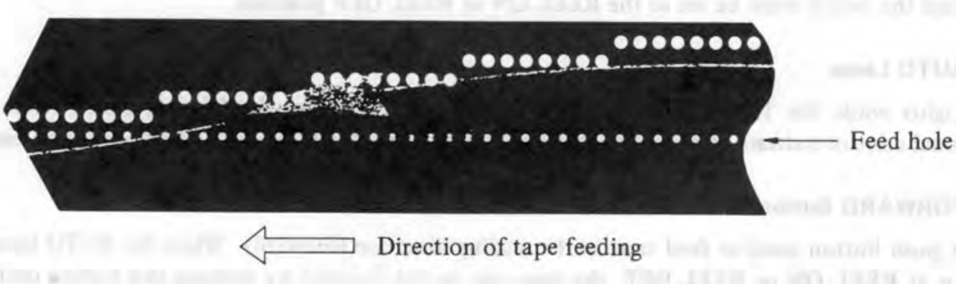


3.3 Tape Reader Handling

(1) Tape loading

(a) Tape Reader without Reels

① Open the Tape Reader Door.
② Set the Tape Reader switch to the RELEASE position.
③ Lift the Light Source Unit, and insert an NC tape between the gap. The tape must be positioned as shown in the figure, when viewed looking downward.

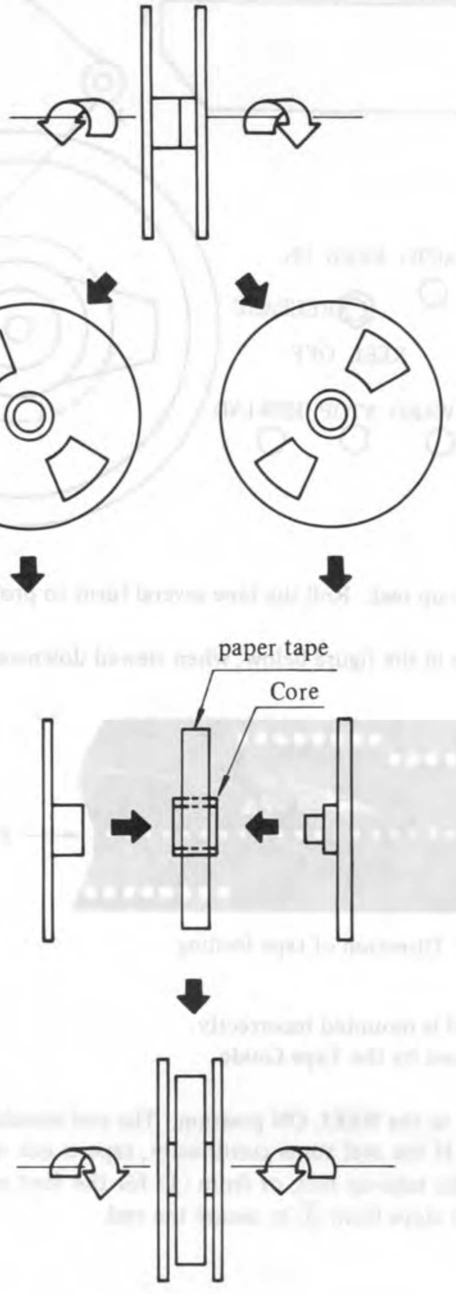


④ Pull the tape until the top of the tape goes past the Capstan roller.
⑤ Check that the NC tape is correctly positioned by the Tape Guide.
⑥ Lower the Light Source.
⑦ Turn the switch to the AUTO position.
⑧ Suspend the top and rear-end of the tape in the Tape Box.
⑨ Close the Tape Reader Door.

(b) Tape Reader with Reels (when reels are not used)
Identical to the steps in (a) except that the switch is set to the REEL OFF position, in step ⑦.

(c) Tape Reader with Reels (when reels are used)

① Correctly reel the NC tape.



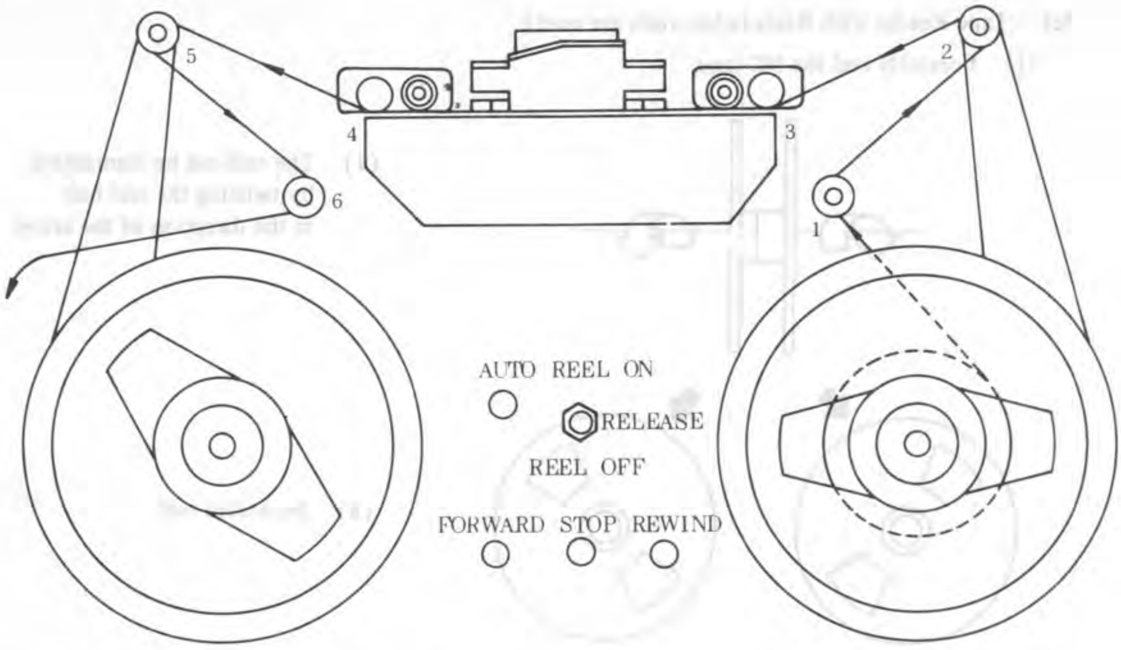
(i) The reel can be dismantled by twisting the reel hub in the direction of the arrow.

(ii) Separated reel.

(iii) Load the NC tape.

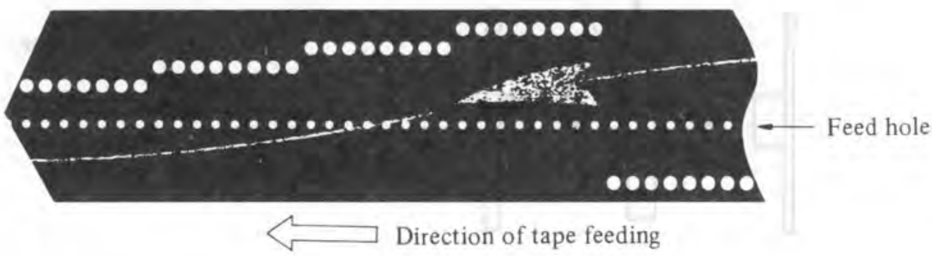
(iv) Twist the reel hub in the direction of the arrows, until the hubs are secured.

② Open the Tape Reader door.
③ Set the control switch to the RELEASE position.
④ Mount the reel with the NC tape on the boss located on the right side, and an empty reel on the left boss. Push the screw pawl inward, mount the reel, line pawl with one of reel slots, pull the pawl outward to secure the reel. The reel with the NC tape must be loaded to allow counterclockwise feeding of the NC tape (see the figure below).
⑤ Lift the Light Source.
⑥ Thread the NC tape on rollers and tension arms.



⑦ Roll the top of the NC tape on the take-up reel. Roll the tape several turns to protect idle running of the reel.

⑧ Check that the tape is received as shown in the figure below, when viewed downward.



If this condition is not obtained, the reel is mounted incorrectly.

⑨ Check that the tape is correctly positioned by the Tape Guide.

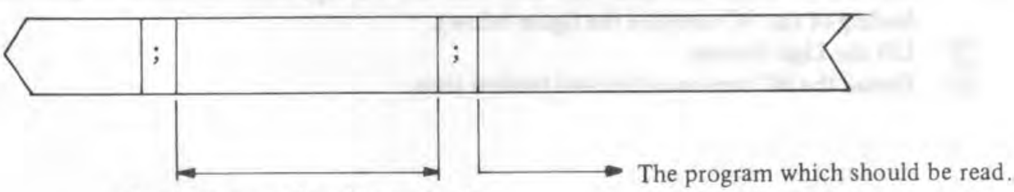
⑩ Lower the Light Source.

⑪ Change the Tape Reader control switch to the REEL ON position. The reel should be turned slightly, and tensioned by the Tension Arms. If the reel turns continually, tape is not received by the reel correctly. Repeat steps from ⑦ for the take-up reel, or from ① for the feed reel. If the reel boss runs idly while the reel stays still, repeat steps from ④ to secure the reel.

⑫ Close the door.

(Note 1) Precautions in tape loading

When the NC tape is loaded, the Label Skip function should be activated to read but skip data until the first End of Block code (CR in EIA code or LF in ISO code) is read. When loading an NC tape, the location within the tape, from which data reading should be started must be properly selected and the NC tape should be set as shown in the figure below.



(Note 2) Disconnection and connection of tape reader connection cable

Don't disconnect or connect NC ↔ tape reader connection cable (signal cable) without turning off the NC and tape reader power supply when using a separate tape reader, otherwise the PCB of the tape reader and master PCB of NC controller may be broken.
Turn off the NC and tape reader power supply before disconnecting or connectiong the connection cable, accordingly.

(2) Unloading the NC tape

(a) Tape Reader without Reels

① Open the tape reader door.
② Turn the switch to the RELEASE position.
③ Lift the Light Source and remove the tape.
④ Lower the Light Source.
⑤ Close the door.

(b) Tape Reader with Reels (when reels are not used)
Identical to the above.

(c) Tape Reader with Reels (when reels are used)

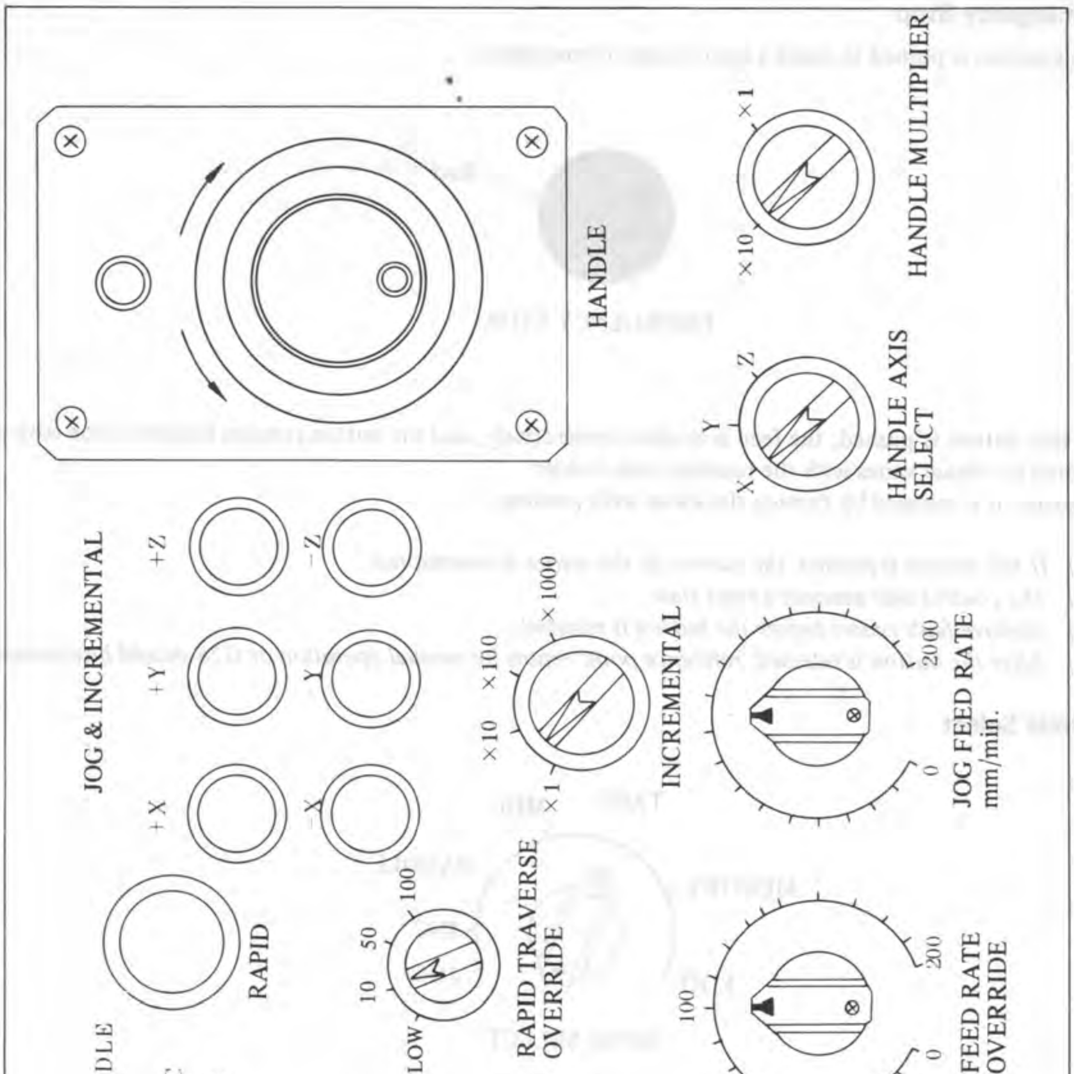
① Open the Tape reader door.
② Push the REWIND button to rewind the tape. When the ER (EIA) or % (ISO) code is read, the rewind stops. Then, push the REWIND button again to rewind the tape completely. Push the STOP button to stop the rewinding.
③ Change the Tape Reader switch to the RELEASE position.
④ Push the boss pawl inward, and pull the reel toward you.
⑤ Close the door.

4. OPERATION OF THE OPERATOR'S PANEL

4.1 Operator's Panel

The operator's panel varies in functioning and switch arrangement between the different tools. Operations of a typical operator's panel is explained. However, for details, refer to the manual issued by the machine manufacturer.

The explanation of the operator's panel for 3 axes control is described in this section. (For the operator's panel with 4th axis and 5th axis, infer from the explanation for 3 axes control similarly.)


HANDLE
HANDLE MULTIPLIER
×10 ×1
HANDLE AXIS SELECT
X Y Z
JOG & INCREMENTAL
+X +Y +Z
−X −Y −Z
RAPID
INCREMENTAL
×1 ×10 ×100 ×1000
JOG FEED RATE
mm/min.
0 2000
RAPID TRAVERSE OVERRIDE
LOW 10 50 100
FEED RATE OVERRIDE
0 100 200
MODE SELECT
MEMORY TAPE MDI HANDLE INC JOG EDIT
CYCLE START
FEED HOLD
SINGLE BLOCK
OFF ON
OPTIONAL BLOCK SKIP
OFF ON
DRY RUN
OFF ON
REFERENCE RETURN
OFF ON
X Y Z
MACHINE LOCK
OFF ON
MANUAL ABSOLUTE
OFF ON
DISPLAY LOCK
LOCK MODE
EMERGENCY STOP

4.2 Emergency Stop

This button is pushed to make a stop in case of emergency.

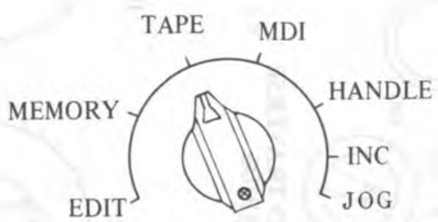
EMERGENCY STOP

If this button is pushed, the feed is stopped immediately, and the button remains latched in the stop position. The method of release varies with the machine tool builder.

Usually, it is released by turning clockwise with pushing.

(Note 1) If this button is pushed, the current to the mtoor is interrupted.
(Note 2) The control unit assumes a reset state.
(Note 3) Remove fault causes before the button is released.
(Note 4) After the button is released, reference point return by manual operation or G28 should be commanded.

4.3 Mode Select



MODE SELECT

This switch is used to specify an operational mode.

Mode	Function
EDIT	The following operations are available. (i) Registration of program to memory (ii) Modification, addition and deletion of program (iii) Punch-out of program in memory, and other program edit.
MEMORY	(i) The program stored in memory can be executed. (ii) The sequence number search of the program in memory can be executed.
TAPE	(i) Operation by commands on NC tape can be executed. (ii) Sequence number search of the program on NC tape can be executed.
MDI	(i) Manual data input can be performed via MDI & DPL panel.
HANDLE	(i) Handle feed can be executed.
INC	(i) Incremental feed can be executed.
JOG	(i) Jog feed can be executed.

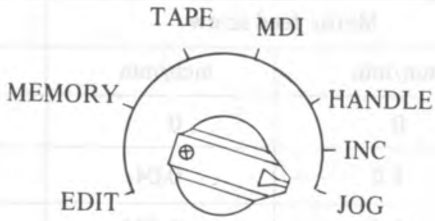
4.4 Operation Relative to Manual Operations

In addition to the automatic operation by programs, the following manual operations are possible, using the switches.

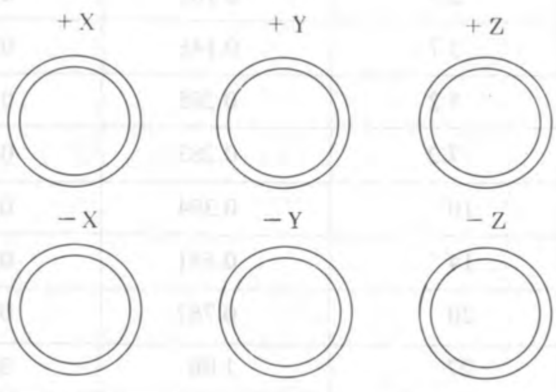
4.4.1 JOG feed

It is possible to move the machine tool.

(1) Set the MODE SELECT switch to JOG position



(2) Select the movable axis

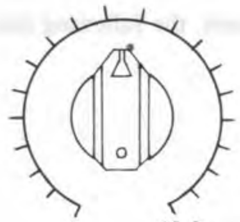


The tool is moved in the selected direction.

(Note 1) In manual operation, 2 axes can be controlled at a time.

(Note 2) Even if the MODE SELECT switch is set to the JOG position under an axis being selected, the machine does not move immediately after turning power on. In this case, an axis is required to be selected again.

(3) Select JOG feed rate



Position on rotary switch	Metric feed screw		Inch feed screw	
	mm/min	inch/min	inch/min	mm/min
0	0	0	0	0
1	1.0	0.04	0.02	0.508
2	1.4	0.055	0.028	0.711
3	2.0	0.079	0.04	1.02
4	2.7	0.106	0.054	1.37
5	3.7	0.146	0.074	1.88
6	5.2	0.205	0.104	2.64
7	7.2	0.283	0.144	3.66
8	10	0.394	0.2	5.08
9	14	0.551	0.28	7.11
10	20	0.787	0.40	10.2
11	27	1.06	0.54	13.7
12	37	1.46	0.74	18.8
13	52	2.05	1.04	26.4
14	72	2.83	1.44	36.6
15	100	3.94	2.00	50.8
16	140	5.51	2.80	71.1
17	200	7.87	4.00	102
18	270	10.6	5.40	137
19	370	14.6	7.40	188
20	520	20.5	10.40	264
21	720	28.3	14.40	366
22	1000	39.4	20.00	508
23	1400	55.1	28.00	711
24	2000	78.7	40.00	1016

(Note 1) The value varies between machine tool builders.
(Note 2) The feed rate error (about ±3%) affects on the feed rate in the table above.

(4) Rapid traverse

It is possible to move the axis in the selected direction at rapid traverse while this button is kept pushed.

RAPID TRAVERSE button

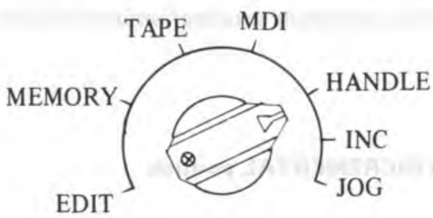
(Note 1) Feed rate, time constant and method of automatic acceleration/deceleration for manual rapid traverse are the same as G00 in programmed command.

(Note 2) When the stored stroke function is provided, the feedrate does not become the rapid traverse rate but remains the jog feedrate even when the RAPID TRAVERSE button is pressed as long as the reference point return is not performed after turning the power on or an emergency stop on the axes with the reference point return function. This is to prevent movement along axes from reaching the stroke end at the rapid traverse rate since the stored stroke limit does not become effective until the manual reference point return is made.

4.4.2 HANDLE feed

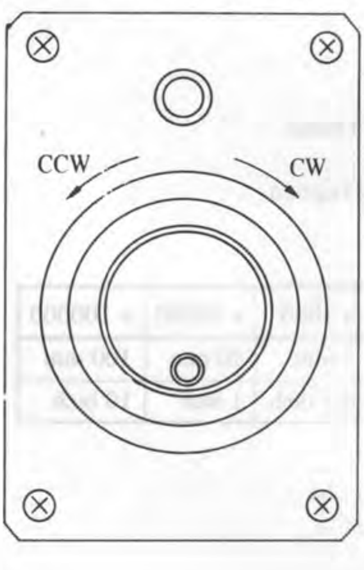
By the use of a manual pulse generator, fine adjustment feed is available for the machine tool.

(1) Set MODE SELECT switch to HANDLE position.



(2) Select the movable axis.

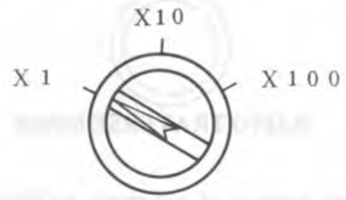
(3) Rotate the handle of manual pulse generator.



Clockwise rotation . . . . + direction
Counterclockwise rotation . . . . – direction
(The direction varies with machine tool builder.)

(4) Movement amount

In some cases, however, the operator's panel is provided with the following selector switch. X10 multiplies the movement amount by 10; X100 by 100.



HANDLE MULTIPLIER

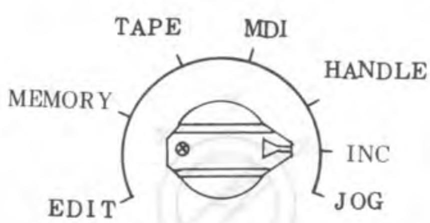
Input system	Movement amount per degree		
	× 1	× 10	× 100
Metric input	0.001 mm	0.01 mm	0.1 mm
Inch input	0.0001 inch	0.001 inch	0.01 inch

(Note 1) If the handle is rotated in excess of 5 rotation/sec, there arises a difference between handle rotation amount and machine movement distance. Namely, don't rotate the handle too rapid.

(Note 2) Rotating the handle too speedily with X100 selected moves the tool or table at rate as speedy as rapid traverse. A sudden stop gives a shock to the machine tool. The Automatic Acceleration/Deceleration option provided applies automatic acceleration/deceleration to manual handle feed also, reducing a mechanical shock.

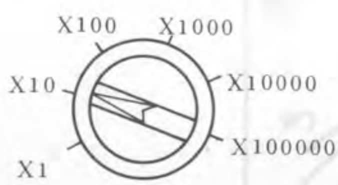
4.4.3 INCREMENTAL feed

(1) Set MODE SELECT switch to INCREMENTAL position.



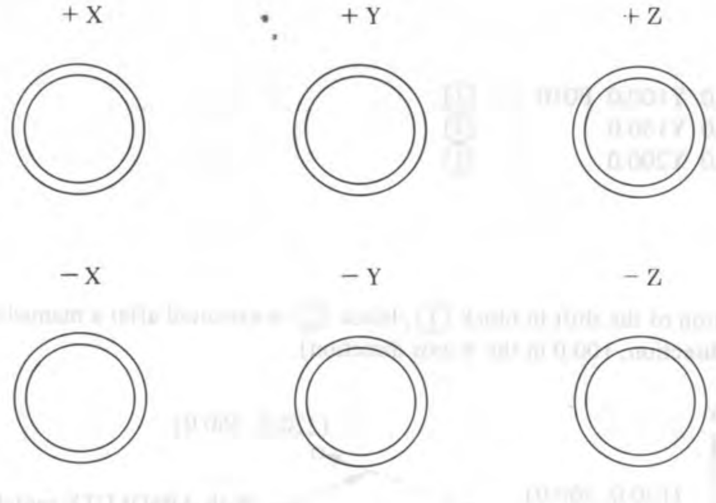
INCREMENTAL feed is available for the machine without manual pulse generator.

(2) Select movement amount



Input system	× 1	× 10	× 100	× 1000	× 10000	× 100000
Metric input	0.001 mm	0.01 mm	0.1 mm	1 mm	10 mm	100 mm
Inch input	0.0001 inch	0.001 inch	0.01 inch	0.1 inch	1 inch	10 inch

(3) Select the movable axis



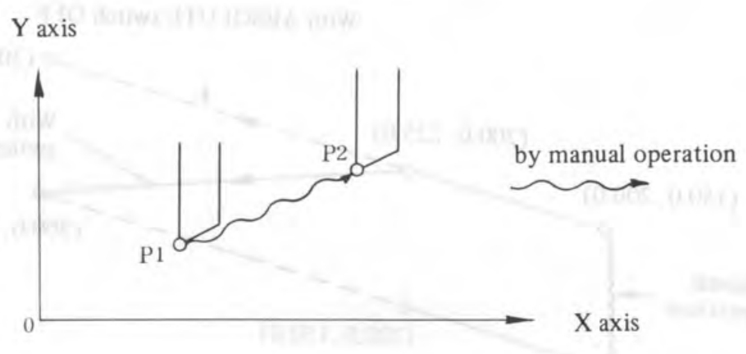
The selected axis is moved for a specified distance by pushing the axis selection switch once.

(Note 1) Incremental feed rate is same as JOG feed rate.
(Note 2) Rapid traverse button is also in effect. Rapid traverse override is in effect in rapid traverse.
(Note 3) Depending on the machine tool, in some cases all of X1 to X100000 are unusable.

4.4.4 Manual absolute

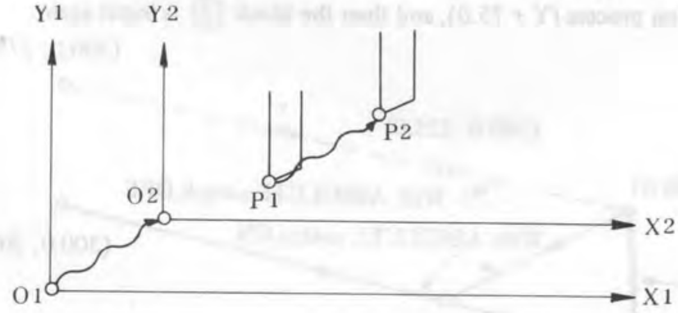
If this switch is set to ON, the movement amount in a manual operation is added to an coordinate value.

(1) Manual absolute switch ON



The coordinate value varies the value of manual operation.

(2) Manual absolute switch OFF

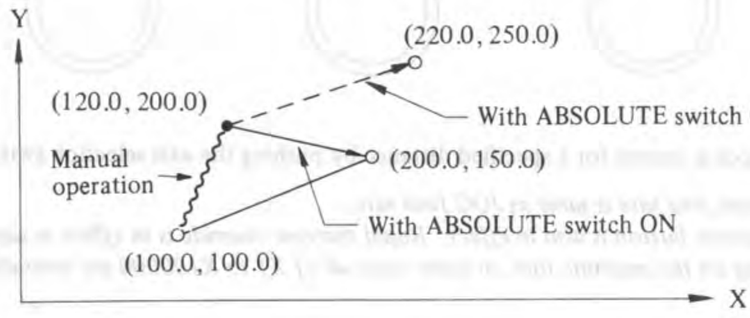


The coordinate value doesn't vary.

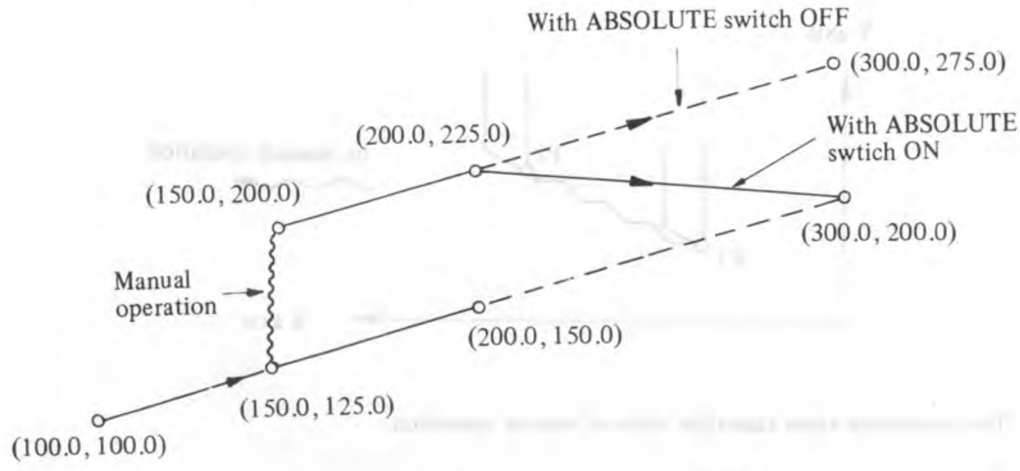
(Example) Suppose the next tape commands:

```
.
.
.
G01 G91 X100.0 Y100.0 F010 ;   ①
        X200.0 Y150.0      ;   ②
        X300.0 Y200.0      ;   ③
.
.
.
```

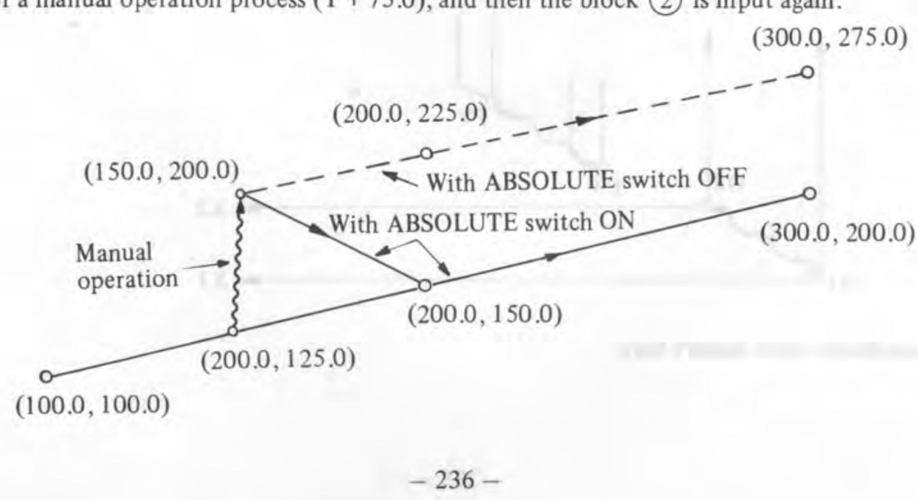
(a) Upon completion of the shift in block ①, block ② is executed after a manual operation process (20.0 in the X-axis direction, 100.0 in the Y-axis direction).



(b) The FEED HOLD button is pushed during the execution of block ②, and the CYCLE START button is again pushed to release the hold state after a manual operation process (Y + 75.0).



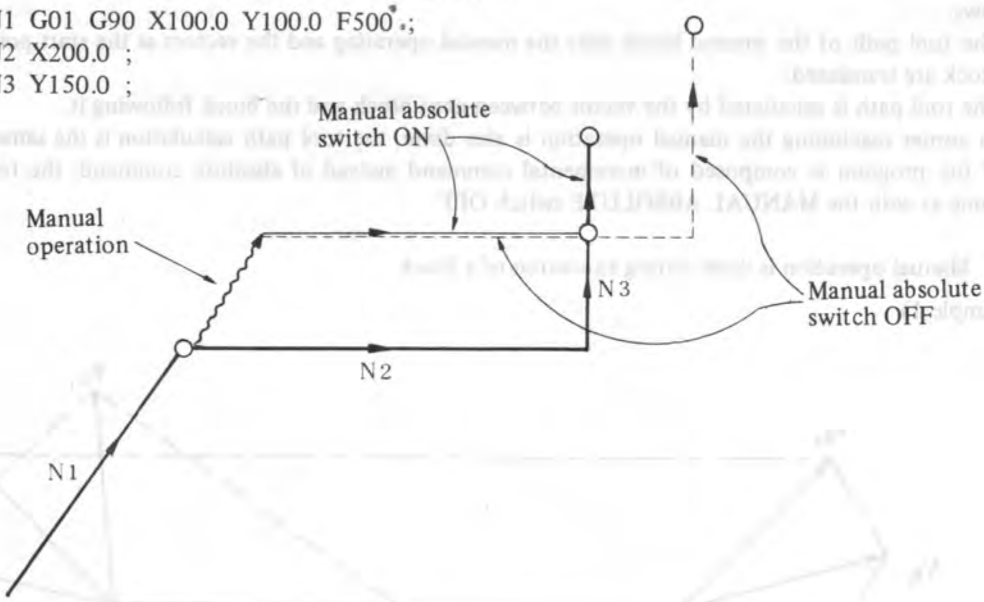
(c) The FEED HOLD button is pushed during the execution of block ②, and a resetting action is taken, after a manual operation process (Y + 75.0), and then the block ② is input again.



240

(d) When the command following manual operation is one axis command, only the commanded axis returns to the programmed absolute position of the axis.

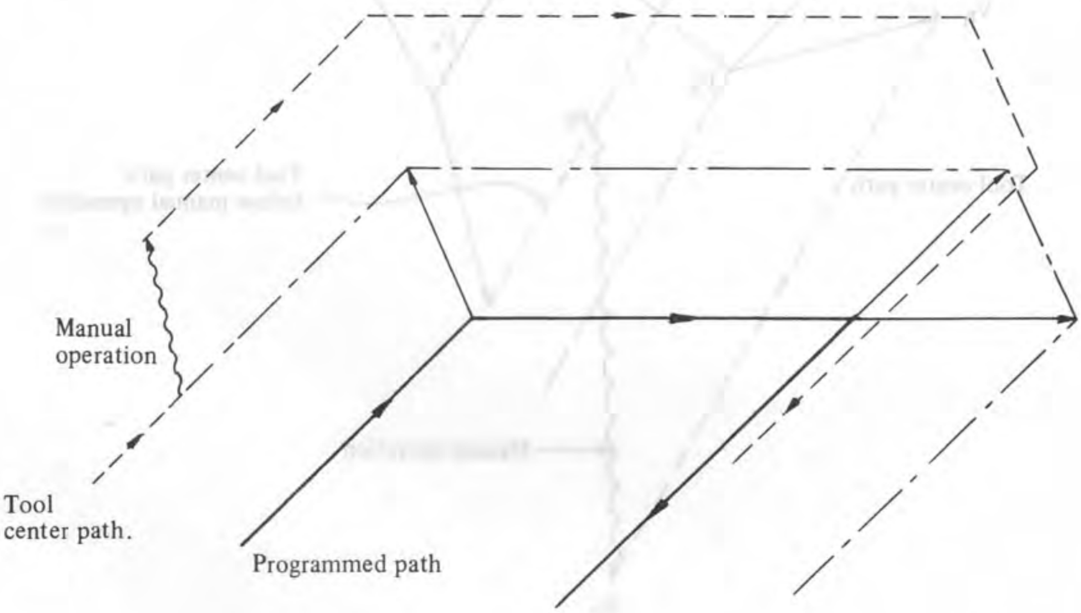
N1 G01 G90 X100.0 Y100.0 F500 ;
N2 X200.0 ;
N3 Y150.0 ;



(e) When the command following the manual operation is incremental command, the axis moves to the same position as when commanded with manual absolute switch OFF.

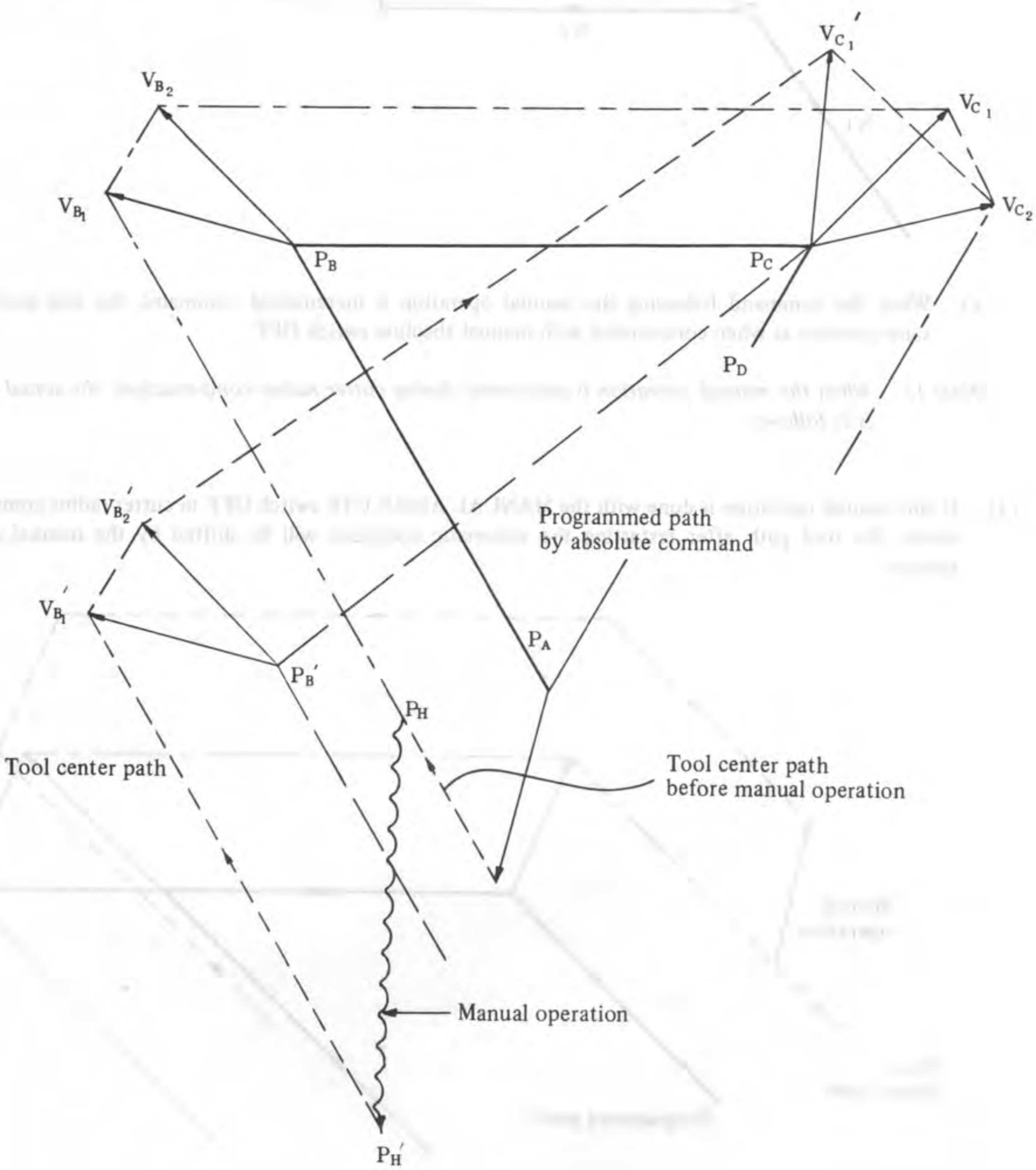
(Note 1) When the manual operation is performed during cutter radius compensation, the actual tool path is as follows:

(1) If the manual operation is done with the MANUAL ABSOLUTE switch OFF in cutter radius compensation mode, the tool path after restarting the automatic operation will be shifted by the manual operation amount.



(2) If the manual operation is performed with the MANUAL ABSOLUTE switch ON in the cutter radius compensation C mode, the tool path after restarting the automatic operation by absolute command is as follows:
The tool path of the present block after the manual operating and the vectors at the start point of the next block are translated.
The tool path is calculated by the vector between next block and the block following it.
In corner machining the manual operation is also done, the tool path calculation is the same as the above.
If the program is composed of incremental command instead of absolute command, the tool path is the same as with the MANUAL ABSOLUTE switch OFF.

(a) Manual operation is done during execution of a block
(Example 1)



246

In the above programmed path ($P_A \rightarrow P_B \rightarrow P_C \rightarrow P_D$), suppose that the tool is translated from P_H between P_A and P_B to $P_{H'}$ by manual operation (after pushing the feed hold button).

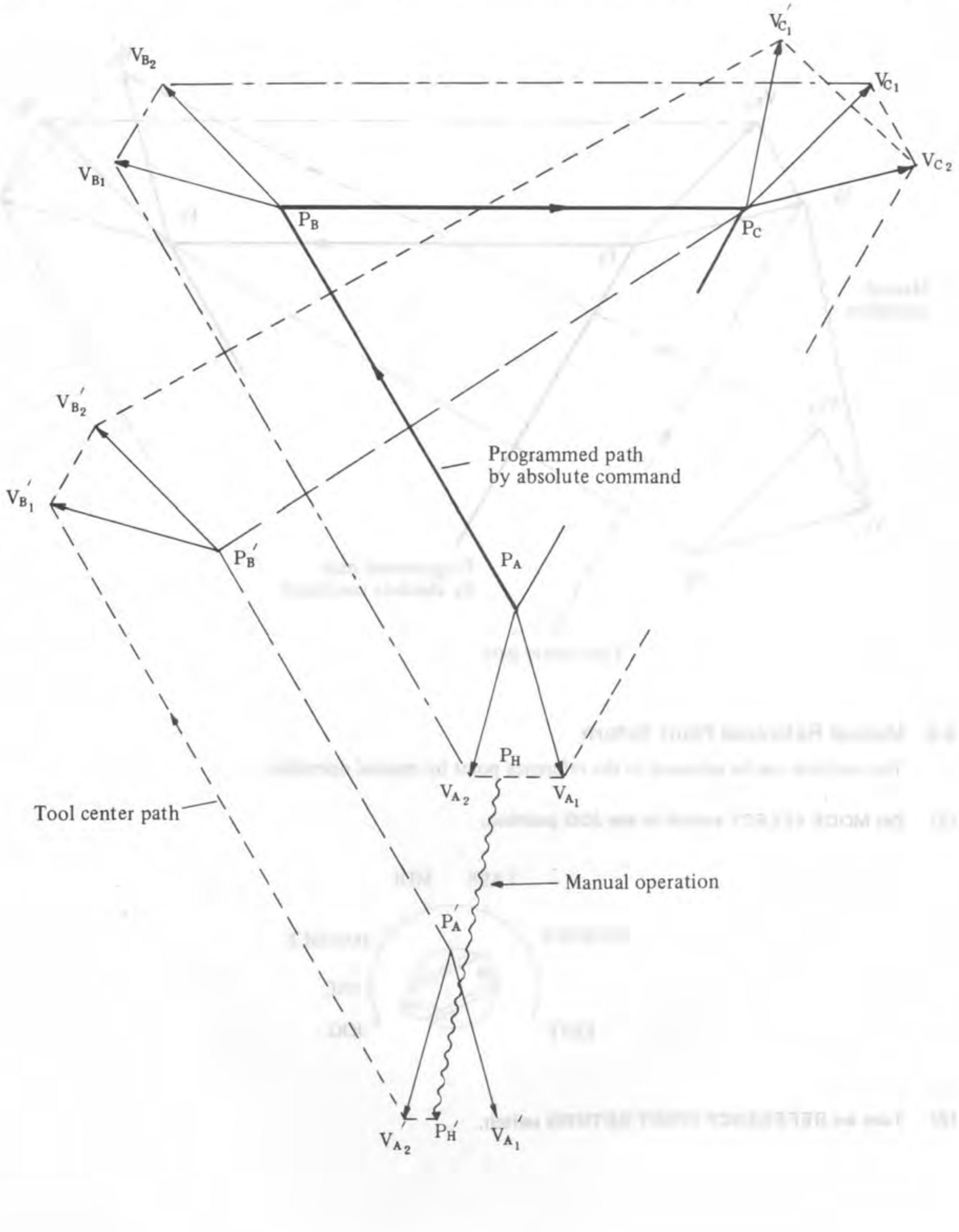
The end point P_B of the present block is translated to the point $P_{B'}$ for the manual operation amount and the vectors (V_{B1}, V_{B2}) on P_B is also translated. (Translated vectors are $V_{B1'}$ and $V_{B2'}$ respectively).

The vectors between the next block (tool path from P_B to P_C) and the following block (from P_C to P_D) need not to be compensated. The new vectors ($V_{C1'}, V_{C2'}$) for compensation are formed by the relation between two blocks (programmed path from $P_{B'}$ to P_C and from P_C to P_D).

However, as the vector $V_{B2'}$ is the same as V_{B2}, the tool offset is not accurately performed at the tool path between $P_{B'}$ and P_C.

For the blocks after point P_C, the tool offset is accurately performed.

(Example 2)



In cutter radius compensation, if during corner machining the manual operation is performed, the tool path after that is determined in the same way as example 1.

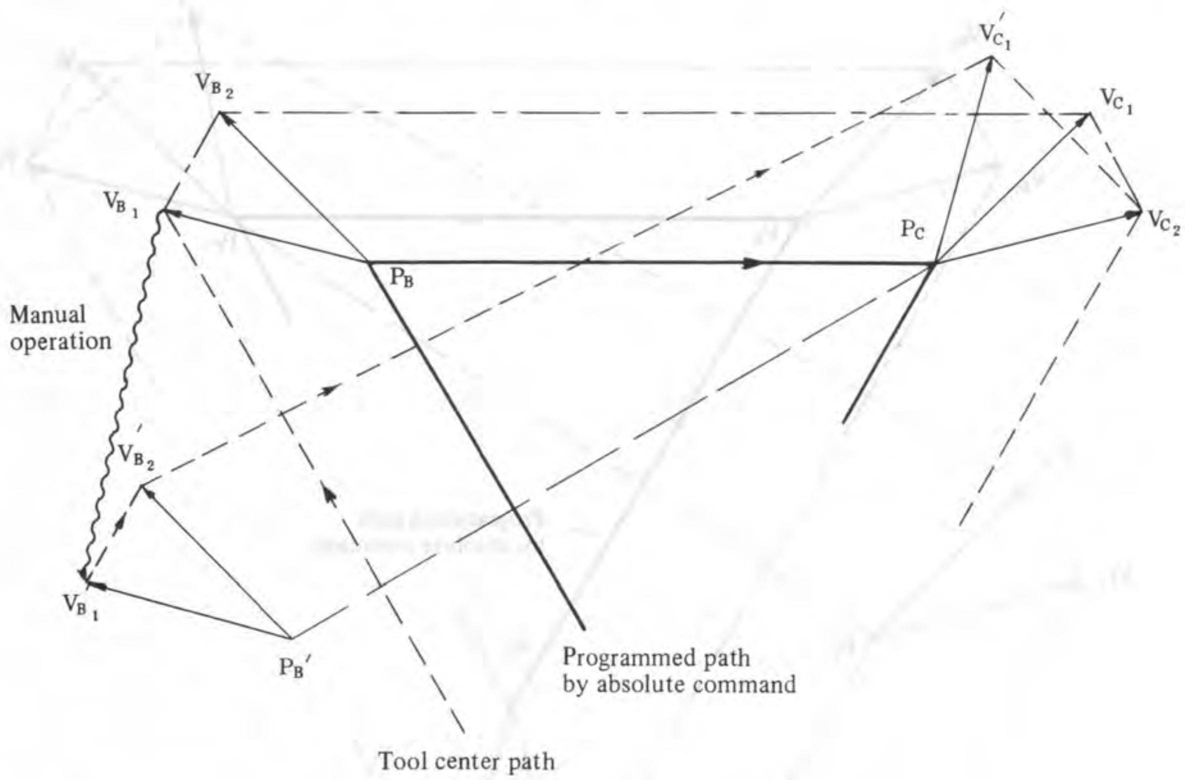
That is, the vectors V_{A2}', V_{B1}' and V_{B2}' in the above figure is calculated by translating the vectors V_{A2}, V_{B1} and V_{B2} for the manual operation amount respectively. And new vectors are calculated by the vectors V_{C1}' and V_{C2}'. For the blocks after point P_C, the cutter radius compensation is accurately performed.

(b) When the manual operation is done after execution by the single block function.

The vectors V_{B1} and V_{B2} at the end point of the present block are translated.

The tool path after that is determined in the same way as item (a).

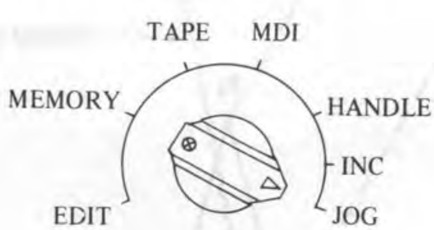
After one block execution by the single block function, MDI operation can be performed. The tool path after the operation is the same as that after manual operation.



4.5 Manual Reference Point Return

The machine can be returned to the reference point by manual operation.

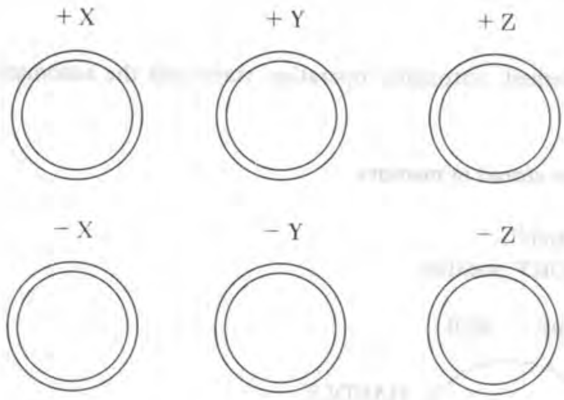
(1) Set MODE SELECT switch to the JOG position.



(2) Turn on REFERENCE POINT RETURN switch.

REFERENCE POINT RETURN switch

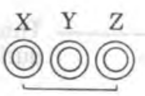
(3) Each axis is moved to the reference point at jog feed.



Push the desired button, until the tool is positioned at the reference point.

The machine moves to the decelerating point at rapid traverse, and then to the reference point at FL speed. Rapid traverse override is also effective during rapid traverse movement.

(4) The machine stops at the reference point with the REFERENCE POINT RETURN COMPLETION LED lighting.



REFERENCE POINT RETURN
COMPLETION LED

(Note 1) After reference point return completion and the LED lights, the machine cannot move by JOG mode if REFERENCE POINT RETURN switch is set to ON.

(Note 2) REFERENCE POINT RETURN COMPLETION LED is extinguished by the following operations.
(1) Moving from the reference point.
(2) Depressing the emergency stop button.

(Note 3) For the distance of return to the reference point, refer to the manual issued by the machine tool builders.

4.6 Operation Relative to Automatic Operation

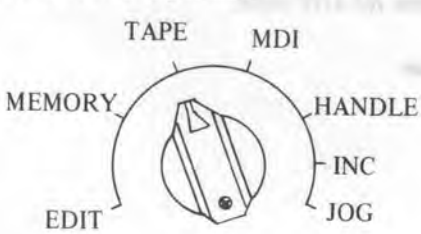
The machine tool can be moved according to programming by automatic operation.

4.6.1 Start of automatic operation

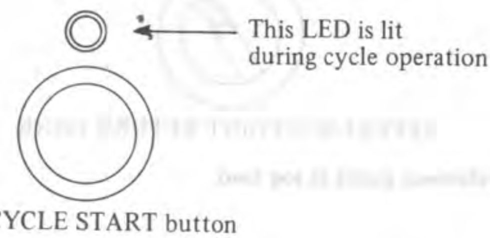
(1) Start of tape operation

(a) Set the program tape to the Tape Reader.
(For the details, refer to item 3.3 in IV OPERATION)

(b) Set the MODE SELECT switch to TAPE position.



(c) Push the CYCLE START button.

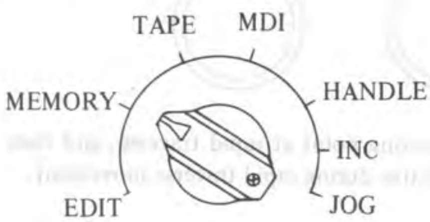


When the CYCLE START button is pushed, automatic operation starts and the automatic operation LED lights up.

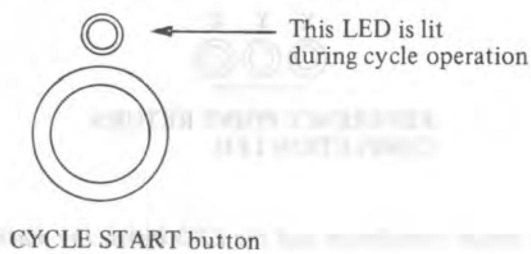
(2) Procedure to start operation from a program stored in memory

(a) Select the program number
Refer to item 5.16 "Program number search".

(b) Set the MODE SELECT switch to MEMORY position.



(c) Push CYCLE START button.



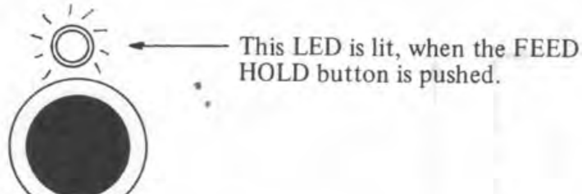
When the CYCLE START button is pushed, automatic operation starts and CYCLE START LED is lit.

(Note 1) When the CYCLE START button is pushed in the Edit mode, a program is read to be registered as in the same way as the READ *button is pushed, by parameter setting.*

(Note 2) In the following cases, the CYCLE START button is ignored.
(a) When FEED HOLD button is pushed.
(b) When EMERGENCY STOP button is pushed.
(c) When RESET signal is turned ON. (For details, contact the machine tool builder.)
(d) When MODE SELECT switch is set to a wrong position. (Other than TAPE, MEMORY, or EDIT mode)
(e) When sequence number is being searched.
(f) When an alarm has occurred.
(g) When automatic operation is selected.
(h) When NC doesn't become READY state.

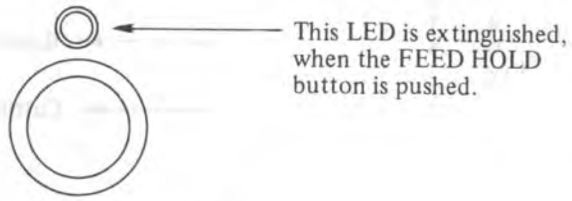
4.6.2 Halting the automatic operation

(1) Push FEED HOLD button.



FEED HOLD button

When FEED HOLD button is pushed, the FEED HOLD LED lights and the CYCLE START LED is extinguished.



CYCLE START button

(a) The feed is stopped after deceleration if the machine tool is moving.
(b) Dwell is not continued even in a feed hold state if the dwell is being executed.
(c) The machine tool is stopped after the operation of the M, S, T or B function is executed.

4.6.3 SINGLE BLOCK

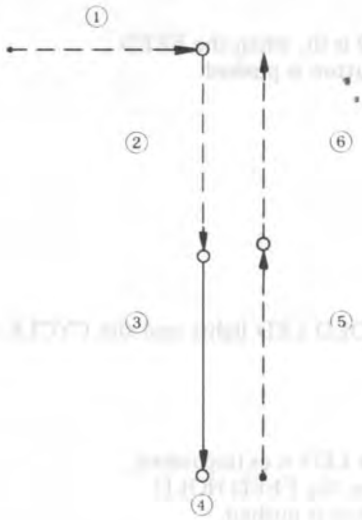
When this switch is turned on, the control executes only one block of tape information each time the cycle start button is depressed.

(1) Turn on SINGLE BLOCK switch.

SINGLE BLOCK switch

The control is stopped after only one block execution. When the CYCLE START button is pushed, the control is stopped after next block execution.

(Note 1) *In G28, G29, or G30 mode, the tool is stopped by single block function at the intermediate point.*
(Note 2) *In canned cycle machining, if the single block function is effective, the tool is stopped at the end point of cycle paths 1, 2 or 6 of canned cycle (See figure below). The FEED HOLD LED lights at the each stop of cycle paths 1 and 2.*
(Note 3) *Single block stop is ineffectife for M98P_____ ; , M99 ; , and block of G65, G66 or G67. However, if the address other than O, N, L and P is commanded in the block of M98 or M99, it is effective.*



(Note) When the repetitive count of canned cycle is not 1, FEED HOLD LED lights at each stop of operation, except for at ⑥ in last cycle.

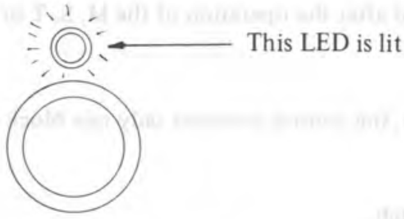
– – – → Rapid traverse

——→ Cutting feed

4.6.4 Restart

(1) Return the mode switch to the desired operating mode (TAPE or MEMORY mode)

(2) Push CYCLE START button.

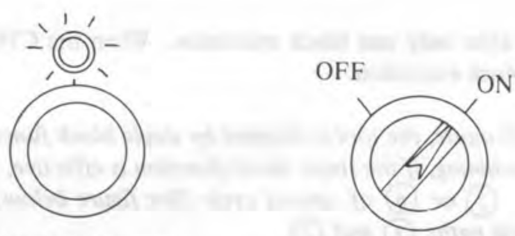


CYCLE START button

When CYCLE START button is pushed, the FEED HOLD LED is extinguished.

4.6.5 Manual operation during automatic operation

(1) Push the FEED HOLD button on the operator's panel, or set the SINGLE BLOCK switch to ON, to make a temporary stop.

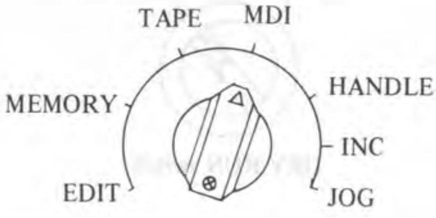


FEED HOLD button SINGLE BLOCK button

(2) Record the coordinate value of the stop position from the position display unit.
(3) Perform the manual operation (see item 4.4.4).
(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 MODE SELECT switch to the position it was at prior to the manual operation.
(6) Push the CYCLE START button.

4.6.6 MDI operation during automatic operation

(1) Set the SINGLE BLOCK switch to ON.
(2) Set the MODE SELECT switch to MDI position.

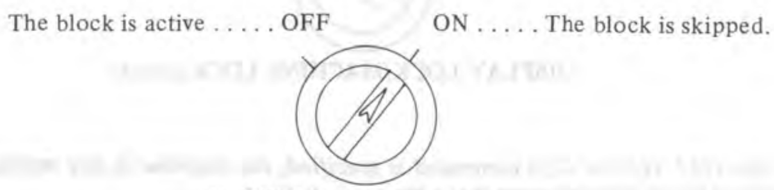


(3) Perform the operation from the Manual Data Input unit.
(4) In order to resume the cycle operation, set the MODE SELECT switch to the prior mode, and push the CYCLE START button on the operator's panel.

(Note 1) Modal data in cycle operation remain and they affect the MDI operation.
(Note 2) Modal data commanded from the MDI unit is effective to the automatic operation after the MDI operation.
(Note 3) Cutter radius compensation C is not performed in MDI operation.
(Note 4) In feed hold status, MDI operation cannot be performed.

4.6.7 Optional block skip

This is the function which allows the control to skip a block of information in which a slash "/n" (n=1 to 9) is punched, as the first character in the block.
Optional block skip switches are provided for each number "n".

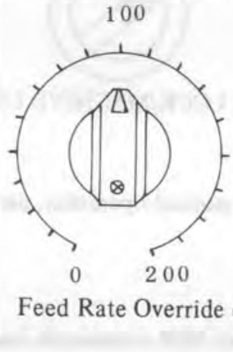


OPTIONAL BLOCK SKIP switch

(Note) The optional block skip function is identified when the block is read into the buffer storage from the paper tape or memory. Therefore, this function is not effective to the block which has been read into the buffer storage.

4.6.8 Feed rate override

An override in increments of 10% from 10 to 200% can be provided for the feed rate specified by the F function.



Feed Rate Override dial

4.6.9 Dry run

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 the machine tool is moved at the following speed.

DRY RUN switch

Rapid traverse button ON/OFF	Rapid traverse	Cutting feed
Rapid traverse button ON	Rapid traverse	Max. JOG feed rate
Rapid traverse button OFF	JOG feed rate (Note 1)	JOG feed rate

(Note 1) Dry run for rapid traverse can be made effective or ineffective by the parameter (RDRN) setting.

4.6.10 Machine lock

With the switch at MACHINE LOCK, move command pulses are suppressed. Consequently, by Cycle Operation or manual operation, the position display is updated as specified, but the machine tool does not move. This function is used for checking a program.

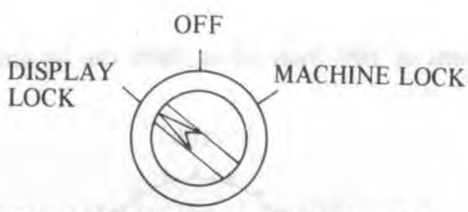
DISPLAY LOCK/MACHINE LOCK switch

(Note 1) When the G27, G28 or G30 command is specified, the machine is not moved to reference point and the REFERENCE POINT RETURN LED is not lighted.

(Note 2) The M, S, T and B functions are executed.

4.6.11 Display lock

With the switch at DISPLAY LOCK, pulses are stopped to the position display unit.



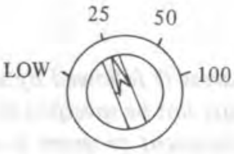
DISPLAY LOCK/MACHINE LOCK switch

When shifting a coordinate system by manual operation, use this switch so as not to enter that shift amount into the display.

4.6.12 Mirror image

The sign of the X, Y or 4th axis tape or MDI commands can be reversed in cycle operation. The movement in manual operation and between the intermediate point and reference point in automatic reference point return are not reversed with this function. Position display depends on the actual tool movement. This function can be performed by using setting parameters via the MDI unit. (See item IV.5.7)

4.6.13 Rapid traverse override



The optional rapid traverse override switch of 100%, 50%, 25%, and Fo can be provided with the machine operator's panel. When the feed rate is 10 m/min and this switch is set to the position of 50%, the actual feed rate becomes 5 m/min. "Fo" is the fixed value (feed rate) specified by each machine tool builder. This function is available in the following cases.

(1) Rapid traverse by G00
(2) Rapid traverse during canned cycle
(3) Rapid traverse in G27, G28, G29 and G30
(4) Manual rapid traverse
(5) Rapid traverse in manual reference point return

4.6.14 External work number search function

Selecting the number of a work to be machined by the switch (Not appeared in the example of operator's panel in this manual) on the machine control panel, with machining programs stored previously in part program memory and pressing the start button, automatically executes the program corresponding to that work number. Use of this function rids the operator of tape replacement and of searching the memory for the program number, helping reduce idle time and operating errors.

(1) Program preparation

To use this function, a program number must be named corresponding to the number of a work to be machined. Namely, with an arbitrary number of 01-31 assigned to each work to be machined, prepare programs of program numbers each of which is written as

O(:) ○ ○ □ □ (O for EIA, : for ISO)
- □ □ — Work number (01-31)
- ○ ○ — arbitrary work number

And memorize them into the part program memory. Each program must start with address O followed by a program number and end with M02, M30 or M99 as shown below.

Also programs not related to work numbers are allowed to be stored.

```
O1001 ;
N001G00 . . . . . . . . . . ;
-----------
-----------
M120M02 ;
```
A program for work number 01

```
O2102 ;
N001G00 . . . . . . . . . . ;
-----------
-----------
N300M30 ;
```
A program for work number 02

```
O4504 ;
N001G00 . . . . . . . . . . ;
-----------
-----------
N080M02 ;
```
A program for work number 04

```
06247 ;
N001G00 . . . . . . . . . . ;           A program not related to work numbers
------------
------------
N034M99 ;
```

(Note 1) Each program must start with address O followed by a program number and end with M02, M30 or M99. But M02, M30 and M99 must not be specified in the halfway of a program. If either of them is specified in the halfway, the subsequent program is assumed as another program. (The sequence number of the block immediately after M02, M30 or M99 is assumed the program number at the time of storing the program into memory.)

(Note 2) Whether how many work numbers are allowed depends on the machine tool builders. Refer to the manual issued from them.

(Note 3) When your machine tool builder has an external work number search function A, the maximum work number allowed is 31. In this case upper two digits of the program numbers which correspond to work numbers must be 00.

(2) Operation

Operation differs from machine tool builder to machine tool builder. Hear describes the standard operation. You must follow the operation which is mentioned by your machine tool builder.

(a) Prepare a tape or a number of tapes programmed in the format mentioned in the previous item and store it or them into the memory. Either of one tape having several programs on it or a number of tapes each of which has one program will do. (Refer to the section on the part program storage & editing function for storing into the memory.)

(b) Set the mode selector switch to MEMORY, then the number (01-31) corresponding to a work to be machined onto the rotary switch on the control panel, and press the start button. The program corresponding to the work number will be executed.

(c) If the start button is pressed after the work number has been set as 00, no program is searched for but the execution starts from the then executable portion. To start from the halfway of a program or to execute a program not related to the work number, press the start button with the work number set 00 after the sequence number search or the program number search.

(Note 4) This function cannot be used in Tape operation and in MDI operation. It can be used in Memory operation only.

(Note 5) When there is no program number corresponding to a work number on the rotary switch in the memory, the alarm (No. 59) will be generated when the start button is pressed.

(Note 6) Even if a workpiece number is selected on the dial, the corresponding program is not always selected. Refer to the manual issued by your machine tool builder for the operation used to select the program. When workpiece number search function A is selected, the NC searches for a program at the start of cycle operation in the reset status.

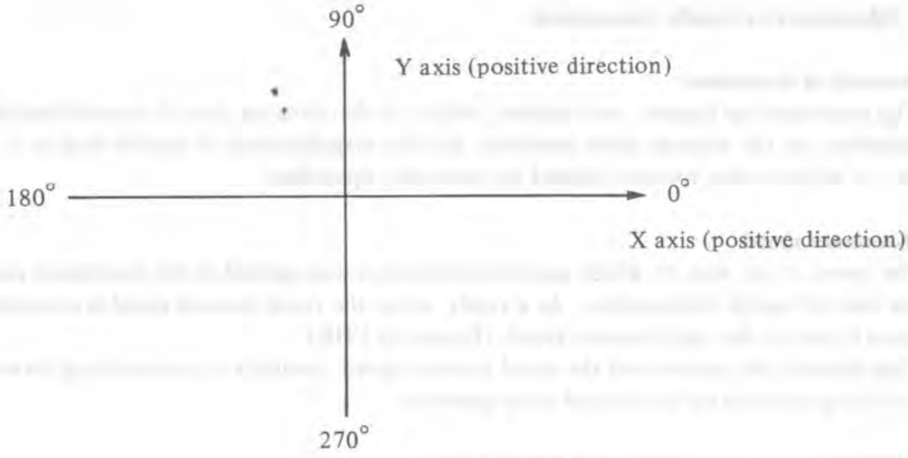
4.7 Manual Arbitrary Angle Feed Function

The machine can be fed manually in any desired angle direction by selecting the angle and feedrate in X-Y plane and pressing start button.

(1) Set mode selector switch to manual arbitrary angle feed mode.

(2) Set an angle with angle selector dial.

An angle is selected among 0 to 71 in BCD codes; 0 to 71 corresponds to 0 to 355° (in increments 5°). Turn on angle strobe signal. If angle strobe signal is remained off, previously selected angle is effected.



Angle 0 is X-axis positive direction; angle 90° is Y axis positive direction as shown in the figure.

(3) Select a feedrate (tangential rate) with the jog feedrate dial.

(4) Press manual arbitrary angle feed start button. The machine will move in the selected angle at the selected feedrate.

If you press the manual rapid traverse button, the machine will be fed at the maximum jog feedrate. While the manual arbitrary angle feed start button is on, the machine is fed but when it is turned off, the machine will stop.

(Note 1) If interlock is applied to either X or Y axis, both axes will stop after deceleration. They will start movemement after interlock is released.

(Note 2) It is possible to intervene manual arbitrary angle feed during cycle operation after stopping the machine with feed hold.

(Note 3) If external deceleration option is equipped, it can be applied to manual arbitrary angle feed. In this case tangential speed becomes equal to the external deceleration speed.

(Note 4) Automatic acceleration and deceleration for feed is applied to manual arbitrary angle feed.

(Note 5) Even if inch metric conversion is made, the manual arbitrary angle feedrate does not change as well as jog feedrate.

4.8 Manual Handle Interruption Function

As regards a specific axis (which is fixed by a parameter) during automatic operation, handle movement can be performed in addition to its automatic movement.

4.8.1 Handle interruption operation

Handle interruption can be performed by turning the manual pulse generator in the following situations.

(1) Mode

TAPE mode, MEMORY mode or MDI mode

(2) Operating state

Handle interruption can be performed during linear, circular interpolation, helical interpolation or sine interpolation except following cases.

(i) Alarm
(ii) Start lock on
(iii) Movement by positioning
(iv) Interlock (when in effect)
(v) There is no move command.

(3) Handle axis selection signal

The handle axis selection signal (HX, HY, HZ, H4, H5) is on (the contact is closed) in the axis in which an attempt has been made to perform handle interruption.

4.8.2 Movement by handle interruption

(1) Amount of movement

The movement by handle interruption, which is the same as that of manual handle feed, is determined by the graduation on the manual pulse generator and the magnification of handle feed (x 1, x 10, and x 100). This movement is added to the amount realized by automatic operation.

(2) Movement speed

The speed in an axis to which handle interruption was applied is the movement speed by automatic operation plus that of handle interruption. As a result, when the rapid traverse speed is exceeded in that axis, the movement speed is tied to the rapid traverse speed. (Parameter CHR)

This discards the excess over the rapid traverse speed, resulting in mismatching between the amount of movement and the graduation on the manual pulse generator.

(3) Relation between movement and signals below

Signal	Movement
Machine lock	Affected, i.e., the tool will not move when machine lock is on.
Display lock	Affected, i.e., the relative coordinate value will not change when display lock is on.
X-axis mirror image	Not affected, i.e., the handle movement is made in the positive direction by rotating the handle in the pluse direction.

(4) Relation between movement and displays below.

Display	Movement
Absolute coordinate value	Not affected, i.e., pulses are not added to the absolute coordinate value by handle interruption.
Relative coordinate value	Added, i.e., pulses are added to the relative coordinate value.
Machine coordinate value	Added, i.e., pulses are added to the machine coordinate value.

(5) Display of amount of movement

Movement by handle interruption can be displayed by Diagnose (DGNOS).

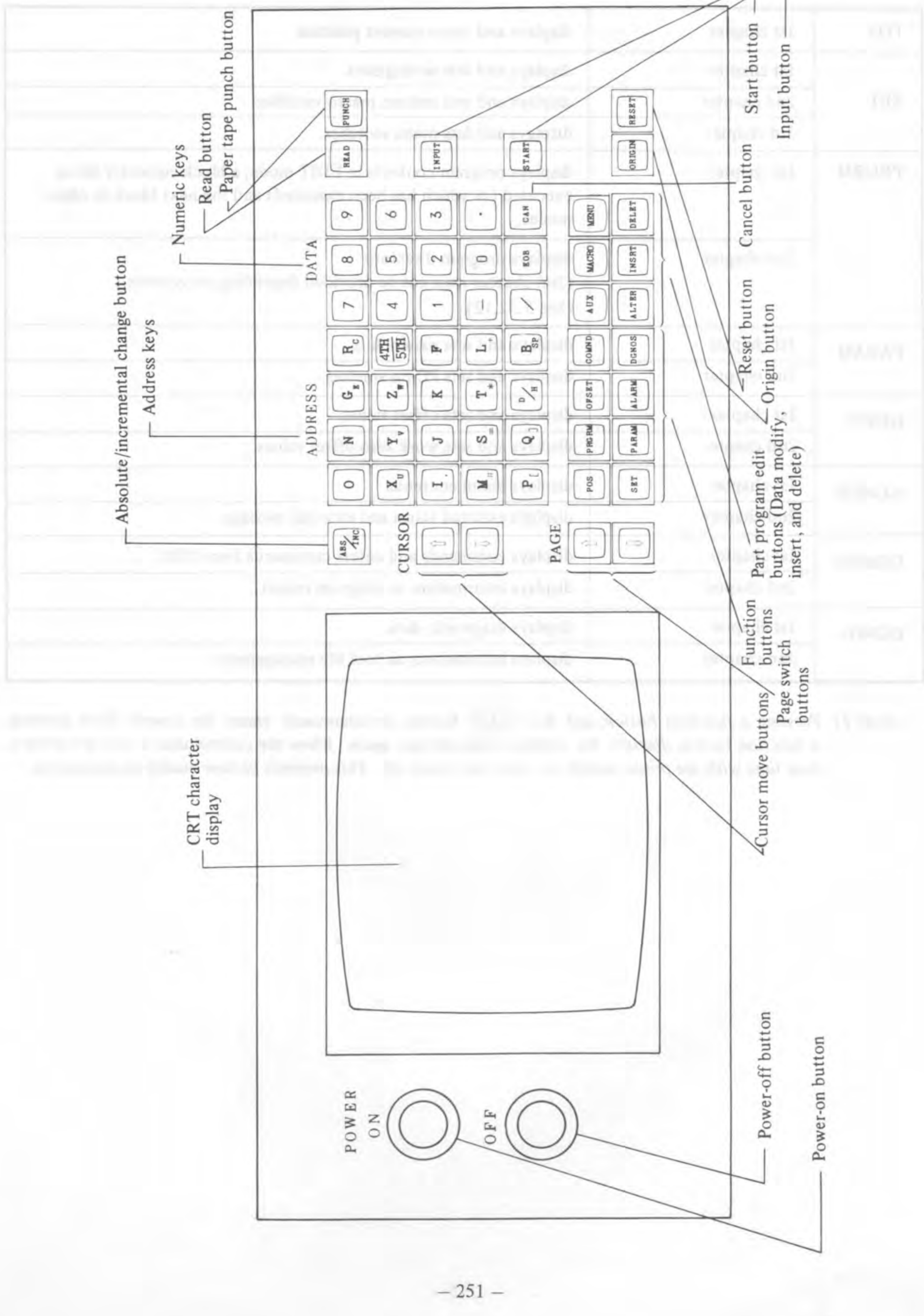
	8	0	5	Movement in X by handle interruption
	8	0	6	Movement in Y by handle interruption
	8	0	7	Movement in Z by handle interruption
	8	0	8	Movement in 4th by handle interruption
	8	0	9	Movement in 5th by handle interruption

Units: 0.001 mm (mm input)
0.0001 inch (inch input)

Note: Movement can be cleared by a clear operation.

5. DISPLAY AND OPERATION ON THE CRT CHARACTER DISPLAYED MDI & DPL PANEL

The MDI & DPL panel, usually attached at the top of the front side of the control unit, consists of a CRT display and buttons as in the following figure.



Function buttons

Function buttons indicate a large item such as a chapter in a book.

When you press a function button twice or third time, 2nd and 3rd chapter (if they exist) of the function is respectively displayed.

A chapter includes several pages and each of them is selected by PAGE button.

The following table shows a list of each function botton and its functions.

POS	1st chapter	displays and resets current position.
SET	1st chapter	displays and sets setting data.
	2nd chapter	displays and sets custom macro variables.
	3rd chapter	displays and sets menu switches.
PRGRM	1st chapter	displays program contents in EDIT mode; a block currentry being executed (or which has been executed) and the next block in other modes.
	2nd chapter	displays program directory (2nd chapter may not be provided depending on systems.) (See 5.32.12)
PARAM	1st chapter	displays and sets parameters.
	2nd chapter	displays and sets PC parameters.
OFSET	1st chapter	displays and sets offset values.
	2nd chapter	displays and sets work zero offset values.
ALARM	1st chapter	displays alarm contents.
	2nd chapter	displays external alarm and external message.
COMND	1st chapter	displays commands and enters commands from MDI.
	2nd chapter	displays information on program restart.
DGNOS	1st chapter	displays diagnostic data.
	2nd chapter	displays information on tool life management.

(Note 1) Pressing a function button and the CAN *button simultaneously erases the screen. Then pressing a function button displays the corresponding picture again. When the control unit is not in use for a long time with the power supply on, turn the screen off. This prevents picture quality deterioration.*

5.1 Status Indication

The status of the system is indicated at bottom right of the screen.

The following indications are displayed.

NOT READY	Indicates that the control unit or the servo system is not ready for operation.
LSK	Indicates the Label Skip status, produced when the power is turned on or when the control unit is reset in other than MDI mode.
BUF	Indicates that a block has been read in, but not yet executed. The block not executed yet disappears when a reset is executed in other than MDI mode.
ABS	Indicates that commands from MDI are of incremental. Pushing [ABS/INC] makes the INC status which is mentioned next.
INC	Indicates that commands from MDI are of incremental. Pushing [ABS/INC] makes the system ABS status which is mentioned above.
ALM	Indicates that an alarm was generated, the type is displayed by pressing the [ALARM] button (this indication is flickering).
EDIT	Indicates that editing is being performed. Stop the editing operation while this indication is on.
SRCH	Indicates that a Sequence Number Search is being performed (this indication flickers).
RESTR	Indicates a period from a Program Restart is commanded until the last axis is returned. (This indication flickers.)

5.2 Keyed Data Display

Input from an address key or from a numeric key is displayed at the bottom left.

Data cannot be keyed in when a picture displayed by pressing the POS or ALARM button is displayed.

If D/H button is pressed twice, address H is selected.

When not editing a program, only one word consisting of an address and a number can be keyed in. Pressing the CAN key cancels the one word.

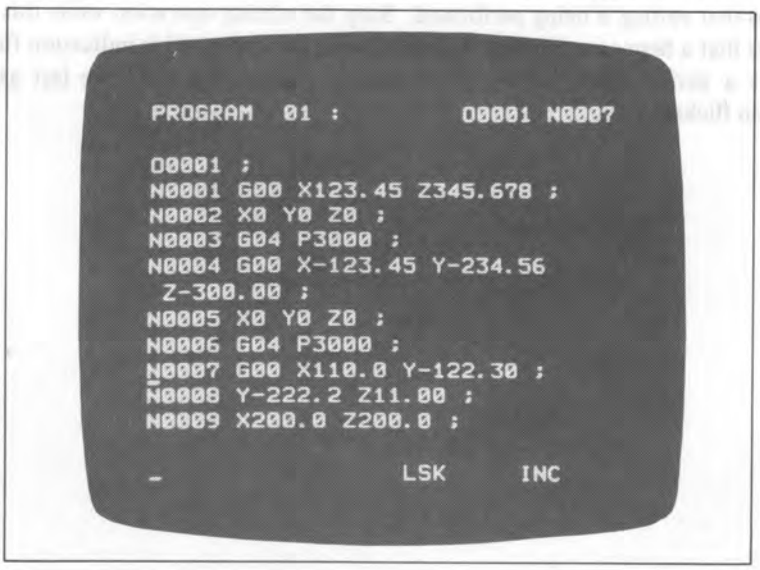
In program editing, one word, several words, one block, and any character string up to 32 characters can be keyed in.

Pressing the CAN key cancels the last character of keyed-in data. If the CAN key is pressed continuously keyed-in characters are canceled one after another.

(Note 1) Pressing the PRGRM *button in the edit mode enables program editing.*

5.3 Program Number and Sequence Number Display

A program number and a sequence number are displayed at the top, as seen in the photo.



262

Indications of program and sequence numbers are as follows.

Mode	Operation	Displayed
Other than EDIT	Other than below	Sequence number executed last
	Sequence number search	Sequence number read moment by moment during search
MEMORY	CURSOR [↑] is pressed with [PRGRM] pressed	Program number at the head of a current program
EDIT	CURSOR [↓] is pressed with [PRGRM] pressed	N's number first encountered, looking at the program from current position of memory in the forward direction
	CURSOR [↑] is pressed with [PRGRM] pressed	N's number first encountered, looking at the program from current position of memory in the reverse direction
	Reset status is executed by pressing [RESET], etc.	Program number at the head of a current program
MEMORY	Program number search	Program number searched for

5.4 Alarm Display (Function button: [ALARM])

When an alarm occurs and ALM is displayed at bottom right, pressing the [ALARM] button displays the alarm message, as in the photo below.

When an operator message is displayed, press the [ALARM] button again to display the alarm message.

See Appendix 7 for the meaning of the alarm number.

(Note) When an alarm occurs the alarm message is usually automatically displayed.

5.5 Operator Message

When an operator message is sent from the machine tool, it is automatically displayed on the screen.

When the operator message is to be displayed after selecting the other screen, push the [ALARM] button. And when the alarm message is displayed, press the [ALARM] button again.

5.6 Display and Reset of a Current Position (Function button: [POS])

(1) Press the [POS] button.

(2) Press a PAGE button. Data will be displayed in one of the following three ways.

(i) In the relative coordinate system

The relative position is displayed in which the position reset by an operator is zero.

Reset

When this is displayed, press the [X] , [Y] , [Z] or [4TH/5TH] key. The address will flicker. Then press the [ORIGIN] button. The relative position of the flickering address will be reset to zero.

(ii) In the work coordinate system

The current value in a programmed coordinate system set with G92, Automatic Coordinate System Setting, or by the following action (Reset) is displayed. For the T-axis, the presently selected tool number is displayed.

Reset ( )

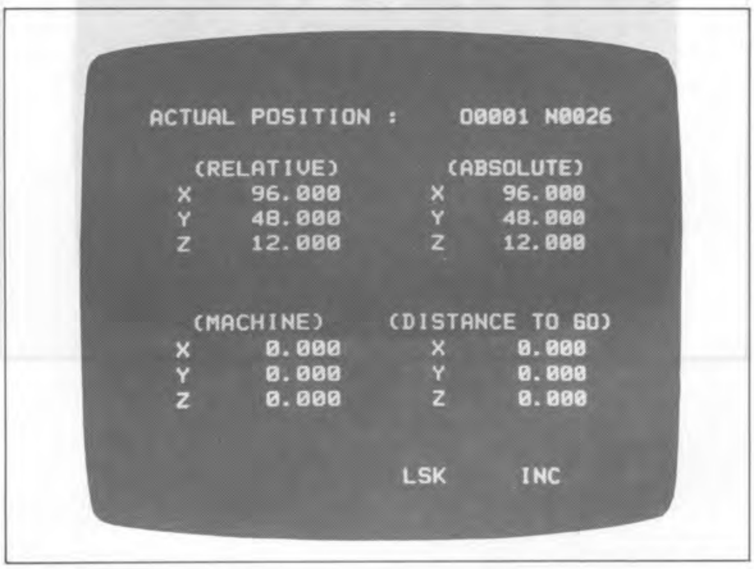
When this is displayed, press the X, Y, Z or 4TH/5TH key. The address will flicker. Then press the ORIGIN button. The current position of the flickering address will be reset to zero.

Note: Reset is possible only in an automatic-operation stopped or a reset status.

(iii) Overall display

The current position in the following coordinate systems is displayed simultaneously:

(a) Position in the relative coordinate system (RELATIVE)
(b) Position in the work coordinate system (ABSOLUTE)
(c) Position in the machine coordinate system (MACHINE)
(d) Residual movement amount (DISTANCE TO GO)



The residual movement amount indicates the movement amount remaining in one block of a command. When the overall display is on, the position of a paticular coordinate system cannot be reset.

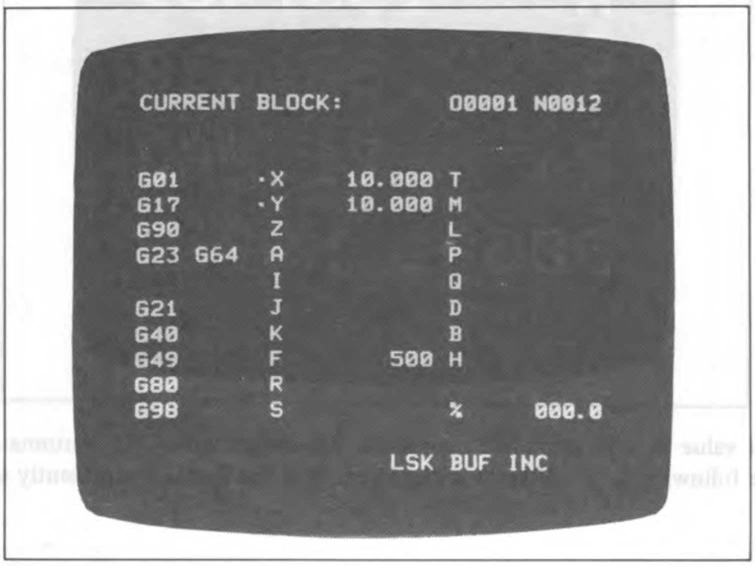
The unit of the machine coordinate system is the same as that of the machine system.

5.7 Command Value Display (Function button: COMND)

(1) Press the COMND button.

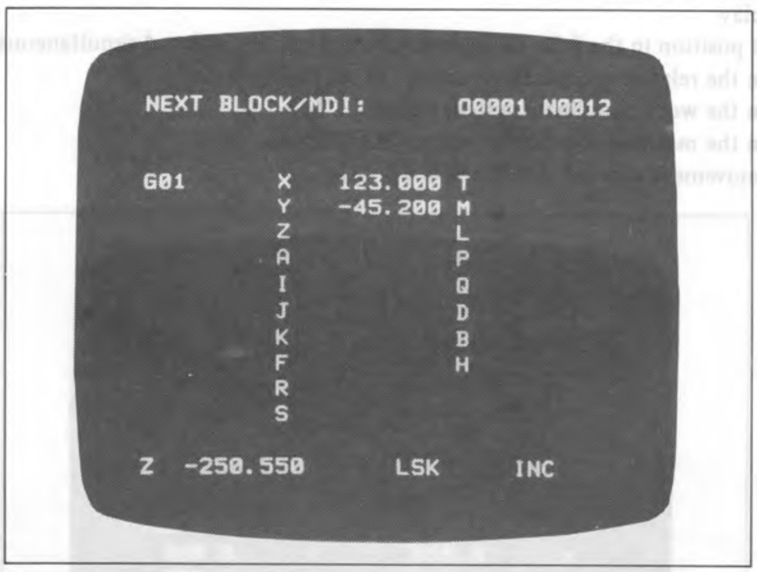
(2) Press a PGAE button. Data will be displayes in the following two ways.

(i) Command values being executed, and modal values previously specified are displayed.

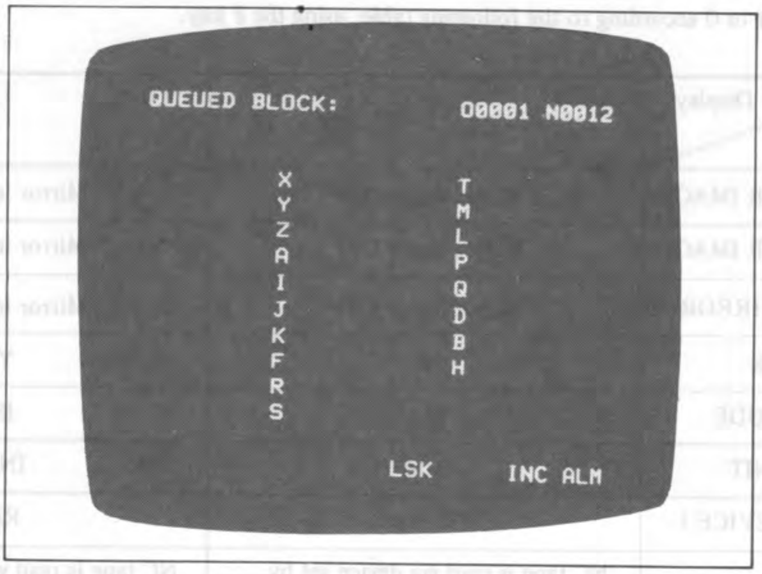


In the above photo, a numeral following the character % indicates the feedrate overridden.

(ii) Command values entered from the MDI or command values to be executed next are displayed.



(iii) The command to be executed next but one is displayed in the offset mode by cutter compensation.



5.8 Setting (Function button: SET)

5.8.1 Display and setting input, output, etc.

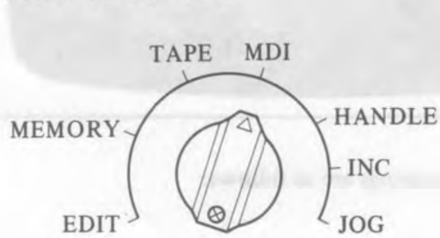
(1) Press the SET button.

(2) Press a PAGE button. Setting and display is performed in the following two ways.

(i) Input and output

Setting ( /no key)

(a) Set the mode selector switch to MDI.



(b) Move the cursor to the item to be changed by pressing a CURSOR button.
The cursor does not move by using the N key.

(c) Enter 1 or 0 according to the following table, using the P key.

Item \ Display	0	1
X MIRROR IMAGE	Mirror image OFF	Mirror image ON
Y MIRROR IMAGE	Mirror image OFF	Mirror image ON
4th axis MIRROR IMAG	Mirror image OFF	Mirror image ON
TV CHECK	No	Yes
PUNCH CODE	EIA	ISO
INPUT UNIT	MM	INCH
INPUT DEVICE 1	NC tape	RMT
INPUT DEVICE 2	NC tape is read via device set by INPUT DEVICE 1, when registering an NC tape into memory.	NC tape is read via a device set by data number 340, when registering a program tape into memory.

Key in the P , 0 or 1 keys and the INPUT button in this order.

(Note 1) *Setting using an option which is not selected is impossible. When no inch/metric switch option is available, for example, INPUT UNIT = 1 cannot be set for a metric machine. Also when no ISO code input option is available, PUNCH CODE = 1 cannot be set.*

(Note 2) *When G20 (INCH input)/G21 (MM input) is executed, the INPUT UNIT is automatically rewritten.*

(Note 3) *The PUNCH CODE specifies ISO code or EIA code in punching, and is not related to input. In input, ISO or EIA code is automatically discriminated.*

(Note 4) *A device to be used for punches is set in data number 341.*

(ii) Others

Displayed numbers and their meanings are as follows:

Data number	Contents
057	Run hour (in hours) (TMHOR)
058	Run hour (in minutes) (TMMIN)
059	Run hour (in seconds) (TMSEC)
067	Retract amount (CYCR) in canned cycle G73 (high-speed peck drilling cycle)
068	Cutting start point in canned cycle G83 (peck drilling cycle)
141	Roun hour (TIME1)
151	X-axis value of stored stroke limit-2 1st vertex
152	Y-axis value of stored stroke limit-2 1st vertex
153	Z-axis value of stored stroke limit-2 vertex
155	X-axis value of stored stroke limit-2 2nd vertex
156	Y-axis value of stored stroke limit-2 2nd vertex
157	Z-axis value of stored stroke limit-2 2nd vertex
180	Sequence number at which execution is to be stopped
319	Various settings (PRG8, MCS8, MPD8, MCS7)
340	Input device selection at program registration time (IDVICE)
341	Output device selection at punch out time (ODVICE)
355	Deceleration distance at the end of a block (Automatic corner override)
356	Deceleration distance at the beginning of a block (Automatic corner override)
407	Scaling factor

(Note 1) These are also set as parameters at the same data numbers.
(Note 2) Refer to items of the data number in Appendix 6 for detailed contents.
(Note 3) Data numbers 340 and 341 are detailed as follows.

	3	4	0	IDVICE
	3	4	1	ODVICE

IDVICE: selects an input device to register a program in memory (is effective when INPUT DEVICE 2 = 1 (RS232C) is set.)

ODVICE: selects an output device for punching.

Setting	I/O device
0	Tape reader for input FACIT PUNCHER for output
1	ASR33/ASR43 for both input and output Sets baud rate and other parameters to 310.
2	RS232C for both input and output Sets baud rate and other parameters to 311.
3	RS232C for both input and output Sets baud rate and other aprameters to 312.
4	RS232C for both input and output Sets baud rate and other parameters to 313.

Both can be set as parameters.

Setting (/no key)

(a) Set the mode selector switch to MDI.

(b) Move the cursor to the item to be changed by pressing a CURSOR button. The cursor does not move by using the N key.

(c) Press the [P] and [numeric keys] and the [INPUT] button in this order.

5.8.2 Display and setting of custom macro variable values

The values of all common variables and of local variables in custom macro body currently being called can be displayed on the CRT.

When a variable value is ⟨vacant⟩ (see Section III 14.2.3), a blank is displayed. When an absolute value exceeds 99999999, OVER FLOW is displayed. When an absolute value is not 0 and under 0.0000001, UNDER FLOW is displayed.

Display

(1) Select SUBPAGE of SETTING
Press the [SET] button for SETTING DISPLAY and press it again.

(2) Display covers six pages. Press the PAGE button to display a particular page.
Page 1 . . . . . . local variables #1 ~ #20 in current degree of multicalling
Page 2 . . . . . local variables #21 ~ L33 in current degree of multicalling
Page 3 . . . . . common variables #100 ~ #119
Page 4 . . . . . common variables #120 ~ #139
Page 5 . . . . . common variables #140 ~ #149
Page 6 . . . . . common variables #500 ~ #509

(3) Move the cursor to the variable number to be displayed.
Method 1 Press the CURSOR button to move the cursor in sequence. Move the cursor beyond the page to shift the screen to the next page.
Method 2 Key in [N] and the [variable number] and press the [INPUT] button.

Setting ( )

(a) Select MDI mode.

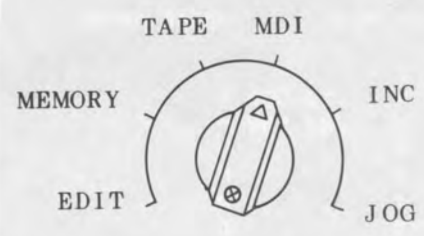
(b) After displaying variables and moving the cursor to the number of the variable to be altered, key in [P] and the [variable value] and press the [INPUT] button.

5.9 Operation from the MDI (Function button: COMND)

One command block can be entered from the MDI for execution.

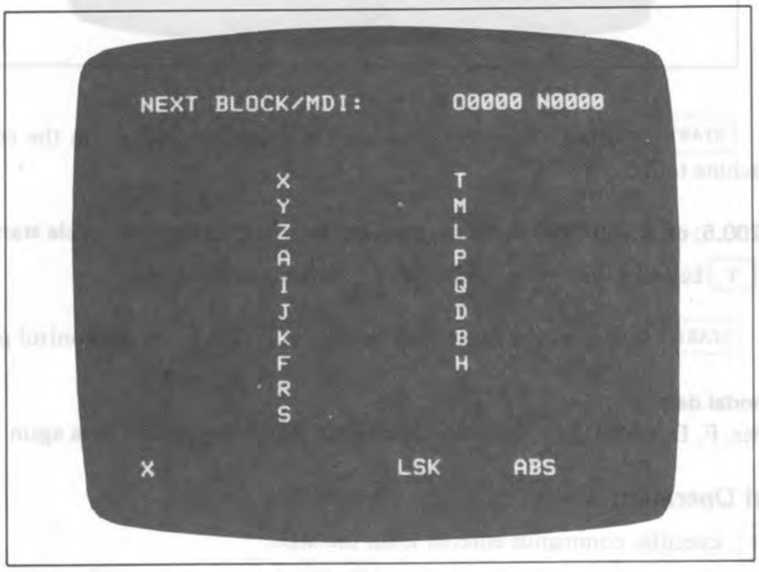
(1) For example, X10.5Y200.5:

(a) Set the mode selector switch to MDI



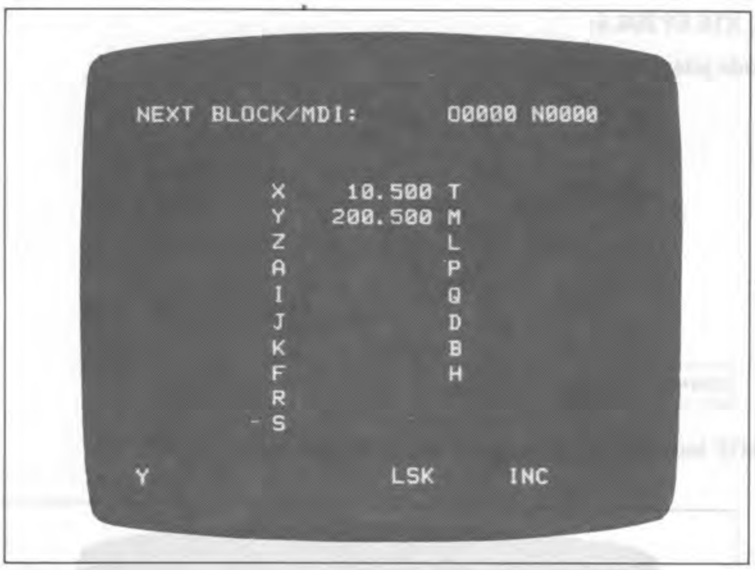
(b) Press the COMND button.

(c) Press a PAGE button to display NEXT BLOCK/MDI at the top left.



(d) Press the X , 1 , 0 , · , 5 keys and the INPUT button in this order. If you are aware of an error in the keyed-in number before pressing the INPUT button, press the CAN button and key in the correct number again. If you notice an error after the INPUT button is pressed, key in the number again from the beginning.

(e) Press the [Y], [2], [0], [0], [.], [5] keys and the [INPUT] button in this order. If you make a mistake keying in the number, follow the above instruction.



(f) Press the [START] button. Press the cycle start button CYCLE START on the control panel (depending on the machine tool).

(2) Canceling Y200.5; of X10.5Y200.5; before pressing the [START] button (cycle start button)

(a) Press the [Y] key and the [CAN] and [INPUT] buttons in this order.

(b) Press the [START] button or the cycle start button CYCLE START on the control panel.

(3) Cancelling modal data

Modal G codes, F, D, and H data cannot be cancelled. Enter the correct data again.

5.10 Starting MDI Operation

Pressing [START] executes commands entered from the MDI.

Note: Depending on the machine tool, pressing the [START] button may start both tape operation and memory operation. Also depending on the machine tool, not pressing this [START] button but pressing the start button on the control panel instead may start MDI operation.

5.11 Reset

Pressing [RESET] executes a reset. This is generally used to release an alarm status.
Pressing the [RESET] button places the NC in the following status.

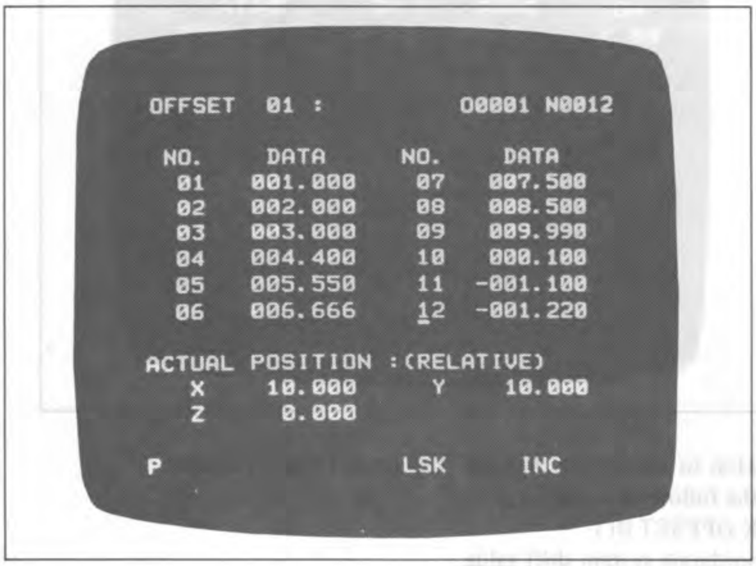
Status before reset		Status after reset
Move commands are being executed.		The tool stops with deceleration and the residual movement disappears.
M, S, T or B is being transmitted.		The transmission sequence stops. Refer to the description issued by the machine tool builder as to what happens to the machine tool at this time.
Buffer storage stores one block	MDI mode	Contents in buffer storage are not deleted.
	Mode other than MDI	Contents in buffer storage are deleted. BUF goes out.

In any case, pressing [RESET] button places the NC in a reset status; and in other than MDI mode, in the Label Skip status.

5.12 Setting and Display of Tool Offset and Cutter Compensation Values (Function button: OFSET)

(1) Press the OFSET button.

(2) Press a PAGE button to display the required page from 3 to 9 pages vailable.
Offset numbers 1 ~ 12 in page 1
Offset numbers 13 ~ 24 in page 2
Offset numbers 25 ~ 32 or 25 ~ 36 (Option) in page 3
Offset numbers 37 ~ 48 in page 4 (option)
⋮
Offset numbers 97 ~ 99 (option) or 97 ~ 108 (option) in page 9
⋮
Offset numbers 193 ~ 200 (option) in page 17.

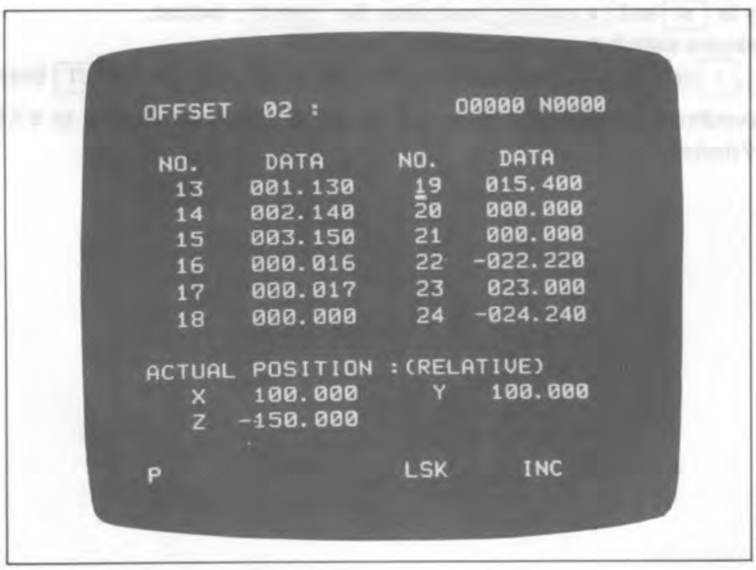


Offset Values in Page 1

(3) Move the cursor to the offset number to be changed.
Method 1 Continuously press the CURSOR button. The cursor will move in sequence. Moving the cursor beyond the page changes the screen to the next page.
Method 2 Key in N and an offset number and press the INPUT button.

(4) Set the mode selector switch to a position other than EDIT.

(5) Key in P , and an offset value and press the INPUT button.
The following photo shows the screen when P , 1 , 5 , . , 4 and INPUT are pressed at offset number 19.



(Note 1) *When the offset value is changed during automatic operation, the new offset value does not become valid until the D code followed by the offset number is commanded.*

(Note 2) *All offset amounts is cleared to zero by O–9999* [INPUT].

5.13 Setting and Display of Work Zero Point Offset Value

(1) Press the [OFSET] button twice to display the page of WORK OFFSET.

(2) Press a PAGE button to display the required page from 2 pages available.
Each page displays the following values:
 (i) Page 1 (WORK OFFSET 01)
  00: Work coordinate system shift value.
  01: Work zero point offset value for work coordinate system 1 (G54).
  02: Work zero point offset value for work coordinate system 2 (G55).
  03: Work zero point offset value for work coordinate system 3 (G56).
 (ii) Page 2 (WORK OFFSET 02)
  04: Work zero point offset value for work coordinate system 4 (G57).
  05: Work zero point offset value for work coordinate system 5 (G58).
  06: Work zero point offset value for work coordinate system 6 (G59).

(3) Move the cursor to a number to be changed.
 Method 1 Continuously press the CURSOR button.
 The cursor will move in sequence. Moving the cursor beyond the page changes the screen to the next page.
 Method 2 Key in [N] and [a number] and press the [INPUT] button.

(4) Set the mode selector switch to a position other than EDIT.

(5) Key in [X], [Y], [Z] or [4TH/5TH], and shift or offset value and press the [INPUT] button.

(Note 1) *Work coordinate system shift value can be set in the range from 0 to ±7.999 mm or from 0 to ±0.7999 inches.*

5.14 Tool Length Measurement

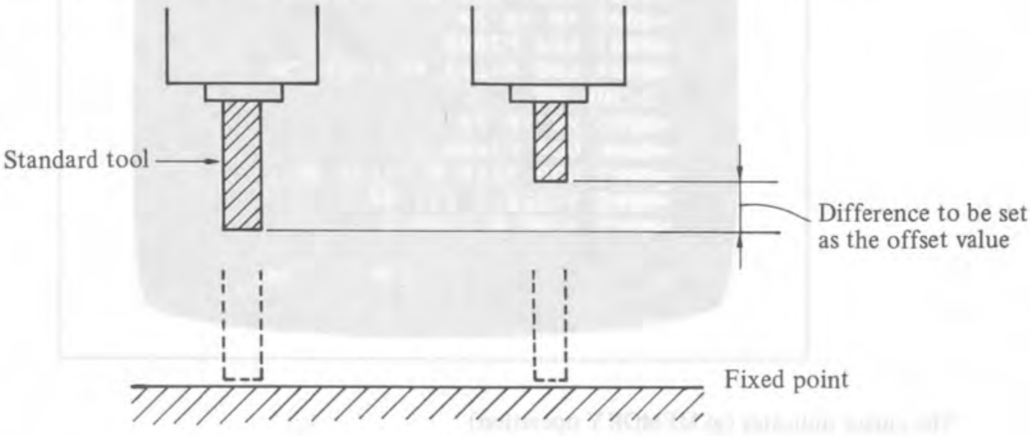
(1) Press the [OFSET] button to select the offset screen.

(2) Bring manually a standard tool to the fixed point on the machine tool (or the fixed point of the work).

(3) Press the [Z] and [ORIGIN] keys to reset the Z-relative coordinate values to zero.

(4) After that, bring manually a tool to be measured to the same fixed point. The difference between the standard tool and the tool to be measured will be displayed in relative position display.

(5) As in offset value setting, move the cursor to the offset number and press the [Z] key and the [INPUT] button without keying in the number. The difference will be input as the offset value.



5.15 Program Display (Function button: [PRGRM])

(1) In EDIT mode

Pressing the [PRGRM] button displays a page containing a selected word in a currently selected program.

See Section IV 5.16 for which program to display. Pressing a PAGE button displays program contents in sequence. Press the PAGE [↓] button displays the contents in the normal direction; the PAGE [↑] button, in the reverse direction.

(Note 1) Switching from another mode to the EDIT mode and pressing the [PRGRM] button displays the contents of a program starting with a block currently being executed or which has been executed. When returning to the head of a program (see Section 5.31.4), however, the contents are displayed from the head of the program.

(2) Memory operation

Pressing the PRGRM button displays the screen containing the block currently being executed.

The cursor indicates (in MEMORY operation)

(a) When flickering,
the block to be executed next.

(b) When not
the block currently being executed or a block which has been executed.

(Note 1) Strictly, the cursor flickers in neither automatic operation nor feed hold status when the buffer storage is empty indicating that a block should be read in the buffer storage next for execution.

(Note 2) When a subprogram is being executed, the same as above is also displayed in TAPE mode.

(Note 3) When a program is started in MEMORY mode after the cursor is moved by pressing the PAGE or CURSOR button in EDIT mode, the block at the cursor position in EDIT mode is read into the buffer storage.

(3) In modes other than the EDIT mode and the Memory mode

Pressing the PRGRM button displays the contents of the block currently being executed or which has been executed on the left, and the contents of the block to be executed next on the right side.

(Note 1) As in G28, G29, canned cycles, and movement at a corner in cutter compensation, when one command block creates movements for several blocks, the same contents are sometimes displayed on both right and left sides.

5.16 Program Number Search (Function button: PRGRM)

One of the programs in memory can be searched for.

O1001	O3054	O1972

⟹ Search for a program number.

(1) Method 1

(a) Select a mode (EDIT or MEMORY).
(b) Press the PRGRM button.
(c) Key O and the program number to be searched for and press the CURSOR ↓ button. At the end of the search, the first page of the program will be displayed.

(2) Method 2

(a) Select a MEMORY mode.
(b) Press the PRGRM button.
(c) Key in O and press the CAN and CURSOR ↓ buttons in this order.
The next registered program will be displayed.

(3) Method 3

(a) Select an EDIT mode.
(b) Press the PRGRM button.
(c) Key in O and CURSOR ↓ buttons in this order.

The next registered program will be displayed.

If you keep pressing the CURSOR ↓ button, numbers of all registered programs are displayed in succession. This method is used to examine registered program numbers.

(Note 1) When all registered program numbers have been displayed, the first one is displayed again.
(Note 2) Searching for a program number clears the contents of buffer storage.

5.17 Registering a Program Tape in Memory (, any function button)

(a) Select a mode (EDIT or MEMORY).
(b) Load the program tape into the tape reader.
(c) When the tape has no program number or when the program number is to be changed, enter the program number. (When the tape has a program number and the program number is not changed, this is not required.)
Key in O and the program number.
(d) Press the READ button. When the program tape is being registered, EDIT is displayed at the bottom of the screen.
(e) Pressing PRGRM displays the first of the registered program.

5.18 Registering One Tape with Programs into Memory (, any function buttons.)

O1111 M02;	O2222 M30;	O3333 M02;	ER (%)
↑ Stops here	↑ Stops here	↑ Stops here	

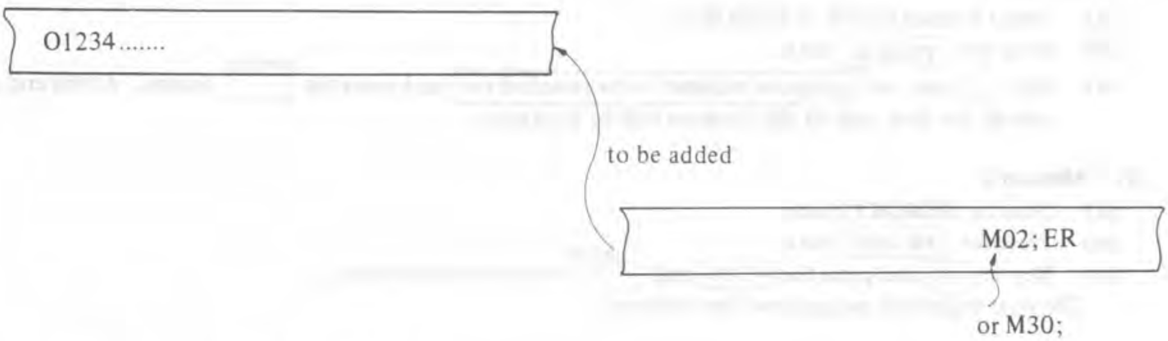
(a) Operation is the same as described in Section 5.16. The tape stops at M02;, at M30;, and at M99;. Then press the READ button.
(b) To change the program number or set a new one, key in O and a program number and press the READ button.
(c) To register programs in succession as far as ER, key in O and –9999 and press the READ button.

(Note 1) The Label Skip status is not entered until ER is read, so when another tape is to be added into the tape reader after reading up to M02;, M30; or M99;, press the RESET button.

(Note 2) *When reading a tape via RS-232C interface, note that the tape does not stop right after M02, M30 or M99 but it stops after a while. In this case, since the NC stores the information beyond code M02, M30 or M99, press* READD *button subsequently.*
However if a reset operation or a mode change is performed in the overrun state, the information following M02, M30 or M99 will be cleared.

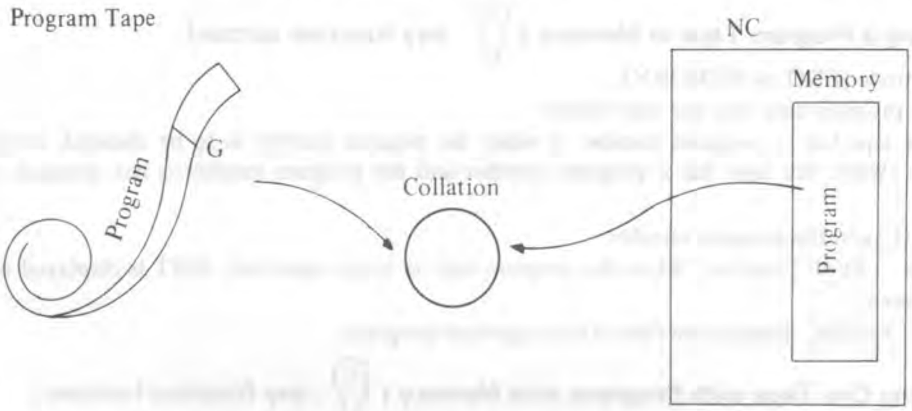
(Note 3) *It is also possible to load all programs by only pressing* READ *button, by setting parameter RDAL (No. 024) to 1.*

5.19 Adding a Program after an Already Registered Program (, any function button.)



(a) Select a mode (EDIT or MEMORY).
(b) Press the PRGRM button.
(c) Set the program tape into the tape reader.
(d) Search for the program to be added.
Key in O and the program number and press the CURSOR ↓ button. (This is not required when the program has been found.)
(e) Press the CAN and READ buttons in this order.

5.20 Collating Programs in Memory and Those on Tape (Function button: PRGRM)



(a) Select a mode (EDIT or MEMORY).
(b) Press the PRGRM button.
(c) Load the program tape to be collated into the tape reader.
(d) Press the / key and the READ button in this order (a slash cannot be keyed in without pressing the PRGRM button).
(e) When the tape contains several programs, collation is performed until ER (%) is encountered.

(Note 1) *Pressing the* EOB *and* READ *buttons in this order collates the currently indicated word by the cursor and subsequent ones in memory and the tape. To collate a tape with no program number and a program registered with a program number entered from the MDI, return to the head of memory, press the* CURSOR ↓ *button once to display the word following the program number, then press the* EOB *and* READ *buttons.*

(Note 2) *A mismatch during collation generates the alarm. Pressing the* PRGRM *button displays the program and the cursor indicates the location of the mismatch. Pressing the* CAN *button releases the alarm, starting collation with the next character.*

5.21 Inputting Programs by Keys

A program written on a process sheet can be directly registered into memory using MDI keys.

(a) Select EDIT mode.

(b) Press the [PRGRM] button. The current program is displayed.

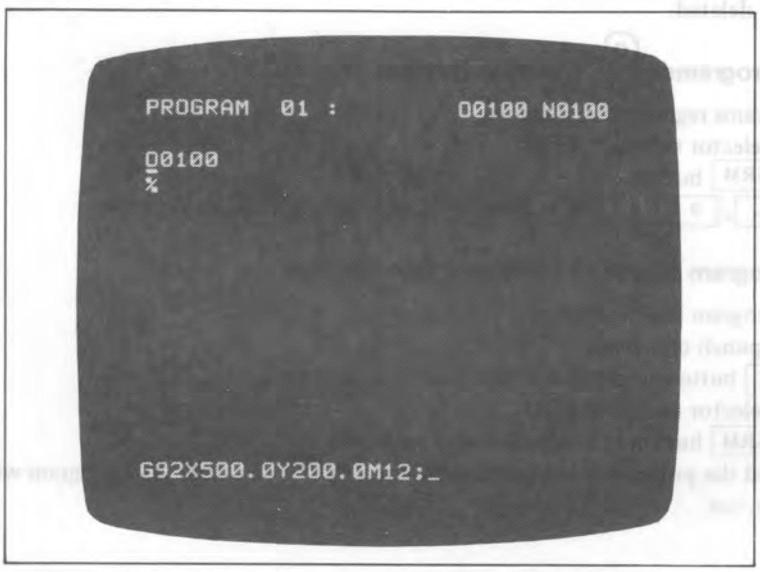
(c) Enter the program number of the program to be registered.
[O] [program number] [INSRT] displays the new screen.

(d) Key in one block according to the process sheet.

(Example) To input G92X500.0Y200.0M12;

[G] [9] [2] [X] [5] [0] [0] [.] [0] [Y]

[2] [0] [0] [.] [0] [M] [1] [2] [EOB]



(e) When a keying error occurs, press the [CAN] key. The character keyed in last will disappear. Keep pressing the [CAN] key to erase characters one by one from the last. A block of more than 32 characters is too long. Separate the block at an appropriate breakpoint.

(f) When the program has been keyed in correctly, press the INSRT button.

(g) Enter blocks one by one in this way.
(h) To correct an entered block, proceed as directed for program editing in Section *5.31*.
(i) For restart, move the cursor to the word keyed in last for continuous input. This operation is exactly the same as the inserting operation.
(j) When the entire program has been input, the operation is complete. To return to the beginning, press the RESET button.

5.22 Deleting a Program (, function button: PRGRM)

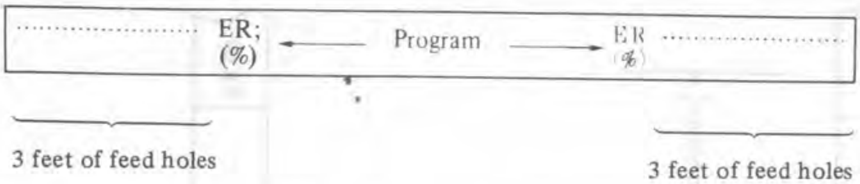
To delete a program registered in memory.
(a) Set the mode selector switch to EDIT.
(b) Press the PRGRM button.
(c) Key in O and the program number and press the DELET button. The program with the keyed in number will be deleted.

5.23 Deleting All Programs (, function button: PRGRM)

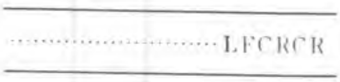
To delete all programs registered in memory.
(a) Set the mode selector switch to EDIT.
(b) Press the PRGRM button.
(c) Key in O , - , 9 , 9 , 9 , and 9 and press the DELET button.

5.24 Punching a Program (Option) (Any function button)

To punch out a program from memory.
(a) Make the tape punch unit ready.
(b) Press the SET button to check that the desired code is set.
(c) Set the mode selector switch to EDIT.
(d) Press the PRGRM button (this is not always required).
(e) Key in O and the program number and press the PUNCH button. The program with the keyed number will be punched out.



(Note 1) *The space code for the TV check is automatically punched.*
(Note 2) *Two CRs are punched after LF in ISO code.*

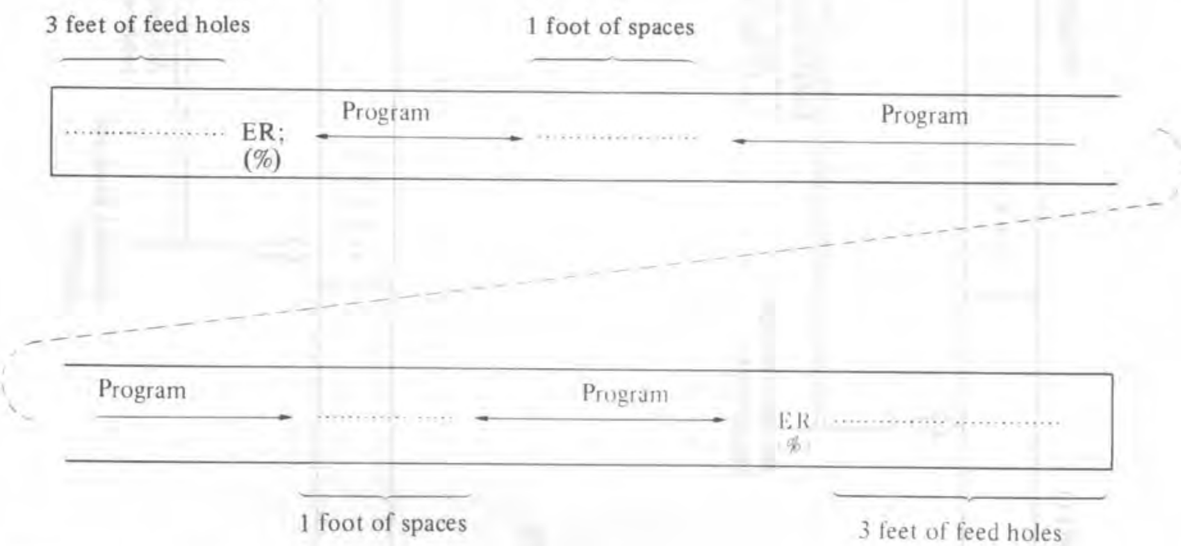


(Note 3) *To stop punching feed holes midway, press the* [CAN] *button.*
(Note 4) *Pressing the* [RESET] *button stops punching.*
(Note 5) *When connecting a tape punch unit to NC, NC and tape punch unit power must be turned off.*

5.25 Punching All Programs (Option) (Any function button)

To punch out all programs from memory.

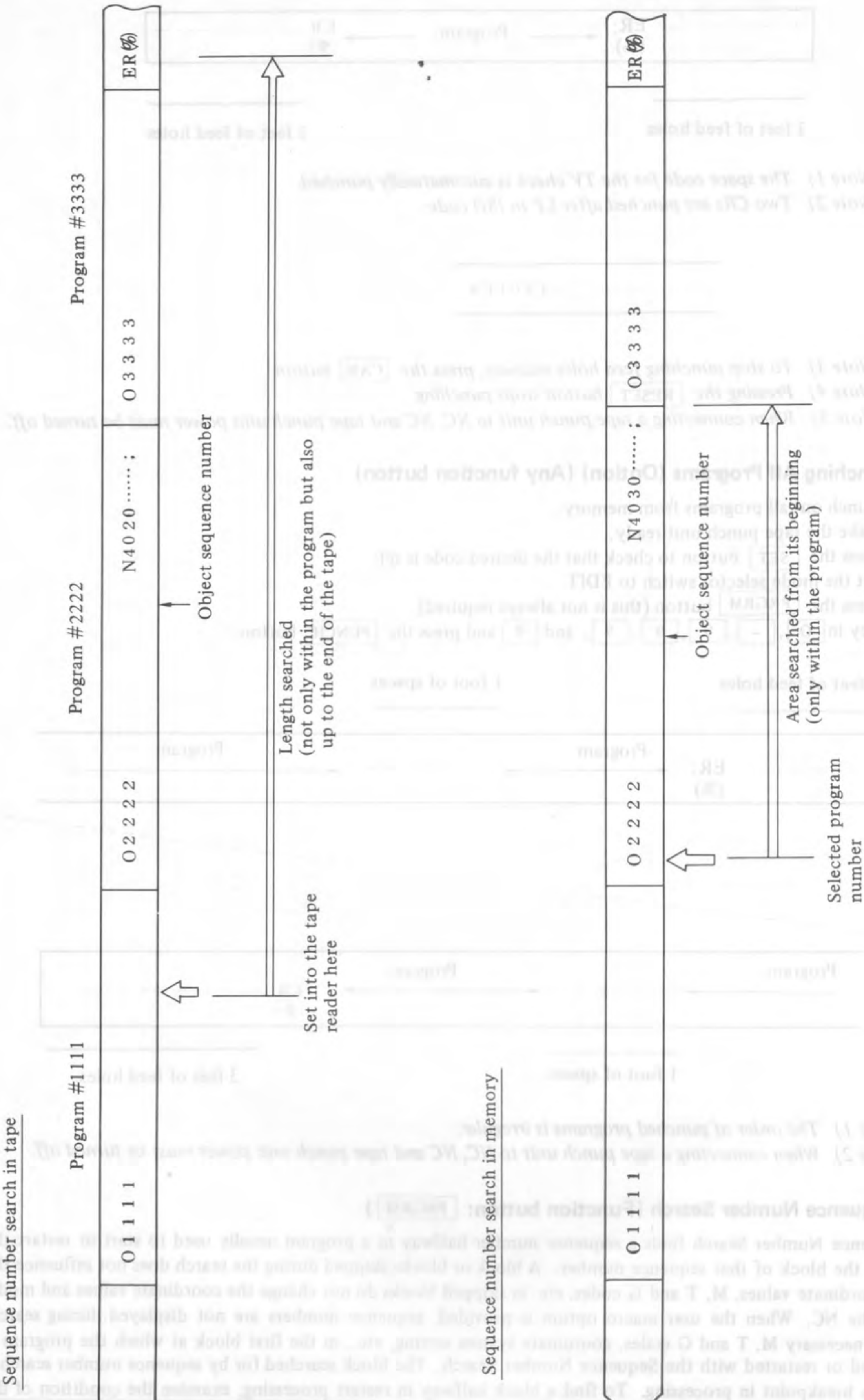
(a) Make the tape punch unit ready.
(b) Press the [SET] button to check that the desired code is set.
(c) Set the mode selector switch to EDIT.
(d) Press the [PRGRM] button (this is not always required).
(e) Key in [O], [-], [9], [9], [9], and [9] and press the [PUNCH] button.



(Note 1) *The order of punched programs is irregular.*
(Note 2) *When connecting a tape punch unit to NC, NC and tape punch unit power must be turned off.*

5.26 Sequence Number Search (Function button: [PRGRM])

Sequence Number Search finds a sequence number halfway in a program usually used to start or restart the program at the block of that sequence number. A block or blocks skipped during the search does not influence the NC; the coordinate values, M, T and G codes, etc. in skipped blocks do not change the coordinate values and modal values of the NC. When the user macro option is provided, sequence numbers are not displayed during search. So specify necessary M, T and G codes, coordinate system setting, etc., in the first block at which the program is to be started or restarted with the Sequence Number Search. The block searched for by sequence number search is usually at a breakpoint in processing. To find a block halfway in restart processing, examine the condition of the machine tool and NC and specify M, S, T and G codes, coordinate system setting, etc. from the MDI & DPL panel as required.

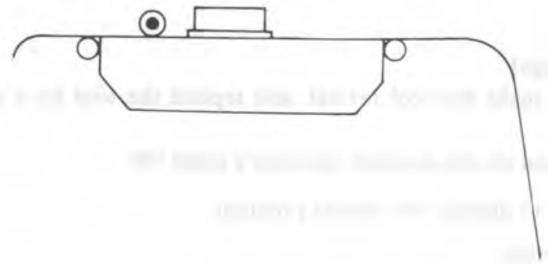


Sequence number search in tape
Program #1111
O 1 1 1 1
Set into the tape reader here
Program #2222
O 2 2 2 2
Length searched (not only within the program but also up to the end of the tape)
N 4 0 2 0 ...... ;
Object sequence number
Program #3333
O 3 3 3 3
ER (%)
Sequence number search in memory
O 1 1 1 1
O 2 2 2 2
Selected program number
Area searched from its beginning (only within the program)
N 4 0 3 0 ...... ;
Object sequence number
O 3 3 3 3
ER (%)

(1) Sequence number search on tape

(a) Set the mode selector switch to TAPE.

(b) Load the program tape into the tape reader.



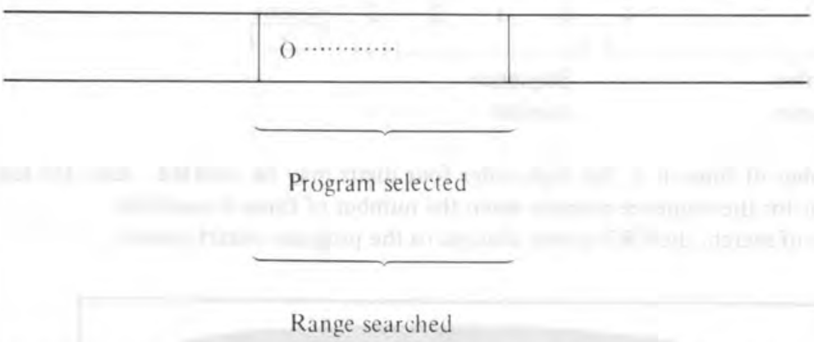
(c) Press the PRGRM button.

(d) Key in N and the sequence number to be searched for and press the CURSOR ↓ button. The sequence number will be searhced for.

(2) Sequence number search in memory

(a) Set the mode selector switch to MEMORY.

(b) Select the program number to which the sequence number to be searched for belongs.



Proceed to (c) when the program contains the sequence number; Otherwise, execute Program Number Search to select a program number to which the sequence number belongs.

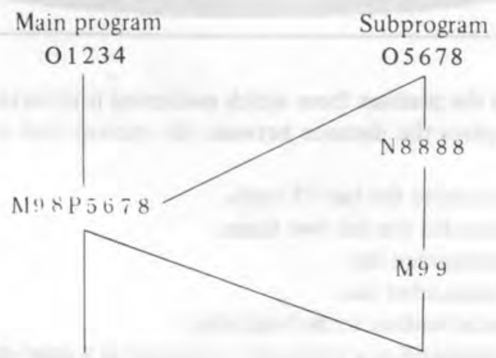
(c) Press the PRGRM button.

(d) Key in N and the sequence number to be searched for and press the CURSOR ↓ button. The sequence number will be searched for.

(Note 1) *Coordinate values and modal data are not altered during a search. Specify this data from the MDI after the search ends as required.*

(Note 2) *During a search, the following items are checked.*
TH check
TV check
Optional Block Skip
Alarm check (03, 04, 05 and 10)

(Note 3) *M98Pxxxx (subprogram call) is not executed during a search. Searching for a sequence number within a subprogram called from a currently selected program in MEMORY mode generates alarm No. 060.*



In the above example, searching for N8888 generates the alarm.

5.27 Program Restart

When the tool has been damaged or when machining is to be restarted after a rest, this function restarts machining from a block to be restarted from, by specifying its sequence number. Furthermore tape check function can be also used.

(1) The tool has been damage (P type)

(a) Press the feed hold button, make the tool retreat, and replace the tool by a new one. Alter offset values, when required.

(b) Set the program restart button on the machine operator's panel ON.

(c) Press the [PRGRM] button to display the current program.

(d) Return program to its beginning.

(i) In memory operation, press the CURSOR [↑] button in memory mode.

(ii) In tape operation, set the beginning of the tape at the tape reader and select the tape mode.

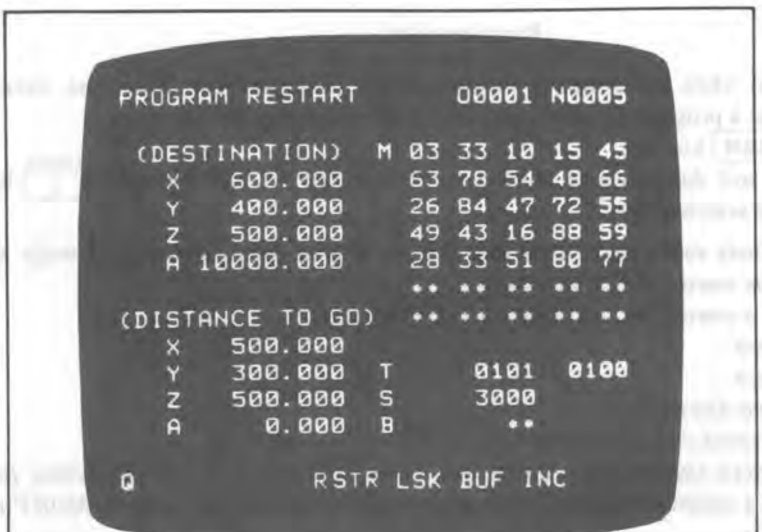
(e) [P] [sequence number] CURSOR [↑] searches the block to be restarted from. When the same sequence number appears many times, for example, when a sequence number is searched for in a subprogram to call it many times, specify the number of times by which the block of that sequence number appears in the high-order four digits and the sequence number in the low-order four digits.

P 1 2 3 4 0 1 2 3 CURSOR [↓]

$\underbrace{1\ 2\ 3\ 4}_{\text{Number of times}}\ \underbrace{0\ 1\ 2\ 3}_{\text{Sequence number}}$

When the number of times is 1, the high-order four digits may be omitted. Also, the leading zero may be omitted, except for the sequence number when the number of times is specified.

(f) On completion of search, the CRT screen changes to the program restart screen.



DESTINATION displays the position from which machining is restarted.
DISTANCE TO GO displays the distance between the current tool position and the position from which machining is restarted.
M displays M codes commanded the last 35 times.
T displays T codes commanded the last two times.
S displays the S code commanded last.
B displays the B code commanded last.
Displays a code commanded earliest at the beginning.
Also a program restart command or a cycle start command in a reset state clears each code.

(g) Set the program restart switch OFF.

294

(h) Look at the screen and issue M, S, T and B codes to be output, if present, via the MDI panel in MDI mode. In this case, the M, S, T or B code which was issued is not displayed on PROGRAM RESTART page.

(i) Return to the memory mode for memory operation; to the tape mode for tape operation. Check if the distance in DISTANCE TO GO is correct, and if the tool collides with the work, etc. when it moves to the position for machining restart. After moving the tool by hand to the position without a fear of collision, if present, press the cycle start button. The tool will move to the position for machining restart under dry run in 4th axis, X, Y and Z in this order and restart machining.

(2) Machining is restarted after (Q type):

(a) The power is turned off.
(b) The emergency stop button is pressed.
(c) The machine stopped instantaneously by the stored stroke limit alarm.
(d) The coordinate system is changed after the last automatic operation. For example,
  (i) G92 is commanded via the MDI panel.
  (ii) The coordinate system is shifted.
  (iii) The Automatic Coordinate System Setting by reference point return is made.
  (iv) The ORIGIN button is pressed.
  (v) The coordinate system is changed by resetting, etc.

(a) Do what should be done such as reference point return, etc. for the above operation before machining restart at the time of power on or releasing emergency stop or stroke limit alarm (see Notes discussed later).
(b) Move the tool by a manual operation to the machining start point of the program and set modal information and coordinates to the same state as when machining restarts.
(c) Set or change offset values if required.
(d) Set the program restart button on the machine operator's panel ON.
(e) Press the [PRGRM] button to display the program. When it is not desired, search for the desired program.
(f) Return the program to its beginning.
  (i) In memory operation, press the [CURSOR ↑] button in memory mode.
  (ii) In tape operation, set the beginning of the tape at the tape reader and select the tape mode.
(g) [Q] [sequence number] [CURSOR ↓] searches for the sequence number of the block to be restarted from. When the same sequence number appears many times during search, specify the number of times by which that sequence number appears in the high-order four digits and the sequence number in the low-order four digits.
(h) On completion of search, the CRT screen changes to the program restart screen.
(i) Set the program restart switch OFF.
(j) Look at the screen and output M, S, T and B codes to be output, if present, via the MDI panel in MDI mode. In this case, the M, S, T or B code which was issued is not displayed on PROGRAM RESTART page.
(k) Check if the tool collides with the work, etc. when the tool moves to the position for machining restart. When there is a fear of collision, move the tool manually to the position without the fear.
(l) Check if the distance in DISTANCE TO GO is adequate.
(m) In memory operation, return to the memory mode; in tape operation to the tape mode. Press the cycle start button. The tool will move to the position for machining restart under dry run in 4th axis, X, Y and Z in this order and restart machining.

(Note 1) [P] [*sequence number*] [CURSOR ↓] *performs no program restart on conditions that:*
(a) No automatic operation is executed after the power is turned on.
(b) No automatic operation is executed after the emergency stop or stroke limit alarm (instantaneous stop) is released.
(c) No automatic operation is executed after the coordinate system is set, altered, or shifted (change of external work zero point offset values).
Of the above conditions,
(a), (b) or reset for alarm 94-97 causes P/S alarm 97.
Coordinate system setting causes P/S alarm 94.
Coordinate system shift causes P/S alarm 95.
Coordinate system change causes P/S alarm 96.
The block from which machining can be restarted is one of the blocks which follows the block at which coordinate system was set or changed at last before machining interruption.

(Note 2) In both P type and Q type, when the tool moves to the position for machining restart in one axis at a time, after completion of motion in one axis, Single Block Stop is possible. But no MDI can be inserted. Manual operation can be inserted. But the axis which has returned does not move.

(Note 3) During search, when the input signals, offset values, and other conditions are not the same as before, the tool cannot return to the same machining start point as before. Also single block switch ON or MEMORY/TAPE mode change-over continues search operation.

(Note 4) When a feed hold is effected during search or when a reset is performed after search, redo the program restart operation from the beginning. However, turn to parameter 007: CLEAR for a reset in the MDI mode after search end.

(Note 5) With the program restart switch ON, cycle start is ignored.

(Note 6) Perform manual operation with Manual Absolute ON at all times, whether before or after machining. When a Program Restart is commanded without resetting after manual operation or when manual operation is performed along the axis which has not yet been returned to the machining restart position, the motion concerned is regarded as that of Manual Absolute ON regardless of Manual Absolute switch ON or OFF.

(Note 7) In principle the tool cannot return to the correct position in cases that:

(a) Manual motion was done with Manual Absolute OFF.

(b) Motion with Machine Lock ON or with Z-axis command cancel was effected.

(c) Mirror image was used.

(d) No coordinate system setting was made at the beginning of incremental programming.

(e) Manual operation was inserted during axis movement for return.

(f) When machine lock is released after program restart is commanded.

(g) When program restart was commanded in the block with skip cutting or in the subsequent blocks till the block with absolute command.

(h) When a coordinate system was created or shifted after search.

In the case of (c), however, with the block for which ON/OFF was switched last and subsequent blocks, P-type return is possible. In this case keep Mirror Image in the same state as interruption time. In any case, note that no alarm is generated.

(Note 8) When the specified block includes only M98, M99, macro call command (M65, G66, G67) or macro statement or when no specified block is searched out, alarm No. 60 is generated.

(Note 9) When program restart is commanded with no reference point return after the power is turned on or after releasing emergency stop or stroke limit alarm (instantaneous stop) and G28 is detected, the P/S alarm (98) occurs.

(Note 10) When a move command is executed by MDI operation before axis movement is performed after search end, the P/S alarm (99) occurs.

(Note 11) Until a final axis (Z) completes return after program restart is commanded, "RSTR" is flickering at the bottom on the CRT screen.

(Note 12) When the block immediately preceding the restart block is of G28, G30 or of incremental command which follows a G28 or G30 block, the absolute position of the 4th axis may be displaced by 360° under the conditions that the 4th axis is rotary axis and the direction of reference point return is negative.

5.28 Sequence Number Comparison and Stop

This function is provided to stop processing after executing commands up to the preset sequence number.

(a) Select the MDI mode.

(b) Depress [SET] button to obtain the setting page.

Set the cursor to setting number 180 by depressing [CURSOR ↑] or [CURSOR ↓] key. (The cursor cannot move using address N.)

(c) Input commands in the order of [P], [sequence number to which execution is to be stopped], and [INPUT].

(d) Select either TAPE or MEMORY mode.

Set the machine tool to be ready for automatic operation.

(e) Depress the cycle start button.

By step (c), the machine tool stops operating after executing data in the block having the preset sequence number.

The preset sequence number is cleared simultaneously when the machine tool stops operating.

For another comparison stop, if desired, repeat the above procedure, starting with (a).

(Note 1) Sequence number 0 cannot be used as the sequence number for comparison stop.

(Note 2) The preset sequence number is cleared by resetting.

5.29 Entering Offset Values from Tape (Any function button)

(1) Format

Offset values can be input from tape in the following format:

G10 P___p___ R___r___ ;

p: Offset number
r: Offset value (Absolute input in G90 mode, incremental input in G91 mode.)
G10 is required for each block. The tape ends with % (ISO) or ER (EIA).

;G10P– ········ ;G10P– ········; ······················;%

(2) Operation

(a) Load a tape in the above-format into the tape reader.
(b) Set the mode selector switch to TAPE.
(c) Press the START button (depending on the machine tool, the start button on the machine side must be pushed)

5.30 Punching Offset Values onto Tape (Option) (Function button: OFSET)

(1) Make the tape punch unit ready.
(2) Set the mode selector switch to EDIT.
(3) Press the OFSET button.
(4) Key in P, -, 9, 9, 9, and 9 and press the PUNCH button. The offset values of all the offset numbers will be punched in the same format as input. (A G90 is punched at the start of the tape and the absolute input is assumed.)

(Note) When connecting a tape punch unit to NC, NC and tape punch unit power must be turned off.

5.31 Parameter Display (Function button: PARAM)

Pressing the PARAM button displays parameters. Parameters cover several pages in display. Press a PAGE button to display a particular parameter. See Appendix 6 for the meanings of parameters.

5.32 Program Editing (Function button: PRGRM)

This function is used to modify a program in memory.
(1) Set the mode selector switch to EDIT.
(2) Press the PRGRM button.
(3) Select a program. When it has been selected, proceed to (4); otherwise, perform a Program Number Search.
(4) Search for the word to be modified. A scan or a word search can be used.
(5) Modify, insert, or delete the word.

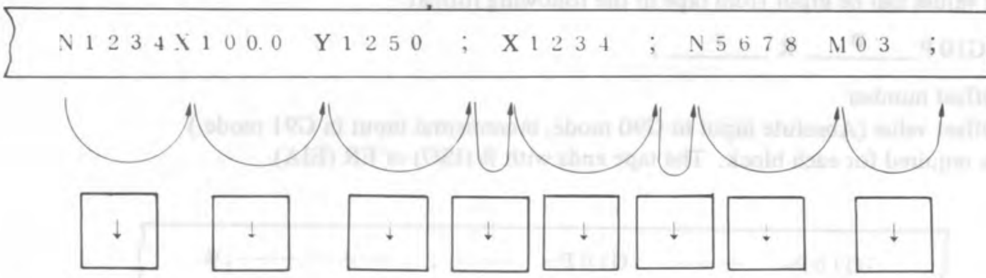
(Note 1) Words and editing unit.
A word includes an address and the number following it. But in user macros, the concept of a word is not clear. Consequently, "editing unit" is used. This is the object of modification and deletion in one operation. One scan causes the cursor to indicate the beginning of the editing unit. In insertion, data is inserted after the editing unit.

Definition of editing unit
(1) From one address to the next address exclusive.
(2) The address is an alphabetic character, IF, WHILE, GOTO, END, DO, =, or ; (EOB).
According to this definition, a word is also one editing unit. The word referred to in the following explanation on editing should strictly be termed the editing unit.

(Note 2) It is not allowable to continue a program again after changing, inserting, or deleting data to the program while temporarily suspending the machining by single block stop, feed hold, etc. during the execution of the program, otherwise the program is not always executed correctly as specified by the program data being displayed on CRT after the continuance of machining.
If it is desired to change memory data by part program editing, change them under the reset condition, or perform a reset operation after editing without fail, before executing the program.

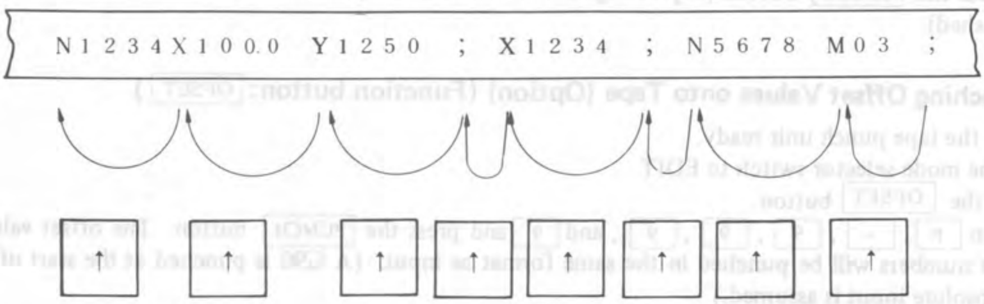
5.32.1 Word scan

(1) Press the CURSOR **[↓] button**



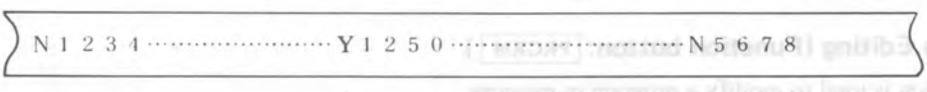
The cursor moves forward word by word on the screen; the cursor is displayed below the address character of a selected word.

(2) Press the CURSOR **[↑] button**



The cursor moves backward word by word on the screen; the cursor is displayed below the address character of a selected word.

(Example)



Scanned here.

(3) Keep pressing the CURSOR **[↓] or the** CURSOR **[↑] button to make a continuous search.**

(4) Pressing the PAGE [↓] button to display the following page and search from the first of that page.

(5) Press the PAGE [↑] button to display the preceding page and search from the first of that page.

(6) Keep pressing the PAGE [↓] or PAGE [↑] button to display one page after another.

5.32.2 Word search

A specified word is searched for from the current position in the forward direction.

N 1 2 3 4 X 1 0 0 0 Y 1 2 5 0 ; X 1 2 3 4 ; N 5 6 7 8 M 0 3 ;

↑ Current word ↑ X1234 to be searched for

⟶ Search direction

(1) Key in [X], [1], [2], [3], and [4].

Note 1: Keying in only X123 does not execute a search for X1234.

2: Keying in X9 does not execute a search for X009. Key in 009 to search for X009.

(2) Pressing the CURSOR [↓] button starts the search. The cursor is displayed below X in X1234 at the end of the search.

5.32.3 Search for addresses only

A specified address is searched for from the current position in the forward direction.

N 1 2 3 4 X 1 0 0.0 Y 1 2 5 0 ; X 1 2 3 4 ; N 5 6 7 8 M 0 3 ;

↑ Current word ↑ M03 to be searched for

(1) Key in [M].

(2) Pressing the CURSOR [↓] button starts the search. The cursor is displayed below M at the end of the search.

(Note 1) Pressing the [CAN] button after keying in a number cancels the number and displays blank. Pressing the [CAN] button only displays CAN.

(Note 2) Neither a Word Search nor an Address Search can be executed with the CURSOR [↑] button.

5.32.4 Returning the cursor to the Head of a Program

O 1 1 0 0 N 0 0 0 1 X 1 2.3 4 ; Y 1 5.6 7 ; G 7 0 X 1 2.5 ; M 0 4

↑ Head ↑ Current word

(1) Method 1

Press the [RESET] button in EDIT mode. The program will be displayed from its head.

(2) Method 2

Execute a Program Number Search.

(3) Method 3

(a) Set the mode selector switch to MEMORY.

(b) Press the [PRGRM] button.

(c) Press the CURSOR [↑] button. For part program editing, return to the EDIT mode.

5.32.5 Inserting a Word

T105 to be inserted

N1234X100.0 Y1250 ; X1234 ; N5678M30 ;

To be searched for

(1) Search for or scan the word immediately before the insertion location

(a) Scan
See Section 5.31.1.

(b) Word Search
See Section 5.31.2. When Y1250 is before the current position, return the cursor to the head of the program first.

(2) Key in [T], [1], [0], and [5] and press the [INSRT] button.

Before

After

(Note 1) When a number but no address is inserted, the inserted number is added to the current word (editing unit). In the above example, inserting 2.5 when the cursor is below Y in Y1250, produces Y12502.5.

(Note 2) Similarly, a number can be added after all addresses such as EOB IF, etc. Inserting 23 when the cursor is below ;, produces ;23. But this is meaningless in programming.

5.32.6 Changing a word ( )

```
N1234X100.0  Y1250T105  ;  X1234  ;
```

↑
To be changed to M15

(1) Search for/scan the word to be changed.
(2) Key in M , 1 , and 5 and press the ALTER button.

```
N1234X100.0  Y1250M15   ;  X1234  ;
```

Modified program

5.32.7 Insertion or correction of words, blocks, and character strings

More than one word, block or character string can be inserted (up to 32 characters). In the previous example (5.32.5), when

T105M20

is to be inserted,
key in T105M20 and press the INSRT button.

Before inserted

After inserted

Similarly, a word indicated by the cursor can be changed to more than one word, block or character string.

(Note 1) When the cursor is below Y in Y1250, inserting 2.5M20 produces Y12502.5M20.
(Note 2) When the cursor is below T in Y1250T105, replacing 2.5M20 produces Y12502.5M20.

5.32.8 Deleting a word

```
N1234X1000.0  Y1250T105 ; X1234 ;
                  ↑
          Y1250 to be deleted
```

(1) Search for/scan the word to be deleted.
(2) Press the [DELET] button.

```
N1234X1000.0  T105 ; X1234 ;
```

Program after deletion

5.32.9 Deleting up to an EOB

```
        Current word
           ↓
N1234X1000.0  Y1250  T105  M13  ; X1234  ;
    |←——— Area to be deleted ———→|
```

Pressing the [EOB] and the [DELET] button deletes up to an EOB and causes the cursor to move to below the address character of the next word.

5.32.10 Deleting blocks

The area from the currently displayed word to the block with a specified sequence number is deleted.

```
            Current word              N2233 to be keyed in
                 ↓                           ↓
N1234 M10 ; M15X10.0 ; ············T0122 ; N2233T1200 ; N3344 Y10.0 ;
               |←——— Area to be deleted ———→|
```

(1) Key in the sequence number of the last block to be deleted.
Key in [N], [2], [2], [3], and [3] in this example.
(2) Press the [DELET] button.

5.32.11 Memory arrangement

Frequent part program editing sometimes causes memory to be used uneconomically so that the data of the specified length tape cannot be stored. This requires Memory Arrangement.

(1) Press the [CAN] and the [ORIGIN] buttons in this order. After arrangement, the number of characters that can still be accommodated is displayed at the bottom of the screen.

(Note 1) When there is one program on a tape, the memory stores the specified length. When many programs are on one tape some areas in memory must be used for their identification.

(Note 2) For speedier part program editing, memory area in excess of characters actually modified or inserted is sometimes consumed. Memory Arrangement eliminates wasteful use of memory.

5.32.12 Display of the program numbers of all the programs registered

Arranging the memory contents discussed in 5.32.11 displays the program numbers of all the programs registered.

5.32.13 Custom macro edition ( )

Custom macros can be edited using shift key (ABS/INC).

Key operation is done as usual under edit mode with the program protect key released. However, note that the following points differ:

(a) ABS/INC key (shift key)

When ABS/INC key is depressed, cursor (key input cursor; at the character position following the last input data) changes from the ordinary _ to ^. If a key with a character at the right lower corner of the key-top is depressed under this state, the character at the right lower corner will be input.

After one character is inputted, the cursor reverts to the ordinary _. The cursor also reverts to the ordinary _ when ABS/INC key is depressed twice.

< Example >

299

(b) How to delete, alter and insert program

On editing cutom macro programs already input, cursor can move only to the following:

- (i) Address
- (ii) / of optional block skip
- (iii) # at the head of the left side of substitution statement
- (iv) (, =, or ;
- (v) Head character of IF, WHILE, GOTO, END, DO

There is a blank space of one character in front of the above on the CRT screen.

Deleting, altering and inserting of program is done in a unit from the first position of cursor to one character before the next position of cursor.

(Example) Position of cursor

N001 X–#100 ;
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X–SQRT[#3/3∗[#4+1]] ;
N004 X–#2 Y#1 ;
N005 #5 =1+2–#10 ;
IF [#1 NE 0] GOTO 10 ;
WHILE [#2 LE 5] DO 1 ;
#[2000+#2] =#2∗10 ;
#2 =#2+1 ;
END 1 ;

(Note 1) Cursor does not stop between ( ).

↑ ↑
Control out/in

(Example)

(#1=100) ;
↑
Cursor does not stop here.

(Note 2) Position of cursor may change according to alteration in program.

(Example) *When*
X100 Y200 ;
if Y200 is altered to 100 using ALTER *key, it becomes*
X100100;

(c) Abbreviation of custom macro words

When altering or inserting custom macro words, the top two characters can be used as abbreviation for the word.

The underlined part of each word below is the abbreviation which can be used as substitute for the whole word:

WHILE, GOTO, END, XOR, AND, SIN, COS, TAN, ATAN, SQRT, ABS, BCD, BIN, FIX, FUP, ROUND.

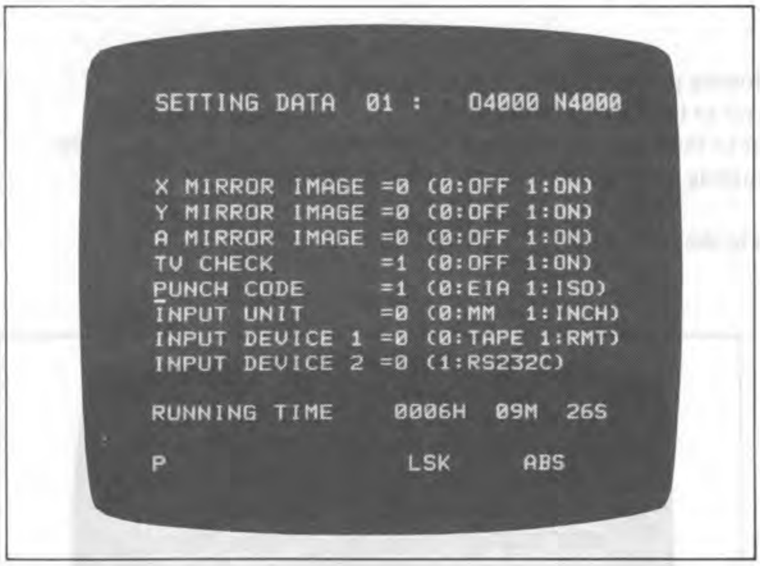
(Example)

When
WH[TA[#1∗AB[#2]]LERO[#3]]
is input as key input data, it is accepted as
WHILE[TAN[#1∗ABS[#2]]LE ROUND[#3]].

5.33 Run Hour Display

Automatic operating times are integrated and displayed on the screen in hours, minutes, and seconds (in units of two seconds in seconds).

Pressing the SET button displays the time as in the photo. When another page appears, press the PAGE button.



(Note 1) *Times to be integrated includes those of automatic operation and not those of single block stop, feed hold stop, etc.*

(Note 2) *Powering off immediately after stopping automatic operation sometimes generates an error of up to six minutes when powering on again.*

(Note 3) *Perform a preset by setting operation, if required. Data number is 57, 58 or 59.*

5.34 Menu Switch Function

Switch functions can be turned on and off in NC memory by using CRT instead of switches on the machine tool operator's panel.

By using this function, the number of switches on the machine tool operator's panel can be reduced.

The following signals can be turned on and off on CRT.

(1) Single block (SBK)
(2) Machine lock (MLK)
(3) Dry run (DRN)
(4) Optional block skip 1 ~ 9 (BDT1 ~ 9)
(5) Mirror image (MIX, MIY, M14, M15)
(6) Display lock (DLK)
(7) Auxiliary function lock (AFL)
(8) Z axis neglect (ZNG)
(9) Manual absolute (ABS)

The signals turned on and off on CRT remain unchanged, even if NC power supply is turned off once after their data have been stored into bubble memory.

These signals are not fully replaced with those on CRT but, they are regarded as being turned on when either machine tool signals or signals being set on CRT are turned on.

Accordingly, these signals can be turned on and off on CRT by deleting optional signals out of the above listed signals from the machine tool operator's panel and opening the circuits for these signal lines.

Setting and display

The CRT status of the above signals can be displayed by the following operation.

Display

(i) Select chapter 3 in setting.
Depress [SET] for setting display, and then, depress [SET] twice.

(ii) Display a necessary page by depressing PAGE button by selecting it from the display on 2 pages.

Page 1 Those other than optional block skip
Page 2 Optional block skip 1 ~ 9

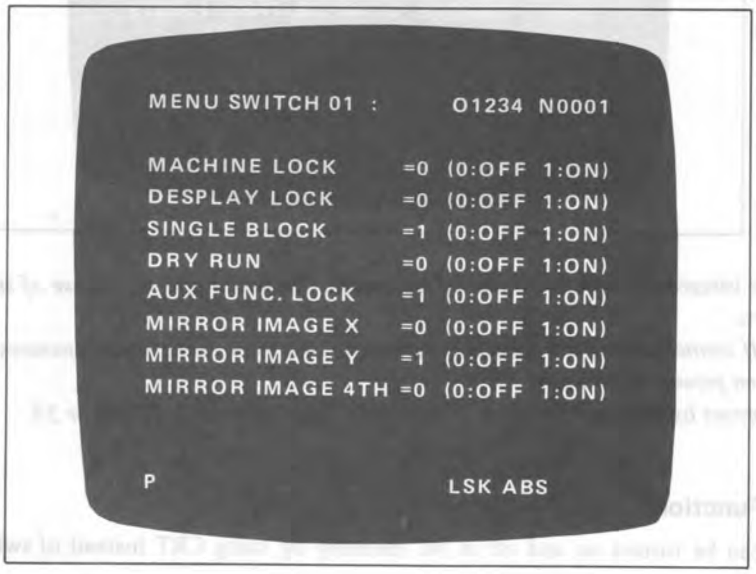
Setting

Observe the following procedure after display procedure.

(iii) Shift the cursor to the item to be changed.
Set the cursor to the item to be changed by depressing [CURSOR ↑] or [CURSOR ↓] key.

(iv) Input 1 for turning on or 0 for turning off after depressing address [P].

Depress keys in the order of (P) $\begin{Bmatrix} (0) \\ (1) \end{Bmatrix}$ (INPUT)



5.35 14″ Color CRT Operation

5.35.1 General

The basic operation remains unchanged from that of 9″ CRT, except for the following items.

(1) Meanings of data displayed on the CRT are color-displayed.
(2) The information displayed on one CRT screen is more than the information in case of 9″ CRT.
(3) Function buttons ([POS] , [PRGRM] , [OFSET] , . . . etc.) in the 9″ CRT serve as software keys, and their meanings are displayed on the CRT in case of 14″ CRT.

The following describes each screen obtained by depressing each software key (9″ CRT's function button).

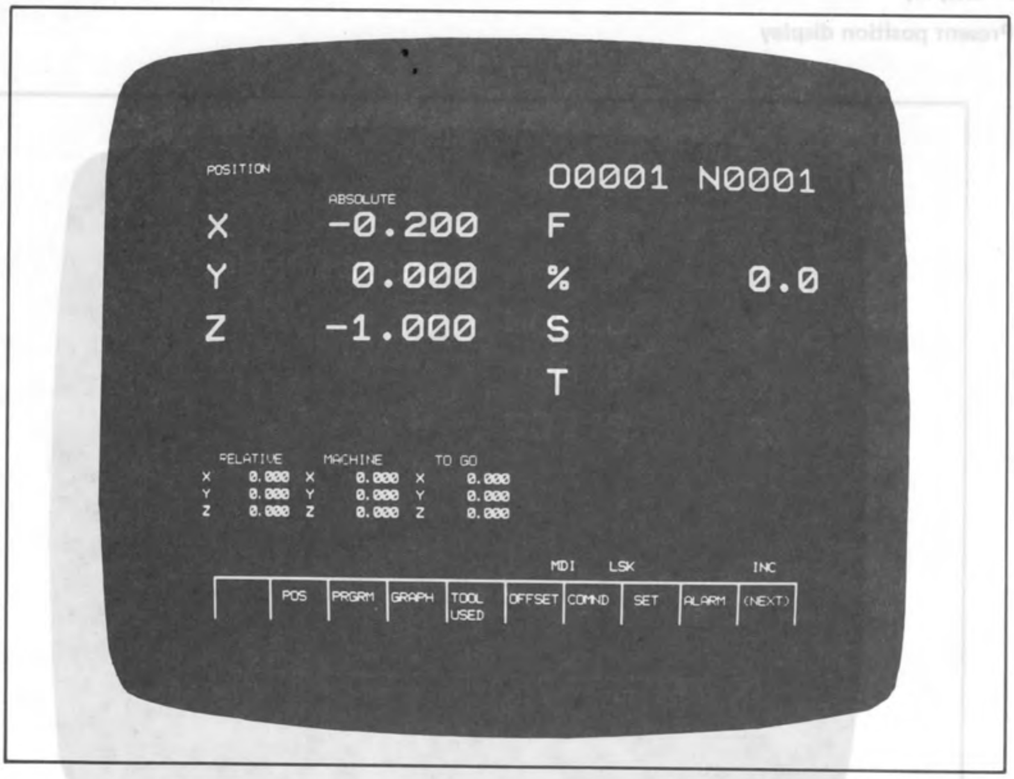
5.35.2 Display

(1) Present position display

By depressing software key POS, the present position in the relative coordinate system is displayed on the CRT. The present positions and residual move amounts in the work coordinate system and machine coordinate system are displayed at the lower part of the CRT.

Also, F, S, T commands and available feedrate override are displayed on the right side of CRT.

(Note) *The display positions of software keys at the bottom part of CRT more or less differ between M95 series and M97 or MA3 series. However, this difference may be left out of consideration from the viewpoints of operations.*
You have only to depress corresponding software keys.
The CRT screen in this manual shows the M97 or MA3 series.



By depressing the PAGE [↓] key in the display on page 289, the work coordinate system position and the relative coordinate system position are replaced with each other on the screen.

(2) Program display

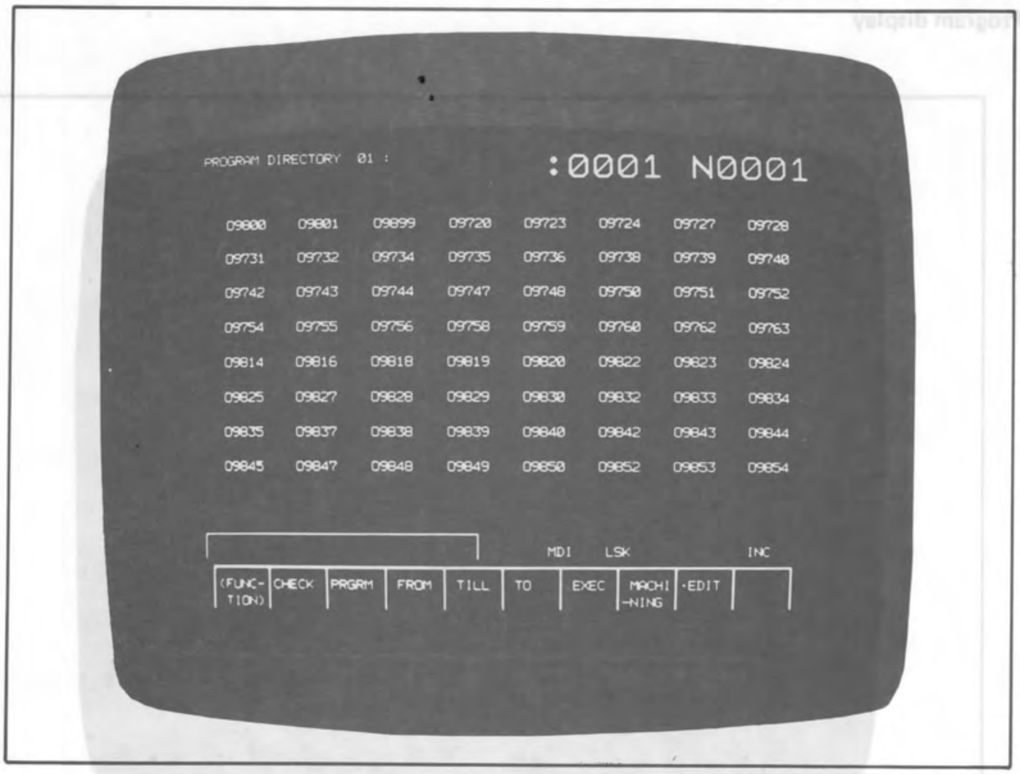
A program is displayed by depressing [PRGRM] software key.

The [*EDIT] display shows that this program is being edited at present.

Also, [*MACHI -NING] display shows that this program is being executed at present.

Either machining program display or editing program display can be selected by depressing [MACHI -NING] or [EDIT] software key.

For using the background EDIT function (option), refer to the supplementary manual (B-54044E-1).



By depressing $\boxed{\text{PRGRM}}$ key once more in the display on page 291, program directory is displayed as shown in the above photo.

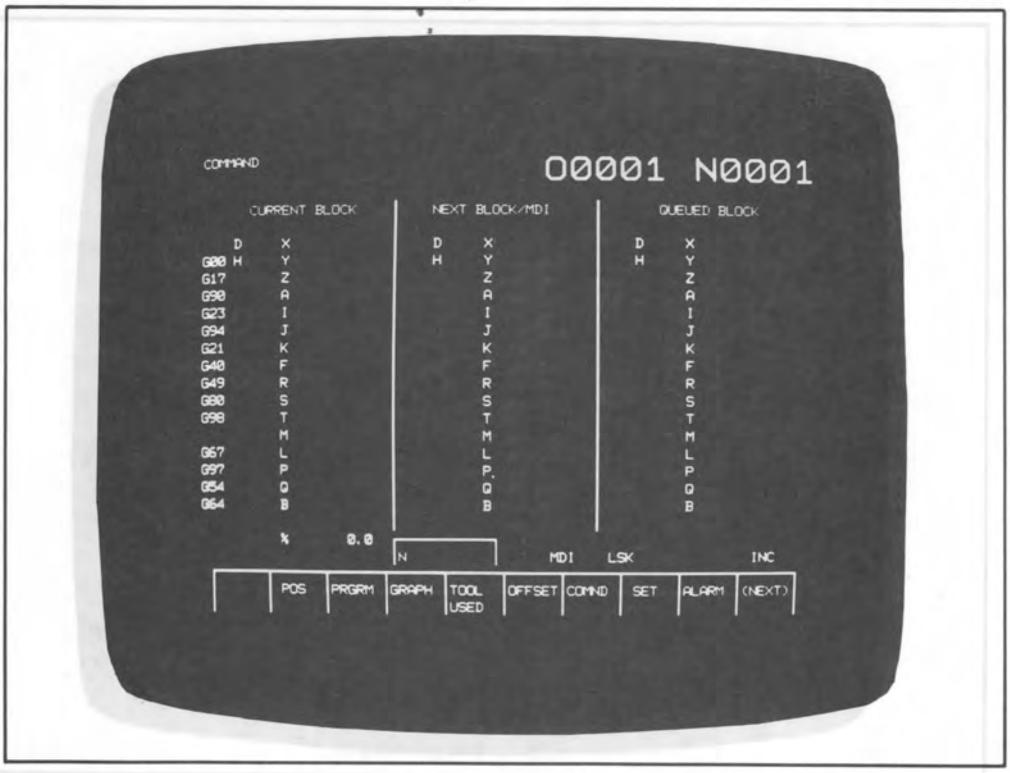
(3) Offset value display

Offset values corresponding to each tool number are displayed by depressing OFSET software key.

Select desired data by depressing PAGE ↑ ↓ and CURSOR ↑ ↓ key.

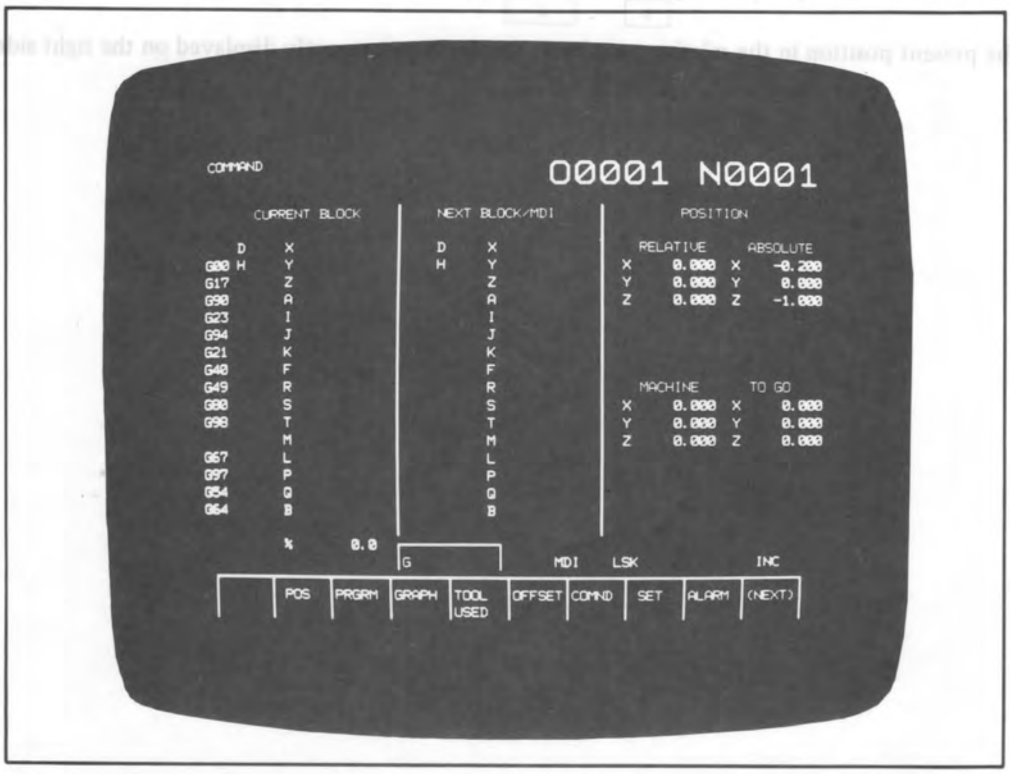
The present position in the relative coordinate system is concurrently displayed on the right side of CRT.

(4) Command display

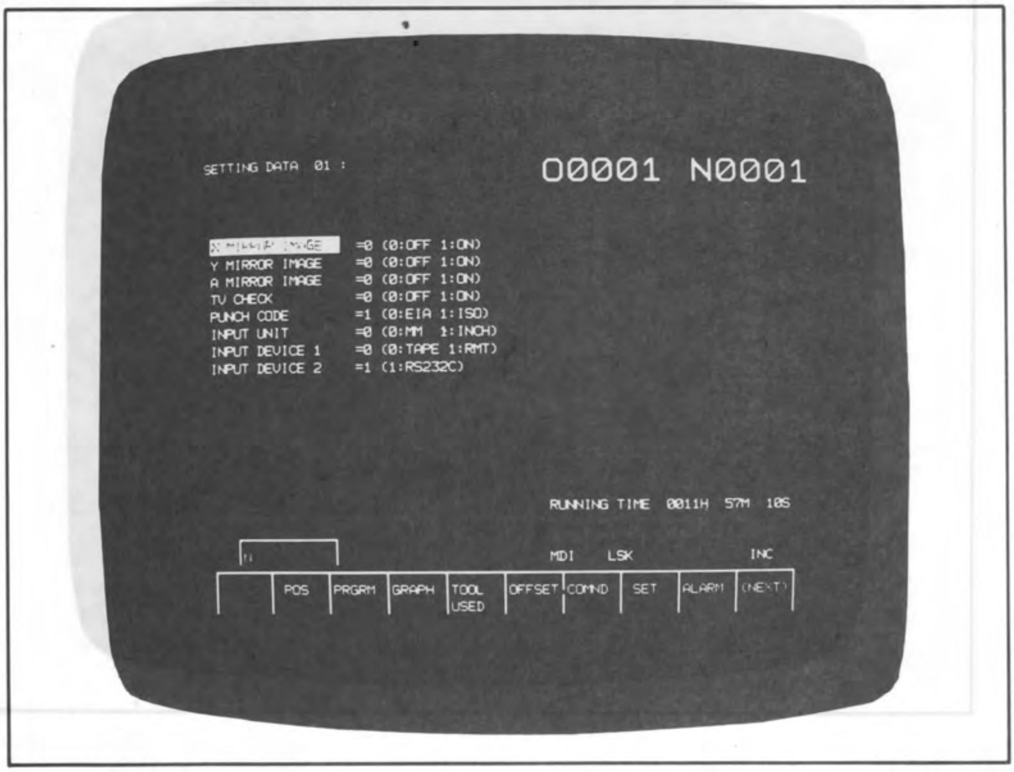


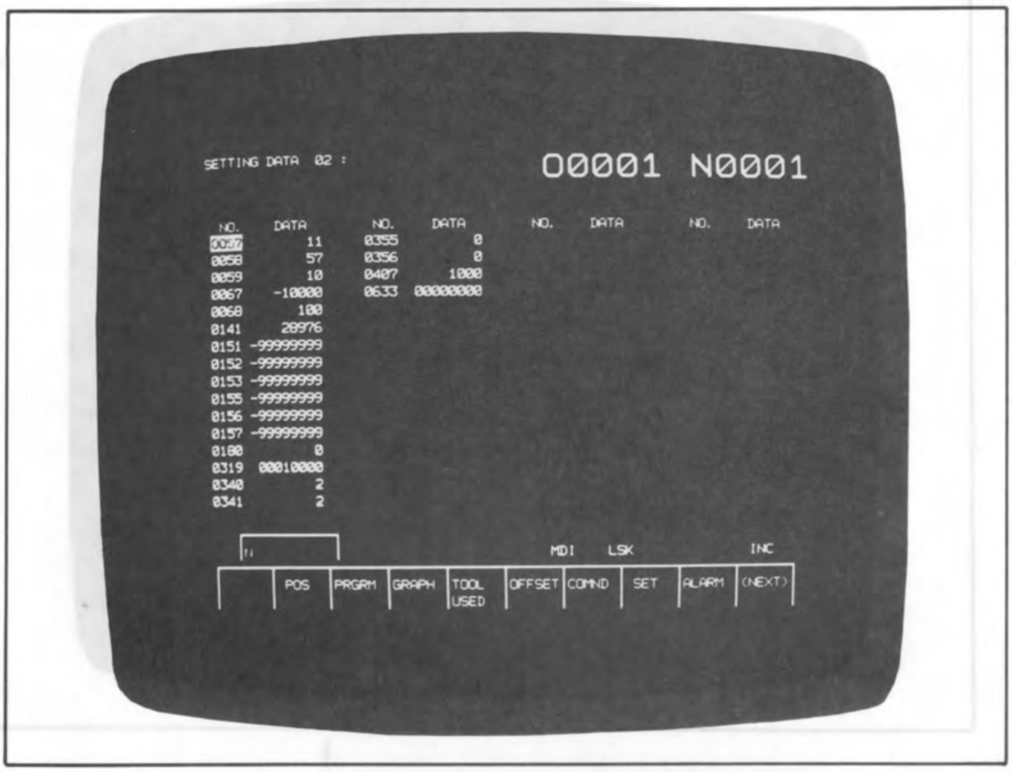
When depressing COMND software key, 9″ CRT's three screen display data appears on one screen as shown in the above figure.

The display is changed as shown below by depressing PAGE ↓ key.

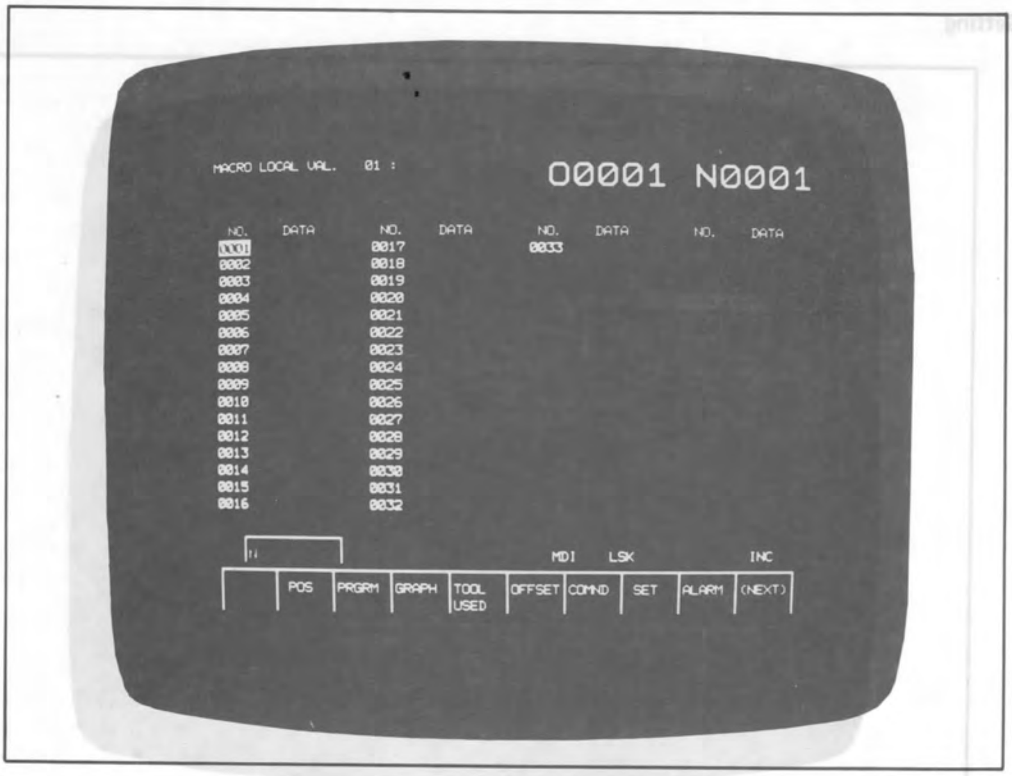


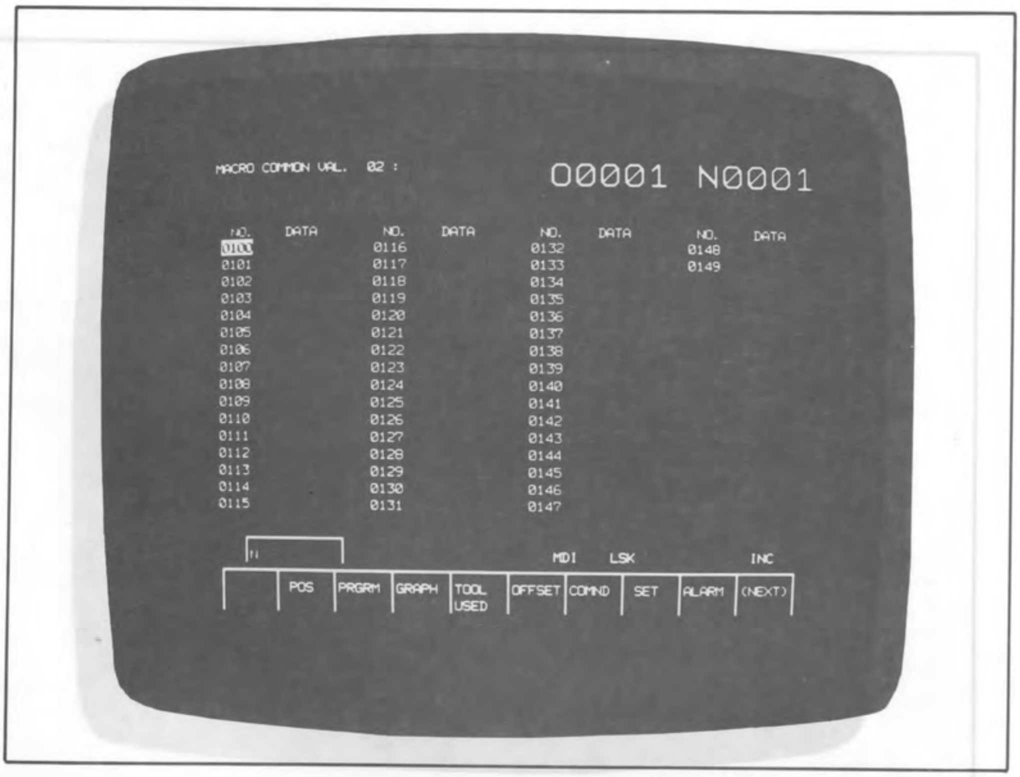
(5) Setting





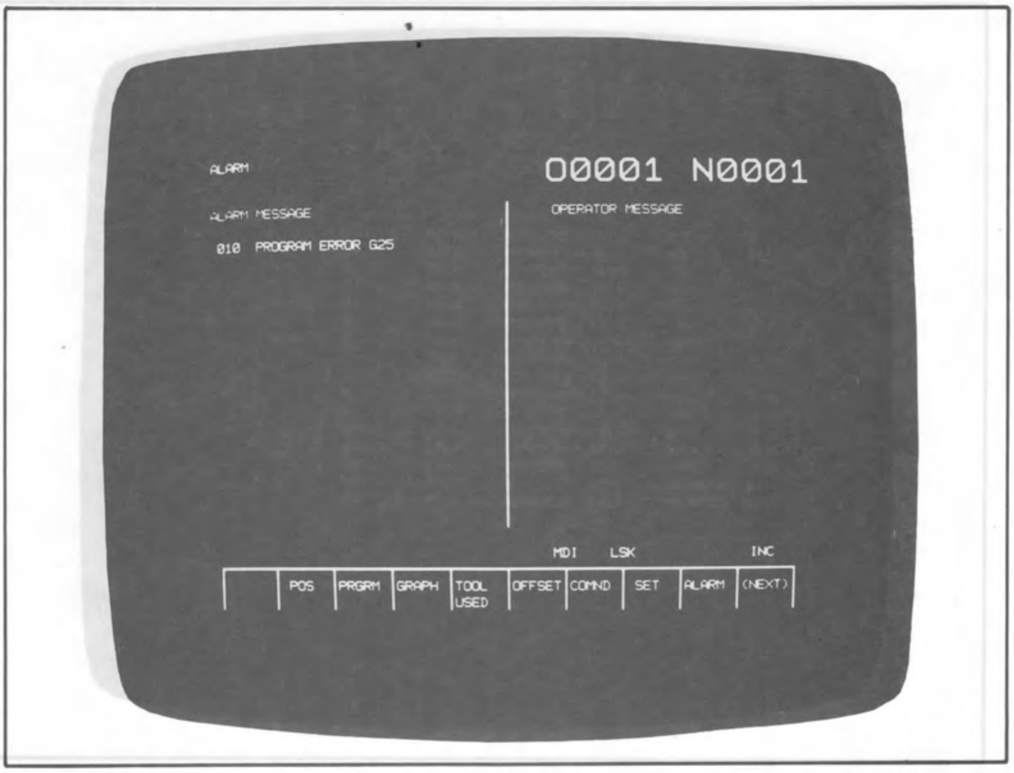
Various setting data are displayed by depressing SET software key. Running time is also displayed at setting 01.





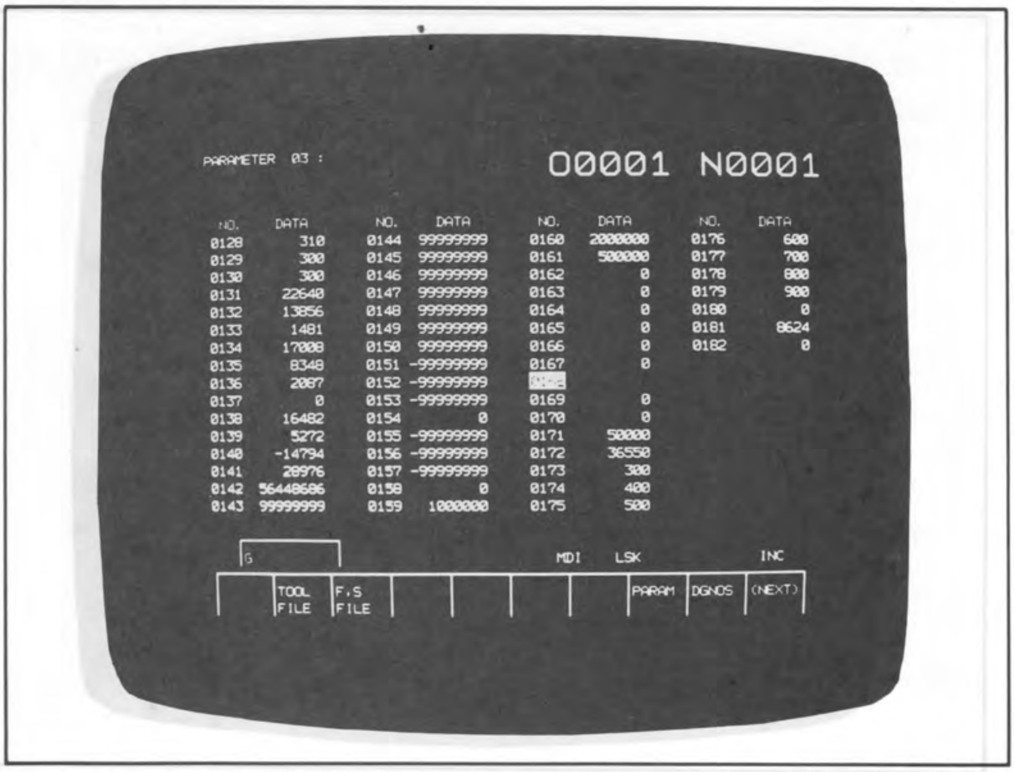
When depressing SET key again, custom macro local variables and common variables are displayed on the CRT screen.

(6) Alarm and operator message display



By depressing $\boxed{\text{ALARM}}$ software key alarm contents and instructions to an operator are displayed concurrently.

(7) Parameter display

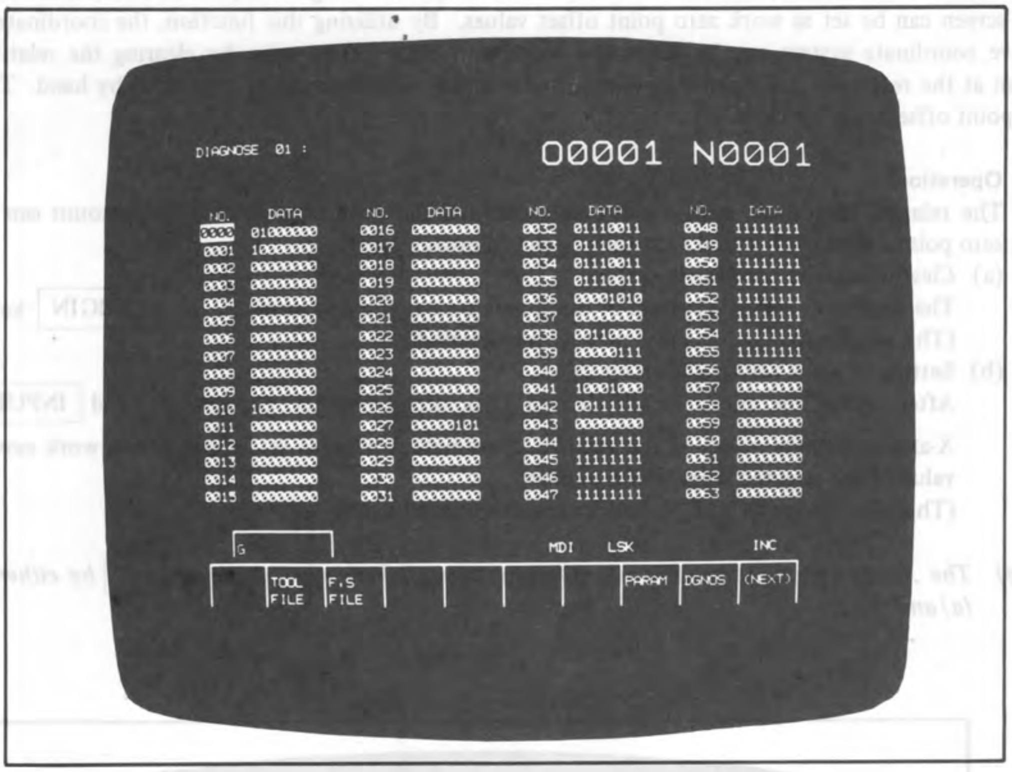


By depressing [(NEXT)] software key, software keys [PARAM] and [DGNOS] are displayed.
By depressing [PARAM] software key, 64 parameters appear on the CRT.

Display desired parameters by depressing PAGE [↑] [↓] and CURSOR [↑] [↓] keys.

306

(8) Diagnostic data display



Sixty-four diagnostic data are displayed by depressing DGNOS software key.

Display desired diagnostic data by depressing PAGE ↑ ↓ and CURSOR ↑ ↓ keys.

307

5.35.3 Direct input of measured work zero point offset values

The coordinate values of the relative coordinate system being indicated on the work zero point offset CRT screen can be set as work zero point offset values. By utilizing this function, the coordinate values of the relative coordinate system can be set as the work zero point offset value by clearing the relative coordinate system at the reference point and then moving the machine tool to the work zero point by hand. Thus, the work zero point offset value easily be set.

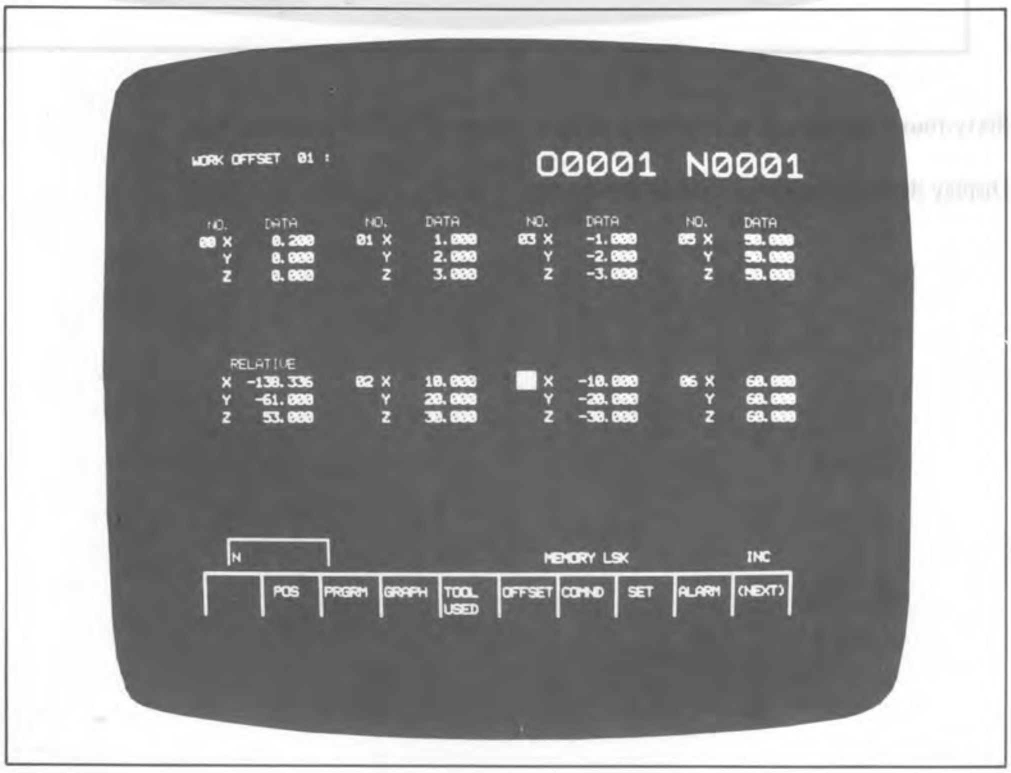
(1) Operation

The relative coordinate system can be cleared, and the work zero point offset amount can be set on the work zero point offset CRT screen according to the following operation.

(a) Clear of relative coordinate system
The relative coordinate system of the X axis is cleared by depressing [X] [ORIGIN] keys.
(This applies to Y, Z, 4, and 5 axes, correspondingly.)

(b) Setting of work zero point offset value
After setting the cursor to the desired work offset number, depress [X] and [INPUT] keys. The X-axis coodinate value of the relative coordinate system is set as the X-axis work zero point offset value of the selected work offset number.
(This also applies to Y, Z, 4, and 5 axes, correspondingly.)

(Note) The X display of the relative coordinate system flickers when depressing [*X*] *by either operation of (a) and (b).*



6. POSITION DISPLAY BY POSITION DISPLAY UNIT

The position display unit displays the present position. The position display reset buttons are attached to the position display unit, with one button for each axis.

When one of these buttons is pushed, the position display for the appropriate axis is cleared.

When programming of absolute zero point (G92) is set, this value can be set in the position display unit by a parameter setting (PPD).

(Note 1) With the DISPLAY LOCK switch at ON, position pulses are not applied to the position display unit. When shifting a coordinate system by manual operation, use this switches so as not enter that shift amount into the display.

(Note 2) In case of inch input, position display is displayed in inch format. In case of metric input, position display is displayed by metric system. When the input system is switched from inch or metric, the reset button should be pushed to make the indication on the position display zero. If the input system is changed while the data is being displayed, only the decimal point is shifted but the data remain unchanged.

(Note 3) The compensation data on the machine tool such as backlash compensation amount are not displayed on the position display unit.

7. OPERATION OF BUBBLE CASSETTE

7.1 General

By adding an input/output interface option, various data can be transferred between NC and bubble cassette (hereinafter referred to as cassette) via RS232C interface. The following data can be transferred in the same manner as in data transfer to and from paper tape reader/puncher.

(1) NC command data
(2) Offset data
(3) NC parameters (including pitch error compensation data)

7.2 Input/Output Operation

Data are transferred to and from the cassette by just the same manner as in paper tape reader/puncher, in principle, and you may presume that the cassette is attached to the RS232C interface instead of paper tape reader/puncher; provided that the cassette involves a concept as a file, and the following filing operation is added.

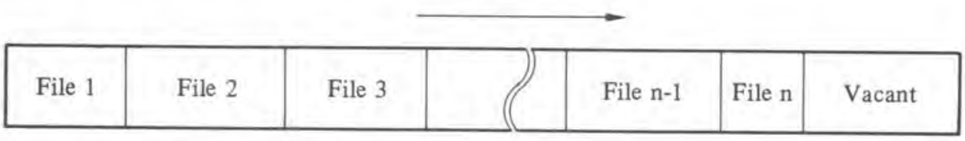
(1) Heading of file
(2) Deletion of file

7.3 What Is a File?

The unit of data to be transferred between NC and cassette by each input/output operation (by depressing READ or PUNCH button) is called "file". Let's consider an NC program output. If O100 PUNCH operation was made, a program having program No. 100 composes a file inside the cassette, and if O-9999 PUNCH (all program output) operation was made, some programs in NC memory compose a file.

Respective files are numbered with ascending file numbers as 2, 3, 4 . . . in the output order, starting with 1 as the start file No. These files correspond to data in the cassette by these file numbers hereafter.

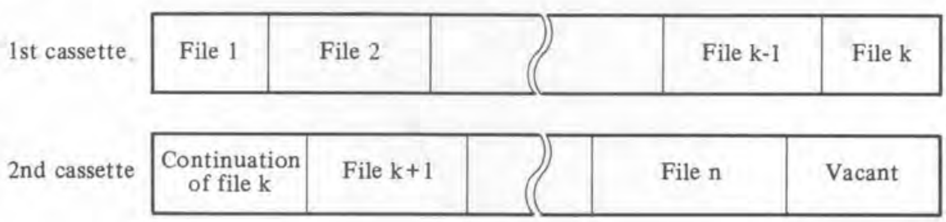
Since it is impossible to visually confirm the correspondence between file numbers and loaded data, it is recommended to enter a file number and its data contents into the "MEMO" column of the cassette at once, each time a data was outputted to the cassette.



MEMO (Example)

File 1	NC parameter
File 2	Offset data
File 3	NC program O0100
.	.
.	.
.	.
File n-1	NC program O0500
File n	NC program O0600

One file can extend over 2 cassettes as illustrated below.



7.4 Heading of File

When data are transferred to and from the cassette, it is necessary to designate a cassette data to be inputted or outputted. This designation is done by the file number, and this designating operation is called "heading of file".

Heading of a file is executed by the following procedure.
(1) Select the EDIT mode (or MEMORY mode).
(2) Select PRGRM function button.
(3) Input N , File No. , and INPUT in this order.
Heading is executed as shown in the following table according to the file numbers in (3).

	File number	Heading contents
1	0 (zero)	The cassette start is headed.
2	1 ~ 9999	A file having file No. 1 to 9999 is headed.
3	–9999	A file next to the file accessed just before is headed.
4	–9998	By this designation, N-9999 is automatically inserted before input/output when data is transferred to and from the cassette. In other words, the next file is headed without fail before data transfer, and data is transferred to and from the file. This designation is modal, and this state is cancelled by designating (1), (2), or (3) or by reset operation.

(Note 1) N0 and N1 designation brings the same result, if a file exists in the cassette. If no file exists, correct heading is not done by N1 designation because of the absence of the first file, while correct heading is done by N0 designation, since the start is headed, irrespective of whether a file is present or absent in the cassette.
If a new cassette is employed or if data are outputted from the start after making all existing files invalid, the heading is done by N0 designation.

(Note 2) No alarm is displayed on the CRT screen, even if the heading is not done correctly because of the absence of files, etc.
If data transfer operation (described later) is done hereafter, alarm 086 is displayed.

(Note 3) When continuously heading files, they can be headed every time by designating N1 ~ N9999 or by designating N1 ~ N9999 first to head the first file and then, by designating N-9999 for the subsequent files. Heading can be done correctly by these ways. However, the required time differs between the two. (The method of designating N-9999 is faster than the other.)

7.5 Deletion of File

A cassette file can be deleted by the following procedure.
(1) Select the EDIT mode.
(2) Select PRGRM of function buttons.
(3) Release the lock key ( ).
(4) Input N , file number , and START in this order. (Input a file number (1 ~ 9999) of the file to be deleted.)
By this operation, the k-th file inputted in (4) is deleted.

(Note 1) Be careful since the file number after deletion is added by –1 about old (k + 1) ~ n files to be k ~ (n – 1) files when the k-th file was deleted. (Change the description in the MEMO column described in 7.3)

File number before deletion		File number after deletion
1 ~ (k – 1)	→	1 ~ (k – 1)
k	→	Deletion
(k + 1) ~ n	→	k ~ (n – 1)

(Note 2) The cassette is provided with a write protect slide. Set this write protect slide upward, and make sure that the red lamp of the adapter lights when the cassette was inserted into the adapter, before strating the deletion.
No alarm is displayed, even if the operation for deletion is made without releasing the write protect slide. However, the deletion is not executed in this case.

7.6 Data Transfer to and from Cassette

Data can be transferred to and from the cassette in the same manner as in paper tape reader/puncher. However, be careful with the following items.

(1) Release the write protect slide of the cassette described in 7.5 (Note 2) without fail when data are outputted to the cassette. Alarm 086 is displayed, if data are outputted without releasing it.

(2) When data are inputted from the cassette, make sure that INPUT DEVICE 2 on page 1 is set to "1" (input from RS232C) when the function button [SET] is depressed.
If INPUT DEVICE 2 is set to "0", data are inputted from paper tape reader.

7.6.1 NC program transfer to and from cassette

(1) NC program output to cassette

An NC program can be outputted to the cassette by just the same operation as in punching to the paper tape puncher.

(1) Select the EDIT mode.
(2) Select [PRGRM] function button.
(3) Input [O], [program No] and [PUNCH] (1 program output) or [O], [–9999] and [PUNCH] (all programs output)

in this order.

By this operation, an NC program is outputted as a new file, after existing files in the cassette.

If the output operation to the cassette was made after heading of a file as described in 7.4, an NC program is outputted as a new file to the headed n-th file.

In this case, the previous state is reserved in 1 ~ (n – 1) files. However, old n-th file and its subsequent files are all deleted.

If an NC program is outputted to new cassette or it is outputted to the start of old cassette after making all existing files invalid, perform the above output operation after heading by N0 designation.

(Example) For [D] output;

(i) When no heading was made;

A	B	C	Vacant

⇩

A	B	C	D	Vacant

(ii) When heading was made;

Headed file (B)

A	B	C	Vacant

⇩

A	D	Vacant

(Note 1) If alarm 086 was produced during the output operation to a cassette, cassette data are restored to the state before output.
If alarm 086 occurred during output operation after heading the n-th file, cassette data are restored up to 1 ~ (n – 1) files.

(2) NC program input from cassette

An NC program can be inputted from the cassette by just the same operation as in reading from paper tape reader.

(i) Select EDIT mode (or MEMORY mode)
(ii) Head a file in which a desired program is loaded (See 7.4).
(iii) Input [O], [–9999], and [READ] (normal) or [O], [program number], and [READ] [when to change the order of inputting programs] in this order.

By this operation, an NC program in the headed file is read into NC memory.

(iv) By selecting the function button [PRGRM], the read program is displayed on CRT.

(Example) For reading files 2 to 4 in cassette into memory;

(Operation example 1)		(Operation example 2)		(Operation example 3)		
N2	[INPUT]	N2	[INPUT]	N2	[INPUT]	—— Heading of file 2
0-9999	[READ]	0-9999	[READ]	0-9999	[READ]	—— Reading of file 2
N3	[INPUT]	N-9999	[INPUT]	N-9998	[INPUT]	—— Heading of file 3
0-9999	[READ]	0-9999	[READ]	0-9999	[READ]	—— Reading of file 3
N4	[INPUT]	N-9999	[INPUT]			—— Heading of file 4
0-9999	[READ]	0-9999	[READ]	0-9999	[READ]	—— Reading of file 4
Slow		Quick		Quick		—— Heading time

7.6.2 Offset data transfer to and from cassette

(1) Offset data output to cassette

Offset data can be outputted to the cassette by just the same operation as in punching to paper tape puncher.

(i) Select EDIT mode.

(ii) Select function button [OFSET].

(iii) Input [P], [–9999], and [PUNCH] in this order.

By this operation, offset data are outputted to the cassette. The file, to which offset data are outputted, is determined in the same manner as in NC program output. (See 7.6.1 (1))

(2) Offset data input from cassette

Offset data can be inputted from the cassette, and set to the NC offset memory by the following operation.

(i) Select EDIT mode.

(ii) Head a file which contains offset data (See 7.4.).

(iii) Input [O], [program number], and [READ] in this order.

If offset data in the file are those outputted in (1), no program number is loaded at the start of data, and a program number must be designated, otherwise alarm 075 is displayed.

By this operation, offset data is read into NC memory as an NC program.

(iv) By selecting the function button [PRGRM], the read offset data (G10P_R_; format) is displayed on CRT.

(v) When this program is executed by selecting the MEMORY mode and cycle start, offset data is set to the NC offset memory.

(vi) Select the [OFSET] function button, and check if the offset data has been set correctly or not.

7.6.3 NC paramter transfer to and from cassette

(1) NC parameter output to cassette

NC parameters can be outputted to the cassette by just the same operation as in punching to paper tape puncher.

(i) Select the EDIT mode.

(ii) Select function button [PARAM].

(iii) Input [P], [–9999], and [PUNCH] in this order.

The NC parameter is outputted to the cassette by this operation. The file, to which the NC parameter is to be outputted, is determined in the same manner as in NC program output. (See 7.6.1 (1))

(2) NC parameter input from cassette

NC parameters can be inputted from the cassette by just the same operation as in reading from paper tape reader.

(i) Head a file which loads the NC parameter. (See 7.4)

(ii) Open the front door of NC, and select the parameter protect switch on the master printed circuit board to ENABLE.

Alarm 100 is displayed.

(iii) Depress the EMERGENCY STOP button on the machine operator's panel.

(iv) Select function button [PARAM].

(v) Input P , –9999 , and READ in this order.
By this operation, the NC parameter is read from the cassette, and set into NC parameter memory.
After reading the parameters, alarm 000 is produced usually.
(vi) Reset the parameter protect switch to DISABLE.
(vii) Release the EMERGENCY STOP button on the machine operator's panel.

(Note 1) Don't execute this operation by end user, in principle.

7.7 Cassette Exchange Request

If data transfer to and from the first cassette is terminated when one file extends over 2 cassettes (See 7.3), the red and green lamps of the cassette adapter alternately flicker to inform the operator of the cassette exchange request.

When these lamps flicker, unload the first cassette, and insert the second cassette. After the second cassette has been inserted, data transfer is automatically continued.

This cassette exchange request is issued, if the second cassette is required halfway in the course of the following operation.

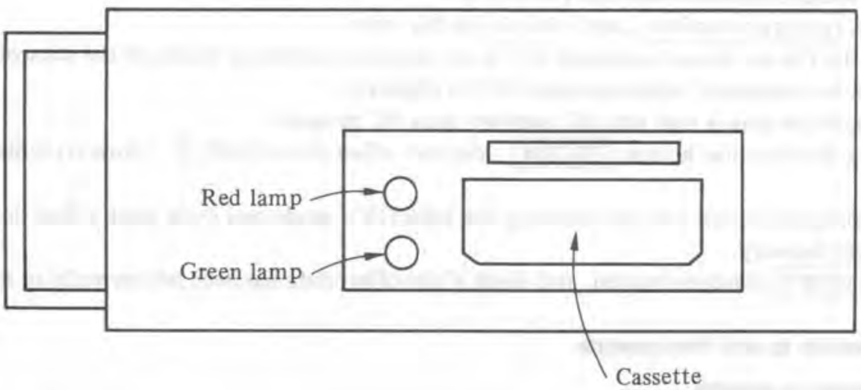
(1) Heading of file
(2) Deletion of file
(3) Data transfer between NC and cassette

(Note 1) Since all cassette exchange processing is done on the cassette adapter side, no particular operation is needed on the NC. You have only to replace the cassette.
(During this time, data transfer processing is being interrupted on the NC.)

(Note 2) Even if NC is reset during the cassette exchange request, it is not reset at once, but it is reset after cassette exchange. The cassette exchange is an only means of releasing the cassette exchange request, accordingly.

7.8 Cassette Adapter Lamp Conditions

The red and green lamps of the cassette adapter indicate operating conditions.



[Cassette adapter as viewed form the top.]

	Lamp conditions	Operating conditions
1	Red lamp lights	Writable: This lamp lights when cassette is inserted after setting the write protect slide of cassette upward.
2	Green lamp lights	Readable: This lamp lights when cassette is inserted.
3	Red lamp flickers	Write in progress.
4	Green lamp flickers	Read or heading of file in progress.
5	Red and green lamps alternately flicker.	Cassette is not inserted, or cassette exchange request is in progress.
6	Red and green lamps concurrently flickers.	File deletion in progress.

314

(Note 1) *All alarms regarding data transfer to and from cassette are displayed as alarm 086. Locate a cause of an alarm, and release it by examining the cassette conditions and operation.*

(Note 2) *When a cassette adapter is connected, to ro disconnected from NC, both NC and cassette adapter power must be turned off.*

7.9 Parameters and Setting on Data Transfer to and from Cassette

Set parameters and setting as described below when data are transferred to and from the cassette.

(1) Selection of input device when data are inputted by RS232C interface

340	3

(2) Selection of output device when data are outputted by RS232C interface

341	3

Set input/output device No. 3 by setting in (1) and (2).

(3) Input/output parameters of input/output device 3 (cassette adapter)

312	1	0	0	1	1	0	0	1
	7	6	5	4	3	2	1	0

Bits 3–0: Baud rate 4800

Bit 4: Stop bit (2 bits)

Bit 5: Control code is used.

Bit 7: Neither start and final feed codes of data nor inter-program space code is outputted.
(This bit is added for effective use of cassette memory.)

(3) is set by a parameter.

V. MAINTENANCE

1. DAILY MAINTENANCE

1.1 Tape Reader Cleaning

(a) Tape reader without reels

Item	Cleaning point	Reference drawing	Cleaning period	Cleaning method
1	Surface of read head (light sensing part)	Fig. 1.1 (a) ①	Daily	Clean with gauze or a thin brush with absolute alcohol.
2	Surface of read head (light emitting part)	Fig. 1.1 (a) ②	Daily	
3	Tape retainer	Fig. 1.1 (a) ③	Daily	
4	Tape path	Fig. 1.1 (a) ④	Daily	
5	Capstan roller	Fig. 1.1 (a) ⑤	Weekly	
6	Guide roller	Fig. 1.1 (a) ⑥	Weekly	
7	Pinch roller	Fig. 1.1 (a) ⑦	Weekly	
8	Assembly under tape path plate	Fig. 1.1 (a) ⑧	Monthly	Clean with a cloth or brush.
9	Inside tape reader cover	Fig. 1.1 (b) ⑨	Monthly	

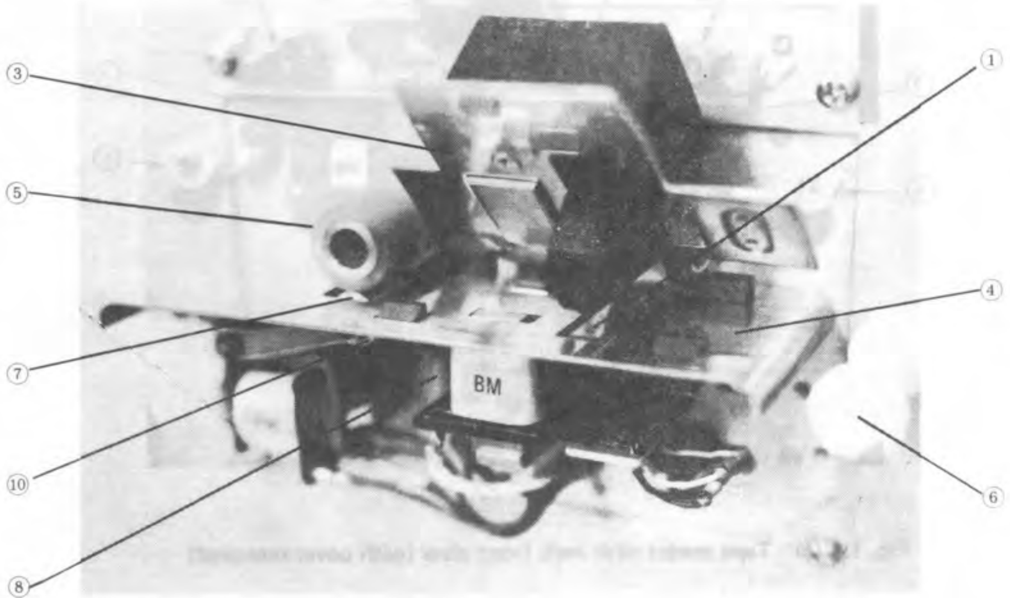


Fig. 1.1 (a) Tape reader without reels front view (with cover removed)

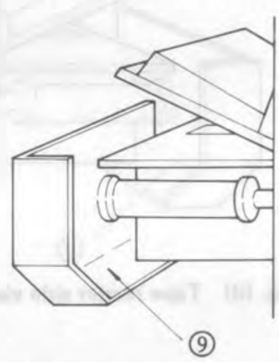


Fig. 1.1 (b) Tape reader side view

(b) Tape reader with reels

Item	Cleaning point	Reference drawing	Cleaning period	Cleaning method
1	Surface of read head (light sensing part)	Fig. 1.1 (c) ①	Daily	Clean with gauze or a thin brush with absolute alcohol.
2	Surface of read head (light emitting part)	Fig. 1.1 (c) ②	Daily	
3	Tape retainer	Fig. 1.1 (c) ③	Daily	
4	Surface of tape path	Fig. 1.1 (c) ④	Daily	
5	Capstan roller	Fig. 1.1 (c) ⑤	Weekly	
6	Guide roller	Fig. 1.1 (c) ⑥	Weekly	
7	Pinch roller	Fig. 1.1 (c) ⑦	Weekly	
8	Assembly under tape path plate.	Fig. 1.1 (c) ⑧	Monthly	Clean with a cloth or a brush.
9	Inside tape reader cover	Fig. 1.1 (d) ⑨	Monthly	

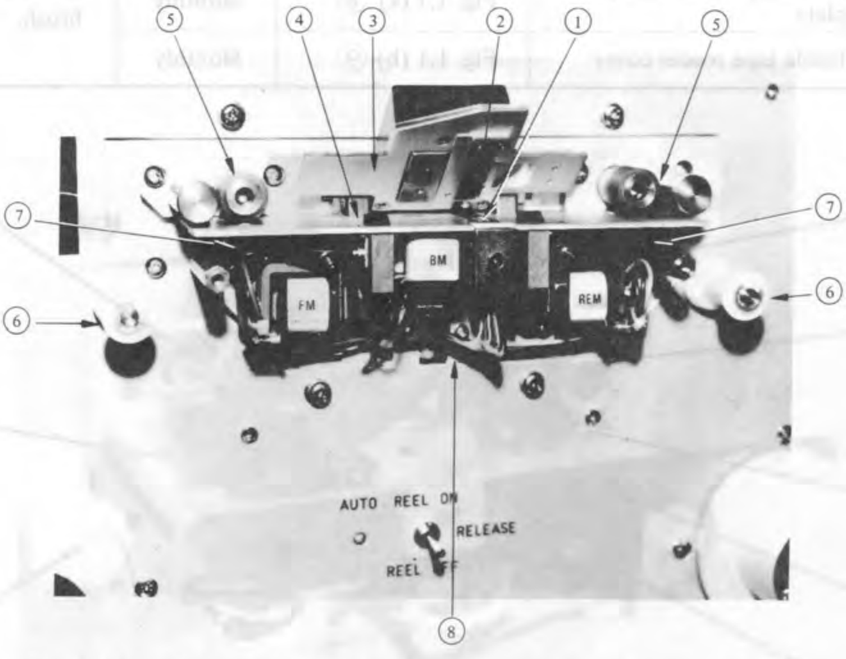


Fig. 1.1 (c) Tape reader with reels front view (with cover removed)

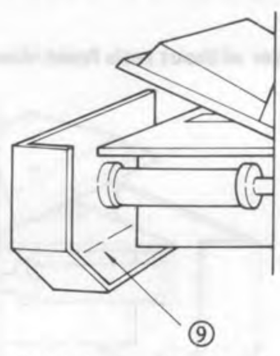
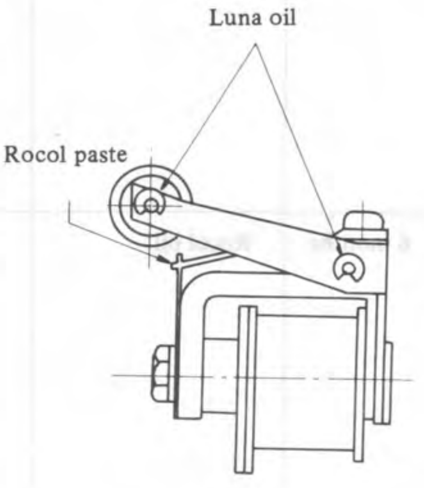


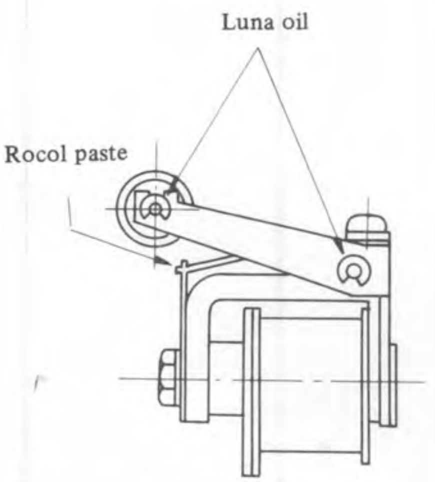
Fig. (d) Tape reader side view

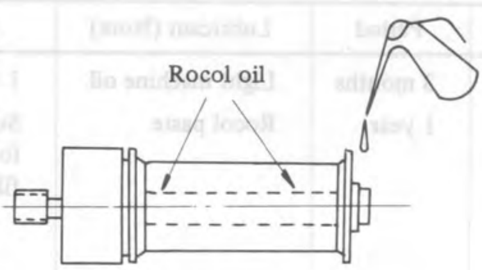
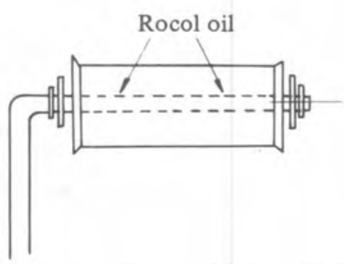
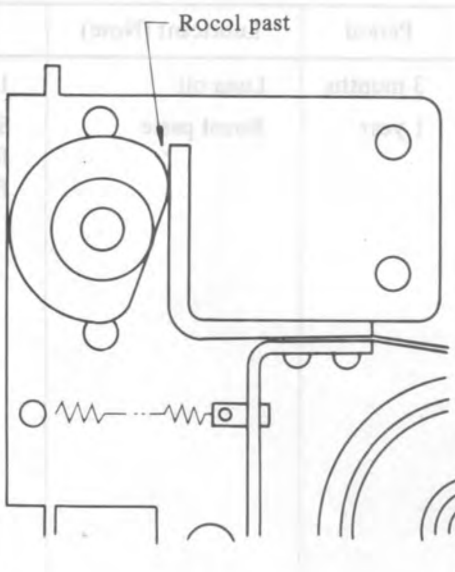
1.2 Tape Reader Lubrication

(a) Tape reader without reels
Routine lubrication points and lubrication periods are as follows:

Item	Lubrication point	Period	Lubricant (Note)	Amount
1	Magnet section  See Fig. 1.1 (a) ⑩	3 months 1 year	Light machine oil Rocol paste	1 drop Sufficient to form a thin film

(b) Tape reader with reels
Routine lubrication points and lubrication periods are as follows:

Item	Lubrication point	Period	Lubricant (Note)	Amount
1	Magnet section  See Fig. 1.2 (b) ①	3 months 1 year	Luna oil Rocol paste	1 drop Sufficient to form a thin film

Item	Lubrication point	Period	Lubricant (Note)	Amount
2	Guide roller  See Fig. 1.2 (b) ②	6 months	Rocol oil	2 ~ 3 drops
3	Tension arm guide roller  See Fig. 1.2 (c) ③	6 months	Rocol oil	2 ~ 3 drops
4	Cam  See Fig. 1.2 (d) ④	3 months	Rocol paste	Sufficient to form a thin film

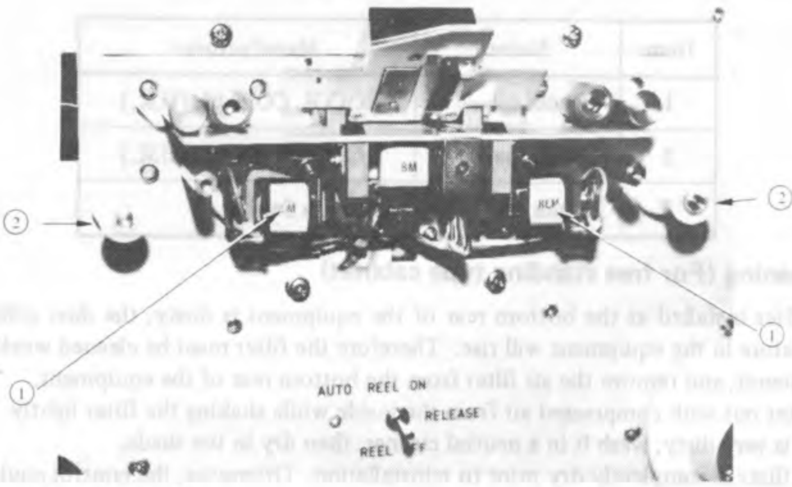


Fig. 1.2 (b) Tape reader with reels front view (with cover removed)

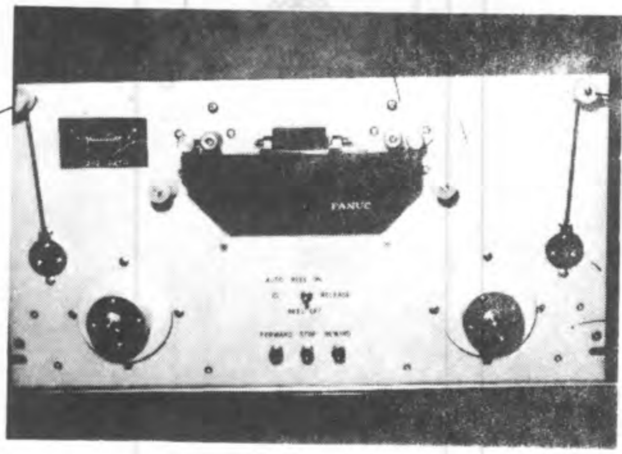


Fig. 1.2 (c) Tape reader with reels front view

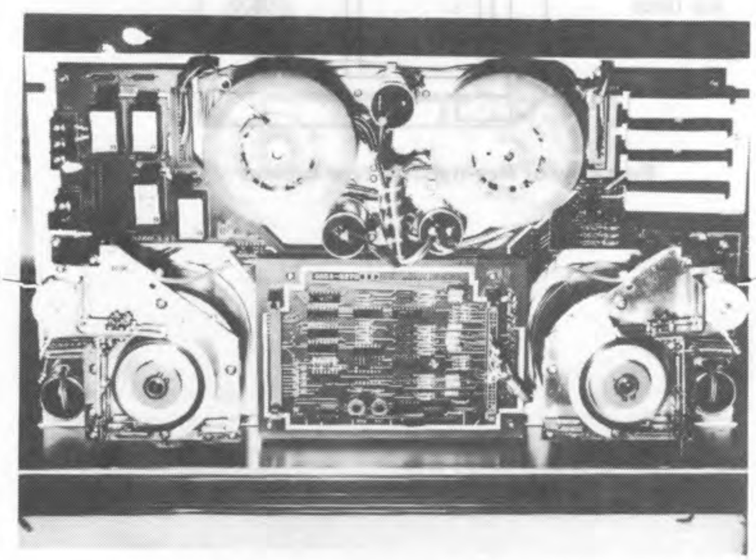


Fig. 1.2 (d) Tape reader with reels rear view

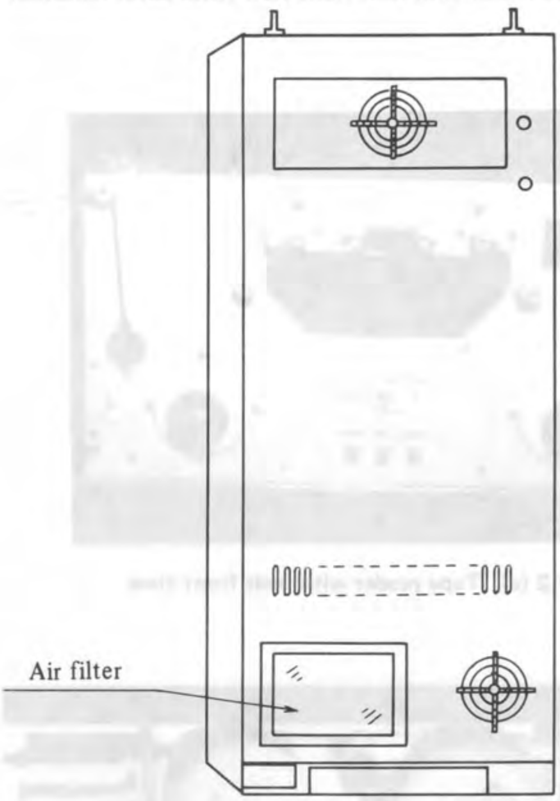
NOTE: Lubricants

Item	Name	Manufacturer
1	Rocol oil	ROCOL CO. Ltd. (U.K.)
2	Rocol paste	ROCOL CO. Ltd. (U.K.)
3	Luna oil	Nippon Sekiyu

1.3 Air Filter Cleaning (For free standing type cabinet)

When the air filter installed at the bottom rear of the equipment is dusty, the dust collection efficiency will drop, and the temperature in the equipment will rise. Therefore the filter must be cleaned weekly, as follows:

(1) Remove the fastener, and remove the air filter from the bottom rear of the equipment.
(2) Blow the air filter out with compressed air from the inside while shaking the filter lightly.
(3) When the filter is very dirty, wash it in a neutral cleaner, then dry in the shade.
(4) Insure that the filter is completely dry prior to reinstallation. Otherwise, the control could be damaged.



Rear view of free-standing type cabinet

2. FUSE CHECK AND REPLACEMENT

When a fuse blows inside the NC unit, find and correct the cause of the failure and replace the fuse. Fuses used in the NC unit are as follows:

2.1 Input Unit

The following two kinds of input units are provided.

(1) For control unit .... Fig. 2.1 (a)
(2) For control unit and servo .... Fig. 2.1 (b)

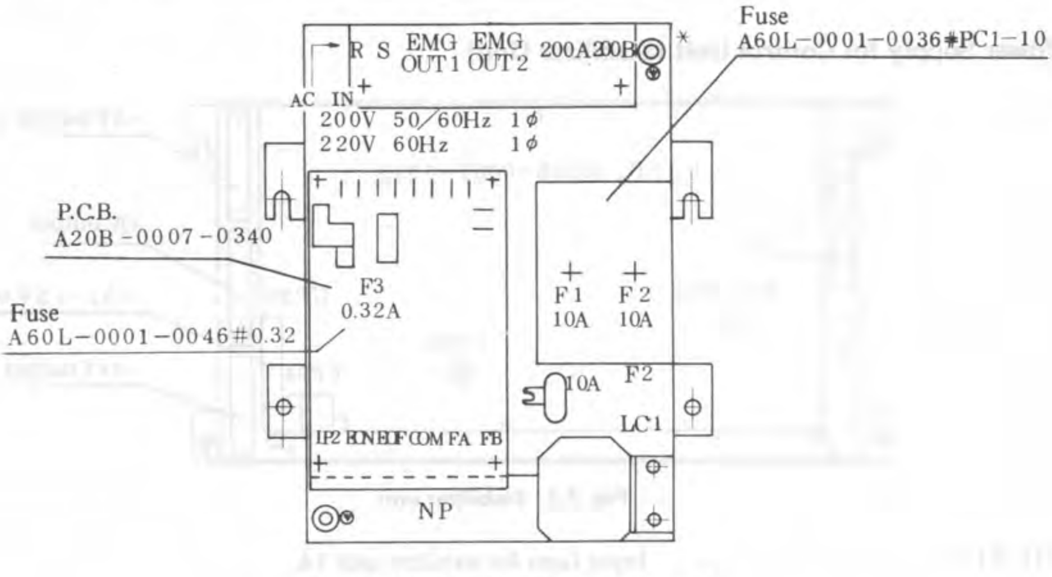


Fig. 2.1 (a) Input unit for control unit

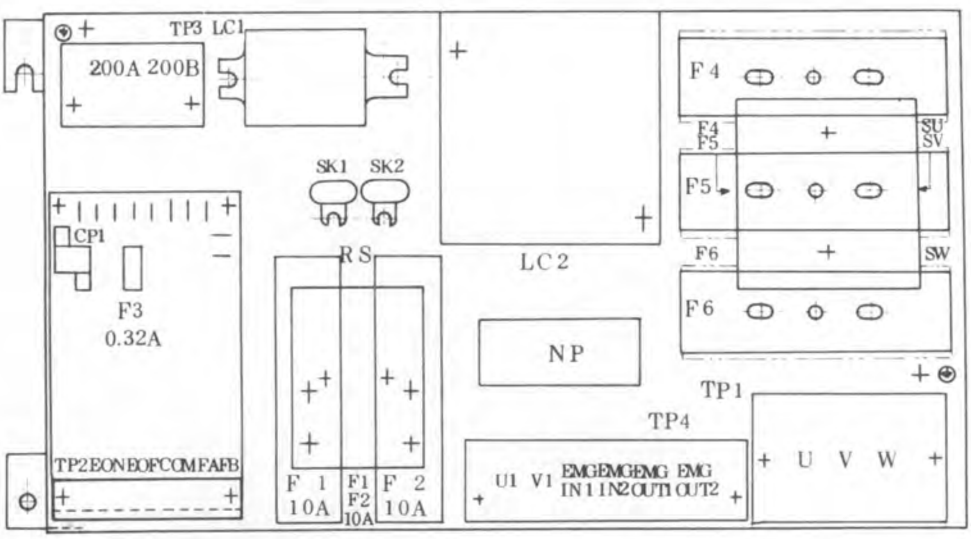


Fig. 2.1 (b) Input unit for control unit and servo unit

F1, F2 .... Input fuses for control unit 10A
F3 .... Fuse for power on/off control circuit 0.32A
F4 ~ F6 .... Input fuses for servo transformer

The rating of fuses used for the servo transformer primary input are listed below.

Power voltage	Fuse type / Transformer capacity (KVA)	Utsunomiya Electric Co., Ltd. PC type	Fuji Electric Co., Ltd. FCF type
200V	1.5KVA	15A	20A
220V	2.5	20	30
~	5	30	30
500V	10	40	40

2.2 Power Supply for Control Unit (Stabilizer Unit)

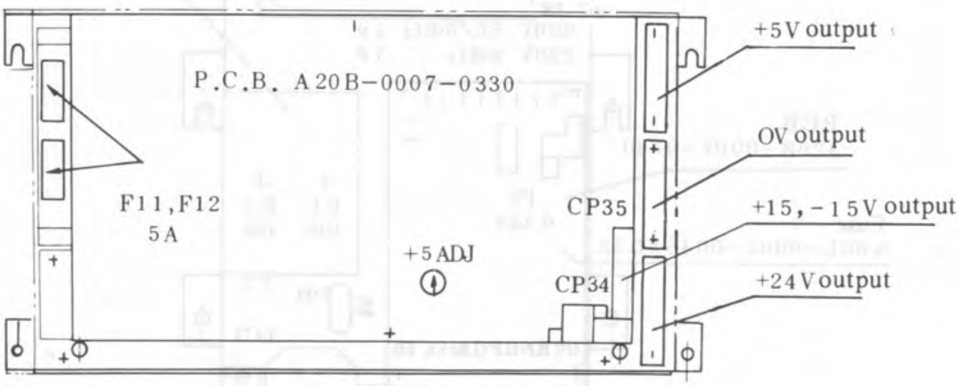


Fig. 2.2 Stabilizer unit

F11, F12 .... Input fuses for stabilizer unit 5A

3. CHECK AND REPLACEMENT OF DC MOTOR BRUSH

A periodic check and replacement of the brushes should be performed in the following way.

(1) Check interval

General machine tools
(Lathe, Milling machine, Machining center, etc.) .... Every year

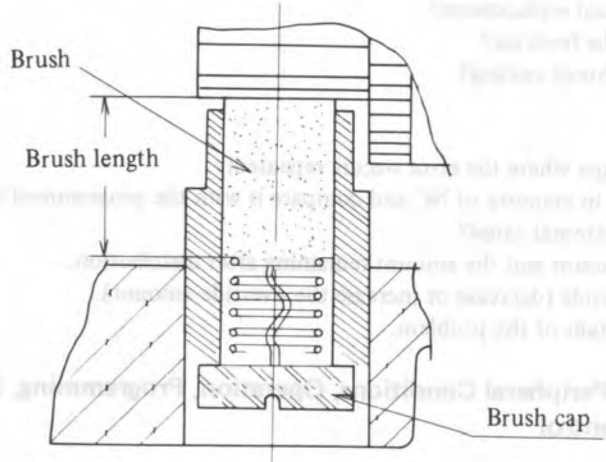
Machine tools with high frequency of acceleration/deceleration
(Turret punch press, etc.) .... Every two months

(2) Standard of brush replacement

When the length of the brush becomes shorter than 10 mm, the brush should be changed.

(3) Method of replacement

Remove the brush cap (screw) and take out the brush.
Thighten the brush cap fully when replacing it.



(Note) Replacement brushes can be obtained from our sales offices.
Part number: A290-0641-V001 (For model 0M and 5M).
A290-0651-V001 (For model 10M, 20M and 30M)

4. TROUBLESHOOTING

4.1 General

(1) Check the type of problem:

In what mode is the controller?
What is displayed on the MDI & CRT?
Is there a positioning error? If so, on which axis and by what amount?
Is there a tool path error? If so, by what amount?
Is the speed abnormal?
Is the problem in an auxiliary function?
What is the alarm number?

(2) Check the frequency of occurence:

When did the problem occur? What is its frequency? (Was another machine also being operated?)
What is the frequency on the same workpieces?
Which program is it ; What is the sequence number?
Is the problem related to a specific mode?
Is the problem related to tool replacement?
Is the problem related to the feedrate?
Is the problem related to thread cutting?

(3) For re-occuring problems:

Go through the program tape where the error occurs repeatedly.
Check the numerical value in memory of NC and compare it with the programmed numerical value.
Is the problem due to an external cause?
Check the stored offset amount and the amount remaining after distribution.
Check the response to override (decrease or increase the override amount).
Ask the operator about details of the problem.

4.2 Checking Input Voltage, Peripheral Conditions, Operation, Programming, Drives, Machine and Interface Control

(1) Check the input voltage:

Are there fluctuations in the input voltage?
Is there a drop in the input voltage?
Is front or rear door opened (door interlock)?
Is there some other device using large amounts of current?
Is there an electric discharge machine or welding machine nearby?

(2) Check peripheral conditions:

What is the temperature of the controller? Did the temperature change? Is it excessive?
Is the filter dirty?
Is the tape reader dirty?
Is there oil or cutting fluid about?
Are there any vibrations?
Is the unit in direct sunlight?

(3) Check for any external causes:

Has the machine recently been repaired or adjusted?
Has the magnetics cabinet recently been repaired or adjusted?
Has the NC unit recently been repaired or adjusted?
Is there a source of noise nearby?
(Example: Cranes, High-frequency machines, Electric discharge machines)
Has a new machine been mounted nearby?
Is the another NC with the same problem?
Has the user adjusted the NC?
Has the same problem occurred before?

(4) Check phases of operation

Has the operator been properly instructed?
Has the operator been replaced?
Is the operator familiar with the program?
Does the program finish too early or was it interrupted?
Does the program contain an incremental command?
Is the tool compensation value correctly set?
If the tool compensation value was changed, was it done correctly?
Does the machine change to another mode of operation?
Is the block skip function used correctly?
Is the tape set correctly?
Are there any tape coding errors?
Has the machine tool been operated incorrectly.

(5) Check punched tapes?

Is there dirt on the tape?
Are there any folds or wrinkles on the tape?
Are joints normal?
Did the program previously run normally?
Was the tape copied from a master tape?
Was it the punched tape?
Was the tape correctly punched?
Is the tape puncher normal?
Was the wrong tape used? (For example, colored tape?)
Is the puncher in good condition?
Was black tape used?

(6) Check the program

Is the program new?
Was the program created according to the OPERATOR'S MANUAL?
Are addresses in the right order?
Does the problem occur in any specific block?
Are the correct speed and lead values set for thread cutting?
Is there space at the beginning and end of thread cutting?
Does the problem occur in the sub-program?
Was the list of tapes created for checking?

(7) Check for changes in operation

Has any change or adjustment been made in the operation procedure?
Has a fuse been blown?
Is the NC in the emergency stop status?
Is the machine tool ready?
Is the NC in the alarm status?
Is the MODE SELECT switch set correctly?
Is the switch on the tape reader set correctly?
Is the override switch set to the zero position?
Is the NC in the machine lock status?
Is the feed hold button pushed?

(8) Check the machine itself

Is the machine properly installed?
Does vibration occur during operation?
Is the tool tip normal?
Is there any offset due to tool exchange?
Is there sufficient backlash compensation?
Are there distortions in any part of the machine due to temperature changes?
Was the workpiece measured correctly?
Was the measurement made at a constant temperature? (1 meter of steel changes 10μ in length at a temperature change of 1 °C)

Are the cables normal (bent, broken or damaged)?
Are the signal lines and power lines separated?

(9) Check the interface control
Are power lines and NC cables mounted separately?
Is the shield normal? Is a noise suppressor attached to the relay, solenoid, and motor?

4.3 NC System Check (No tools required)

(1) Check control unit external conditions
Is there damage to the cabinet?
Is the MDI & DPL unit normal?
Is the filter clean?
Is the reader clean?
Is the door of the reader closed?
Was operation made with the door open?
Check that chips accumulated on the cabinet did not fall inside when the door was opened.

(2) Check the tape reader
Is the tape reader dirty?
Do the feed and the brake magnets operate normally?

(3) Check inside the control unit
Is there dirt in the control unit?
Is the fan motor normal?
Is there corrosion?

(4) Check the power unit
Is the unit correctly connected?
Are all fuses OK?
Is the circuit breaker normal?
Is the voltage within the allowable range?
Are the shield and cable duct grounded correctly?
Is the wiring path OK?
Are all terminals fully tightened?

(5) The grounding
Is the grounding connection OK?
Is the shield ground OK?

(6) Check all cables
Are cable connectors fully pushed in?
Are there any abnormalities in internal cables?
Are there any abnormalities in external cables?
Are there any scratches, bends or breaks?

(7) Check printed circuit boards
Are all PCBs mounted properly?
Is the plug connector OK?
Are physical conditions normal (no distortions, etc.)?
What is the PCB edition?
Are connections between printed circuit boards good?

(8) Check the MDI & DPL unit:
Do the push buttons operate normally?
Is the tape cable normal?

4.4 NC Status

When it seems that the NC is executing nothing even during cycle operating status, the current status of the NC is displayed on CRT screen at the diagnostic number 700 or 701 by selecting DIAGNOSTIC page using DGNOS function button.

	7	0	0

	CSCT	CITL	COVZ	CINP	CDWL	CMTN	CFIN
7	6	5	4	3	2	1	0

When a digit is a 1, the corresponding status is effective.

CFIN: The M, S, or T function is being executed.
CMTN: A move command in the cycle operation is being executed.
CDWL: Dwell is being executed.
CINP: An in-position check is being executed.
COVZ: Override is at 0%.
CITL: STLK or Interlock is on.
CSCT: The controller is waiting for the speed arrival signal of the spindle to turn on.

	7	0	1

		CRST				CTRD	
7	6	5	4	3	2	1	0

CTRD: The controller is reading the NC command from the tape reader.
CRST: One of the following: emergency stop, remote reset, reset & rewind or the reset button on the MDI & DPL panel is on.

APPENDIXES

APPENDIX 1 TAPE CODES USED IN PROGRAMMING

ISO code										EIA code										Meaning
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		
0			○	○		∘				0			○			∘					Numeral 0
1	○		○	○		∘			○	1						∘			○		Numeral 1
2	○		○	○		∘		○		2						∘		○			Numeral 2
3			○	○		∘		○	○	3				○		∘		○	○		Numeral 3
4	○		○	○		∘	○			4						∘	○				Numeral 4
5			○	○		∘	○		○	5				○		∘	○		○		Numeral 5
6			○	○		∘	○	○		6				○		∘	○	○			Numeral 6
7	○		○	○		∘	○	○	○	7						∘	○	○	○		Numeral 7
8	○		○	○	○	∘				8					○	∘					Numeral 8
9			○	○	○	∘			○	9				○	○	∘			○		Numeral 9
A		○				∘			○	a		○	○			∘			○		Address A
B		○				∘		○		b		○	○			∘		○		?	Address B
C	○	○				∘		○	○	c		○	○	○		∘		○	○	?	Address C
D		○				∘	○			d		○	○			∘	○				Address D
E	○	○				∘	○		○	e		○	○	○		∘	○		○		Address E
F	○	○				∘	○	○		f		○	○	○		∘	○	○			Address F
G		○				∘	○	○	○	g		○	○			∘	○	○	○		Address G
H		○			○	∘				h		○	○		○	∘				?	Address H
I	○	○			○	∘			○	i		○	○	○	○	∘			○		Address I
J	○	○			○	∘		○		j		○		○		∘			○	?	Address J
K		○			○	∘		○	○	k		○		○		∘		○			Address K
L	○	○			○	∘	○			l		○				∘		○	○		Address L
M		○			○	∘	○		○	m		○		○		∘	○				Address M
N		○			○	∘	○	○		n		○				∘	○		○		Address N
O	○	○			○	∘	○	○	○	o		○				∘	○	○			Address O
P		○		○		∘				p		○		○		∘	○	○	○		Address P
Q	○	○		○		∘			○	q		○		○	○	∘					Address Q
R	○	○		○		∘		○		r		○			○	∘			○		Address R
S		○		○		∘		○	○	s			○	○		∘		○			Address S
T	○	○		○		∘	○			t			○			∘		○	○		Address T
U		○		○		∘	○		○	u			○	○		∘	○				Address U
V		○		○		∘	○	○		v			○			∘	○		○	?	Address V
W	○	○		○		∘	○	○	○	w			○			∘	○	○			Address W
X	○	○		○	○	∘				x			○	○		∘	○	○	○		Address X
Y		○		○	○	∘			○	y			○	○	○	∘				?	Address Y
Z		○		○	○	∘		○		z			○		○	∘			○		Address Z
DEL	○	○	○	○	○	∘	○	○	○	Del		○	○	○	○	∘	○	○	○	*	Delete (cancel an error punch).
NUL						∘				Blank						∘				*	Not punched. Can not be used in significant section in EIA code.
BS	○				○	∘				BS			○		○	∘		○		*	Back space
HT					○	∘			○	Tab			○	○	○	∘	○	○		*	Tabulator
LF or NL					○	∘		○		CR or EOB	○					∘					End of block
CR	○				○	∘	○		○											*	Carriage return
SP	○		○			∘				SP				○		∘				*	Space
%	○		○			∘	○		○	ER					○	∘		○	○		Absolute rewind stop
(			○		○	∘				(2–4–5)				○	○	∘		○			Control out (a comment is started)
)	○		○		○	∘			○	(2–4–7)		○			○	∘		○			Control in (the end of a comment)
+			○		○	∘		○	○	+		○	○	○		∘				*	Positive sign
–			○		○	∘	○		○	–		○				∘					Negative sign
:			○	○	○	∘		○													Colon
/	○		○		○	∘	○	○	○	/			○	○		∘			○		Optional block skip
.			○		○	∘	○	○		.		○	○		○	∘		○	○		Period (A decimal point)
#	○		○			∘		○	○											*	Sharpe
$			○			∘	○													*	Dollar sign
&	○		○			∘	○	○		&					○	∘	○	○		*	Ampersand
’			○			∘	○	○	○											*	Apostrophe
*	○		○		○	∘		○												*	Asterisk
,	○		○		○	∘	○			,			○	○	○	∘		○	○	*	Comma
;	○		○	○	○	∘		○	○											*	Semicolon
<			○	○	○	∘	○													*	Left angle bracket
=	○		○	○	○	∘	○		○											*	Equal
>	○		○	○	○	∘	○	○												*	Right angle bracket
?			○	○	○	∘	○	○	○											*	Question mark
@	○	○				∘														*	Commercial at mark
”			○			∘		○												*	Quotation
[	○	○		○	○	∘		○	○											*	Left brace
]	○	○		○	○	∘	○		○											*	Right brace

(Note 1) *: *When read in the comment zone, the codes are read into memory. When read in the significant data zone, the codes are ignored.*

(Note 2) ?: *When read in the comment zone, the codes are read into memory. When read in the significant data zone, an alarm is generated.*

(Note 3) *When custom macro option is used, the following codes can also be used in significant data zone.*
*[,], #, *, =, B, C, H, J, V and Y in ISO code.*
[,], &, a code set by parameter, B, C, H, J, V and Y in EIA code.

(Note 4) *Codes not in this table are ignored if their parity is correct.*

(Note 5) *Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment zone.*

(Note 6) *A character with all eight holes punched does not generate TH alarm even in EIA code.*

APPENDIX 2 G FUNCTION TABLE

The following G codes are available.

G code	Group	Function	Basic or option
G00	01	Positioning (Rapid traverse)	B
G01		Linear interpolation (Feed)	B
G02		Circular interpolation CW	O
G03		Circular interpolation CCW	O
G04	00	Dwell	B
G07		SINE interpolation (imaginary axis specification)	O
G09		Exact stop check	B
G10		Offset value setting, work zero point offset value setting	O
G17	02	XY plane selection	O
G18		ZX plane selection	O
G19		YZ plane selection	O
G20	06	Input in inch	O
G21		Input in mm	O
G22	04	Stored stroke limit ON	O
G23		Stored stroke limit OFF	O
G27	00	Reference point return check	O
G28		Return to reference point	O
G29		Return from reference point	O
G30		Return to 2nd, 3rd and 4th reference point	O
G31		Skip cutting	O
G33	01	Thread cutting	O
G40	07	Cutter compensation cancel	B
G41		Cutter compensation left	O
G42		Cutter compensation right	O
G43	08	Tool length offset + direction	O
G44		Tool length offset – direction	O
G49		Tool length offset cancel	O
G45	00	Tool offset increase	O
G46		Tool offset decrease	O
G47		Tool offset double increase	O
G48		Tool offset double decrease	O
G50	11	Scaling off	O
G51		Scaling on	O
G52	00	Local coordinate system setting	O
G54	14	Work coordinate system 1 select	O
G55		Work coordinate system 2 select	O
G56		Work coordinate system 3 select	O
G57		Work coordinate system 4 select	O
G58		Work coordinate system 5 select	O
G59		Work coordinate system 6 select	O
G60	00	Single direction positioning	O
G61	15	Exact stop check mode	B
G62		Automatic corner override effective	O
G64		Cutting mode	B
G65	00	Custom macro simple call	O

G code	Group	Function	Basic or option
G66	12	Custom macro modal call	O
G67 ◤		Custom macro modal call cancellation	O
G68	16	Coordinate system rotation ON	O
G69 ◤		Coordinate system rotation OFF	O
G73	09	Peck drilling cycle	O
G74		Counter tapping cycle	O
G76		Fine boring	O
G80 ◤		Canned cycle cancel	O
G81		Drilling cycle, spot boring	O
G82		Drilling cycle, counter boring	O
G83		Peck drilling cycle	O
G84		Tapping cycle	O
G85		Boring cycle	O
G86		Boring cycle	O
G87		Back boring cycle	O
G88		Boring cycle	O
G89		Boring cycle	O
G90 ◤	03	Absolute programming	B
G91 ◤		Incremental programming	B
G92	00	Programming of absolute zero point	B
G94 ◤	05	Per minute feed	B
G95 ◤		Per revolution feed	O
G96	13	Constant surface speed control	O
G97 ◤		Constant surface speed control cancel	O
G98 ◤	10	Return to initial point in canned cycle	O
G99		Return to R point in canned cycle	O

B: basic O: option

(Note 1) *The G codes marked with ◤ are initial G codes in each group. That is, when the power is turned on or when the reset button is pressed when the system parameter specifying initialization of G codes at reset is effective, those G codes are set.*
For G22 and G23, G22 is selected after the power was turned on. After resetting, G22 or G23, the one effective before resetting is set.
For G00 and G01, G43, G44 and G49, G90 and G91, G94 and G95, either is selected for the initial G codes by setting a parameter (G00, G43, G44, G90, G95).
For G20 and G21, the one in effect before cutting power or pressing the reset button is selected.

(Note 2) *The G codes in group 00 are not modal. They are effective only in the block in which they are specified.*

(Note 3) *An alarm (No. 010) occurs when a G code not listed in the above table is specified. When an optional G code not stored in the controller is specified, alarm (No. 010) occurs. However, G38 and G39 are ignored.*

(Note 4) *A number of G codes can be specified in a block even if they do not belong to the same group. When a number of G codes in the same group are specified, the G code specified last is effective.*

(Note 5) *If any G code in group 01 is specified in a canned cycle mode, the canned cycle is automatically cancelled and the G80 condition is entered. However G codes in group 01 is not affected by any of the canned cycle G codes.*

(Note 6) *G 70 and G71 can be used by parameter setting (GSP) instead of G20 and G21 respectively (Special G code).*

(Note 7) *A G code from each group other than 00, is in effect at all times.*

APPENDIX 3 TABLE OF RANGE OF COMMAND VALUE

	Input in mm / Output in mm	Input in inch / Output in mm	Input in mm / Output in inch	Input in inch / Output in inch
Least input increment	0.001 mm 0.001 deg	0.0001 inch 0.001 deg	0.001 mm 0.001 deg	0.0001 inch 0.001 deg
Maximum stroke (Value from the reference point)	±99999.999 mm	±99999.999 mm	±3937.0078 inch	±9999.9999 inch
Maximum programmable dimension	±99999.999 mm ±99999.999 deg	±3937.0078 inch ±99999.999 deg	±99999.999 mm ±99999.999 deg	±9999.9999 inch ±99999.999 deg
Feed per minute (At cutting feed rate override 100%)	1~15000 mm/min.	0.01~600.00 inch/min.	1~15000 mm/min.	0.01~600.00 inch/min.
Rapid traverse rate (Separate for each axis)	30~15000mm/min.	30~15000mm/min.	3.0~600.0 inch/min.	3.0~600.0 inch/min.
Upper limit of value of cutting feed rate Manual rapid traverse rate Fo	6~15000mm/min.	6~15000mm/min.	0.6~600.0 inch/min.	0.6~600.0 inch/min.
Manual jog feed rate	1~2000 mm/min.	0.04~78.7 inch/min.	0.5~1016 mm/min.	0.02~40 inch/min.
Coordinate value of 2nd reference point (Value from reference point)	0~±99999.999mm	0~±99999.999mm	0~±3937.0078 inch	0~±9999.9999 inch
Tool offset amount	0~±999.999mm	0~±99.9999 inch	0~±999.999 mm	0~±99.9999 inch
Minimum value in incremental feed	0.001 mm	0.0001 inch	0.001 mm	0.0001 inch
Backlash compensation value	0~0.255 mm	0~0.255 mm	0~0.0255 inch	0~0.0255 inch
Pitch error compensation value	0~±0.007 mm	0~±0.007 mm	0~±0.0007 inch	0~±0.0007 inch
Area of stored stroke limit (Value from reference point)	±99999.999 mm	±99999.999 mm	±3937.0078 inch	±9999.9999 inch
Dwell	0~99999.999 sec	0~99999.999 sec	0~99999.999 sec	0~99999.999 sec

APPENDIX 4 NOMOGRAPHS

4.1 Tool Path at Corner Arc

(1) Outline

Because of the delay of the servo system (owing to the exponential acceleration/deceleration during cutting or the positioning system using the DC servo motor), the actual tool path (tool center path) does not coincide with the programmed path as shown in the Fig. 4.1 (a).

The time constant of the exponential acceleration/deceleration T1 is fixed zero on the 6M.

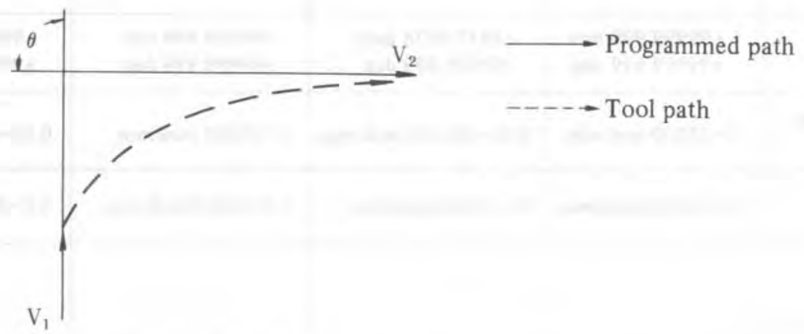


Fig. 4. 1 (a) Tool path in the corner

The tool path at the corner is determined by the following parameters.

(a) Cutting feed rate (V_1 and V_2)
(b) Corner angle (θ)
(c) Exponentical acceleration/deceleration time constant in cutting feed (T_2)
(d) Loop gain of positioning system.
(e) The buffer is provided or not.

In this appendix, the tool paths for proper parameters are illustrated by using these parameters and theoretically analizing the tool path.

Programming should be made so that the accuracy of the finished work shape is satisfied taking the above items into consideration.

If the accuracy is not theoretically satisfied, the dwell command should be inserted after the first block not to read the next block until the commanded speed becomes zero.

(2) Analysis

The tool path shown in the Fig. 4. 1 (b) will be mentioned under the following conditions.

(a) Cutting feed rate is constant after and before the corner.
(b) The buffer is provided to the NC.
(Error depends on the reading speed of the tape reader, the number of characters in the next block, etc.)

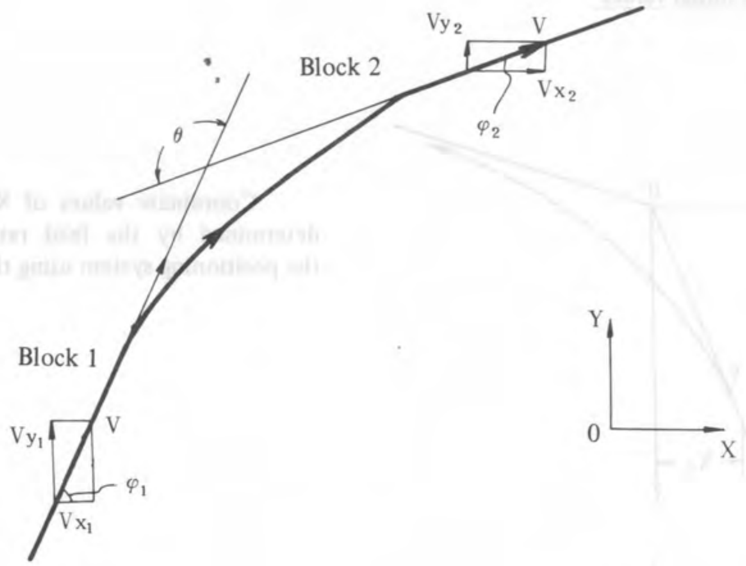


Fig. 4. 1 (b) Tool Path

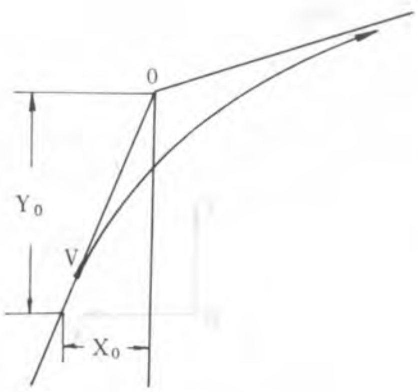
Conditional expressions

$Vx_1 = V.\cos\varphi_1$
$Vy_1 = V.\sin\varphi_1$
$Vx_2 = V.\cos\varphi_2$
$Vy_2 = V.\sin\varphi_2$
$\pi - (\varphi_1 - \varphi_2) = \theta$

Meanings of symbols

V	:	Cutting feed rate across corner
Vx_1	:	Feed rate component in X axis in the block 1
Vy_1	:	Feed rate component in Y axis in the block 1
Vx_2	:	Feed rate component in X axis in the block 2
Vy_2	:	Feed rate component in Y axis in the block 2
θ	:	Corner angle
φ_1	:	The angle between X axis and the block 1
φ_2	:	The angle between X axis and the block 2

Calculation of initial values



Coordinate values of X_0 and Y_0 in the figure are determined by the feed rate and the time constant of the positioning system using the DC servo motor.

Fig. 4. 1 (c) Initial values

$$X_0 = Vx_1\,(T_1 + T_2) \quad \cdots\cdots\cdots \quad (1)$$

$$Y_0 = Vy_1\,(T_1 + T_2) \quad \cdots\cdots\cdots \quad (2)$$

T_1 : Exponential acceleration/deceleration time constant ($T_1 = 0$)

T_2 : Time constant for the positioning system (Inverse of the position loop gain)

Tool path analysis

Velocity vector components at the corner (Vx and Vy) are described by following expressions.

$$Vx(t) = (Vx_2 - Vx_1)\left[1 - \frac{Vx_1}{T_1 + T_2}\left\{T_1 \cdot \exp\left(-\frac{t}{T_1}\right) - T_2 \cdot \exp\left(-\frac{t}{T_2}\right)\right\} + Vx_1\right]$$

$$= Vx_2\left[1 - \frac{Vx_1}{T_1 + T_2}\left\{T_1 \cdot \exp\left(-\frac{t}{T_1}\right) - T_2 \cdot \exp\left(-\frac{t}{T_2}\right)\right\}\right] \quad \cdots\cdots \quad (3)$$

$$Vy(t) = \frac{Vy_1 - Vy_2}{T_1 + T_2}\left\{T_1 \cdot \exp\left(-\frac{t}{T_1}\right) - T_2 \cdot \exp\left(-\frac{t}{T_2}\right)\right\} + Vy_2 \quad \cdots\cdots \quad (4)$$

The coordinate values of the tool at the time t is calculated by the following expressions.

$$X(t) = \int_0^t Vx(t)dt - X_0$$

$$= \frac{Vx_2 - Vx_1}{T_1 - T_2}\left\{T_1^2 \cdot \exp\left(-\frac{t}{T_1}\right) - T_2^2 \cdot \exp\left(-\frac{t}{T_2}\right)\right\} - Vx_2\,(T_1 + T_2 - t)$$

$$\cdots\cdots \quad (5)$$

$$Y(t) = \int_0^t Vy(t)dt - Y_0$$

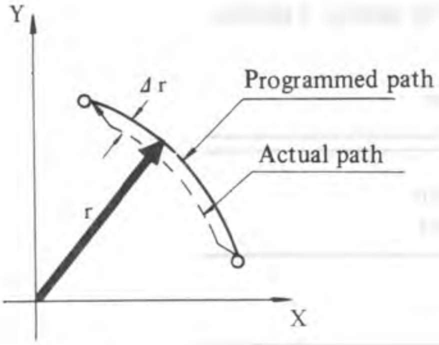
$$= \frac{Vy_2 - Vy_1}{T_1 - T_2}\left\{T_1^2 \cdot \exp\left(-\frac{t}{T_1}\right) - T_2^2 \cdot \exp\left(-\frac{t}{T_2}\right)\right\} - Vy_2\,(T_1 + T_2 - t)$$

$$\cdots\cdots \quad (6)$$

4.2 Error in Radius Direction in Circular Cutting

When using DC servo motor, a delay by the positioning system occurs between input and output axes.

In linear interpolation, the tool moves on a specified segment of a line; thus no error is generated. In circular interpolation, especially in high-speed circular cutting, an error is generated in the radius direction. This error can be calculated by the procedure described below.



Δr: Maximum error in radius direction (mm)
v: Feedrate (mm/sec.)
r: Radius of arc (mm)
T_1: Time constant for cutting feed
T_2: Time constant for positioning system (Inverse of the position loop gain)

$$\Delta r = \frac{1}{2} \cdot (T_1^2 + T_2^2) \cdot \frac{v^2}{r} \quad \cdots\cdots (1)$$

For actual processing, a processing radius, r (mm), of the workpiece and an allowable error, Δr (mm), is given, and the allowable speed limit, v (mm/sec.), can be calculated by formula (1).

Since acceleration/deceleration time constant for cutting feed differs with machine tool builders, please refer to the manual issued by your machine tool builder.

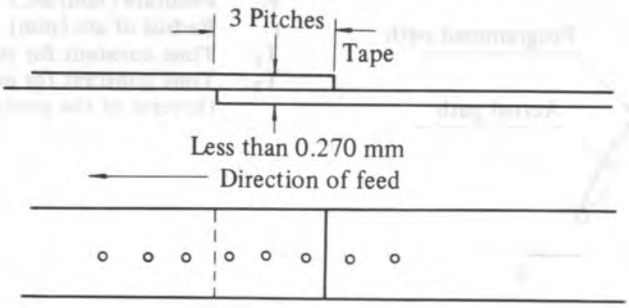
The time constant for servo system is $T_2 = 33 \times 10^{-3}$ (sec), for DC servo motors.

APPENDIX 5 TAPE JOINING

If a loop of punched tape is to be made or tape is broken or in similar cases, two tape sections must be joined. Join these tape section as described below.

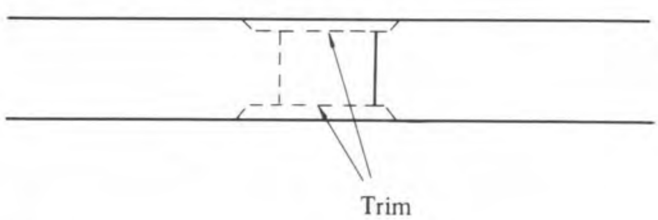
(a) Butt the two tape sections and paste them together, placing the section on the side on which the tape is fed, on top of the other.

(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 smooth them.



(e) Make sure punched holes are not clogged up with the paste.

APPENDIX 6 PARAMETERS

Display and Setting Procedure of Parameter

Parameters must be set correctly so that the DC servo motor and the machine tool are given their full characteristics, specifications and functions, when the NC is combined with the machine tool. Since parameters differ with machine tool, refer to the parameter table prepared by machine tool builder for the contents. End user must not change parameters.

6.1 Parameter display procedure

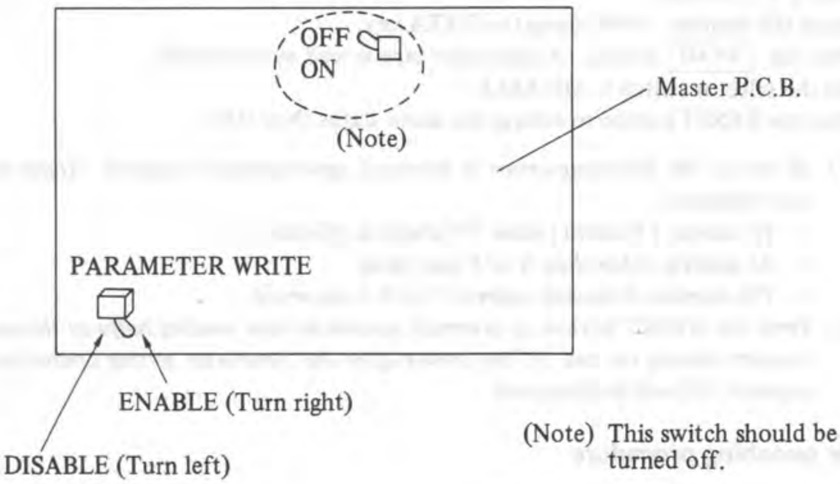
(a) Press the PARAM button.

(b) Press the PAGE button to select the page. N a parameter number INPUT can also be used to select a screen.

6.2 Parameter setting procedure

(1) Setting from the MDI

(a) Set the selector switch on the master P.C.B. to ENABLE.



(Note) This switch should be turned off.

(b) Set the mode switch to MDI.

(c) Press the PARAM button.

(d) Keying in N and parameter number to be set and pressing the INPUT button, selects the page of that parameter number and moves the cursor to below the number (the PAGE button and the CURSOR button can also be used).

(e) Key in P and data to be set and press the INPUT button. When you are aware of an error during keying-in, press the CAN button.

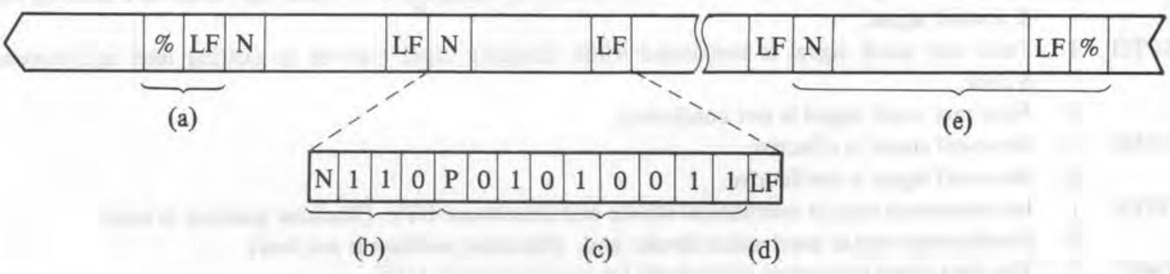
(f) Check whether the data was set correctly.

(g) Turn the PARAMETER WRITE switch on the master PCB to DISABLE after all parameters were set and confirmed.

(h) Push a reset button to release alarm status (number 100).

(2) Setting from a tape

Parameters can also be entered from a tape. Prepare a tape for parameter setting as follows.



(a) Punch %LF (for ISO code) or ER CR (for EIA code) at the start of the tape.
(b) Punch the parameter number following address N after the LF (or CR).
(c) Punch the parameters corresponding to the parameter number following address P.
(d) Punch LF (or CR).
Repeat (b) through (d) the required number of times. Each block must start with a parameter number following address N.
Leading zeros can be omitted for parameters following address P.
(e) Punch LF% or (or CRER) at the end to stop reading parameters.

Parameters not punched on the tape do not change after the tape is read into the NC. So you can prepare several tapes; one for backlash compensation data, one for pitch error compensation data, etc.

A parameter tape is read into the NC as follows:

① Set the selector switch on the master PCB to ENABLE. Alarm No. 100 will be displayed on the CRT.
② Load the parameter tape in the tape reader.
③ Push the EMERGENCY STOP button. Confirm the LABEL SKIP status. If the LABEL SKIP status is not in effect, push the EMERGENCY STOP button in a mode other than MDI mode to enter LABEL SKIP status.
④ Push the [PARAM] button.
⑤ Push [P] button.
⑥ Input the number −9999 using the DATA key.
⑦ Push the [READ] button. A parameter tape is read automatically.
⑧ Set the selector switch to DISABLE.
⑨ Push the RESET button to release the alarm status (No. 100).

(Note 1) If one of the following errors is detected, tape reading is stopped. Alarm numbers, however, are not displayed.
- *TH alarm, TV alarm (when TV check is effective).*
- *An address other than N or P was input.*
- *The number following address N or P is incorrect.*

(Note 2) Press the RESET button to interrupt parameter tape reading halfway through. Some parameters require turning on and off the power after the parameter setting operations. At this time alarm number 000 will be displayed.

6.3 Parameter punching procedure

(a) Connect the puncher to the tape punch unit.
(b) Select punching code, EIA/ISO.
(c) Set the MODE SELECT switch to EDIT.
(d) Push the [PARAM] button.
(e) Input [P] [−] [9] [9] [9] [9] [PUNCH].

(Note) When a puncher is connected or disconnected to the NC, both NC and puncher power must be turned off.

6.4 Parameters

	0	0	6

ORWD	SCTO	EENB	OTCS	FMIC	MDL	MIC	SCW
7	6	5	4	3	2	1	0

ORWD 1: The rewind signal is output only when a tape is rewound by the reset & rewind signal.
0: The rewind signal is output for a certain time (100 msec) even when tape is not rewound by the reset & rewind signal.

SCTO 1: Feed rate reach signal is confirmed while changing rapid traverse to cutting feed and commanding S code.
0: Feed rate reach signal is not confirmed.

EENB 1: Servo-off signal is effective.
0: Servo-off signal is ineffective.

OTCS 1: Instanteneous stop at mechanical stroke end (Hardware OT). (Machine position is lost)
0: Decelerated stop at mechanical stroke end. (Machine position is not lost)

FMIC 1: The least input increment of feedrate for metric input is 1/10.
0: The least input increment of feedrate for metric input is not 1/10.

MDL 1: The least display increment of position display unit for metric input (inch input) is 0.01 mm (0.0001 inch).
0: The least display increment of position display unit for metric input (inch input) is 0.001 mm (0.0001 inch).
MIC 1: The least input increment for metric input (inch input) is 0.01 mm (0.0001 inch).
0: The least input increment for metric input (inch input) is 0.001 mm (0.0001 inch).
SCW 1: The least command increment is 0.0001 inch. (Machine tool: inch system)
0: The least command increment is 0.001 mm. (Machine tool: metric system)

	0	0	7	ADFT	EOM	CINP	DCS	CLER	TVC	PPD	RDRN
				7	6	5	4	3	2	1	0

ADFT 1: Automatic drift is compensated for.
0: Automatic drift is not compensated for.
EOM 1: When M30 is specified, M30 is sent to the machine tool, then when the NC receives the FIN signal, the execution continues from the head of the program.
If the NC does not receive the FIN signal but the external reset signal, execution returns to the head of the program and the NC is reset. (Memory operation mode)
0: When M30 is specified, M30 is sent to the machine tool but execution does not return to the head of a program unless the NC receives the reset & rewind signal. (Memory operation mode)
CINP 1: NC control goes to the next block after the specified feedrate becomes zero with deceleration and the NC confirms that the machine position is at the specified position at the intersection of two blocks other than two cutting feed blocks or when error detection is effective. (Confirmation is called in-position check.)
0: NC control goes to the next block after the specified feedrate becomes zero with deceleration at the intersection of two blocks other than two cutting feed blocks or when error detection is effective. (No in-position check is made.)
DCS 1: The system is generated by the START button on the MDI panel without going through the machine tool. (In the MDI mode only)
0: The start signal is sent to the machine tool by pressing the START button on the MDI panel. Then when the NC receives the start signal issued from the machine tool, the system is generated.
CLER 1: The NC is cleared by the Reset button, External Reset or Reset & rewind signal.
0: The NC is reset by the Reset button, External Reset or Reset & rewind signal. (Refer to appendix 8 for reset and clear status.)
TVC 1: The TV check of the information in the comment zone is ineffective.
0: The TV check of the information in the comment zone is effective.
PPD 1: The indication of the position display unit is preset by G92.
0: The indication of the position display unit is not preset by G92.
RDRN 1: Dry run is effective for rapid traverse command.
0: Dry run is not effective for rapid traverse command.

	0	0	8	ICR		GSP	G44	G90	G95	G43	G00
				7	6	5	4	3	2	1	0

ICR 1: "LF" is punched as the EOB code in the ISO code system.
0: "LF CR CR" is punched as the EOB code in the ISO code system.
GSP 1: Special G code is used.
0: Standard G code is used.
G90 1: G90 is selected when the power is turned on or the system is cleared.
0: G91 is selected when the power is turned on or the system is cleared.
G95 1: G95 is selected when the power is turned on or the system is cleared.
0: G94 is selected when the power is turned on or the system is cleared.
G00 1 G00 is selected when the power is turned on or the system is cleared.
0: G01 is selected when the power is turned on or the system is cleared.

G44, G43

Parameter setting		Initial G code of group 08
G44	G43	
1	0	G44
0	1	G43
0	0	G49

0	0	9	FIX2	RWL	MCF	FMFS	FCUT	ILVL	EFR1	TDRN
			7	6	5	4	3	2	1	0

FIX2 1: M code is output in canned cycle. (canned cycle II)
0: Signals SSP, and SRV are output in canned cycle. (canned cycle I)
RWL 1: Forbidden area of stored stroke limit 2 is outside.
0: Forbidden area of stored stroke limit 2 is inside.
MCF 1: The external function signal (EF) is output at completing positioning in canned cycle G81. (without Z axis movement)
0: The external function signal (EF) is not output at completing positioning in canned cycle G81. (with Z axis movement)
FMFS 1: FMF signal is sent out twice to the machine side in canned cycle.
0: FMF signal is sent out once in canned cycle.
FCUT 1: The move command in X and Y axes in canned cycle depends on the G codes in group 01.
0: The move command in X and Y axes in canned cycle is that of rapid traverse.
ILVL 1: Initial level point can be changed by using RESET or ORIGIN button.
0: Initial level point cannot be changed by using RESET or ORIGIN button.
EFR1 1: External function signal (EF) is output through photo coupler. (Not used.)
0: External function signal (EF) is output through reed relay.
TDRN 1: Dry run in thread cutting is effective.
0: Dry run in thread cutting is ineffective.

0	1	0	TCW	CWM	SOV	TLCC	OFSD	SOVC	REDT	ISOT
			7	6	5	4	3	2	1	0

TCW, CWM The sign of S-4 digit analog output signal.

Parameter		Output Signal	
TCW	CWM	M03	M04
0	0	+	+
0	1	–	–
1	0	+	–
1	1	–	+

SOV 1: Spindle override is effective.
0: Spindle override is ineffective.
TLCC 1: When the offset value is changed in the G43 or G44 mode, it is effective from the next block.
0: When the offset value is changed in the G43 or G44 mode, it is effective from the next appearing H or D code.
OFSD 1: D code is used for tool offset (G45 ~ G48).
0: H code is used for tool offset (G45 ~ G48).
SOVC 1: The spindle override is clamped to 100% during tapping.
0: The spindle override is not clamped to 100% even during tapping.

REDT 1: The part program is registered in memory by pushing the cycle start button in the EDIT mode.
0: It is not registered.

IOST 1: The manual rapid traverse rate becomes effective even if reference point return is not performed when the stored stroke limit option is used.
0: The manual rapid traverse rate does not become effective until the reference point return is performed.

	0	1	1

DGNE	SETE	DECI	SSPB	NPRD	VCT	SUPM	ADLN
7	6	5	4	3	2	1	0

DGNE 1: Data output is effective in DIAGNOSE.
0: Data output is ineffective in DIAGNOSE.

SETE 1: Setting is possible even when the LOCK key is locked.
0: Setting is not possible when the LOCK key is locked.

DECI 1: Deceleration signal "1" in reference point return indicates deceleration.
0: Deceleration signal "0" in reference point return indicates deceleration.

SSPB 1: The rotation of the spindle stops when the spindle stop input signal SSTP is 0.
0: The rotation of the spindle stops when the spindle stop input signal SSTP is 1.

NPRD 1: Decimal point programming and the display are not used. (Not used)
0: Decimal point programming and the display are used.

VCT 1: Cutter compensation vector can be specified by address I, J or K. (Must not be used.)
0: Cutter compensation vector cannot be specified by addresses I, J and K. (Always calculated automatically.)

SUPM 1: Start-up B type is effective in cutter radius compensation C.
0: Start-up A type is effective in cutter radius compensation C.

ADLN 1: Additional axis is a linear axis.
0: Additional axis is a rotary axis.

	0	1	2

ZGM4	ZGMZ	ZGMY	ZGMX	ZM4	ZMZ	ZMY	ZMX
7	6	5	4	3	2	1	0

ZGMX, ZGMY, ZGMZ, ZGM4 Reference point return method of each axis.
1: Magneswitch method
0: Grid method

ZMX, ZMY, ZMZ, ZM4 Reference point return direction of each axis and the initial backlash direction when turning on the power.
1: Reference point return and the backlash are performed in the negative direction.
0: Reference point return and the backlash are performed in the positive direction.

(Note 1) For the axis functioning in reference point return, the reference point return direction and the initial backlash direction are the same. For another axis, this parameter specifies only the initial backlash direction.

(Note 2) The backlash compensation is performed for the first time when motion along the axis moves in the direction opposite to the direction set by this parameter after the power is turned on.

	0	1	3

PSG2	PSG1		PHS				
7	6	5	4	3	2	1	0

PSG2, PSG1 Gear ratio of spindle and position coder

Magnification	PSG2	PSG1
× 1	0	0
× 2	0	1
× 4	1	0
× 8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle rotations}}{\text{Number of position coder rotations}}$$

PHS Primitive setting of phase shift amount in DSCG method (Resolver/Inductosyn).

1: The phase shift amount is not set automatically.

0: The phase shift amount is set automatically.
Once this is done, PHS = 1 is assigned automatically.

Address	7	6	5	4	3	2	1	0
0 1 4		DMRX			GRDX			
0 1 5		DMRY			GRDY			
0 1 6		DMRZ			GRDZ			
0 1 7		DMR4			GRD4			
4 1 0		DMR5			GRD5			

DMRX, DMRY, DMRZ, DMR4, DMR5

Detect multipling ratio for each axis.

Setting code			Multipling ratio	
			Pulse coder	Resolver, Inductosyn
0	0	0	1/2	1/8
0	0	1	1	1/4
0	1	0	1	1/4
0	1	1	2	1/2
1	0	0	3/2	3/8
1	0	1	3	3/4
1	1	0	2	1/2
1	1	1	4	1

GRDX, GRDY, GRDZ, GRD4, GRD5

Capacity of reference counter for each axis.

Setting code				One cycle capacity
0	0	0	1	2000
0	0	1	0	3000
0	0	1	1	4000
0	1	0	0	5000
0	1	0	1	6000
0	1	1	1	8000
1	0	0	1	10000

If the code other than codes in the above table is set, capacity is set to 8000.

	0	1	8

	DIC	IPL2	CPF2	CPF1
	3	2	1	0

DIC 1: Input unit 1/10
0: Input unit 1/1
IPL2 1: Interpolation unit 1/2 (0.0005 mm, 0.00005 inch)
0 Interpolation unit 1/1 (0.001 mm, 0.0001 inch)
CPF2, CPF1 Pulse frequency of a backlash compensation (Common for all axes)

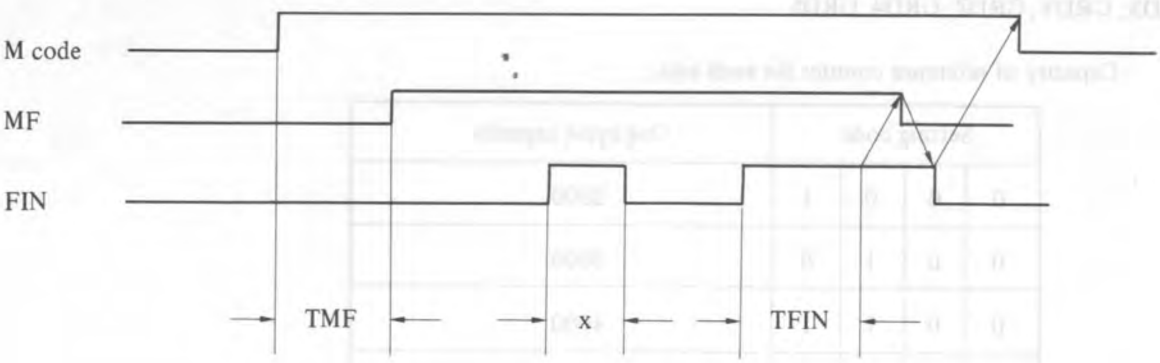
Frequency (kHz)	CPF2	CPF1
32	0	0
64	0	1
128	1	0
256	1	1

(Set surely the frequency to 256 Hz)

	0	1	9

TMF				TFIN			
7	6	5	4	3	2	1	0

TMF Time from M, S, T, B code issue to MF, SF, TF, BF issue 16 ~ 256m sec (16m sec increment)
TFIN Time of reception width of FIN 16 ~ 256 m sec (16 m sec increment)



Because of X < TFIN
it is neglected.

TMF	TFIN	Parameter setting			
16 (msec)	More than 16 (msec)	0	0	0	0
32	" 32	0	0	0	1
48	" 48	0	0	1	0
64	" 64	0	0	1	1
80	" 80	0	1	0	0
96	" 96	0	1	0	1
112	" 112	0	1	1	0
128	" 128	0	1	1	1
144	" 144	1	0	0	0
160	" 160	1	0	0	1
176	" 176	1	0	1	0
192	" 192	1	0	1	1
208	" 208	1	1	0	0
224	" 224	1	1	0	1
240	" 240	1	1	1	0
256	" 256	1	1	1	1

	0	2	0

CLSI			ZTN5	ZTN4	ZTNZ	ZTNY	ZTNX
7			4	3	2	1	0

CLSI 1: Servo position LSI is not checked.
2: Servo position LSI is checked.

ZTNX, ZTNY, ZTNZ, ZTN4, ZTN5
Provision of reference point return function for each axis
1: Provided
0: Not provided

	0	2	1

G84S	SFOU	EDMZ	EDMY	EDMX	EDPZ	EDPY	EDPX
7	6	5	4	3	2	1	0

G84S 1: Method B is effective in canned cycle G74 and G84 with S-12 bit output A or S analog output A.
0: Method A is effective in canned cycle G74 and G84 with S-12 bit output or S analog output.
(Refer to connecting manual of FS6M for method A/B.)

SFOU 1: When the gear change is not performed at S-12 bit output A or S analog output A, SF signal is output.
0: In the above case, SF signal is not output.

EDMX, EDMY, DMZ
1: External deceleration is effective for rapid traverse and cutting feed in (–) direction of each axis.
0: External deceleration is effective only for rapid traverse in (–) direction.

EDPX, EDPY, EDPZ
1: External deceleration is effective for rapid traverse and cutting feed in (+) direction of each axis.
0: External deceleration is effective only for rapid traverse in (+) direction.

	0	2	2

	SIJ	PMXY2	PMXY1	RS43	FXCD	TAPSG	FXCS
7	6	5	4	3	2	1	0

SIJ Specifying method of retracting direction and amount in canned cycle G76 and G87.
1: Addresses I and J specify the retracting direction and amount.
0: Address Q specifies the amount and the parameters PMXY1 and PMXY2 specify the direction.

PMXY1, PMXY2
Retracting direction in canned cycle G76 and G87. (These parameters are effective only when the parameter SIJ is zero.)

PMXY2	PMXY1	Retracting direction
0	0	+ X
0	1	– X
1	0	+ Y
1	1	– Y

RS43 1: Offset vector in G43 and G44 modes remain effective after resetting.
0: Offset vector in G43 and G44 modes are cleared by reet.

FXCD 1: Dwell command is effective in canned cycle G74 and G84.
0: Dwell command is ineffective in canned cycle G74 and G84.

TAPSG 1: Tapping signal is output in canned cycle G84.
0: Tapping signal is not output.

FXCS 1: In canned cycle G74 and G84, spindle rotates CW or CCW without M05 signal output.
(It is effective only when FIX2 is 1.)
0: In canned cycle G74 and G84, spindle rotates CW or CCW after M05 signal is output.

	0	2	3

HIPRP	EX10D						
7	6	5	4	3	2	1	0

HIPRP 1: When very small blocks continue, their paths are not displayed but processing rate is increased. (Must not be used)
0: Their paths are displayed normally.
EX10D 1: Unit of external zero point offset amount is multiplied by 10.
0: It is not multiplied by 10.

	0	2	4	PML2	PML1	DLME	RDAL	ADW2	ADW1	ADW0	FMT
				7	6	5	4	3	2	1	0

PML2, 1 Pitch error compensation magnification. (all axes common)
The value, with this magnification multiplied to the set compensation value, is outputted.

PML2	PML1	Magnification
0	0	x 1
0	1	x 2
1	0	x 4
1	1	x 8

DLME 1: When a program is loaded in memory, all stored programs are canceled automatically.
0: When a program is loaded in memory, all stored programs are not canceled automatically.

RDAL 1: When a program is loaded in memory, all programs always are loaded.
0: When a program is loaded in memory, whether only one program is loaded or all programs are loaded is selected by operation of MDI.

ADW2, 1, 0 In the case with an additional axis provided, a setting of address in punching out or in displaying on CRT.

ADW2	ADW1	ADW0	Address
0	0	0	A
0	0	1	B
0	1	0	C
0	1	1	U
1	0	0	V
1	0	1	W

FMT 1: Performs software parity check as regards part program editing areas.
0: Does not perform.

This parameter is set to 1 automatically, when the condition has become ready for checking. This parameter cannot be turned on/off externally.

	0	2	5	MUSR	MCYL	MSUB	MPRM				TSE
				7	6	5	4	3	2	1	0

MUSR 1: Custom macro interrupt function is used.
0: Custom macro interrupt function is not used.

MCYL 1: Custom macro interrupt is executed during cycle operation.
0: Custom macro interrupt is not executed during cycle operation.

MSUB 1: Subprogram type custom macro interrupt.
0: Macro type custom macro interrupt.

(Note) Values of local variables used in called macro program do not change by subprogram type custom macro interrupt. Values of local variable used in called macro program differ from those used in the current program by macro type custom macro interrupt.

MPRM 1: Two M codes used to make custom macro interrupt effective and ineffective are set by parameters.
0: M96 and M97 are used for custom macro interrupt.

TSE 1: Status trigger method is used for custom macro interrupt.
0: Edge trigger method is used for custom macro interrupt.

*(Note) Macro interruption is made as long as interrupt signal is input on status trigger method.
Macro interruption is made only when interrupt signal is turned on with edge trigger method.*

	0	2	6

7	6	5	4	3	2	1	0
	FHDL	NGMP	OFFVY		OGE		CKIM

FHDL 1: Movement amount per scale of manual pulse generator is fixed to 0.01 mm/0.001 inch and is not influenced by signals MP1, MP2. (Not used)
0: Movement amount per scale of manual pulse generator varies according to MP1, MP2 signals.

NGMP Movement amount per scale of manual pulse generator is as follows. (Not used)

NGMP	MP2	MP1	Movement amount
0	0	0	0.001 mm/0.0001 inch
0	0	1	0.01 mm/0.001 inch
0	1	0	0.1 mm/0.01 inch
0	1	1	0.1 mm/0.01 inch
1	0	0	0.01 mm/0.001 inch
1	0	1	0.001 mm/0.0001 inch
1	1	0	0.1 mm/0.01 inch
1	1	1	0.1 mm/0.01 inch

OFFVY 1: Servo alarm is not generated, even if VRDY is on before PRDY is output.
0: Servo alarm is generated, if VRDY is on before PRDY is output.

Parameters mentioned below and after should be set by decimal number.

OGE 1: Memory arrangement is not executed during program search.
0: Memory arrangement is executed during program search.

CKIM 1: Ignores the turning on or off of the machine lock signal during automatic operation (keeps the same state when cycle start was applied).
0: Makes the machine lock signal valid immediately.

(The machine lock signal is always made valid in the manual mode.)

	0	2	7	CMRX

	0	2	8	CMRY

	0	2	9	CMRZ

	0	3	0	CMR4

355

	4	1	4

CMR5

CMRX, CMRY, CMRZ, CMR4, CMR5

Command multiply for each axis.

(1) When parameter 0316 ACMR = 0 (Standard specification)

Setting code	Multiply
1	0.5
2	1
4	2
10	5
20	10

Refer to table mentioned in parameter 014~017, 410 for setting value.

(Note) If a code not listed in this table is set, the multiply is 1.

(2) When parameter 0316 ACMR =1 (Arbitrary command multiply)

(i) Setting method when command multiply ratio ranges from 1/2 to 1/27

$$\text{Setting value} = \frac{1}{\text{Command multiply ratio}} + 100$$

(ii) Setting method when command multiply ratio ranges from 2 to 48.

Setting value =2 x (Command multiply ratio)

(Note) Set such a value that command multiply ratio be an integer in case of (ii) above.

	0	3	1

VLOCX

	0	3	2

VLOCY

	0	3	3

VLOCZ

	0	3	4

VLOC4

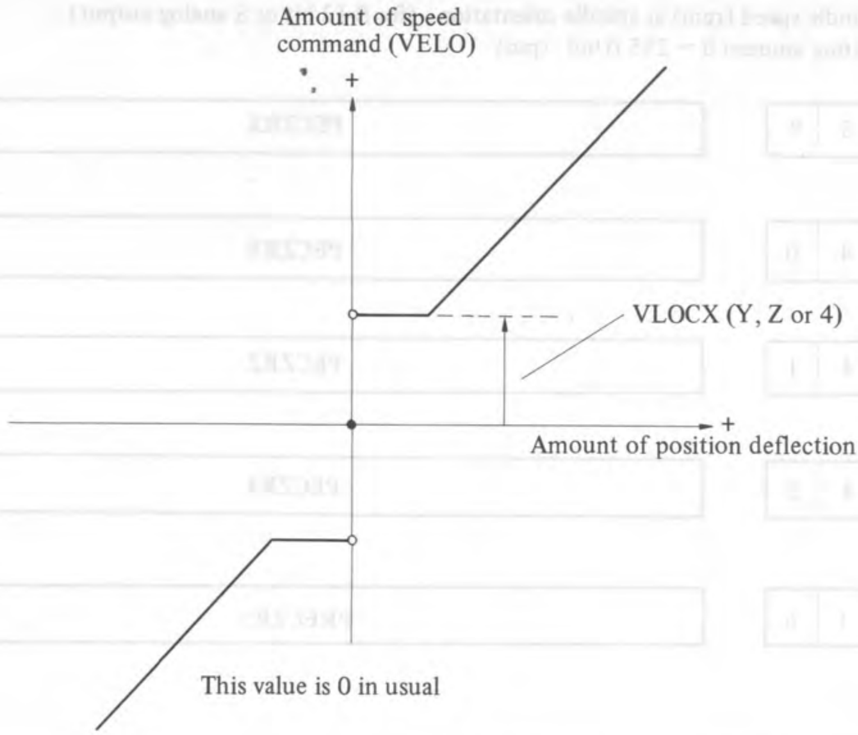
	4	1	5

VLOC5

VLOCX, VLOCY, VLOCZ, VLOC4, VLOC5

Clamp of feed command value of each axis.

Setting amount 0 ~ 7



	0	3	5

MBUF1

	0	3	6

MBUF2

MBUF1, MBUF2

Up to two M codes are set. If this code is read, a block following the M code is not read in the buffer storage.
Setting amount 00 ~ 97

	0	3	7

SPGST

SPGST Spindle motor rev. at spindle gear shift. (for S-12 bit or S analog output A/B).
Setting amount 0 ~ 255.

$$\text{Setting value} = \frac{\text{Gear shift spindle motor rev. number}}{\text{Spindle motor maximum rev. number}} \times 4095$$

	0	3	8

SPSOR

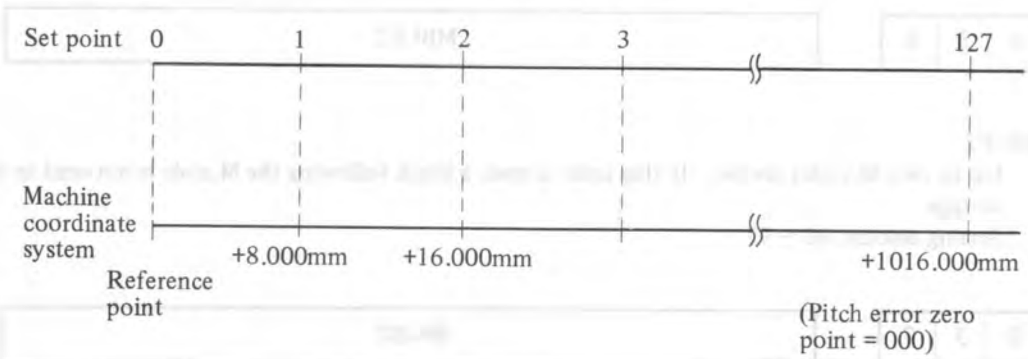
357

SPSOR Spindle speed (rpm) at spindle orientation. (for S-12 bit or S analog output)
Setting amount 0 ~ 255 (Unit: rpm)

	0	3	9	PECZRX
	0	4	0	PECZRY
	0	4	1	PECZRZ
	0	4	2	PECZR4
	4	1	6	PRECZR5

PECZRX ~ 5

Setting of pitch error zero point of each axis.
Setting amount 0 ~ 127.
Set a point which corresponds to the reference point.
For example, if point 0 is specified, point 1 is at +8.000mm position and point 127 is at +1016.000mm.
And compensation is made over the range from 0 to 1016.000mm.



Here, the pitch error compensation interval, which is set by parameter with parameter number 163 ~ 166, is 8000.

	0	4	3	UMMCD4
	0	4	4	UMMCD5
	0	4	5	UMMCD6
	0	4	6	UMMCD7

	0	4	7	UMMCD8

	0	4	8	UMMCD9

	0	4	9	UMMCD10

	0	5	0	UMMCD11

	0	5	1	UMMCD12

	0	5	2	UMMCD13

UMMCD4 ~ UMMCD13

An M code to call custom macro. (Ten M codes)
(Argnment assignment can be used.)
Setting value 01 to 97

	0	5	3	MACINTON

MACINTON

An M code to make custom macro interrupt effective.
Setting value 03 ~ 97
This parameter is effective only when parameter 025MPRM = 1.

	0	5	4	MACINTOF

MACINTOF

An M code to make custom macro interrupt ineffective.
Setting value 03 ~ 97
This parameter is effective only when parameter 025 MPRM = 1.

	0	5	7	TMHOR (Hour)

	0	5	8	TMMIN (Minute)

	0	5	9	TMSEC (Second)

It sets and displays the run hour (Time when STL lamp is lighting) in the unit of hr, min and sec.

TMHOR Indication in hour 0 ~ 255 (One hour increment)

TMMIN Indication in minute 0 ~ 59 (One minute increment)

TMSEC Indication in second 0 ~ 58 (Two seconds increment)

Run hour (flickering time of STL lamp) is displayed in hours, minutes and seconds.
The run hour is held in the bubble memory even if the power is cut off. But as the reading-in of the run hour into the bubble memory is made every 6 minutes, the time less than 6 minutes is cleared when the power is turned off. When you wish to preset the run hour to zero, follow the normal setting operation.

	0	6	0	IDXUNT

IDXUNT Least angle for indexing the index table

Set Value 1 to 255
Unit deg

This is significant only when parameter 314-5 (IFIX) is 1. If, at this time, a distance which is not an integer multiple of the value specified in IDXUNT, a PS alarm occurs (No. 180). Also, if the value specified by the coordinate system setting (G92), automatic coordinate system setting parameter value, or work zero point offset parameter value is not an integer multiple of the value specified in IDXUNT, a PS alarm occurs (No. 180) when the value is used.

	0	6	1	F1DN

FIDN In case of F 1-digit command, it is the constant to determine variations of feed rate when the manual pulse generator is turned by one scale.

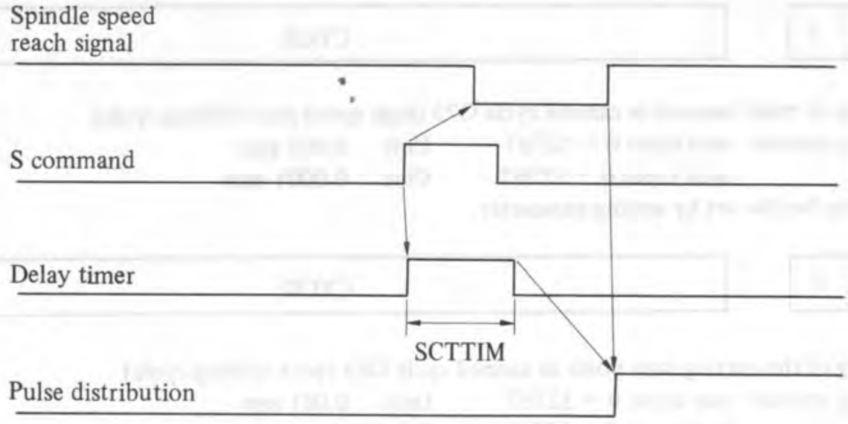
$$\triangle F = \frac{Fmax\ i}{100n} \quad (i = 1, 2)$$

It sets n in the above expression. Namely, it sets how many times the manual pulse generator is to be turned for the feed rate to reach to Fmax i.
Setting amount: 1 ~ 127
The Fmax i in the above expression is the upper limit value of the feed rate of F 1-digit command, and is set to parameter 065, 066.

Fmax 1: The feed rate upper limit value of F1 ~ F4 (065)
Fmax 2: The feed rate upper limit value of F5 ~ F9 (066)

	0	6	2	SCTTIM

SCTTIM Delay timer setting when the spindle speed reach signal is checked.
Set the time interval from execution of the S function to a check start of the spindle speed reach signal.
Setting amount 0 ~ 255 Unit: msec.



0	6	3				ADL5	EX5NG	HIR5	AP5	G605
			7	6	5	4	3	2	1	0

ADL5 1: 5th axis is linear axis.
0: 5th axis is rotary axis.
EX5NG 1: 5th axis neglect signal is effective.
0: 5th axis neglect signal is ineffective.
HIR5 1: 5th axis handle interruption is effective.
0: 5th axis handle interruption is ineffective.
AP5 1: 5th axis automatic coordinate system setting is effective.
0: 5th axis automatic coordinate system setting is ineffective.
G605 1: 5th axis single direction positioning approach direction is minus.
0: It is plus.

0	6	4						SCLZ	SCLY	SCLX
			7	6	5	4	3	2	1	0

SCLX, Y, Z
Scaling function is:
1: Effective
0: Ineffective

0	6	5	F1DMAX1
0	6	6	F1DMAX2

FIDMAX1, FIDMAX2
The upper limit value of feed rate of F 1-digit command.
F1DMAX1: The feed rate upper limit value of F1 ~ F4
F1DMAX2: The feed rate upper limit value of F5 ~ F9
Setting amount: 0 ~ 15000 Unit: mm/min (mm output)
0 ~ 6000 Unit: 0.1 inch/min (inch output)
Refer to parameter 061

	0	6	7	CYCR

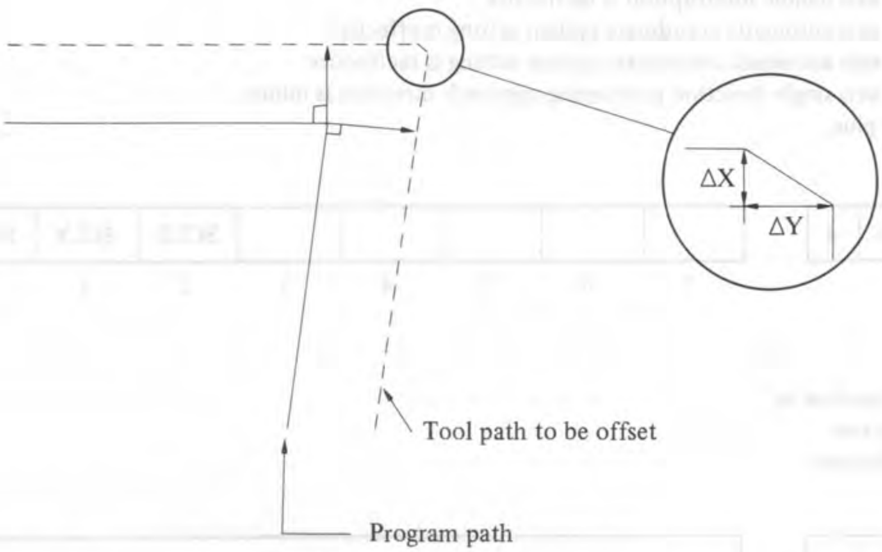
CYCR
Setting of relief amount in canned cycle G73 (high speed peck drilling cycle).
Setting amount mm input 0 ~ 32767 Unit: 0.001 mm
inch input 0 ~ 32767 Unit: 0.0001 inch
This can be also set by setting parameter.

	0	6	8	CYCD

CYCD
Setting of the cutting start point in canned cycle G83 (peck drilling cycle)
Setting amount mm input 0 ~ 32767 Unit: 0.001 mm
inch input 0 ~ 32767 Unit: 0.0001 inch
This can be also set by setting parameter.

	0	6	9	CRCDL

CRCDL
When tool moves along the outside of acute angle close to 90° with cutter radius compensation, limitations on ignoring a small movement amount.
Setting amount mm input 0 ~ 16383 (Unit: 0.001 mm)
inch input 0 ~ 16383 (Unit: 0.0001 inch)



If $\Delta X <$ CRCDL and $\Delta Y <$ CRCDL, the small movement is ignored. It prevents the workpiece from being affected by stepping the tool at the corner.

	0	7	0	INPX

	0	7	1	INPY

	0	7	2	INPZ

	0	7	3	INP4

425	INP5

INPX, INPY, INPZ, INP4, INP5
Width of imposition for each axis.
Setting amount 0 ~ 32767 (Detect unit)

074	STPEX
075	STPEY
076	STPEZ
077	STPE4
426	STPE5

STPEX, STPEY, STPEZ, SPTE4, STPE5
Limitation value of position deflection amount at stoppage for each axis.
Setting amount 0 ~ 32767 (Detect unit)

078	SERRX
079	SERRY
080	SERRZ
081	SERR4
427	SERR5

SERRX, SRRRY, SERRZ, SERR4, SERR5
Limitation value of position deflection amount during movement for each axis.
Setting amount 0 ~ 32767 (Detect unit)

082	GRDSX

	0	8	3	GRDSY

	0	8	4	GRDSZ

	0	8	5	GRDS4

	4	2	1	GRDS5

GRDSX, GRDSY, GRDSZ, GRDS4, GRDS5

Setting of grid shift amount of each axis.
Setting amount 0 to ± 32767 (Detect unit)
When the reference point is shifted, the sign of this parameter is necessary.

	0	8	6	LPGMX

	0	8	7	LPGMY

	0	8	8	LPGMZ

	0	8	9	LPGM4

	4	2	2	LPGM5

LPGMX, LPGMY, LPGMZ, LPGM4, LPGM5

Setting of servo gain multiple of each axis.

$$\text{Setting amount} = 2048 \times \frac{E}{L} \times \alpha \times 1000$$

$$E = \begin{cases} 7 \quad [\mathrm{V}] & \text{(For motor with 7V and 1000 rpm)} \\ 3.5 \quad [\mathrm{V}] & \text{(For motor with 7V and 2000 rpm)} \end{cases}$$

L Machine movement amount per motor revolution (mm or inch)
α Detect unit (mm or inch)
(Example) 2mm per motor revolution at 1000 rpm/7V

$$\text{Setting value:} \quad 2048 \times \frac{7}{2} \times 0.001 \times 1000 = 7168$$

(At detect unit 0.001 mm)

	0	9	0	LPGIN

LPGIN Setting of position control loop gain
Setting amount 1 to 9999 (Unit: 0.01 $\sec^{-1}$)

	0	9	1	JOGF

JOGF JOG feed rate at rotary switch position 10.

Setting amount 1 to 150 Unit: mm/min, deg/min (mm output)
1 to 60 Unit: 0.1 inch/min, 0.1 deg/min (inch output)
1 to 60 Unit: deg/min (inch output)

(Note) Whether the unit on additional axis is 0.1 deg/min or 1 deg/min depends on parameter ROT10 (No. 306).
Also refer to parameter ADNW (318) and JOGFAD (348).

	0	9	2	RPDFX
	0	9	3	RPDFY
	0	9	4	RPDFZ
	0	9	5	RPDF4
	4	2	8	RPDF5

RPDFX, PRDFY, RPDFZ, RPDF4, RPDF5

Rapid traverse of each axis.

Setting amount

30 to 15000 Unit: mm/min (mm output)
30 to 6000 Unit: 0.1 inch/min (inch output)
30 to 15000 Unit: deg/min (for rotational axis)

	0	9	6	LINTX
	0	9	7	LINTY
	0	9	8	LINTZ
	0	9	9	LINT4
	4	2	9	LINT5

LINTX, LINTY, LINTZ, LINT4, LINT5

Time constant of linear acceleration/deceleration of each axis.
(for rapid traverse)
Setting amount 8 to 4000 (Unit: msec.)

	1	0	0	EXPTX

	1	0	1	EXPTY

	1	0	2	EXPTZ

	1	0	3	EXPT4

	4	3	0	EXPT5

EXPTX, Y, Z, 4, 5

Time constant of exponential acceleration/deceleration in manual feeding in order of X, Y, Z and 4th axis.

Setting amount 8 to 4000 Unit: msec.

	1	0	5	FEEDT

FEEDT Time constant of exponential acceleration/deceleration in feed.

Setting amount 8 to 4000 Unit: msec.

	1	0	6	FEDMX

FEDMX Upper speed of cutting feed

(a) In case of parameter ADNW=0 (No. 318) (Specification A)

Tangential feedrate is limited so as not to exceed the set feedrate for all axes in common.

Setting amount: 6~15000 Unit: mm/min, deg/min (metric output)
6~ 6000 Unit: 0.1 inch/min, 0.1 deg/min (inch output)
6~ 6000 Unit: deg/min (inch output)

(Note) Whether the unit on additional axis is 0.1 deg/min or 1 deg/min in inch output depends on parameter ROT10 (No. 306).

(b) In case of parameter ADNW=1 (No. 318) (Specification B)

Effective for X, Y and Z axes. Feedrate along each axis is limited so as not to exceed the set feedrate during linear interpolation or tangential feedrate is limited so as not to exceed the set feedrate during circular interpolation.

Setting amount 6~15000 Unit: mm/min (metric output)
6~ 6000 Unit: 0.1 inch/min (inch output)

(See parameter 366 for the additional axis)

	1	0	7	EXDEC

EXDEC Speed at external deceleration (common to all axes)

Setting amount 6 to 15000 Unit: mm/min (mm output)
6 to 6000 Unit: 0.1 inch/min (inch output)

	1	0	8		FEDFL

FEDFL The lower limit speed (FL) on exponential acceleration/deceleration in feed.
Setting amount 6 to 15000 Unit: mm/min (mm output)
6 to 6000 Unit: 0.1 inch/min (inch output)
Normally, this amount is set to "0".

	1	0	9		JGFLX

	1	1	0		JGFLY

	1	1	1		JGFLZ

	1	1	2		JGFL4

	4	3	1		JGFL5

JGFLX, Y, Z, 4, 5
The lower limit speed (FL) on exponential acceleration/deceleration in manual feeding X axis and Z axis in order.
Setting amount 6 to 15000 Unit: mm/min (mm output)
6 to 6000 Unit: 0.1 inch/min (inch output)

	1	1	3		RPDFL

RPDFL Least speed of rapid traverse override (Fo) (Common to all axes)
Setting amount 6 to 15000 Unit: mm/min, deg/min (metric output)
6 to 6000 Unit: 0.1 inch/min, 0.1 deg/min (inch output)
6 to 6000 Unit: deg/min (inch output)

(Note) Whether the unit on additional axis is 1 deg/min or 0.1 deg/min in inch output depends on parameter ROT10 (No. 306).

	1	1	4		ZRNFL

ZRNFL Low feed speed at reference point return (FL speed) (Common to all axes)
Setting amount 6 to 15000 Unit: mm/min, deg/min (metric output)
6 to 6000 Unit: 0.1 inch/min, 0.1 deg/min (inch output)
6 to 6000 Unit: deg/min (inch output)

(Note) Whether the unit on additional axis is 1 deg/min or 0.1 deg/min in inch output depends on parameter ROT10 (No. 306).

	1	1	5	BKLX

	1	1	6	BKLY

	1	1	7	BKLZ

	1	1	8	BKL4

	4	3	2	BKL5

BKLX, BKLY, BKLZ, BKL4, BLK5

Backlash amount of each axis

Setting amount 0 to 255 Unit: 0.001 mm (metric output)

0 to 255 Unit: 0.0001 inch (inch output)

	1	1	9	SPDLC

SPDLC Set spindle speed offset compensation value, that is, compensation value of zero offset of spindle speed command voltage.

(for S analog output A/B)

Setting amount 0 to ±8191 Unit: VELO

	1	2	1	TLCNEG

TLCNEG Tool life management neglect number

Setting value 1 ~ 255

	1	2	4	DRFTX

	1	2	5	DRFTY

	1	2	6	DRFTZ

	1	2	7	DRFT4

	4	2	3	DRFT5

DRFTX, DRFTY, DRFTZ, DRFT4, DRFT5

Compensation amount of drift generated in servo loop of each axis.
Setting amount 0 to ±500 Unit: VELO
In case of automatic drift compensation set by parameter
ADFT in parameter number 007, this amount is changed automatically.

	1	2	8	PHAZX

	1	2	9	PHAZY

	1	3	0	PHAZZ

	1	3	1	PHAZ4

	4	2	4	PHAZ5

PHAZX, PHAZY, PHAZZ, PHAZ4, PHAZ5

Amount of servo phase shift of each axis.
Data corresponding with phase of signal returning from phase detector is automatically set (for resolver, inductosyn).
Setting amount 0 to 500

	1	3	2	GRLMAX

GRLMAX Setting of the max. spindle speed at low speed gear (for S-12 bit A or S analog output A).
Set the spindle speed when the velocity command voltage is 10V.
Setting amount 1 to 9999 Unit: rpm.

	1	3	3	GRHMAX

GRHMAX Setting of the max. spindle speed at high speed gear (for S-12 bit A or S analog output A).
Set the spindle speed when the velocity command voltage is 10V.
Setting amount 1 to 9999 Unit: rpm.

	1	3	4	GRHMIN

GRHMIN Setting of the lower limit of the spindle speed when using high speed gear. (for S-12 bit A or S analog output A).
Setting amount 1 to 9999 Unit: rpm.

(Note) If the same spindle speed as this setting is specified, low speed gear is selected.

	1	3	5	SPDMIN

SPDMIN Setting of the lower limit of output value to the spindle motor. (for S-12 bit A/B or S analog output A/B).

$$\text{Setting value} = \frac{\text{Lower limit of spindle motor speed}}{\text{Max. spindle motor speed}} \times 4095$$

Setting amount 1 to 4095

	1	3	6	SPDMAX

SPDMAX Setting of the upper limit of output value to the spindle motor.
(for S-12 bit A/B or S analog output A/B).

$$\text{Setting value} = \frac{\text{Upper limit of spindle motor speed}}{\text{Max. spindle motor speed}} \times 4095$$

Setting amount 1 to 4095

	1	4	0	PSANGN

PSANGN Data of adjustment for gain in spindle analog output A/B.
Set the data of adjustment for gain in spindle analog output A/B.
Setting amount: 700 to 1250
Standard amount: 1000
Setting procedure is as follows.
(1) Set the standard amount '1000'.
(2) Command max. value of S analog signal (10V).
(3) Measure the output signal voltage.
(4) Setting amount is as follows.

$$\text{Setting amount} = \frac{10.0}{\text{Measured voltage (V)}} \times 1000$$

(5) After this parameter setting, confirm the output signal voltage is 10V when the max. value of S analog signal is commanded.

	1	4	1	TIME1

TIME1 Preset time for use.
Setting amount 0 to 32767 (Unit: 0.1 H)
Setting is also effective.

	1	4	2	TIME2

TIME2 Preset time for use.
Setting amount 0 to 99999999 (Unit: 0.1 H)

	1	4	3	LT1X1

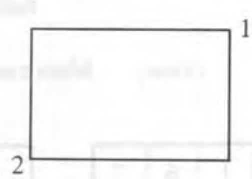
	1	4	4	LT1Y1

	1	4	5	LT1Z1

146	LT141
⋮	⋮
433	LT151
147	LT1X2
148	LT1Y2
149	LT1Z2
150	LT142
434	LT152
151	LT2X1
152	LT2Y1
153	LT2Z1
155	LT2X2
156	LT2Y2
157	LT2Z2

LT □□□

- Represent nth top in square zone
- Represent axis
- Represent nth stroke limit



Set stroke limit mentioned above.
Setting amount 0 to ± 99999999 Unit: 0.001 mm (Metric output)
0 to ± 99999999 Unit: 0.0001 inch (inch output)
SETTING is also effective in the range of 151 to 157.

	1	5	9	REF2X
	1	6	0	REF2Y
	1	6	1	REF2Z
	1	6	2	REF24
	4	3	5	REF25

REF2X, REF2Y, REF2Z, REF24, REF25

Distance from second reference point to first reference point of each axis.

Setting amount	0 to ± 99999999	Unit:	0.001 mm (metric output)
	0 to ± 99999999	Unit:	0.0001 inch (inch output)
	0 to ± 99999999	Unit:	0.001 deg (rotation axis)

	1	6	3	PECINTX
	1	6	4	PECINTY
	1	6	5	PECINTZ
	1	6	6	PECINT4
	4	3	6	PECINT5

PECINTX-5

Pitch error compensation interval for each axis.

Setting amount	8000 to 20000000	Unit:	0.001 mm (metric output)
	4000 to 20000000	Unit:	0.0001 inch (inch output)
	6000	Unit:	0.001 deg (rotation axis)

(Note) When zero is set to this parameter, the compensation is not performed.

	1	6	7	ATCLZV

ATCLZV Z-axis stroke limit in the negative direction

Setting range	0 to ±99999999	Unit 0.001 mm	(Metric output)
	0 to ±99999999	Unit 0.0001 inch	(Inch output)

	1	6	8	Secret number

Register a secret number to be locked in advance.
Setting range: 1 to 99,999,999

Note 1: Whenever parameter 168 is zero, an unlocked state is provided. Turning off/on the NC power does not effect locking. Note that any number other than zero should not be set in parameter 168 for programs not to be locked.

	1	7	1	F1DF1

	1	7	2	F1DF2

	1	7	3	F1DF3

	1	7	4	F1DF4

	1	7	5	F1DF5

	1	7	6	F1DF6

	1	7	7	F1DF7

	1	7	8	F1DF8

	1	7	9	F1DF9

F1DF1, 2, 3, 4, 5, 6, 7, 8, 9

The feed rate to F 1-digit commands F1 ~ F9 in order.

Setting amount 0 to 150000 Unit: 0.1 mm/min. (mm input)
0 to 60000 Unit: 0.01 inch/min. (inch input)

These feed rates can also be set as setting parameter.

Also, in F 1-digit command, if the feed rate is changed by turning the manual pulse generator, the value of this parameter is also changed according to it.

	3	0	5	FL4	FLZ	FLY	FLX	G604	G60Z	G60Y	G60X
				7	6	5	4	3	2	1	0

FLX, Y, Z, 4

Whether the axis is parallel with the 5th axis or not.
(NC/TC option)
1: Parallel axis
0: Not parallel axis

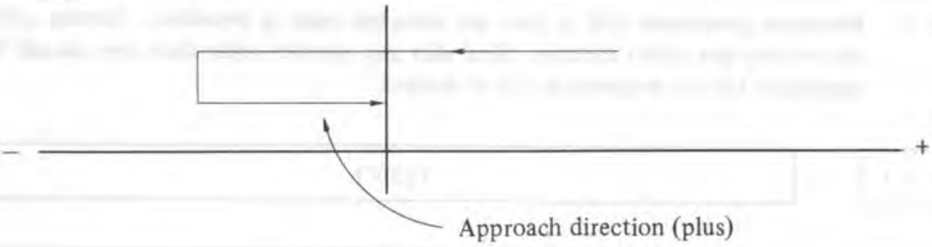
373

G60X, Y, Z, 4

The approach direction by single direction positioning of X axis, Y axis, Z axis and the 4th axis in order.

1: The approach direction is the minus direction.

0: The approach direction is the plus direction.



	3	0	6

SKPF	CHR		SFRV	NEOP	ROT10	TMCR	SALM
7	6	5	4	3	2	1	0

SKPF 1: On skip cutting command (G31), the feed rate comes to FL speed subjected to parameter (No. 342) setting.
0: On skip cutting command (G31), the feed rate follows F code.

CHR 1: Feedrate is limited to rapid traverse during handle interruption.
0: It is not limited to rapid traverse.

SFRV 1: Polarity of analog voltage changes by signal SRV in G84 or G74.
0: Polarity of analog voltage does not change by signal SRV in G84 or G74.

(Note) This parameter is effective only when parameter TCW (No. 010 - bit 7) is 1.

NEOP 1: When a tape is registered in memory, M02, M30, and M99 are not counted as the program end.
0: When a tape is registered in memory, M02, M30, and M99 are counted as the program end.

ROT10 1: The unit of feed rate parameter (091, 106, 113, 114) is 1 deg/min for inch output.
0: It is 0.1 deg/min.

TMCR 1: User macro is called by T code.
0: No user macro is called by T code.

SALM 1: Alarm is occured when S command exceeding lower limit or upper limit of output to spindle motor is executed in S4-digit binary 12 bit output A/analog output A.
0: Alarm is not occurred and the value of lower limit or upper limit is limited.
Refer to parameter 135 and 136, too.

	3	0	7

	EX4NG	SFOB	SCDB	GRST		TLCD	EXRMT
7	6	5	4	3	2	1	0

EX4NG 1: Additional axis disregard signal 4NG is effective.
0: Additional axis disregard signal 4NG is ineffective.

SFOB 1: SF is not issued in S-12 bit output B or in S analog output B.
0: SF is issued in S-12 bit output B or in S analog output B.

SCDB 1: Lower 2 digits of 4 digit S code are issued to B21 to B38 in S4-digit 12-bit output B or analog output B. (This cannot be used if B3-digit function is equipped.)
0: They are not issued.

GRST 1: All the execution data of all groups is cleared when tool change reset signal is input.
0: All the execution data of a group in which tool life has ended is cleared when tool change reset signal is input.

TLCD 1: Tool length compensation is applied to a command axis.
0: Tool length compensation is applied to Z-axis.

EXRMT 1: Selection between RMT and PTR is possible from machine tool.
0: Selection between RMT and PTR is impossible from machine tool.

	3	0	8

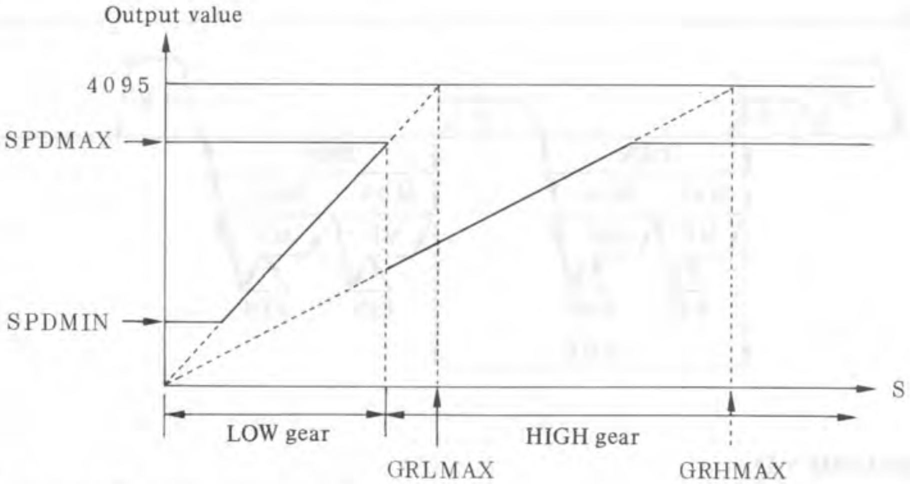
DIOM	MSFT	LGCM	MANP	RSTB	UWKZ	CFMF	
7	6	5	4	3	2	1	0

DIOM 1: DI and DO can be read and written by macro variables. (Not used.)
0: They can not be read and written by macro variables.

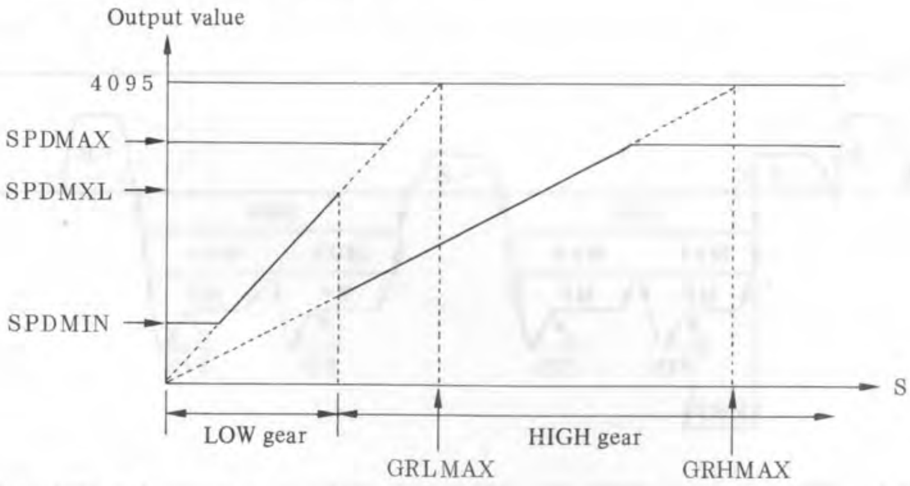
MSFT 1: When the option of user macro is equipped, shift key is valid.
0: Shift key is invalid.

LGCM 1: Specifies the value of parameter SPDMXL (No. 365) for the number of revolutions for low gear/high gear switching (Method B).
0: Specifies the low gear maximum number of revolutions for low gear/high gear switching. (Method A).
(Valid with S-12 bit output A/S analog output A)

Method A (When parameter LGCM = 0)



Method B (When parameter LGCM = 1)



GRLMAX (Parameter No. 132)
GRMMAX (Parameter No. 133)
SPDMIN (Parameter No. 135)
SPDMAX (Parameter No. 136)
APDMXL (Parameter No. 365)

MANP 1: When there is no decimal point in argument of user macro, the argument is regarded as integer. (Not used)

0: When there is no decimal point in argument of user macro, the argument is regarded as value with decimal point obeyed the regulations.

RSTB 1: On resetting by an emergency stop, external reset, or reset & rewind, no resetting signal is issued during the reset.

0: On resetting by an emergency stop, external reset, or reset & wind, a resetting signal is issued during the reset.

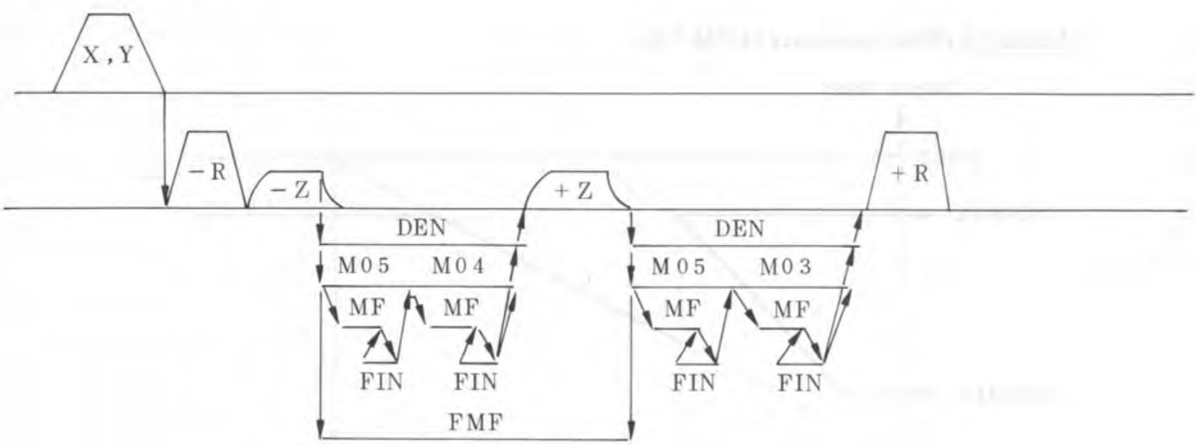
UWKZ 1: When work zero point offset value is changed, absolute coordinate value is charged immediately. (Must not be used).

0: When it is changed, absolute coordinate value is changed at the first preprocessing.

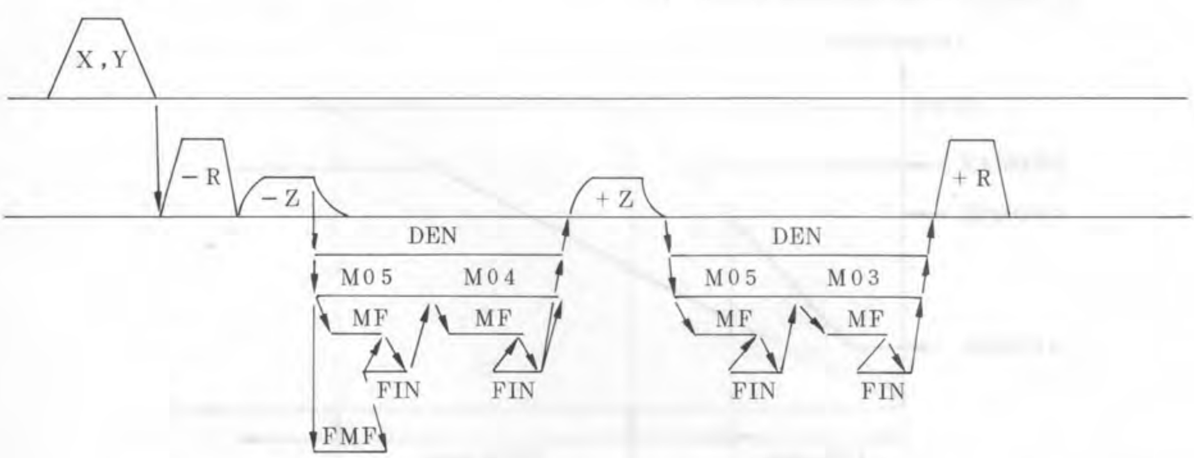
CFMF 1: Turns off FMF by M05 FIN.
(Time chart B)

0: Holds FMF up to the R point level.
(Time chart A)

Time chart A (when CFMF = 0)



Time chart B (when CFMF = 1)



Note 1: When parameter FXCS (No. 022-0) = 1, M05 is not output, but M04 or M03 is output directly.

Note 2: In the case of G74, M04 and M03 are reversed.

3	0	9	TLSK	GST2	GST1	LCTM	AP4	APZ	APY	APX
			7	6	5	4	3	2	1	0

TLSK 1: inputs group No. for tool skip.
0: inputs no group No.

GST1, GST2: specify the number of group that can be registered.

GST1	GST2	Number of groups
0	0	1 ~ 16
0	1	1 ~ 32
1	0	1 ~ 64
1	1	1 ~ 128

LCTM 1: specifies tool life by time.
0: specifies tool life by frequency.

APX, Y, Z, 4

Automatic coordinate system setting (option) for X, Y, Z and 4th axes are effective or ineffective in order.

1: Automatic coordinate system setting is effective.
0: Automatic coordinate system setting is ineffective.

Refer to parameters 375 to 382, too.

3	1	0	NFED1		RSCB1	STP21	BAD1			
3	1	1	NFED2		RSCB2	STP22	BAD2			
3	1	2	NFED3		RSCB3	STP23	BAD3			
3	1	3	NFED4		RSCB4	STP24	BAD4			
			7	6	5	4	3	2	1	0

NFED 1, 2, 3, 4

On I/O devices 1, 2, 3, 4 in order, feed is executed or not.

1: executed
0: not executed

RSCB1, 2, 3, 4

On I/O devices 1, 2, 3, 4 in order, whether control codes (DC1 ~ DC4) are used or not.

1: The control codes are not used.
0: The control codes are used.

STP21, 2, 3, 4

On I/O devices 1, 2, 3, 4 in order, the stop bit is to be 2 bits/1 bit.

1: The stop bit is to be 2 bits.

0: The stop bit is to be 1 bit.

BAD 1, 2, 3, 4

It sets a baud rate on I/O devices 1, 2, 3, 4 in order.

(Note) *Set the baud rate of data transportion between NC and robot to BAD4 when the robot is used.*

Baud rate	BAD1, 2, 3, 4			
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

(Note) *Refer to parameter 340, 341.*

	3	1	4

IM15	MINT	IFIX	IRND	H4	HZ	HY	HX
7	6	5	4	3	2	1	0

IM15 1: Commands for axis B are always regarded as absolute commands whether in G90 mode or in G91 mode, and the rotation direction is positive. When M15 is specified, the rotation direction is negative.

0: Commands for axis B are regarded as absolute or incremental commands, respectively, depending on whether the mode is G90 or G91. The rotation direction is determined as either positive or negative in the same way as for linear axis;M15 is not significant in this case.

MINT 1: Assigned NC statement is executed after the current block is completed. (Custom macro interrupt type II)

0: Assigned NC statement is executed immediately. (Custom macro interrupt type I)

IFIX 1: When the specified angle is not an integer multiple of the least angle for indexing the index table, a PS alarm occurs. (Parameter 060 must also be specified.)

0: Commands for axis B can be issued independently of the least angle for indexing the index table.

IRND 1: The axis B absolute coordinate value is rounded to 360°.

0: The axis B absolute coordinate value is not rounded to 360°.

HX, Y, Z, 4 Handle interruption is

1: effected

0: not effected

	3	1	5

PRT	SLOW	BDEG	IDXB	SSCR	SSCA2	SSCA1	SSCA0
7	6	5	4	3	2	1	0

PRT 1: None is output as leading zeros in DPRNT command.

0: Spaces are output as leading zeros in DPRNT command.

SLOW 1: Minimum spindle speed clamp value is set to all gears in common during constant surface speed control (No. 347)

0: Minimum spindle speed clamp value is set to each gear individually during constant surface speed control. (No. 343, 344, 345, 346)

BDEG 1: B axis input unit 0.001° (B1 = 0.001°)

0: B axis input unit 1° (B1 = 1°)

IDXB 1: Index table indexing sequence B

0: Index Table indexing sequence A

SSCR 1: Surface speed is calculated based on the end point of the current block in a rapid traverse block.

0: Surface speed is calculated based on the current position of the tool in a rapid traverse block.

SSCA2, SSCA1, SSCA0

An axis for which constant surface speed control is performed.

SSCA2	SSCA1	SSCA0	Axis
0	0	0	X
0	0	1	Y
0	1	0	Z
0	1	1	4
1	0	0	5

	3	1	6

CDSCG		ACMR	DSCG5	DSCG4	DSCGZ	DSCGY	DSCGX
7	6	5	4	3	2	1	0

CDSCG 1: It performs no frequency check of DSCG feedback (Resolver, Inductosyn)

0: It performs frequency check of DSCG feedback (Resolver, Inductosyn)
(After field adjustment, this parameter should always be set to "0".)

ACMR 1: Setting of special CMR is possible (Must not be used).

0: It is impossible.

DSCGX, Y, Z, 4, 5

It sets the type of a position detecting system of X axis, Y axis, Z axis and the 4th axis in order.

1: Resolver or inductosyn.

0: Pulse coder.

Following combination is possible.

DSCG4	DSCGZ	DSCGY	DSCGZ	Result
1	0	0	0	X, Y, Z : Pulse coder Additional axis: Resolver/inductosyn.
0	1	1	1	X, Y, Z : Resolver/inductosyn Additional axis: Pulse coder

(Note) Pulse coder and resolver/inductosyn cannot be mixed among X, Y and Z axes.

	3	1	7

UM#8	UM#7	UM#6	UM#5	UM#4	UM#3	UM#2	UM#1
7	6	5	4	3	2	1	0

UM#1–8 On EIA code, it registers the code corresponding to "#" used in custom macro.

Example: UM#8 to UM#1 = 01001001

The code having a hole in channels 1, 4 and 7 is counted as "#" of the EIA code. Address code which has been used can not be set.

(UM#8 to UM#1 = 00000000 mean that '#' is not used.)

379

	3	1	8	PRG9	MSC9	MPD9			NSRH	RSTL	ADNW
				7	6	5	4	3	2	1	0

PRG9 1: Program numbers 9000 to 9899 can not be edited.
0: Program numbers 9000 to 9899 can be edited.

MSC9 1: If the mode is single block mode during execution of program numbers 9000 to 9899, single block stop is effective in executing macro format of user macro.
0: Even if the mode is single block mode during execution of program numbers 9000 to 9899, single block stop is ineffective in executing macro format of user macro.

MPD9 1: The contents of the program is not displayed during execution of program numbers 9000 to 9899.
0: The contents of the program is displayed during execution of program numbers 9000 to 9899.

NSRH 1: Signal OP is not issued during sequence number search.
0: Signal OP is issued during sequence number search.

RSTL 1: When a program is stored in memory by cycle start under EDIT mode, signal STL is not issued.
0: It is issued.

ADNW 1: B specification on feed rate.
0: A specification on feed rate.

[B specification]

(1) Jog feed rate

Jog feed rate along additional axis (rotary axis) can be set by other parameter (No. 348) than that along basic X, Y, Z axes.

However, if the additional axis is moved with another axis or if the additional axis is a linear axis (parameter ADLN = 1 (No. 011, 0-bit)), the jog feed rate along the additional axis is the same as that along other axes. (parameter JOGF No. 091).

(2) Upper limit feed rate

Feed rate along each axis can be limited not to exceed the parameter set feed rate during linear interpolation. (G01). The feed rate to be limited can be set separately between X, Y, Z axes and additional axis.

Tangential feed rate is limited not to exceed the parameter set value as usual during circular interpolation.

[A specification]

(1) Jog feed rate

Feed rate along additional axis is set by parameter No. 091 in common to other axes.

(2) Feed rate upper limit

Feed rate is limited so that the tangential feed rate does not exceed the set feed rate in common to all axes.

	3	1	9	PRG8	MCS8	MPD8					MCS7
				7	6	5	4	3	2	1	0

These can also be set as setting parameter.

PRG8 1: Program Nos. 8000 to 8999 can not be edited.
0: Program Nos. 8000 to 8999 can also be edited.

MCS8 1: If the mode is single block mode during execution of program numbers 8000 to 8999, single block stop is effective in executing macro format of user macro.
0: Even if the mode is single block mode during execution of program numbers 8000 to 8999, single block stop is ineffective in executing macro format of user macro.

MPD8 1: The contents of the program is not displayed during execution of program numbers 8000 to 8999.
0: The contents of the program is not displayed during execution of program numbers 8000 to 8999.

MCS7 1: If the mode is single block mode during execution of program numbers 0001 to 7999, single block stop is effective in executing macro format of user macro.
0: Even if the mode is single block mode during execution of program numbers 0001 to 7999, single block stop is ineffective in executing macro format of user macro.

	3	2	0	UMMCD1
	3	2	1	UMMCD2
	3	2	2	UMMCD3

UMMCD1, 2, 3
It sets up to 3M codes to call user macro.
Setting amount 01 ~ 97
(With M00 the user macro can not be called. Even when 00 are set, it is equivalent to no setting.)

	3	2	3	UMGCD0
	3	2	4	UMGCD1
	3	2	5	UMGCD2
	3	2	6	UMGCD3
	3	2	7	UMGCD4
	3	2	8	UMGCD5
	3	2	9	UMGCD6
	3	3	0	UMGCD7
	3	3	1	UMGCD8

	3	3	2	UMGCD9

UMGCD0, 1, ... , 9

It sets up to 10 G codes to call user macro.
Setting amount 01 ~ 99
(G00 can not call the user macro. Even when 00 is set, it is equivalent to no setting.)

	3	3	3	AOVMDR

AOVMDR Least deceleration ratio of inner circular cutting speed. (MDR)
Range 1 to 100% Standard setting 1

	3	3	4	AOVOR

AOVOR Deceleration ratio of inner corner automatic override.
Range 1 to 100% Standard setting 50

	3	3	5	AOVTH

AOVTH Inner angle of inner corner automatic override.
Range 1 to 179° Standard setting 91

	3	3	6	POSTNX

	3	3	7	POSTNX

	3	3	8	POSTNZ

	3	3	9	POSTN4

	4	1	7	POSTN5

POSTNX, Y, Z, 4, 5

Setting of approach amount for single direction positioning of X axis, Y axis, Z axis and the 4th axis in order.
Setting amount 0 to 255 Unit: 0.01 mm (mm output)
0 to 255 Unit: 0.001 inch (inch output)

	3	4	0

IDVICE

	3	4	1

ODVICE

IDVICE It selects an input device when a program is registered in memory. (When it has been set as INPUT DEVICE 2 = 1 (RS232C) of setting, this setting becomes valid.)

ODVICE It selects an output device on punching out.

Set value	Input/Output Device
0	On input, tape reader On output, FACIT PUNCHER
1	On both input and output, ASR33/ASR43. Parameters such as baud rate, etc. should be set to parameter No. 310.
2	On both input and output, RS232C. Parameters such as baud rate, etc. should be set to parameter No. 311.
3	On both input and output, RS232C. Parameters such as baud rate, etc. should be set to parameter No. 312.
4	On both input and output, RS232C. Parameters such as baud rate, etc. should be set to parameter No. 313.

Above parameters can also be set as setting parameter.

	3	4	2

PSKPFL

PSKPFL FL speed of skip cutting (common to all axes)
Setting amount 6 to 15000 Unit: 1 mm/min. (mm output)
6 to 6000 Unit: 0.1 inch/min. (inch output)

	3	4	3

GRMIN1

	3	4	4

GRMIN2

	3	4	5

GRMIN3

	3	4	6

GRMIN4

GRMIN1 ~ 4

Minimum spindle speed for each gear in the constant surface speed control mode.
Setting value 0 to 9999 Unit: RPM
Effective when parameter 315 SLOW = 0

	3	4	7

LOWSP

LOWSP Minimum spindle speed in the constant surface speed control mode.
(Common to each gear).
Setting value 0 to 9999 Unit: RPM
Effective when parameter 315 SLOW = 1

	3	4	8

JOGFAD

Jog feed rate at rotary switch position 10 along additional axis (rotary axis). (B specification).
Setting amount: 1 to 150 Unit: deg/min
Refer to parameter JOGF (parameter No. 091).

(Note) See parameter ADNW (No. 318) for B specification.

	3	5	5

AOVLE

AOVLE Deceleration distance Le of inner corner automatic override.
Range 0 to 3999 Unit: 0.1 mm Metric input
0.01 inch Inch input
This data can be set by setting operation.

	3	5	6

AOVLS

AOVLS Deceleration distance Ls of inner corner automatic override.
Range 0 to 3999 Unit: 0.1 mm Metric input
0.01 inch Inch input
This data can be set by setting operation.

	3	5	7

EXOFSX

	3	5	8

EXOFSY

	3	5	9

EXOFSZ

	3	6	0

EXOFS4

	4	4	2

EXOFS5

EXOFSX, Y, Z, 4, 5

The external work zero point offset amount of X axis, Y axis, Z axis, the 4th and the 5th axis in order.

Setting amount 0 to ± 7999 Unit: 0.001 mm (mm input)
0 to ± 7999 Unit: 0.0001 inch (inch input)

This parameter is normally automatically set by the input from the machine side. (External data input function)

	3	6	1	PGMAX1

	3	6	2	PGMAX2

	3	6	3	PGMAX3

	3	6	4	PGMAX4

PGMAX1, 2, 3, 4

Setting of the main spindle Max. speed when Gears 1, 2, 3, 4 are selected in order (for S12 bit output B, S analog output B). It sets the main spindle speed when the speed command voltage is 10V.

Setting amount 1 to 9999 rpm.

	3	6	5	SPDMXL

SPDMXL Specifies $4{,}095 \times \frac{\text{Allowable maximum spindle motor speed}}{\text{Maximum spindle motor speed}}$

The setting range is 0 to 4,095.
This setting is significant only when parameter LGCM (No. 308-5) = 1.

	3	6	6	FEDMXAD

FEDMXAD Feed rate upper limit along additional axis. (effective if parameter ADNW=1 (No. 318))

Setting amount 6 to 15000 Unit: deg/min (rotary axis)
6 to 15000 Unit: mm/min (metric output)
6 to 6000 Unit: inch/min (inch output)

	3	6	7	REF3X

	3	6	8	REF3Y

	3	6	9	REF3Z

	3	7	0	REF34

389

	4	3	8

REF35

REF3X, Y, Z, 4, 5

The distance of the 3rd reference point from the 1st reference point of X axis, Y axis, Z axis, the 4th axis, the 5th axis in order.

Setting amount 0 to ±99999999 Unit: 0.001 mm (mm output)
0 to ±99999999 Unit: 0.0001 inch (inch output)

	3	7	1

REF4X

	3	7	2

REF4Y

	3	7	3

REF4Z

	3	7	4

REF44

	4	3	9

REF45

REF4X, Y, Z, 4, 5

The distance of the 4th reference point from the 1st reference point of X axis, Y axis, Z axis, the 4th axis and the 5th axis in order.

Setting amount 0 to ±99999999 Unit: 0.001 mm (mm output)
0 to ±99999999 Unit: 0.0001 inch (inch output)

	3	7	5

PPRTMX

	3	7	6

PPRTMY

	3	7	7

PPRTMZ

	3	7	8

PPRTM4

	4	4	0

PPRTM5

PPRTMX, Y, Z, 4, 5

The coordinates for automatic coordinate system setting on mm input of X axis, Y axis, Z axis, the 4th axis the 5th axis in order. It sets the distance from the zero point of the coordinate system to be set to the 1st reference point in the mm system.

Setting amount 0 to 99999999 Unit: 0.001 mm

When inch/metric conversion option is equipped with, parameter Nos. 379 to 382 should be also set.

Only the axis that is set to be effective for automatic coordinate system setting in parameter 309 is effective.

	3	7	9	PPRTIX
	3	8	0	PPRTIY
	3	8	1	PPRTIZ
	3	8	2	PPRTI4
	4	4	1	PPRTI5

PPRTIX, Y, Z, 4, 5

The coordinates for automatic coordinate system setting on inch input of X axis, Y axis, Z axis the 4th axis the 5th axis in order. It sets the distance from the zero point of the coordinate system to be set to the 1st reference point in the inch system.

Setting amount 0 to 99999999 Unit: 0.0001 inch

When inch/metric conversion option is equipped with, parameter Nos. 375 to 378 should be also set.

Only the axis that is set to be effective for automatic coordinate system setting in parameter 309 is effective.

	3	8	3	ZOFS1X
	3	8	4	ZOFS1Y
	3	8	5	ZOFS1Z
	3	8	6	ZOFS14
	4	4	3	ZOFS15

ZOFSIX, Y, Z 4, 5

The 1st work zero point offset amount of X axis, Y axis, Z axis, the 4th and the 5th axis in order. (G54)

Setting amount 0 to 99999999 Unit: 0.001 mm (mm input)

0 to 99999999 Unit: 0.0001 inch (inch input)

These parameters are usually input using OFSET function button.

389

	3	8	7	ZOFS2X

	3	8	8	ZOFS2Y

	3	8	9	ZOFS2Z

	3	9	0	ZOFS24

	4	4	4	ZOFS25

ZOFS2X, Y, Z, 4, 5

The 2nd work zero point offset amount of X axis, Y axis, Z axis, the 4th axis and the 5th axis in order. (G55)

Setting amount 0 to 99999999 Unit: 0.001 mm (mm input)
0 to 99999999 Unit: 0.0001 inch (inch input)

These parameters are usually input using OFSET function button.

	3	9	1	ZOFS3X

	3	9	2	ZOFS3Y

	3	9	3	ZOFS3Z

	3	9	4	ZOFS34

	4	4	5	ZOFS35

ZOFS3X, Y, Z, 4, 5

The 3rd work zero point offset amount of X axis, Y axis, Z axis, the 4th axis and the 5th axis in order. (G56)

Setting amount 0 to 99999999 Unit: 0.001 mm (mm input)
0 to 99999999 Unit: 0.0001 inch (inch input)

These parameters are usually input using OFSET function button.

	3	9	5	ZOFS4X

	3	9	6	ZOFS4Y

	3	9	7	ZOFS4Z
	3	9	8	ZOFS44
	4	4	6	ZOFS45

ZOFS4X, Y, Z 4, 5

The 4th work zero point offset amount of X axis, Y axis, Z axis, the 4th axis and the 5th axis in order. (G57)

Setting amount 0 to 99999999 Unit: 0.001 mm (mm input)
0 to 99999999 Unit: 0.0001 inch (inch input)

These parameters are usually input using OFSET function button.

	3	9	9	ZOFS5X
	4	0	0	ZOFS5Y
	4	0	1	ZOFS5Z
	4	0	2	ZOFS54
	4	4	7	ZOFS55

ZOFS5X, Y, Z, 4, 5

The 5th work zero point offset amount of X axis, Y axis, Z axis, the 4th axis and the 5th axis in order. (G58)

Setting amount 0 to 99999999 Unit: 0.001 mm (mm input)
0 to 99999999 Unit: 0.0001 inch (inch input)

These parameters are usually input using OFSET function button.

	4	0	3	ZOFS6X
	4	0	4	ZOFS6Y
	4	0	5	ZOFS6Z
	4	0	6	ZOFS64

	4	4	8	ZOFS65

ZOFS6X, Y, Z, 4, 5
The 6th work zero point offset amount of X axis, Y axis, Z axis, the 4th axis, the 5 axis in order. (G59)
Setting amount 0 to 99999999 Unit: 0.001 mm (mm input)
0 to 99999999 Unit: 0.0001 inch (inch input)
These parameters are usually input using OFSET function button.

	4	0	7	SCRATE

SCRATE Scaling rate
Set value 1 to 9999 Unit: 0.001

	4	0	8	Lock/Unlock

Entering the same number as that of parameter 168 effects unlocking. Entering a different number effects locking.

	4	1	1		ZGM5	ZM5
					1	0

ZGM5 5th axis reference point return method
1: Magneswitch method
0: Grid method

ZM5 1: Reference point return direction and backlash initial direction on the 5th axis is minus.
0: They are plus.

	4	1	2		ADW52	ADW51	ADW50	AD5B	AD5A	AD4B	AD4A
				7	6	5	4	3	2	1	0

ADW52, 1, 0 Name of 5th axis

ADW52	ADW51	ADW50	Used character
0	0	0	A
0	0	1	B
0	1	0	C
0	1	1	U
1	0	0	V
1	0	1	W

AD4A, AD4B, AD5A, AD5B
Parallel axis with 4th or 5th axis

5th axis		4th axis		Basic axes parallel with 4th or 5th axis
AD5B	AD5A	AD4B	AD4B	
0	0	0	0	X axis
0	1	0	1	Y axis
1	0	1	0	Z axis
1	1	1	1	None

	4	1	3

CLCL		WNSAA				SELB	SEL

CLCL 1: Local coordinate system is cancelled when manual reference point return is performed.
0: It is not cancelled.

WNSAA 1: A program with its program number being 00 in the higher order two digits and a specified number in the lower order two digits is searched by external work number search A function.
0: A program with its program number being a specified number in the lower order two digits is searched by external work number search A function.

SELB 1: Spec B in multi-handle function.
0: Spec A or C in multi-handle function.

SEL 1: Axes can move only if the contacts of handle axis selection signals are closed. Set 1 for specs B and C and also A if the axis selection signals are to be made valid.
0: Handle axis selection signals are invalid.

	4	1	8

HXNO + HDCNT

HDCNT: The number of the MPGs to be used. Must be specified for all the specs A, B, and C.
Setting value: 2 or 3

HXNO: When using Spec B, specify the MPG number (n-th) to be used for X-axis.
Setting value: 1, 2 or 3

Specify HXNO for the second order position and HDCNT for the first order position.
(Example) If X-axis is operated by the 1st MPG and the number of MPGs is 3: Specify 13.

	4	1	9

HZNO + HYNO

HYNO: In case of Spec B, specify the MPG number (n-th) for Y-axis.
Setting value: 1, 2 or 3

HZNO: In case of Spec B, specify the MPG number (n-th) for Z-axis.
Setting value: 1, 2 or 3

Specify HZNO for the second order position, and HYNO for the first order position.
(Example) If Y-axis is to be operated by the 2nd MPG and Z-axis is to be operated by the 3rd MPG:
Specify the value 32.

	4	2	0	H5NO + H4NO

H4NO: In case of Spec B, specify the MPG number (n-th) for the 4th axis.
Setting value: 1, 2 or 3

H5NO: In case of Spec B, specify the MPG number (n-th) for the 5th axis.
Setting value: 1, 2 or 3

Specify H5NO for second order position, and H4NO for the first order position.
(Example) If the 4th axis is to be operated by the 1st MPG and the 5th axis is to be operated by the 3rd MPG:
Specify 31.

	6	3	3	ROTR						

ROTR 1: Whether R specification is absolute or incremental depends on G90 or G91.
0: R specification is absolute.

	7	1	6	RTANGL

(Setting operation is possible)

RTANGL Angle of coordinate system rotation
Setting value 0 ~ 360000 Unit 0.001 deg.
This value becomes effective when R is not specified in a G68 block.

1000 ~ 1127: Setting of pitch error compensation amount for X axis.
Setting amount 0 ~ ±7

2000 ~ 2127: Setting of pitch error compensation amount for Y axis.
Setting amount 0 ~ ±7

3000 ~ 3127: Setting of pitch error compensation amount for Z axis.
Setting amount 0 ~ ±7

4000 ~ 4127: Setting of pitch error compensation amount for 4th axis.
Setting amount 0 ~ ±7

5000 ~ 5127: Setting of pitch error compensation amount for 5th axis.
Setting amount 0 ~ ±7

The list of parameters which can be set by the setting operation (address SET) is as follows:

Data Number	Content
000	Setting on I/O (RS232C, RMT, INCH, ISO, TVON, REV4, REVY, REVX)
057*	Run hour (TMHOR) (unit: hour)
058*	Run hour (TMMIN) (Unit: minute)
059*	Run hour (TMSEC) (Unit: 2 seconds)
067*	Retract amount (CYCR) in canned cycle G73 (high speed peck drilling cycle)
068*	Cutting start point (CYCD) in canned cycle G83 (peck drilling cycle)
141*	Run hour (TIME1)
151*	Stored stroke limit 2 Amount of X axis at first top
152*	Stored stroke limit 2 Amount of Y axis at first top
153*	Stored stroke limit 2 Amount of Z axis at first top
155*	Stored stroke limit 2 Amount of X axis at second top
156*	Stored stroke limit 2 Amount of Y axis at second top
157*	Stored stroke limit 2 Amount of Z axis at second top
180*	Sequence number at which execution is to be stopped. (Sequence number comparison and stop)
319*	Various setting (PRG8, MSBL)
340*	Selection of input device at storing the program (IDVICE)
341*	Selection of output device at punch out (ODVICE)
355*	Deceleration distance at the end of a block (Automatic corner override)
356*	Deceleration distance at the beginning of a block (Automatic corner override)
407*	Scaling factor

- Select the function key to SET.
- BLANK is displayed in Data No. except as mentioned above.
- Data No. with * is set to the same Data No. in function key PARAM.
- About content, refer to the explanation of Parameter in the same Data No.

APPENDIX 7 ALARM LIST

Number	Content	Remarks
000	Re-apply the power after the parameter was input. (Parameter numbers 012 to 018, 027 to 034, 082 to 090, 124 to 131, 316)	
001	TH alarm (a character with incorrect parity was input in the significant information zone). Correct the tape.	
002	TV alarm (the number of characters in a block is odd). This alarm will be generated only when the TV check is effective. Correct the tape.	
003	Data exceeding the maximum allowable number of digits was input. (See the Section on max. programmable dimensions.)	
004	A numeral, the sign (–) or a decimal point was input without an address at the beginning of a block. (See section III. 10.11 if user macro option is equipped).	
005	The address was not followed by appropriate data but was followed by another address or EOB code.	
006	Sign "–" input error. (Sign "–" was input after an address with which it can't be used. Or two or more "–" signs were input.)	
007	Decimal point " . " input error. (A decimal point was input after an address with which it can't be used. Or two or more decimal points were input.)	
008	The tape reader control switch was set to other than AUTO (without reel) or REEL ON, OFF (with reel).	
009	An invalid character was input in the significant information zone (E).	
010	An invalid G code was specified. (This alarm is also generated when a G code that cannot be used by the controller is specified.)	
011	The feedrate was not specified for cutting feed or the feedrate was inadequate.	
014	A synchronous feed has been specified without thread cutting/synchronous feed option.	
015	The number of the commanded axes exceeded that of simultaneously controlled axes.	
017	The move command of the additional axis was commanded without additional axis control option.	
018	An additional axis was commanded with the other axes simultaneously without additional axis simultaneous control option.	
021	The axis not included in the selected plane (by using G17, G18, G19) was commanded in circular command.	
022	In a circular interpolation command radius designation was performed for an NC which is not equipped with the radius designation option.	
023	In a circular interpolation command by radius designation, zero or a negative value was specified for address R.	
027	Tool length compensation has been applied to an axis without cancelling former tool length compensation.	
028	Two or more axes have been specified in the same direction in a circular interpolation command.	

Number	Content	Remarks
029	An offset value exceeded 6 digits. The offset value should be reset.	
030	The tool offset number was too large for the D, or H code in cutter compensation, tool length compensation or tool offset.	
031	In an offset value input command (by G10 or by custom macro input command), the numerical value under the address P specifying an offset number was too large or P was missing.	
032	In an offset value input command (by G10 or by custom macro input command), the specified offset value was too large.	
033	A point of intersection cannot be determined for cutter compensation C. Or cutter compensation B was programmed on the inside corner less than 90°.	
034	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation.	
035	Skip cutting (G31) was specified in cutter compensation mode.	
036	G45 to G48 (Tool offset) was commanded in cutter compensation mode.	
037	The plane selected (by using G17, G18 or G19) is changed in cutter compensation mode.	
038	Overcutting will occur in cutter compensation because the arc start point or end point coindices with the arc center.	
041	Overcutting will occur in cutter compensation.	
044	One of G27 to G30 was commanded in canned cycle mode.	
045	With NC without reference point return option, ATC cycle (M06) was specified.	
046	2nd, 3rd, and 4th reference point return commands specify other than P2, P3, and P4 respectively.	
047	G code G27 to G30 was specified for an axis which has no reference point.	
048	G30 was specified without performing reference point return after the power was turned on or an emergency stop was executed. A move command was executed without performing reference point return after the power was turned on or after an emergency stop was executed in an NC which includes optional stored stroke limit.	
058	The rpm value is larger than the maximum spindle speed or less than the minimum spindle speed in S4-digit binary 12-bit/analog output A.	
059	The selected workpiece number program cannot be found. (External workpiece number selection function A.)	
060	The specified sequence number was not found in the sequence number search or in the program restart.	
065	Scaling magnification is specified as other than 1-99999.	
066	Scaling is applied and as a result, movement value, coordinate value, circular arc radius, etc. exceed maximum programmable dimension.	
067	G51 (scaling ON) is specified in cutter compensation mode.	
070	The memory area is insufficient.	

Number	Content	Remarks
071	The address searched for was not found.	
072	The number of programs to be stored exceeds 95 or 191.	
073	The program number has already been used.	
074	The program number is other than 1 to 9999.	
075	The program number and sequence number were not found in the first block of the program.	
076	Address P was not specified in a block which includes a M98, G65 or G66 command.	
077	The subprogram was called 3 times (5 times with custom macro option).	
078	The program number (specified program number when called by G, M or T) or the sequence number which was specified by address P in a block which includes an M98, M99, G65, G66 and call by G, M or T was not found. Or the sequence number specified in a GOTO statement was not found.	
079	Stored program and tape contents do not coincide. (Collating program)	
084	Part program editing cannot be performed because of inproper start, end or move destination specification in extended part program edit function.	
085	In reading by the RS232C interface, the number of bits of input data does not match or the baud rate is not correct.	
086	In reading or in output by the RS232C interface, a transmission or I/O device problem occurred.	
087	In reading by the RS232C interface, after DC3 (stop code) was transmitted, data over 10 characters long was input.	
090	The grid method reference point return cannot be performed normally because one-revolution signal (or reference zero point signal for the linear scale) is not input from the pulse coder.	
091	The grid method reference point return cannot be executed normally, because the feedrate is too low to synchronize the one-revolution signal (or reference zero point signal for the linear scale) of the pulse coder with the reference counter.	
092	Motion along the axis specified in G27 has not yet returned to the reference point.	
094	No P type can be specified for program restart (because after program interruption, programming of absolute zero point, ORIGIN, etc. have been operated for).	
095	No P type can be specified for program restart (because after program interruption, external zero offset value has changed).	
096	No P type can be specified for program restart (because after program interruption, the work zero offset value has changed.)	
097	No P type can be specified for program restart (because after the power was turned on or after an emergency stop or stroke limit alarm (instantaneous stop) was released, automatic operation was not executed).	

396

Number	Content	Remarks
098	When program restart was specified with no reference point return after the power was turned on or after the emergency stop or stroke limit alarm was released, G28 was found during block search by a program restart command.	
099	A move command was performed in the MDI mode after the search for program restart.	
100	The parameter enable switch is on. Push the reset button after turning off the switch.	
101	The power was turned off while rewriting the contents of memory in a part program storage / editing operation. When this alarm is generated, you must turn on the power while pushing the DELET and RESET button to clear the memory.	
102	Power has been turned off during tool management data writing.	
110	The absolute value of data of fixed point representation was not within the valid range.	
111	The exponent of data of floating point representation exceeds the upper bound.	
112	Division by zero was specified.	
113	A function that cannot be used by custom macro A was used.	
114	A format error (except in (Format)) occurred.	
115	A value not defined as a variable number was assigned.	
116	The left side of a substituted sentence was invalid.	
118	Nesting of brackets exceeds the upper limit (5).	
119	The argument of SQRT is negative. The argument in BCD is a negative value or is composed of data other than 0 to 9.	
122	Macro nesting is not within the valid range (1 ~ 4).	
123	A mcaro control command was used in tape mode.	
124	DO-END statements do not correspond one to one.	
125	A format error occurred in (Formula).	
126	$1 \leq n \leqq 3$ is not satisfied in a DO n statement.	
127	An NC command and macro command are mixed.	
128	$0 \leq n \leqq 9999$ is not satisfied in a GO TO n statement.	
129	An invalid address was used in (Argument assignment).	
130	In external data input, the greater-address data contained an error.	
131	In an external alarm message, five or more alarms occurred.	
132	In an external alarm message clear, no corresponding alarm number existed.	
133	In an external alarm message or an external operator message, the smaller-address data contained an error.	
134	The coordinate system rotation plane is not of arc or cutter compensation C.	

Number	Content	Remarks
140	The group No. exceeds the maximum value (16, 32, 64 or 128).	
141	The tool group specified in the machining program is not set.	
142	The number of tools in the group exceeds the maximum value that can be registered.	
143	No T code is stored in the block of the program to set tool groups.	
144	H99 or D99 was specified when not using tools of the group.	
145	The T code following M06 does not match the T code corresponding to the tool group being used.	
146	P and L were not specified at the beginning of the program to set the tool group.	
147	The number of the tool groups exceeds the maximum value.	
148	Parameter No. 333, 334, 335 is out of allowable range.	
160	A program being executed has been edited. Or a program being edited has not been selected as a program to be edited. Search the program by program number search before editing.	
170	Programs with numbers 8000 to 8999 and 9000 to 9899 are being edited, but parameter setting inhibits these programs from being edited. (See parameters 318-PRG9, and 319-PRG8.)	
180	A value lower than the decimal point has been specified when the decimal point is used for the axis B command. The specified angles is not an integer multiple of the least angle for indexing the index table.	
181	Axis X, Y, or Z is specified simultaneously with axis B.	
190	Invalid axis has being specified in constant surface speed control.	
210	A movable part of the machine tool has touched the X axis plus stroke limit switch.	
211	A movable part of the machine tool has touched the X axis minus stroke limit switch.	
212	While moving along the X axis in the plus direction, the tool entered the forbidden area of stored stroke limit 1.	
213	While moving along the X axis in the minus direction, the tool entered the forbidden area of stored stroke limit 1.	
214	While moving along the X axis in the plus direction, the tool entered the forbidden area of stored stroke limit 2.	
215	While moving along the X axis in the minus direction, the tool entered the forbidden area of stored stroke limit 2.	
220	A movable part of the machine tool touched the Y axis plus stroke limit switch.	
221	A movable part of the machine tool touched the Y axis minus stroke limit switch.	
222	When moving along the Y axis in the plus direction, the tool entered the forbidden area of stored stroke limit 1.	
223	When moving along the Y axis in the minus direction, the tool entered the forbidden area of stored stroke limit 1.	

Number	Content	Remarks
224	When moving along the Y axis in the plus direction, the tool entered the forbidden area of stored stroke limit 2.	
225	When moving along the Y axis in the minus direction, the tool entered the forbidden area of stored stroke limit 2.	
230	A movable part of the machine tool touched the Z axis plus stroke limit switch.	
231	A movable part of the machine tool touched the Z axis minus stroke limit switch.	
232	While moving along the Z axis in the plus direction, the tool entered the forbidden area of stored stroke limit 1.	
233	While moving along the Z axis in the minus direction, the tool entered the forbidden area of stored stroke limit 1.	
234	While moving along the Z axis in the plus direction, the tool entered the forbidden area of the stored stroke limit 2.	
235	While moving along the Z axis in the minus direction, the tool entered the forbidden area of the stored stroke limit 2.	
240	A movable part of the machine tool touched the 4th axis plus stroke limit switch.	
241	A movable part of the machine tool touched the 4th axis minus stroke limit switch.	
242	While the 4th axis was moving in the plus direction, it entered the forbidden area of the stored stroke limit 1.	
243	While the 4th axis was moving in the minus direction, it entered the forbidden area of the stored stroke limit 1.	
250	The movable part of the machine touched the 5th axis plus stroke limit switch.	
251	The movable part of the machine touched the 5th axis minus stroke limit switch.	
252	While the 5th axis was moving in the plus direction, it entered the forbidden area of the stored stroke limit 1.	
253	While the 5th axis was moving in the minus direction, it entered the forbidden area of the stored stroke limit 1.	
400	The controller received the X, Y or Z axis overload signal.	
401	The READY signal (VRDY) of the velocity controller has turned off.	
402	The control received the additional axis overload signal.	
403	The READY signal (VRDY) of the additional axis velocity control has turned off.	
404	The READY signal (VRDY) of the velocity controller did not turn off even though the READY signal (PRDY) of the position controller turned off. The READY signal (VRDY) of the velocity controller turned on even though the READY signal (PRDY) of the position controller has not yet turned on.	
405	Reference point return might not be performed correctly because of NC or servo system faults. Re-try manual reference point return.	
407	The READY signal (VRDY) of the 5th axis velocity control unit has turned off.	

399

Number	Content	Remarks
410	The value of the position deviation amount on the X axis is larger than the value allowed while the machine is stopped.	
411	The value of the position deviation amount on the X axis is larger than the value allowed while the machine is moving.	
412	The drift amount on the X axis is excessive. (Exceeds 500VELO)	
413	The value of the position deviation amount on the X axis exceeded ±32767. Or the velocity command value of the DA converter is out of the range of +8191 to −8192. Incorrect settings will cause this alarm.	
414	A fault in the position detection system of resolver or inductosyn in the X axis was detected.	
415	A feedrate exceeding 511875 detection unit/sec. was specified for the X axis. An invalid CMR setting causes this error.	
416	The position detector of the X-axis pulse coder is faulty (burn-out alarm).	
417	X axis servo position LSI is faulty.	
420	The value of the position deviation amount on the Y axis is larger than the value allowed while the machine is stopped.	
421	The value of the position deviation amount on the Y axis is larger than the value allowed while the machine is moving.	
422	The drift amount on the Y axis is excessive. (Exceeds 500VELO).	
423	The value of the position deviation amount on the Y axis exceeded ±32767. Or the velocity command value of the DA converter is out of the range of +8191 to −8192. Incorrect settings will cause this alarm.	
424	A fault in the position detection system of resolver or inductosyn on the Y axis was detected.	
425	A feedrate exceeding 511875 detection units/sec. was specified for the Y axis. An invalid CMR setting causes this error.	
426	The position detector of the Y-axis pulse coder is faulty (burn-out alarm),	
427	Y axis servo position LSI is faulty.	
430	The value of the position deviation amount on the Z axis is larger than the value allowed while the machine is stopped.	
431	The value of the position deviation amount on the Z axis is larger than the value allowed while the machine is moving.	
432	The drift amount on the Z axis is excessive. (Exceeds 500VELO).	
433	The value of the position deviation amount on the Z axis exceeded ±32767. Or the velocity command value of the DA converter is out of the range of +8191 to −8192. Incorrect settings will cause this alarm.	
434	A fault in the position detection system of resolver or inductosyn in the Z axis was detected.	
435	A feedrate exceeding 511875 detection unit/sec. was specified for the Z axis. An invalid CMR setting causes this error.	
436	The position detector of the Z-axis pulse coder is faulty (burn-out alarm).	

Number	Content	Remarks
437	Z axis servo position LSI is faulty.	
440	The value of the position deviation amount on the additional axis is larger than the value allowed while the machine is stopped.	
441	The value of the position deviation amount on the additional axis is larger than the value allowed while the machine is moving.	
442	The drift amount on the additional axis is excessive. (Exceeds 500VELO).	
443	The value of the position deviation amount on the additional axis exceeded ±32767. Or the velocity command value of the DA converter is out of the range of +8191 to −8192. Incorrect settings will cause this alarm.	
444	A fault in the position detection system of resolver or inductosyn on the additional axis was detected.	
445	A feedrate exceeding 511875 detection units/sec. was specified for the additional axis. An invalid CMR setting causes this error.	
446	The position detector of the additional axis pulse coder is faulty (burn-out alarm).	
447	Additional axis servo position LSI is faulty.	
450	The content of error register of the 5th axis is larger than the value allowed while the machine is stopped.	
451	The content of error register of the 5th axis is larger than the value allowed while the machine is moving.	
452	Drift of 5th axis is excessive. (Exceeds 500VELO)	
453	The contents of the error register of the 5th axis exceeded ±32767. Or the velocity command value of the DA converter is out of the range of +8191 to −8192. Incorrect setting will cause this alarm.	
454	The resolver/inductosyn position detecting system of 5th axis is trouble.	
455	A feedrate exceeding 511875 detection units/sec. was commanded in the 5th axis. An incorrect setting of CMR causes this alarm.	
456	5th axis pulse coder position feedback is abnormal. (Disconnection alarm)	
457	5th axis servo position LSI is faulty.	
600	A data transfer error took place in the connection unit.	
601	Slave ready has turned off.	
602	The PC program has not yet been loaded. (PC-Model A only)	
603	The correspondence between NC and PC is invalid or was interrupted.	
604	No hold is effective for the PC model B MPU.	
605	A system error occurred in the PC model B MPU (watch dog timer alarm).	
606	A RAM/ROM parity error occurred in the PC model B MPU.	
607	A data transfer error took place in MDI & CRT.	

Number	Content	Remarks
700	Master PCB overheated.	
701	The additional axis PCB overheated.	
702	The X, Y or Z-axis DC motor overheated.	
703	The 4th axis DC motor has overheated.	
704	The 5th axis DC motor has overheated.	
900	A fault occurred in the bubble device)	NC alarm
901	A fault occurred in the bubble device (The initial point in the bubble was not detected immediately after power was turned on.)	NC alarm
902	A fault occurred in the bubble device (Page size error, undefined command).	NC alarm
903	A fault occurred in the bubble device (Transfer missing)	NC alarm
904	A fault occurred in the bubble device (Parity error).	NC alarm
905	A fault occurred in the bubble device (No marker).	NC alarm
906	A fault occurred in the bubble device (Many defect loops).	NC alarm
907	A fault occurred in the bubble device (Data cannot be written correctly).	NC alarm
908	A fault occurred in the bubble device (Data cannot be written correctly).	NC alarm
909	A fault occurred in the bubble device (Bubble device stop).	NC alarm
910	A RAM parity error (low byte) occurred.	NC alarm
911	A RAM parity error (high byte) occurred.	NC alarm
912	A fault occurred in the bubble device (Abnormal signal is being issued).	NC alarm
920	A system error (watch dog timer alarm) occurred.	NC alarm
930	A CPU error (0, 3, 4, type interrupt generation) occurred.	NC alarm
940	A memory error for offset value occurred (abnormally large offset value set.) Set the correct value in the specified offset number.	NC alarm
950	Clock alarm (Master PCB clock is abnormal)	NC alarm
960	The memory area to temporary load system control commands is insufficient.	NC alarm.
961	CPU error (INT command execution)	NC alarm
996	RAM is not mounted although an option which requires an additional RAM is equipped.	NC alarm
997	A ROM parity error (PC ROM) occurred.	NC alarm
998	A ROM pair error (Basic ROM) occurred.	NC alarm
999	A ROM pair error (no correspondence between high and low) occurred.	NC alarm

APPENDIX 8 STATUS AT TURNING POWER ON, AT RESET AND AT CLEAR

O : The status is not changed or the movement is continued.
X : The status is canceled or the movement is interruped.

	Item	At turning power on	Clear state	Reset state
Setting data	Offset amount	O	O	O
	Setting data	O	O	O
	Parameter	O	O	O
Data	Program in the memory	O	O	O
	Contents in the buffer	X	X	In MDI mode O Other than MDI mode X
	Sequence number display	X	O (Note 1)	O (Note 1)
	One-shot G code	X	X	X
	Modal G code	Initial G code (G20/G21 remains before turning power off)	Initial G code (G20/G21 is not changed)	No change
	F function	Zero	Zero	O
	S, T, M, B function	X	O	O
	Address L (Repetitive count)	X	X	In MDI mode O Other than MDI modeX
Coordinate system	Work coordinate value	Zero	O	O
Coordinate system	Movement	X	X	X
	Dwell	X	X	X
	Sending of M, S, T, B code	X	X	X
	Tool length compensation	X	By parameter "RS43"	In MDI mode O Other than MDI mode, by parameter "RS43"
Executing movement	Cutter radius compensation	X	X	In MDI mode O Other than MDI mode X
	Memorization of called subprogram number	X	X (Note 2)	In MDI mode O Other than MDI mode X (Note 2)
	Rewind	X	X	X

	Item	At turning power on	Clear state	Reset state
Display LED and output signal	ALM	If there is no alarm, extinguishes	Same as left	Same as left
	NOT READY	X	X (flickering by emergency stop)	X (flickering by emergency stop)
	LSK	Lights	Lights	In MDI mode O Other than MDI mode Light
	BUF	Extinguishes	Extinguishes	In MDI mode O Other than MDI mode Extinguishes
	Reference point return LED	X	O (In emergency stop X)	O (In emergency stop X)
	S, T, B code	X	O	O
	M code	X	X	X
	M, S, T, B strobe signal	X	X	X
	Spindle rotation signal (S-12 bit/S analog)	O	O	O
	NC ready signal (MA, MB)	ON	O	O
	Servo ready signal	ON (Other than servo alarm)	ON (Other than servo alarm)	ON (Other than servo alarm)
	Cycle start LED	X	X	X
	Feed hold LED	X	X	X

(Note 1) When returning to the start of program, the main program number is displayed.

(Note 2) When the NC is reset during subprogram execution, the control returns to the block next to subprogram call in main program.

The control can return to the start of the main program. However, subprogram cannot be executed from the half way.

APPENDIX 9 STORED PITCH ERROR COMPENSATION FUNCTION

9.1 Function

Pitch error can be compensated by the least command increment for each axis. This function becomes effective after reference point return.

9.2 Specification

The tool position after reference point return is called compensation zero point. The compensation amount must be set to the parameter corresponding to the each compensation point.

(1) Compensating axis: X, Y, Z and additional axis

(2) Number of compensation points
Linear axis: 128 points (X, Y, Z)
Rotational axis: 61 points (additional axis)

(3) Compensation amount
0 to ±7 x (Compensation magnification) (Lease command increment)
Compensation magnification x 1, x 2, x 4, x 8 (common to all axes)

(4) Compensation interval

	Min. setting interval	Max. setting interval	Unit
Metric system	8000	20000000	0.001 mm
Inch system	4000	20000000	0.0001 inch

(Max. compensation range = Setting interval x 128)

Set the proper compensation interval according to the relation between max. compensation range within the above range and the machine stroke.

When the additional axis is used as a rotational axis, set 6000 as compensation interval. If other value is set, the compensation sometimes cannot be performed correctly.

In this case, the feed rate along the rotational axis should be 110000 deg/min (31.2 rpm) or less.

If the value less than min. setting value of interval is set in pitch error compensation for linear axis, the compensation sometimes cannot be performed correctly. In the above case, lower the feedrate.

9.3 Parameter

Parameters corresponding to pitch error comepnsation should be set in MDI mode or in emergency stop status. The parameter contents and parameter number are as follows.

(1) Pitch error compensation magnification

		2	4

Parameter number

PML2	PML1						

The value, with this magnification multiplied to the set compensation value, is outputted.

PML2	PML1	Magnification
0	0	x 1
0	1	x 2
1	0	x 4
1	1	x 8

(Common to all axes)

(2) Pitch error zero point

		3	9	PECZRX
		4	0	PECZRY
		4	1	PECZRZ
		4	2	PECZR4
	4	1	6	PECZR5

PECZRX ~ 5 Pitch error zero point

Pitch error zero point is the zero point for pitch error compensation. It corresponds to the reference point in the machine coordinate system. Set the value from 0 to 127 for each axis according to the machine tool.

(3) Pitch error compensation interval

0	1	6	3	PECINTX
0	1	6	4	PECINTY
0	1	6	5	PECINTZ
0	1	6	6	PECINT4
	4	3	6	PECINT5

PECINTX ~ 5 Pitch error compensation interval

Set the positive amount 8000 or more in metric system, or 4000 or more in inch system.
For rotational axis, set '6000'.
When zero is set to these parameter, compensation is not performed.

(4) Compensation amount

Compensation amount of each axis and point is set to a corresponding parameter number.

Axis name	Parameter number
X axis	1000 ~ 1127
Y axis	2000 ~ 2127
Z axis	3000 ~ 3127
4th axis	4000 ~ 4127
5th axis	5000 ~ 5127

For the other parameter number shown above, the compensation amount cannot be set. Setting amount is 0 to ±7 and the other values are neglected.

(Example)

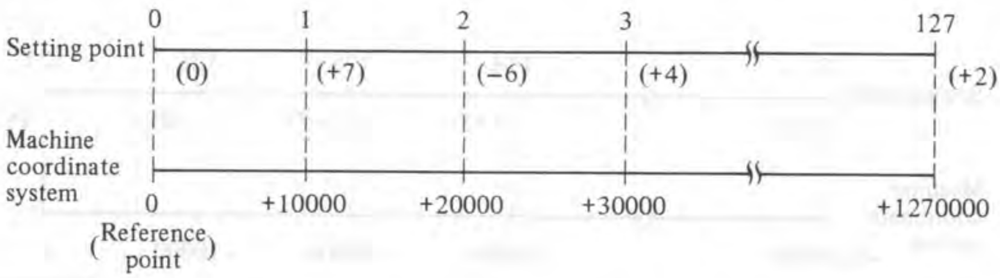
1	1	2	0	—							7
				7	6	5	4	3	2	1	0

In the above example, this indicates that the compensation data –7 is set to the setting point 120 of X axis. As the displayed parameter number can be changed one by one, by using CURSOR ↑ and ↓ keys, the compensation data of right or left setting point can be displayed.

9.4 Example

(1) Example 1

Pitch error zero point = 0
Compensation interval = 10000



The beginning of the compensation table corresponds to the reference point. The set point 1 is at +10000 position and the other setting points correspond to the positions on the machine coordinate system which is shifted +10000 each other. Therefore, setting point 128 is at +1270000.

Compensation amount when the machine moves from 0 to +10000 is set to the setting point 1 and that when the machine moves from +10000 to +20000 is set to the setting point 2. Generally, compensation amount when the machine moves from (n - 1) x (compensation interval) to n x (compensation interval) is set to the setting point n.

In the above example, compensation data is

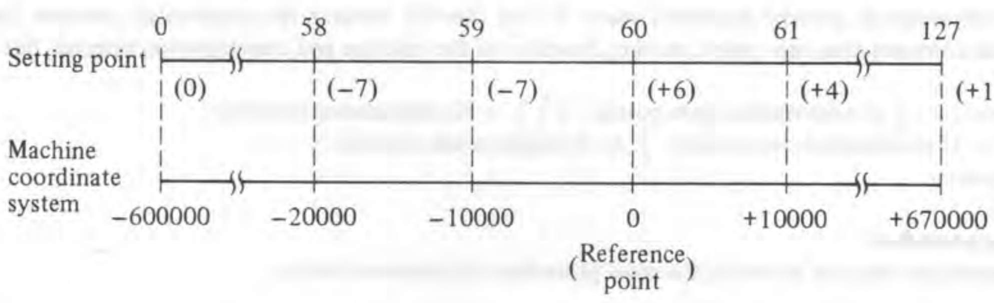
- 7 between 0 and 10000
+6 between 10000 and 20000
- 4 between 20000 and 30000

If the machine moves from the reference point to +30000, total compensation amount is as follows.

(+7) + (-6) + (+4) = +5

(2) Example 2

Pitch error zero point = 60
Compensation interval = 10000



The setting point 60 corresponds to the reference point. The setting point 61 is at +10000 position and the other setting points corresponds to the position which is shifted +10000 each other.

And, the setting point 59 is at –10000 from the reference point and the other setting points correspond to the positions which is shifted -10000 each other, that is the setting point 0 is at -600000.

Compensation amount when the machine moves from (n - 61) x (compensation interval) to (n - 60) x (compensation interval) is set to the setting point n.

In the above example, compensation data is as follows.

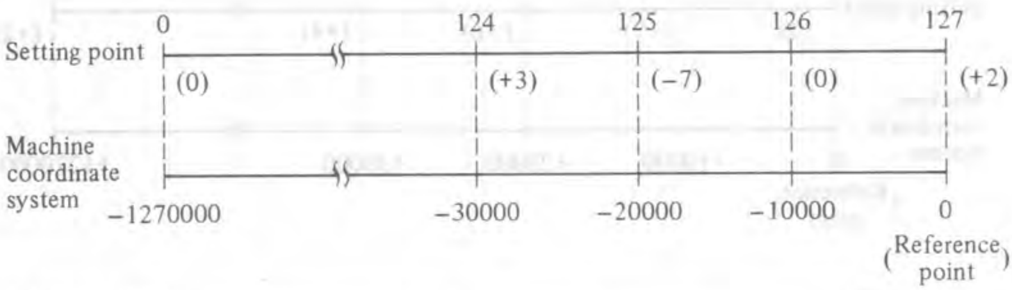
+7 between -30000 and - 20000
+7 between -20000 and - 10000
- 6 between -10000 and 0
- 4 between 0 and +10000

If the machine moves from -30000 to +10000, total compensation amount is as follows.

(-7) + (-7) + (+6) + (+4) = (-4)

(3) Example 3

Pitch error zero point = 127
Compensation interval = 10000



The end of the compensation table corresponds to the reference point and the setting point 126 is at -10000 from the reference point and the other setting points correspond to the position which is shifted -10000 each other.

Compensation amount when the machine moves from -10000 to 0 is set to the setting point 127 and that when the machine moves -20000 to -10000 is set to the setting point 126. Generally, compensation amount when the machine moves from (n - 128) × (compensation interval) to (n - 127) × (compensation interval) is set to the setting point n.

In the above example, compensation data is as follows.

- 3 between - 40000 and - 30000
+7 between - 30000 and - 20000
0 between - 20000 and - 10000
- 2 between - 10000 and 0

If the machine moves from -40000 to the reference point, total compensation amount is as follows.

(+3) + (-7) + (0) + (+2) = (-2)

9.5 Setting Procedure for Compensation Data

The compensation amount described above do not directly concern the relationship between the reference point and the compensation zero point, moving direction of the machine and compensation interval, but mechanical error.

between [n - { (Compensation zero point) + 1 }] × (Compensation interval)
and { n - (Compensation zero point) } × (Compensation interval)

for setting point n.

(1) Setting procedure

Compensation data can be set by the same procedure of parameter setting.

(a) Delection of compensation amount

If –9999 is input to the parameter on any one of parameter number of desired axis, all compensation amount of desired axis become zero.

(b) Punch out of compensation amount

They are punched out by the same procedure of parameter punch out. However, in the procedure, input -9998 instead of -9999.

Punching out of the desired axis cannot be performed.

(2) Precautions in setting

(a) Compensation interval (Parameter number 163 to 166)

If the compensation interval is positive value, compensation is performed by the set value. If it is negative value, compensation is performed by the absolute value of it. And if it is zero, the compensation for the axis is not performed.

(Even if it is negative value, it is displayed as positive value.)

(b) The pitch error compensation becomes effective after reference point return. If reference point return is not performed, it is not effective after parameter setting. The parameter setting should be performed after turning power on and before reference point return. If the parameter for pitch error compensation is altered after reference point return, perform reference point return again.

(c) Pitch error compensation amount
(Parameter number 1000 ~ 4127)
The limitation of the compensation amount is as follows. The following amount should be within ±127.
(Pitch error compensation amount) × (Compensation magnification) × CMR

(Note) CMR: Command multiply
For details, refer to parameter with parameter number 27 to 30.

If the above amount is out of ±127, the compensation is not performed correctly. If the compensation amount which is out of ±127 is necessary, divide it among setting points before and after it.

9.6 Pitch Error Compensation for Rotational Axis

When the pitch error compensation is executed in 4th axis (rotation axis), the parameter setting should be done as follows.

Parameter number	Contents	Setting value
042	Compensation zero point	0
166	Compensation interval	6000

That is, the circle is divided into 60 segments and compensation is performed per 6 degrees rotation. The compensation data should be set to the following parameters.

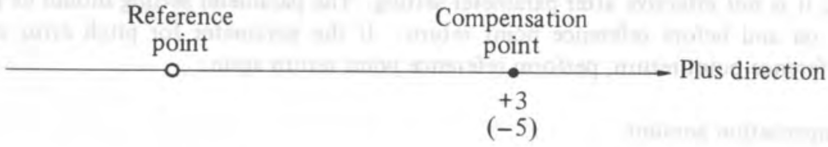
Parameter number	Setting value
4000	Compensation amount between -6 and 0 deg.
4001	Compensation amount between 0 and 6 deg.
4002	Compensation amount between 6 and 12 deg.
≀	≀
4059	Compensation amount between 348 and 354 deg.
4060	Compensation amount between 354 and 360 deg.

The same value should be set to parameter number 4000 and 4060. The pitch error compensation of rotational axis must be performed at rapid traverse of 110000 deg./min. (31.2 rpm) or less.

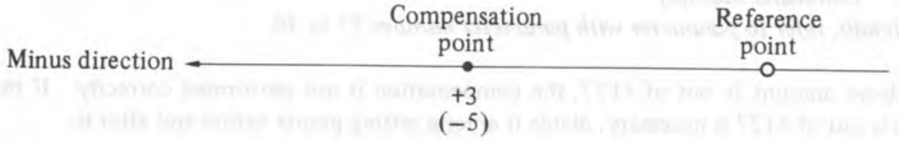
Sign of compensation amount

The sign of pitch error compensation amount is decided against moving direction. If positive compensation value is set, the move amount is increased by it. If negative compensation value is set, the move amount is decreased by it. That is, when the machine move amount has plus or minus error against the move command, minus or plus compensation amount should be set respectively.

(Example)



When the machine arrives at the compensation point moving in plus direction, the move amount is compensated by +3 (-5). In minus direction, the move amount is compensated by -3 (+5).



When the machine arrives at the compensation point moving in plus direction, the move amount is compensated by +3 (-5). In minus direction, the move amount is compensated by -3 (+5).
That is, the sign of the compensation amount is decided by the moving direction in compensation. It does not concern the position of the compensation zero point.

APPENDIX 10 OPERATION TABLE

Classification	Function	Key Switch	Parameter Enable Switch	Mode Switch	Function key	Operation
Clear	Bubble Memory All Clear		○	Power ON	–	[O] + [DELET]
	Clearing Parameter		○	Power ON	–	[CAN] + [DELET]
	Clearing Stored Program			Power ON	–	[RESET] + [DELET]
Data Input from Tape	Parameter		○	Emergency Sw. ON	PARAM	[P] → –9999 → [READ]
	Offset Value			TAPE	–	[START]
	Program Input	○		EDIT	–	([O] → Program No.) → [READ]
	Add Program	○		EDIT	–	[O] → [CAN] → [READ]
	Many Program Registration			EDIT	–	[O] → –9999 → [READ]
	Pitch Error Compensation		○	Emergency Sw. ON	PARAM	[P] → –9999 → [READ]
Data Input fron. MDI	Parameter		○	MDI	PARAM	[N] → Parameter No. → [INPUT] → [P] → Data → [INPUT] → Parameter Enable Sw. OFF → [RESET]
	Offset value	○		Any mode (without EDIT)	OFSET	[N] → Offset No. → [INPUT] → [P] → Offset Data → [INPUT]
	Setting Data	○		MDI	SET	CURSOR to Set No. → [P] → Data → [INPUT]
Tape Punch	Parameter			EDIT	PARAM	[P] → –9999 → [PUNCH] (Note)
	Offset value			EDIT	OFSET	[P] → –9999 → [PUNCH]
	Pitch Error Compensation			EDIT	PARAM	[P] → –9998 → [PUNCH]
	All Programs			EDIT	–	[O] → –9999 → [PUNCH]
	One Program			EDIT	–	[O] → Program No. → [PUNCH]
Search	Program No. Search (MEMORY mode only)			EDIT	PRGRM	① [O] → Program No. → [↓] (CURSOR) ② [O] → [CAN] → [↓] (CURSOR)

411

Classification	Function	Key Switch	Parameter Enable Switch	Mode Switch	Function Key	Operation
Search	Sequence No. Search (MEMORY mode only)			MEMORY	PRGRM	Program No. Search → [N] → Sequence No. Search → [↓] (CURSOR)
	Sequence No. Search (TAPE mode)			TAPE	PRGRM	[N] → Sequence No. → [↓] (CURSOR)
	Address word search (MEMORY mode only)			EDIT	PRGRM	Searching Address and Data Input → [↓] (CURSOR)
	Address search (MEMORY mode only)			EDIT	PRGRM	Searching Address → [↓] (CURSOR)
Program Editing	Deletion of all Programs	○		EDIT	PRGRM	[O] → −9999 → [DELET]
	Deletion of a Program	○		EDIT	PRGRM	[O] → Program No. → [DELET]
	Deletion of several Blocks	○		EDIT	PRGRM	[N] → Sequence No. → [DELET]
	Deletion of a Block	○		EDIT	PRGRM	Search the Block to be deleted → [EOB] → [DELET]
	Deletion of a word	○		EDIT	PRGRM	Search the word to be deleted → [DELET]
	Alteration of a word	○		EDIT	PRGRM	Search the word to be altered → Address → Data → [ALTER]
	Insertion of a word	○		EDIT	PRGRM	Search the word before the place in the program → Address → Data → [INSRT]
	Arrangement of Memory	○		EDIT	(PRGRM)	[CAN] → [ORIGIN]
Comparison	Comparison in Memory with Tape			EDIT	(PRGRM)	[/] → [READ]
	Comparison from Current Position			EDIT	(PRGRM)	[EOB] → [READ]
Bubble cassette	Heading of file — Heading of cassette			EDIT mode	PRGRM	[N] → 0 (Zero) → [INPUT]
	Heading of file — Next file			EDIT mode	PRGRM	[N] → −9999 → [INPUT]
	Heading of file — Automatic heading of next file			EDIT mode	PRGRM	[N] → −9998 → [INPUT] (Modal)
	Heading of file — File no. designation			EDIT mode	PRGRM	[N] → File no. → [INPUT]
	Deletion of file	○		EDIT mode	PRGRM	[N] → File no. (k) → [START] File no. (k + 1) ~ n → k − (n − 1)
	Output of one program			EDIT mode	PRGRM	[O] → Program no. → [PUNCH]
	Output of all programs			EDIT mode	PRGRM	[O] → −9999 → [PUNCH]
	Loading of programs			EDIT mode	–	Heading of file → [O] → −9999 → [READ]
	Output of offset data			EDIT mode	OFSET	[P] → −9999 → [PUNCH]
	Input of offset data			EDIT mode	–	Heading of file → [O] → Program no. → [READ] → Program execution
	Output of parameters			EDIT mode	PARAM	[P] → −9999 → [PUNCH]
	Input of parameters		○	Emergency stop ON	PARAM	Heading of file → [P] → −9999 → [READ]

412

APPENDIX 11 PROGRAM KEY LOCK

11.1 Introduction

Programs 9000 to 9899 can be locked. While locked, these programs cannot be displayed, edited, and punched out. This function protects the secrecy of special programs created by the machine builder by using macros and prevents them from being erased by mistake.

11.2 Program Number

Only programs 9000 to 9899 can be locked. According to a certain method, described later, they are locked automatically. Therefore, programs not to be locked must be assigned with numbers other than 9000 to 9899.

11.3 Locked Conditions

In a locked state (see Section 4), programs 9000 to 9899 are as follows:

(1) The contents are not displayed even during execution.
(2) Program number search in the edit (memory) mode cannot be performed (alarm 071). Therefore, nor can editing.
(3) Memory arrangement is not carried out.
(4) The program numbers are not included in All Program Number Display.
(5) The programs cannot be punched out (even by All Program Punch-Out).
(6) The programs cannot be cancelled (even by All Program Cancel).
(7) The programs cannot be registered in the memory (alarm 170).

11.4 Locking and Unlocking Method

(1) Set the secret numbers (1 to 99,999,999) in parameter 168 in advance. Note the contents of this parameter. Setting zeros does not effect locking.

Note 1: This parameter can be set only in an unlocked state.
Note 2: This parameter cannot be cancelled even by All Parameter Clear.
Note 3: This parameter is cleared to zero by Bubble All Clear, that is, an unlocked state is entered.

(2) For unlocking, enter into parameter 408 the same number as that set in parameter 168.

Note 1: The contents of this parameter are not displayed.
Note 2: This parameter is not stored in the bubble memory.

(3) To lock following unlocking, take either of the following actions.
(a) Enter into parameter 408 a number different from that set in parameter 168.
(b) Turn the NC power off once and on again.

Parameter No.

	1	6	8	Secret number

Register a secret number to be locked in advance.
Setting range: 1 to 99,999,999

	4	0	8	Lock/Unlock

Entering the same number as that of parameter 168 effects unlocking. Entering a different number effects locking.

Note 1: Whenever parameter 168 is zero, an unlocked state is provided. Turning off/on the NC power does not effect locking. Note that any number other than zero should not be set in parameter 168 for programs not to be locked.

11.5 Notes

(1) If the set secret number (parameter 168) has become unknown, take the following action.

(a) Perform Bubble All Clear (unlocked).

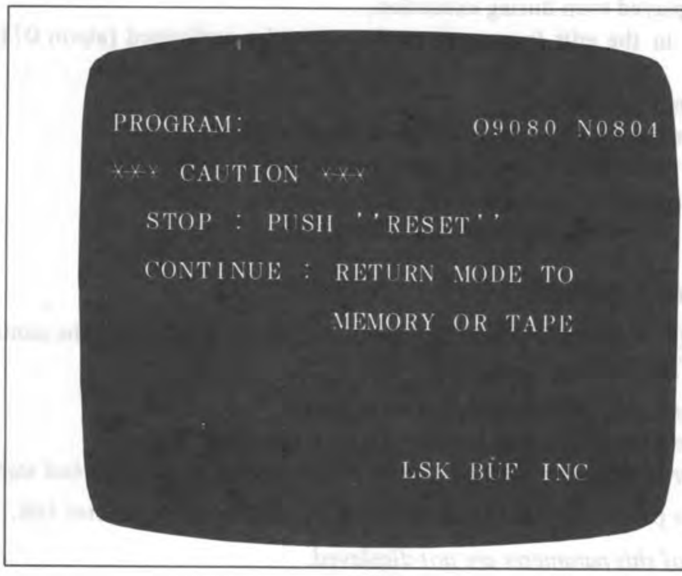
(b) Enter all parameters (except parameter 168).

(c) Store the secret program in the memory.

(d) Set a secret number in parameter 168 (locked).

(2) For replacement of the faulty bubble memory, also follow the operation (2) above. For quick recovery, secret programs must be attached to the NC equipment.

(3) If the mode is changed to EDIT during execution of a locked program, the PRGRM screen will be displayed as follows. That is, the program will not be displayed. Execute reset before displaying and editing of other programs. (Executing reset does not permit the program to be continued.) Executing reset returns the screen to an ordinary program display. To continue the program, return the mode to MEMORY or TAPE and resume machining by pressing the cycle-start button.



(4) After registering or editing programs 0900 to 09899, call out a different program and effect locking. Locking, while programs 09000 to 09899 are displayed, provides the screen with the figure above in the edit mode. Applying reset displays the program.

APPENDIX 12 CUSTOM MACRO INTERRUPT FUNCTION

1. OUTLINE

Entering an interrupt signal (UNIT) from the machine during execution of a program permits call-out of another program. This function is called the macro interrupt function.

The interrupt command is executed by specifying, in programming,

M96 Pxxxx ; (Custom macro interrupt on)
M97 ; (Custom macro interrupt off)

This function permits any execution block of the program to call out another program, thus enabling program operations according to the ever changing conditions.

[Applications]

1. Processing for tool error detection is initiated by an external signal.
2. Another program is inserted in a machining series without the stopping of machining operation.
3. The current machining information is read at fixed time intervals, etc.

Thus, adaptive control-like applications are possible.

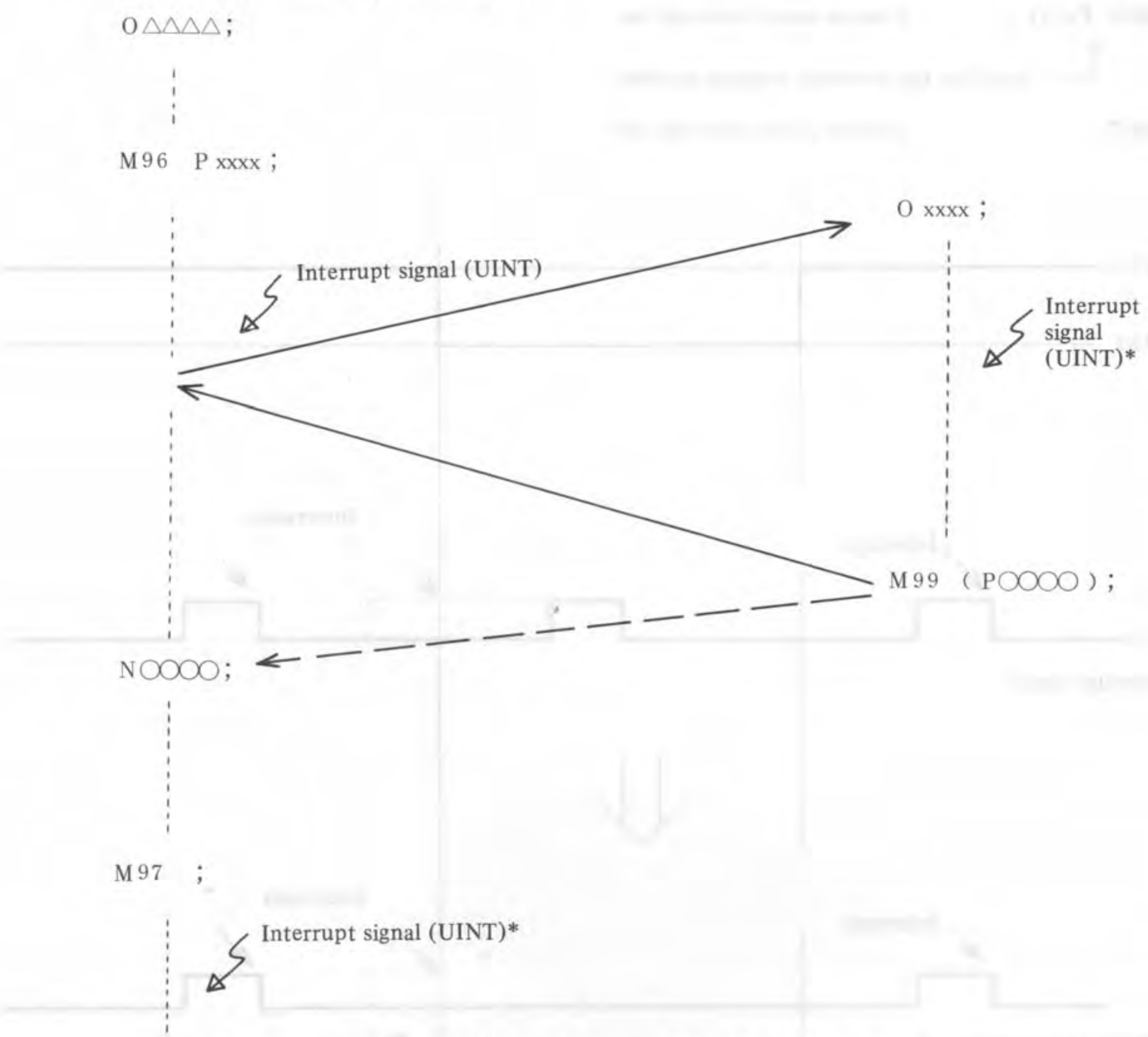


Figure 1 Outline of custom macro interruption

The figure shows that specifying M96 Pxxxx in programming permits a program specified by Pxxxx to interrupt the current program and be executed, when the interrupt signal (UNIT) is entered. The interrupt signal is ignored during execution of the interrupt program and after M97 (interrupt signals marked *).

415

2. COMMANDING

2.1 Effective Conditions

Custom macro interruption can be used only during program execution. Therefore, it is effective when:

1. The memory, tape, or MDI mode has been selected;
2. The start lamp (STL) is on;
3. The custom macro interruption is not in progress.

The macro interruption cannot be applied during manual operations (jog, step, handle, etc.).

2.2 Command Format

The custom macro interrupt function is, in principle, executed by making the interrupt signal effective and ineffective by M96 and M97. That is, the interrupt signal entered until M97 is commanded or the NC is reset, after M96 is commanded, initiates the custom macro interruption. On the other hand, entering the interrupt signal after M97 is commanded or after the NC is reset, does not initiate the custom macro interruption, but the interrupt signal entered until M96 is commanded is ignored.

[Format]

M96 Pxxxx ; Custom macro interrupt on

└── Specifies the interrupt program number.

M97 ; Custom macro interrupt off

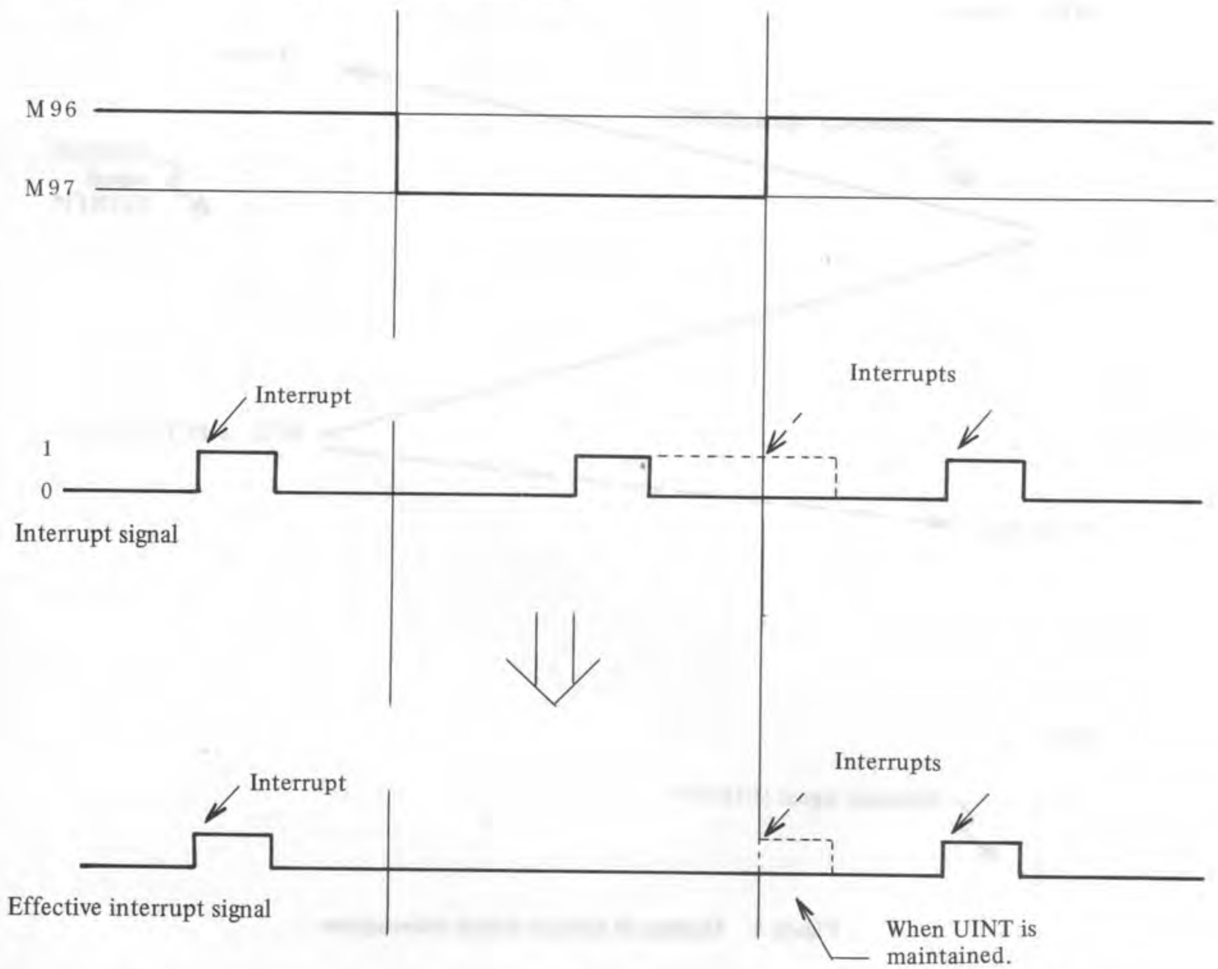


Figure 2.2 Relations between M96, M97, and interrupt signal

The interrupt signal becomes effective after M96 is entered. It is ignored after M97 is commanded, but if the signal entered after M97 is commanded is maintained until M96 is commanded, it initiates the custom macro interruption immediately after M96 is commanded.

3. DETAILED DESCRIPTIONS

3.1 Custom Macro Interruption • Enable/Mask

When the custom macro interrupt function is not to be used, it can be assumed that the program is not to be modified. Thus, the parameter which selects between Enable and Mask for the custom macro interrupt function is available (025-MUSR).

When this parameter specifies that the custom macro interrupt function is to be masked, M96 and M97 become ordinary M codes and are output externally, but when it specifies that the function is to be enabled, they are processed internally and are not output externally.

3.2 Subprogram-type Interrupt and Macro-type Interrupt

The custom macro interrupt method includes subprogram type and macro type. Thus, the parameter which selects between the two types is available (025-MSUB).

- Subprogram type
  The interrupt program is called out as a subprogram. That is, the local variable level does not change before and after the interruption. Also, this interruption is not added to the degrees of multiple subprogram calling.

- Macro type
  The interrupt program is called out as a custom macro. That is, the local variable level changes before and after the interruption. Also, this interruption is not added to the degrees of multiple macro calling.

A subprogram call and a custom macro call executed in the interrupt program are added to their respective degrees of multiple calling.

The execution program cannot pass arguments in the custom macro interruption, even if it is macro type.

3.3 Custom Macro Interruption Control M Codes

The custom macro interruption is, in principle, controlled by M96 and M97. However, they can be used for other uses (M functions, macro call-out, etc.), depending on the machine manufacturer. Therefore, the parameter which can set these M codes is available (025-MPRM).

When this parameter is designed to set the M codes, set them in:

parameter #053 for custom macro interrupt on;
parameter #054 for custom macro interrupt off.

When parameter MPRM is designed for not setting of M codes, M96 and M97 become the custom macro control M codes, regardless of the contents of #053 and #054.

In either case, the custom macro interruption control M codes are processed internally and are not output externally.

It is not preferable, from the viewpoint of program compatibility, that M codes other than M96 and M97 control custom macro interruption.

3.4 Custom Macro Interruption and NC Command

Two types of custom macro interruption are considered: interruption performed by stopping an NC command being executed prematurely; and interruption performed after completion of that block. Therefore, the parameter which selects between the interrupts during, and after the completion of, the block is available (314-MINT).

If this parameter selects an interrupt during execution of a block (type I):

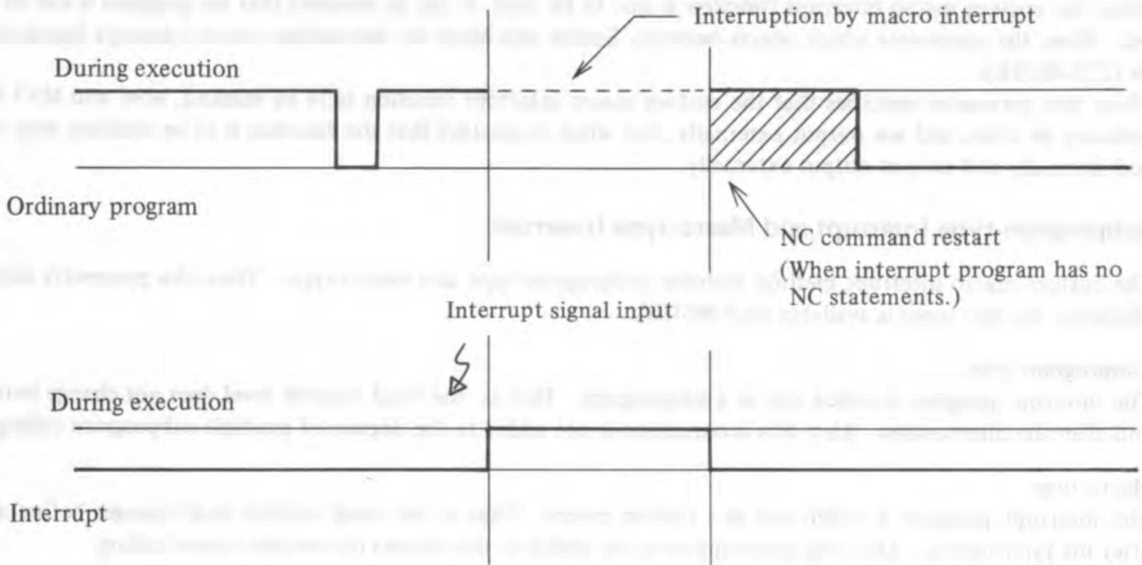
1. Entering the interrupt signal interrupts the movement or dwell being executed and executes the interrupt program;
2. If <u>NC statements exist</u> in the interrupt program, the interrupted block of command disappears and interrupt programs are executed. When having returned to the original program, the execution continues from the next block on;
3. If <u>NC statements do not exist</u> in the interrupt program, when having returned to the original program by M99, the execution continues from the interrupted command on.

If this parameter selects an interrupt after completion of a block (type II):

1. Entering the interrupt signal executes the interrupt program without interrupting the command being executed;

2. If NC statements exist in the interrupt program, they are executed after completion of the command being executed.

In either case, note that control passes to the interrupt program as soon as the interrupt signal is input.



(Figure 3.4-a)

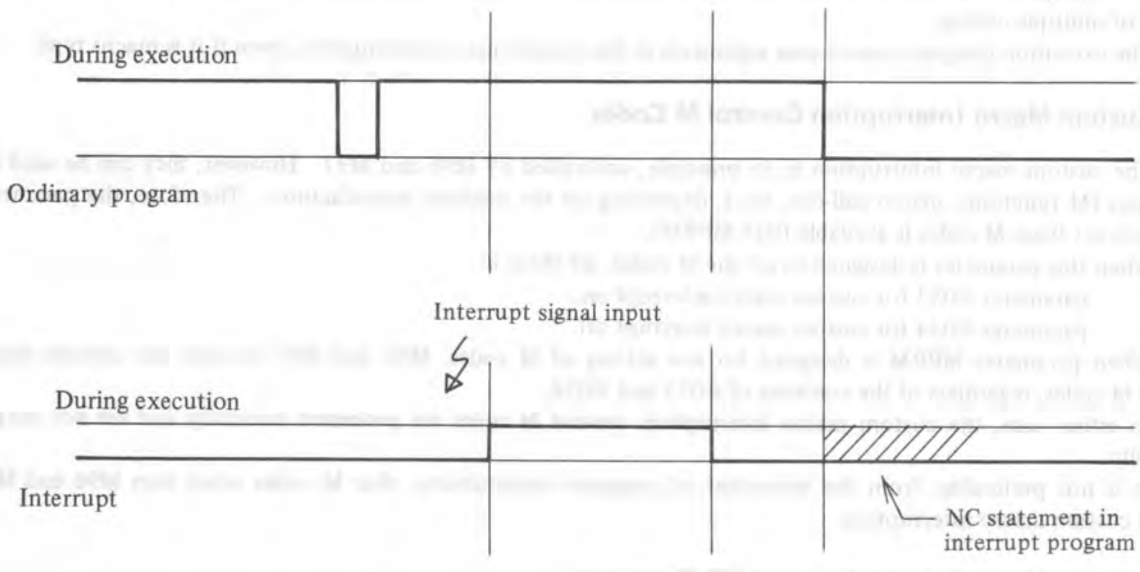


Figure 4 Custom macro interruption and NC command (Figure 3.4-b)

The figure shows an interrupt during execution of a block (3.4-a) and an interrupt after completion of the block (3.4-b). In either case, the interrupt is executed when the interrupt signal has been input.

3.5 Acceptance of Custom Macro Interrupt Signal (UNIT)

There are two methods of accepting the custom macro interrupt signal: status trigger and edge trigger. The status trigger makes the signal effective when it is on; the edge trigger makes it effective when it turns from off to on. The parameter selects between the two types (025-TSE).

When this parameter specifies the status trigger, if the interrupt signal is on (1) when it has become effective, the custom macro interrupt occurs. Therefore, while the signal is on, the interrupt program can be executed repeatedly.

When this parameter specifies the edge trigger, the interrupt signal becomes effective only when it turns on, and the interrupt program is terminated in a moment (programs of macro statements alone, etc.). Therefore, when the status trigger is not suitable, or when the custom macro interrupt is to be performed only once through the whole program (the interrupt signal stays on), this type may be used.

Either type will provide the same effect in practice, except for special uses, such as noted above. (There are no differences, e.g., one type takes longer than the other from the input of the signal to the execution of the custom macro interrupt.)

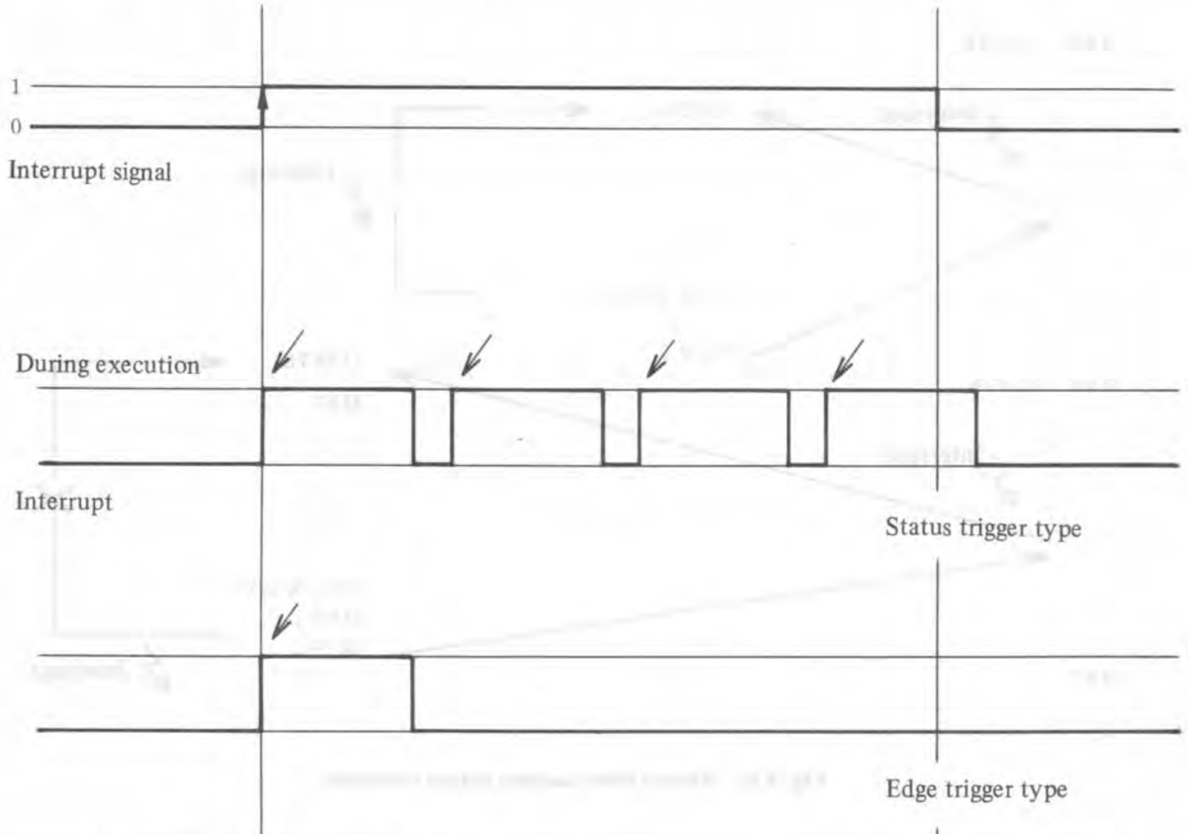


Figure 5 Custom macro interrupt signal

The status trigger type executes the custom macro interrupt when the signal is on; the edge trigger type executes it when the signal turns on. Therefore, the example above shows that four interrupts are executed in the former type; only one in the latter type.

3.6 Return from Custom Macro Interrupt

M99 commands return from the custom macro interrupt to the original program. Also, address P specifies a sequence number in the program to be returned to. In this case, the corresponding program is searched from its beginning and the sequence number appearing first is returned to (same as M98).

Another interrupt does not occur during execution of a custom macro interrupt program, but M99 clears this situation. When M99 is specified independently*, it is executed before completion of the preceding command. Therefore, the custom macro interrupt also becomes effective for the last command of the interrupt program. If this is inconvenient, control the custom macro interrupt by M96 and M97 in the program.

* : block with M99 alone

The M99 block, which consists of addresses O, N, P, L, or/and M alone, is assumed to be the same block as the preceding block in programming. Therefore, the single block stop is not performed. In programming,

Gxx Xxxxx ;
M99 ; and
Gxx Xxxxx M99 ;
provide the same effect.
(They differ in whether Gxx is executed or not before M99 is known.)

419

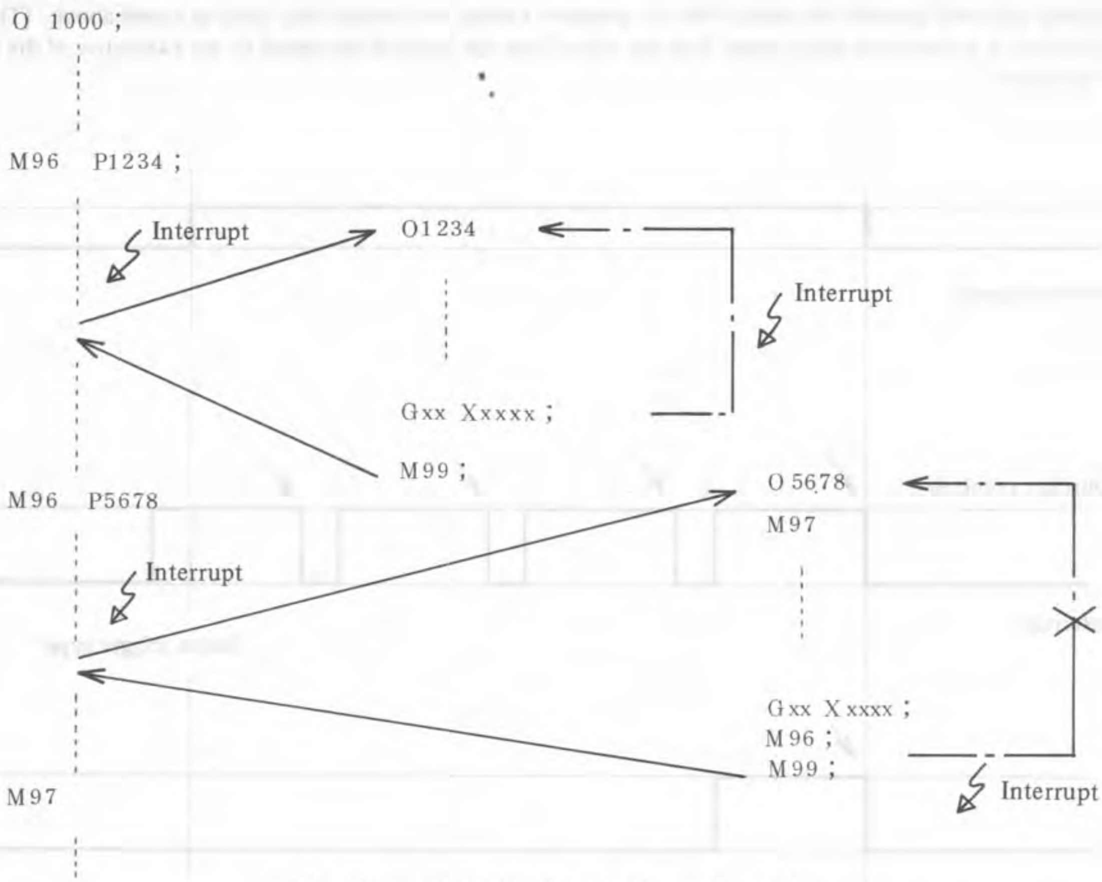


Fig. 3.6 Return from custom macro interrupt

A custom macro interrupt does not overlap with another which is being executed. That is, if an interrupt occurs, another is prohibited automatically; if M99 is executed, the custom macro interrupt becomes effective again. Since the M99 block is executed independently before completion of the preceding block, it can also interrupt the Gxx block of O1234, and if the signal is input, the O1234 program is executed again. On the other hand, the O5678 program is controlled by M96 and M97 and the interrupt becomes effective after it returns to O1000.

3.7 Custom Macro Interruption and Modal Information

The custom macro interrupt, unlike the ordinary program call, is initiated by the interrupt signal. Therefore, it is not desirable that modal information is modified in the interrupt program and as a result, the original program is affected. Thus, even if modal information is modified in the interrupt program, when the original program is returned to by M99, the modal information is restored to that before interruption.

When the interrupt program returns to the original program by M99 Pxxxx, modal information can be controlled in the program. Therefore, the modal information modified in the interrupt program is taken over. (Conversely, when modal information in the original program is to be taken over, movement after return may vary depending on the modal information at the time of interruption.) Therefore, in this case:

1) Command modal information in the interrupt program

or

2) Command necessary modal information at the returned point.

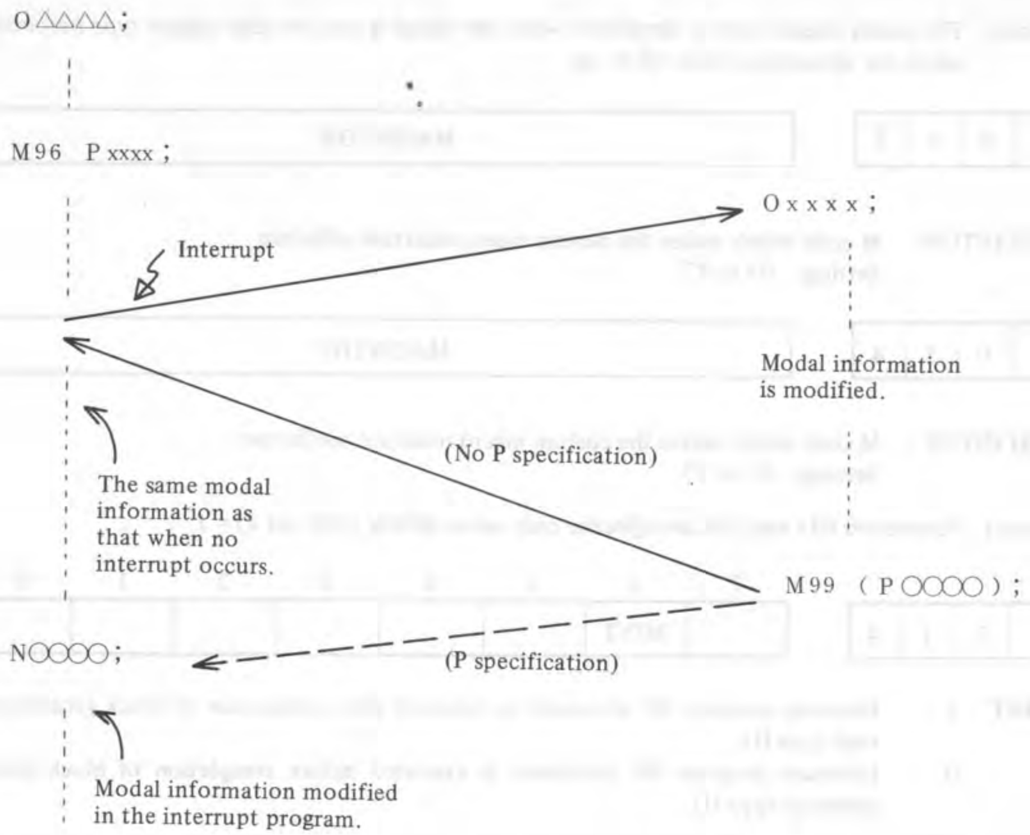


Fig. 3.7 Custom macro interruption and modal information

When modal information is modified in the interrupt program:

1) Return by M99

The modal information before interruption remains valid, and the modal information modified in the interrupt program is neglected;

2) Return by M99 Pcooo

The modal information modified in the interrupt program also becomes valid after return (same as M98, etc.).

Parameters

					7	6	5	4	3	2	1	0
	0	2	5		MUSR		MSUB	MPRM				TSE

MUSR 1 : Custom macro interrupt function is used.
0 : Custom macro interrupt function is not used.
MSUB 1 : Subprogram-type custom macro interrupt
0 : Macro-type custom macro interrupt
MPRM 1 : Custom macro interrupt control M code is parameter-set.
0 : Custom macro interrupt is controlled by M96 and M97.

(Note) The custom macro interrupt is a part of the Custom Macro B function: without the Custom Macro B option, the custom macro interrupt cannot be utilized.
Parameters 053 and 054 can be set only when MPRM = 1. (Of course, the Custom Macro B option is required.)

TSE 1 : Status trigger-type custom macro interrupt
0 : Edge trigger-type custom macro interrupt

(Note) *The status trigger type is significant when the signal is on; the edge trigger type becomes significant when the signal turns from off to on.*

	0	5	3	MACINTON

MACINTON : M code which makes the custom macro interrupt effective
Settings: 03 to 97

	0	5	4	MACINTOF

MACINTOF : M code which makes the custom macro interrupt ineffective
Settings: 03 to 97

(Note) *Parameters 053 and 054 are effective only when MPRM (309, bit 4) = 1.*

				7	6	5	4	3	2	1	0
	3	1	4		MINT						

MINT 1 : Interrupt program NC statement is executed after completion of block (custom macro interrupt type II).

0 : Interrupt program NC statement is executed before completion of block (custom macro interrupt type II).

(Note) *With the type I, when the interrupt signal is input, the block being executed is interrupted, and the interrupt program is executed. Movement after returning to the original program varies, depending on whether or not the interrupt program includes an NC statement.*

i) When an NC statement is included,
the remaining portion (movement and dwell time) of the interrupted block disappears.

ii) When an NC statement is not included,
the interrupted block is executed continuously.

In either case, the miscellaneous function being issued is output correctly.

With tye type II, even when the interrupt signal is input, the block being executed is not interrupted but the interrupt program is executed. If an NC statement is included in the interrupt program it is executed after completion of interrupted block.

5. DIAGNOSE

				7	6	5	4	3	2	1	0
	1	2	0					UINT			

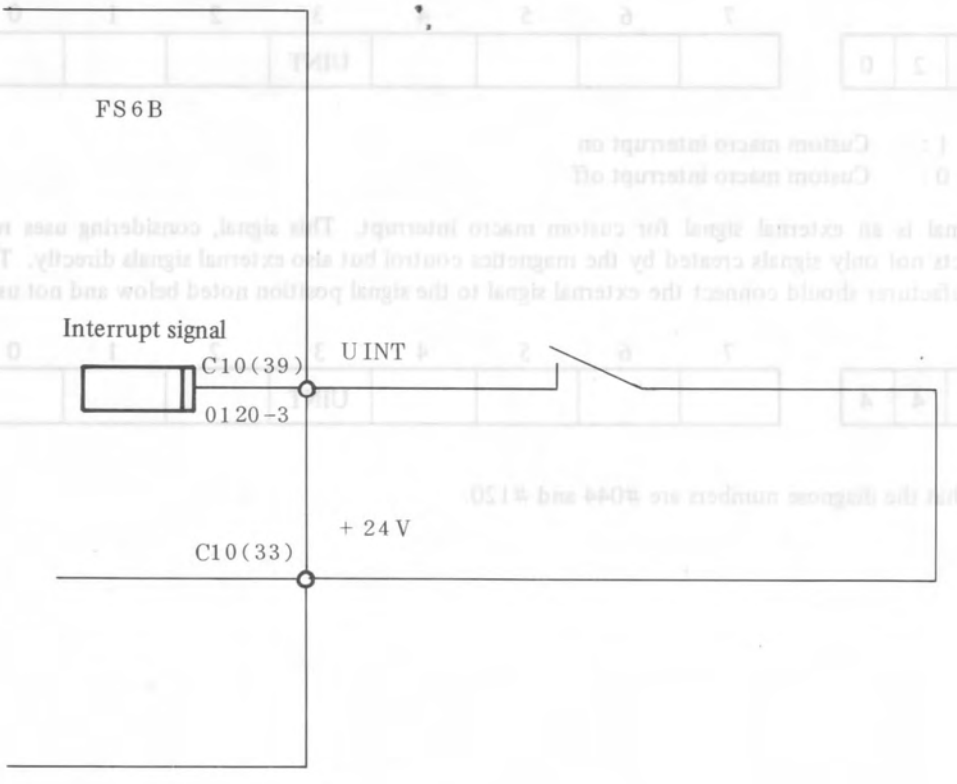
UNIT 1 : Custom macro interrupt on
0 : Custom macro interrupt off

This signal is an external signal for custom macro interrupt. This signal, considering uses requiring high velocity, detects not only signals created by the magnetics control but also external signals directly. Therefore, the machine manufacturer should connect the external signal to the signal position noted below and not use it for other uses.

				7	6	5	4	3	2	1	0
	0	4	4					UINT			

Verify that the diagnose numbers are #044 and #120.

6. INTERNAL CONNECTION DIAGRAM



To connect this signal, connection unit 2 is required.

7. APPLICATION EXAMPLES

7.1 The External Signal Initiates Processing of Detected Tool Failure

[Specifications]
Failure recovery is carried out promptly, even during a cycle motion.

[Parameter setting]

TSE = 0	:	Edge trigger type
MUSR = 1	:	Custom macro interrupt function is used.
MSUB = *	:	Subprogram-/macro-type custom macro interrupt
MPRM = *	:	Selection between the interrupt control M codes
MINT = 0	:	Interrupt program is executed before completion of block.

[Explanation]
If the interrupt signal is to turn on when a failure is detected, the machine holds the signal until a special operation is conducted. If the edge trigger type is selected, the interrupt is executed only once, and diagnose 120 detects whether or not it has been executed.

Subprogram type/macro type and selection of control M codes, whichever are each more convenient in programming, are set by parameter.

7.2 Another Machining Sequence is Inserted in a Series of Machining Operations, without Stopping the Machining Sequence Being Executed.

[Specification]
A short machining sequence is to be inserted in a long machining program being executed. A conventional single block stop, MDI intervention, etc. are awkward to execute, when resuming the original program.

[Parameter setting]

TSE = *	:	Selection between the trigger types
MUSR = 1	:	Interrupt function is used.
MSUB = 0	:	Macro-type interrupt
MPRM = *	:	Selection between the interrupt control M codes
MINT = 1	:	Interrupt program is executed after completion of block.

[Explanation]
Selection between the trigger types and between the interrupt control M codes, whichever are each more convenient, is set by parameter. In order for the custom macro interrupt not to affect the block being executed, the interrupt is prohibited during execution of the block, and the macro-type interrupt is taken. The interrupt program restores modal information, machine position, etc. at the time of interruption, when returning to the original program, whereby any program can be performed. If fixed, the program can be specified directly by M96 Pxxxx; if not, it can be called out by M98 P#100, etc.

7.3 Machining Information is Read at Fixed Time Intervals.

[Specifications]
For machining conditions management, machining information is transmitted externally at fixed time intervals. The machining sequence is not to be affected.

[Parameter setting]

TSE = 0	:	Edge trigger type
MUSR = 1	:	Interrupt function is used.
MSUB = 0	:	Macro-type interrupt
MPRM = *	:	Selection between the interrupt control M codes
MINT = 0	:	Interrupt program is executed before completion of block.

[Explanation]
Suppose that the interrupt program includes no NC statements. The edge trigger type is taken (because the status trigger type repeats interrupts continuously while the signal is on), and turning on and off the signal at each time interval initiates the interrupt program once. Since the interrupt is possible during execution of blocks, as specified above, interrupts are carried out each time the signal turns from off to on. Machining information, such as modal information and position information, is output externally by the use of Custom Macro DO.

The interrupt program is carried out concurrently with the original program block; the machining operation stops for a moment, when the interrupt program is not completed, even if the original program block is completed.

7.4 The Same Program Is Selectively Used for Ordinary Cutting and for Special Cutting.

[Specifications]

A special motion is to be added at each execution of blocks. It is inconvenient to specify such a command in programming.

[Parameter setting]

TSE = 1	:	Status trigger type
MUSR = 1	:	Interrupt function is used.
MSUB = *	:	Selection between the subprogram and macro types
MPRM = *	:	Selection between the interrupt control M codes
MINT = 1	:	Interrupt program is executed after completion of block.

[Explanation]

The interrupt program is commanded as:

```
Oxxxx ;

M97 ;          Prohibits interrupt.
 .
 .
 .
 .
M96 ;          Enables interrupt.
M99 ;
```

The interrupt signal stays on; the status trigger type is taken, whereby the custom macro interrupt is executed each time one block is completed in the original program. The interrupt program specifies in advance a special motion to be executed. M97 makes interrupts invalid for program portions for interrupts not to be executed.

APPENDIX 13. MULTI-HANDLE FUNCTION

1. General

More than one axes can be fed at a time manually if several manual pulse generators (MPG) are connected. (Up to three MPGs can be connected.)

The following specifications are provided based on the number of axes, number of manual pulse generators, and handle axis selection input signals.

(1) Spec A

If the number of axes is identical to the number of manual pulse generators, they correspond as follows:

1st MPG → X axis
2nd MPG → Y axis
3rd MPG → Z axis

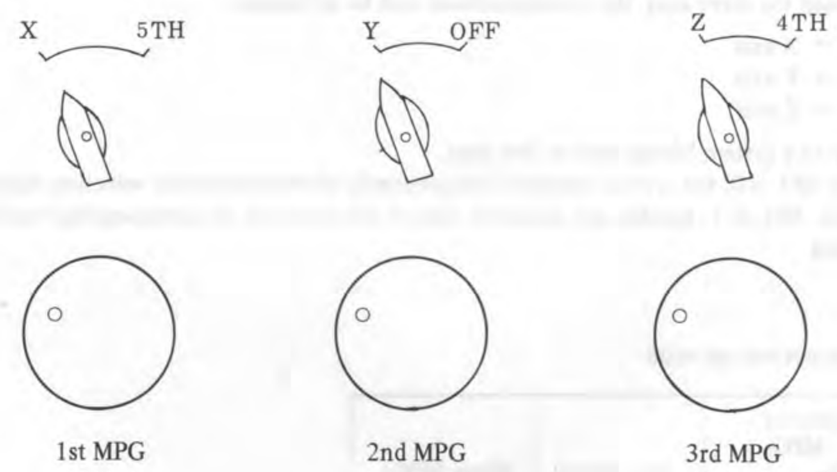
By using a parameter (SEL), these can be made valid only when the handle axis selection signal (HX, HY, or HZ) is closed.

(2) Spec B

Spec B is used when the number of MPGs is smaller than the number of axes and the correspondences between the axes and the MPGs are fixed. Correspondence between the respective axes and MPGs must be determined beforehand by the parameters.

(Example) Three MPGs for five axes

Axis	MPG
X	1st MPG
Y	2nd MPG
Z	3rd MPG
4th axis	3rd MPG
5th axis	1st MPG



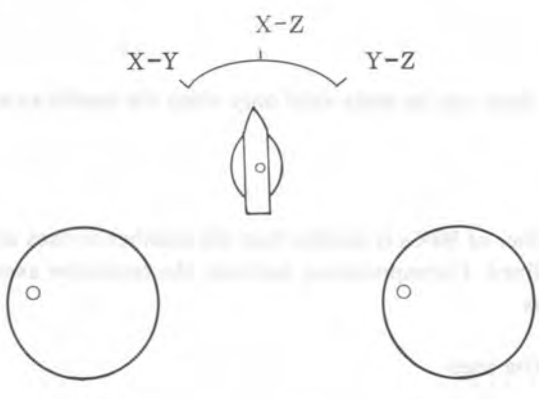
(3) Spec C

Spec C is used when the number of MPGs is smaller than the number of axes and correspondences between the axes and the MPGs is not fixed.

(Example) Two MPGs for three axes

Handle axis selection			1st MPG	2nd MPG
HX	HY	HZ		
O	O	X	X axis	Y axis
O	X	O	X axis	Z axis
X	O	O	Y axis	Z axis

O : Contact is closed.
X : Contact is open.



1st MPG 2nd MPG

2. SPECIFICATION DETAILS

All specs become valid when the handle mode is selected as the same as conventional handle function. If the handle interruption function is added the operation conforms to the handle interruption specifications. The movement and magnification per graduation are the same as in case of the conventional handle function.

(1) Spec A

If three MPGs are used for three axes, the correspondence will be as follows:

1st MPG → X axis
2nd MPG → Y axis
3rd MPG → Z axis

This does not apply to a system having four or five axes.
When the parameter SEL is 0, the system operates independently of the handle axis selection signals.
When the parameter SEL is 1, handles are operated only if the contacts of corresponding handle axis selection signals are closed.

(2) Spec B

The following combinations are valid:

Number of MPGs / Number of axes	Two MPGs	Three MPGs
Three axes	O	
Four axes	O	O
Five axes	O	O

O : Valid combinations

Use the parameters HXNO, HYNO, HZNO, H4NO, and H5NO to specify the correspondences between the axes and the MPGs.
The axes move by the MPG handle specified by the parameters when the contacts of corresponding axis selection signals are closed.

(Note) *If one MPG is used for more than one axes, the axes move simultaneously when the contacts of corresponding axis selection signals are closed.*

(3) Spec C

The following combinations are valid:

Number of axes \ Number of MPGs	Two MPGs	Three MPGs
Three axes	O	
Four axes	O	O
Five axes	O	O

O : Valid combinations

This is valid only if the number of closed handle axis selection signals is identical to the number of MPGs. The axes move by the MPGs specified in the sequence of X, Y, Z, 4th, and 5th axes.

(Note) *If the number of closed handle axis selection signals is not equal to the number of handles, none of the axes moves.*

(Example) Three MPGs for four axes

Handle axis selection				1st MPG	2nd MPG	3rd MPG
HX	HY	HZ	H4			
O	O	O	X	X axis	Y axis	Z axis
O	O	X	O	X axis	Y axis	4th axis
O	X	O	O	X axis	Z axis	4th axis
X	O	O	O	Y axis	Z axis	4th axis

O : Contact closed
X : Contact open

3. PARAMETERS

Specify the following parameters according to the specs:

	4	1	3

7	6	5	4	3	2	1	0
						SELB	SEL

SEL 1: Axes can move only if the contacts of handle axis selection signals are closed. Set 1 for specs B and C and also A if the axis selection signals are to be made valid.
0: Handle axis selection signals are invalid.

SELB 1: Spec B
0: Spec A or C

	4	1	8

HXNO + HDCNT

HDCNT: The number of the MPGs to be used. Must be specified for all the specs A, B, and C.
Setting value: 2 or 3

HXNO: When using Spec B, specify the MPG number (n-th) to be used for X-axis.
Setting value: 1, 2 or 3

Specify HXNO for the second order position and HDCNT for the first order position.
(Example) If X-axis is operated by the 1st MPG and the number of MPGs is 3: Specify 13.

	4	1	9

HZNO + HYNO

HYNO: In case of Spec B, specify the MPG number (n-th) for Y-axis.
Setting value: 1, 2 or 3

HZNO: In case of Spec B, specify the MPG number (n-th) for Z-axis.
Setting value: 1, 2 or 3

Specify HZNO for the second order position, and HYNO for the first order position.
(Example) If Y-axis is to be operated by the 2nd MPG and Z-axis is to be operated by the 3rd MPG: Specify the value 32.

	4	2	0

H5NO + H4NO

H4NO: In case of Spec B, specify the MPG number (n-th) for the 4th axis.
Setting value: 1, 2 or 3

H5NO: In case of Spec B, specify the MPG number (n-th) for the 5th axis.
Setting value: 1, 2 or 3

Specify H5NO for second order position, and H4NO for the first order position.
(Example) If the 4th axis is to be operated by the 1st MPG and the 5th axis is to be operated by the 3rd MPG: Specify 31.

APPENDIX 14 INPUT/OUTPUT INTERFACE

The following input/output interfaces are available for FANUC SYSTEM 6M-MODEL B.

This material describes a connection information for connection between NC and I/O device through Input/Output interface.

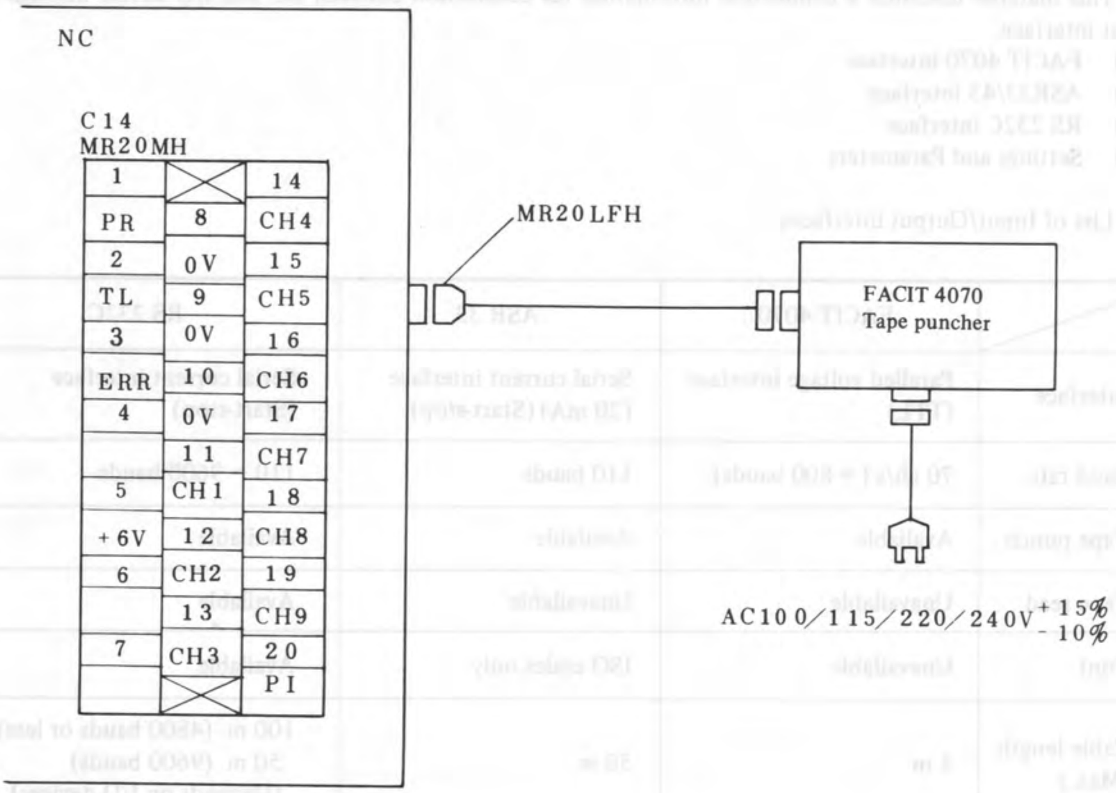
(1) FACIT 4070 interface
(2) ASR33/43 interface
(3) RS 232C interface
(4) Settings and Parameters

List of Input/Output interfaces:

	FACIT 4070	ASR 33	RS 232C
Interface	Paralled voltage interface (TTL)	Serial current interface (20 mA) (Start-stop)	Serial current interface (Start-stop)
Baud rate	70 ch/s (= 800 bauds)	110 bauds	110 ~ 9600 bauds
Tape punch	Avaliable	Available	Available
Tape read	Unavailable	Unavailable	Available
Print	Unavailable	ISO codes only	Available
Cable length (Max.)	5 m	50 m	100 m (4800 bauds or less) 50 m (9600 bauds) (Depends on I/O devices)
Remarks			RS 232C is an EIA standard, not an I/O device name.

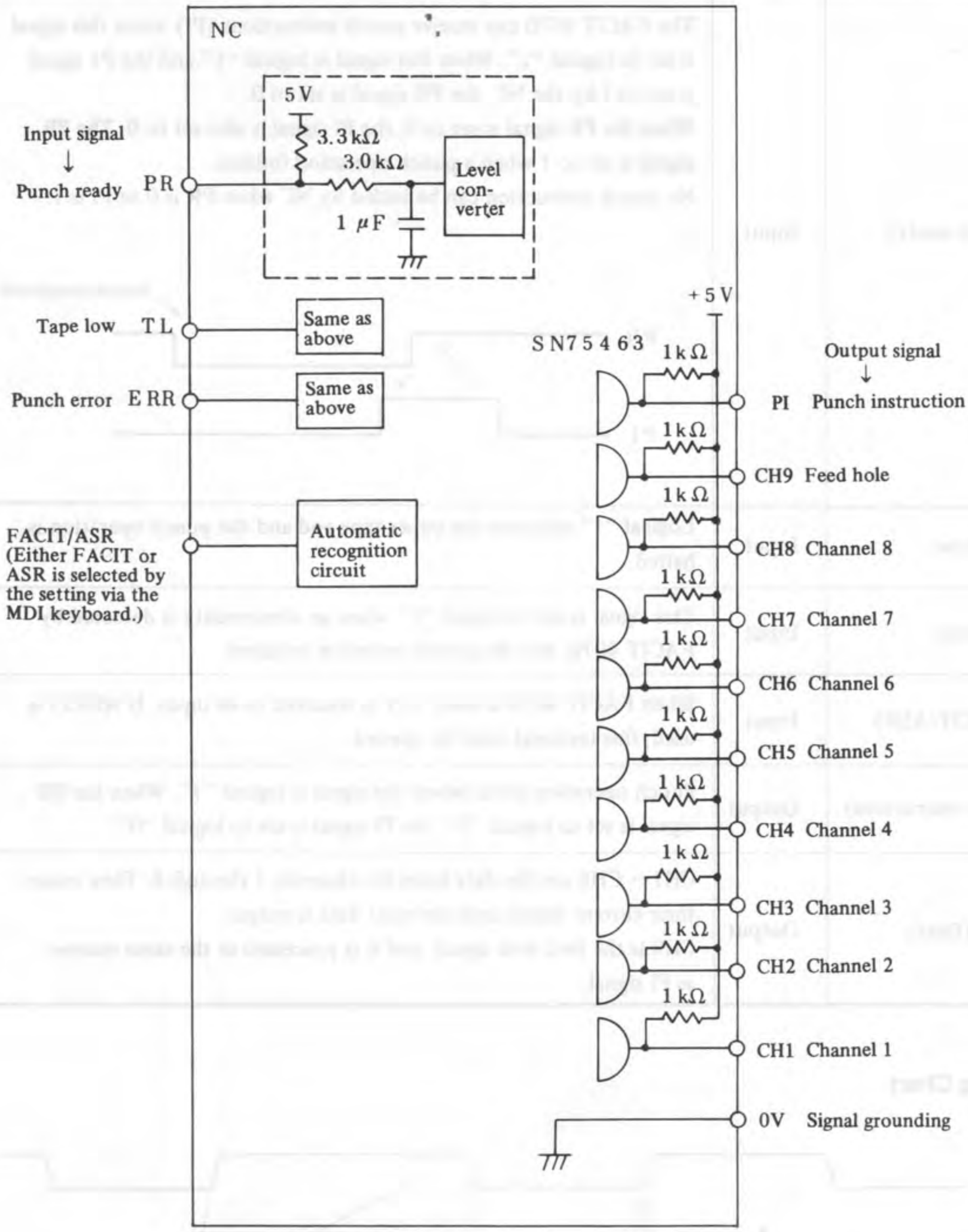
1. FACIT 4070 INTERFACE

1.1 Connection to NC



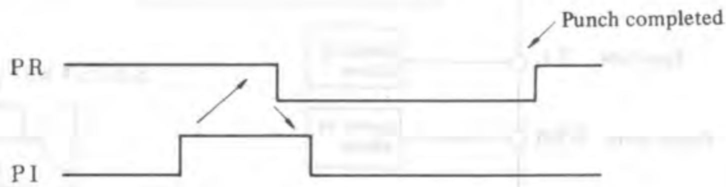
(Note) The input/output interface option does not include the connecting cable running between the FACIT 4070 and NC.

1.2 Signal Description

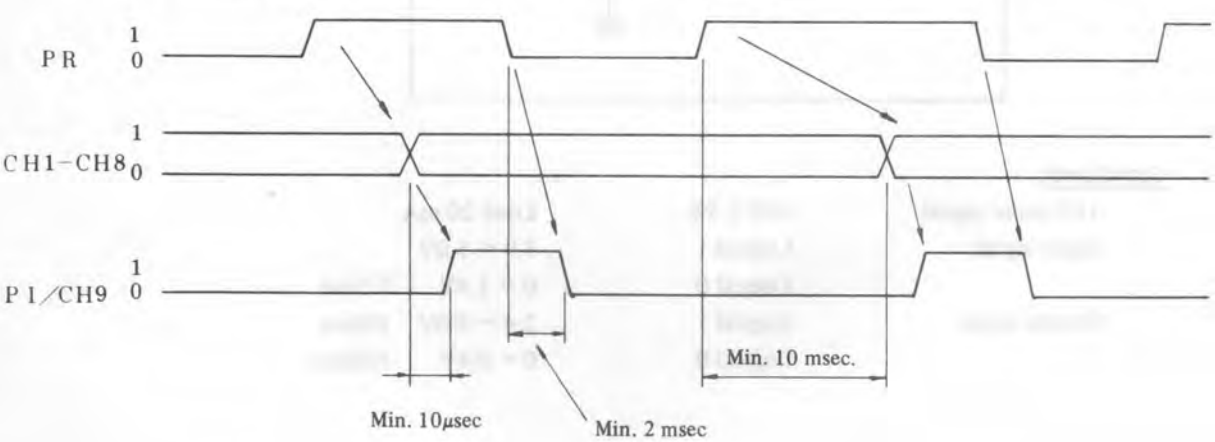


Signal level

+6V input signal	+6V ± 5%	Load 20 mA	
Input signal	Logical 1	3.9 ~ 5.0V	
	Logical 0	0 ~ 2.4V	3.2mA
Output signal	Logical 1	2.4 ~ 5.0V	100μA
	Logical 0	0 ~ 0.4V	100mA

Signal name	I/O	Description
PR (Punch ready)	Input	The FACIT 4070 can receive punch instructions (PI) when this signal is set to logical "1". When this signal is logical "1" and the P1 signal is set to 1 by the NC, the PR signal is set to 0. When the PR signal is set to 0, the PI signal is also set to 0. The PR signal is set to 1 when a punch operation finishes. No punch instruction can be issued by NC when PR is 0 or PI is 1. 
TL (Tap low)	Input	Logical "1" indicates the paper tape end and the punch operation is halted.
ERR (Error)	Input	This signal is set to logical "1" when an abnormality is detected by FACIT 4070, and the punch operation is halted.
+6V (FACIT/ASR)	Input	When FACIT 4070 is used, +6V is required to be input. If ASR33 is used, this terminal must be opened.
PI (Punch instruction)	Output	Punch operation starts when this signal is logical "1". When the PR signal is set to logical "0", the PI signal is set to logical "0".
CH1 ~ 9 (Data)	Output	CH1 ~ CH8 are the data holes for channels 1 through 8. They retain their current status until the next data is output. CH9 is the feed hole signal, and it is processed in the same manner as PI signal.

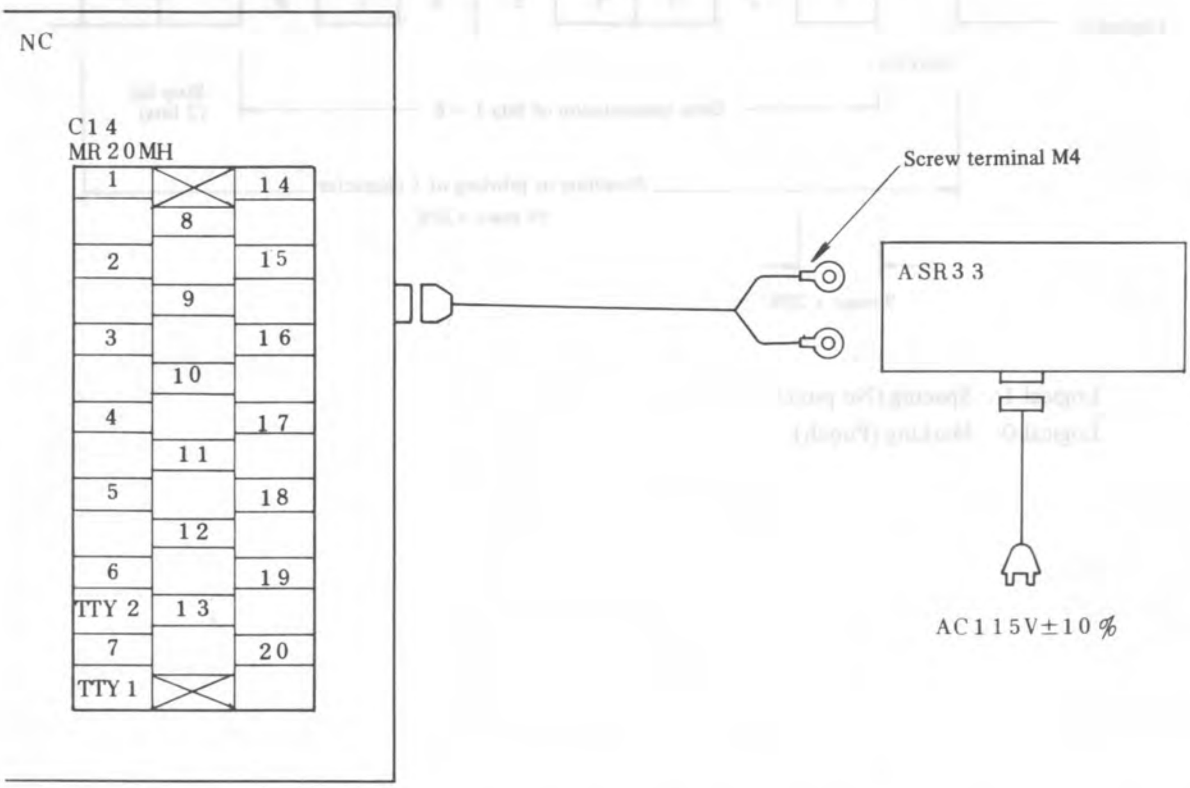
1.3 Timing Chart



434

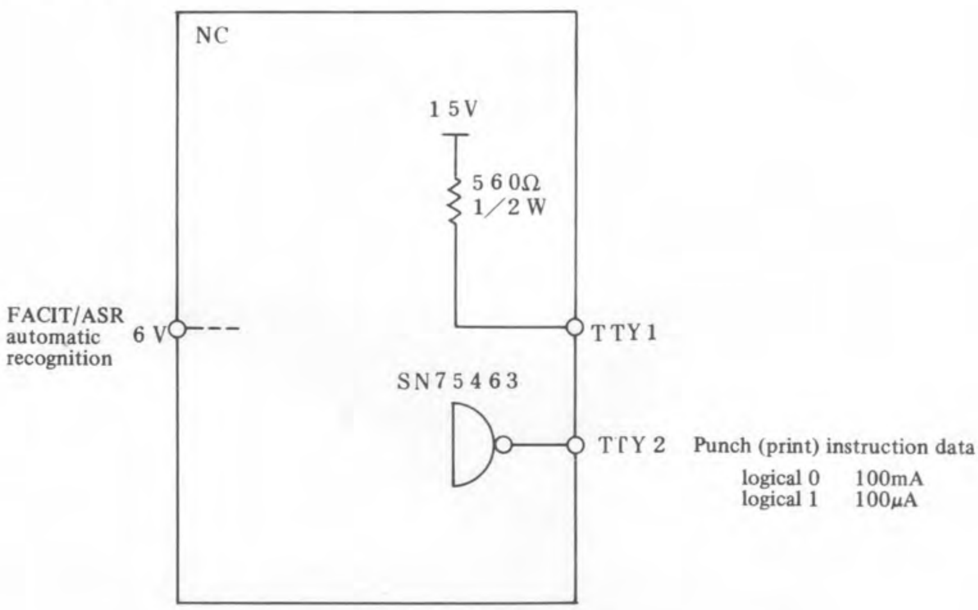
2. ASR 33 INTERFACE

2.1 Connection to NC

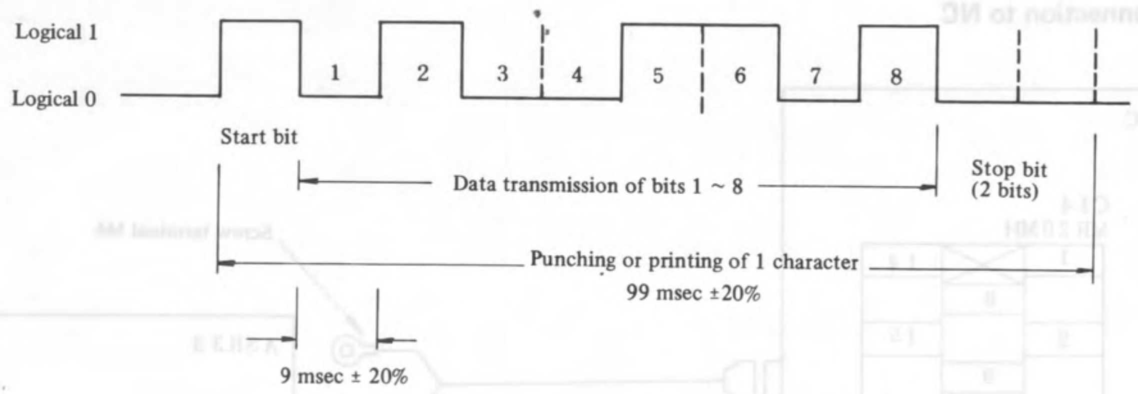


(Note) The input/output interface option does not include the connecting cable running between the ASR 33 and the NC.

2.2 Signal Description



(Note) The +6V for FACIT/ASR automatic recognition must be open when the ASR 33 is used.



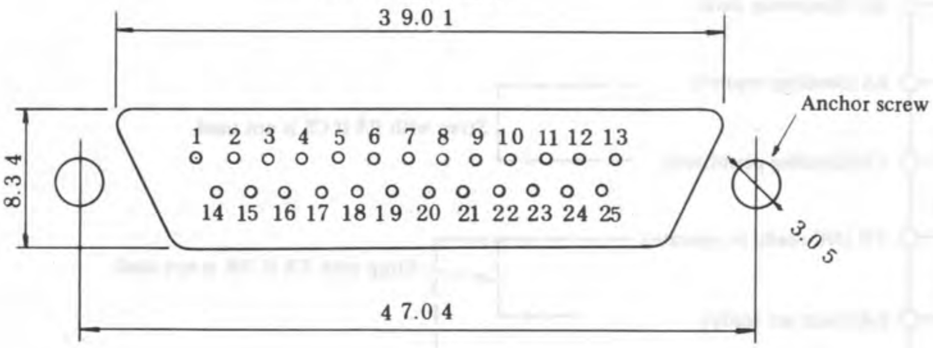
Logical 1: Spacing (No punch)
Logical 0: Marking (Punch)

3. RS 232C INTERFACE

3.1 Connection to NC

NC

Units: mm



Connector made by Japan Aviation Electronic Industry, Ltd. DBM-25S

Lock D20418-J2

DMB−2 5 S

C15 (female)

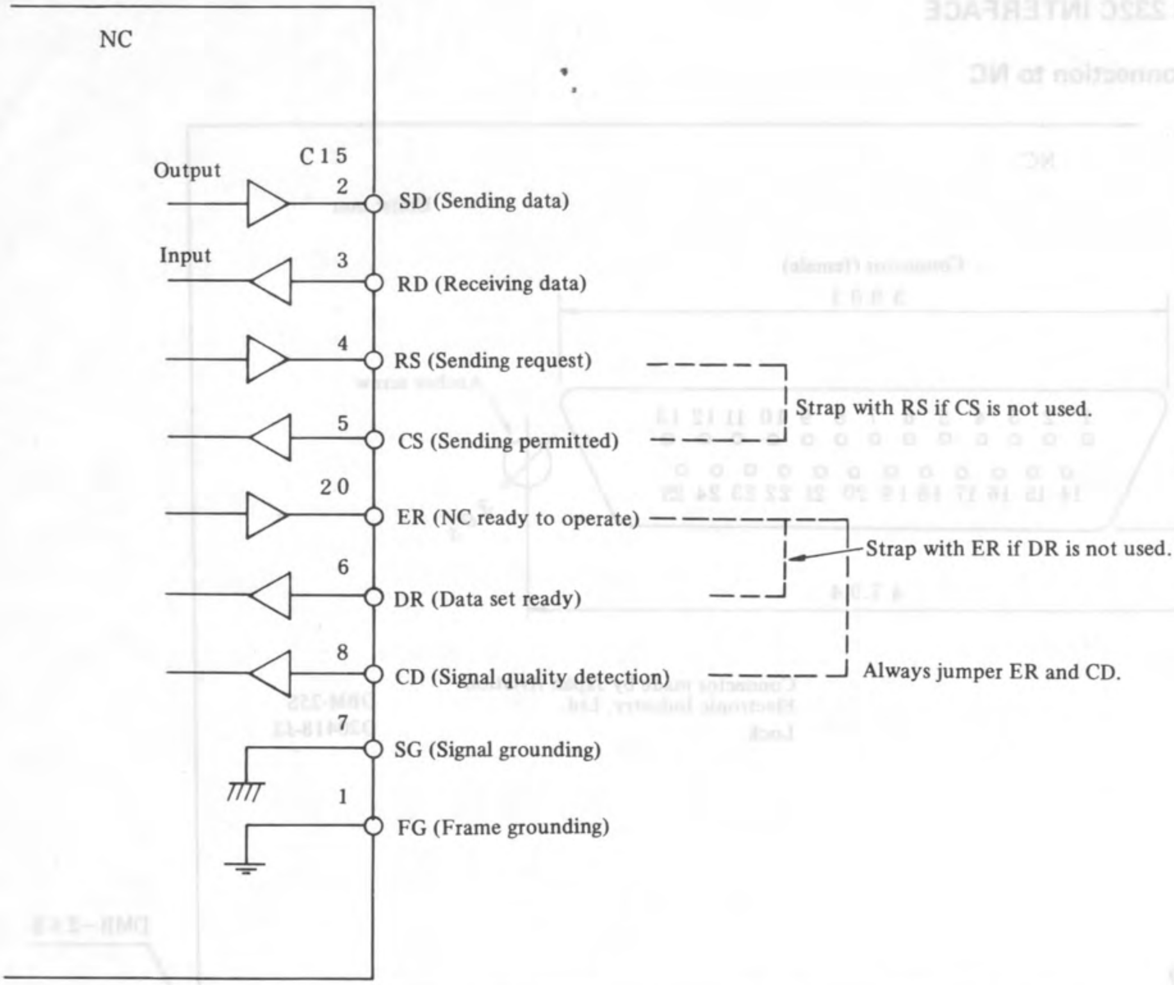
1	2	3	4	5	6	7	8	9	10	11	12	13
FG	SD	RD	RS	CS	DR	SG	CD					

14	15	16	17	18	19	20	21	22	23	24	25
						ER					+24N

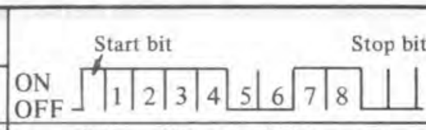
C 1 5 — To I/O device

(Note) A cable for connection to I/O device is not included in this option. It must be prepared by machine tool builder.

437


NC
Output
Input
C15
2
SD (Sending data)
3
RD (Receiving data)
4
RS (Sending request)
5
CS (Sending permitted)
Strap with RS if CS is not used.
20
ER (NC ready to operate)
6
DR (Data set ready)
Strap with ER if DR is not used.
8
CD (Signal quality detection)
Always jumper ER and CD.
7
SG (Signal grounding)
1
FG (Frame grounding)

3.2 Signal Description

Signal name	RS232C circuit number	Pin number	I/O *	Description
SD	103	2	Output	Sending data  (When ISO code "0" is sent)
RD	104	3	Input	Receiving data
RS	105	4	Output	Sending request: This signal is set to on when NC starts sending data and is turned off when transmission ends.
CS	106	5	Input	Sending permitted: When both this signal and the DR signal are set, the NC can send data. If I/O device processing is delayed by a punching operation, etc., NC data sending can be stopped by turning off this signal after sending two characters, including the data being sent currently. If this signal will not be used, make sure to strap this signal circuit to the RS signal circuit.
DR	107	6	Input	Data set ready: When I/O device is ready to operate, this signal is set. This signal should usually be connected to the signal indicating I/O device power supply being on. (ER signal of I/O device). See Note 1. The NC transfers data when this signal is set. If the signal is turned off during data transfer, alarm 086 is issued. If the DR signal will not be used, make sure to strap this signal circuit to the ER signal circuit.
ER	108.2	20	Output	NC ready to operation: This signal is set when the NC is ready to operate. I/O device should regard the SD signal as being significant when the ER signal is set.
CD	109	8	Input	Signal quality signal: Since this signal is not used in connections with I/O device, the signal circuit must be strapped, inside the connecting cable, to the NC ER signal circuit.
SG	102	7		Signal grounding
FG	101	1		Frame grounding

(Note 1) *When an alarm is issued during data transfer or an I/O device abnormality is detected, the I/O device's ER signal is usually turned off. So by connecting this signal circuit to the NC DR signal circuit, the data transfer alarm can be displayed.*

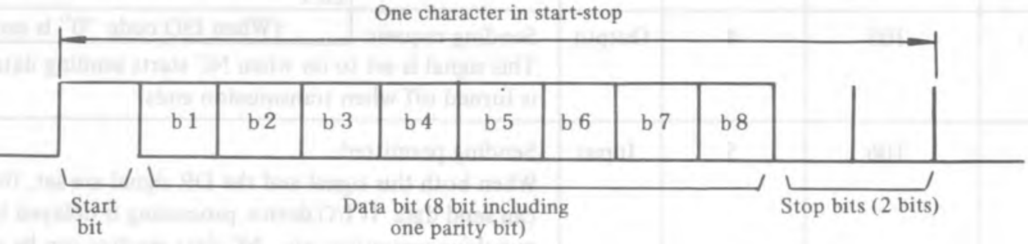
(Note 2) *Signals are set and cleared based on the following:*

	–3V or lower	+3V or higher
Function	OFF	ON
Signal condition	Marking	Spacing

3.3 Transmission Method

(1) Start-stop

Generally, two transmission methods are available at the RS-232C interface. The System 6 series uses the start-stop method. With this method, start and stop signals are output before and after each data bit.



(2) Codes

The RS 232C interface transmission codes are as follows:

① EIA code and Control codes DC1 ~ 4

or ② ISO code and Control codes DC 1 ~ 4
(ISO code is optional.)

The connected I/O device must be able to recognize the following control codes.

	Character	8	7	6	5	4		3	2	1
DC 1	Tape reader start				o		o			o
DC 2	Tape punch designation				o		o		o	
DC 3	Tape reader stop	o			o		o		o	o
DC 4	Tape punch release				o		o	o		

(Note) The listed control codes are used for both EIA and ISO.

When the I/O device is equipped with an ISO/EIA converter, the I/O device must satisfy the specification shown in Table 3.3.

Table 3.3

ISO code										EIA code											Meaning
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		
0			O	O		o				0			O			o					Numeral 0
1	O		O	O		o			O	1						o			O		″ 1
2	O		O	O		o		O		2						o		O			″ 2
3			O	O		o		O	O	3				O		o		O	O		″ 3
4	O		O	O		o	O			4						o	O				″ 4
5			O	O		o	O		O	5				O		o	O		O		″ 5
6			O	O		o	O	O		6				O		o	O	O			″ 6
7	O		O	O		o	O	O	O	7						o	O	O	O		″ 7
8	O		O	O	O	o				8					O	o					″ 8
9			O	O	O	o			O	9				O	O	o			O		″ 9
A		O				o			O	a		O	O			o			O		Address A
B		O				o		O		b		O	O			o		O			″ B
C	O	O				o		O	O	c		O	O	O		o		O	O		″ C
D		O				o	O			d		O	O			o	O				″ D
E	O	O				o	O		O	e		O	O	O		o	O		O		″ E
F	O	O				o	O	O		f		O	O	O		o	O	O			″ F
G		O				o	O	O	O	g		O	O			o	O	O	O		″ G
H		O			O	o				h		O	O		O	o					″ H
I	O	O			O	o			O	i		O	O	O	O	o			O		″ I
J	O	O			O	o		O		j		O		O		o			O		″ J
K		O			O	o		O	O	k		O		O		o		O			″ K
L	O	O			O	o	O			l		O				o		O	O		″ L
M		O			O	o	O		O	m		O		O		o	O				″ M
N		O			O	o	O	O		n		O				o	O		O		″ N
O	O	O			O	o	O	O	O	o		O				o	O	O			Not used at significant data zone in ISO code. Assumed as program number in EIA code.
P		O		O		o				p		O		O		o	O	O	O		Address P
Q	O	O		O		o			O	q		O		O	O	o					″ Q
R	O	O		O		o		O		r		O			O	o			O		″ R
S		O		O		o		O	O	s			O	O		o		O			″ S
T	O	O		O		o	O			t			O			o		O	O		″ T
U		O		O		o	O		O	u			O	O		o	O				″ U
V		O		O		o	O	O		v			O			o	O		O		″ V
W	O	O		O		o	O	O	O	w			O			o	O	O			″ W
X	O	O		O	O	o				x			O	O		o	O	O	O		″ X
Y		O		O	O	o			O	y			O	O	O	o					″ Y
Z		O		O	O	o		O		z			O		O	o			O		″ Z
DEL	O	O	O	O	O	o	O	O	O	Del		O	O	O	O	o	O	O	O	*	Delete (cancel erroneous hole)
NUL						o				Blank						o				*	No holes. Not used in significant data zone in EIA code.
BS	O				O	o				BS			O		O	o		O		*	Back space
HT					O	o			O	Tab			O	O	O	o	O	O		*	Tabulator
LF or NL					O	o		O		CR or EOB	O					o					End of block
CR	O				O	o	O		O											*	Carriage return
SP	O		O			o				SP				O		o				*	Space
%	O		O			o	O		O	ER					O	o		O	O		Absolute rewind stop
(			O		O	o				(2-4-5)				O	O	o		O			Control out (start of comment)
)	O		O		O	o			O	(2-4-7)		O			O	o		O			Control in (end of comment)
+			O		O	o		O	O	+		O	O	O		o				*	Plus sign
−			O		O	o	O		O	−		O				o					Minus sign
:			O	O	O	o		O													Assumed as program numbe. in ISO code.
/	O		O		O	o	O	O	O	/			O	O		o			O		Optional block skip
.			O		O	o	O	O		.		O	O		O	o		O	O		Decimal point
#	O		O			o		O	O											*	Sharp
$			O			o	O													*	Dollar symbol
&	O		O			o	O	O		&					O	o	O	O		*	Ampersand
'			O			o	O	O	O											*	Apostrophe
*	O		O		O	o		O												*	Asterisk
,	O		O		O	o	O			,			O	O	O	o		O	O	*	Comma
;	O		O	O	O	o		O	O											*	Semicolon
<			O	O	O	o	O													*	Left angle bracket
=	O		O	O	O	o	O		O											*	Equal mark
>	O		O	O	O	o	O	O												*	Right angle bracket
?			O	O	O	o	O	O	O											*	Question mark
@	O	O				o														*	Commercial at mark
″			O			o		O												*	Quotation mark
DC1				O		o			O											*	Tape reader start
DC2				O		o		O												*	Tape punch designation
DC3	O			O		o		O	O											*	Tape reader stop
DC4				O		o	O													*	Tape punch release

The codes marked with * are ignored when they are read in the NC.

441

(Note 1) *When the I/O device is equipped with an ISO/EIA converter, the following items must be noted in the Table 3.3.*

Control out (Comment field start)

Control in (Comment field end)

EIA code	(- - - - - - - - -	)	CR	O - - - - - -
	↑	↑	↑	↑
	*1	*1	*2	*3
	↓	↓	↓	↓
ISO code	(- - - - - - - - -	)	LF	:

*1 *Left parenthesis (of the ISO code punches holes at bits 2, 4 and 5 when used in the EIA code. Right parenthesis) of the ISO code punches holes at bits 2, 4 and 7 when used in the EIA code.*

*2 *EIA code* CR is LF in ISO code.

*3 *EIA code* O *is : in ISO code.*

(Note 2) *Control codes DC1 ~ 4 are transmission codes output from the NC. So they need not to be punched on the NC tape.*

(3) Transmission rate (Baud rate)

The transmission rate (Baud rate) is the number of bits transferred per second.

In the system 6 series, the following baud rates are available depending on the system parameter.

50, 100, 110, 150, 200, 300, 600, 1200, 2400, 4800, 9600.

Example: Baud rate: 110

When using one start bit and two stop bits (totalling 11 bits per character):

Transmission characters/second $= \frac{110}{11} =$ 10 characters/second (Maximum)

(4) Cable length

The cable length required when using the RS-232C interface depends on the I/O device type. Consult with the device manufacturers for actual connecting cable lengths.

When cable A (A66L-0001-0041) (See Connecting Manual) is used, cable length is as follows by the specification of NC

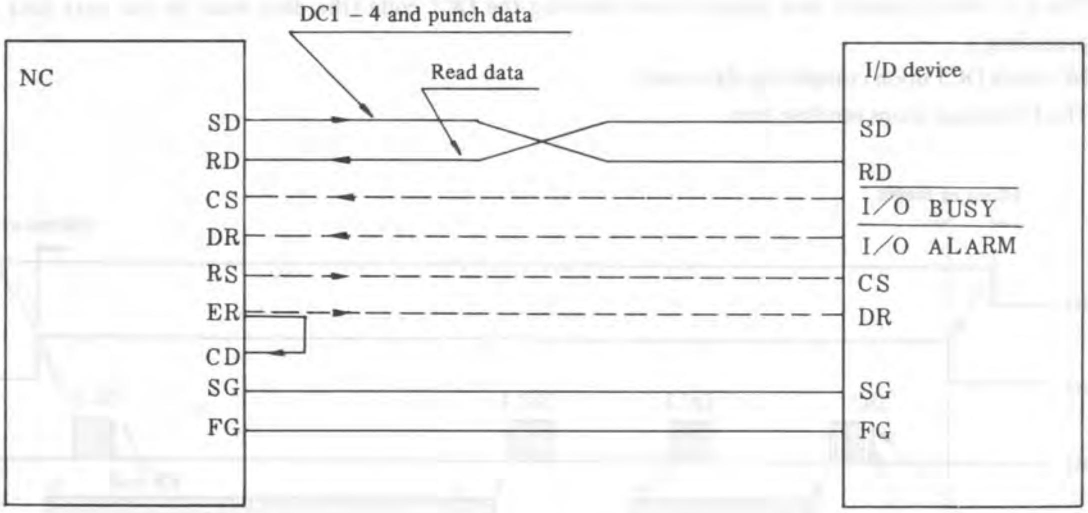
100 m or less . . . . 4800 bauds or less
50 m or less . . . . 9600 bauds

(5) Connection to I/O device

NC and I/O device are generally connected as shown in Fig. 3.3 (a). In this interface, control codes DC 1-4 are used.

1) NC can control I/O device by issuing codes DC1-4.

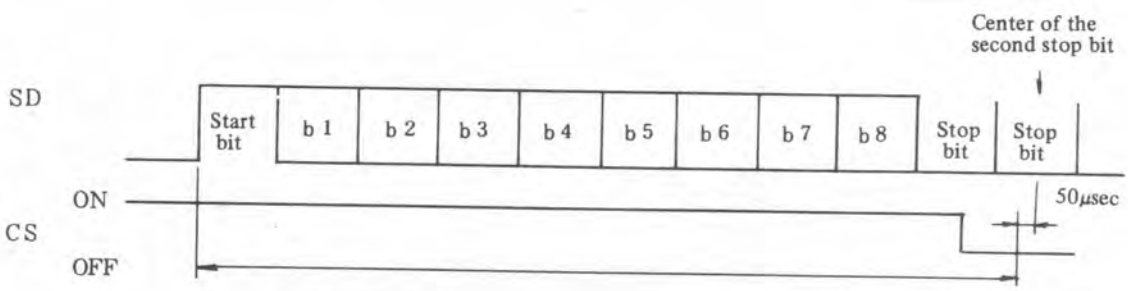
2) When I/O processing falls behind the pace of the NC signals (When NC issues data)
   (i) I/O device can temporarily stop NC data output by using the NC's CS signal. In this case follow the requirment shown in Fig. 3.3 (b).
   Data output stops within two characters including a currently transmitting character when CS signal is input to NC.
   When CS signal is turned on again, data transmission starts.
   (ii) If control code DC3 is input to NC, NC stops data output within ten characters. When control code DC1 is input to NC, NC starts sending data again.



DC1 : Tape reader start
DC2 : Tape punch designation
DC3 : Tape reader stop
DC4 : Tape punch release

Note: Signals indicated by dashed lines are connected to the I/O device only as needed. If they are not used, these signals must be strapped within the connecting cable as explained in the section on signal descriptions.

Fig. 3.3 (a)

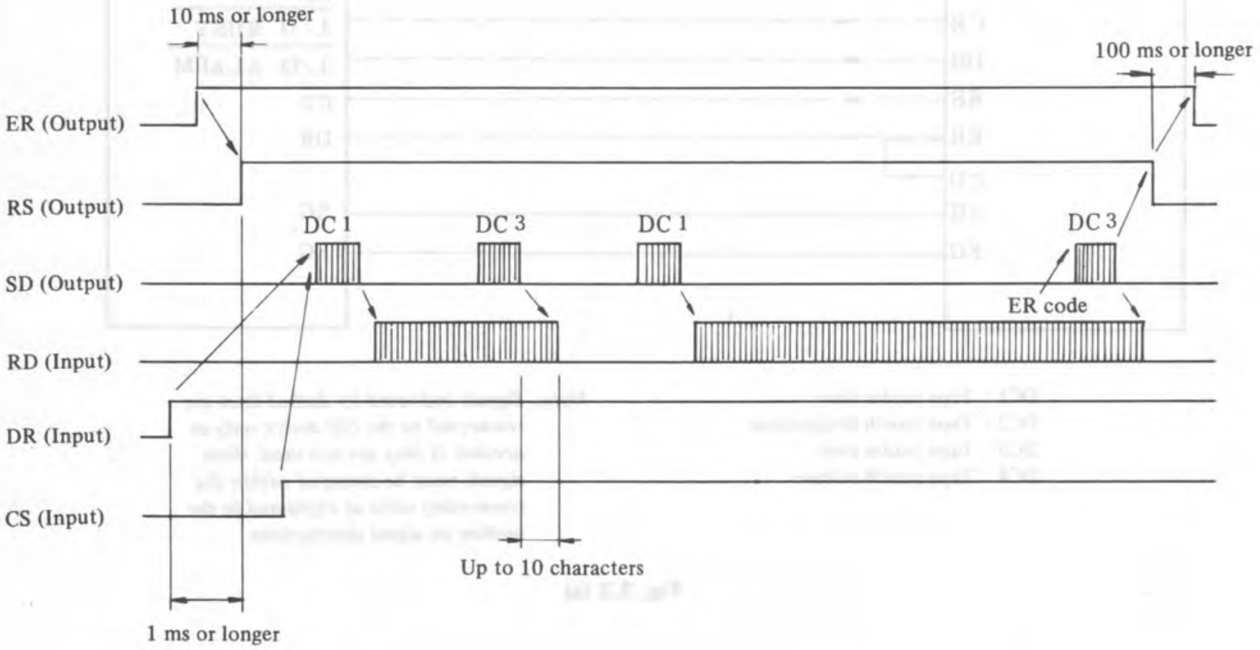


(There is no prescription on timing when switching from OFF to ON)

Fig. 3.3 (b)

Time chart when the NC receives data (Read into memory)

(1) NC outputs DC1.
(2) The I/O device starts sending data upon receiving DC1.
(3) NC sends DC3 when NC processing is delayed.
(4) The I/O device stops sending data to NC after receiving DC3.
The device may send up to 10 characters after receiving DC3. If it sends more than 10 characters, alarm 087 will occur.
(5) NC reissues DC1 upon completing delayed processing.
(6) The I/O device restarts data output upon receiving the DC1 code (the data must be the next data to the preceding.)
(7) NC sends DC3 upon completing data read.
(8) The I/O device stops sending data.



444

Time chart when the NC sends data (Punch out)

(1) NC outputs DC2.
(2) NC outputs punch data in succession.
(3) When data processing is delayed at the I/O device, including the character currently being sent if the CS signal is turned off.
  (i) Data output stops within two characters including a currently transmitting character when CS signal is input to NC.
  When CS signal is turned on again, data transmission starts. (See Fig. A)
  (ii) If control code DC3 is input to NC, NC stops data output within ten characters. When control code DC1 is input to NC, NC starts sending data again. (See Fig. B)
(4) The NC starts sending the next data if the CS signal is turned on after the I/O device completes data processing.
(5) The NC issues DC4 upon completing data output.

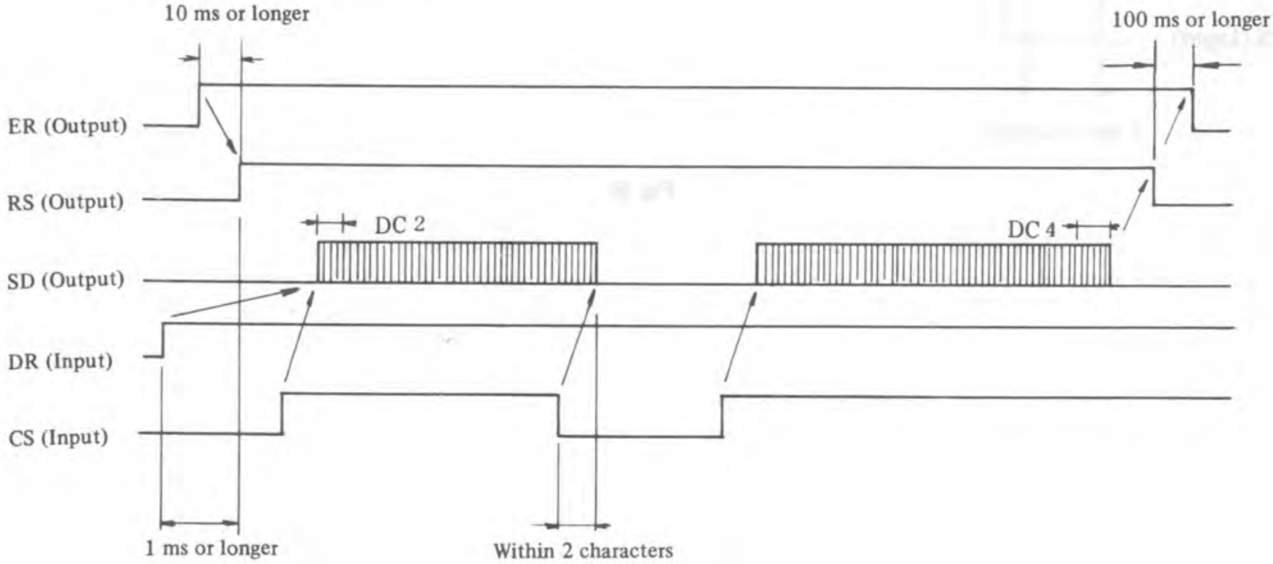


Fig. A

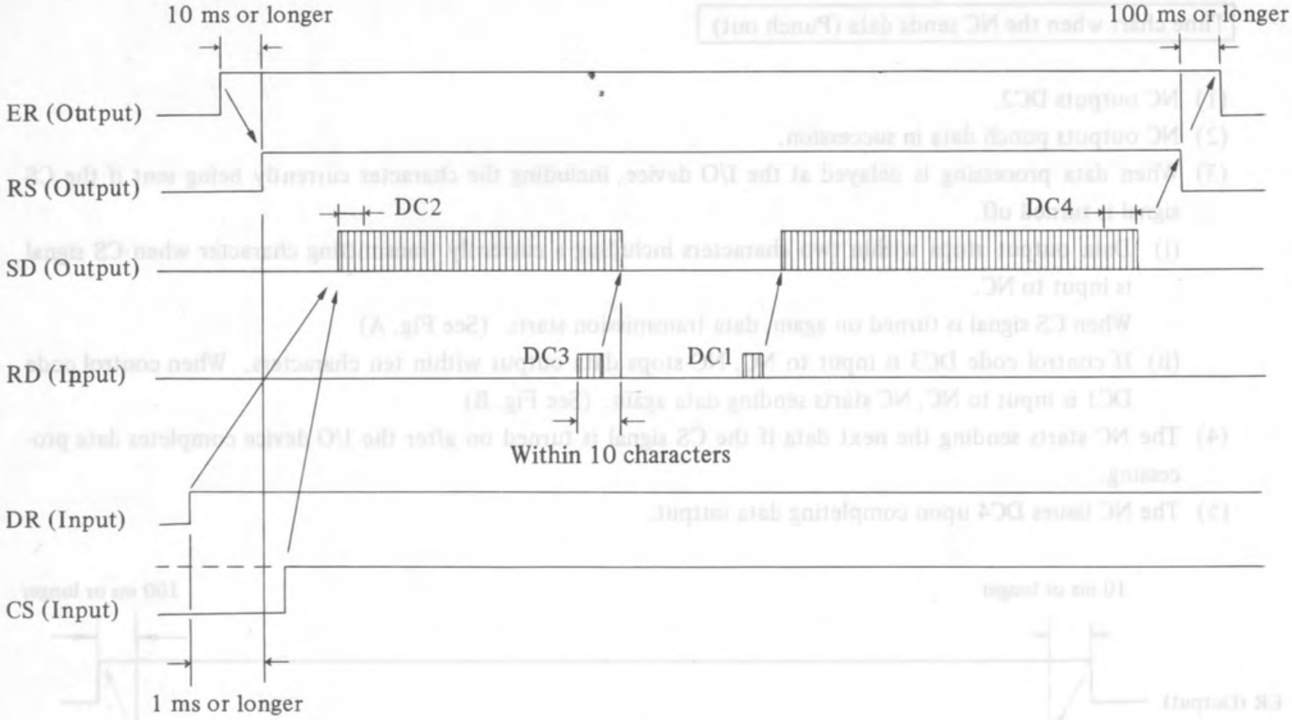

10 ms or longer
100 ms or longer
ER (Output)
RS (Output)
DC2
DC4
SD (Output)
DC3
DC1
RD (Input)
Within 10 characters
DR (Input)
CS (Input)
1 ms or longer

Fig. B

446

4. SETTINGS AND PARAMETERS

4.1 Setting (Refer to IV 5.8 for setting operation.)

Set INPUT DEVICE 2 to 1 on SETTING No. 1 screen.

Item \ Display	0	1
TV CHECK	No	Yes
PUNCH CODE	EIA	ISO
INPUT UNIT	MM	INCH
INPUT DEVICE 1	NC tape	RMT
INPUT DEVICE 2	NC tape is read via device set by INPUT DEVICE 1, when registering an NC tape into memory.	NC tape is read via a device set by data number 340, when registering a program tape into memory.

Set an I/O device No. to 340 and 341 on SETTING No. 2 screen.

	3	4	0	IDVICE

IDVICE It selects an input device on registering a program in memory. (When INPUT DEVICE 2 = 1 (RS232C) of setting has been set, this setting becomes valid.)

Set value	
0	Tape reader
1	ASR33/43 (parameter such as a baud rate, etc. should be set to parameter No. 310.)
2	RS232C (parameter such as a baud rate, etc. should be set to parameter No. 311.)
3	RS232C (parameter such as a baud rate, etc. should be set to parameter No. 312.)
4	RS232C (parameter such as a baud rate, etc. should be set to parameter No. 313. Also used for data transfer when Robot interface option is provided.)

	3	4	1	ODVICE

ODVICE It selects an output device on punching out.

Set value	
0	FACIT PUNCHER
1	ASR33/ASR43 (parameters such as a baud rate, etc. should be set to parameter No. 310.)
2	RS232C (parameters such as a baud rate, etc. should be set to parameter No. 311.)
3	RS232C (parameters such as a baud rate, etc. should be set to parameter No. 312.)
4	RS232C (parameters such as a baud rate, etc. should be set to parameter No. 313. Also used for data transfer when Robot interface option is provided.)

Above parameters can also be set as setting parameter.

4.2 Parameters (Refer to Appendix 6 for setting procedure of parameters)

				7	6	5	4	3	2	1	0
	0	0	8	ICR							

ICR 1: LF is punched for an EOB code in ISO code.
0: LFCRCR is punched for an EOB code in ISO code.

				7	6	5	4	3	2	1	0
	0	1	0							REDT	

REDT 1: Storage of program to memory is started by pressing the cycle start button in EDIT mode.
0: Storage of program to memory is not started by pressing the cycle start button in EDIT mode.

				7	6	5	4	3	2	1	0
	0	2	4			DLME	RDAL				

DLME 1: When programs are stored in memory, all previously stored programs are canceled automatically.
0: When programs are stored in memory, all previously sotred programs are not canceled.
RDAL 1: When programs are stored in memory, always all programs are stored.
0: When programs are stored in memory, whether one program is stored or all programs are stored is selected by operation of MDI.
(All the programs is stored by [O] [–] [9] [9] [9] [9] [READ])

				7	6	5	4	3	2	1	0
	3	0	6					NEOP			

NEOP 1: When registering a tape in memory, M02, M30 or M99 is not counted as the end of program.
0: When registering a tape in memory, M02, M30 or M99 is counted as the end of program.

3	1	0	NFED1		RSCB1	STP21	BAD1			
3	1	1	NFED2		RSCB2	STP22	BAD2			
3	1	2	NFED3		RSCB3	STP23	BAD3			
3	1	3	NFED4		RSCB4	STP24	BAD4			
			7	6	5	4	3	2	1	0

NFED 1, 2, 3, 4 In order of I/O device 1, 2, 3 and 4, feed is executed or not.
1: Executed
0: Not executed

RSCB 1, 2, 3, 4 Whether control codes (DC1 ~ DC4) are used or not on I/O devices 1, 2, 3, and 4 in order.
1: The control codes are not used.
0: The control codes are used.

STP2 1, 2, 3, 4 In order of I/O devices 1, 2, 3 and 4, the stop bit is to be 2 bits/1 bit.
1: The stop bit is to be 2 bits.
0: The stop bit is to be 1 bit.

BAD 1, 2, 3, 4 In order of I/O devices 1, 2, 3 and 4, it sets a baud rate.

Baud rate	BAD 1, 2, 3, 4			
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

ASR33 = 110 bauds
ASR43 = 300 bauds
(110 bauds)
Bubble cassette = 4800 bauds

Note 1. Refer to parameters 340, 341.
Note 2. Parameter number 313 is used for data transmission between NC and robot with robot control option.

3	1	8							RSTL	
			7	6	5	4	3	2	1	0

RSTL 1: Signal STL is not issued by cycle start operation in EDIT mode to load programs into memory.
0: It is issued.

449

	3	4	0

IDVICE

IDVICE It selects an input device on registering a program in memory. (When INPUT DEVICE 2 = 1 (RS232C) of setting has been set, this setting becomes valid.)

Set value	
0	Tape reader
1	ASR33/43 (parameter such as a baud rate, etc. should be set to parameter No. 310.)
2	RS232C (parameter such as a baud rate, etc. should be set to parameter No. 311.)
3	RS232C (parameter such as a baud rate, etc. should be set to parameter No. 312.)
4	RS232C (parameter such as a baud rate, etc. should be set to parameter No. 313. Also used for data transfer when Robot interface option is provided.)

	3	4	1

ODVICE

ODVICE It selects an output device on punching out.

Set value	
0	FACIT PUNCHER
1	ASR33/ASR43 (parameters such as a baud rate, etc. should be set to parameter No. 310.)
2	RS232C (parameters such as a baud rate, etc. should be set to parameter No. 311.)
3	RS232C (parameters such as a baud rate, etc. should be set to parameter No. 312.)
4	RS232C (parameters such as a baud rate, etc. should be set to parameter No. 313. Also used for data transfer when Robot interface option is provided.)

Above parameters can also be set as setting parameter.

Revision Record

FANUC SYSTEM 6M-MODEL B OPERATOR'S MANUAL B-54044E

Edition	Date	Contents	Edition	Date	Contents
02	1984, 01	1. Local coordinate system setting function has been explained in III 5.17. 2. Coordinate system rotation function has been explained in III 6.8. 3. Assignment of variable names has been explained in III 10.3.3. 4. Macro call M codes have been explained in III 10.7.7. 5. Index table indexing function has been explained III 12. 6. 14 color CRT operation has been explained in IV 5.35. 7. Multi-handle function has been explained in Appendix 13. 8. Input/output interface has been explained in Appendix 14. 9. Some parameters and alarms on additional functions have been added.			
01	1982, 12				

453

Oct., 1985

FANUC LTD

454