Instructions
TOE-C843-5
July, 1980

YASNAC® 3000G

OPERATOR'S MANUAL

YASKAWA Electric Mfg. Co., Ltd.

TABLE OF CONTENTS

Page

1. INTRODUCTION 1

2. PROGRAMMING 1

2.1 TAPE FORMAT 1

2.1.1 Tape Format 1
2.1.2 Address and Function Characters 3
2.1.3 Buffer Register 4
2.1.4 TV Check (Tape vertical parity check) 4
2.1.5 Optional Block Skip ("/" code) 4

2.2 SEQUENCE NUMBER 5

2.2.1 4-Digit Sequence Number 5

2.3 MOVE COMMAND 5

2.3.1 Least Output Increment 5
2.3.2 Least Input Increment 5
2.3.3 Maximum Programmable Value 6

2.4 RAPID TRAVERSE RATE 6

2.4.1 Rapid Traverse Rate 6
2.4.2 Range of Rapid Traverse Rate 6

2.5 CUTTING FEED 6

2.5.1 Feed Function A (F-function A) 6
2.5.2 Feed Function B (F-function B) 7
2.5.3 Feed per Revolution A (mm/rev command A)† 7
2.5.4 Feed per Revolution B (mm/rev command B)† 8
2.5.5 Summary of Feed Function 9

2.6 AUTOMATIC ACCELERATION AND DECELERATION 9

2.6.1 Acceleration and Deceleration of Rapid Traverse and Manual Feed 9
2.6.2 Acceleration and Deceleration of Cutting Feed 9

2.7 MISCELLANEOUS FUNCTIONS (M-FUNCTION) 9

2.7.1 M Codes for Stop (M00, M01, M02, M30) 9
2.7.2 M Codes for Other Internal Processes† 10
2.7.3 Other M Codes 12
2.7.4 Subprogram Function (M98, M99)† 12

2.8 SPINDLE-SPEED FUNCTION (S-FUNCTION) 16

2.8.1 S 2-Digit Command 16
2.8.2 S 4-Digit† Command 16

2.9 TOOL FUNCTION (T-FUNCTION) 17
2.9.1 T 2-Digit 17
2.9.2 T 4-Digit † 17
2.10 TOOL OFFSET 17
2.10.1 Outline of Tool Offset 17
2.10.2 Storage of Tool Offset Values 17
2.10.3 H- and D-function (H, D codes) 18
2.11 B-FUNCTION † 19
2.12 SIMULTANEOUSLY CONTROLLABLE AXES 19
2.12.1 Three-Axis Control 19
2.12.2 Four-Axis Control† 19
2.13 ADDITIONAL 4TH AXIS † 20
2.13.1 Rotary Axis (A, B or C Axis) 20
2.13.2 Linear Axis (U, V or W Axis) 20
2.13.3 4th Axis Command Address 20
2.14 PREPARATORY FUNCTIONS (G-FUNCTION) 21
2.14.1 List of G Codes and Groups 21
2.14.2 Positioning (G00) 23
2.14.3 Linear Interpolation (G01) 23
2.14.4 Circular Interpolation (G02, G03) 24
2.14.5 Dwell (G04) 29
2.14.6 Circle Cutting (G12, G13)† 30
2.14.7 Helical Cutting (G14, G15)† 32
2.14.8 Plane Designation (G17, G18, G19)† 34
2.14.9 Program Copy (G25) 34
2.14.10 Reference Zero Check (G27)† 36
2.14.11 Automatic Return to Reference Zero (G28)† 36
2.14.12 Return from Reference Zero (G29)† 37
2.14.13 Tool Radius Compensation C (G40, G41, G42)† 38
2.14.14 Tool Position Offset B (G43, G44, G49) 55
2.14.15 Tool Position Offset A (G45 to G48) 59
2.14.16 Canned Cycles (G73, G76, G80 to G89)† 64
2.14.17 Absolute/Incremental Designation (G90, G91) 77
2.14.18 Programming of Absolute Zero Point (G92) 77
2.14.19 Feed Function Designation (G94, G95)† 78
2.14.20 Designation of Canned Cycle Return Point (G98, G99)† . . . 78

3. NC TAPE PUNCHING 79

3.1 TAPE CODE 79
3.1.1 Tape Code 79
3.1.2 EIA/ISO/Auto-Select 80
3.2 PROGRAMMING 80
3.2.1 Process Sheet 80
3.2.2 General Program Form 81
3.2.3 Precaution in Programming 81

3.3 NC TAPE 82
3.3.1 Paper Tape 82
3.3.2 Punching of NC Tape 82
3.3.3 Checking of NC Tape 82
3.4 NC TAPE HANDLING 83
3.4.1 Splice Tape 83
3.4.2 Keeping of NC Tape 83

4. NC OPERATOR'S PANEL 85

4.1 PUSHBUTTONS, KEYS, AND LAMPS 85
4.1.1 POWER ON/OFF Pushbuttons 85
4.1.2 Indicating Lamps 85
4.1.3 FUNCTION Select Keys 86
4.1.4 ADDRESS Select Keys and Display 86
4.1.5 NUMBER Setting Keys and Display 86
4.1.6 DATA Keys 87
4.1.7 WR (Write) Key 87
4.1.8 AS (Address Search) Key 87
4.1.9 Sequential Search Keys ↓ and ↑ 87
4.1.10 ERS (Erase), INS (Insert), ALT (Alter), and EOB (End of Block) Keys 87
4.1.11 START Key 88
4.1.12 RESET Key 88
4.1.13 ORG Key 88
4.1.14 Universal Display 88
4.1.15 TAPE FEED and SYSTEM NO. Switches 88
4.2 POWER ON/OFF OPERATION 89
4.2.1 Turning on Power 89
4.2.2 Turning off Power 89
4.2.3 Remote Turning ON/OFF Buttons 90
4.3 DISPLAY AND WRITING OF COMMAND DATA 90
4.3.1 Display of Command Data 90
4.3.2 Writing Command Data by MDI 91
4.3.3 Current Position Display 92
4.3.4 Incremental Value Display 93
4.3.5 Display of Tool Offset Value 93
4.3.6 Writing of Tool Offset Value 93
4.3.7 Display of Current Tool Offset Number 94
4.3.8 Parameter Display 95
4.3.9 Writing Parameters 95
4.3.10 Operation Time Display 96
4.3.11 Parameter Writing for Mirror-Image Axis 96
4.3.12 Alarm and Status Code Display 100
4.3.13 Display of Input/Output Signals 103
4.3.14 Address Search 104
4.3.15 TV Check (Vertical Parity Check) 104
4.3.16 Current Position Display Unit† 104
4.4 STORING TOOL OFFSET VALUES FROM NC TAPE† 105

4.5 PART PROGRAM STORAGE† 105
4.5.1 Storing Part Program from NC Tape† 105
4.5.2 Part Program Modification from NC Tape† 106
4.5.3 Storing Part Program from MDI† 107
4.5.4 Address Display of Tape Memory (Parameter No. 99) 108

4.6 EDIT† 108
4.6.1 Display of Stored Part Program† 108
4.6.2 Editing Stored Part Program† 109
4.6.3 Outline of Edit Operation† 111

4.7 PUNCHOUT OPERATION† 112
4.7.1 Tape Puncher† 112
4.7.2 Punchout of NC Tape† 113
4.7.3 Punchout of Tool Offset Value† 114
4.7.4 Sequence of Tape Data Storing and Punching Operation† . . 114

4.8 COLLATING OF STORED PROGRAM AND OFFSET VALUE 114
4.8.1 Collating of Stored Program† 114
4.8.2 Collating of Stored Offset Value† 114

4.9 OUTLINE OF OPERATION IN THE EDT MODE† 115

4.10 LEADSCREW ERROR COMPENSATION† 116

4.11 STROKE CHECK† 118

5. TAPE READER COMPARTMENT 120

5.1 TAPE READER COMPARTMENT 120
5.1.1 TAPE FEED and SYSTEM NO. Switches 120
5.1.2 Tape Reader 120
5.1.3 Tape Tumble Box 121

5.2 TAPE REELS† 121
5.2.1 Tape Reels† 121
5.2.2 6-Inch Tape Reels 121
5.2.3 8-Inch Tape Reels 122

6. CONTROL STATION FOR MACHINE 126

6.1 SWITCHING UNITS ON THE CONTROL STATION 126
6.1.1 MODE SELECT Switch 126
6.1.2 CYCLE START Pushbutton and Lamp 126
6.1.3 FEED HOLD Pushbutton and Lamp 126
6.1.4 EMERGENCY STOP Pushbutton 126
6.1.5 HANDLE Dial (Manual Pulse Generator)† 127
6.1.6 HANDLE AXIS Select Switch† 127
6.1.7 MANUAL PULSE MULTIPLY Select Switch† 127
6.1.8 JOG Pushbuttons 128
6.1.9 JOG FEEDRATE Switch 128
6.1.10 RAPID TRAVERSE RATE OVERRIDE Switch† 128
6.1.11 FEEDRATE OVERRIDE Switch 130

6.1.12 OVERRIDE CANCEL Switch† 130
6.1.13 ZERO RETURN Switch† (Manual Zero Return) 130
6.1.14 ZERO POSITION Lamps† (For X, Y, Z, and 4th Axes) . . . 130
6.1.15 SINGLE Block Switch 130
6.1.16 DISPLAY LOCK/MACHINE LOCK Switch 130
6.1.17 DRY RUN Switch 131
6.1.18 OPTIONAL BLOCK SKIP Switch 131
6.1.19 OPTIONAL STOP Switch 131
6.1.20 MANUAL ABSOLUTE Switch 131
6.1.21 M-FUNCTION LOCK Switch (Miscellaneous Function Lock) 132
6.1.22 Z-AXIS FEED NEGLECT Switch 132
6.1.23 FEED CODE LOCK Switch (F-code Lock) 132
6.1.24 BLOCK RETURN Switch and BLOCK RETURN END Lamp† 132
6.1.25 MIRROR IMAGE AXIS Selector Switch† 133
6.1.26 ERROR DETECT ON/OFF Switch† 133
6.1.27 TOOL OFFSET AUTO-STORE Pushbutton and Lamp† . . . 133
6.1.28 EDIT LOCK Switch† 134
6.1.29 START LOCK Input† (or Switch) 134
6.1.30 AXIS INTERLOCK Input 134
6.1.31 TRAVERSE RATE CLAMP Input† 134

6.2 OPERATION FOR MACHINING 135

6.2.1 Manual Return to Reference Zero 135
6.2.2 Block Return† 135
6.2.3 Automatic Tool-offset Storing† 137

7. OPERATION PROCEDURE 140

7.1 INSPECTION BEFORE TURNING ON POWER 140

7.2 TURNING ON POWER 140

7.3 MANUAL OPERATION 140

7.4 PREPARATION FOR LEADSCREW ERROR COMPENSATION AND STROKE CHECK† 141

7.5 PREPARATION FOR AUTOMATIC OPERATION 141

7.6 OPERATION IN TAPE AND MEMORY MODE 142

7.7 MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION . 142

7.8 AUTOMATIC OPERATION IN MDI MODE 143

7.9 MDI OPERATION INTERRUPTING AUTOMATIC OPERATION . . . 143

7.10 PREPARATION FOR TURNING OFF POWER 143

7.11 TURNING OFF POWER 143

8. MAINTENANCE 145

8.1 ROUTINE INSPECTION SCHEDULE 145

8.1.1 Tape Reader 146
8.1.2 Control Panel 146
8.1.3 Servomotor and DC Motor for Spindle 146
8.1.4 Battery 147

8.2 REPLACEMENT OF BATTERY 148

8.3 POWER FUSES 149

8.3.1 Fuses of Composite Control Power Supply Unit 149
8.3.2 Fuses of Servo Unit 150

8.4 THERMAL OVERLOAD RELAY OF SERVO UNIT 150

8.5 OTHERS 151

8.5.1 Molded-case Circuit Breakers (MCB) 151
8.5.2 Power Receptacles for Maintenance Tools 151

8.6 TROUBLE CAUSES AND REMEDIES 151

8.6.1 On-line Diagnostics 151
8.6.2 Alarm Codes and Remedies 151
8.6.3 Input/Output Signals 152
8.6.4 Before Maintenance Call 160

APPENDIX-1 INTERFACE FOR M-, S-, T-, AND B-CODE 161

APPENDIX-2 SPINDLE CONTROL IN CANNED CYCLES 169

APPENDIX-3 LIST OF PARAMETERS 173

APPENDIX-4 TROUBLE CAUSES AND REMEDIES 185

1. INTRODUCTION

YASNAC 3000G is a low cost, high performance CNC specifically designed for machining centers to direct a three- or four-motion machine tool. It incorporates the latest microelectronics technology in every design feature, which remarkably upgrades the basic functions and offers a wide variety of optional functions.

The improved transistorized PWM servos combined with the optimum pulse distribution control provides faster and higher accuracy of machining.

The totally-enclosed, dustproof enclosure protects all components from the attack by rugged industrial environment. This contributes to longer-lasting control reliability.

On-line diagnostics of YASNAC 3000G identifies the failed spot and minimizes down time.

2. PROGRAMMING

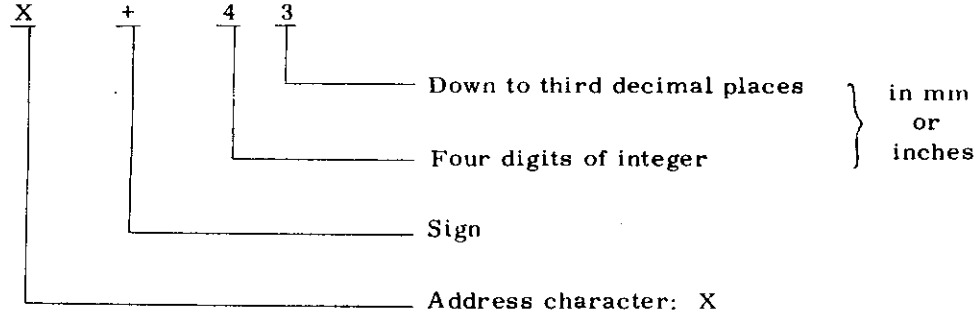
2.1 TAPE FORMAT

2.1.1 TAPE FORMAT

A variable block format conforming to JIS# B 6313 is used for YASNAC 3000G.

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

EXAMPLE



Note: A decimal point must be omitted in actual programming.

The leading zeros can be suppressed for all address codes. Plus signs need not be programmed, but all minus signs must be programmed.

In the manual, EOB code in a program example is represented by an asterisk (*). In actual programming, CR (EIA code) or LF/NL (ISO code) should be used instead of the asterisk (*).

Japanese Industrial Standard

Table 2.1.1 Tape Format

Items \ Least input increment		Metric		Inch†	
		0.01 mm	0.001 mm	0.001 inch	0.0001 inch
Sequence Number		N4		N4	
Preparatory Function		G2		G2	
Move Command	X	X + 42	X + 43	X + 33	X + 34
	Y	Y + 42	Y + 43	Y + 33	Y + 34
	Z	Z + 42	Z + 43	Z + 33	Z + 34
	α (4th axis)†	β + 42	β + 43	β + 42	β + 43
		γ + 42	γ + 43	γ + 33	γ + 34
Arc Radius Designation	G02, G03 in X direction (Radius of G12 to G15)	I + 42	I + 43	I + 33	I + 34
	in Y direction	J + 42	J + 43	J + 33	J + 34
	in Z direction	K + 42	K + 43	K + 33	K + 34
	Radius Designation†	R + 42	R + 43	R + 33	R + 34
Helical Cutting†	Lead	K42	K43	K33	K34
	No. of turns	L7		L7	
Feedrate	mm/min inch/min	F4 or F41		F31 or F32	
	mm/rev† inch/rev†	F22 or F23		F13 or F14	
Tool Offset No.	H	H2		H2	
	D	D2		D2	
Tool Function		T2 T4†		T2 T4†	
Spindle-Speed Function		S2 S4†		S2 S4†	
Miscellaneous Function		M2		M2	
B-Function†		B3		B3	
Dwell		P43		P43	
Canned Cycles†	Cutting Feed Start Point	R + 42	R + 43	R + 33	R + 34
	Deep Hole Drilling Depth	Q42	Q43	Q33	Q34
	Dwell	P43		P43	
	No. of Cycles	L7		L7	
Subprogram†	Sequence Number	P4		P4	
	No. of Cycles	L7		L7	
G25 Program Copy†	Start Sequence No.	P4		P4	
	End Sequence No.	Q4		Q4	
	No. of Cycles	L7		L7	
End of Block		*		*	

Notes:

1. Functions with † are optional.
2. Only the numbers of digits are shown in the above table. Some of them cannot exceed the maximum programmable value. Refer to the description of each item.
3. Least input increment (0.01/0.001 mm) is parameter-switched.
4. Feedrate is designated by either mm (inch) per minute or mm (inch) per revolution, which can be switched by G code. (G94, G95)
5. β is A, B, or C axis; γ is U, V, or W axis.
6. Inch input is available for the control equipped with an optional Inch/Metric selection. Inch/Metric are parameter-switched.
7. Instead of Deep Hole Drilling Depth Q, addresses I (1st depth), J (Decrement) and K (Final depth) may be used for designating deep hole drilling cycles.

2.1.2 ADDRESS AND FUNCTION CHARACTERS

Address characters and the meanings are shown in Table 2.1.2.1.

Function characters and the meanings are shown in Table 2.1.2.2.

Table 2.1.2.1 Address Characters

B: Basic
O: Optional

Address Characters	Meanings	Section
A	Additional rotary axis parallel to X-axis	O
B	Additional rotary axis parallel to Y-axis	O
C	Additional rotary axis parallel to Z-axis	O
D	Tool radius offset number	O, B
E	Unused	
F	Cutting feedrate	B
G	Preparatory function	B
H	Tool position offset number	B
I	X-coordinate of arc center Radius for circle cutting/helical cutting	B O
J	Y-coordinate of arc center	B
K	Z-coordinate of arc center Helix lead for helical cutting	B O
L	Numbers of canned cycles, subprogram (M98), subprogram copies, helical cutting cycles	O
M	Miscellaneous functions	B
N	Sequence number	B
O	Same as N (EIA only)	B
P	Dwell time, Sequence No. of subprogram (M98, 99), Sequence No. for start of program copy (G25)	B, O O
Q	Sequence No. for G73, G83 depth of cut, shift of G76, G87, End of program copy (G25)	O
R	Radius designation of a circular arc for G02 and G03 R-point for canned cycles	O, B
S	Spindle function	B
T	Tool function	B
U	Additional linear axis parallel to X-axis	O
V	Additional linear axis parallel to Y-axis	O
W	Additional linear axis parallel to Z-axis	O
X	X-coordinate	B
Y	Y-coordinate	B
Z	Z-coordinate	B

Table 2.1.2.2 Function Characters

EIA	ISO	Meanings
Blank	NuL	Error in significant data area in EIA Disregarded in ISO
BS	BS	Disregarded
Tab	HT	Disregarded
CR	LF/NL	End of block
	CR	Disregarded
SP	SP	Disregarded
ER	%	Rewind stop
UC		Disregarded
LC		Disregarded
	(	Control out
	)	Control in
+	+	Disregarded
-	-	Negative sign
0 to 9	0 to 9	Numerals
a to z	A to Z	Address characters
0	:	Regarded as N
/	/	Optional block skip
Del	DEL	Disregarded (Including All Mark)

Notes:

1. Characters other than the above cause error in significant data area.
2. Information between Control Out and Con-In is ignored as insignificant data.
3. Tape code (EIA or ISO) can be switched by parameter.
4. Label Skip function: After switching on the power supply, or after making RESET operation, Label Skip function becomes effective, and LABEL SKIP lamp on the NC operation panel lights. In this condition, all tape information is ignored before the first EOB code is read. When an EOB code is read, LABEL SKIP lamp goes out automatically, and a significant data area is initiated.

2.1.3 BUFFER REGISTER

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

Buffer capacity is 96 characters including EOB. A command of block exceeding 96 characters causes errors and the alarm code "14" is displayed. The following characters are not read into buffer register and are not restricted by buffer capacity.

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

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

2.1.4 TV CHECK (TAPE VERTICAL PARITY CHECK)

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

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

2.1.5 OPTIONAL BLOCK SKIP ("/" CODE)

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

2.2 SEQUENCE NUMBER

2.2.1 4-DIGIT SEQUENCE NUMBER

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

Sequence number is represented by four digits from 0001 to 9999 with the preceding N. The leading zeros can be suppressed.

EXAMPLE

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

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

2.3 MOVE COMMAND

2.3.1 LEAST OUTPUT INCREMENT

The least output increment is the minimum unit of movement by which the machine can move and is represented in millimeters per pulse (or in deg/pulse†).

Table 2.3.1 Least Output Increment

	Linear axis	Rotary axis†
X, Y, Z axis	0.001 mm/ pulse	———
4th axis†	0.001 mm/ pulse	0.001 deg/ pulse

Note: Some machines have the least output increment of 0.002 mm/pulse. Refer to the specifications given by the machine tool builder.

2.3.2 LEAST INPUT INCREMENT

The least input increment is the minimum unit that can be programmed and is represented in millimeters (or in inches† or degrees†).

Table 2.3.2 Least Input Increment

	x 10	x 1
Metric system	0.01 mm	0.001 mm
Inch system†	0.001 inch	0.0001 inch
Degree†	0.01 deg.	0.001 deg.
	↑	↑
Contents of Parameter No. 88	"1"	"0"

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

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

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

NOTES:

- If NC tape programmed by 0.001 mm is fed into or stored in an equipment set by 0.01 mm increment, the machine will move ten times the intended dimensions.
- If the increment system is switched when the contents of NC tape are stored in memory†, the machine will move by ten times or one tenth of the commanded dimensions.
- When the stored program is punched out on the tape†, the stored figures are punched out "as stored" regardless of switching of the increment system.

2.3.3 MAXIMUM PROGRAMMABLE VALUE

Maximum programmable values of move command are shown below.

Table 2.3.3 Maximum Programmable Values

Increment system	x 10	x 1
Metric	±8388.60 mm	±8388.607 mm
Inch†	±330.260 inches	±330.2601 inches
Degree†	±8388.60 deg.	±8388.607 deg.

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

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

Note: The machine may not function properly if a move command over the maximum programmable value is given. The above maximum programmable values also apply to distance command addresses I, J, K (R, Q)† in addition to move command addresses X, Y, Z (α†).

2.4 RAPID TRAVERSE RATE

2.4.1 RAPID TRAVERSE RATE

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

Rapid traverse rate is set with parameter No. 94. To the rapid traverse rate, some override traverse rates can be applied. (Refer to 6.1.10 RAPID TRAVERSE RATE OVERRIDE Switch†)

2.4.2 RANGE OF RAPID TRAVERSE RATE

Rapid traverse rate is set for each axis in the following step.

Step of rapid traverse rate--7.5 mm/min
(or deg/min†)

The upper limit of rapid traverse rate is 15 m/min (or 41.6 rev./min†). However, the upper limits are subject to the design of the servomotor or machine used. The optimum rate must be determined in accordance with the specifications given by the machine tool builder.

2.5 CUTTING FEED

2.5.1 FEED FUNCTION A (F-FUNCTION A)

The four digits following the address F are for commanding tool feed rates minute (mm/min). Table 2.5.1.1 shows the programmable range of the F code.

Table 2.5.1.1 F code (mm/min-A)

	Format	Range of Feedrate	Meanings
Metric System	F4	F1 to F5400	1 to 5400 mm/min
Inch† System	F31	F1 to F2125	0.1 to 212.5 inch/mm

Note: Where the 4th axis is a rotary axis, refer to 2.14.3 Linear Interpolation (G01).

- Machine tool builder's specifications may fix the maximum cutting feedrate. If so, the relevant maximum value is set for parameter No. 75. Even when F command in excess of this set limit value is given, the feedrate is fixed in accordance with specifications.
- Values of F command at linear or circular interpolation represent the tangential feedrate when two axes are simultaneously controlled.

EXAMPLE 1

G91 (Incremental command designation)

G01 X4000 Y3000 F500 *

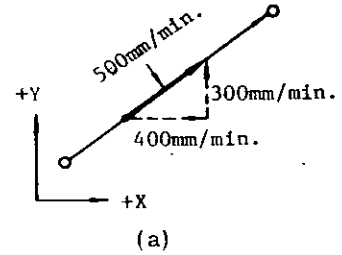
In the above case, the feedrate is:

F = 500 (mm/min)

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

Z-axis component

X-axis component



(a)

Fig. 2.5.1.1

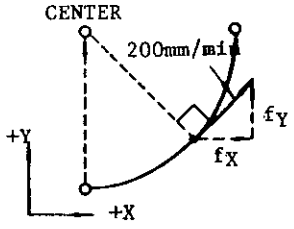
EXAMPLE 2

G03 X··· Y··· I··· F200 *

In the above case, the feedrate is:

F = 200 mm/min

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



(b)

Fig. 2.5.1.2

· Values of F command at linear interpolation represent the tangential feedrate when three axes are simultaneously controlled.

EXAMPLE

G01 X··· Y··· Z··· F400 *

In the above case, the feedrate is:

F = 400 = $\sqrt{fx^2 + fy^2 + fz^2}$ (mm/min)

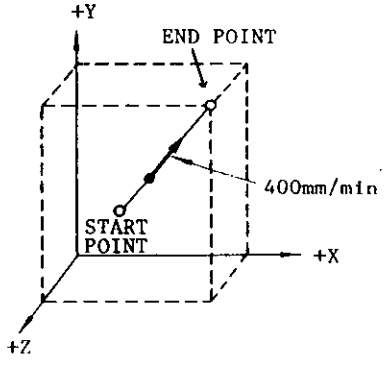


Fig. 2.5.1.3

· Values of F command at linear interpolation represent the tangential feedrate when four axes are simultaneously controlled .

F (mm/min) = $\sqrt{fx^2 + fy^2 + fz^2 + f\alpha^2}$

NOTES:

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

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

EXAMPLE

F-250 * . . . Wrong

2.5.2 FEED FUNCTION B (F-FUNCTION B)

The following feed function may be adopted in place of the feed function A described above when parameter No. 87 is set to "15."

· Commanding tool feed per minute (mm/min) in five digits following the address F.

· The command range of the F code is as follows.

Table 2.5.2.1 F Code (mm/min-B)

	Format	Range of Feedrate	Meanings
Metric	F41	F1 to F54000	0.1 to 5400.0 mm/min
Inch†	F32	F1 to F21259	0.01 to 212.59 inch/min

As shown above, the feed function B can be treated similarly with the feed function A, except for the number of digits of commanded value.

2.5.3 FEED PER REVOLUTION A (MM/REV COMMAND A)†

When a spindle pulse generator is installed†, the feedrate per revolution function can be used, and a new G code will be used for this function. Before F function for feedrate per revolution is given, G code of F group shown below must be designated. When the power supply is switched on, G94 is in effect.

G code of D group	Function
G94	Feedrate per minute (mm/min) designation
G95	Feedrate per revolution (mm/rev) designation

Note: For details of these G codes, refer to 2.14.19 Feed Function Designation (G94, G95)†

Since F code is modal, the code is effective until the next F code is given. However, when G94/G95 are switched over, F code must be designated again.

After the designation of G95, the feedrate-of-tool per spindle-revolution can be given by 4 digits following F. The command range of the F code is as follows.

Table 2.5.3.1 F Code (mm/rev.-A)

	Format	Range of Feedrate	Meanings
Metric	F22	F1 to F9999	0.01 to 99.99 mm/rev.
Inch†	F13	F1 to F3936	0.001 to 3.936 inch/rev.

However, the programming of feedrate is restricted by the spindle speed (S) as shown below.

F	x	S	≦	5400
(mm/rev)		(rpm)		(mm/min)

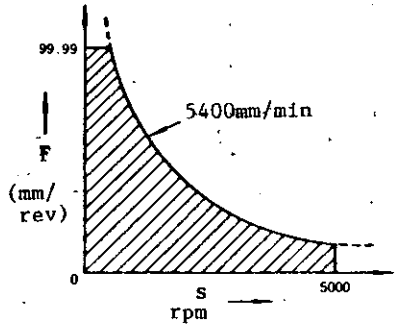


Fig. 2.5.3.1 Restriction of Feedrate (F) and Spindle Speed (S)

EXAMPLE

G95 S500 (rpm) *

G01 Y10000 F60 *

In the above case, the feedrate is:

F x S = 0.60 mm/rev x 500 rpm

= 300 mm/min

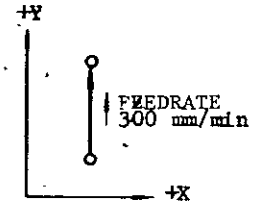


Fig. 2.5.3.2

Note: When G94 command is given, feedrate in mm per minute can be given by F-function A.

2.5.4 FEED PER REVOLUTION B† (MM/REV COMMAND B)

Instead of the feed per revolution A described above, the following feed per revolution B may optionally be incorporated when parameter No. 87 is set to "15."

After designating G95, tool feed per each revolution of the spindle(mm/rev) can be given by the 5 digits following the F address.

Table 2.5.4.1 shows the programmable range of F code.

Table 2.5.4.1 F Code (mm/rev.-B)

	Format	Range of Feedrate	Meanings
Metric	F23	F1 to F99999	0.001 to 99.999 mm/rev.
Inch†	F14	F1 to F39366	0.0001 to 3.9366 inch/rev.

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

F	x	S	=	5400
(mm/rev)		(rpm)		(mm/min)

Note: When G94 command is given, feedrate in mm per minute can be given by F-function B.

2.5.5 SUMMARY OF FEED FUNCTION

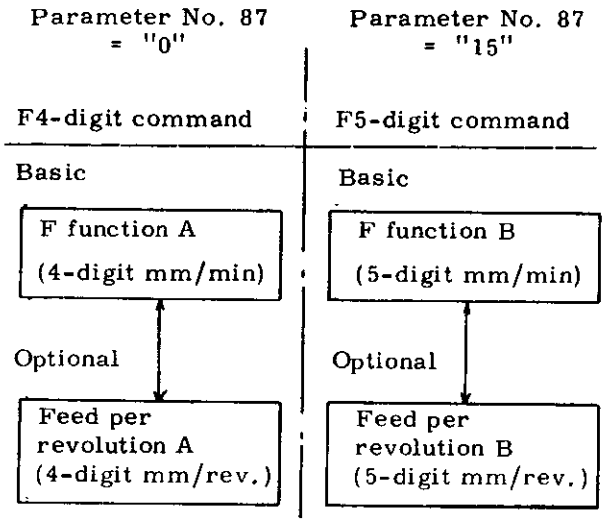


Fig. 2.5.5

2.6 AUTOMATIC ACCELERATION AND DECELERATION

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

2.6.1 ACCELERATION AND DECELERATION OF RAPID TRAVERSE AND MANUAL FEED

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

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

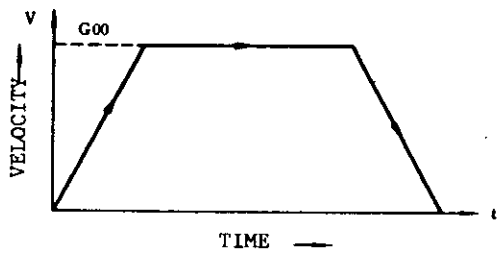


Fig. 2.6.1

2.6.2 ACCELERATION AND DECELERATION OF CUTTING FEED

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

- Cutting feed (G01 to G03)

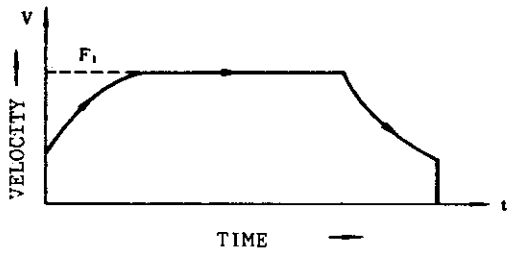


Fig. 2.6.2

2.7 MISCELLANEOUS FUNCTIONS(M-FUNCTION)

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

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

- M00 (Program Stop)

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

M01 (Optional Stop)

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

- M02 (End-of-Program)

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

- M30 (End-of-Tape)

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

NOTES:

- When M00, M01, M02 or M30 is given, it prevents the control from reading ahead the next block of information. The single decoded signal is fed in addition to the 2-digit BCD output for M codes. For the timing of output, refer to the Appendix 1.
- Whether M00, M01, M02 or M30 executes spindle stop, coolant off or some other executions, refer to the machine tool builder's manual.
- Whether the control is automatically reset or rewinds the tape (or memory), is determined by the following state.
  (a) Input signal of the control "EOP" (internal reset input) is wired for "ON" or not.
  (b) Input signal of the control "RWD" (rewind input) is wired for "ON" or not.

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

2.7.2 M CODES FOR OTHER INTERNAL PROCESSES†

M90 through M99 are for internal processes. Even when they are given, no external output signal (BCD and decoded output) will be output.

1. M94/M95(Mirror image ON/OFF)

   M94 ··· Mirror image OFF
   M95 ··· Mirror image ON

   - With these codes, mirror image operation can be started and stopped at any desired point on the program. These commands must always be made on a single block.
   - M94 and M95 are modal. When the power supply is turned on, M94 (OFF) is in effect.
   - The axis on which mirror images are to be effected is specified by parameter No. 91(or mirror image axis designation switch†.) For this procedure, refer to 4.3.11 Parameter Writing for Mirror-Image Axis (and 6.1.25 MIRROR IMAGE AXIS Selector Switch†).
   - When M95 is given, the subsequent blocks will control the machine in mirror-image fashion, that is, movements in the specified coordinate direction will be reversed.

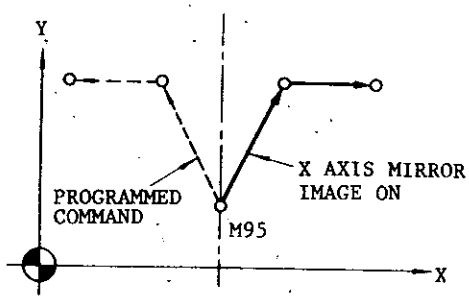


Fig. 2.7.2.1

With both the absolute and increment move commands, the same mirror image effect will be obtained. The start point of the respective block constitutes the mirror point.

- When M94 is given, mirror image effect will be cancelled on the subsequent blocks. Mirror image operation must be started and cancelled at the same position.

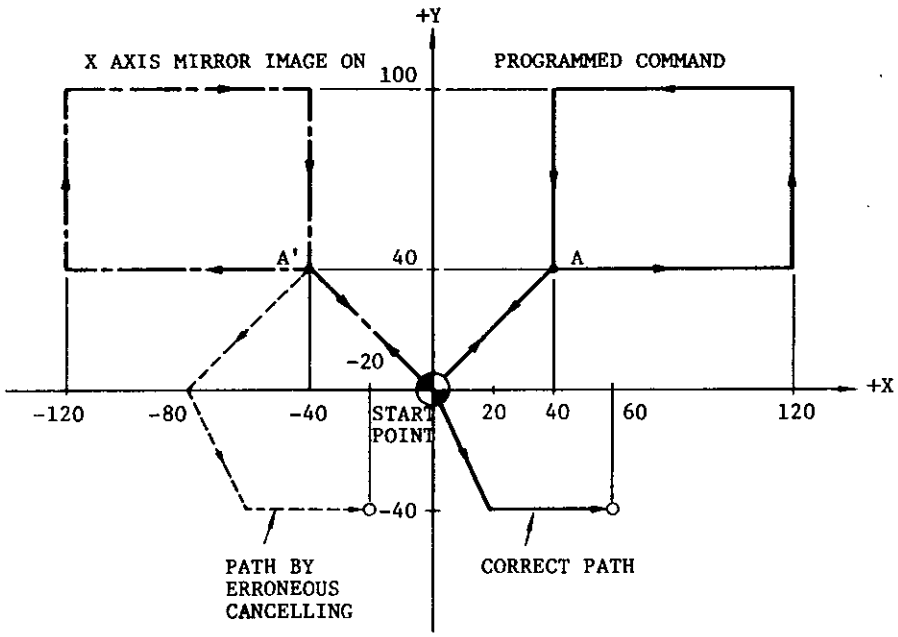
NOTES:

- When G28 or G29 is used to change tools or for ending machining processes, make sure to cancel the mirror image effect by means of M94. If mirror image effect is not cancelled when G28 or G29 is given, an input error results.
- The mirror image effect is not effective on the offset movement resulting from the tool offset function B. It is effective on the offset movement resulting from the tool offset function A and tool radius compensation C.
- Do not switch the designation of mirror image axis during operations under M95 (ON) mode.
- Displayed current position by POS key in mirror image fashion indicates the actual motion of tool. Displayed data by COM key show programmed commands.

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

· Program must be made so that mirror image operation starts and stops at the same position. If the start position and the stop position are not the same, movements of the machine after cancelling mirror image will be shifted by the difference between the both positions.

EXAMPLE




```
N01 G92 X0 Y0 *
N02     M95 * ..... Mirror image on.
N03 G90 G01 X4000 Y4000 F300 *
N04     X12000 *
N05     X10000 *
N06     X4000 *
N07     X4000 *
                    <= If "M94 *" is programmed here, the
N08     X0 Y0 *        tool moves on the dotted line.
N09     M94 * ..... Correct mirror image off.
N10     X2000 Y-4000 *
N11     X6000
.
.
.
```

Fig. 2.7.2.2

2. M96/M97 (Compensation C circular path ON/OFF)*

M96 ... Compensation C circular path ON
M97 ... Compensation C circular path OFF

- In the G41 or G42 cutter radius compensation mode, when M96 is given, the tool moves along a circular path around a corner with an angle of 180° or larger. In the M97 mode, the tool does not move along a circular path at the corner, but moves along two intersecting straight lines intersecting at a calculated intersecting point shifted from the programmed contour by the tool radius.

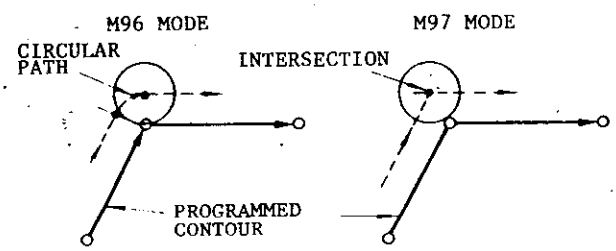


Fig. 2.7.2.3

- M96 and M97 are modal. When the power is turned on, M96 takes effect.

- M96 and M97 are effective on the following move command blocks.

```
G01 X... Y... F...
    M96 (or M97) *      ┐ Effective from
(G01) X... Y... *       ┘ the corners of these 2 blocks

G01 X... Y... F... *
    M96 (or M97) *      ┐ Effective from
(G01) X... Y... *       ┘ the corners of these 2 blocks
```

3. M98/M99 (Subprogram function)†

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

2.7.3 OTHER M CODES

For using M codes, other than those mentioned above, refer to the machine tool builder's manual.

Table 2.7.3 Typical Examples of M codes for Machine

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

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

2.7.4 SUBPROGRAM FUNCTION (M98, M99)†

The sequence of operation can be altered by this option which is available for the control supplied with part program storage. The M98 and M99 commands can specify the sequence number of the block to be performed next. (The previous or the following block.) With M98 code, up to 8 levels of subprogram jumps are possible.

1 Jump Instruction: P... M99 *

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

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

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

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

Address L specifies the number of repetitions of the subprogram. The maximum programmable value of the address L is 8388607. After execution, the command returns to the main program.

When address L is omitted, the subprogram is executed only once. M98 may also be given in a subprogram to jump to another subprogram. With the first jump from the main program counted as the simplex jump, jumps are possible up to eight levels.

3 Automatic Return Instruction from subprogram: M99 *

At the end of a subprogram, M99 is given in a single block. When M99 is given in the subprogram to which a jump was made under the command of M98, the block in the main program following the block in which the jump was given is called back automatically.

4 Return Block Instruction from Subprogram: P··· M99 *

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

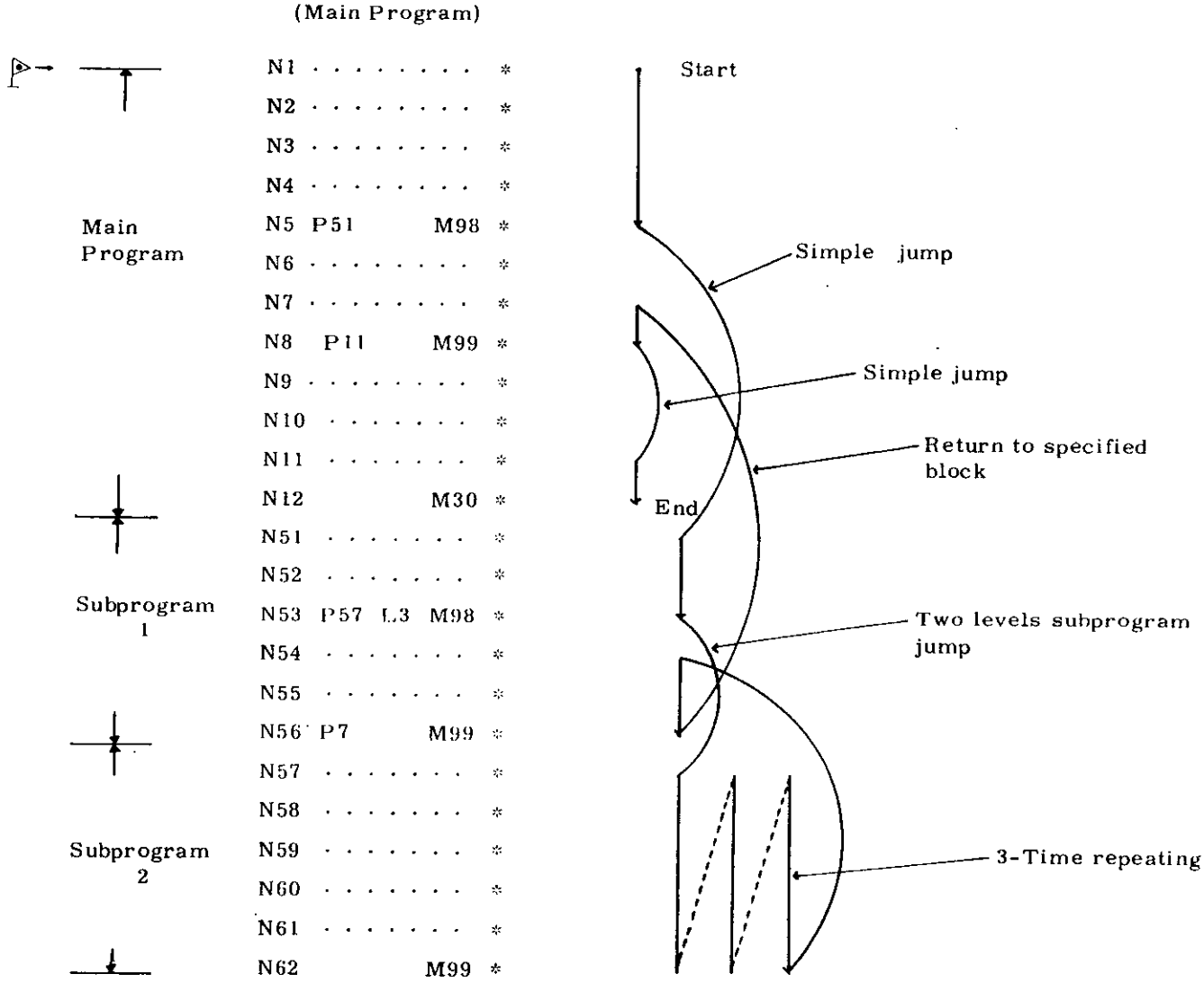


Fig. 2.7.4.1

NOTES:

- "Subprogram" simply means the program specified by M98 command. Therefore, no special consideration is required for storing and editing of subprograms.
- M98 cannot be used in a subprogram more than 8 levels. Even when the program copy (G25) is used in combination, the limit is 8. If more than 8 levels are programmed, an alarm code "15" will be displayed.
- Signals M98 and M99 are not sent to the machine.
- The sequence number designated by the address P is searched from the first address of memory and the execution of program starts from the first block of the sequence number searched. Therefore, the same sequence number should not be used.
- When the sequence number designated by the address P is not found, it causes an error.
- Addresses P and N designate sequence numbers. Up to four-digit number can be used for sequence numbers and leading zeros can be suppressed.
- The sum of the characters of the main program and the subprogram should not exceed the memory capacity of the control.
- Address L cannot be used together with M99 command.
- The remaining number of repetitions of a subprogram is displayed when POS key is depressed, the address L is set on the address indicator, and "01" through "08" is set as NUMBER, during the operation of repeating command
  P··· L··· M98 *
  The digit set as NUMBER means the number of levels that the subprogram has been run.
- Subprograms cannot be executed by writing M98 in MDI mode.
- Perpetual program runs are possible by using M99 command in a main program. When the RESET button is depressed, the program returns to the first address of memory. Refer to Fig. 2.7.4.2 (a).
- Programs which are frequently used can be stored and retrieved. Refer to Fig. 2.7.4.2 (b). When M codes for rewind (M02 or M30) are used, the program returns to the initial address. When M codes for reset are given, the following program is executed as a main program.

5 M98 command from punched tape

M98 may be given from punched tape to jump to a subprogram. In this case, all the subprograms to which jumps are to be made must be stored in the tape memory. After a jump has been made to a subprogram in the tape memory, the rules governing the subprograms as described above take effect. After executing the subprogram, the program returns to the block that follows the jump instruction.

NOTES:

- After the subprogram runs, the program returns to the punched tape and the return block instruction "P··· M99 *" cannot be used. P command is neglected.
- Simple jump command M99 cannot be given from a punched tape.
  If given, it is neglected.
- If an M code involving rewinding M code (M02 or M30) is given in a subprogram to which a jump is made, the tape will be rewound.
  Pay attention to this feature.

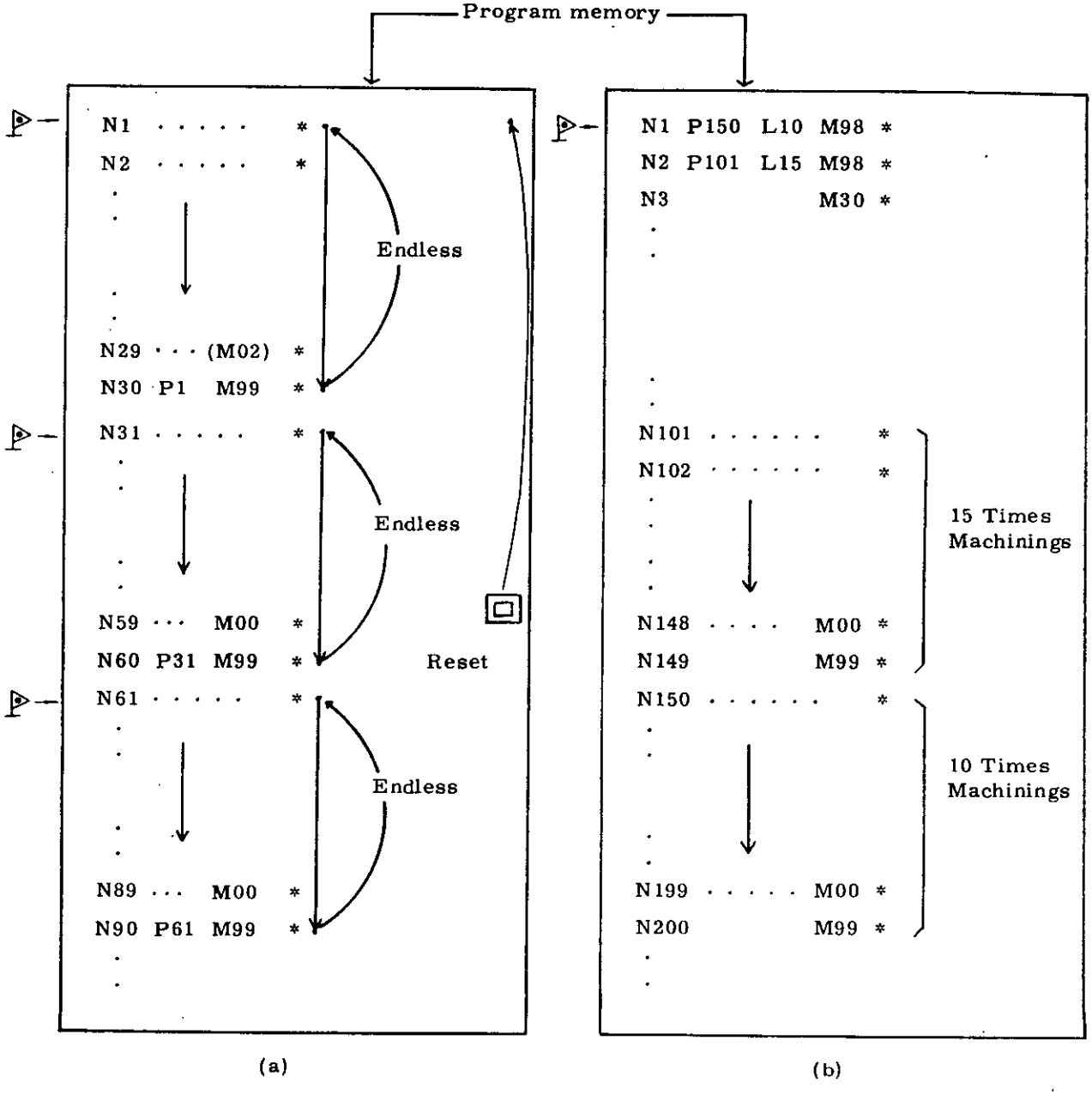

Program memory
N1 . . . . . *
N2 . . . . . *
N29 . . . (M02) *
N30 P1 M99 *
Endless
N31 . . . . . *
N59 . . . M00 *
N60 P31 M99 *
Endless
Reset
N61 . . . . . *
N89 . . . M00 *
N90 P61 M99 *
Endless
(a)
N1 P150 L10 M98 *
N2 P101 L15 M98 *
N3 M30 *
N101 . . . . . *
N102 . . . . . *
N148 . . . . M00 *
N149 M99 *
15 Times Machinings
N150 . . . . . *
N199 . . . . . M00 *
N200 M99 *
10 Times Machinings
(b)

Fig. 2.7.4.2

2.8 SPINDLE-SPEED FUNCTION(S-FUNCTION)

2.8.1 S 2-DIGIT COMMAND

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

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

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

EXAMPLE

```
G00 S11 M03 *
          ... S command
              Spindle CW
    X... Y... Z... *          S11: Effective
G01 Z... F... *
```

```
G00 X... Y... Z... M05 *
          ... Spindle stop
    ... M03 *                 S11: Effective
    X... Y... Z... *
```

```
G01 Z... F... *
    S22 *
    X... Y... F... *          S22: Effective
```

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

2.8.2 S 4-DIGIT† COMMAND

- Four digits following S (S□□□□) are used to specify the spindle speed in rpm.
- The S command becomes effective when the input signal of S command completion (SFIN) is turned on after given. For details of timing, etc., refer to APPENDIX 1 S 4-digit command.
- When S command is given in a block together with M03 (spindle forward running) or the M04 (reverse running), the control proceeds to the next block after the spindle speed reaches the speed given by the S code. For details, refer to the machine tool builder's manual.

EXAMPLE

S1000 M03 *

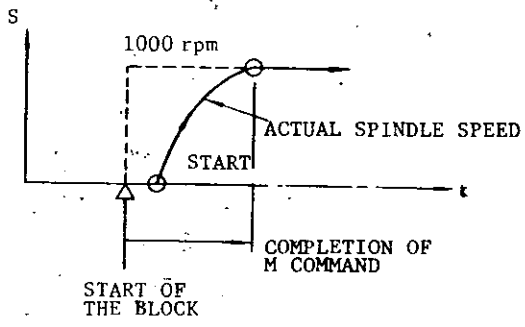


Fig. 2.8.2

- S commands are modal. Although the spindle stops at the M05 command, the S command is retained. Therefore, when M03 (or M04) is given, the spindle runs according to the S command.
- When S command is changed after the spindle start by M03 or M04, S command should be given within the range of spindle speed (High or Low) selected.

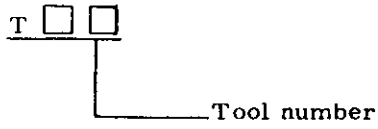
NOTES:

- The lower limit of the spindle speed depends on the spindle drive. If a speed lower than the manufacturer's specification is given, the machine will stop. Refer to the machine tool builder's manual for the low-speed limit. Negative S commands must not be programmed.
- When S and 4 digits are given for spindle speed, the output for the spindle drive is given in one of the two following ways.
  - 12 Bits binary non-contact output
  - Analog D/A converter output (±10 V Max.)

2.9 TOOL FUNCTION(T-FUNCTION)

2.9.1 T 2-DIGIT

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



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

When a move command and a T code are issued simultaneously,

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

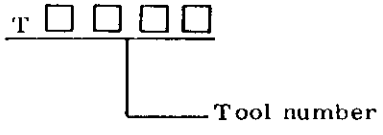
depending on the design of the machine.

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

- T codes are modal, and therefore, once they are given, they remain effective until another T command is given.
- T code commands are generally for making automatic tool changers (ATC) to select the tool number to be used next. Therefore, they can be given without regard to the G, H or D codes which are for offsetting for the length or radius of the tool currently in use.

2.9.2 T 4-DIGIT†

Four digits following the address T specifies the tool number.



- Leading zeros may be omitted.
- This tool code is the same as the T 2-digit codes, except for the increased number of digits.

2.10 TOOL OFFSET

2.10.1 OUTLINE OF TOOL OFFSET

Tool offset commands are in the following three groups.

- Tool position offset B (for compensation for tool length and work location)

  This offset is for compensating the difference of tool positions in X, Y, and Z (4th axis†) axes. This instruction is executed from the block in which G43, G44 are given together with H or D code. The instruction is cancelled by G49 command.

- Tool position offset A (for simple compensation for tool radius)

  This command is for compensating for tool radius or for offsetting the tool. It is effective in X, Y, and Z axes (4th axis†). It is executed only to the blocks in which G45 through G48 commands are given together with H or D codes.

- Tool radius compensation C† (for complicated tool radius compensation)

  This code is for tool radius compensation functions effective to any work contours. It is effective to X-Y plane (Y-Z†, or Z-X plane†). The command is executed the moment G41 or G42 is given together with an H or D code. The command is cancelled by G40 code. For details of these compensations, refer to 2.14 PREPARATORY FUNCTIONS (G-FUNCTIONS).

2.10.2 STORAGE OF TOOL OFFSET VALUES

For the three groups of offsets, all the necessary offset values must be stored in memory beforehand.

Up to 99 offset values can be stored in the tool offset memory.

Offset value storage: 99 max.

The setting range of offset values ia as follows.

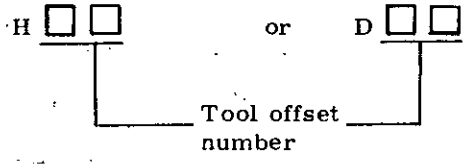
Table 2.10.2.1

	Range of offset values
Metric†	0 to ±999.999 mm
Inch†	0 to ±40.0000 inch
Degree†	0 to ±999.999 deg.

For the procedures of storing values into memory, refer to 4.3.6 Writing of Tool Offset Value.

2.10.3 H- AND D-FUNCTION (H, D CODES)

Two digits, following the address H or D, specify tool offset numbers.



The tool offset numbers 01 through 99 directly correspond to the 99 offset-value memory numbers. That is, when certain numbers are designated, the corresponding offset values stored in the offset memories will be used to offset the tools.

Tool offset numbers 00 (H00 or D00) have different meanings depending on the respective offset functions. For details, refer to the descriptions on the respective G functions.

NOTES:

- The addresses H and D used to designate tool offset numbers are equivalent in all respects. When H (or D) is given, the system interprets that D (or H) is also given.
- However, programmers may conveniently distinguish H from D, thus:

  H code ... For tool length compensation

  D code ... For tool radius compensation.

Furthermore, more conveniently for programming, numbers may also be distinguished as follows.

H01 through H50 ... For tool length compensation

H51 through D99 ... For tool radius compensation

Table 2.10.3

Offset method	G code	H or D code	Offset value memory
Tool position offset B	G43 G44 G49	H 01 D 99	No. / Offset value 01 02 03 04 . . . 96 97 98 99
Tool position offset A	G45 G46 G47 G48		
Tool dia. compensation C (Intersection computing system)	G40 G41 G42		

2.11 B-FUNCTION†

B-function and T 4-digit† commands cannot be used simultaneously.

Three digits following the address B give index table positions.

The actual index positions corresponding to the respective B codes depend on the machine tool builder. For this, refer to the specifications of the machine tool builder. When a B-function is given together with a move command in one block,

- the B command is executed simultaneously with the move command, or
- B command is executed after the execution of the move command,

depending on the design of the machine tool. For this refer to the specifications of the machine tool builder.

B codes are modal. When one B code is given, it remains effective until another B-command is given.

NOTES:

- B-function standard interface is in 3-digit BCD output. For output timing, answer back (FIN) signal, etc., refer to Appendix I.
- With MDI operation on NC panel, "4" is used to specify address for B codes. Therefore, when the control has B-function, the 4th axis control cannot be added.

2.12 SIMULTANEOUSLY CONTROLLABLE AXES

2.12.1 THREE-AXIS CONTROL

The following axes are simultaneously controllable.

Table 2.12.1

	Simultaneously controllable axes
Positioning G00	X, Y, and Z axes
Linear interpolation G01	X, Y, and Z axes
Circular interpolation G02, G03	Two axes, XY, YZ, or ZX (Note)
Circle cutting† G12, G13	Two axes, X and Y
Helix cutting† G14, G15	Circle in XY-plane and linear feed in Z-axis direction.
Manual control	One axis, X, Y, or Z

Note: Where only one axis is specified, and therefore, no circular interpolation plane is determined, the currently effective plane-designating G code (G17 through G19 *) is effective. Where the control is not equipped with optional plane-designating G code†, all X or Y single-axis commands are interpolated in the XY plane, and Z single-axis commands will create an alarm "15" condition. For details, refer to "Circular interpolation (G02, G03)."

2.12.2 FOUR-AXIS CONTROL†

The following axes are simultaneously controllable.

Table 2.12.2

	Simultaneously controllable axes
Positioning G00	X, Y, Z, and α axes
Linear interpolation G01	X, Y, Z, and α axes
Circular interpolation G02, G03	Two axes, XY, YZ, ZX, Xα, Yα, or Zα (Note)
Circular cutting† G12, G13	Two axes, X and Y
Helix cutting† G14, G15	Circle in XY-plane and linear feed in Z-axis direction.
Manual control	One axis, X, Y, Z, or α.

Notes:

- The axis α represents any one of axes A, B, C, U, V and W, selected as the 4th axis.
- When only one axis is specified and therefore no circular interpolation plane is determined, the currently effective plane-designating G code (G17 through G19†) is effective. The designation of α axis alone will create an alarm "15" condition. Where the control is not equipped with optional plane-designating G code†, all X or Y single-axis commands are interpolated in the XY plane, and Z single-axis commands will create an alarm "15" condition.

2.13 ADDITIONAL 4TH AXIS †

An additional 4th axis can be incorporated. In this manual, the 4th axis is referred to as α-axis, and represents any of the 6 axes, A, B, C, U, V and W.

2.13.1 ROTARY AXIS (A, B OR C AXIS)

The rotary axis is defined as follows.

Table 2.13.1

Rotary axis	Definition
A axis	Rotary axis parallel to X-axis
B axis	Rotary axis parallel to Y-axis
C axis	Rotary axis parallel to Z-axis

Note: In this manual, any one of the three axes, A, B and C, is referred to as β-axis.

The unit of output increment and input increment for β-axis is "deg." instead of "mm" used with linear axes. For the other respects, the treatments are the same as those in mm. (Metric system)

Even when inch system is selected by parameter, the values for the β-axis remains "deg." unit. The control does not convert β-axis coordinate commands. However, feedrate command F is converted. (Refer to 2.14.3 Linear Interpolation)

2.13.2 LINEAR AXIS (U, V OR W AXIS)

The linear axes are defined as follows.

Table 2.13.2.1

Linear axis	Definition
U-axis	Linear axis parallel to X-axis
V-axis	Linear axis parallel to Y-axis
W-axis	Linear axis parallel to Z-axis

Note: In this manual, linear axes either U, V or W are indicated by γ axis.

The unit output increment and input increment for γ-axis is the same as the other linear axes, X, Y and Z. No discrimination is necessary.

When inch system is selected by parameter, input values must be in inches for γ-axis.

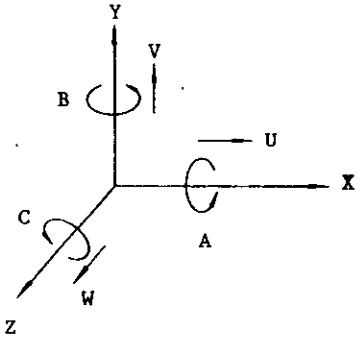


Fig. 2.13.1 4th Axis in Right-hand Coordinate System

Table 2.13.2.2

	Rotary axis (β)	Linear axis (γ)
Least output increment	0.001 deg/pulse	0.001 mm/pulse
Least input increment	0.01/0.001 deg.	0.01/0.001 mm
Maximum programmable value	±8388.607 deg.	±8388.607 mm
Rapid traverse rate	[ ] deg./min	[ ] deg./min
Cutting feedrate	[ ] deg./min	[ ] deg./min
Manual feedrate	[ ] deg./min	[ ] deg./min

2.13.3 4TH AXIS COMMAND ADDRESS

For the 4th axis address to be programmed in the NC tape, use one axis specified by the machine tool builder out of the A, B, C, U, V and W. However, the control makes no discrimination in reading the tape program, and accepts any of the addresses A, B, C, U, V and W.

When writing and editing programs by MDI operation, select "A" as the 4th address designation.

When punching out programs from memory, one of the addresses, specified by the machine tool builder, out of A, B, C, U, V and W, is used as the 4th axis address. For this, refer to the specifications of the machine tool builder.

2.14 PREPARATORY FUNCTIONS(G-FUNCTION)

2.14.1 LIST OF G CODES AND GROUPS

The preparatory functions are programmed with the address G and two digits. Table 2.14.1.3 shows the list of G codes. There are nine groups of G codes. An "initial state" which the control assumes at the time it is switched on or depressed RESET button is shown in Table 2.14.1.1

Table 2.14.1.1

Group	Initial state	When reset	Remarks
A	G00	(Note 2)	Basic function group
B	G17	No change	Plane designation
C	G40	G40	Tool radius compensation
D	G80	G80	Canned cycle
E	G90 (Note 1)	No change	Absolute/incremental
F	G94	No change	Feedrate per minute/per revolution
G	G98	No change	Canned cycle return point
Δ	Unspecified	Unspecified	Non-modal
ΔΔ	Unspecified	Unspecified	Modal

Notes:

1. G90 is the initial state when the parameter No. 78 is "0." When it is "1," the initial state is G91.
2. G00 is the initial state when the parameter No. 09 is "0." When it is "1," no change occurs.

G codes in groups A through G are modal. When a modal G code is once programmed, it remains effective until another G code in the same group is given.

G codes in the delta group (Δ) are non-modal. They are effective only in the block containing them. G codes in the double delta group (ΔΔ) are modal only for each specified axis.

Two or more G codes in groups A through G can be programmed in one block. However, when two or more G codes of the same group are programmed in one block, the last given G code has priority.

G codes in the delta group cannot be programmed in combination with other G codes in one block. They must be programmed in an separate block. (Exceptions to this rule are shown in Table 2.14.1.2.)

When an A-group G code is given in canned cycle mode (G73, G76, G81 through G89), the canned cycle is cancelled, and the D group code becomes G80.

When a reset is made during a tool radius compensation (G41, G42) or canned cycle execution, the G40 or G80 state is assumed.

In normal command data display, only one of G codes in the A or D group, and a part of the delta (Δ) group are displayed. When NUMBER is set to 99, all G code groups can be displayed. (Refer to 4.3.1 Display of Command Data.) In the EDT mode, all G codes given in the block are displayed, regardless of the NUMBER setting.

G43, G44 and G49 codes in the ΔΔ group and G45 through G48 in the Δ group can be given together with the following G codes in the A group, within the same block.

Table 2.14.1.2

		Programmable G code combinations
ΔΔ	G43, G44, G49	G00, G01
Δ	G45 to G48	G00, G01, G02, G03

Table 2.14.1.3 List of G codes

G Code	Group	Function	B: Basic O: Optional
G00#	A	Positioning	B
G01	A	Linear interpolation	B
G02	A	Circular interpolation, CW	B
G03	A	Circular interpolation, CCW	B
G04	Δ	Dwell	B
G12	Δ	Circle cutting, CW	O
G13	Δ	Circle cutting, CCW	O
G14	Δ	Helical cutting, CW	O
G15	Δ	Helical cutting, CCW	O
G17#	B	XY plane designation	O
G18	B	ZX plane designation	O
G19	B	YZ plane designation	O
G25	Δ	Program copy	O
G27	Δ	Reference zero check	O
G28	Δ	Automatic return to reference zero	O
G29	Δ	Return from reference zero	O
G40#	C	Tool radius compensation cancel	O
G41	C	Tool radius compensation, left	O
G42	C	Tool radius compensation, right	O
G43	ΔΔ	Tool position offset B, plus direction	B
G44	ΔΔ	Tool position offset B, minus direction	B
G45	Δ	Tool position offset A, extension	B
G46	Δ	Tool position offset A, retraction	B
G47	Δ	Tool position offset A, double extension	B
G48	Δ	Tool position offset A, double retraction	B
G49	ΔΔ	Tool position offset B, cancel	B
G73	D	Canned cycle #10	O
G76	D	Canned cycle #11	O
G80#	D	Canned cycle cancel	O
G81	D	Canned cycle #1	O
G82	D	Canned cycle #2	O
G83	D	Canned cycle #3	O
G84	D	Canned cycle #4	O
G85	D	Canned cycle #5	O
G86	D	Canned cycle #6	O
G87	D	Canned cycle #7	O
G88	D	Canned cycle #8	O
G89	D	Canned cycle #9	O
G90	E	Absolute command designation	B
G91	E	Incremental command designation	B
G92	Δ	Programming of absolute zero point	B
G94#	F	Feedrate per minute (mm/min) designation	O
G95	F	Feedrate per revolution (mm/rev.) designation	O
G98#	G	Return to initial level for canned cycles	O
G99	G	Return to R point level for canned cycles	O

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

2.14.2 POSITIONING (G00)

G00 X··· Y··· Z··· ($\alpha^{\dagger}$···) *

where α = A, B, C, U, V, or W

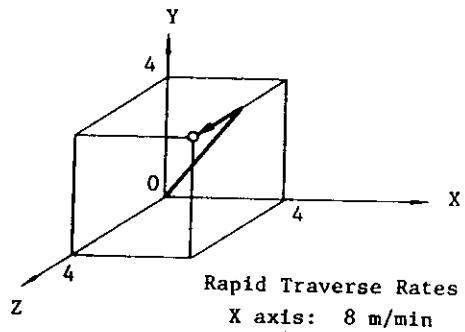
With this command, the tool is moved simultaneously in the three axial directions (four axial directions†) in the respective rapid traverse rates to the specified coordinate position. When a certain axis is missing in the command, the tool does not move in the axial direction of that axis.

Rapid traverse rates differ with each axis and are determined by the machine tool builder. Refer to the specifications of the machine tool builder.

The tool moves independently on each specified axis, and therefore, the resultant motion is generally not-linear. When programming a rapid traverse, take this into consideration to avoid any interference of the tool with the workpieces or other objects.

EXAMPLE

G00 X4000 Y4000 Z4000 *



Rapid Traverse Rates
X axis: 8 m/min
Y axis: 8 m/min
Z axis: 4 m/min

Fig. 2.14.2

2.14.3 LINEAR INTERPOLATION (G01)

G01 X··· Y··· Z··· ($\alpha^{\dagger}$···) F··· *

where α = A, B, C, U, V, or W

With this command, the tool is moved simultaneously in the three (four†) axial directions resulting in a linear motion. When a certain axis is missing in the command, the tool does not move in the axial direction of that axis.

Feedrate is specified by an F code. The feedrate in the component axial directions are so controlled that the resultant feedrate becomes the specified feedrate.

$$F = \sqrt{Fx^2 + Fy^2 + Fz^2 + F\alpha^2}$$

(where Fx, Fy··· are feedrates in the X, Y··· directions.)

The end point can be programmed either in absolute coordinates or in incremental values with G90 or G91 respectively. (Refer to 2.14.17 Absolute/Incremental Designation (G90, G91).)

If no F code is given in the block containing the G01 or in preceding blocks, the block constitutes an input error.

PROGRAM EXAMPLE

G01 X4000 Y4000 Z4000 F100 *

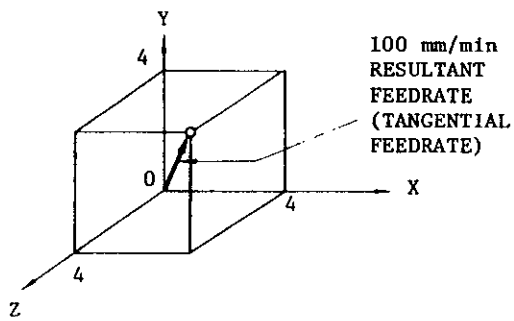


Fig. 2.14.3

Where the optional 4th axis is a rotary axis (A, B or C), for the same F code, the feedrates in the basic three axis directions (X, Y and Z), and the rotary axis feedrate are as follows.

Table 2.14.3

F function			In minimum F command unit	
			Feedrate in basic three axes (X, Y, Z)	Feedrate around rotary axes (A, B, C)
Metric system	Feedrate per min.	F4	1 mm/min.	1 deg./min.
		F4.1	0.1 mm/rev.	0.1 deg./min.
	Feedrate per rev.	F2.2	0.01 mm/rev.	0.01 deg./rev.
		F2.3	0.001 mm/rev.	0.001 deg/rev.
Inch system	Feedrate per min.	F3.1	0.1 inch/min.	2.54 deg./min.
		F3.2	0.01 inch/min.	0.254 deg./min.
	Feedrate per rev.	F1.3	0.001 inch/rev.	0.0254 deg./rev.
		F1.4	0.0001 inch/rev.	0.00254 deg./rev.

Note: When the 4th axis is a linear axis (U, V or W), its feedrate is equivalent to that of the basic three axes (X, Y and Z).

2.14.4 CIRCULAR INTERPOLATION (G02, G03)

The tool is moved along a circular path on the XY, ZX or YZ plane with the following commands. (Where a 4th axis is used†, in any of the planes containing that 4th axis.)

Table 2.14.4.1

G02 (G03)	X··· Y···	I··· J··· (or R···)	F···	*	XY plane
G02 (G03)	Z··· X···	K··· I··· (or R···)	F···	*	ZX plane
G02 (G03)	Y··· Z···	J··· K··· (or R···)	F···	*	YZ plane
Circular interpolation	Coordinate of end point	Coordinate of arc center with respect to start point (or radius value)	Feedrate	EOB	—

The direction of circular motion is specified as follows.

G02: CW (clockwise)
G03: CCW (counterclockwise)

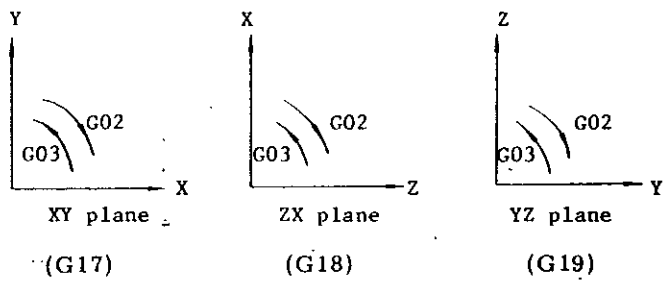


Fig. 2.14.4.1

The plane on circular interpolation is determined automatically by the combination of the address codes used X, Y or Z (or $\alpha^{\dagger}$) and I, J or K. Where no plane can be determined due to the omission of address codes or otherwise, the specifications of plane by means of G17, G18 or G19 becomes effective. In principle, when giving circular interpolation command (G02 or G03), the interpolation plane is designated by G17, G18 or G19 in advance.

G17: XY plane
G18: ZX plane
G19: YZ plane

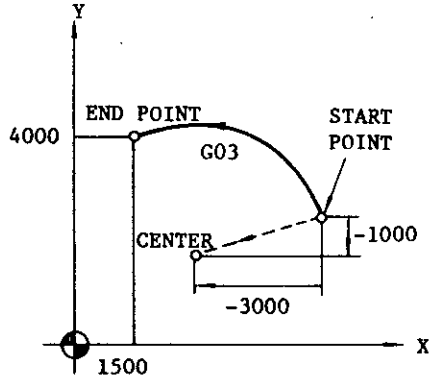
These G codes specify the tool radius compensation planes (G41/G42), in addition to the circular interpolation planes. Unless otherwise specified, XY plane (G17) is automatically selected when power is switched on.

The end point of the arc can be programmed in either absolute values or incremental values, with the code G90 or G91. However, the arc center is always programmed in incremental values with respect to the start point, irrespective of code G90 or G91.

EXAMPLES

G03 X1500 Y4000 I-3000 J-1000 F150 *

(a) Absolute designation (G90)



G03 X-4000 Y2000 I-3000 J-1000 F150 *

(b) Incremental designation (G91)

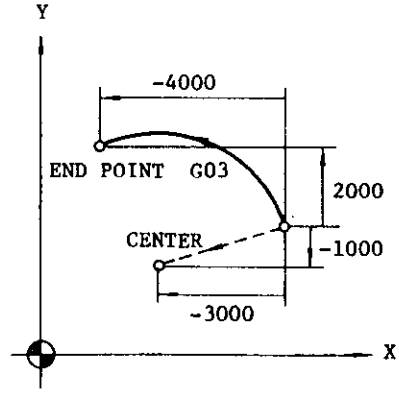


Fig. 2.14.4.2

Circular interpolation by radius

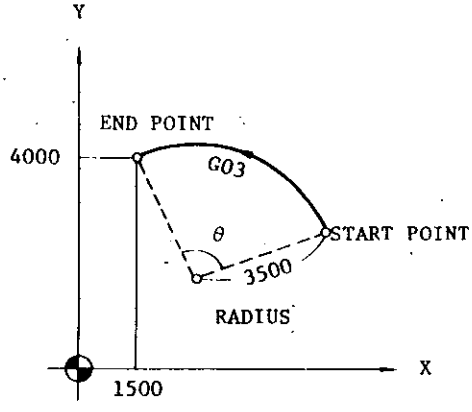
Instead of specifying the coordinates of the arc center by I, J or K, the radius of the arc may be directly specified with the address R. In this case, R is specified without a sign, and the center angle of the arc must be smaller than 180°.

EXAMPLES

G03 X1500 Y4000 R3500 F150 *

(a) Absolute designation (G90)

Center angle θ < 180°



G03 X-4000 Y2000 R3500 F150 *

(b) Incremental designation (G91)

Center angle θ < 180°

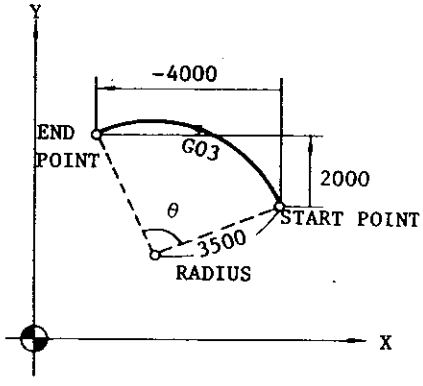


Fig. 2.14.4.3

Where an optional 4th axis is used, circular interpolation is possible in any of the planes including the 4th axis.

Table 2.14.4.2

G02 (G03)	X··· α···	I··· J··· (or R···)	F···	*	Xα plane†
G02 (G03)	Z··· α···	K··· I··· (or R···)	F···	*	Zα plane†
G02 (G03)	Y··· α···	J··· K··· (or R···)	F···	*	Yα plane†
Circular interpolation	Coordinate of end point	Coordinate of arc center with respect to start point (or radius value)	Feedrate	EOB	—

Where α = A, B, C, U, V or W

Circular arc on a plane, including the 4th axis, is defined on a plane on which the 4th axis constitutes the vertical axis, as shown below.

G02: CW (clockwise)
G03: CCW (counterclockwise)

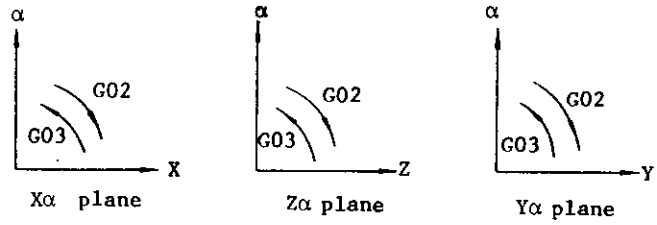
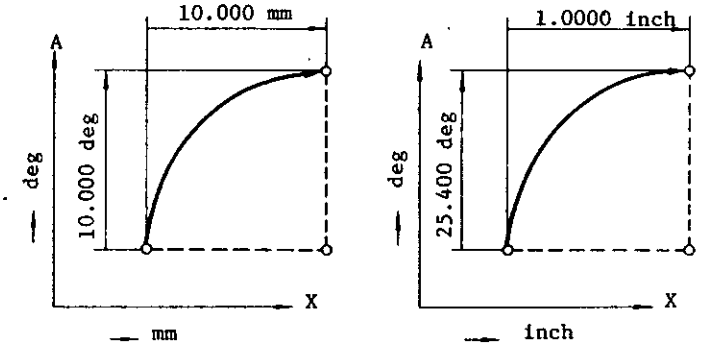


Fig. 2.14.4.4

Where the 4th axis is a rotary axis (A, B or C), the designation of degree values for the circular interpolation is different between the metric system and inch system, because in the inch system, all designations for linear axes are multiplied by 2.54 by the control but angular designation in degrees for rotary axis is not multiplied by 2.54.



G91 *

G02 X10000 A10000
I10000 F200 *

(a) Metric command

G91 *

G02 X10000 A25400
I10000 F200 *

(b) Inch command

Fig. 2.14.4.5

To be more specific, in the inch system, a rectangular coordinate system with 0.0001 inch (linear axis) and 0.00254 deg. (rotary axis) is assumed, and circular interpolation is effected as circular arcs on this plane. In this way, the correct relationship between the coordinate designations for the arc center and for the end point is maintained.

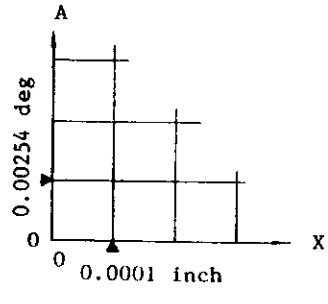


Fig. 2.14.4.6

NOTES:

- In a G02 (G03) block, when the plane specified by the address codes X, Y or Z (α†), and I, J or K is different from the plane designated by G17, G18 or G19, the designation by the address codes has priority.

  (X or I) and (Y or J): XY plane
  (Z or K) and (X or I): ZX plane
  (Y or J) and (Z or K): YZ plane
  X and α : Xα plane
  Z and α : Zα plane
  Y and α : Yα plane

- When circular interpolation involving a 4th axis is programmed, the arc end point coordinates cannot be omitted.

- Where only one axis is specified by the address codes, circular interpolation is made on the plane specified by G17, G18 or G19. If three or more axes are specified, this is interpreted as an input error.

EXAMPLE

G91 G17 G02 X200 I100 F50 *

⇩

G02 X200 Y0 I100 J0 F50 *

EXAMPLE

G17 G02 X100 Y200 Z300 I100 F50 *

⇩

Input error "0515"

- The feed designation by F codes represents velocities in the tangential direction of the arcs.

- With the YASNAC 3000G, a circular arc covering two or more quadrants can be programmed with one block. Commands for completely closed circles can be also programmed.

EXAMPLE

G00 X0 Y0 *

G02 X0 Y0 I1000 J0 F100 *
... One complete circle

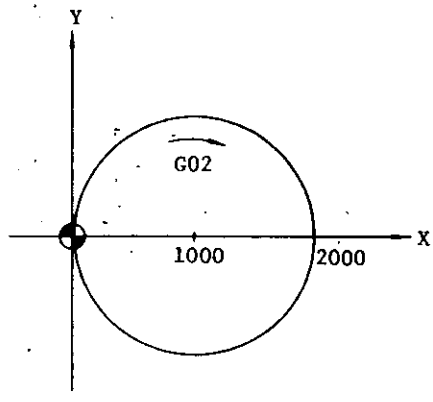
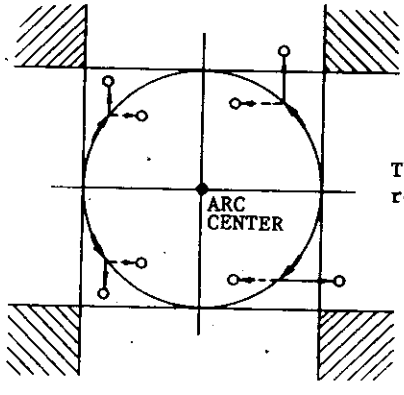


Fig. 2.14.4.7

- If the end point of an arc is not correctly specified and does not lie on the circle specified by radius, the tool moves along the path shown below.



The end point is represented by "°."

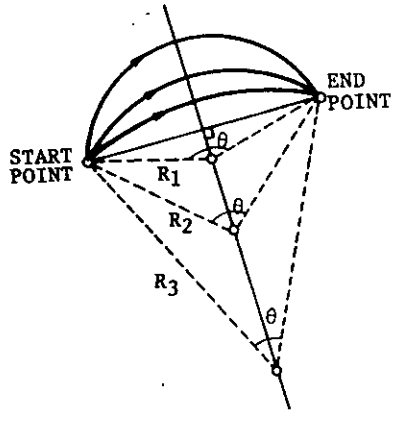
Fig. 2.14.4.8

If the end point is designated in the shaded area, no alarm condition is created, but the tool keeps on moving in circle endlessly. Especially when giving tool compensation, etc., the coordinates of the end point and the center must be accurate.

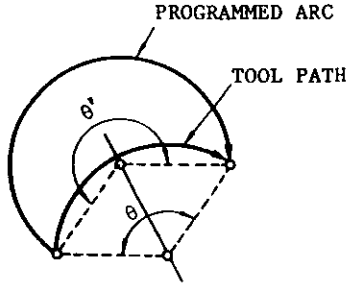
- When designating circular interpolation using the radius R command, the center angle cannot exceed 180°. If a radius shorter than half the distance between the start and end points is designated, this constitutes an input error.

$\theta < 180°$

$$R > \frac{\text{Distance between start and end points}}{2}$$



(a)



$\theta' > 180°$
$\theta < 180°$

(b)

Fig. 2.14.4.9

- If an arc with a center angle larger than 180° is given as a circular interpolation through radius R designation by mistakes, the tool path will be as shown in Fig. 2.14.4.9 (b) corresponding to the arc having the opposite center angle θ.

- With a circular interpolation by radius R designation, if one or both end point coordinates are omitted, the tool path will be defined on the plane designated by the plane designation codes.

- With all circular interpolation blocks, codes related to arc center such as I, J, K and R cannot be omitted. If no radius or arc center is specified, this is interpreted as a zero value.

2.14.5 DWELL (G04)

G04 P... *

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

Dwell is programmed as an independent block.

The maximum length of time which can be designated with address P is as follows.

Table 2.14.5

	Metric system		Inch system	
Least input increment	0.01 mm	0.001 mm	0.001"	0.0001"
Max. dwell time	8388.607 sec.		8388.607 sec.	

EXAMPLE

G04 P2500 *

Dwell time: 2.500 sec.

An exact-stop-check is made when the dwell time with address P is zero or is omitted.

EXAMPLE

G04 P0 * or G04 *

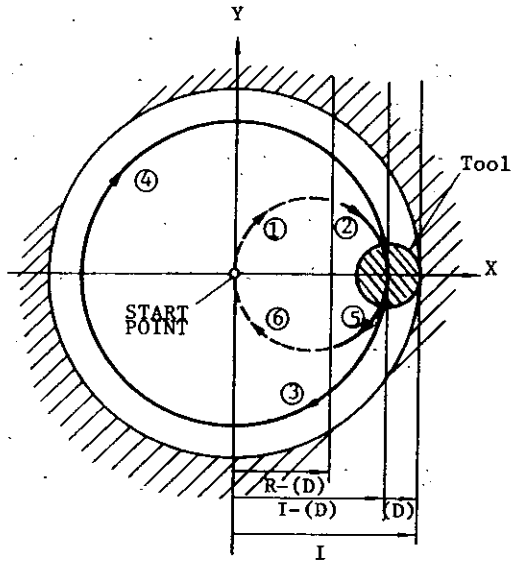
The exact-stop-check is the function of stopping the advance to the next block while discharging the servo-lag pulses. Corners are rounded to a certain extent in normal cutting feed because of smooth connection between blocks in normal cutting feed, but a sharp corner can be cut if an exact-stop-check or a dwell is used.

2.14.6 CIRCLE CUTTING (G12, G13)†

G12 I··· R··· D··· F··· *

(G13)

With this command, circle cutting is programmed.



Tool path

G12: ① → ② → ③ → ④ → ⑤ → ⑥

G13: ⑥ → ⑤ → ④ → ③ → ② → ①

(D) represents a set value of tool radius compensation.

G12: Clockwise (CW)

G13: Counterclockwise (CCW)

I: Raidus of finished circle (incremental value with sign)

R: Rapid traverse section (incremental value with sign)

D: Tool radius compensation No.

F: Cutting feed rate

Fig. 2.14.6.1

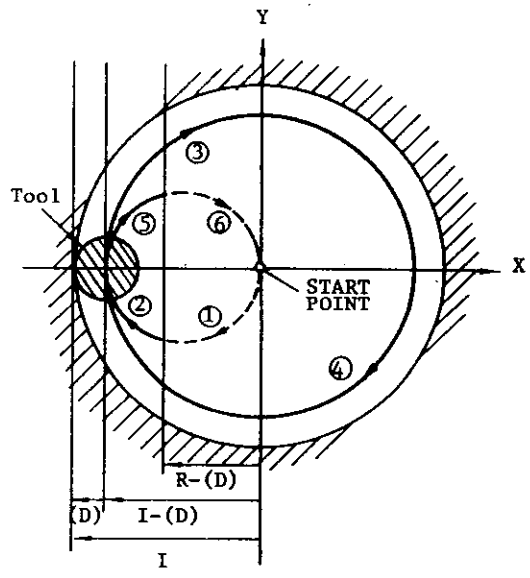
Circle cutting is performed on the XY plane, and a circle is cut with the tool moved from the center of the circle as indicated above.

In sections (①, ⑥) designated by R, the tool moves at a rapid traverse rate regardless of the F code. This is for reducing the air cut time, and the rapid traverse rate is set by a parameter according to the type of machine. (Parameter No. "66")

When R is zero or is omitted, the tool moves over the entire path at cutting feedrate designated by the F code.

In circle cutting (G12, G13), compensation for the tool radius is made regardless of the programming of G41, G42 (tool radius compensation). Program G12, G13 in G40 (tool radius compensation cancel) mode.

When the signs of I, R and (D) are properly considered, a circle can be cut in the negative direction of X axis. I, R and (D) are all negative in the example shown below. Cutting the Y direction is impossible.



Tool path

G12: ① → ② → ③ → ④ → ⑤ → ⑥

G13: ⑥ → ⑤ → ④ → ③ → ② → ①

(D) represents a set value of tool radius compensation.

G12: Clockwise (CW)

G13: Counterclockwise (CCW)

I: Radius of finished circle (incremental value with sign)

R: Rapid traverse section (incremental value with sign)

D: Tool radius compensation No.

F: Cutting feed rate

Fig. 2.14.6.2

There is the following restriction between radius I of the finished circle and the value of rapid feedrate section R. An input error occurs when this restriction is neglected. (→ "15")

$$|R| < |I|$$

An error also occurs when the signs of R and I are different.

In circle cutting (G12, G13), the tool radius compensation number (D) must be programmed. If omitted, D or H code programmed in the previous block becomes effective.

EXAMPLE

(a) G00 Z-4000 *
G12 I5000 R4000 D10 F300 *
G00 Z4000 *

D10 = 10.0 mm

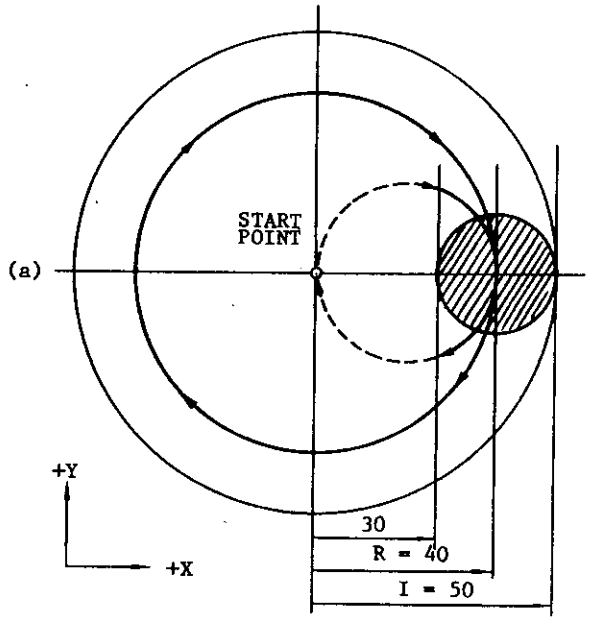


Fig. 2.14.6.3

(b) G00 Z-3000 *
G12 I-5000 R-4000 D15 F300 *
G00 Z3000 *

D15 = 8.0 mm

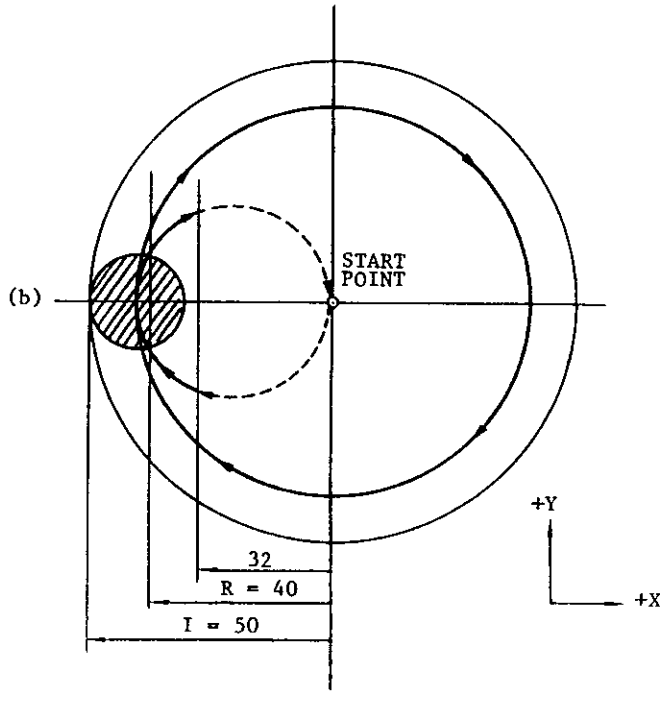


Fig. 2.14.6.4

2.14.7 HELICAL CUTTING (G14, G15)†

G14 I··· K··· L··· R··· D··· F··· *
(G15)

This command executes circular interpolation in the XY plane and linear interpolation in the Z direction simultaneously.

G14: Clockwise (CW)

G15: Counterclockwise (CCW)

I: Radius of circle (incremental value with sign)

K: Lead of spiral (incremental value with sign)

L: Number of spirals (integer)

R: Rapid traverse section (incremental value with sign)

D: Tool radius compensation No.

F: Cutting feedrate

where $K \leq 6.8 \left|I - (D)\right|$

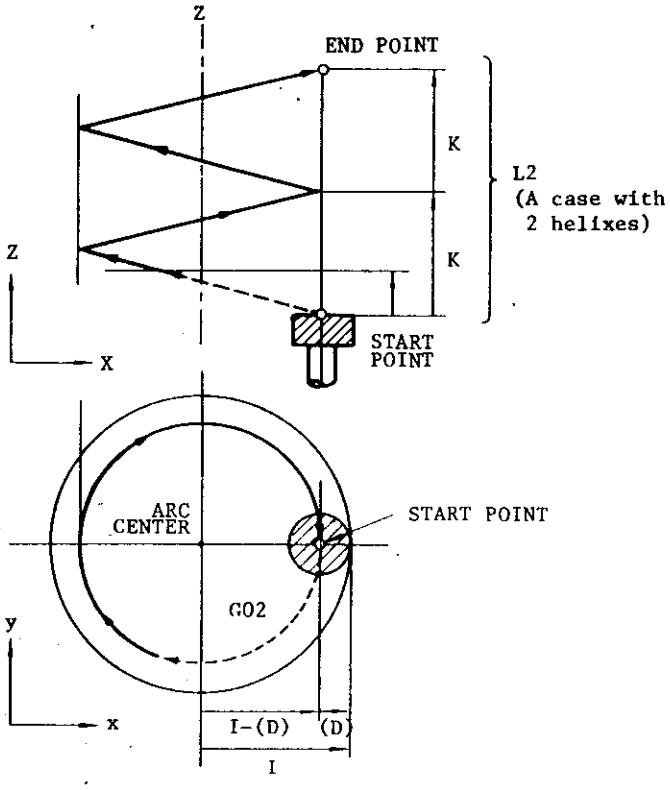


Fig. 2.14.7.1

Helical cutting is limited to circular motion on the XY plane and linear motion in the Z direction.

The direction of movement on the Z axis is determined by the sign of K. When K is positive, the tool moves in the direction of positive Z axis.

I designates the radius of the cutting circle from the center of the helical circle. The start point is limited to the point on the X axis to the right or the left of the center of the helical circle. Start cannot be made on Y axis.

L represents the number of helixes, and only an integer without sign should be programmed. If the desired number of turns of the helix is not an integer, make an adjustment by shifting the Z axis position of the start point.

In the section designated by R, the tool moves at rapid traverse regardless of the F code. This is for reducing the air cut time, and the speed is set by a parameter according to the type of machine. (Parameter No. "66")

When R is zero or is omitted, the tool moves over the entire path in cutting feedrate designated by F code. The feedrate designated by an F code represents the speed of circular arc compensation on the XY plane, and the tool moves in synchronization with it in the direction.

In helical cutting (G14, G15), tool radius compensation is made regardless of designation of G41, G42. Program G14, G15 in G40 (tool radius compensation cancel) mode.

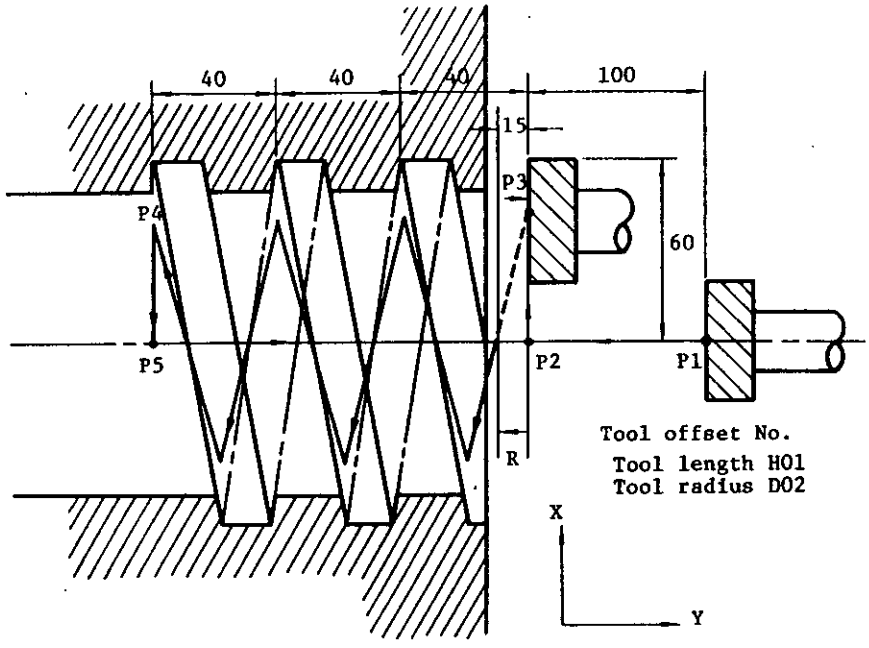
There is the following restriction between spiral lead K and rapid traverse section R. Exceeding of the restriction causes an input error.

$$|R| < |K|$$

An error also occurs when the signs of R and K are different.

In the block containing G14 (G15), the tool radius compensation number (D) must be programmed. If omitted, D or H code programmed in the previous block becomes effective.

EXAMPLE (Helical cutting)



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

Example of program (in the case of incremental command)

N123 G00 G43 Z-10000 H01 *	P1 → P2
N124 G46 X60000 D02 *	P2 → P3
N125 G15 I6000 K-4000 L3 R-1500 F150 *	P3 → P4
N126 G00 G46 X-6000 *	P4 → P5
N127 G49 Z22000 *	P5 → P1

Note: Helical cutting starts from P3, and the tool moves at rapid traverse rate in the air cut section designated by R.

Fig. 2.14.7.2

2.14.8 PLANE DESIGNATION (G17, G18, G19)†

The plane for making circular interpolation and tool radius compensation is designated by G codes G17/G18/G19.

G17: XY plane
G18: ZX plane
G19: YZ plane

However, for the plane for making circular interpolation, address characters X, Y, Z, (α)†, I, J and K take priority over the designation made by G17, G18 and G19. The latter designation is valid only when the interpolation plane is not determined due to omission of address characters or the like.

The axes for performing circle cutting, helical cutting and canned cycles are designated independently those of stated above.

The move command in each axis can be programmed regardless of the plane designation by G17/G18/G19.

For example, if

G17 Z··· *

is designated, motion is on Z axis.

The plane for making tool radius compensation by command G41 or G42 is univocally determined by G17, G18 or G19. It is not possible to designate compensation plane including the fourth axis.

The XY plane (G17) is selected when the power is turned on.

2.14.9 PROGRAM COPY (G25)

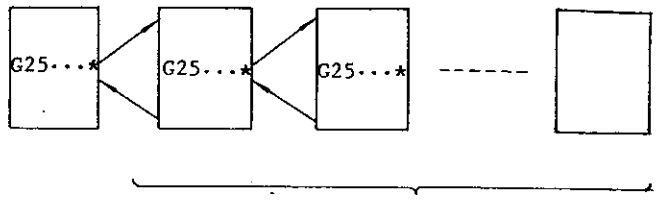
G25 P·^{p}· Q·^{q}· L·$^{\ell}$· *

This command permits the programs with sequence numbers p through q to be repeatedly executed ℓ times.

G25: Program copy
P: Start sequence number
Q: End sequence number
L: Number of times to be repeated (runs once if omitted)

G25 can be programmed from tape or memory, but it is necessary to store the program to be copied in memory. Exercise care so that the sequence numbers do not overlap.

G25 may be programmed in the program to be copied. This is called multiple program copy and such copy is possible up to eighth level.

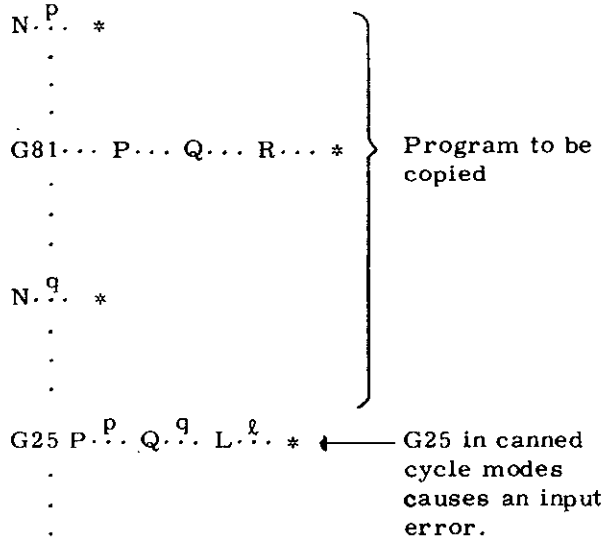


Max. 8th level

Fig. 2.14.9.1

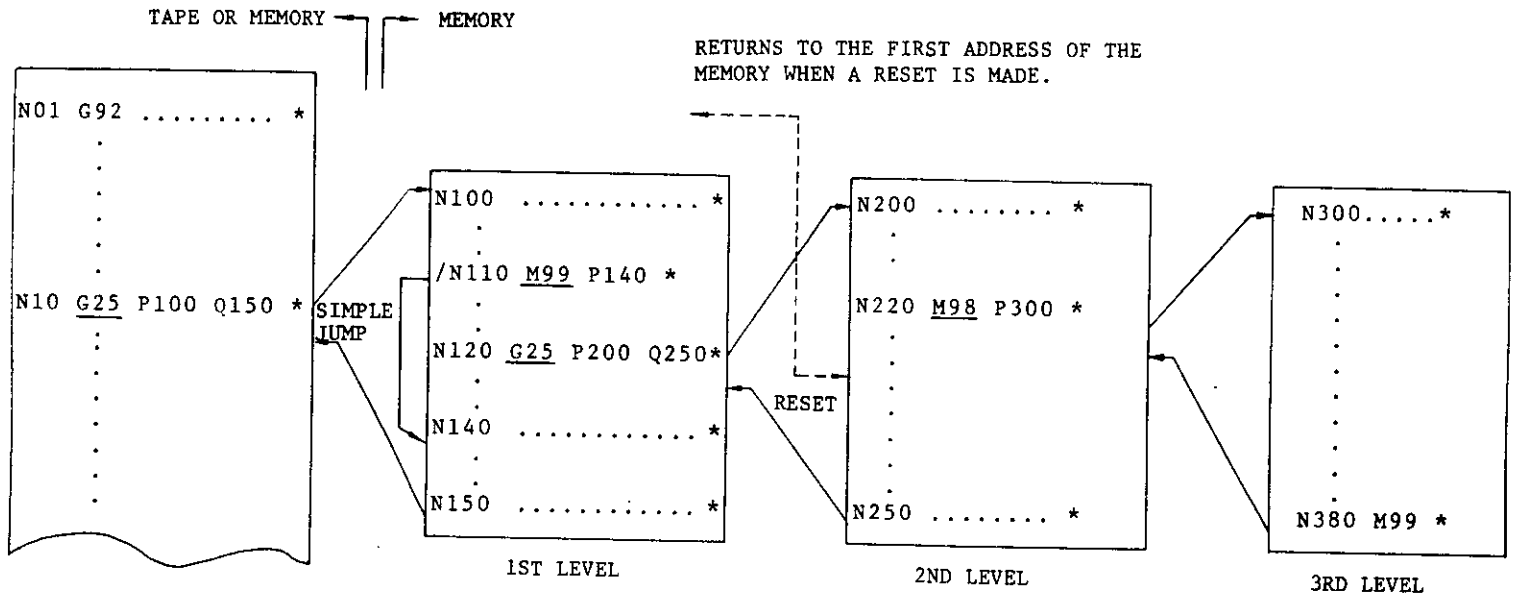
However, it is not possible to program G25 in the last block of the program to be copied. An input error occurs in this case.

Program copy cannot be programmed in canned cycle modes (G73, G76, G81 to G89). If it is programmed, an input error occurs because P, Q values of the canned cycles are changed. However, canned cycles can be programmed in the program to be copied.



NOTES:

- It is possible to program a simple jump command M99 in the program to be copied. That is, it becomes the function of M99 in the main program. (See program example.)
- The precautions for a G25 are the same as those of M98. Refer to "NOTE" in 2.7.4 Subprogram Function.



Notes:

1. M98 can be used in the program copied by G25.
   Even when G25 and M98 are used together, eight levels cannot be extended.
2. When a jump to another level is made by M99, an endless loop program is created and escape from this loop becomes impossible.
   Keep this in mind.

Fig. 2.14.9.2 Example of Program Copy

2.14.10 REFERENCE ZERO CHECK (G27)†

This function checks if the tool which has been programmed to start at the reference zero of the machine and return to the reference zero point accurately.

G27 X··· Y··· Z··· (α†···) *

When this command is given, the tool moves at rapid traverse rate along the three axes (four axes†) simultaneously to the programmed point, and a check is made if this position is reference zero point. However, positioning and check are not executed for an axis for which a coordinate command was omitted.

ZERO POSITION lamps light up if this position matches reference zero point. Automatic operation continues if maching is made with all of the designated axes. If there is any axis with which matching is not made, a reference zero error (alarm code "24") occurs, and automatic operation is interrupted. (CYCLE START lamp goes out.)

If G27 is given during tool radius compensation, matching cannot be made because the tool is positioned to the point offset by the compensation value. Use G27 with tool radius compensation cancelled.

Reference zero point is a fixed point of the machine and the return to this point can be made by "manual return to reference zero" or "G28 automatic return to return to reference zero." See 6.2.1 Manual Return to Reference Zero.

The mirror image is effective to the direction of movement by a G27 command. Program G27 in M94 (mirror image off) mode to avoid a mismatch error.

2.14.11 AUTOMATIC RETURN TO REFERENCE ZERO (G28)†

G28 X··· Y··· Z··· (α†···) *

With this command, the tool can be programmed to return to reference zero. The tool moves to the programmed position at rapid traverse rate, and then makes "return to reference zero" automatically.

Tool can be programmed to move simultaneously in three axes (4 axes†). However, the tool will not move in the direction for which a coordinate instruction is omitted.

The position of the tool commencing a return to reference zero should be within a region in which return to reference zero is possible (before deceleration LS). (Alarm code "24")
It is possible to program the tool to move within a region in which the return to reference zero is possible by the preceding programming of positioning command.

EXAMPLE

G28 Y··· Z··· * (In the case of absolute input)

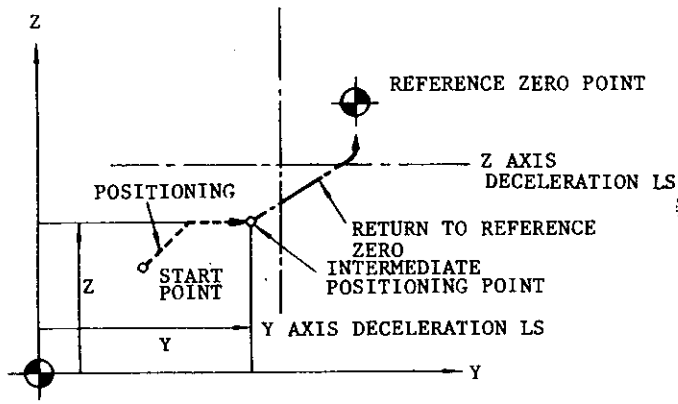


Fig. 2.14.11

"Return to reference zero motion" means the sequence of motions equal to that from commencement of movement of the manual return to reference zero to the return to reference zero point.

NOTES:

- An input error occurs when G28 is given in tool radius compensation mode (G41, G42) or canned cycle mode. (→ "0215")
- An input error occurs when G28 is given in mirror image (M95). (→ "1215")
- Tool offset is not cancelled by G28 command. But G28 should be given after cancelling tool offset as a rule.

When the content of parameter No. 86 is "1," the control can performs "rapid return to reference zero" instead of "automatic return to reference zero" described above. The operation is as follows:

- After positioning to intermediate positioning point B, direct positioning to reference zero is made by rapid traverse. Return to reference zero is faster than with the normal "automatic return to reference zero" using the deceleration limit switches of each axis.
- Rapid return to reference zero is possible even when point B is outside the region in which return to reference zero is possible.
- However, rapid return to reference zero is possible only after normal return to reference zero has been completed with all the axes by manual return to reference zero or by a G28 command after power switching on.
- Rapid return to reference zero is possible only with a G28 command, and it does not affect manual return to reference zero.
- Where the control is provided with the fourth axis, rapid return to reference zero is made if normal return to reference zero of X, Y and Z axes has already been completed, unless the fourth axis is designated in the G28 command. If the fourth axis is programmed, normal return to reference zero is executed unless normal return to reference zero in all four axes has been completed.

2.14.12 RETURN FROM REFERENCE ZERO (G29)†

This code is used to return the tool to its original position after return to reference zero by automatic return to reference zero, along the same path.

G28 Y··· Z··· ; (Point B) — Point A → B → C (Reference zero point)

G29 Y··· Z··· ; (Point D) — Point C → B → D

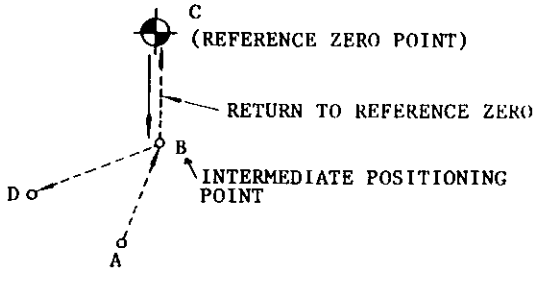


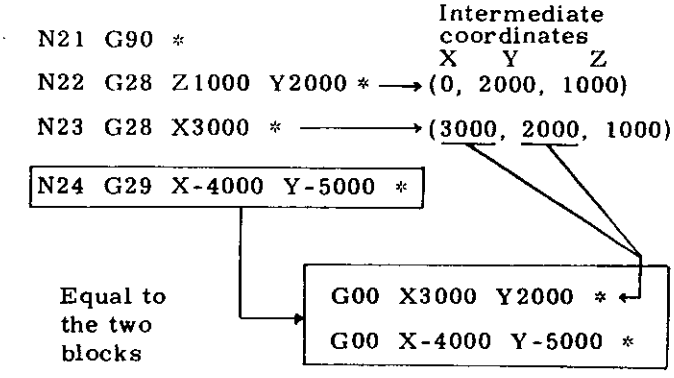
Fig. 2.14.12.1

When G29 is programmed, it is not necessary to consider the distance between point B and C in the program. Particularly when an incremental instruction is used, this is effective for returning tool to the original position, after returning to reference zero.

Movement of C → B and of B → D is made at rapid traverse rate simultaneously along three axes (simultaneously four axes†) by G29. However, in an axis for which a coordinate instruction was omitted, the tool will not move.

If G28 is programmed a number of times, the final coordinates of point B which last G28 creates is effective for the move of G29.

EXAMPLE 1 (In the case of absolute input)



EXAMPLE 2

```
N31 G91 ;
N32 G28 Z··· ;
N33 G28 X2000 Y4000 ;
N34 M06 ;
N35 G29 X4000 Y-4000 ;
 .
 .
 .
```

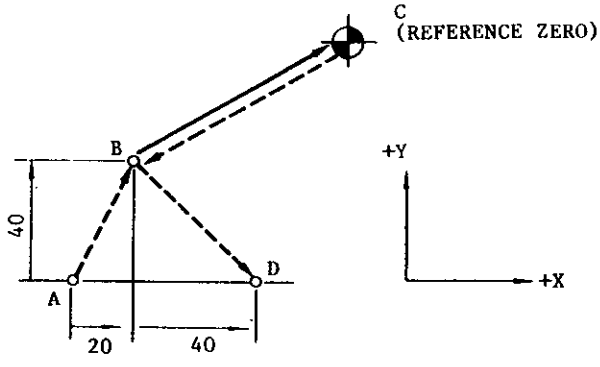


Fig. 2.14.12.2

NOTES:-

- An input error occurs if G29 is programmed in tool radius compensation mode (G41, G42) or during canned cycle mode (G73, G76 G81 to G89). (→ "0215")
- An input error occurs if G29 is given without execution of G28 after the control is turned on (→ "1215")
- In principle, cancel tool offset before programming G28 or G29. If they are programmed when offset is also effective, intermediate positioning point B will also be offset, and the tool passes point B'.

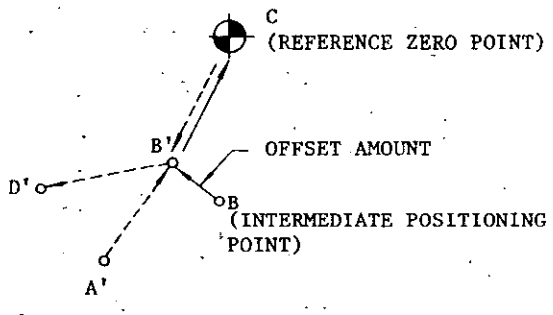


Fig. 2.14.12.3

- An input error occurs if G29 is given during mirror image (M95). (→ "1215")
- The following command or operation must not be taken because intermediate positioning point B of G28 does not meet with that of G29.

(1) The following operations are made between G28 and G29 commands.

- Setup of absolute zero (G92, ORG key)
- Machine lock
- Manual operation at Manual Absolute Off

(2) G28 and G29 are commanded in the blocks following the block containing M94 which cancels mirror image at the different point from the starting point of mirror image.

(3) G28 and G29 are commanded after manual operation at Manual Absolute Off.

2.14.13 TOOL RADIUS COMPENSATION C (G40, G41, G42)†

It is possible to specify the radius of the tool and to cause automatic tool path offset by this value. Store the offset value (tool radius value) in the offset value memory in advance by MDI, and program the tool offset number correspond to the tool radius value by a D code in the program.

1. Designation of compensation direction and of D code

Tool radius compensation is programmed with G41, G42 and is cancelled by G40. G41 and G42 indicate the directions of tool offset with respect to the direction of movement.

G code of tool radius compensation

G code	Group	Meaning
G40	C	Cancellation of tool radius compensation
G41	C	Tool radius compensation, left
G42	C	Tool radius compensation, right

Note: When the power is turned on, G40 is effective.

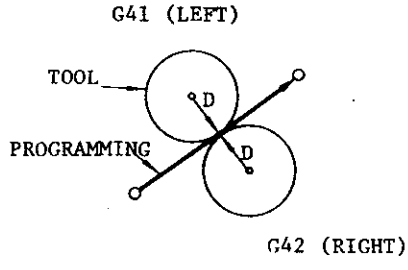


Fig. 2.14.13.1

Note that the directions of compensation (right, left) indicated above are reversed when the sign of the tool radius value in the offset memory designated by a D code is negative. Make sure to designate a D code in the block containing G41, G42 or in a preceding block. If D00 is commanded, tool radius will be regarded as "0."

Switching between G41 and G42 can be made in the compensation mode. Details will be given in item 5 below.

2. Designation of compensating plane

The plane in which tool radius compensation is made is designated by G17, G18, G19. They are G codes of B group. The XY plane (G17) is in effect at the time power is turned on.

G codes for designation of planes

G code	Group	Meaning
G17	B	XY plane
G18	B	ZX plane
G19	B	YZ plane

Note: When the power is turned on, G17 is effective.

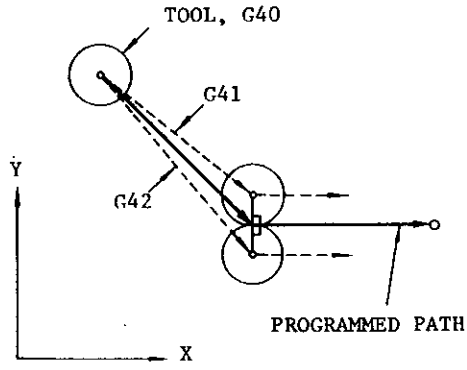
Make sure to designate a G code for plane designation in the same block as that of G41, G42 or in a preceding block. Plane designation cannot be made in a compensation mode. It is not possible to apply tool radius compensation in a plane including the fourth axis[†].

3. Method of entry into compensation mode

When G41 (G42) is programmed, the tool moves to an offset position with the distance equal to the radius. The offset position is on the normal line at the start point of the block immediately after G41 (G42). If no coordinate instruction is programmed in the block of G41 (G42), movement is made by the offset value only. Because G41 (G42) accompanies a movement, it is necessary to program G00 or G01 for a G code in group A. An input error (alarm code "0615") occurs if a G code other than G00, G01 is programmed.

EXAMPLE (A)

(a) G17 G01 F··· *
G41 (G42) D··· X··· Y··· *
X··· *



(b) G17 G01 F··· *
G41 (G42) D··· X··· Y··· *
G02 X··· Y··· J··· *

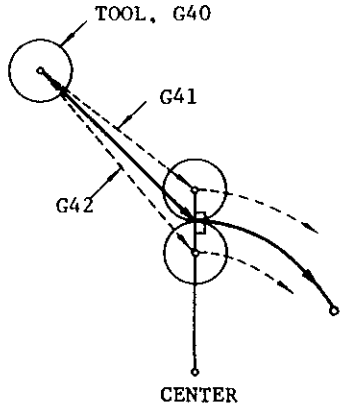
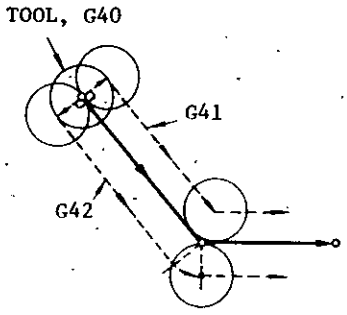


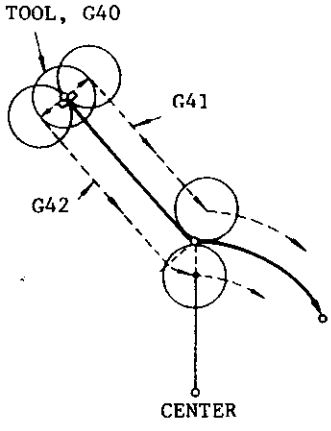
Fig. 2.14.13.2

EXAMPLE (B)

(c) G17 G01 F··· *
G41 (G42) D··· *
X··· Y··· *
X··· *



(d) G17 G01 F··· *
G41 (G42) D··· *
X··· Y··· *
G02 X··· Y··· J··· *



(e) G17 G01 F··· *
G41 (G42) D··· *
G02 X··· Y··· J··· *

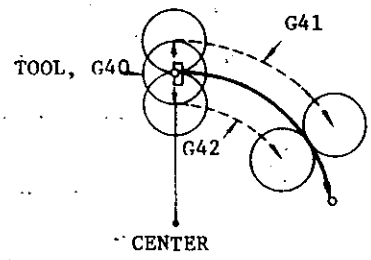


Fig. 2.14.13.3

Pay attention to the fact that offset is made on the normal line to the program line determined by the block after G41 (G42) at the start point in all of the examples (a) to (e) above. When the movement on the compensation plane is not programmed in the block after G41 (G42), the next one block is read ahead and the compensation start with the block. Input error occurs if move commands on the compensation plane are not programmed in more than two blocks. (→ "0615")

When compensation entry is programmed in the G00 mode, positioning movement is made independently by each axis to the offset point. Take care not to make the tool interfere with the work.

4. Movement in compensation mode

When after the tool radius compensation is programmed by G41, G42, the tool moves along the offset path until the instruction G40 is given. As calculation of the path is automatically made by the control, designate only the shape of the workpiece in the program. The tool path is controlled as follows depending on the angle between blocks.

a. Inside corner (180° or less):
Intersection computing type

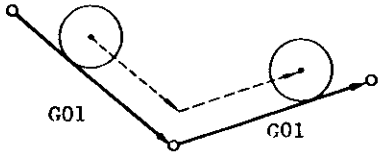


Fig. 2.14.13.4

b. Outside corner (over 180°):
Circular path type (in the case of M96)

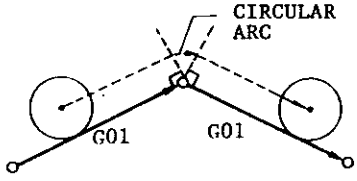


Fig. 2.14.13.5

In this case, movement of circular path is included in the former block.

Code M97 can be used to machine the outside corner by the intersection calculation, depending on the work. Refer to 2.7.2 M codes for Other Internal Processes† for details.

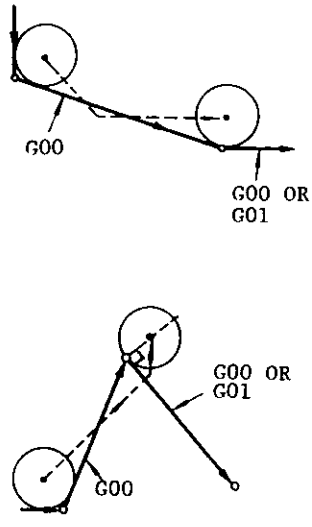
M96 ··· Tool radius compensation circular path ON

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

Normally, M96 is used for this operation, however, when there is a possiblity of an "overcut" in cutting special shapes with the M96, M97 should be used.

· Movement in G00 mode

The instruction G00 positions tools independently along each axis toward the final offset position. Care should be taken on the cutter path.



(In M96 mode)

Fig. 2.14.13.6

· Shape requiring care

Do not program a wedge shape having an accute angle.

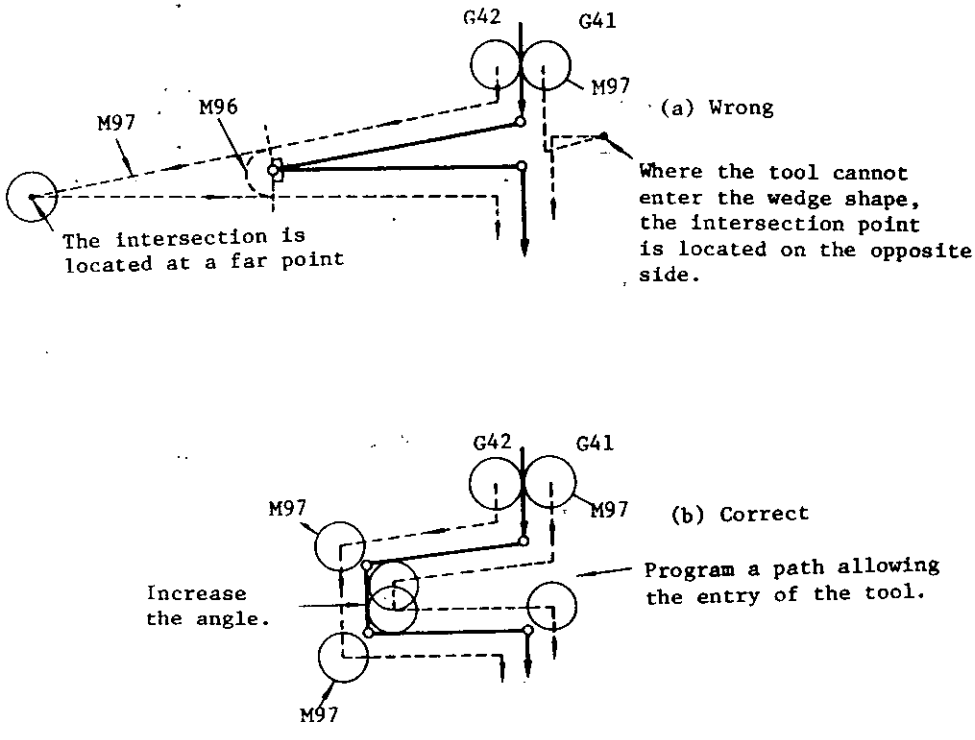


Fig. 2.14.13.7

5. Command involving no movement in compensation mode

· The control normally reads two blocks ahead during tool radius compensation mode and calculates the tool path. If either of these blocks give no coordinate instructions such as G04 (dwell), the control reads the block further ahead and makes calculations. When coordinate instructions are missing in two or more blocks, tool radius compensation becomes impossible and accurate tool path cannot be obtained. Therefore, in a program where G41 or G42 is used, ensure that, after them, two or more blocks without movement command in the compensation plane will not follow.

EXAMPLE

```
G17 G01 G41 X··· Y··· D··· F··· *
X··· Y··· *
  .
  .
  .
X··· Y··· *
G04 P1000 *  <-------- Blocks without movement
X··· Y··· *            in compensation plane.
  .
  .
  .
  .
  .
  .
  .
X··· Y··· *
Z··· *
X··· Y··· *
  .
  .
  .
X··· Y··· *
G40 X··· Y··· *
```

If no movement instruction is programmed in two continuous blocks, offset in the block immediate before them is made on the normal line at the end point. Where movement in the the compensation plane cannot be programmed in two or more continuous blocks for retracting in the third axis or the like, and offsetting on the normal line is not satisfactory, a dummy block can be inserted by I, J or K.

EXAMPLE

```
N001 G17 G01 G41 X··· Y··· D··· F··· *
N002 X··· Y··· *
        .
        .                    } XY plane
        .
N010 X··· Y··· *
N011 I··· J··· *  · · ·  Dummy block
N012 Z··· *
        .
        .                    } Z axis
        .
N019 Z··· *
N020 X··· Y··· *
        .
        .                    } XY plane
        .
N029 X··· Y··· *
N030 G40 X··· Y··· *
```

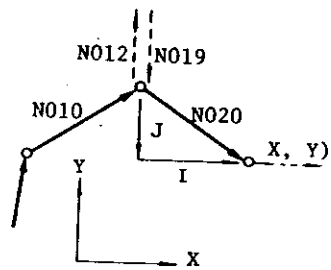


Fig. 2.14.13.8

The dummy block is not programmed for actual movement but it only provides data required for tool radius compensation calculation. In the example indicated above, an instruction that is the same as the first block (N020) of restarted movement of the XY plane after movement of Z axis is programmed as a dummy by I, J. I, J, K are used as the addresses of this dummy instruction, and they correspond to X, Y, Z axes respectively. Suitably use them in accordance with the plane designation.

I: Dummy for X axis command
J: Dummy for Y axis command
K: Dummy for Z axis command

} Programmed in incremental values

If X··· Y··· of N020 is in absolute values in the above example, give an instruction by converting into incremental values.

Note: Make a dummy block as follows if the object of the dummy block is circular interpolation.

EXAMPLE

N050 G01 X··· Y··· *

N051 G01 I(b) J(-a) * . . . Dummy block

N052 Z··· *
N059 Z··· * } Z axis

N060 G03 X··· Y··· I(a) J(b) * . . . Circular interpolation

N061 G01 X··· Y··· *

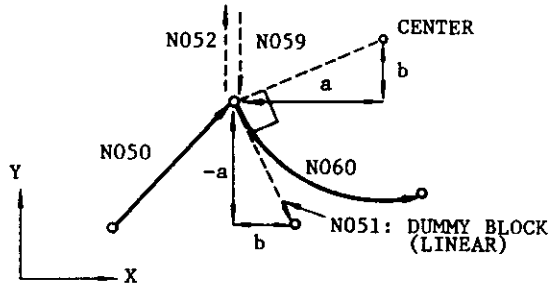


Fig. 2.14.13.9

That is, insert a linear dummy block that gives the tangential direction at the start point of the circular interpolation program block as shown above. Exercise care with the sign of the dummy block data depending on the shape of the circle. The tool stops at point A by the dummy block in preparation for the next circular command.

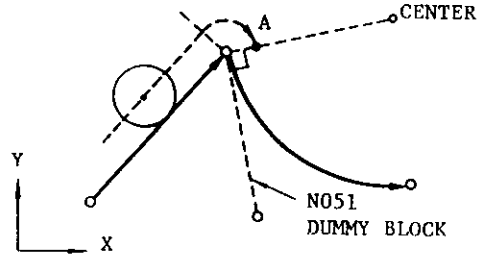


Fig. 2.14.13.10

· Switching between G41 and G42 in compensation mode

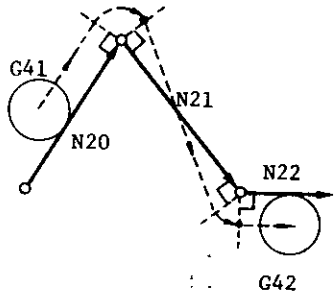
In this compensation mode, direct switching between G41 and G42 is possible without making cancellation with G40.

EXAMPLE

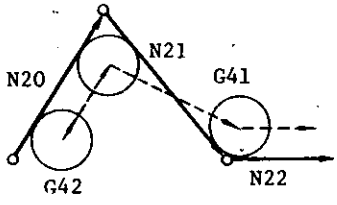
N10 G17 G01 F··· *
N11 G41 (G42) D··· *
.
.
.
N20 G01 X··· Y··· F··· *

N21 G42(G41) X··· Y··· * — Block of switching

N22 X··· *



(a) G41 → G42
(M96 mode)



(b) G42 → G41

Fig. 2.14.13.11

Note: The movement idential to what is shown above is also obtained when block N21 is split into two blocks as follows.

```
G42 (G41) *
     X··· Y··· *
```

· Change of tool radius value in compensation mode

New D code cannot be effective in a compensation mode. Compensation is made with the initially programmed D code, and it remains effective until it is cancelled by G40.

However, it is possible to change the tool radius value by changing the contents of the offset memory of the initially designated D code. The changing is made by MDI. The change becomes operative from the block read ahead into the buffer register after making the change.

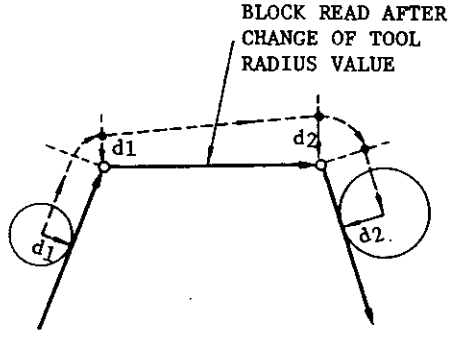


Fig. 2.14.13.12

· Method of cancellation of compensation

G40 is the command for cancelling tool radius compensation and for positioning or feeding the tool to just programmed end point. In this case, the tool moves to a point on the normal line at the end point of the block immediately before the block containing G40. Therefore, no portion will be left unmachined even when a cancellation with sharp angle is programmed. Because G40 accompanies cancelling movement, program it in the G00 or G01 mode like G41, G42. An input error occurs if group A other than G00, G01 is used.

EXAMPLE A

(a) G41 (G42)

```
    .
    .
    .
G01 X···  F···  *
G40 X···  F···  *
```

(b) G41 (G42)

```
    .
    .
    .
G02 X···  Y···  I···  J···  *
G01 G40 X···  Y···  *
```

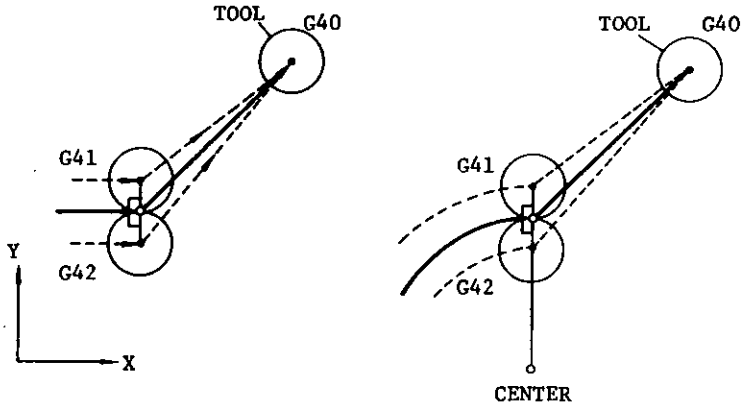


Fig. 2.14.13.13

EXAMPLE B

(c) G41 (G42)

```
    .
    .
    .
G01 X···  F···  *
    X···  Y···  *
G40 *
```

(d) G41 (G42)

```
    .
    .
    .
G02 X···  Y···  I···  J···  *
G01 X···  Y···  *
G40 *
```

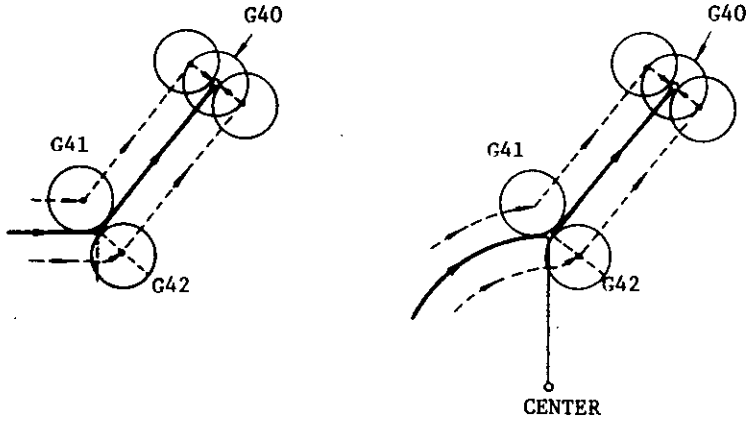


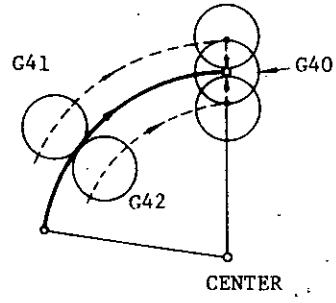
Fig. 2.14.13.14

(e) G41 (G42)

.
.
.

G02 X··· Y··· I··· J··· *

G01 G40 *



(e)

Fig. 2.14.13.14 (Continued)

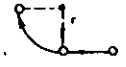
In all cases (a) through (e) described above, the tool reaches the programmed end point via the offset position on the normal line at the end point of the block immediately before G40.

· Cautions and remarks in tool radius compensation

A. Restriction of maximum programmable value

The following restrictions are imposed on the coordinate command value during tool radius compensation.

Table 2.14.13.1

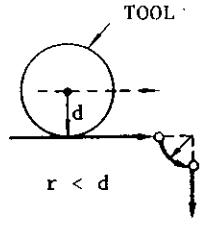
Condition		Restriction
Length of linear interpolation		$\ell \leqq \pm 5931.64\ (0)$ mm
Junction between linear line and circular arc		$r \leqq \pm 8388.60\ (7)$ mm
Junction between circular arcs		$r \leqq 2965.81\ (8)$ mm I, J, K $\leqq 2097.15\ (0)$ mm

B. Programmed shapes that produce input errors

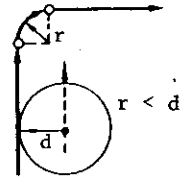
Input errors occur with the following programmed shapes. (→ "15")

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

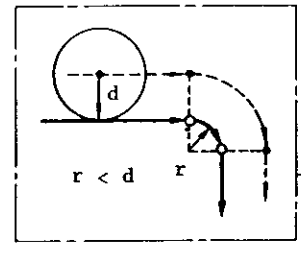
Programmed arc radius $r \leqq$ tool radius d



(a) Inside compensation error



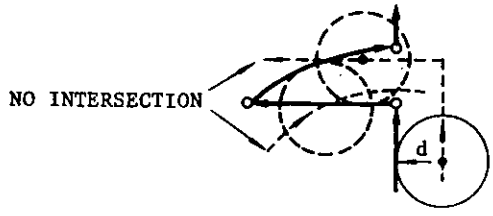
(b) Inside compensation error



Outside compensation is correctly made even when r < d

Fig. 2.14.13.15

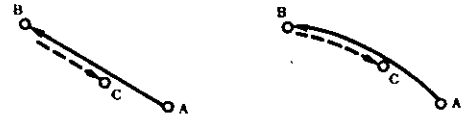
(2) When no intersection point exists on the locus of the offset tool center.



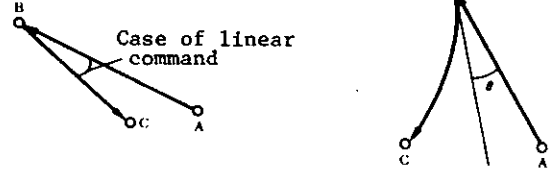
No-intersection error occurs when tool radius is too large relative to the programmed shape.

Fig. 2.14.13.16

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



(a) Reversing command



Note: With the circular arc command, tangent angle θ alone is insufficient.

(b) Command close to reversing

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

C. Input errors occur when the following G codes are programmed in the compensation mode.

Table 2.14.13.2

	Prohibited G codes
G codes producing input errors	G12, G13, G14, G15, (G17 to G19)
	G28, G29
	G73, G76, G81 to G89
	G92

* If a "reset operation" is performed in the compensation mode, compensation is cancelled and G40 remains.

D. Tool radius compensation C is applied to the movement path offset by tool position offsets B and A. However, in principle, avoid applying compensation C to the path using tool position offset A for compensation of tool radius.

E. When programming G41, G42 and G40, G00 or G01 and an F code should be programmed in the same block or in a preceding block.

F. An input error occurs if a G code, G17 to G19 of plane designation for changing the compensation plane is programmed during compensation.

G. Program circle cutting (G12, G13), helical cutting (G14, G15) and canned cycles (G73, G76, G80 to G89) in the tool radius compensation cancell mode. Circle cutting and helical cutting incorporate tool radius compensating functions in themselves. Input errors occur when they are programmed in the compensation mode.

H. Tool radius compensation is also possible on circular interpolation by radius R designation.

I. Program copy (G25) and subprogram (M98, M99) can be programmed in the compensation mode.

J. Compensation is applied to the projection to the compensation plane designated by G17, G18 or G19 when simultaneous movement along three axes (four axes†) is programmed in compensation mode.

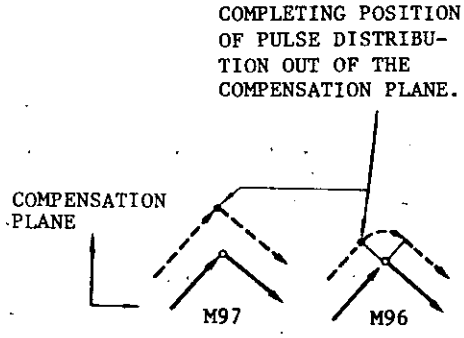


Fig. 2.14.13.18

K. Input error occurs when circular interpolation is programmed out of the plane designated by G17, G18 or G19 (→ 0615).

L. Offset position may be temporarily modified by programming a dummy block using addresses I, J, K.

```
(G42)
 .
 .
 .
N100 G01 X... Y... *
N101      I... J... *
N102      X... *
```

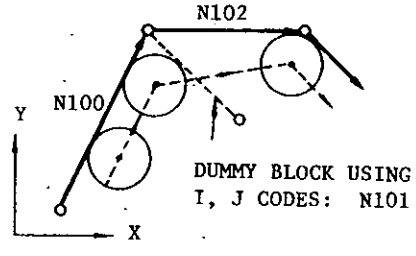


Fig. 2.14.13.19

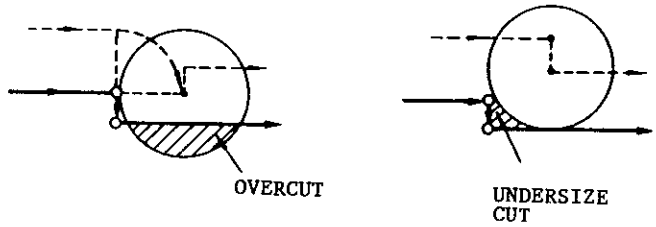
M. Reading ahead of blocks is prohibited when M00, M01 (M02, M30) commands are given, and compensation is usually interrupted. Continuation of correct compensation is secured by programming I, J, K in a dummy block immediately before M00, M01 to avoid interruption.

```
(G41)
 .
 .
 .
N200 G01 X... Y... *
N201      I... J... *
N202      M00 *
N203      X... Y... *
```

Command movement data N203 using I, J

N. Up to 99 radius values can be stored in the the offset memory in total for the tool radius compensation, together with the values for other compensation. Make designation by a D code or a H code. The maximum programmable value of tool radius compensation is ±999.999 mm (or ±40.000 inch) (Refer to 2.10.2 Storage of Tool Offset Values.)

O. Overcut occurs if compensation is programmed on a step less than the tool radius in M96 mode. Keep this in mind. Although undersize cut occurs with the G97 mode, it is better than overcut with the M96 mode.



(a) M96 mode (b) M97 mode

Fig. 2.14.13.20

P. Even in M96 mode, the tool moves directly toward point B without making circular path, if both ΔX and ΔY are smaller than a fixed value as shown below. The fixed value in this case is the value set by parameter No. "77."

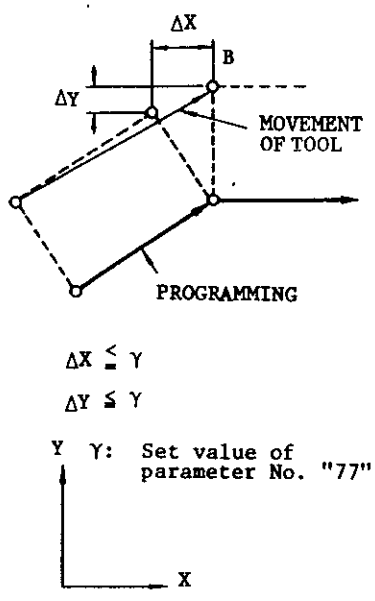


Fig. 2.14.13.21

- Intervention of MDI operation in compensation mode (See NOTE.)

It is possible to reprogram the following data in MDI mode after suspending the block, by turning on the SINGLE BLOCK switch in the compensation mode of G41 or G42.

Programmable data: F, M, S, T and B† codes
Programmable block: Blocks in the buffer register to be executed next

Automatic operation is resumed when the mode is set to the original automatic operation mode (TAPE or MEM) and then Cycle Start is made. Do not make Cycle Start in the MDI mode.

- Intervention to active buffer in compensation mode (See NOTE.)

The data given below can be programmed in the compensation mode of G41 or G42 with procedures identical to those of MDI operation, after turning on the SINGLE BLOCK switch to suspend the block, and then, selecting the RAPID or JOG mode.

Programmable data: F, M, S, T and B† codes
Programmable block: In addition to the block of instructions of the active buffer just executed

When the CYCLE START button is pushed in the RAPID or JOG mode after programming, the instructions are immediately executed and signals such as BCD output are sent out. Automatic operation can be resumed when CYCLE START is made after returning to the original automatic operation mode.

NOTE: In the operation described in these items, the following M codes cannot be written.

M00, M01, M02, M30, M91 to M99

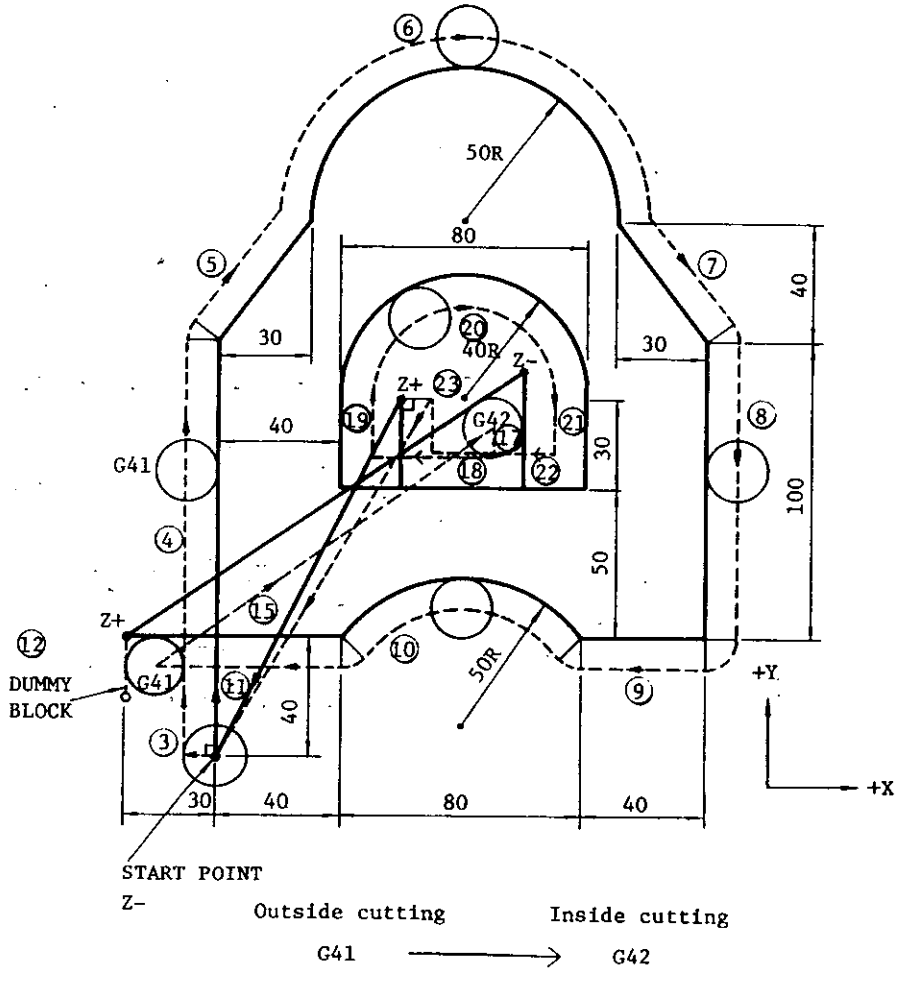


Fig. 2.14.13.22

①	G91 G01 Z-2500 F150 *	———	Incremental; Z axis lowering
②	G17 (G01) F300 *	———	XY plane designation, feed command
③	G41 D21 *	———	Tool radius compensation start command with tool offset No. 21
④	Y14000 *	←	Offset to a point on the normal line of start point of this block.
⑤	X3000 Y4000 *		
⑥	G02 X10000 I5000 *		Outside cutting
⑦	G01 X3000 Y-4000 *		
⑧	Y-10000 *		
⑨	X-4000 *		
⑩	G03 X-8000 R5000 *	———	R designation circular arc
⑪	G01 X-7000 *		

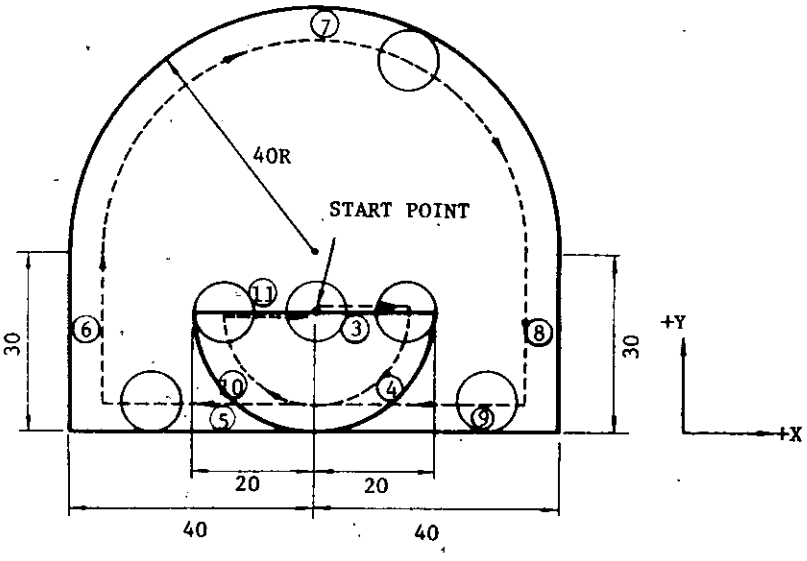
Fig. 2.14.13.22

⑫	J-2000 *	———	Dummy block (for modifying offset position)
⑬	Z2500 *	———	Z axis elevation ┐ Two blocks without movement on the designated plane.
⑭	M01 *	———	Optional stop ┘
⑮	G42 (G01) X13000 Y9000 F2000 *		Switching of direction of compensation (left → right)
⑯	Z-2500 F150 *		Z axis lowering
⑰	Y-4000 F300 *	┐	
⑱	X-6000 *		
⑲	Y3000 *		Inside cutting
⑳	G02 X8000 I4000 *		
㉑	G01 Y-3000 *		
㉒	X-6000 *	┘	
㉓	Y3000 *	———	Offset to a point on the normal line of end point of this block
㉔	Z2500 *	———	Z axis elevation
㉕	G40 (G01) X-6000 Y-12000 F2000 *		Compensation cancel command

The same effect is obtained even when the commands in parentheses are not made. They are entered for ease of understanding.

Fig. 2.14.13.22 (Continued)

EXAMPLE B




```
                (G40)
①    G91 G01 Z-2500 F150 *
②        G17 F300 *
③        G42 D20 X2000 *
④        G02 X-2000 Y-2000 I-2000 *
⑤        G01 X-4000 *
⑥            Y3000 *
⑦        G02 X8000 I4000 *
⑧        G01 Y-3000 *
⑨            X-4000 *
⑩        G02 X-2000 Y2000 J2000 *
⑪    G40 G01 X2000 *
⑫            Z2500 *
```

With the inside cutting in EXAMPLE A, the double cutting allowance at the cutting start and cutting end varies with the tool radius. An inside cutting case with zero double cutting allowance regardless of the cutter radius is shown in EXAMPLE B.

Fig. 2.14.13.23

2.14.14 TOOL POSITION OFFSET B (G43,G44,G49)

Tool position offset B adds or sustracts the stored tool offset values to or from the coordinate command value of the designated axes. It is mainly used for tool length compensation in Z axis and offset for work mounting position in X, Y, ($\alpha^{\dagger}$) axes.

1. G codes of tool position offset B

G code	Group	Meaning
G43	ΔΔ	Positive direction
G44	ΔΔ	Negative direction
G49	ΔΔ	Cancel

2. G43, G44 have effect on the axis programmed in the block, and once these instructions are given, they remain valid until they are cancelled by G49. They are modal for each axis unlike G codes of other Δ groups.

3. G49 cancels tool position offset B of all the axes simultaneously.

4. When programming G43, G44 designate the tool offset number by an H (or D) code simultaneously with designation of the axis.
··· Pair designation of axis and H code is necessary.

EXAMPLE

```
G01  G43  Z···  H01 *   } Offset H01 is
       .                  applied to Z axis
       .
       .
     G43  X···  H02 *   } Offset H02 is
       .                  applied to X axis
       .
       .
G00  G49 *
```

5. In the case where tool position offset B is programmed to two or more axis and then cancellation is made for one axis only, program H00 for this axis.

EXAMPLE

```
G01  G43  Z···  H01 *   } Offset H01 is applied
       .                  to Z axis.
       .
       .
     G44  X···  H02 *   } Offset H02 is
       .                  applied to X axis.
       .
       .
     G43  Y···  H03 *   } Offset H03 is
       .                  applied to Y
       .                  axis.
       .
     G44  X···  H00 *
       .
       .
       .
G00  G49 *
```

6. When programming G43, G44, G49, the G code of group A should be G00 or G01. An input error occurs if a command is given in G02, G03 or the like.

7. Direction of offset

The direction of offset is determined by the sign of the tool offset value designated by an H code in addition to the G code.

Table 4.14.14.1

	Sign of tool offset value	
	Positive	Negative
G43	Offset in positive direction	Offset in negative direction
G44	Offset in negative direction	Offset in positive direction

Note: Generally, tool offset values should preferably be in positive values for the sake of clarity.

8. Offset value

A. The tool is offset by the designated tool offset value from the command target point (the point before offset) both with absolute and incremental instructions.

B. If the tool offset value is changed by designating a new H code during offset, the tool is offset by the new offset value from the new target point (the point before offset), regardless of the previous offset.

EXAMPLE:

(Both command value and tool offset value are given in 0.001 mm.)

	Tool offset value	Position in absolute coordinate value		
		X	Y	Z
G92 X0 Y0 Z0 * →	— →	0	0	0
G90 G00 *	—	0	0	0
G43 Z1000 H01 * →	H01 = 100 →	0	0	1100
G43 X2000 H02 * →	H02 = 200 →	2200	0	1100
G44 Y3000 H03 * →	H03 = 300 →	2200	2700	1100
G44 Z4000 H04 * →	H04 = 400 →	2200	2700	3600
G43 X1000 H05 * →	H05 = 500 →	1500	2700	3600
G43 Y2000 H00 * →	— →	1500	2000	3600
G91 G01 F400 * →	— →	1500	2000	3600
G44 Z-1000 H02 * →	H02 = 200 →	1500	2000	2800
G44 X2000 H02 * →	H02 = 200 →	2800	2000	2800
G43 Y2000 H03 * →	H03 = 300 →	2800	4300	2800
G43 Z-3000 H04 * →	H04 = 400 →	2800	4300	400
G44 X-3000 Y-4000 H03 * →		-300	-300	400
G43 X0 Y0 H00 * →		0	0	400
G00 G49 * →		0	0	0

9. Program "0" as the axis movement instruction, when movement only by tool offset values in incremental designation (G91) is wanted.

   G91 G01 G43 Z0 H01 F... *

   ... Offset is made along Z axis in positive direction only by tool offset value.

   G91 G00 G44 X0 Y0 H02 *

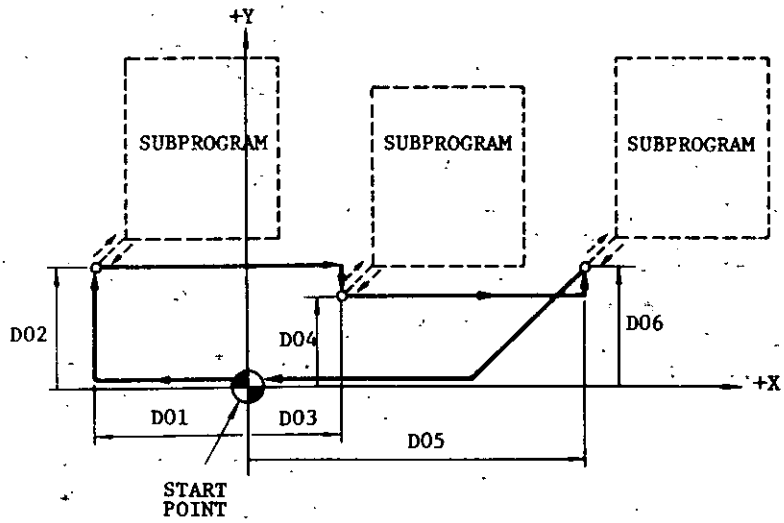
   ... Offset is made along both X and Y axes in negative direction by tool offset value.

   It is meaningless to give a sign to "0."

   In the absolute mode (G90), program the current coordinate command value of the respective axis. That is, program so that the incremental movement value becomes zero.

NOTES:

- If the offset value is changed by MDI in the offset mode, this change becomes operative from the block read next.
- Tool position offset A or tool radius compensation C can be applied in addition to tool position offset B.
- The direction of offset by tool position offset B is determined by G43, G44 regardless of designation of mirror image.
- Tool position offset B is independent of G codes (G17/G18/G19) for plane designation.
- G43, G44, G49 cannot be programmed in the canned cycle mode. An input error occurs when they are programmed.
- When G92 is programmed in an offset mode, programming of absolute zero point is made with the offset value added. Program G92 in cancelled state in principle.
- During automatic operation, display of the tool offset number (H code number) valid for each axis by tool position offset B is possible. See 4.3.7 Display of Current Tool Offset Number.




```
N005 G92 X0 Y0 Z10000 *
     G90 G00
         G43 X0 D01 *
         G43 Y0 D02 *
         M98 P100 L1 *
     G90 G00
         G43 X0 D03 *
         G43 Y0 D04 *
         M98 P100 L1 *
     G90 G00
         G43 X0 D05 *
         G43 Y0 D06 *
         M98 P100 L1 *
     G90 G00
         G49 X0 Y0 *
```

SUBPROGRAM

```
N100 G91 *
     G01 Z-5000 F200 *
     G17 F300 *
     G41 D20 X··· Y··· *
      .
      .
      .
      .
      .
     G40 X··· Y··· *
     M99 *
```

Fig. 2.14.14

2.14.15 TOOL POSITION OFFSET A (G45 TO G48)

Tool position offset A is for extending or reducing the movement value designated in the program by the values in the tool offset memory, and is mainly used for tool radius compensation for square patterns. Therefore, this function is not required with controls equipped with G40, G41, G42 (tool radius compensation C).

1. G codes of tool position offset A

G code	Group	Meaning
G45	Δ	Extension
G46	Δ	Reduction
G47	Δ	Expansion by double
G48	Δ	Reduction by double

2. G45 to G48 extend and reduce the movement value programmed in the block, in the direction of movement by the tool offset value. Extension or reduction is made only in the block in which G45 to G48 are programmed and movements in other blocks are unaffected. Therefore, to restore extended or reduced values to the original program values, an extension or reduction in the opposite direction must be programmed eventually.

3. Make program command by incremental designation (G91) for the sake of making the above operation clear. When the command is given by absolute designation (G90), extension and reduction are made along the direction of movement to the movement value from the end point of the preceding block, to the command target point. That is, extension and reduction are made to the incremental movement amount. The programming may become complicated.

4. When programming G45 to G48, designate the tool offset number by a D code simultaneously with axis designation. Because D codes are modal, they may be omitted if the same D code is used. Store the tool radius value in the tool offset value memory.

EXAMPLE

```
     G91
(1)  G00 G46 X... Y... D01 *      ...  Reduction
(2)  G01 G47 Y... (D01) F... *    ...  Extension by double
(3)      G47 X... (D01) *         ...  Extension by double
(4)      G47 Y... (D01) *         ...  Extension by double
(5)      G47 X... (D01) *         ...  Extension by double
(6)  G00 G46 X... Y... (D01) *    ...  Reduction
```

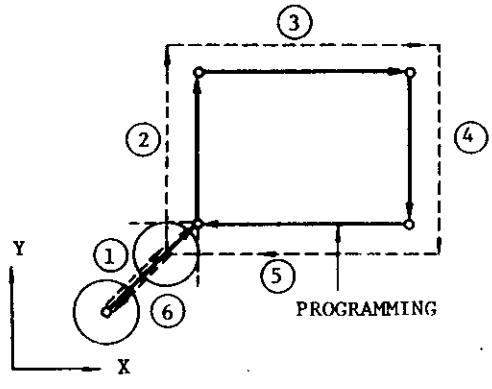


Fig. 2.14.15.1

5. Extension and reduction

Extension or reduction is determined by the sign of the tool offset value designated by a D code in addition to the G code.

Table 2.14.15.1

	Sign of tool offset value	
	Positive	Negative
G45	Extension	Reduction
G46	Reduction	Extension
G47	Extension by double	Reduction by double
G48	Reduction by double	Extension by double

Note: In general, tool offset value should be "positive."

6. Values of extension and reduction

A. Programmed incremental move values are extended or reduced by the designated tool offset values or by twice their values.

G91 G00 G47 X6000 D10 * D10 = 2000

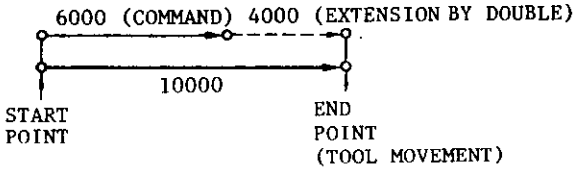
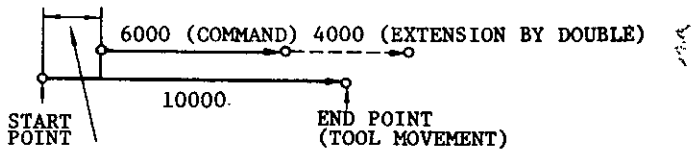


Fig. 2.14.15.2

B. Where extension or reduction is applied to an axis in the preceding block and the start point has already been offset, the total movement value is identical to that described above, but the distance is measured from the offset start point.

With an instruction same as that described above:



Offset value by preceding block

Fig. 2.14.15.3

Note: Where the tool offset value is larger than the programmed movement value, the direction of movement may be reversed when extension or reduction is applied.

G46 X1000 D10 * D10 = 2000

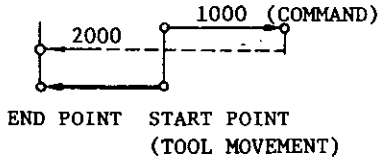


Fig. 2.14.15.4

7. The above applies to X and Y axes, but G45 to G48 may also be programmed to Z axis in the same manner. However, because they are used as the position offset to Z axis, the program is easier to understand if H codes are used instead of D codes.

8. Application to circular interpolation

If I, J, K are programmed in the block with G45 to G48, extension or reduction is made respectively in the same directions as X, Y Z. Therefore, cutter radius compensation is possible with 1/4 circle or 3/4 circle only.

G91

G45 G02 X5000 Y5000 I5000 D10 *

D10 = 2000

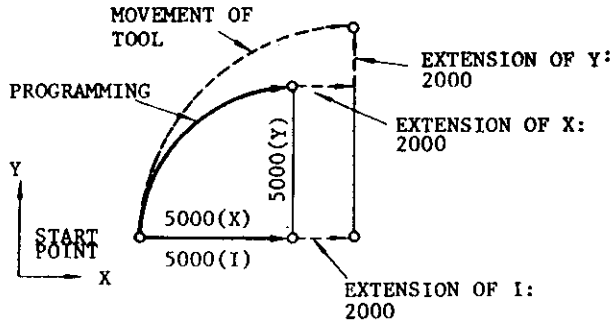


Fig. 2.14.15.5

In practice, correct radius compensation of circular arc is made if an offset is applied in the preceding block.

G91 G01 F... *

G46 X... Y... D10 *

G45 Y... *

G45 G02 X... Y... I... *

G01 X... *

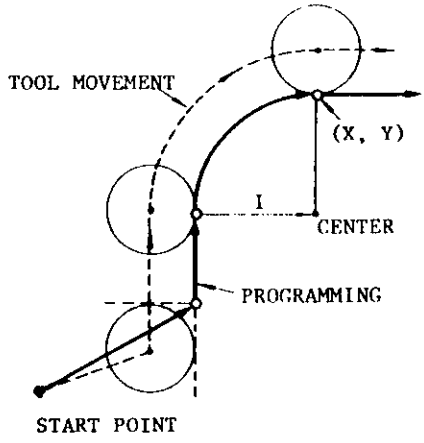


Fig. 2.14.15.6

Note: When it is necessary to program 1/2 circle or 4/4 circle, assemble them using 1/4 circle and 3/4 circle.

9. When programming G45 to G48, the G code of group A can be given together in the same block. An input error occurs if instruction is given with other G codes.

10. When only movement by offset in the incremental designation (G91) is required, program "0" as the axis movement instruction.

G91 G01 G45 X0 Y0 D10 F... *

... Movement is made in the positive direction along both X and Y axis by the offset value with D10.

G91 G00 G46 X0 D11 *

... Movement is made in the negative direction along X axis by the offset value with D11.

It is meaningless to give a sign to "0."

NOTES:

· When G45 to G48 are programmed as the simultaneous movement instruction along two axes, extension or reduction is made in the two axes. Overcut or undersize cut will occur if this is applied to cutting. Keep this in mind.

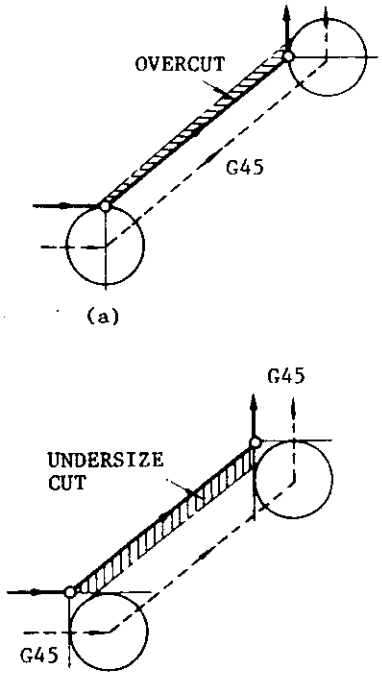
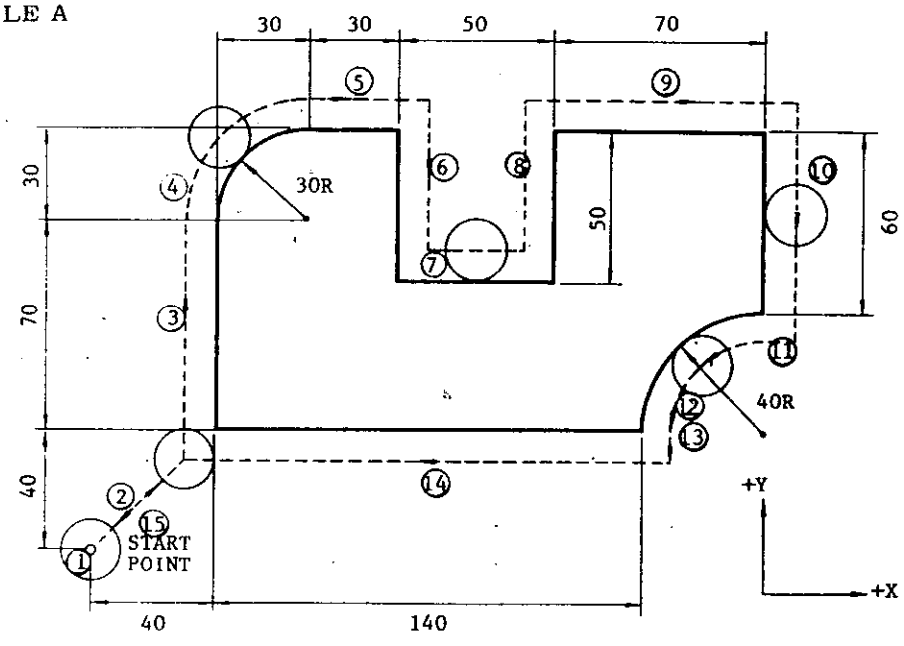


Fig. 2.14.15.7

- Even when the offset value is changed by MDI, the offset instruction previously programmed will not be affected. It becomes operable when G45 to G48 are programmed thereafter.
- This tool position offset A can be applied in addition to the tool position offset B.
- Mirror image can be applied to tool position offset A. That is, it is possible to perform symmetrical cutting with this offset applied.
- Tool position offset A is independent of G codes (G17/G18/G19) of plane designation.
- G45 to G48 can not be programmed in the canned cycles mode. An input error will occur if this is programmed.
- If G92 is programmed in the offset mode, programming of absolute zero point is made with the offset value added.
  In principle, program G92 after returning the offset value to the original value by programming extension or reduction in the opposite direction.
- During automatic operation, the offset distance in each axis from the programmed end point by tool position offset A can be displayed. Refer to 4.3.7 Display of Current Tool Offset Number.

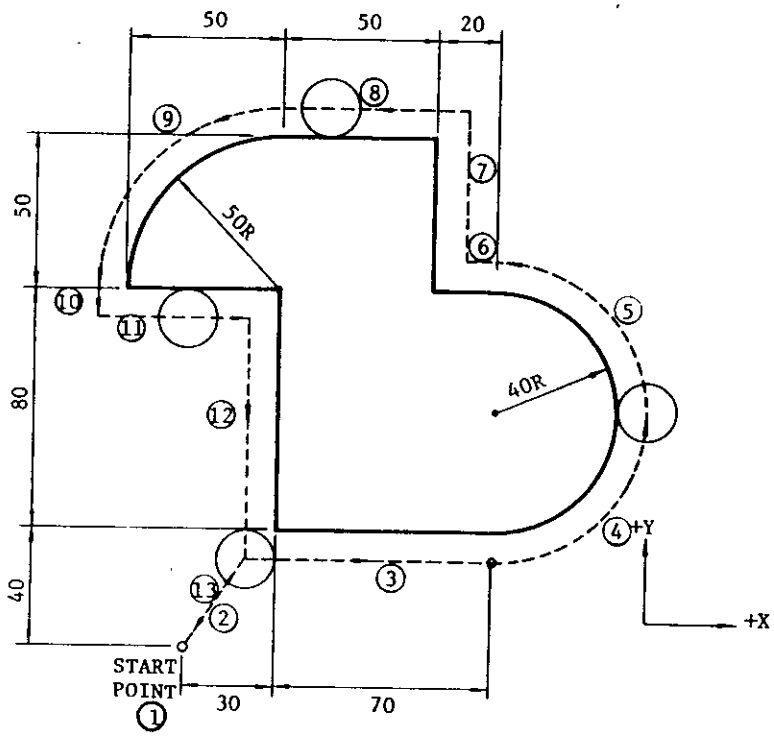
EXAMPLE A




```
①  G91 G01 Z-2500 F150 *
②      G46 X4000 Y4000 D10 F300 *
③      G45 Y7000 *
④  G45 G02 X3000 Y3000 I3000 *
⑤  G45 G01 X3000 *
⑥          Y-5000 *
⑦      G48 X5000 *
⑧          Y5000 *
⑨      G47 X7000 *
⑩      G47 Y-6000 *
⑪      G46 X0 *
⑫  G46 G03 X-4000 Y-4000 J-4000 *
⑬  G46 G01 Y0 *
⑭      G47 X-14000 *
⑮      G46 X-4000 Y-4000 *
⑯          Z2500 *
```

Fig. 2.14.15.8

EXAMPLE B




```
①  G91 G01 F300 *
②      G46 X3000 Y4000 D15 *
③      G45 X7000
④  G45 G03 X4000 Y4000 J4000 *
⑤  G45 G03 X-4000 Y4000 I-4000 *
⑥  G46 G01 X-2000 *
⑦          Y5000 *
⑧      G45 X-5000 *
⑨  G45 G03 X-5000 Y-5000 J-5000 *
⑩  G46 G01 Y0 *
⑪          X5000 *
⑫          Y-8000 *
⑬      G46 X-3000 Y-4000 *
```

Fig. 2.14.15.9

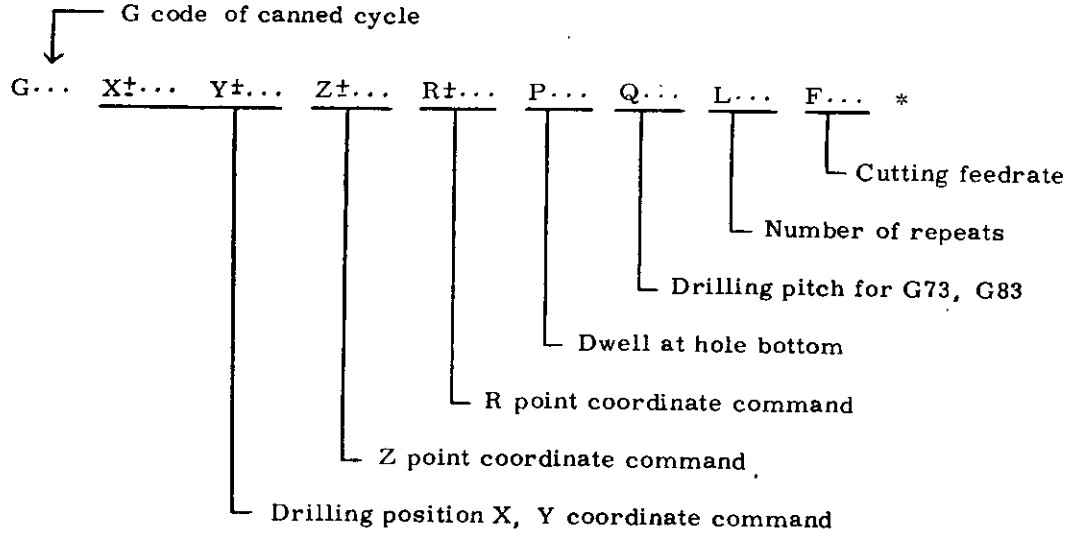
2.14.16 CANNED CYCLES (G73, G76, G80 TO G89)†

- Canned cycles (G73, G76, G80 to G89) are simplified programs that contain specific movements over a number of blocks in one block.
- Nine types of cycles, G73, G76 and G81 to G89 are available, and G80 code is commanded for cancelling them. They are modal G codes which belong to group D.

Table 4.14.16.1 Canned Cycles

G code	Plunging	At hole bottom	Retraction	Application
G73	Wood pecker feed	-	Rapid traverse	High speed deep hole drilling
G76	Cutting feed	Spindle indexing → Shift	Rapid traverse → Shift, spindle start	Boring
G80	-	-	-	Cancel
G81	Cutting feed	-	Rapid traverse	Drilling
G82	Cutting feed	Dwell	Rapid traverse	Spot facing
G83	Wood pecker feed	-	Rapid traverse	Deep hole drilling
G84	Cutting feed	Spindle reversing after dwell	Spindle reversing after cutting feed	Tapping
G85	Cutting feed	-	Cutting feed	Boring
G86	Cutting feed	Spindle stop	Rapid traverse → Spindle start	Boring
G87	Cutting feed	Spindle stop	Manual retraction → Spindle start	Boring
G88	Cutting feed	Spindle stop after dwell	Manual retraction → Spindle start	Boring
G89	Cutting feed	Dwell	Cutting feed	Boring

· Command format



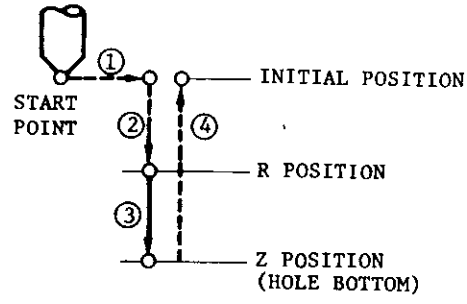


Fig. 4.14.16.1

Operations ① through ④ are executed in one cycle with the commands shown above.

① Positioning the drilling position (X, Y)
② Rapid traverse to R point
③ Drilling to Z point
④ Re turn to R point or to initial point

In the next and subsequent blocks, the G codes of the canned cycles are modal, and program each cycle with changes only in the portions newly specified by the following commands.

X±··· Y±··· * ——— Change in drilling position

X±··· Y±··· Z±··· * —— Change in Z point

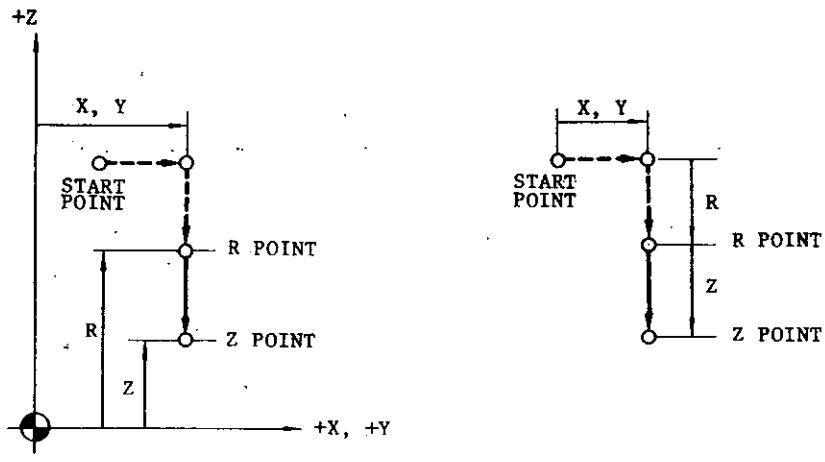
X±··· Y±··· P··· * —— Change in dwell time

For cancelling the canned cycles, program G80 (canned cycle cancel) or a G code of group A in the next block.

· G codes of canned cycles and their function are indicated in Table 2.14.16.2. Both cases of return to R point (G99) and return to initial point (G98) are indicated. For proper programming of these returns, program G98 or G99 in the same block as the canned cycle instruction or in a preceding block. G98, G99 are modal G codes of G group. See 2.14.20 Designation of Canned Cycle Return Point for details.

· Absolute/incremental designation

Address data X±···, Y±···, Z±···, R±··· of coordinates command of canned cycles follow the rules for G90 (absolute designation) or of G91 (incremental designation). That is, the above address data are defined as shown below by G90 or G91 programmed in the same block as the canned cycle instruction or in a preceding block.



(a) With G90 (absolute)

(b) With G91 (incremental)

Fig. 2.14.16.2

EXAMPLE

A. G98 G90 G81 X··· Y··· Z-7000 R-4000 F··· ;

··· Return to initial point, absolute

B. G99 G91 G81 X··· Y··· Z-7000 R-4000 F··· ;

··· Return to point, incremental

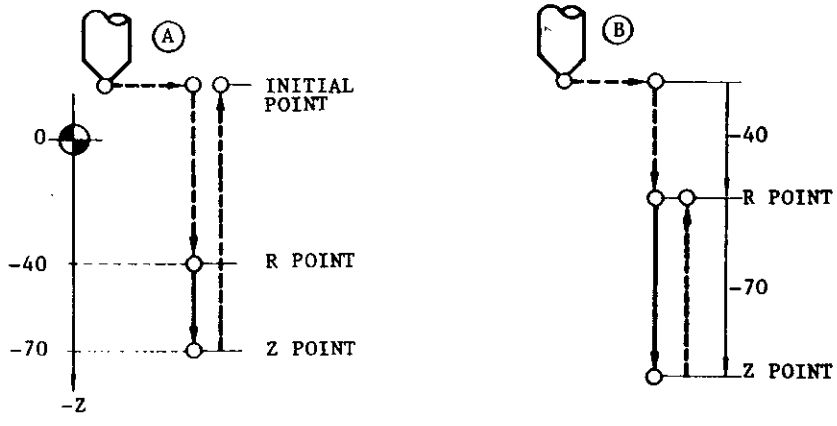


Fig. 2.14.16.3

C. N1 G92 X0 Y0 Z0 *

N2 G28 G90 G81 X1000 Y1000

Z-5000 R-2000 F100 * ⟶

N3 G91 X2000 R-3000 * ⟶

N4 Z-5000 * ⟶

N5 G99 Z-4000 R-5000 * ⟶

N6 G80 G00 Z5000 * ⟶

Absolute position		Remarks
Z =	R =	
-5000	-2000	Canned cycle start
-5000	-3000	Change in X, R
-8000	-3000	Change in Z
-9000	-5000	Change in R, Z
Tool moves to Z = 0		Canned cycle cancel

Newly programmed addresses only are changed including the case where switching is made from G90 to G91 such as N2 → N3 indicated in the above case. As for the non-programmed addresses, the positions programmed in the earlier blocks are maintained.

Note: Since address P, Q, I, J and K are modal in canned cycle mode, if once commanded, they are effective until the canned cycle is cancelled.

Table 2.14.16.2

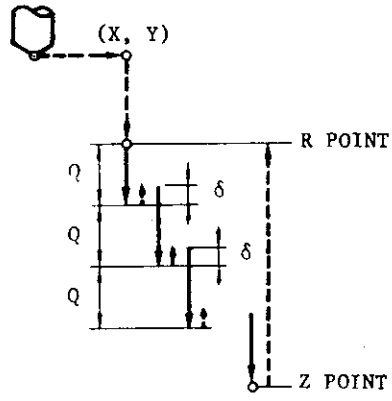
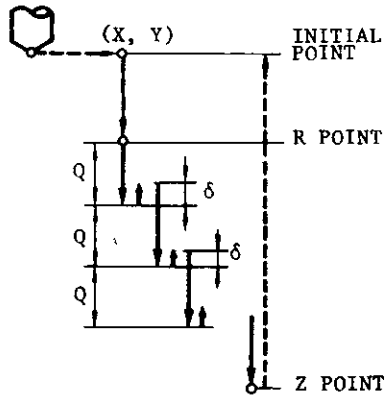
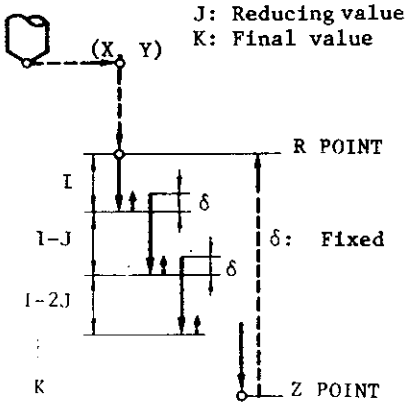
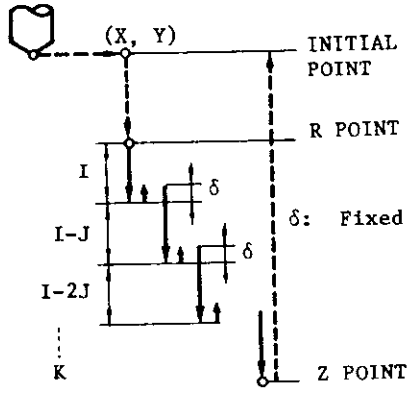
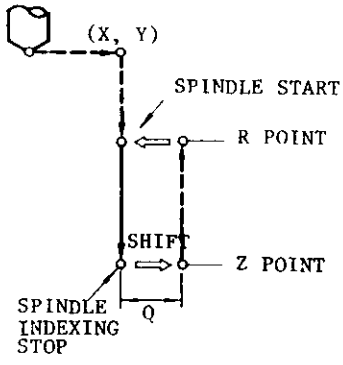
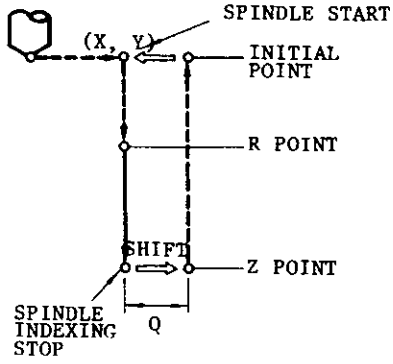
-	With G99 (return to R point)	With G98 (return to initial point)
G73 (Fixed pitch) High speed deep hole drilling	G73 X··· Y··· Z··· R··· Q··· L··· F··· * 	
G73 (Variable pitch) High speed deep hole drilling	G73 X··· Y··· Z··· R··· I··· K··· L··· F··· * I: Initial value J: Reducing value K: Final value 	
G76 Boring	G76 X··· Y··· Z··· R··· Q··· L··· F··· * 	

Table 2.14.16.2 (Continued)

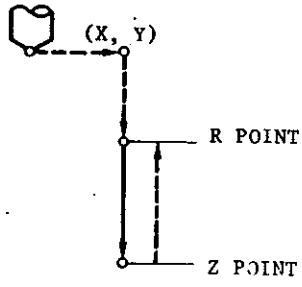
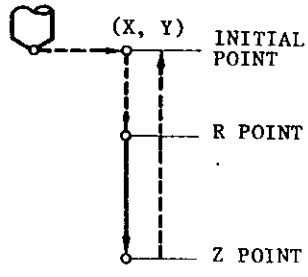
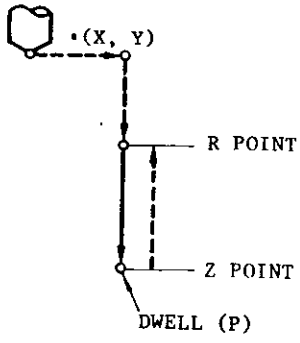
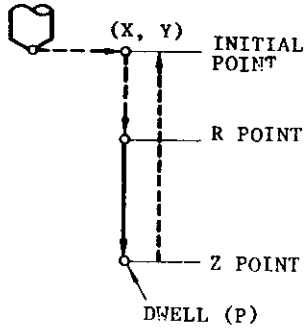
	With G99 (return to R point)	With G98 (return to nitial point)
G80 Cancel	G80 X···	
G81 Drill	G81 X··· Y··· Z··· R··· L··· F··· * 	
G82 Spot facing	G82 X··· Y··· Z··· R··· P··· L··· F··· * 	

Table 2.14.16.2 (Continued)

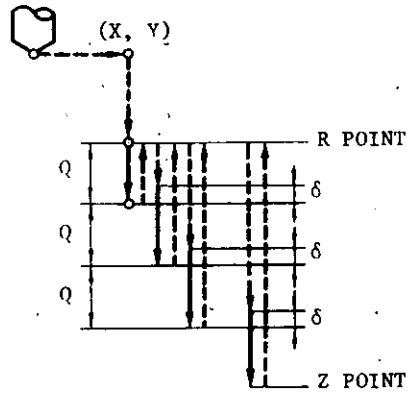
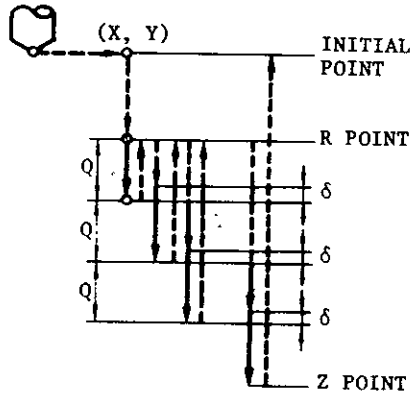
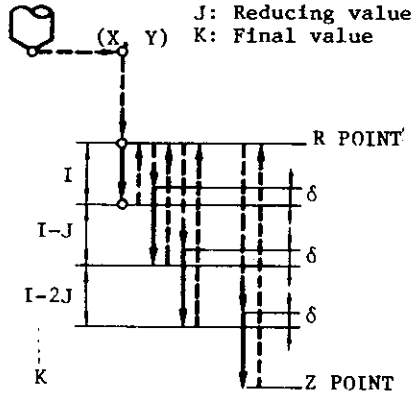
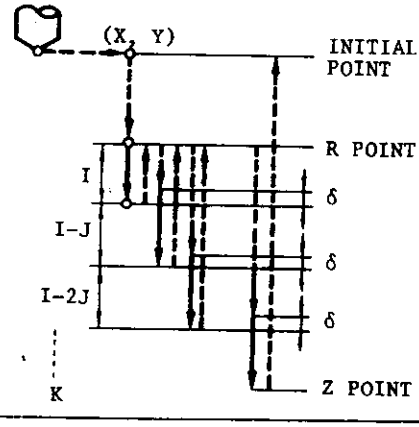
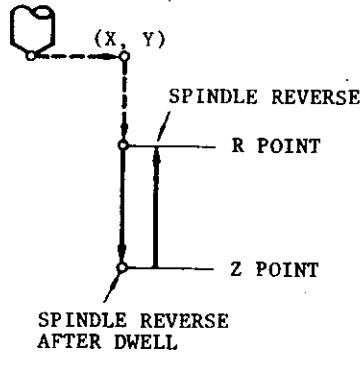
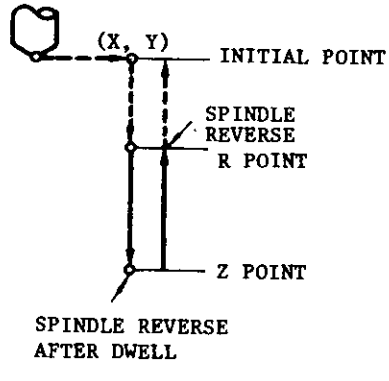
	With G99 (return to R point)	With G98 (return to initial point)
G83 (Fixed pitch) Deep hole drilling	G83 X··· Y··· Z··· R··· Q··· L··· F··· * 	
G83 (Variable pitch) Deep hole drilling	G83 X··· Y··· Z··· R··· I··· J··· K··· L··· F··· * I: Initial value J: Reducing value K: Final value 	
G84 Tapping	G84 X··· Y··· R··· P··· L··· F··· * 	

Table 2.14.16.2 (Continued)

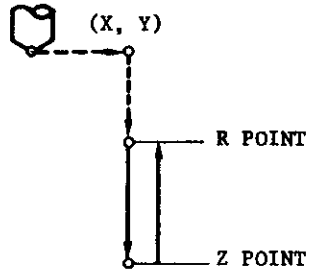
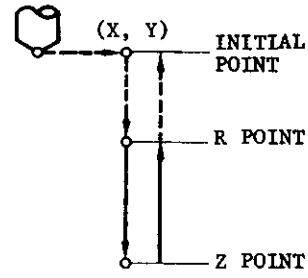
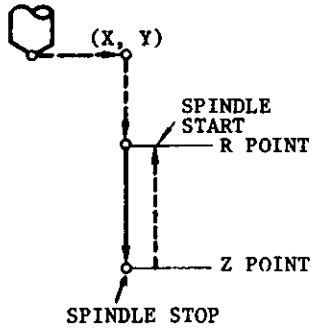
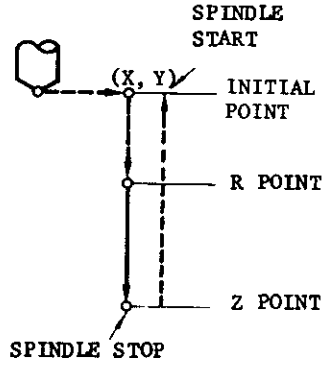
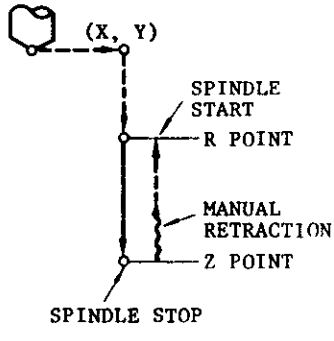
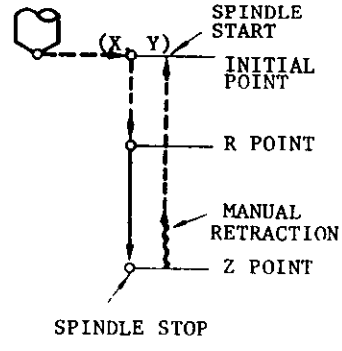
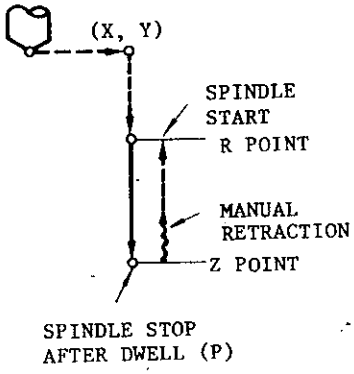
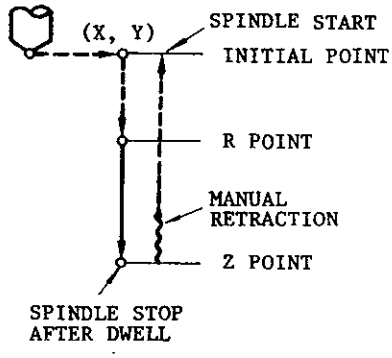
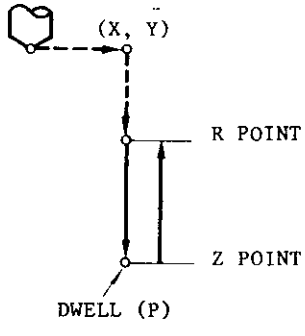
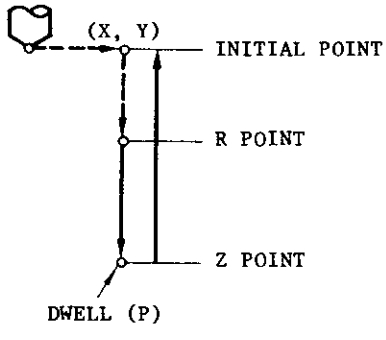
	With G99 (return to R point)	With G98 (return to initial point)
G85 Boring	G85 X··· Y··· Z··· R··· L··· F··· * 	
G86 Boring	G86 X··· Y··· Z··· R··· L··· F··· * 	
G87 Boring	G87 X··· Y··· Z··· R··· Q··· L··· F··· * 	

Table 2.14.16.2 (Continued)

	With G99 (return to R point)	With G98 (return to initial point)
G88 Boring	G88 X··· Y··· Z··· R··· P··· L··· F··· *  	
G89 Boring	G89 X··· Y··· Z··· R··· P··· L··· F··· *  	

· Variable pitch command (G73, G83)

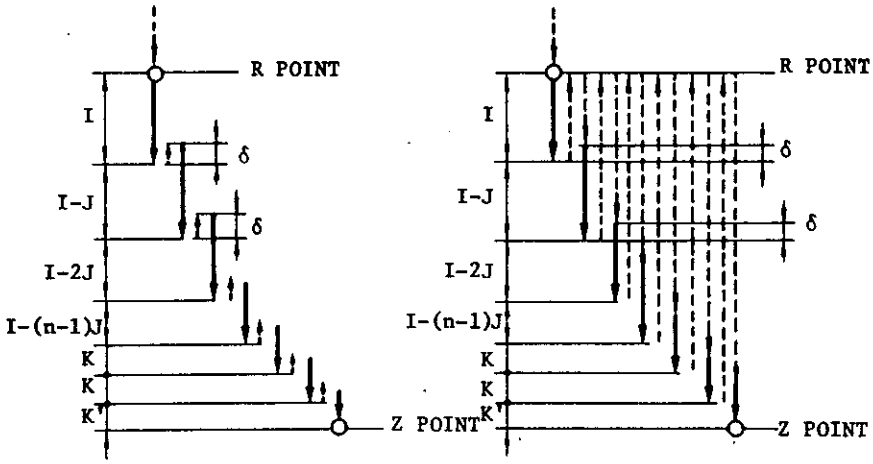
In the deep hole drilling cycles of G73 and G83, variable drilling pitch can be programmed with addresses I, J, K instead of address Q for programming a constant drilling pitch.

I: Initial value

J: Reducing value in 2nd and subsequent plunges

K: Final value

} Command is given without signs



K': Final plunge to Z point

$K' \leq K$

(a) With G73 (b) With G83

Fig. 2.14.16.4

The value of δ is fixed both with variable pitch and fixed pitch.

	δ
Metric system	1.0 mm
Inch system†	0.1 inch

NOTES:

· Q, I, J, K are modal during canned cycle modes and are effective until the canned cycle is cancelled. Specify them without signs.

· Variable pitch can also be programmed by address Q instead of I. Furthermore, when instructions Q, I, J, K are given simultaneously, drilling cycle is executed with variable drilling pitch with Q as the initial value.

EXAMPLE

G91 G73 X··· Y··· R-3000 Z-5500
I1000 ··· J100 K400 F··· *

Drilling pitch

1st plunge	···	10 mm	⇦ I1000
2nd plunge	···	9 mm	
3rd plunge	···	8 mm	
4th plunge	···	7 mm	
5th plunge	···	6 mm	
6th plunge	···	5 mm	
7th plunge	···	4 mm	⇦ K400
8th plunge	···	4 mm	
9th plunge	···	2 mm	⇦ K'
Total		55.00 mm	⇦ Z-5500

· Dwell command in canned cycles

In the canned cycles of G82, G84, G88 and G89, the dwell is programmed by the digits following the address P.

Maximum programmable value:
P8388607 = 8388.607 sec.

Measurement of dwell starts at the time of termination of pulse distribution to Z point. When P is not programmed or when the designated time is zero, the next operation commences immediately after termination of pulse distribution to Z point. ... This is different from dwell using G04. P is modal during canned cycles.

· Spindle control during canned cycles

With G84, G86, G87, G88 and G76, spindle control signals are sent for spindle stop, spindle reversing, etc. at hole bottom and at return positions. See Appendix 2 "Spindle Control in Canned Cycles" for the interface and timing of these signals.

· Intervention by manual operation during canned cycles

In G87 and G89, the automatic operation is suspended after spindle stop at hole bottom. In this case, select manual operation mode (JOG or RAPID), and pull the drill up from the hole bottom in the direction of Z axis by manual operation. Return to the original automatic operation mode (TAPE or MEM) after ensuring that the tool is free from interference, and start cycle. The drill returns to R point or initial point and automatic operation is resumed.
This movement of automatic return to R point or initial point is not affected by the manual absolute switch, and the drill always returns to R point or initial point.

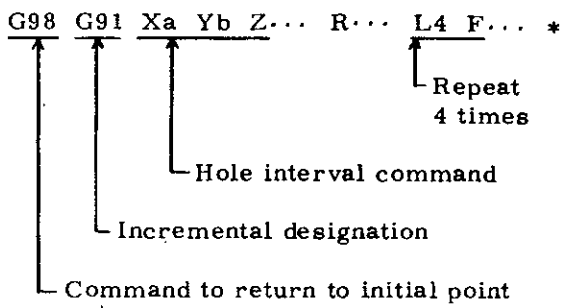
· Number of repeat commands (L)

For drilling multiple holes at equal intervals, command can be given in one block by programming the number of repeats by address L.

Maximum programmable value:
L8388607 = 8388607 times

The L command is non-modal, and is valid only in the programmed block. If L is not given, the operation is performed only one time.

EXAMPLE



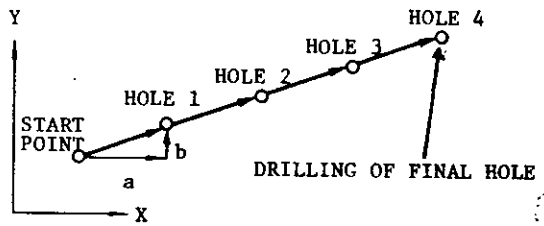


Fig. 2.14.16.5

Note: If the canned cycles are executed with G90 (absolute designation), hole drilling is repeated in the same position. Exercise care in this respect.

NOTES:

· When the canned cycles are executed by turning on the SINGLE BLOCK switch, a temporary stop is made in an intermediate position, and the FEED HOLE lamp lights up.

(1) After positioning to (X, Y) point
(2) After positioning to R point
(3) After termination of each cycle, if L command has been given,

Lighting of this FEED HOLE lamp indicates that operation is in the process of a cycle, and it does not go out even when the operation mode is switched to manual mode. It goes out when a cycle start is made in automatic mode. The single block stop after the completion of canned cycles is as usual, and the FEED HOLD lamp does not light up.

· Be sure to designate R point and Z point by programming R and Z before entering the canned cycle mode. R point and Z point are cleared when canned cycles are cancelled.

- When executing canned cycles with the address data changed, the block requires any of the following address command. The canned cycles will not be executed otherwise.

X, Y, Z, R, Q, I

- An input error occurs when any M, S, T, B† codes are programmed in the blocks of canned cycles. However, an independent block using any M, S, T, B† codes can be inserted in the middle of the canned cycle mode.

G82 X··· Z··· R··· F··· S20 * ——— Input error

S20 * ——————— Correct

X··· Y··· *

·

·

·

- An input error occurs when any of the following G codes are programmed in the canned cycle mode.

· G codes of Δ and ΔΔ group except for G04. · G codes of C group (G41, G42)

When programming G92, G27, G28 etc., make sure to cancel the canned cycles in advance. Cancellation is made when a G code of group A or G80 is programmed.

- An independent block of dwell (G04) can be programmed in the middle of the canned cycle mode. Dwell is executed properly.

- An input error occurs when canned cycles are programmed in the tool radius compensation C mode (G41, G42).

- Start of spindle forward or reverse (M03 or M04) should be executed by automatic operation commands before entering canned cycles.
Do not enter into canned cycles after manually switching the spindle between forward and reverse.

- Execution of subprogram (M98) in canned cycle mode

In a canned cycle mode, M98 P··· L··· * can be programmed to call up subprogram and the canned cycle is continued in the subprogram. Address P must be reprogrammed when a canned cycle requiring dwell is given in the subprogram, because the address P (sequence No. of the first block of subprogram) with M98 command destroys the contents of address P for designation of dwell time.
Addresses X, Y, Z, R, Q or I should not be programmed in the block containing M98 or input error occurs. (→ "0415")

NOTES:

- Programming consideration of M98 in the canned cycle mode is the same as those of the normal M98 mentioned in 2.7.4 Subprogram Function (e.g. Restriction of execution more than eight levels, M98 command from punched tape and the like.)

- Program Copy by G25 should not be made in canned cycle mode. If made, input error occurs. (→ 0415)

- Address L for designation of repetition number of subprograms is nonmodal. But described below is a special case that the address L is retained temporarily.

EXAMPLE

G91 G81 X1000 R-2000 Z-3000 F100 *

L3 * ··· The canned cycle is not executed because X, Y, Z, R, Q or I is not designated in this block. The L3 is retained.

X2000 * ··· The canned cycle G81 is executed 3 times using the retained L3. After the execution the L3 is diminished.

As mentioned above, address L in canned cycle is retained until executed actually.

EXAMPLE

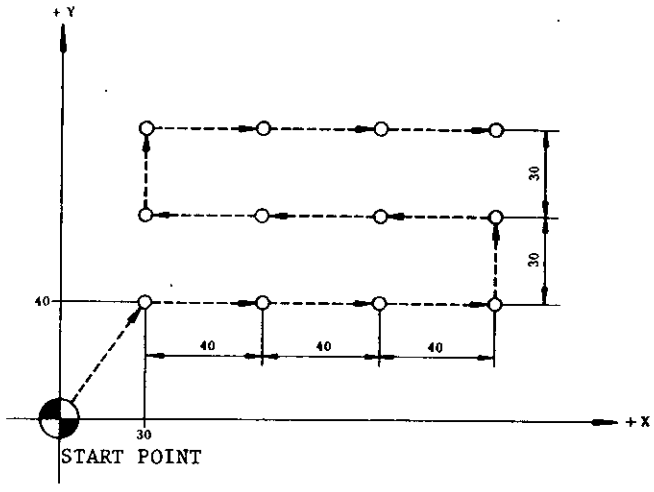


Fig. 2.14.16.6

```
N10 G92 X0 Y0 Z0 *
N11 G90 G98 *              . . .  Return to initial point, Absolute
N12 G81 X3000 Y4000 R-2000 Z-3000 F200 *   . . .  Drilling cycle
N13     M98 P100 *         . . .  Jump to subprogram
N14 G00 X0 Y0 *
N15     T05 *              . . .  Tapper selection
N16     M06 *              . . .  Tool change
N17 G84 X3000 Y4000 R-2000 Z-3000 F2000 *  . . .  Tapping cycle
N18     M98 P99 *          . . .  Jump to subprogram (Note)
N19 G00 X0 Y0 *
 .
 .
 .
N99     P0 *               . . .  Reprogram of dwell time (Note)
N100 G91 X4000 L3 *
N101     Y3000 *
N102     X-4000 L3 *                Subprogram for drilling
N103     Y3000 *                    position pattern.
N104     X4000 L3
N105 G90 G80 *
N106     M99 *
```

Note: Format for reprogram of dwell time is shown. In practice, when the address P specifies "0" in N099 as described here, N18 M98 P100 * may be programmed instead of them. This is because, address P in M98 block is nonmodal and after the execution of the block, the contents of address P are reset to zero.

2.14.17 ABSOLUTE/INCREMENTAL DESIGNATION (G90, G91)

These G codes are for designating whether the movement data following the axis address are in absolute value or incremental value.

· G90 · · · Absolute designation

In the block including G90 and in the subsequent blocks, the movement data which follow addresses X, Y, Z, $\alpha^{\dagger}$ are regarded as absolute values.

G90 G00 X··· Y··· Z··· *
··· Absolute designation

· G91 · · · Incremental designation

In the block including G91 and in the subsequent blocks, said data area is regarded as incremental values.

G91 G01 X··· Y··· F··· *
··· Incremental designation

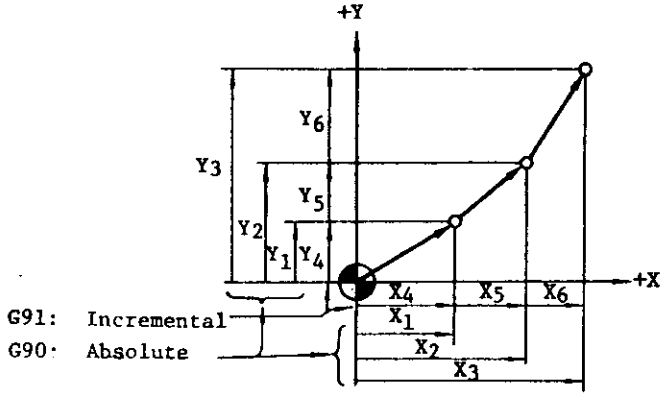


Fig. 2.14.17

· G90, G91 are modal G codes of E group.

· If both G90 and G91 are programmed in the same block, the G code which was programmed last is valid.

NOTE:

· The initial state of these G codes when the power is turned on can be designated by parameter No. "78."

Parameter No. 78	Initial state
"0"	G90
"1"	G91

2.14.18 PROGRAMMING OF ABSOLUTE ZERO POINT (G92)

It is necessary to program the absolute zero point before programming movement command. When an absolute zero point is programmed, one absolute coordinate system is determined, and all absolute movement commands programmed thereafter will move the tool on the programmed coordinate.

· G92 X··· Y··· Z··· ($\alpha^{\dagger}$···) *

With this command, the current position of the tool is programmed in the control as absolute coordinate point (X, Y, Z, $\alpha^{\dagger}$). That is, program the distance (with sign) from the desired absolute coordinate zero position (0, 0, 0, $0^{\dagger}$) to the current position. In other words, G92 command is for designating the position of the "absolute zero point."

EXAMPLE

G92 X50000 Y30000 Z40000 *

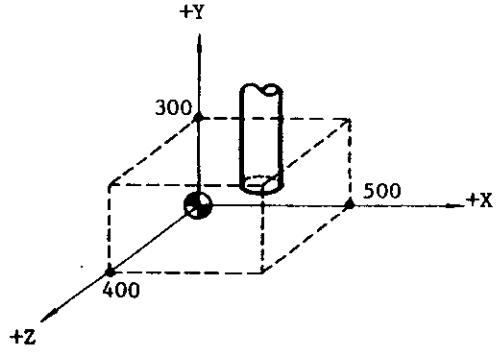


Fig. 2.14.18

· G92 is a G code of non-modal group which is valid only in the programmed block. It is not possible to program other G codes, F, M, S, T, $B^{\dagger}$ codes in the same block.

NOTES:

- In principle, program G92 in the state where all tool offset modes are cancelled.
- When the power is turned on, the current position of the tool is set as absolute zero point (0, 0, 0, 0†). Make sure to reprogram absolute coordinate by G92 before executing the automatic operation.
- The programmed absolute zero point is not affected by reset operation. Perform any of the following operations for resetting the absolute zero point.
  1. Use ORG key (see 4.1.13).
  2. Write G92 X0 Y0 Z0 α† 0 * in MDI mode, and then execute.
  3. Turn the power off and on again.

2.14.19 FEED FUNCTION DESIGNATION (G94, G95)†

These G codes are for selecting whether to designate the feed in mm/min. or in mm/rev. prior to programming the F code for feed, in the case where the control is equipped with feed per revolution† function.

- When G94 * is programmed, the F code programmed thereafter is executed in mm/min. (or inch/min.†, deg/min.†).
- When G95 * is programmed, the F code programmed thereafter is executed in mm/rev. (or inch/rev.†, deg/rev.†).
- G94, G95 are modal G codes of F group, and G94 is selected when the power is turned on.
- When switching between G94 and G95 is made, the previously programmed F code is cancelled. Therefore, a new F code must be programmed.

2.14.20 DESIGNATION OF CANNED CYCLE RETURN POINT (G98, G99)†

The G codes are for designation of the returning point on completion of canned cycles.

- G98 ··· Return to initial point

  The tool returns to the "initial point" on the Z axis at the beginning of the canned cycles, on completion of canned cycles of G73, G76, G81 to G89.

- G99 ··· Return to R point

  The tool returns to the programmed "R point" on completion of canned cycle.

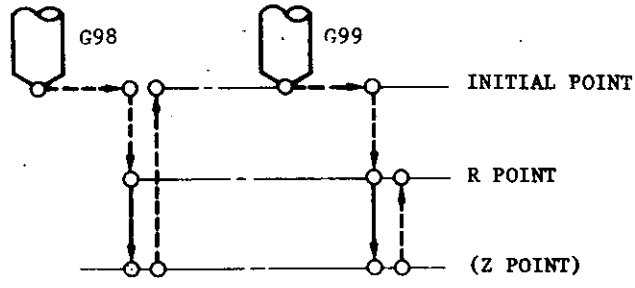


Fig. 2.14.20

- G98, G99 are modal G codes of group G. G98 (return to initial point) is selected when the power is turned on.

NOTE: If the next canned cycles are continuously programmed after return to R point on connection of the preceding canned cycles instructions, the canned cycles are executed with this R point as the start point. In general when drilling many holes, program G99 (R point) before the first drilling and program G98 before the final drilling.

3. NC TAPE PUNCHING

3.1 TAPE CODE

3.1.1 TAPE CODE

Available codes for punching a paper tape are:

EIA code (EIA RS-244-A).
ISO code (ISO R840).

Punching patterns according to these codings are shown in Table 3.1.1.

Before programming, select the code to be used.

Table 3.1.1 Tape Codes

EIA CODE									CHARACTERS		ISO CODE							
8	7	6	5	4	○	3	2	1			8	7	6	5	4	○	3	2	1
		○			○				0				○	○		○			
					○			○	1		○		○	○		○			○
					○		○		2		○		○	○		○		○	
			○		○		○	○	3				○	○		○		○	○
					○	○			4		○		○	○		○	○		
			○		○	○		○	5				○	○		○	○		○
			○		○	○	○		6				○	○		○	○	○	
					○	○	○	○	7		○		○	○		○	○	○	○
				○	○				8		○		○	○	○	○			
			○	○	○			○	9				○	○	○	○			○
	○	○			○			○	a	A		○				○			○
	○	○			○		○		b	B		○				○		○	
	○	○	○		○		○	○	c	C	○	○				○		○	○
	○	○			○	○			d	D		○				○	○		
	○	○	○		○	○		○	e	E	○	○				○	○		○
	○	○	○		○	○	○		f	F	○	○				○	○	○	
	○	○			○	○	○	○	g	G		○				○	○	○	○
	○	○		○	○				h	H		○			○	○			
	○	○	○	○	○			○	i	I	○	○			○	○			○
	○		○		○			○	j	J	○	○			○	○		○	
	○		○		○		○		k	K		○			○	○		○	○
	○				○		○	○	l	L	○	○			○	○	○		
	○		○		○	○			m	M		○			○	○	○		○
	○				○	○		○	n	N		○			○	○	○	○	
	○				○	○	○		o	O	○	○			○	○	○	○	○
	○		○		○	○	○	○	p	P		○		○		○			
	○		○	○	○				q	Q	○	○		○		○			○
	○			○	○			○	r	R	○	○		○		○		○	
		○	○		○		○		s	S		○		○		○		○	○
		○			○		○	○	t	T	○	○		○		○	○		
		○	○		○	○			u	U		○		○		○	○		○
		○			○	○		○	v	V		○		○		○	○	○	
		○			○	○	○		w	W	○	○		○		○	○	○	○
		○	○		○	○	○	○	x	X	○	○		○	○	○			
		○	○	○	○				y	Y		○		○	○	○			○
		○		○	○			○	z	Z		○		○	○	○		○	
					○				Blank	NUL						○			
		○		○	○		○		BS		○				○	○			
		○	○	○	○	○	○		Tab	HT					○	○			○
○					○				CR	LF/NL					○	○		○	
									—	CR	○				○	○	○		○
			○		○				SP		○		○			○			
				○	○		○	○	ER	%	○		○			○	○		○
	○	○	○	○	○	○			UC	—									
	○	○	○	○	○		○		LC	—									
									—	(			○		○	○			
									—	)	○		○		○	○			○
	○	○	○		○				+				○		○	○		○	○
	○				○				−				○		○	○	○		○
	○				○	○	○		o	:			○	○	○	○		○	
		○	○		○			○	/		○		○			○	○	○	○
	○	○	○	○	○	○	○	○	Del	DEL	○	○	○	○	○	○	○	○	○
○	○	○	○	○	○	○	○	○	All Mark		○	○	○	○	○	○	○	○	○

3.1.2 EIA/ISO/AUTO-SELECT

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

Where the contents of parameter No. 80 are "1," the control will automatically adjust so as to read the tape with EIA or ISO coding by sensing first EOB code at the beginning of the tape with label skipped.

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

"0" means EIA code.
"1" means ISO code.

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

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

3.2 PROGRAMMING

3.2.1 PROCESS SHEET

The programming is performed with the process sheet. It is recommended that the process sheet to match final specifications should be made by users in a form easy to perceive and rewrite. Fig. 3.2.1 shows an example of the process sheet.

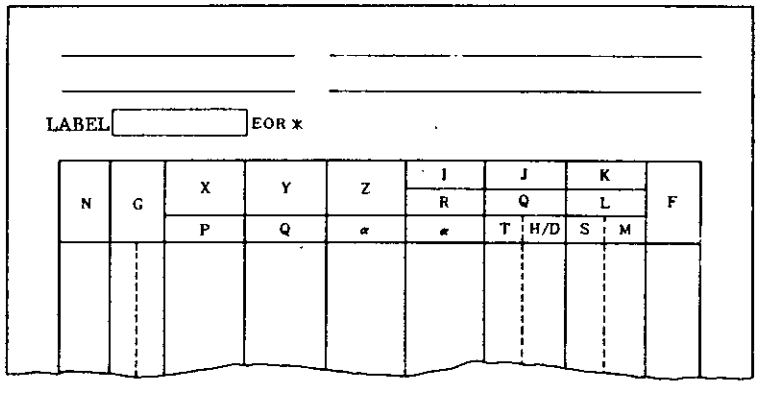


Fig. 3.2.1

3.2.2 GENERAL PROGRAM FORM

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

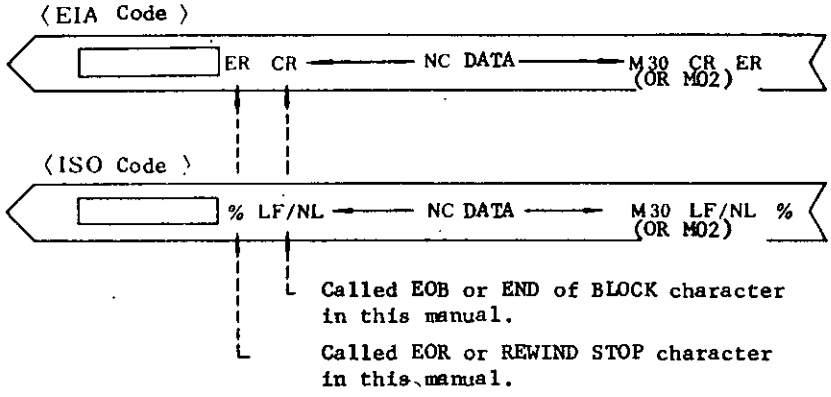


Fig. 3.2.2

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

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

Where storing NC tape data into memory, with the label skipped, the memory stores the data between the first EOB code and the next EOR code. Therefore, EOR code at the end of tape must not be omitted.

3.2.3 PRECAUTION IN PROGRAMMING

A block ends with EOB (End-of-Block) character. EOB character is represented by CR in EIA code and LF/NL in ISO code. In this manual, mark * is substituted for them to read easily this manual.

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

When M02 or M30 is commanded, automatic operation# is stopped. In most cases, the control is reset, or rewinds the tape (or memory) automatically. As the details are determined by the machine, refer to the machine tool builder's manual.

NOTE: See 2.7 MISCELLANEOUS FUNCTIONS

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

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

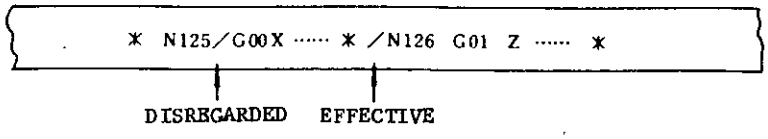
Where the tape vertical parity check (TV check) is made, number of characters in a block must be even. If odd, it should be made even by using "SP" character.

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

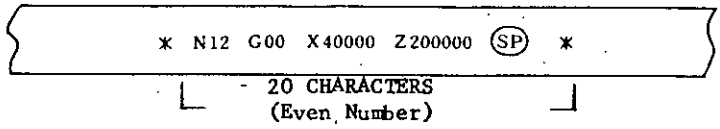
The maximum allowable number of characters in a block is 96. The disregarded characters such as "Del, BS and Tab" are not included in them.

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

a. Position of Optional Block Skip



b. Making the Number of Characters even for TV Check



(Odd number causes tape vertical parity error.)

c. The Maximum Allowable Number of Characters in a Block

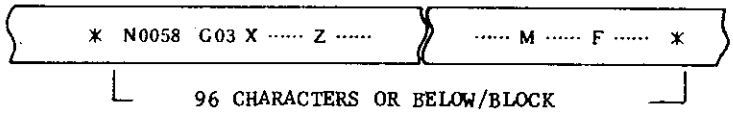


Fig. 3.2.3

3.3 NC TAPE

3.3.1 PAPER TAPE

Eight-channel paper tape for computers complying with JIS#-6243 is used as standard. The dimensions are 25.4 ±0.08 mm (1 inch) width and 0.108 mm (0.0042 inch) thickness.

It is recommended that the color of the tape is black or gray, and not of high transparency. If the tape with high transparency is used, the tape reader may misread it.

3.3.2 PUNCHING OF NC TAPE

NC tape must be punched out with the tape puncher for EIA code or ISO code according to contents of process sheet.

When punching the tape, provide the feed holes at both end parts of the tape. The feed holes part should be more than 70 cm in length for 6" reel†, and 1 m or more, for 8" reel.

3.3.3 CHECKING OF NC TAPE

NC tape can be checked by using the following functions.

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

Japanese Industrial Standard

3.4 NC TAPE HANDLING

3.4.1 SPLICE TAPE

To splice NC tapes, stick a splice tape(0.08 mm thickness) with sprocket holes, or fully perforated one on the one side of the NC tapes. Before using the spliced NC tape, make sure that the sprocket holes are in position. The overlapped part of tapes should not be extremely thick, and do not use the rigid adhesive agent without flexibility.

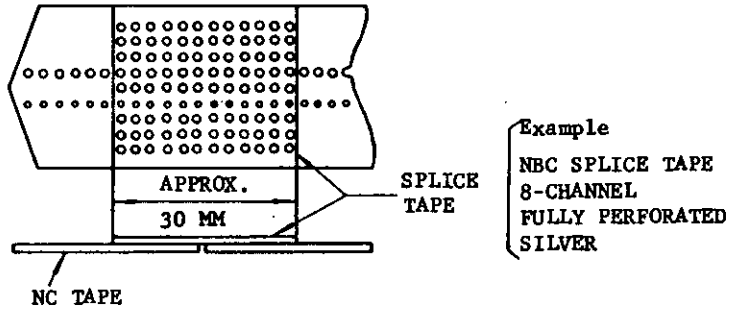


Fig. 3.4.1 Splicing of NC Tapes

3.4.2 KEEPING OF NC TAPE

To maximize life of NC tape, protect NC tape from moisture and oil, and do not handle the tape with oil-stained gloves.

Properly kept tapes will permit 300 times of reading and rewinding.

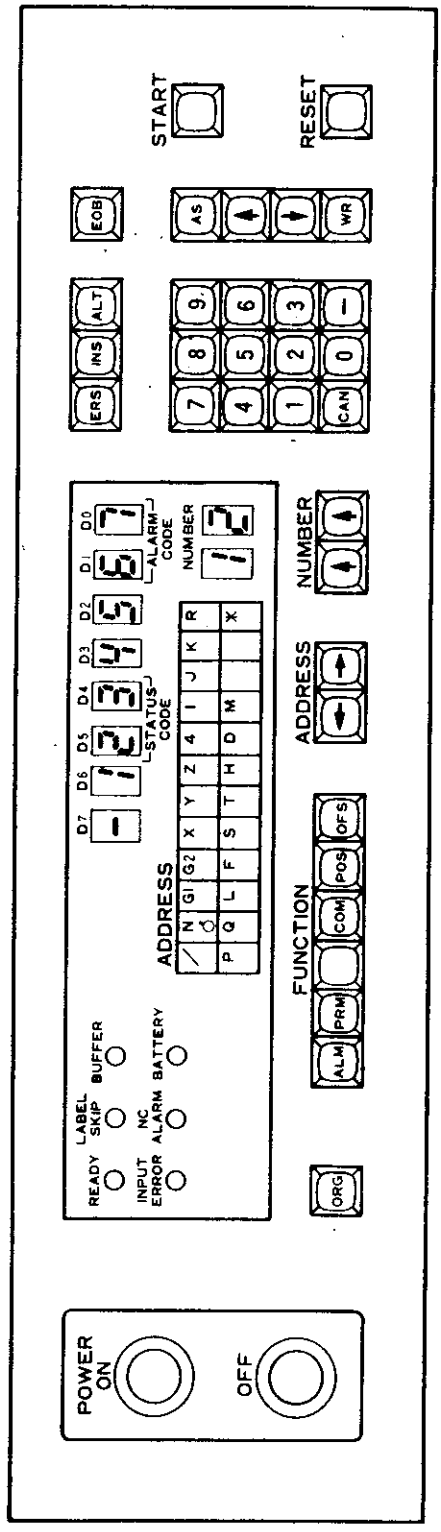


Fig. 4.1.0 Standard NC Operator's Panel

4. NC OPERATOR'S PANEL

4.1 PUSHBUTTONS, KEYS, AND LAMPS

4.1.1 POWER ON/OFF PUSHBUTTONS

· Double-action POWER ON pushbutton

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

· POWER OFF pushbutton

Depress it to remove servo power and control power.

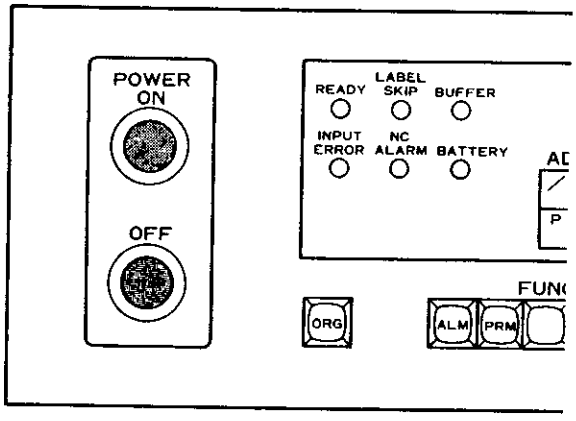


Fig. 4.1.1

4.1.2 INDICATING LAMPS

· READY lamp

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

· LABEL SKIP lamp

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

In the MEM or EDT mode, the illuminated lamp indicates that memory or tape is rewound.

This display is related to only tape or memory information but not to MDI operation.

BUFFER lamp

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

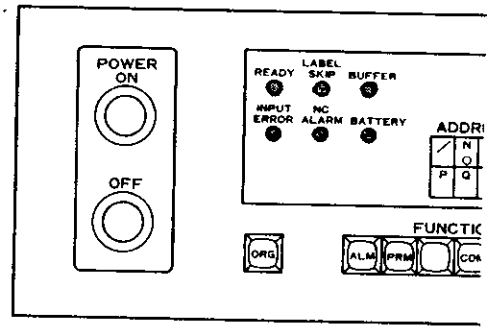


Fig. 4.1.2

· INPUT ERROR lamp

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

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

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

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

· NC ALARM lamp

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

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

· BATTERY lamp

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

4.1.3 FUNCTION SELECT KEYS

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

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

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

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

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

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

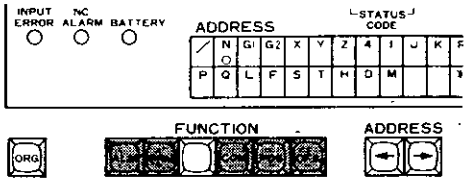


Fig. 4.1.3

4.1.4 ADDRESS SELECT KEYS AND DISPLAY

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

· [→] Key

Each depression of the key moves the address indicator to the right by one address. If the key is held depressed for more than 0.5 second, the indicator will continue to move automatically until the key is released.

· [←] Key

Each depression of the key moves the address indicator to the left by one address. If the key is held depressed for more than 0.5 second, the indicator will continue to move automatically until the key is released.

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

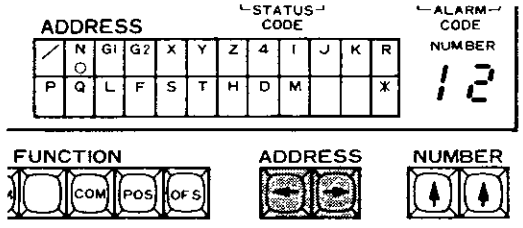


Fig. 4.1.4

4.1.5 NUMBER SETTING KEYS AND DISPLAY

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

· [↑] Key at right

Each depression of the key increases the digital display by 1. If the key is held in for more than 0.5 second, the number will continue to increase automatically until the key is released.

· [↑] Key at left

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

By pushing the both NUMBER keys simultaneously, NUMBER is set back to "00." At that time depressing the both keys simultaneously again makes the NUMBER 99.

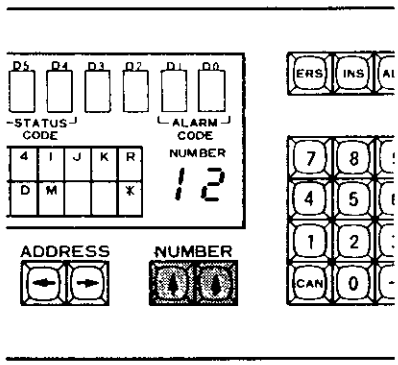


Fig. 4.1.5

4.1.6 DATA KEYS

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

4.1.7 WR (WRITE) KEY

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

4.1.8 AS (ADDRESS SEARCH) KEY

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

4.1.9 SEQUENTIAL SEARCH KEYS [↓] AND [↑]

Sequential search keys [↓] and [↑] located between AS and WR keys are to search forward or reverse through memory or tape contents for a particular block.

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

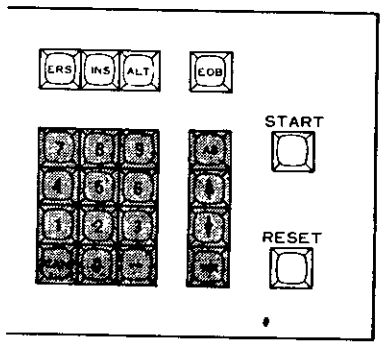


Fig. 4.1.9

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

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

· ERS (erase) key

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

· INS (insert) key

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

· ALT (alter) key

It is to modify a block of data in the memory.

· EOB (end of block) key

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

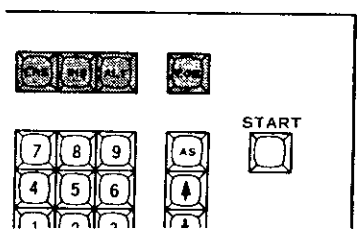


Fig. 4.1.10

4.1.11 START KEY

It is used to start tape punching and program storing from tape in the EDT mode.

4.1.12 RESET KEY

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

- Move command cancel
- Buffer register clear
- Alarm code release if the cause eliminated
- Tool offset cancel
- Miscellaneous function cancel
- Label Skip function ON
- Memory pointer rewind
- Sequence number reset
- RST signal transmission to the machine
- G-code of A group changed to G00 (where parameter No. 99 = "0"), G-code of C group to G40, and G-code of D group to G80

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

- Current position value of each axis
- Modal G-codes (except G-code of C and D group), and G-code of A group where parameter No. 09 = "1"
- F commands
- S 4-digit commands (S 2-digit commands are affected.)
- Memory contents such as tool offset values and parameter data

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

4.1.13 ORG KEY

ORG key is used to set the current position of an axis at "0." The operation of the ORG key gives the same result of executing G92 X0 Y0 Z0 (α0†)*.

1. Select any of manual modes (RAPID, JOG, STEP, and HANDLE).
2. Any FUNCTION key is permitted to be set.
3. Select the address of the axis desired.
4. Depress the ORG key.
   Then the current position of the selected axis is set at "0."
5. Repeat steps 2 to 4 to set the current position of the other axes at "0."

NOTE: ORG key is not effective during operation or while BUFFER lamp is on: data in the next block are held in buffer register.

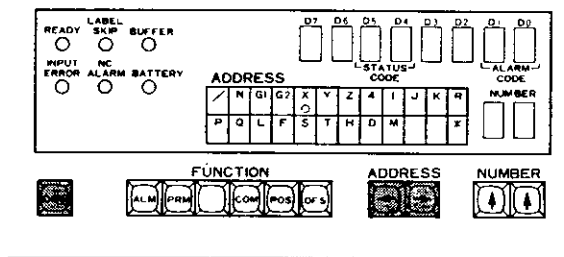


Fig. 4.1.13

4.1.14 UNIVERSAL DISPLAY

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

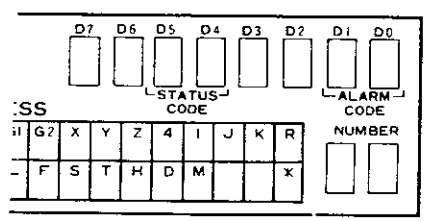


Fig. 4.1.14

4.1.15 TAPE FEED AND SYSTEM NO. SWITCHES

These switches are mounted above the tape reader.

- TAPE FEED switch

  It is to feed and rewind the tape manually with the control at standby.

Setting the switch to F (forward) causes the tape to feed. To rewind the tape, set the switch to R (reverse).

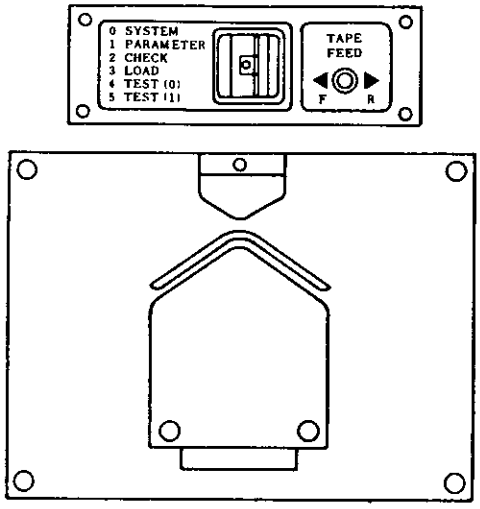


Fig. 4.1.15

· SYSTEM NO. switch

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

Setting at:

"0" SYSTEM--

Is for usual operation and prevents writing parameters is prevented.

"1" PARAMETER--

Is to write parameters and at this position prevents Cycle Start.
Set the switch back to "0" when parameters have been written.

"2" CHECK--

Is to collate the system program stored with the system tape.

"3" LOAD--

Allows storing maintenance tape data into the control.

"4" TEST (0)--

Permits usual operation. Diagnosing of the memory contents and checking of Reference Zero position are omitted.

"5" TEST (1)--

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

4.2 POWER ON/OFF OPERATION

4.2.1 TURNING ON POWER

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

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

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

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

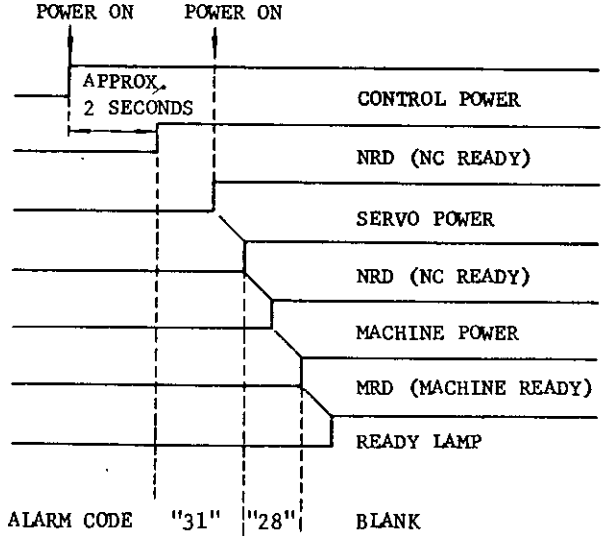


Fig. 4.2.1 Sequence of Turning on Operation

4.2.2 TURNING OFF POWER

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

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

2. Depress the POWER OFF button to cut off control power.

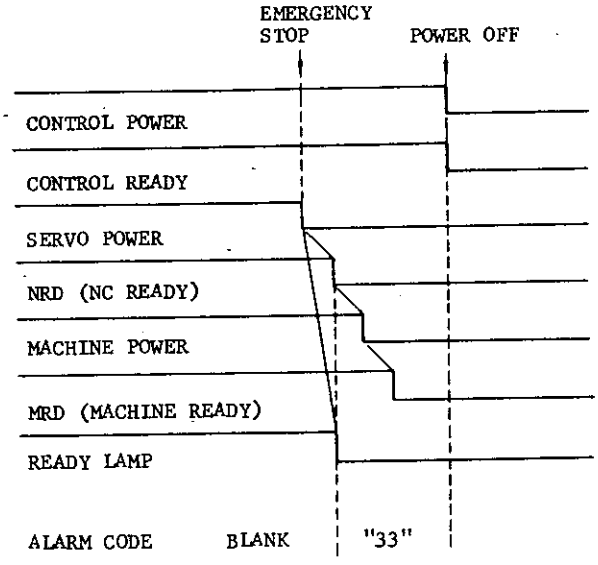


Fig. 4.2.2 Sequence of Turning Off Operation

4.2.3 REMOTE TURNING ON/OFF BUTTONS

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

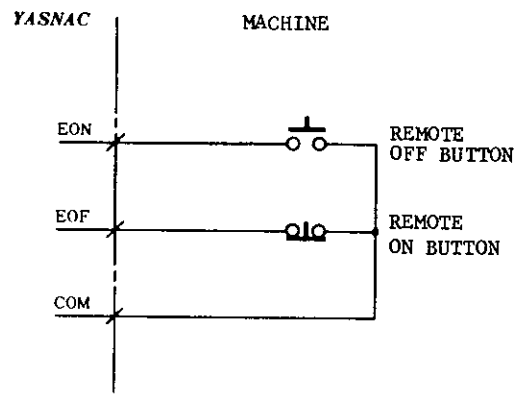


Fig. 4.2.3 Connection of Remote Power ON/OFF Buttons

4.3 DISPLAY AND WRITING OF COMMAND DATA

4.3.1 DISPLAY OF COMMAND DATA

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

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

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

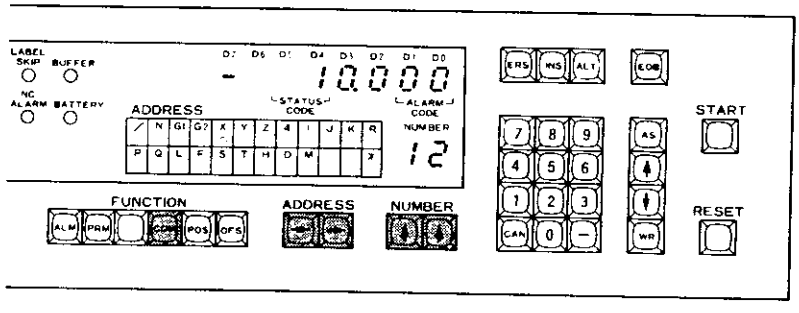


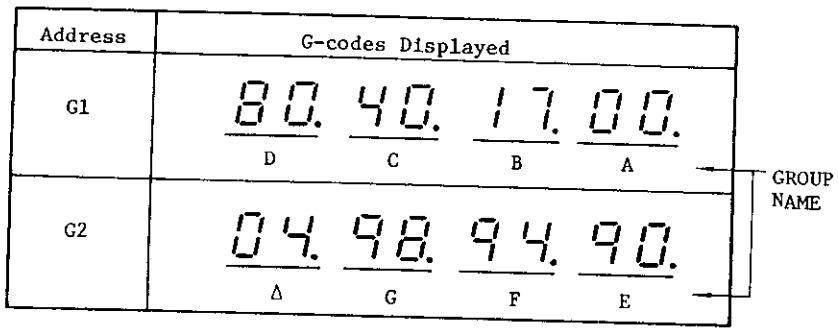
Fig. 4.3.1.1

· To display all address characters in the selected block

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

· To display all G-codes

All the current G-codes are displayed by setting NUMBER at "99" and selecting address G1 or G2.



Notes:

1. Double depressions of both NUMBER keys make the NUMBER 99.
2. Display for G code of Δ group is blank when it is not specified.

Fig. 4.3.1.2 Example of All G-code Display

· Usual G-code display

When the NUMBER is set at other than 99 except in the EDIT mode, the current G code among listed below will be displayed on the extreme right of the universal display regardless of addresses G1 and G2.

Addresses	G-codes Displayed
G1, G2	A Group, D Group, Δ Group (except G25, G43 to G48)

Fig. 4.3.1.3

· G-code display in the EDIT mode

Selecting addresses G1 or G2 in the EDIT mode displays stored G-codes in a pointed block according to G1 or G2 on their respective group position on the universal display shown in Fig. 4.3.1.2. The display is made independently of operating NUMBER keys.

4.3.2 WRITING COMMAND DATA BY MDI

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

1. Set the MODE select switch to MDI.
2. Depress the COM key and it will light up.
3. Select the address character with ADDRESS key. G code can be written by selecting either one of G1 and G2.

   Modal command data already executed is displayed on the universal display.
4. Key in the new data through the DATA keyboard.

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

   The data just keyed in enters the buffer register as a new command data. The coordinate value displayed are modified with the tool offset value.
6. Repeat steps 3 through 5 until a block has been written.
7. Depress the CYCLE START pushbutton, and the input commands are executed.

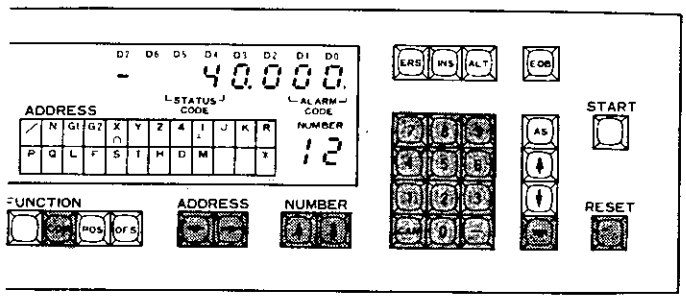


Fig. 4.3.2

NOTES:

- Depressing the RESET key during writing operation by MDI deletes all the data already written.
- When writing G-codes other than A and D group, check the written value on the universal display by setting the number at "99" with the NUMBER key. G-codes of other than A and D group and some of * group are not displayed by usual display operation.
- Depressing COM key while writing command data by MDI displays all the addresses of the data already written on the address display. See 4.3.1 Display of Command Data.
- The data of codes except F, M, S, T and B† cannot be changed by MDI operation as far as Cutter Radius Compensation C is made by G41 or G42.

To write slash "/" code, set the address indicator to slash "/," input 1 using DATA key, and depress the WR key.

4.3.3 CURRENT POSITION DISPLAY

The current position of X-, Y-, or Z-axis can be displayed at any time in all modes. Operating procedure is as follows.

1. Depress POS key, and it lights up.
2. Set the ADDRESS to X, Y, or Z using ADDRESS keys.

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

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

Where parameter No. 79 = "0"

- Displayed current position is the same as that on the current position display unit†.
- Universal display shows total amount of movement of the tool by manual and automatic operation. It cannot be reset by G92.

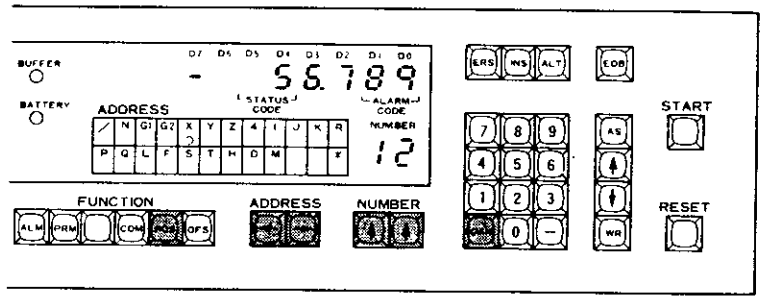


Fig. 4.3.3

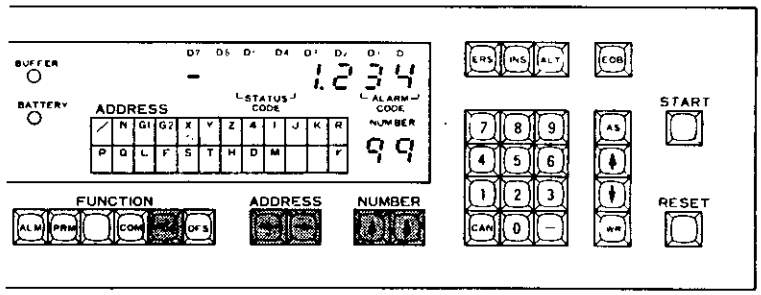


Fig. 4.3.4

- The display is updated even if the LOCK MODE switch is set to MACHINE LOCK position.
- To reset the displayed data, depress the POS and CAN keys simultaneously.
- The RESET buttons on the current position display unit † do not affect the universal display.

Where parameter No. 79 = "1"

- Displayed data is automatically set up by G92 command. The universal display shows the current position based on the programmed coordinate system as far as manual operation does not interrupt.
- Displayed data cannot be cleared by any buttons on the operator's panel except ORG key.
- The display is updated even if the LOCK MODE switch is set to MACHINE LOCK position.

4.3.4 INCREMENTAL VALUE DISPLAY

Incremental values of the axis movement are displayed by setting the NUMBER at 99 in addition to selecting the address of the axis to be displayed and pressing the POS key.

Displayed shows:

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

4.3.5 DISPLAY OF TOOL OFFSET VALUE

The tool offset value stored in memory can be displayed at any time in every mode even during automatic operation.

Operating procedure is as follows:

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

   The selected offset value will be displayed on the universal display.

The control recognizes addresses H and D as the same when displaying or writing tool offset value. If the address other than H or D has been selected, the display will be blank.

4.3.6 WRITING OF TOOL OFFSET VALUE

Writing or modification of the tool offset value is always possible independently of operation mode. Writing tool offset value is completed by writing the incremental value. The written value just keyed in is added algebraically to the stored offset value and displayed as a new offset.

The procedure is as follows:

1. Depress the OFS key, and it lights up.
2. Select the address character H or D using the ADDRESS key.

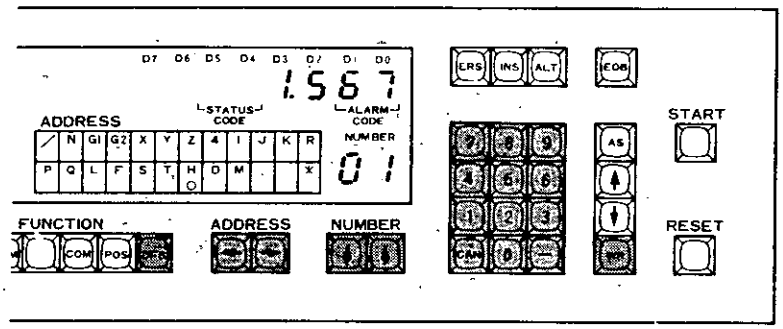


Fig. 4.3.6

3. Set the tool offset number using the NUMBER key.

   The selected offset value is displayed.

4. Key in offset value to be added from the DATA keyboard. The display shows the data just keyed in, flickering the last significant digit. To correct the data just keyed in, depress the CAN key and input correct data.

5. Depress the WR key. Then the displayed data will be added algebraically to the stored value. The summed value is a new tool offset value.

NOTES:

- To enter the new offset data instead of incremental value, depress both OFS key and CAN key simultaneously before handling DATA keyboard. The offset value stored will be cancelled and the display shows 0.
- Tool offset values stored in memory will not be erased by turning off the power.
- Writing and modification of tool offset values can be always possible in any mode including automatic operation mode.
- When the offset value is changed during automatic operation, the blocks in buffer register and active register are executed with an old one. The new offset value is effective in the next block.

4.3.7 DISPLAY OF CURRENT TOOL OFFSET NUMBER

- Display of tool offset number for tool position offset B

  To display the current tool offset number for tool position offset B (G43, G44) during automatic operation, proceed as follows.

  1. Depress the OFS key and set the NUMBER at 00.

  2. Select the address character H.

     Current tool offset number for each axis will be displayed on the universal display. See Fig. 4.3.7.1.

  Note: If the address D is selected in step 2, the display shows the current tool offset number during cutter radius compensation (G41, (G42) mode.

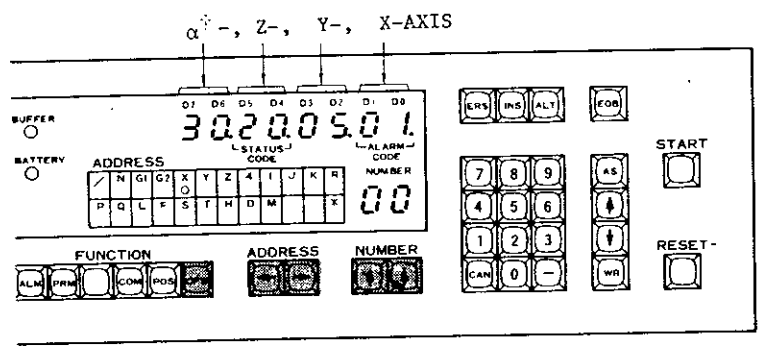


Fig. 4.3.7.1

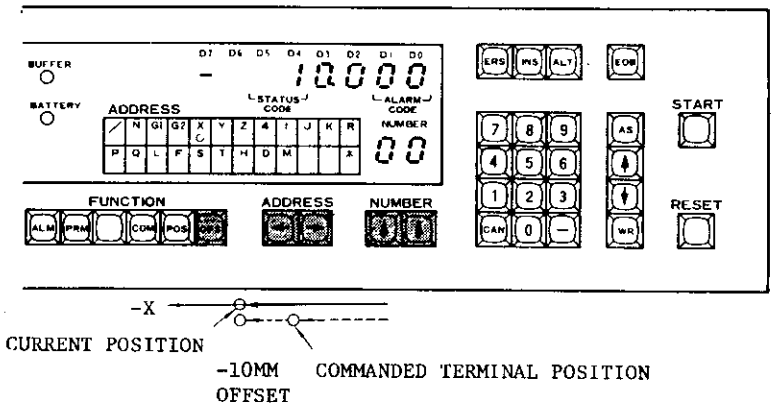


Fig. 4.3.7.2

· Display of tool offset value for tool position offset A

To display the current tool offset value to which tool position offset A is functioning, proceed as follows.

1. Depress the OFS key and set NUMBER at 00.

2. Select the address character X, Y, Z or 4.

   Then the display shows the current offset value for the selected axis. The value displayed means the departure of the tool from the commanded position without being offset.

4.3.8 PARAMETER DISPLAY

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

1. Depress the PRM key, and it lights up.

2. Set the parameter number using NUMBER key.

3. Select the address character of the axis with the ADDRESS key when displaying the parameters requiring axes selection.

   The parameter data will be displayed as it is entered on the universal display. If the address character except for axis X, Y, Z, or 4 is selected for parameters requiring axis selection, the display will be blank.

4.3.9 WRITING PARAMETERS

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

1. Set the SYSTEM NO. switch at "1."

2. Depress the PRM key, and it is illuminated.

3. Set the parameter number using the NUMBER key.

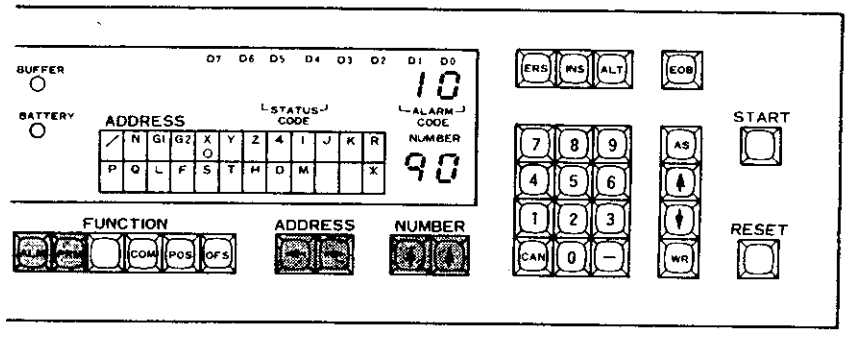


Fig. 4.3.8

4. Select the address character of the axis with the ADDRESS key when writing the parameter requiring axis selection.

   The parameter data stored will be displayed on the universal display.

5. Key in the parameter data from the DATA keyboard.

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

6. Depress the WR key. The data just keyed in will be stored into memory as a new parameter.

7. Repeat steps 3 to 6 until the necessary parameters have been set.

8. Set back the SYSTEM NO. switch to "0."

9. Depress the RESET key.

NOTES:

- Data of parameters, once stored in memory, are not erased by turning off the power.
- With parameter No. at 00, totalized time of automatic operation will be displayed. For details, refer to 4.3.10 Operation Time Display.
- Setting parameter number at 99 displays active address of the tape memory with number of characters on the universal display. See 4.5.4 Address Display of Tape Memory.
- Data of parameter numbers 01 to 98 can be displayed at anytime. They cannot be changed unless SYSTEM NO. switch is set at "1."
- If any of parameter data is accidentally destroyed, alarm code "17" will be displayed.
- SYSTEM NO. switch is provided above the tape reader.
- Where the SYSTEM NO. switch is set at other than "0" and "4," CYCLE START button cannot start the operation. On completion of writing parameter data, never fail to set back the SYSTEM NO. switch to 0. The switch is not usually set at 4 which is for test.

4.3.10 OPERATION TIME DISPLAY

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

1. Depress the PRM key, and it lights up.

2. Set the parameter No. at 00 with NUMBER key.

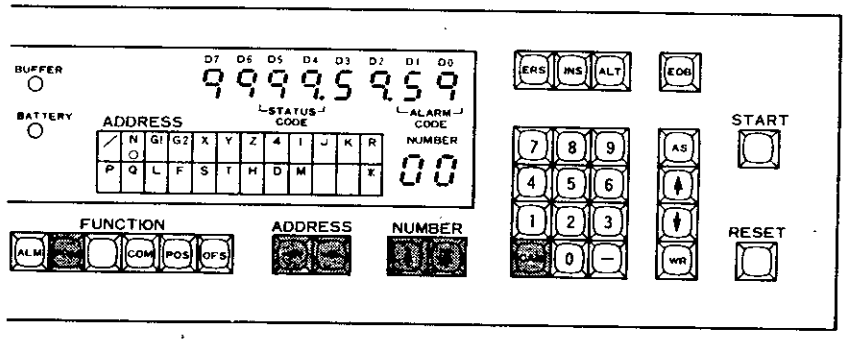
   Operation time is displayed in hours, minutes, and seconds. Operation time means the total cumulative time while CYCLE START lamp is on. It is not cleared by turning off power.

3. To reset the display, depress the PRM key and CAN key simultaneously.

4.3.11 PARAMETER WRITING FOR MIRROR-IMAGE AXIS

The mirror-image axis is designated by parameter. The Mirror-Image function on the designated axis is effective when M95 (Mirror-Image ON) command is given. The procedure for designation of mirror-image axis is as follows.

1. Display the data of parameter No. 91 by using the procedure in 4.3.8 Parameter Display. Never rail to select the address of the axis to be displayed.



(Operation Time: 9999 Hours 59 Minutes 59 Seconds)

Fig. 4.3.10

第4·3·9表　パラメーター覧(1/3)

Table 4·3·9 PARAMETER TABLE (1/3)

NO.	ADD-RESS	意味 MEANING	NO.	ADD-RESS	意味 MEANING	NO.	ADD-RESS	意味 MEANING
00		稼動時間表示 Operation Time Display 9999. 時(H) 59. 分(M) 59. 秒(S)	10†	○		30†	○	ピッチエラー補正量倍率 Multiply for Leadscrew Error Compen. '1' = 1倍
			11†	○				
01		未使用 Not Used	12†	○		31†	○	ピッチエラー補正原点番号 Reference Point No. for L. E. C. '20' = Parameter No. 20
			13†	○				
02		ドライラン時早送り速度 Rapid Traverse Rate for Dry Run '0' RAPID, '1' = JOG	14†	○		32†	○	ピッチエラー補正の補正ピッチ Compensational Pitch for L. E. C '1' = 0.001 mm
			15†	○				
03†		ストロークチェック入切 Stroke Check ON-OFF '0' = OFF, '1' = ON	16†	○		33†	○	第1ストロークチェック座標(最大値) First Stroke Check Point (Max) '1' = 0.001 mm
			17†	○	ピッチエラー補正量			
04†		マシンロック時, ストロークチェック入切 Stroke Check ON-OFF for Machine Lock '0' = OFF, '1' = ON	18†	○	Leadscrew Error Compen.	34†	○	第1ストロークチェック座標(最小値) First Stroke Check Point (Min) '1' = 0.001 mm
			19†	○	Values			
05†		ピッチエラー補正 入切 Leadscrew Error Compen. ON-OFF '0' = OFF, '1' = ON	20†	○	'1' = 0.001 mm	35†	○	第2ストロークチェック座標(最大値) Second Stroke Check Point (Max) '1' = 0.001 mm
			21†	○				
06†		サイクルスタート前の原点復帰 Reference Zero Return before Cycle Start '0' = 不要, '1' = 要 Not Required Required	22†	○		36†	○	第2ストロークチェック座標(最小値) Second Stroke Check Point (Min) '1' = 0.001 mm
			23†	○				
07		未使用 Not Used	24†	○		37	○	未使用 Not Used
			25†	○				
08			26†	○		38	○	
			27†	○				
09		RESET時のAグループGコード状態 A Group G code When Resetting '0' = G00, '1' = 保存 Held	28†	○		39†	○	ストロークチェック領域指定 Area Select for Stroke Check (第4·11項領域チェック機能を参照 Refer to 4·11 Stroke Check)
			29†	○				

第4・3・9表　パラメーター一覧 (2/3)

Table 4・3・9 PARAMETER TABLE (2/3)

NO.	ADD-RESS	意味 MEANING	NO.	ADD-RESS	意味 MEANING	NO.	ADD-RESS	意味 MEANING
40†			60	○	バックラッシュ補正開始方向 Backlash Comp. Starting Direction '0' = from (+), '1' = from (−)	70†		GR0 主軸DA出力値 Spindle DA Output '2047' = 10 V
41†								
42†			61†	○	原点復帰方向 Reference Zero Return Direction '0' = to (+), '1' = to (−)	71†	○	GR1, GR1′ 主軸最大回転数 Spindle Max. RPM '1' = 1 RPM
43†		シーケンス制御用パラメータ Parameter for Optional†						
44†		Machine Interface '0' = OFF, '1' = ON	62†	○	原点復帰最終速度 Reference Zero Return Final Speed '1' = 7.5 mm/min	72†	○	GR2, GR2′ 主軸最大回転数 Spindle Max. RPM '1' = 1 RPM
45†								
46†			63	○	ポジションエラー領域 Position Error Zone '1' = 0.001 mm	73†	○	GR3, GR3′ 主軸最大回転数 Spindle Max. RPM '1' = 1 RPM
47†								
48†	○		64	○	サーボエラー領域 Servo Error Zone '1' = 2^nパルス (Pulses)または2^nパルス (Pulses) (機種による) Depend on the Control	74†	○	GR4, GR4′ 主軸最大回転数 Spindle Max. RPM '1' = 1 RPM
49†	X	システム番号 System Number						
50†			65	○	仕様指定用パラメータ (バイナリー) Designation of Optional Features (Binary)	75	○	切削送り最高速度 Max. Feedrate '1' = 1/16 mm/min
51†								
52†			66†		G12～G15 R指定区間送り速度 R Zone Feedrate '1' = 1/16 mm/min	76†		G76シフト方向 Shifting Direction '1' = X+, '2' = X−, '4' = Y+, '8' = Y−
53†		シーケンス制御用タイマ定数 Timer Constant for Optional†						
54†		Machine Interface 'N' = (16n − 16) msec	67†	○	手動ハンドル送り最大速度 Max. Feedrate for Handle '1' = 7.5 mm/min	77†		補正Cコーナ円弧逃りスキップ値 '1' = 最小設定単位 (1倍) Least Input Increment (×1)
55†								
56†			68		切削送り時定数 Time Constant for Feed 'N' = (32n + 32) msec	78		G90/G91 イニシャルセット Initial Set '0' = G90, '1' = G91
57†								
58†	○	未使用 Not Used	69		切削送り速度バイアス Velocity Bias for Feed '1' = 2 Kpps	79		G92 表示プリセット入切 Display-preset ON-OFF '0' = OFF, '1' = ON
59†	○							

第4・3・9表　パラメーター一覧 (3/3)

Table 4・3・9 PARAMETER TABLE (3/3)

NO.	ADD-RESS	意味 MEANING	NO.	ADD-RESS	意味 MEANING
80		EIA/ISOの自動判別入切 - Auto Select '0' = OFF, '1' = ON	90	○	バックラッシュ補正量 Backlash Value '1' = 0.001 mm
81		TVチェック入切 TV Check ON-OFF '0' = OFF, '1' = ON	91	○	ミラーイメージ軸指定 Mirror Image Axis ON-OFF '0' = OFF, '1' = ON
82		EIA/ISOコード指定 Code Designation '0' = EIA, '1' = ISO	92†	○	外部減速速度 (RPG) Traverse Rate Clamp Speed (RPG) '1' = 7.5 mm/min
83†		MM/INCH指定 Designation '0' = MM, '1' = INCH	93†	○	外部減速速度 (FG) Traverse Rate Clamp Speed (FG) '1' = 1/16 mm/min
84		タッチブザー入切 Touch Buzzer ON-OFF '0' = OFF, '1' = ON	94	○	早送り速度 Rapid Traverse Rate '1' = 7.5 mm/min
85		未使用 Not Used	95	○	早送り加減速定数 Accel./Decel. Time Const. for RT '1' = 125/8 mm/sec²
86		高速原点復帰入切 Rapid Reference Return ON-OFF '0' = OFF, '1' = ON	96†	○	原点復帰クリープ速度 Reference Zero Return Approaching Speed '1' = 7.5 mm/min
87		送り指令単位 Feedrate Designation '0' = F4.0, '15' = F4.1 mm/min	97†	○	原点復帰最終距離 Reference Zero Return Final Stroke '1' = 0.001 mm
88		入力指令10倍入切 Command Data × 10 ON-OFF '0' = OFF, '1' = ON	98	○	MF, SF, TF, BF 送出遅れ時間 SFb 送出遅れ時間 Delay Time for MF, SF, TF, BF, and SFb '1' = 1 msec
89		未使用 Not Used	99†		メモリポインタ表示 Memory Pointer Display '1' = 1 ch

備考 1)
Notice 1)

備考 NOTICE

1) No. 00および99は表示専用であり，書込みはできません。表示は常時可能です。

No. 00 and 99 are for display only. These cannot be written, but can be displayed at anytime.

2) No. 01～98はSYSTEM No. スイッチによってインタロックされており，同スイッチを '1' の位置にして書込みを行ないます。

No. 01 to 98 are interlocked by SYSTEM No. switch, and can be changed only when the position of SYSTEM No. switch is '1.'

3) † 印の付されたパラメータはオプションです。
Parameters with † mark are optional.

4) ADDRESS部の○は軸の指定を要するパラメータです。

Parameter with ○ mark in ADDRESS requires the selection of axis.

Parameter No. 91	Meanings
"0"	Mirror-image axis OFF
"1"	Mirror-image axis ON

2. Write "1" in order to specify the displayed axis as the mirror-image axis, following the operation procedure in 4.3.9 Parameter Writing. The Mirror-Image function is on when M95 command is given and off at M94 command.

NOTES:

- Mirror-image axis can be specified for X, Y, Z, or ($\alpha^{\dagger}$) axis.
- For setting the MIRROR IMAGE AXIS selector switch†, refer to the table below for the combined results of setting parameter No. 91 and the selector switch.

Table 4.3.11

Parameter No. 91	MIRROR IMAGE AXIS Selector Switch	Resultant ON/OFF state of Mirror-Image Axis
"0" = OFF	OFF	OFF
"0" = OFF	ON	ON
"1" = ON	OFF	ON
"1" = ON	ON	ON

Note: Resultant ON/OFF state cannot be displayed on the display.

4.3.12 ALARM AND STATUS CODE DISPLAY

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

NOTES:

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

  Refer to 8.5 Trouble Causes and Remedies.
- Status code will be displayed, if any, simultaneously with alarm code.
- Where the SYSTEM NO. switch is set at other than zero, the set number is shown on the extreme left of the universal display.
- I/O signal will replace alarm code by operating the NUMBER key. Alarm code will be displayed again by depressing the ALM key again. See 4.3.13 Display of I/O Signals.

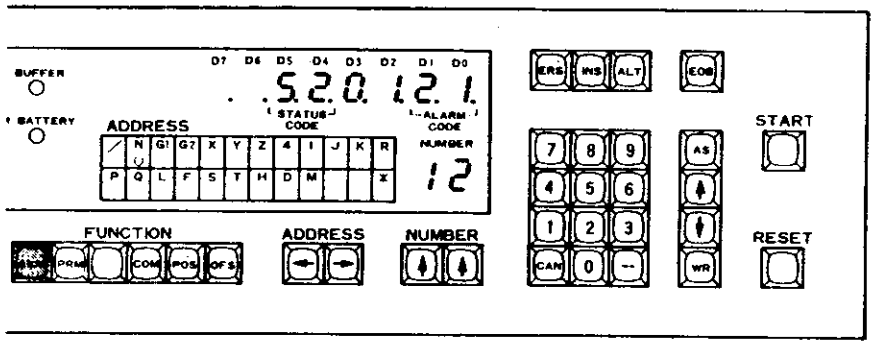


Fig. 4.3.12

Table 4.3.12.1 List of Alarm Codes and Status Codes

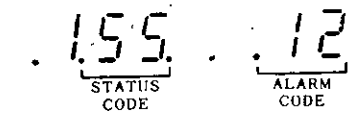
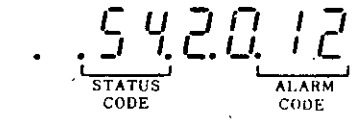
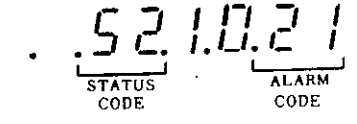
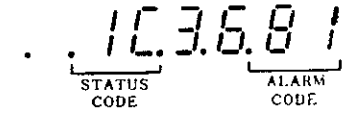
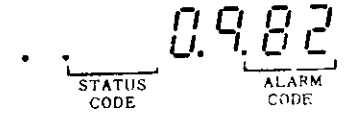
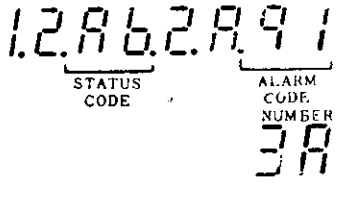
Alarm Code	Causes	Alarm Code	Causes
11	Excessive temperature rise in the panel.	□27	Positioning error.
□12	Tape/Memory horizontal parity error.	28	Machine unready.
13	Tape vertical parity error.	31	Servo power unsupplied.
□14	Format error.	32	Control unit unready.
□15	Data error.	33	Emergency stop.
16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

Status Code	Status	Remarks
□51	Performing M-, S-, T-, and/or $B^{\dagger}$-function.	-
52	Distributing pulses. Dwelling.	-
□53	Performing M-, S-, T-, and/or $B^{\dagger}$-function and distributing pulses.	51 + 52
54	Reading tape.	-
□55	Performing M-, S-, T-, and/or $B^{\dagger}$-function and reading tape.	51 + 54
56	Distributing pulses and reading tape.	52 + 54
□57	Performing M-, S-, T-, and/or $B^{\dagger}$-function distributing pulses and reading tape.	51 + 52 + 54
58	Waiting for canned cycle's FIN signal.	-

Notes:

1. □ of alarm code is filled with a digit indicating the axis whose error is detected. 1: X-axis, 2: Y-axis, 4: Z-axis, (8: $\alpha^{\dagger}$-axis)
   Where the error is detected in more than one axis, total of axis codes is shown.
2. For alarm code "12," additional two digits indicate the memory IC number.
3. □ of alarm code "14," "15" shows the detailed error causes.
4. □ of status code is filled with a digit indicating M, S, T, or B function which is being executed.
   1: M, 2: S, 4: T, (10: $B^{\dagger}$)
   Where the error is detected in more than one function, total of function codes is shown.

Table 4.3.12.2 Display Example of Alarm Code Combined with Status Code

Alarm and Status Codes on Universal Display	Meanings
	Horizontal parity error (alarm code: "12") occurs while executing M-function and reading tape (status code: "55" = "51" + "54") in the TAPE mode.
	Horizontal parity error (alarm code: "12") occurs while reading tape memory (status code: "54") in the MEM mode. Defective IC number is 20.
	Overtravel (alarm code: "21") of Y (2) and α (8) axes (Total 10) occurs in the automatic operation.
	Operation cannot be continued due to erroneous operation of CPU (alarm code "81"). The active address is IC 36.
	Diagnostics for memory detect memory error (alarm code "82). Defective ROM number is "09."
	Contents of system memory are different from those of source tape (alarm code "91") during collating operation with SYSTEM NO. switch set at 2. Memory address is "12AB," memory contents, "2A," and tape contents "3A."

4.3.13 DISPLAY OF INPUT/OUTPUT SIGNALS

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

Procedure is as follows:

1. Depress the ALM key, and it lights up.

   Alarm and/or status code will be shown on the universal display.

2. Select diagnostic number of I/O signals using NUMBER key and ADDRESS key.

   Then I/O signal "1" or "0" replaces the displayed alarm code on the universal display. Operation of NUMBER key makes I/O signal display mode automatically.

NOTES:

- For diagnostic number of each signal, refer to Table 8.5.3 List of Input/Output Signals.
- Signal is shown by "1" or "0."

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

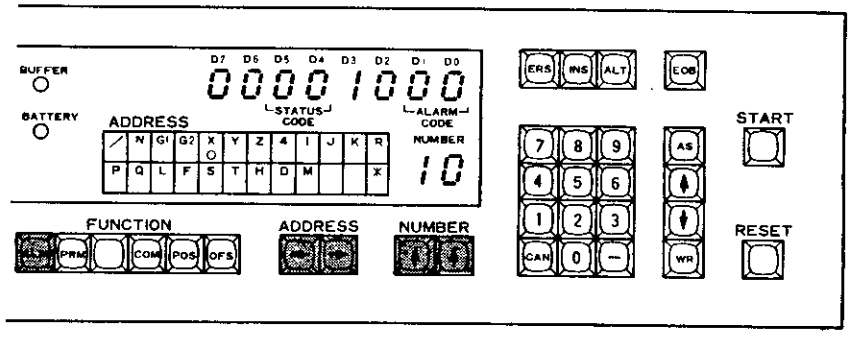
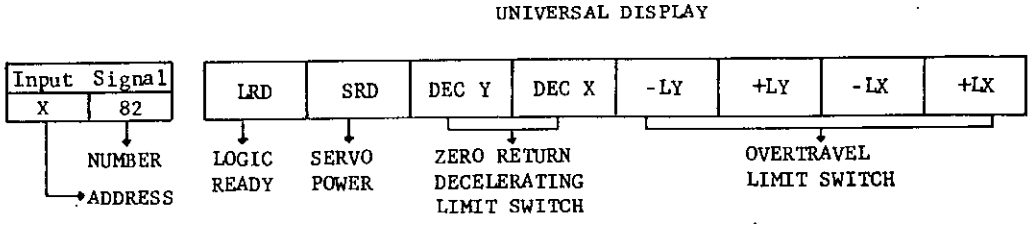


Fig. 4.3.13.1



Selecting address X and number "82" causes input/output signals shown above to be displayed by "1" or "0" on the universal display.

Fig. 4.3.13.2 Example of Input/Output Signal Display

4.3.14 ADDRESS SEARCH

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

1. Set the MODE SELECT switch to TAPE, MEM, or EDT position.
2. Depress the COM key, and it is lit.
3. Depress the RESET key. Then the LABEL SKIP lamp will be on, and the memory is rewound.
4. Set the address character by ADDRESS key.

   In the EDT mode, set the NUMBER key at other than "00."
5. Set the data through the DATA keyboard.

   The display shows the data as it is entered, flickering the last significant digit. To correct the data just entered, depress the CAN key and key in the correct one.
6. Depress the AS key to start address search.

   When the block containing the selected address data is encountered, BUFFER LAMP lights up indicating that search operation is completed.
7. To stop the search operation, depress the RESET key.
8. In the MEM and EDT modes, block-by-block search is allowed by using [↓] and [↑] keys.

NOTES:

- In the TAPE and MEM modes searched block of data enters the buffer register. Coordinate values are modified with tool offset values.
- In the EDT mode, searched block enters edit-buffer register without being modified with tool offset value and so on. But BUFFER lamp is on likewise in Tape and MEM modes.
- Leading zeros may be suppressed for all address characters including N in address search operation. For example, N12 means N012.
- All commands including modal one except the block to be searched are ignored. Only searched block data enters the buffer register.
- Operation begins with searched block by depressing CYCLE START pushbutton after address search in the TAPE or MEM mode has been accomplished.
- Reset the control before Cycle Start in MEM mode after Address Search in the EDT mode.

  The operation starts from the first address of the memory.

4.3.15 TV CHECK (VERTICAL PARITY CHECK)

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

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

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

4.3.16 CURRENT POSITION DISPLAY UNIT†

The control may be provided with a current position display unit for three axes (X, Y, Z) or for four axes (X, Y, Z, 4). The unit may be mounted on the panel front or separately as remote display unit.

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

- Depress the RESET button causes the display to be zero.
- The indication is not affected by the G92 command.
- The indication is updated even with the MACHINE LOCK switch on, but not changed with the DISPLAY LOCK† switch on.
- Even if servopower is off by pressing the EMERGENCY STOP button, the indication follows tool movement.

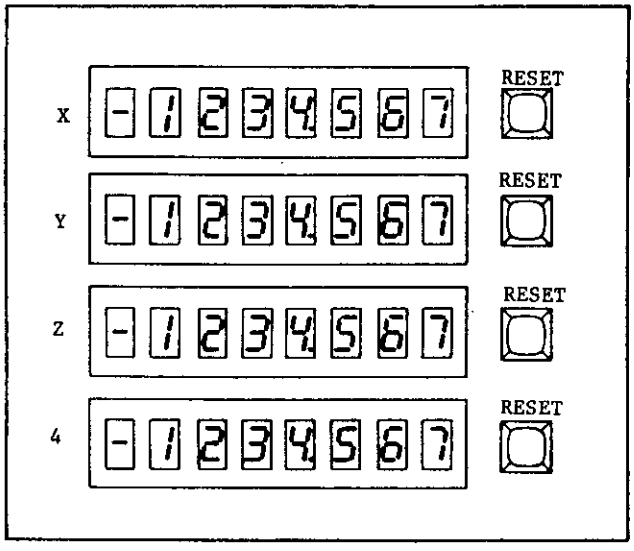


Fig. 4.3.16

4.4 STORING TOOL OFFSET VALUES FROM NC TAPE†

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

Tape format for tool offset values is:

LABEL *

H1 X··· *
H2 X··· *
H3 X··· *
D4 X··· *
D5 X··· *
.
.
.

ER (or %) · · · · · Rewind stop code

Notes:

1. Both address H and D can be used for the format of tool offset values.
2. Tool offset values are placed after X.

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

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

4.5 PART PROGRAM STORAGE†

4.5.1 STORING PART PROGRAM FROM NC TAPE†

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

Table 4.5.1

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

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

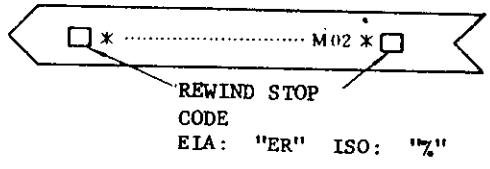


Fig. 4.5.1.1

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

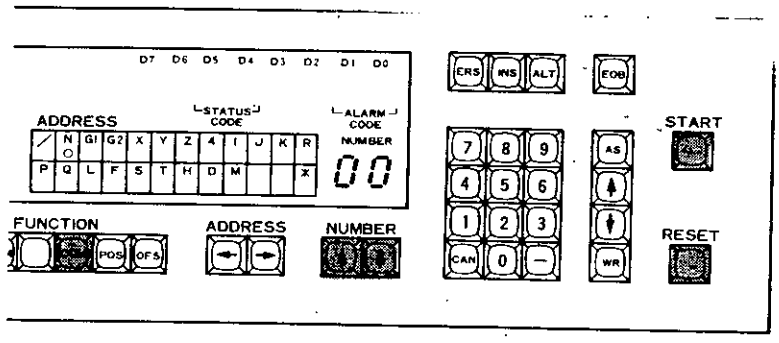


Fig. 4.5.1.2

Follow the procedure below to store the punched tape data.

1. Set MODE SELECT switch to EDT.
2. Depress the RESET key. Then memory is rewound and LABEL SKIP lamp is on. This is a preparation to store part program orderly from the first address of the memory.
3. Depress the COM key, and it lights up. Any FUNCTION key except OFS is allowed to be set. However, use COM key normally.
4. Set the NUMBER at other than "61" and "62," using the NUMBER key.
5. Set the punched tape onto the tape reader unit. Be sure that Label Skip function is effective.
6. Depress the START key.

   Then tape reader starts and part program is stored into the memory. It automatically stops when rewind stop code has been read.

4.5.2 PART PROGRAM MODIFICATION FROM NC TAPE†

To modify the block already stored in the middle of the program, use the following procedure.

The already stored data in the block to be modified is deleted by storing a new data.

1. Set the MODE SELECT switch to EDT.
2. Search the block in which the new program will be stored. See 4.3.14 Address Search.
3. Depress the COM key, and it lights up.
4. Set the NUMBER at other than "61" and "62" using the NUMBER key.
5. Apply the new tape onto the tape reader unit.

   Be sure that Label Skip function is effective.
6. Depress the START key.

   New tape data is stored into memory beginning from the searched block continuously until a rewind stop code is read.

NOTE:

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

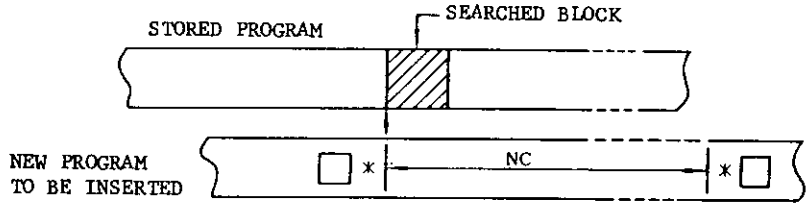


Fig. 4.5.2

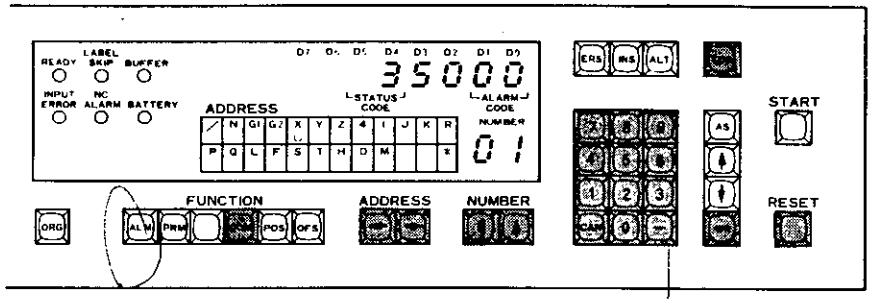


Fig. 4.5.3

4.5.3 STORING PART PROGRAM FROM MDI†

1. Set the MODE SELECT switch to EDT.

2. Depress the RESET key.

   Memory is rewound, and LABEL SKIP lamp is on. It indicates that the control is ready to store the programs from the first address in the memory.

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

4. Set the NUMBER at other than "00."

   With the NUMBER set at "00" in the EDT mode, free address setting is not allowed because the addresses already stored in the buffer register are scanned.

5. Select the address character using the ADDRESS key.

6. Key in the data through the DATA keyboard.

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

7. Depress the WR key

   The data just keyed in is registered as a new command data, and flickering stops.BUFFER lamp lights up.

8. Repeat steps 4 through 6 until a block of data has been written.

9. Check the input data, referring to 4.3.1 Display of Command Data. Then depress the EOB key.

   The data of a block in buffer register is stored into memory, and BUFFER lamp goes out.

10. Repeat steps 4 through 9 to complete the part program storage. Store the block including "M02" or "M30" command as the final block of the part program.

11. Check the program stored after depressing the RESET key.

    See 4.6.1 Display of Stored Part Program.

12. Correct the data, if necessary, referring to 4.6.2 Editing Stored Part Program.

13. Depress the RESET key again to rewind memory. LABEL SKIP lamp will light up.

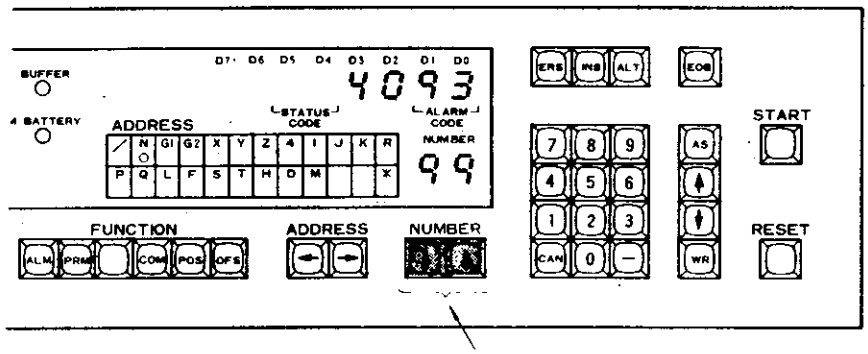


Fig. 4.5.4

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

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

EXAMPLE:

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

  Depressing both NUMBER keys simultaneously two times makes the NUMBER "99."

4.6 EDIT †

4.6.1 DISPLAY OF STORED PART PROGRAM †

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

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

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

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

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

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

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

   Notes:

   - Depressing [+] ADDRESS key at the block end causes the first address of the block to appear again.
   - Depressing [-] ADDRESS key makes the scanning of the addresses from the last.
   - Depressing both ADDRESS keys always moves address indicator to the first address of the block.

8. Depress the [↓] key to read out the next block. Check the command data repeating steps 6 and 7.

9. Depressing [↑] key calls the preceding block. Selected block can be searched with the AS key.

10. On completion of reading out the memory and checking the command data, depress the RESET key to rewind the memory.

4.6.2 EDITING STORED PART PROGRAM †

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

ERS (erase) . . . To delete block

INS (insert) . . . To insert block

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

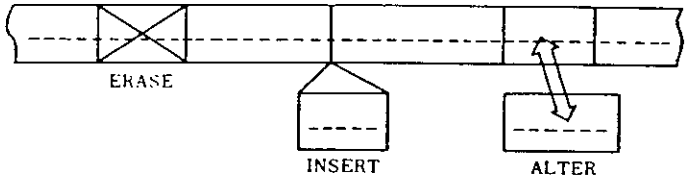


Fig. 4.6.2.1

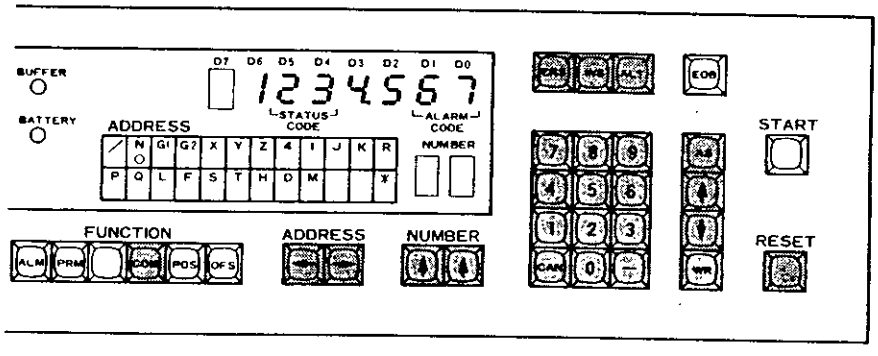


Fig. 4.6.2.2

· Deleting part program block (ERASE key)

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

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

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

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

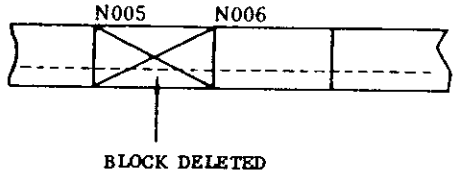


Fig. 4.6.2.3

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

· Adding part program block (INSERT key)

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

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

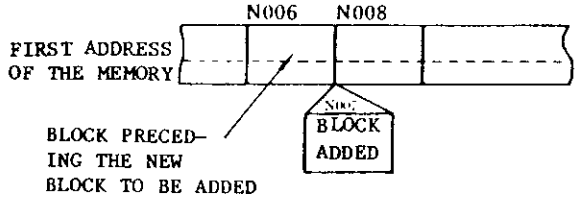


Fig. 4.6.2.4

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

3. Insert the data of a block according to steps 3 and 4 in 4.3.2 Writing Command Data by MDI.

4. Depress the INS key again.

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

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

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

· Modifying part program block (ALTER key)

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

   BUFFER lamp will be lit.

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

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

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

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

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

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

NOTE: More than one G-code can be commanded in a block. Depressing the CAN key deletes all the G-codes in a block.

4.6.3 OUTLINE OF EDIT OPERATION†

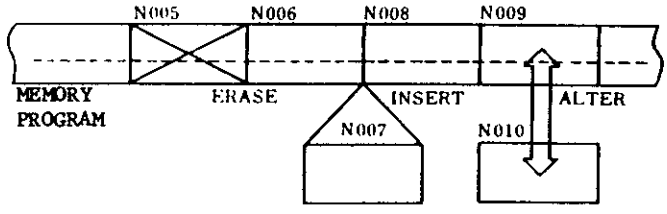


Fig. 4.6.3

Table 4.6.3

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

Notes:

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

4.7 PUNCHOUT OPERATION†

4.7.1 TAPE PUNCHER†

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

Tape Puncher

Type FACIT 4070

Punching Speed: 75 characters/second

Dimensions:

432 wide x 220 deep x 198 high (mm)

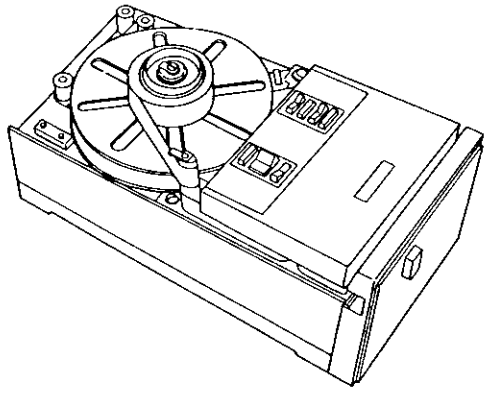
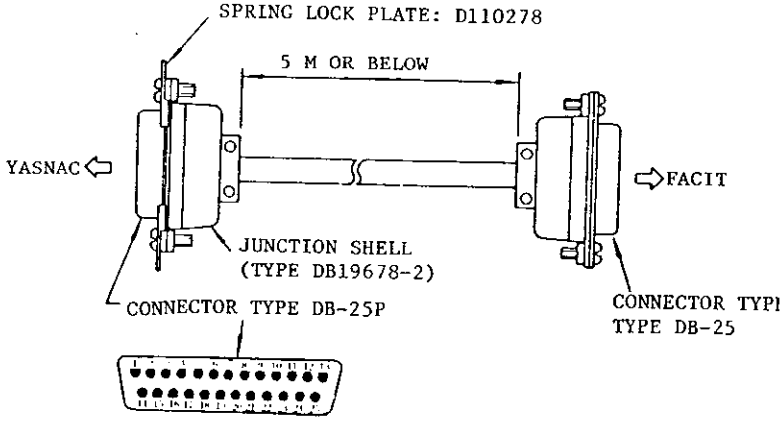


Fig. 4.7.1.1



CONNECTIONS BETWEEN THE CONTROL AND THE PUNCHER

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

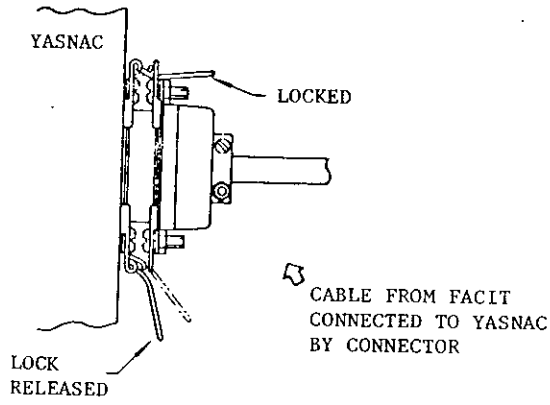


Fig. 4.7.1.2

Operation of FACIT Punchers

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

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

Switch is set with position indicator placed at ▲.

Fig. 4.7.1.3

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

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

4. Turn off the control.

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

5. Connect AC power supply to FACIT 4070.

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

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

8. Turn on the control.

   The FACIT puncher is ready to operate.

4.7.2 PUNCHOUT OF NC TAPE†

1. Set the MODE SELECT switch to EDT.

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

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

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

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

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

6. Depress the START key.

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

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

   To resume the operation, take the steps from 1.

NOTES:

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

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

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

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

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

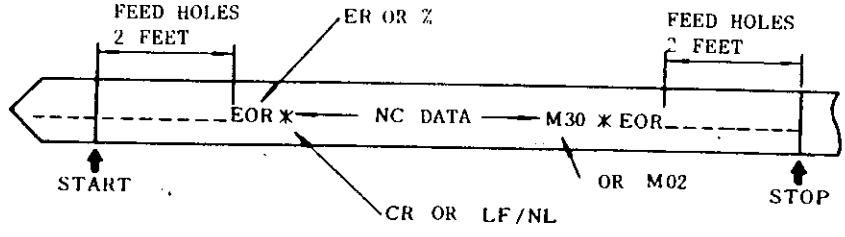


Fig. 4.7.2

4.7.3 PUNCHOUT OF TOOL OFFSET VALUE†

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

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

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

   To resume the operation, take the steps from 1.

4.7.4 SEQUENCE OF TAPE DATA STORING AND PUNCHING OPERATION†

Select EDT mode.

↓

Depress RESET key.

↓

Selection of FUNCTION keys.

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

↓

Set the NUMBER at:

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

↓

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

Fig. 4.7.4

4.8 COLLATING OF STORED PROGRAM AND OFFSET VALUE

4.8.1 COLLATING OF STORED PROGRAM†

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

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

   Memory is rewound and LABEL SKIP lamp is illuminated.
3. Depress the COM key, and it will light up.
4. Set the NUMBER at "62" which is the function number for collating operation.
5. Set the NC tape to the tape reader.

   Be sure that LABEL SKIP lamp remains on.
6. Depress the START key.

   The tape reader starts reading, and the memory contents are collated with the tape data. When it reads rewind stop code, it automatically stops.
7. If disagreement with NC tape is detected, INPUT ERROR lamp lights up and tape reader stops. Alarm code "18" is displayed.

NOTES:

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

4.8.2 COLLATING OF STORED OFFSET VALUE†

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

1. Set the MODE SELECT switch to the EDT.
2. Depress the RESET key, and LABEL SKIP lamp will go on.
3. Depress the OFS key, and it will be on.
4. Set the NUMBER at "62" which is the function number for collating operation.
5. Set the source tape to the tape reader.

   Be sure that LABEL SKIP lamp remains on.

6. Depress the START key with the OFS key held in.

   Tape reader starts reading, and memory contents are collated with the tape data. It automatically stops when rewind stop code is encountered.

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

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

4.9 OUTLINE OF OPERATION IN THE EDT MODE†

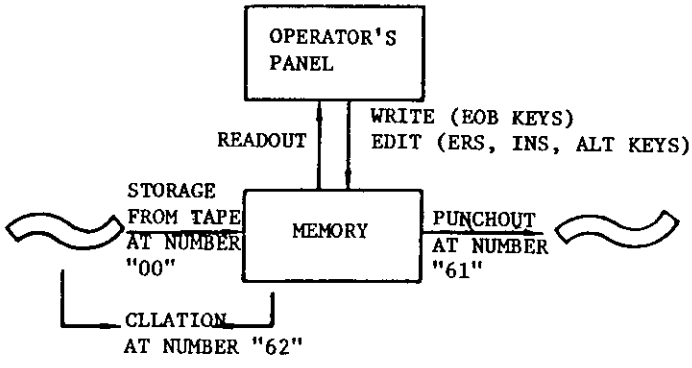


Fig. 4.9.1

Table 4.9.1

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

4.10 LEADSCREW ERROR COMPENSATION †

Errors in leadscrew pitch can be compensated by using this option which provides storage for the compensation data. For this operation, the control must be provided with the return to reference zero option.

1. Storage for leadscrew error compensation

   20 Sets of parameters (parameter No. 10 to No. 29) are available for storing the compensation values. Since these parameters can be designated with the axis, the compensation values corresponding to 20 points for each axis can be stored.

No.	Address	Meanings
10	X Y Z α	Comepnsation values.
11	X Y Z α	
. . .		
29	X Y Z α	

Fig. 4.10.1

2. Compensation values

   Table 4.10.1 shows the meaning of the contents of parameter No. 10 to No. 29.

Table 4.10.1

	Compensation Values
Unit	"1" = 0.001 mm
Setting Range	0 to ±133
Absolute/Incremental	Absolute value of error

Difference between two adjacent compensation points (e.g. increment values) must be less than plus or minus 7.

3. Measuring of compensation amount

   A description of the method of measuring compensation values for X axis is given below. For the other axes, the same procedure should be taken.

A. Determine the value of compensation interval by dividing the effective stroke of X axis into twenty equal parts. (The effective stroke of X axis is determined with the positions of overtravel limit switches on the negative and positive direction end of X axis.

B. Return the machine to reference zero point by manual operation.

C. Compensation points can be determined both in negative and positive directions with the interval of $1/2P_L$, P_L, P_L ··· .

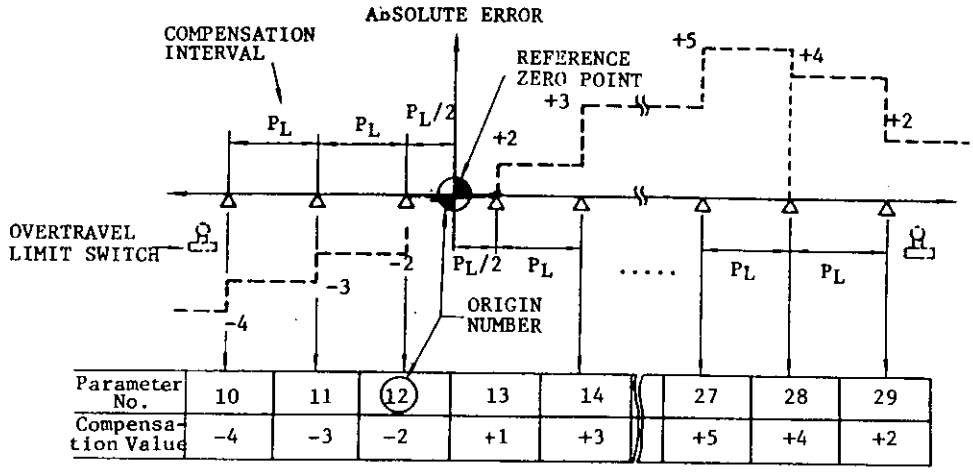
D. Assign "10" to the extreme end of compensation points in the negative direction of X axis and number the remaining points from 11 to 29 to the positive direction of X axis. That is, for simplification, this compensation point numbers are corresponded to the parameter numbers for storing the compensation values.

E. Move the machine by MDI operation by P_L intervals and measure the absolute error values at each compensation position using RASER measuring devices.

F. Arrange the measured values so that the error at the reference zero point is to be zero. (Refer to Fig. 4.10.2.)

NOTES:

- Reference zero point must be located between the compensation points No. 10 and No. 29.
- The difference between two adjacent compensation points (e.g. increment values) must be less than plus or minus 7.



Parameter No.	10	11	12	13	14	27	28	29
Compensation Value	-4	-3	-2	+1	+3	+5	+4	+2

Fig. 4.10.2

4. Writing of compensation values

Set "0" to the parameter No. 05 before writing compensation values.

A. Write the compensation values for each axis to the parameter No. 10 to No. 29.

B. Write the origin number to the parameter No. 31.

Origin number = the next parameter No. of reference point in the negative direction

In the case of Fig. 4.10.2, write "12" to the parameter No. 31.

C. Write the compensation interval (P_L) to the parameter No. 32. In the actual case, the figure to be written should be "2" or more.

5. Multiplier of compensation values

A. When the measured values are directly written to the parameters No. 10 to No. 29, write "1" to the parameter No. 30. The compensation is performed after the written values have been multiplied by "one."
Note that the compensation will not be performed if "0" is written to the parameter No. 30.

B. If the compensation values are too big figures to be written, write the figures divided by "n" to the parameter No. 30. The compensation is performed after the written values have been multiplied by "n." "n" must not be greater than "5" ($n \leq 5$).

6. Leadscrew error compensation operation

A. The following conditions must be satisfied for the operation of leadscrew error compensation.

- The contents of parameter No. 05 are "1" (Leadscrew error compensation on)
- Manual or automatic return to the reference zero has been performed for the pitch error compensation axis (not set by parameter No. 30 to "0") after power on.

Note: When the contents of parameter No. 06 are "1," if CYCLE START button is depressed before performing reference zero return, the alarm code "23" will be displayed.

B. When the conditions above described are satisfied, the leadscrew error compensation is effective to any motions in automatic and manual operations.

NOTES:

- When the optional fourth axis is a rotary axis, the compensation is not effective to the fourth axis, but the fourth axis is a linear axis, it is effective.
- When the fourth axis is a rotary axis, even if the contents of parameter No. 06 are "1," the CYCLE START button can be depressed without return to the reference zero of rotary axis.

4.11 STROKE CHECK

Stroke Check is used during automatic operation to see if tool positions at each block end will be out of the specified prohibit area. If the tool is to be in the prohibit area at a block end, INPUT ERROR lamp lights and the machine stops before executing a block command. (Alarm code "15" is displayed.)

1. Writing Area

Specify one or two areas outlined by coordinate values written into parameter numbers 33 to 36 in order to check tool positions. Of two areas, one is called first area, and the other, second area.

· First Area

Write a distance with a sign from the reference zero point to the plus-end of the area on a selected axis using parameter No. 33. Write a distance with a sign from the reference zero point to the minus-end of the area on the axis using parameter No. 34.

· Second Area

Write a distance with a sign from the reference zero point to the plus-end of the area on a selected axis using parameter No. 35. Write a distance with a sign from the reference zero point to the minus-end of the area on the axis using parameter No. 36.

NOTE: Plus-end of the area may be in the minus side facing the reference zero point and vice versa.

Table 4.11.1

	Parameter No.	Address	Meanings
First Area	33	X	First stroke check point (Max.) "1" = 0.001 mm
		Y	
		Z	
		4	
	34	X	First stroke check point (Min.) "1" = 0.001 mm
		Y	
		Z	
		4	
Second Area	35	X	Second stroke check point (Max.) "1" = 0.001 mm
		Y	
		Z	
		4	
	36	X	Second stroke check point (Min.) "1" = 0.001 mm
		Y	
		Z	
		4	

Values for setting parameter numbers are the distance from the reference zero point with a sign. "±" = ±0.001 mm

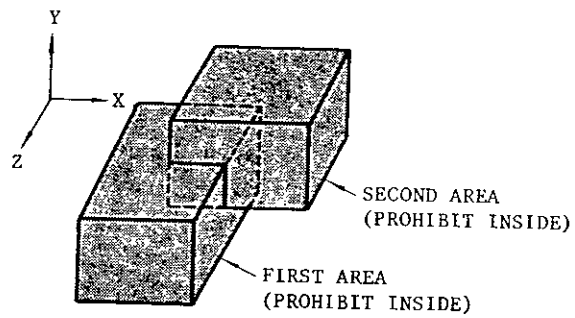
2. Determination of Prohibit Area

Set the parameter No. 39X to determine whether inside or outside of written area is prohibited for programming tool position. If a tool is to be in the prohibit area at a block end, INPUT ERROR lamp lights up.

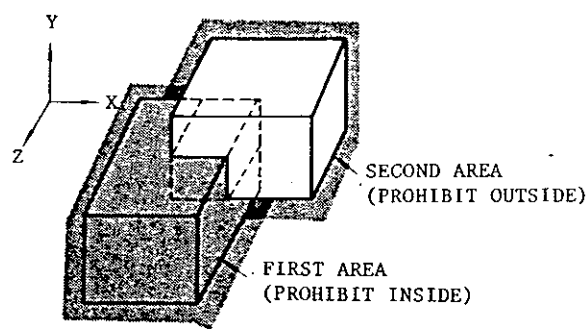
Table 4.11.2

Parameter No. 39X	Prohibit Area	
	First Area	Second Area
"0"	Outside	Outside
"1"	Inside	Outside
"2"	Outside	Inside
"3"	Inside	Inside

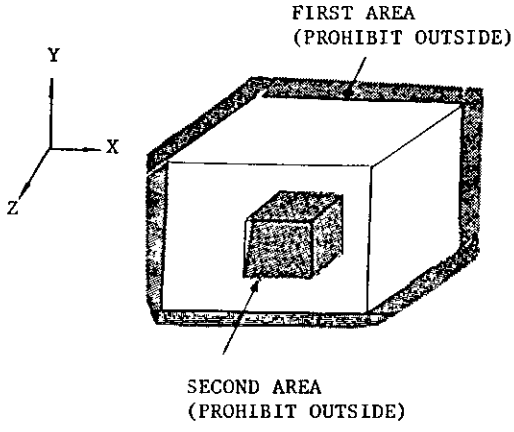
· Inside/Inside Designation
(Parameter No. 39X = "3")



· Inside/Outside Designation
(Parameter No. 39X = "1")



· Outside/Inside Designation
(Parameter No. 39X = "2")



NOTES:

· Boundary line of each axis is contained in the prohibit area.

· Listed below are meanings of values written into parameter No. 39X.

	Inside	Outside
First Area	"1"	"0"
Second Area	"2"	"0"

INHERENT CODE
FOR EACH AREA

EX.

First Area Prohibit Inside		Second Area Prohibit Outside		Data of Parameter No. 39X
"1"	+	"0"	=	"1"

3. Number of Areas

Number of areas in which the tool position is checked can be determined by parameter No. 39Y.

Parameter No. 39Y	Meanings
"1"	Tool position checked only in first area
"2"	Tool position checked in first and second areas

Note: Data of parameter No. 39Y set at "0" cannot execute Stroke Check.

4. Automatic Operation with Stroke Check Function

a. The following conditions make stroke check effective

· Parameter No. 03 = "1": Stroke Check on.

· Manual or Automatic Return to Reference Zero has been executed on each axis after supplying power to the control.
Where parameter No. 06 = "1," operating the Cycle Start displays alarm code "23" before executing Reference Zero Return.

b. Where the Stroke Check is on, it is detected during automatic operation that tool position programmed is in the specified prohibit area, INPUT ERROR lamp lights up, and machine motions stop before executing the block command. (Alarm code "15" is displayed.)

NOTES:

· Where parameter No. 04 = "1," Stroke Check is effective even at Machine Lock ON, if parameter No. 03 = "1."

· Where the 4th axis is for rotating, Stroke Check is disabled on the 4th axis.

· Where the 4th axis is for linear motion, Stroke Check is effective. If the 4th axis is assumed, for example, to be a linear W-axis parallel to Z-axis, the area is defined by the space outlined by X, Y, and W parameters. The check points on W-axis can be freely set, however, the check points on X and Y axes are set on the same ones determined for Z-axis spindle.

· Where the 4th axis is for rotating, Cycle Start is possible without Reference Zero Return operation even if parameter No. 06 = "1."

5. TAPE READER COMPARTMENT

5.1 TAPE READER COMPARTMENT

5.1.1 TAPE FEED AND SYSTEM NO. SWITCHES

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

· TAPE FEED switch

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

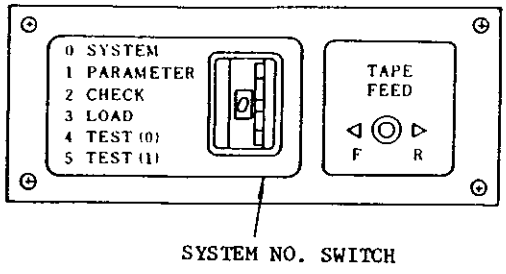


Fig. 5.1.1

· SYSTEM NO. switch

This switch has been fixed at "0" for normal operation and does not need operation. Parameter writing is made with the switch set at "1." For details on its setting, see 4.1.15 TAPE FEED and SYSTEM NO. switches and 4.3.9 Writing Parameters.

5.1.2 TAPE READER

· Light source

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

· Tape reading head and tape feeding part

Phototransistor is imbedded in the tape reading head and covered with glass. Scratch or dust on the glass can create errors in the readout. Make it clean periodically. See 8.1 ROUTINE INSPECTION. Feed holes on the tape should be set to the sprocket of the tape feeder.

· Tape bail

Push up the tape bail magnet to release tape bail, mount the tape, and push down the tape bail slowly. The tape reader will not operate until the tape bail is pushed down.

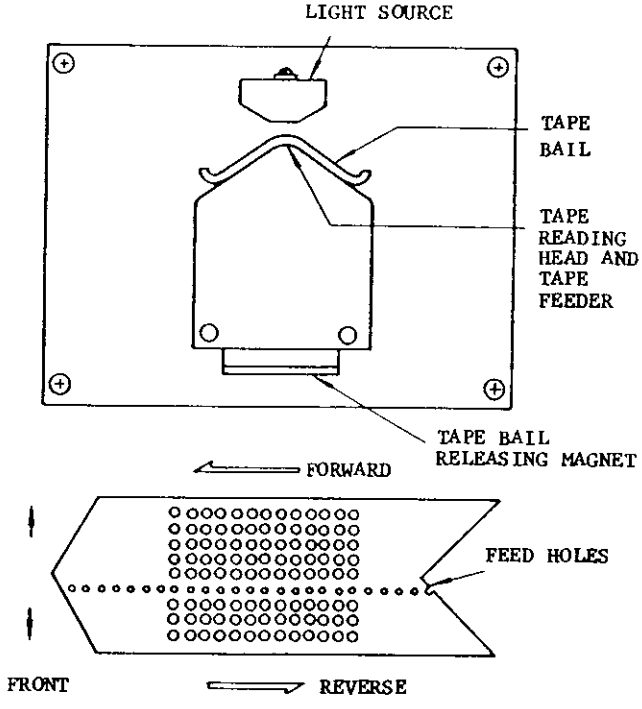


Fig. 5.1.2

5.1.3 TAPE TUMBLE BOX

Tape tumble box is provided below the tape reader to accomodate NC tape. The NC tape is easily taken out by pulling a polyester tape mounted inside the box as shown in Fig. 5.1.3. When the NC tape cannot be taken out, remove screws of tape outlet cover mounted on the lower part of the box periodically referring to 8.1 ROUTINE INSPECTION.

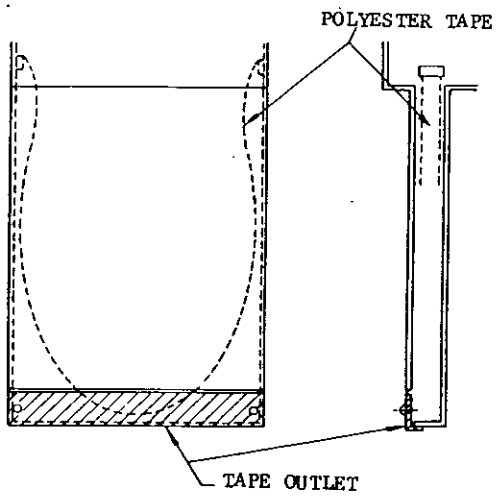


Fig. 5.1.3 Tape Tumble Box

5.2 TAPE REELS†

5.2.1 TAPE REELS†

Available as an option are two types of tape readers according to the tape reels listed below.

Table 5.2.1

Tape reels	Reel diameter	Length of NC tape accomodated in case of 0.0042 inch (0.108 mm) thick tape
6-inch reel	6 inches	262 feet (80 meters)
8-inch reel	8 inches	590 feet (180 meters)

5.2.2 6-INCH TAPE REELS

Place the tension arm on the arm rest when tape reels are not used, or tape is mounted on or removed from the tape reels. To do so removes power from the reel motor and permits the reels to be freely handled.

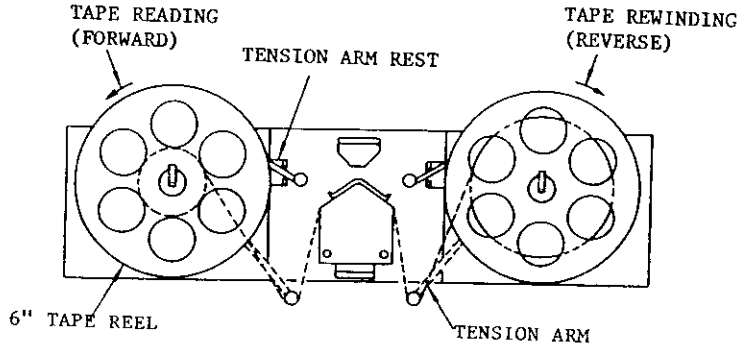
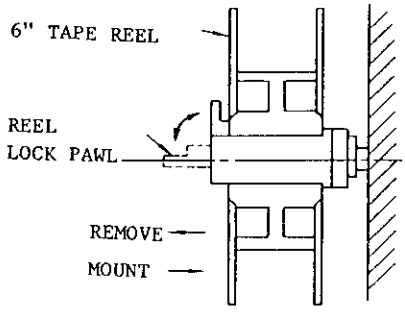


Fig. 5.2.2.1



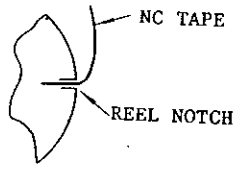


Fig. 5.2.2.2

To mount the NC tape, proceed as follows.

1. Take out the right hand side reel by pulling the reel lock pawl.

2. Wind the NC tape on the removed reel manually after trapping the tape end in the reel notch.

3. Place the reel, which is holding the tape, on the right spindle. Then push the reel lock pawl to its groove to fix the reel.

4. Draw the NC tape approximately 3 feet (1 m) and thread it into the tape reader.

5. Attach the NC tape, which is inserted into the reader, to the left reel.

   a. Take out the left hand side reel by pulling the reel lock pawl.

   b. Wind the NC tape two or three times on the removed reel manually after catching the tape end in the reel notch, so that tape does not loosen.

   c. Place the reel, which is holding the tape, on the left spindle. Then push the reel lock pawl to its groove to fix the reel.

6. Holding the reel not to loosen the tape, detach the tension arm from the arm rest and push it down slowly. Fig. 5.2.2.1 shows the tape properly mounted on the tape reels.

   The tape reader is now ready to run.

NOTES:

- When removing the tape from the reels, the following care should be observed.
  - Place the tension arm on the arm rest holding the reel to be removed.
  - Push up the tape bail of the tape reader before removing the tape from the reel.
- Never fail to be sure that reel spindle lock pawl is fully engaged in its groove before operating the tape reader.

5.2.3 8-INCH TAPE REELS

When tape reels are not used, or tape is mounted on or removed from the tape reels, turn off the reel motor power switch. This action removes power from the reel motor and permits the reels to be freely handled.

Tape readers with 8-inch tape reels are composed of the following:

- Reel motor power switches

  Turning on the reel motor power switch applies power to the reel motor.

- Guide rollers

  Two guide rollers are provided for each reel. They are used to guide the tape to be fed and wound easily.

- Tension arms

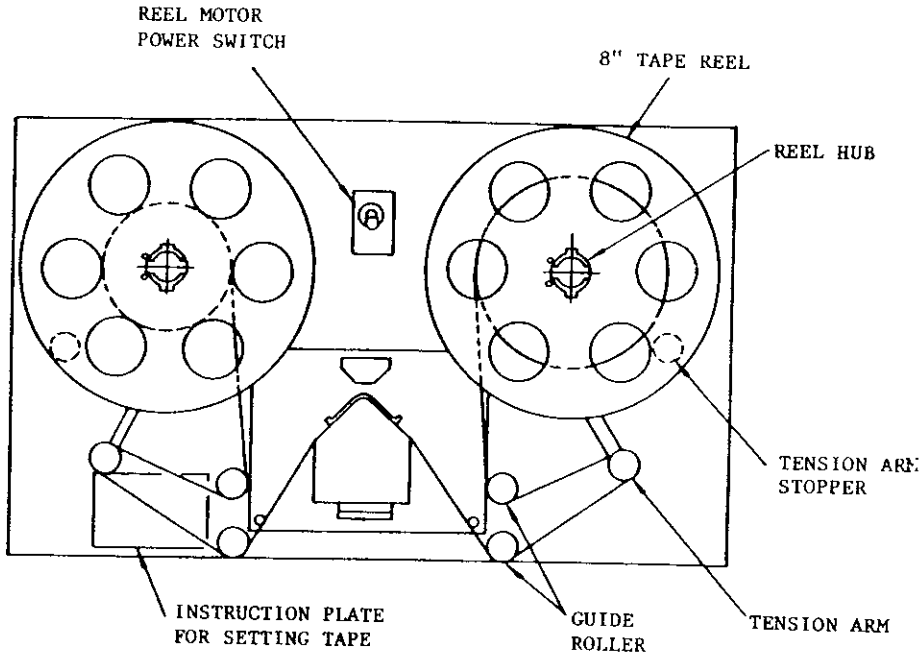


Fig. 5.2.3.1

· Tension arm stoppers

The tension arm is locked on the tension arm stopper when the NC tape is not set on the tape reel.

· 8" Tape reels

8" Tape reels can accomodate 590 feet (180 m) NC tape.

· Reel-hub

Reel hub is used to mount and fix the tape reel at the reel spindle.

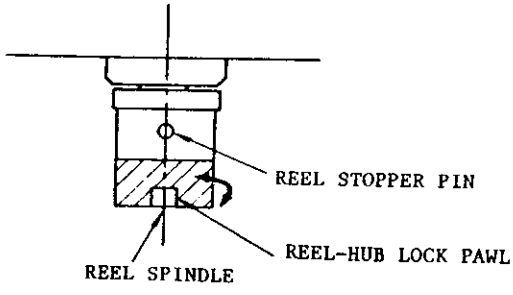


Fig. 5.2.3.2

· Instruction plates

The instruction plate shows that the proper setting of the tape over the guide rollers and tension arms.

To mount the NC tape, proceed as follows.

1. To remove the right reel, turn the reel lock pawl with the reel held until it is caught in the groove in reel bail. Then remove the reel bail, tape core, and inside reel at one time. See Fig. 5.2.3.3.

2. Wind the NC tape on the removed tape core manually after trapping the tape end in the core notch.

   If the core without groove is used, fit the tape end to the core with cellophane-tape.

3. Match the position of reel stopper pin with that of reel lock pawl by turning the pawl. See Fig. 5.2.3.2.

4. Set the inside reel on the reel-hub matching the position of the bail groove with that of hub lock pawl.

5. Mount the core, which is holding the tape, on the inside reel.

6. Place the reel bail on the reel-hub matching the position of the bail groove with that of reel lock pawl. Turn the lock pawl until the pawl is engaged between the two small bosses on the reel bail.

   The right tape reel is properly mounted as shown in Fig. 5.2.3.4.

7. Draw the NC tape approximately 5 feet (1.5 m).

8. Insert the NC tape into the tape reader through the guide rollers, tension arm on the right hand side.

9. Attach the NC tape, which is coming out of the tape reader, to the left reel in the same manner described in steps 1 to 6. Never fail to wind the NC tape two or three times on the left core so that tape does not loosen.

10. Draw the tape coming out of the tape reader over the guide rollers and tension arm following the instructions in the instruction plate for setting tape.

11. Turn the right and left reels so that each tension arm locates at the center of the guide rollers and the tension arm stopper.

    Fig. 5.2.3.1 shows the NC tape properly mounted on both tape reels.

12. Turn on the reel motor power switches.

    The tension arm may move at that instant, however, it will immediately stop.

NOTES:

· When tape reels are not be used, turn off the reel motor power switch.

· To remove the tape from the tape reels, proceed as follows.

  a. Turn off the reel motor power switch.

  b. Remove the tape from the tension arm.

  c. Push up the tape-bail of the tape reader.

· When the reel motor power switch is on, do not turn the tape reel with the NC tape set.

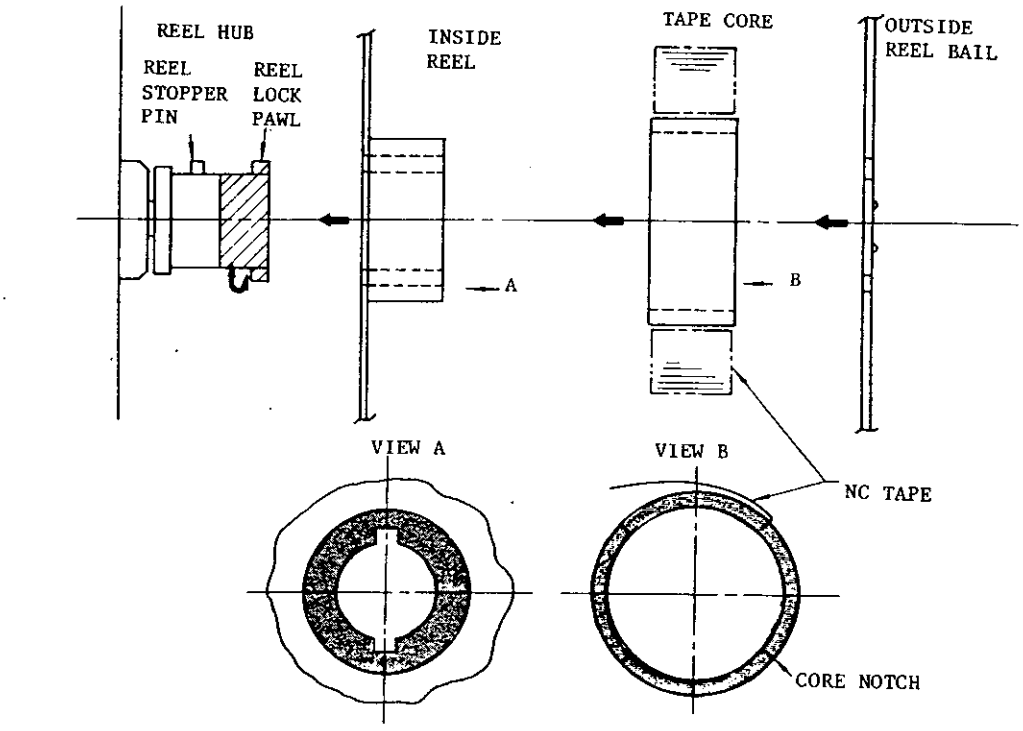


Fig. 5.2.3.3

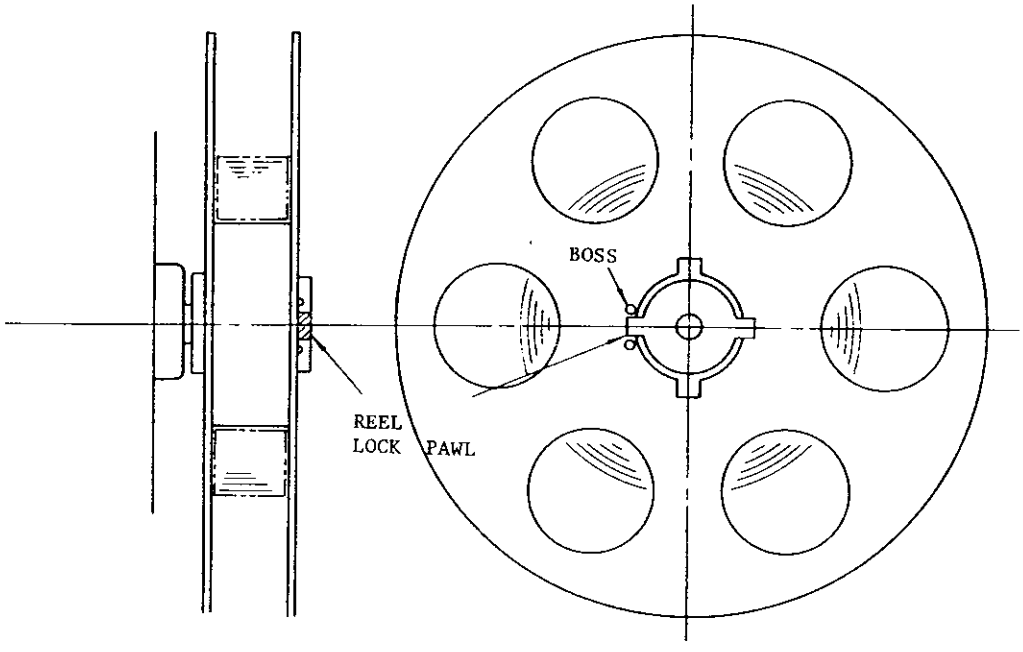


Fig. 5.2.3.4

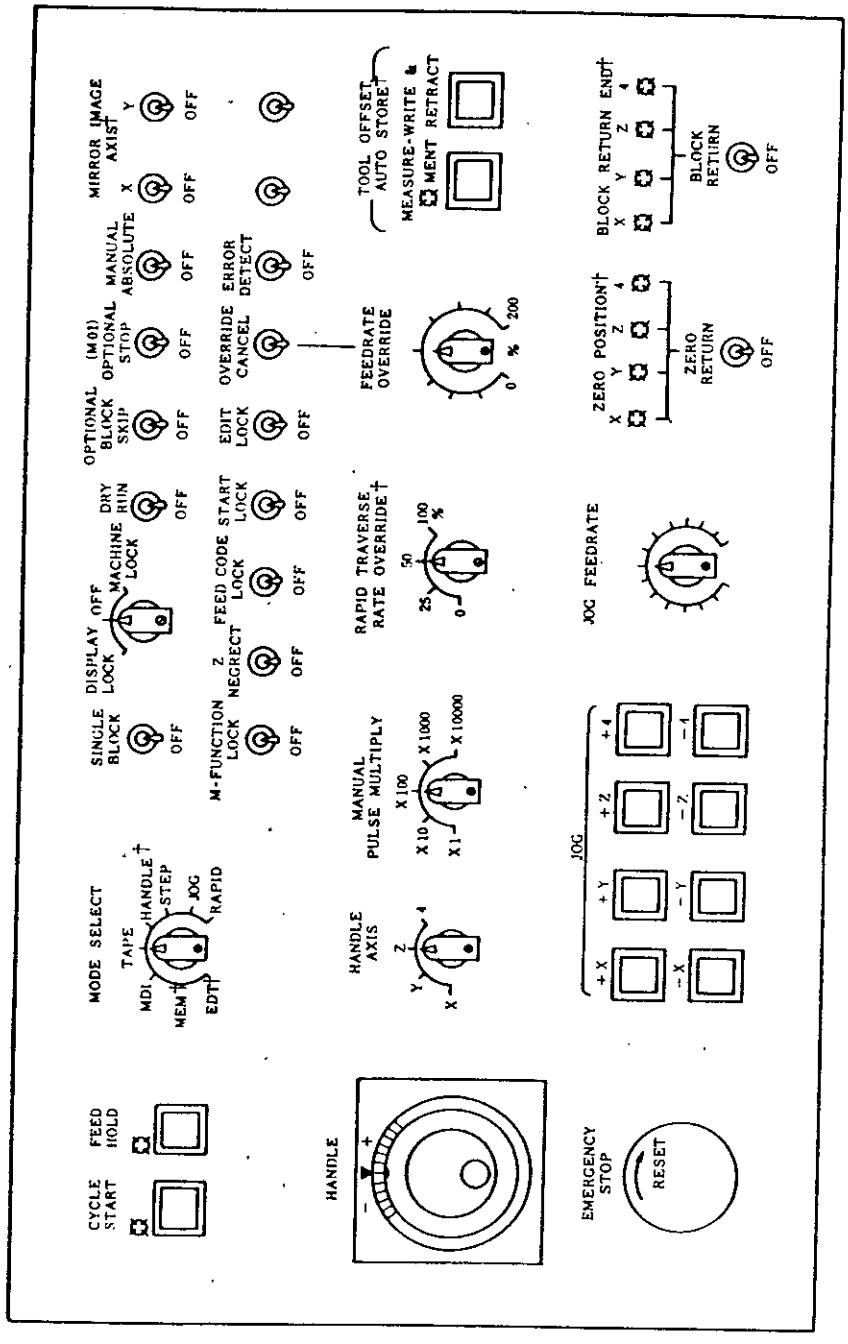


Fig. 6.1.0 Layout Example of Switching Units on Control Station

6. CONTROL STATION FOR MACHINE

6.1 SWITCHING UNITS ON THE CONTROL STATION

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

6.1.1 MODE SELECT SWITCH

This switch gives the operator a choice among the following eight modes of operation (RAPID, JOG, STEP, HANDLE, TAPE, MDI, MEM, EDT). RAPID, JOG, and HANDLE modes are called manual operation mode, and TAPE, MDI, and MEM, automatically operation mode in this manual.

RAPID: Allows the tool to traverse rapidly or return to reference zero by manual operation.

JOG: Allows the tool to feed continuously by manual operation. Feedrate is set by JOG FEEDRATE switch.

STEP: Allows the tool to feed manually by step each time JOG pushbutton is depressed.

HANDLE†: Allows the tool to feed by operating the manual pulse generator†.

TAPE: To automatically control the system from NC tape.

MDI: Allows the operator to insert the block of data through the DATA keyboard and control the system automatically with the data.

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

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

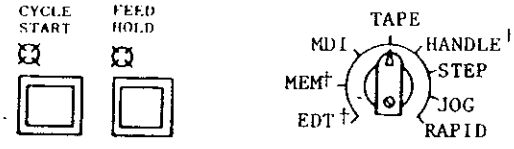


Fig. 6.1.1

6.1.2 CYCLE START PUSHBUTTON AND LAMP

Depress this pushbutton to start the system in the automatic operation mode (TAPE, MDI and MEM). The CYCLE START indicating lamp lights when automatic operation starts. Depress it again to start the operation after temporary stop by operating FEED HOLD pushbutton or MODE SELECT switch.

6.1.3 FEED HOLD PUSHBUTTON AND LAMP

When the FEED HOLD pushubtton is depressed during automatic operation, the feedrate is decreased immediately and machine motion is stopped. Feedhold is not active during tapping by G84, however, it functions during positioning before tapping. Feedhold does not interrupt a dwell by G04.

If it is depressed while M-, S-, T-, or B†-function without move command is being executed, the FEED HOLD lamp will light, but these functions will be executed until finished. On completion of the function, the lamp goes off and machine operation is stopped. Depress the CYCLE START pushbutton to resume the operation after temporary stop by operating FEED HOLD pushbutton.

FEED HOLD lamp is automatically illuminated when the machine stops temporarily during canned cycles if SINGLE BLOCK switch is set on.

6.1.4 EMERGENCY STOP PUSHBUTTON

Depress this pushbutton to immediately stop all machine movement in an emergency. The servo power is turned off and the machine is stopped immediately by dynamic brake. The NC ALARM lamp lights and alarm code "33" is displayed.

To recover the syst em from an emergency stop after the cause has been removed, take the following procedure.

1. Turn the EMERGENCY STOP pushbutton clockwise to release the locking.
2. Depress the RESET key. Alarm code "31" replaces "33."
3. Turn on the servo power again by depressing POWER ON pushbutton. NC ALARM LAMP is extinguished and READY lamp lights up.

The operation is effective in the reverse order of steps 2 and 3. Use this switch also for turning off the system.

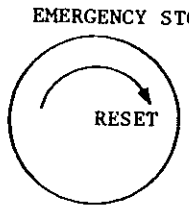


Fig. 6.1.4

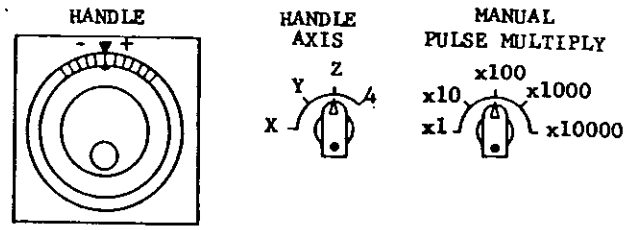


Fig. 6.1.5

6.1.5 HANDLE DIAL† (MANUAL PULSE GENERATOR)

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

1. Set the MODE SELECT switch to the HANDLE.
2. Select the axis to be operated with HANDLE AXIS select switch.
3. Set the move amount per graduation of the dial by setting MANUAL PULSE MULTIPLY switch.
4. Rotate the dial to move the selected axis.

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

6.1.6 HANDLE AXIS SELECT SWITCH†

This switch is used to select an axis to be operated. For operation of the switch, see 6.1.5 Handle Dial†

6.1.7 MANUAL PULSE MULTIPLY SELECT SWITCH†

This switch is used to:

- Select the value from Table 6.1.7.1 corresponding to a single graduation of the HANDLE dial in the HANDLE mode.
- Select the move amount (1 step) from Table 6.1.7.2 corresponding to each depression of JOG pushbutton in the STEP mode.

Table 6.1.7.1 Selection of Move Amount in the HANDLE Mode

Magnification	Metric	Inch†	Rotating Angle†
x 1	0.001 mm/ graduation	0.0001 inch/ graduation	0.001 deg/ graduation
x 10	0.01 mm/ graduation	0.001 inch/ graduation	0.01 deg/ graduation
x 100 x 1000 x 10000	0.1 mm/ graduation	0.01 inch/ graduation	0.1 deg/ graduation

Table 6.1.7.2 Selection of Move Amount in the STEP Mode

Magnification	Metric	Inch †	Rotating Angle †
x 1	0.001 mm/step	0.0001 inch/step	0.001 deg/step
x 10	0.01 mm/step	0.001 inch/step	0.01 deg/step
x 100	0.1 mm/step	0.01 inch/step	0.1 deg/step
x 1000	1.0 mm/step	0.1 inch/step	1.0 deg/step
x 10000	10.0 mm/step	1.0 inch/step	10.0 deg/step

6.1.8 JOG PUSHBUTTONS

This pushbutton is used to feed the tool manually.

- With any of pushbuttons +X, -X, +Y, -Y, +Z, or -Z (+4, -4) held in the RAPID mode, the axis can be moved rapidly until the button is removed.
- This pushbutton moves the tool at the speed set by JOG FEEDRATE switch in the JOG mode.
- Each time the pushbutton is depressed in the STEP mode, the tool is moved by the value per step set by MANUAL PULSE MULTIPLY select switch. Maximum feedrate per step is determined by parameter No. "67."

NOTE: A JOG pushbutton works on an axis. If more than one JOG button is depressed, motion is stopped.

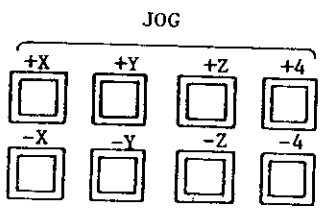


Fig. 6.1.8

6.1.9 JOG FEEDRATE SWITCH

The JOG FEEDRATE switch is used to select the jog feedrate in the JOG mode. Up to 32 steps of feedrate can be specified. Jog feedrate depends on the machine tool. For definite values, refer to the machine tool builder's manual. See Table 6.1.9.

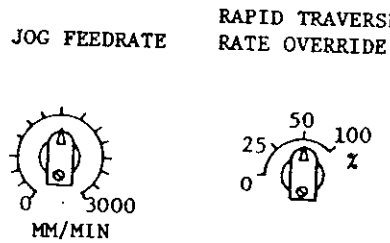


Fig. 6.1.9

6.1.10 RAPID TRAVERSE RATE OVERRIDE SWITCH †

This switch is used to adjust the traverse rate by 0, 25, 50, and 100%. 100% Rate is the rapid traverse rate set by parameter No. "94." The switch is effective both in automatic operation including G00 command and in manual operation (RAPID mode).

Table 6.1.9 Jog Feedrate

Step	A mm/min (deg†/min)†	B
0	1	
1	2	
2	3	
3	4	
4	6	
5	8	
6	10	
7	12	
8	15	
9	20	
10	25	
11	30	
12	40	
13	50	
14	60	
15	80	

Step	A mm/min (deg†/min)†	B
16	100	
17	120	
18	150	
19	200	
20	250	
21	300	
22	400	
23	500	
24	600	
25	800	
26	1000	
27	1200	
28	1500	
29	2000	
30	2500	
31	3000	

Notes:

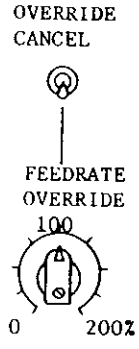
- Jog feedrate depends on the machine tool.
  For definite values, refer to the machine tool builder's manual.
- Feedrate of the fourth axis†, if provided, is shown by deg/min.

6.1.11 FEEDRATE OVERRIDE SWITCH

In the automatic operation mode (TAPE, MEM, MDI), this switch is used to adjust the feedrate by 10% from 0 to 200% of the programmed feedrate specified with an F function at whatever position the switch may be set. Where OVERRIDE CANCEL switch is set on, the tool will be moved at the programmed feedrate by F code regardless of setting the switch.

Table 6.1.11 FEEDRATE OVERRIDE

STEP	%	STEP	%
0	0	11	110
1	10	12	120
2	20	13	130
3	30	14	140
4	40	15	150
5	50	16	160
6	60	17	170
7	70	18	180
8	80	19	190
9	90	21	200
10	100		



6.1.12 OVERRIDE CANCEL SWITCH†

Turning on the OVERRIDE CANCEL switch prevents the function of FEEDRATE OVERRIDE switch.

6.1.13 ZERO RETURN SWITCH† (MANUAL ZERO RETURN)

Turning on this switch causes the tool to return to the reference zero by manual operation. For operation, refer to 6.2.1 Manual Zero Return.

6.1.14 ZERO POSITION LAMPS† (FOR X, Y, Z, AND 4TH AXES)

These lamps indicate that the tool is positioned at the reference zero. When the tool has been positioned at reference zero on each axis manually, by G28 automatically, or by G27 for checking, the lamps for each axis light up. The lamps go off when the tool leaves the reference zero.

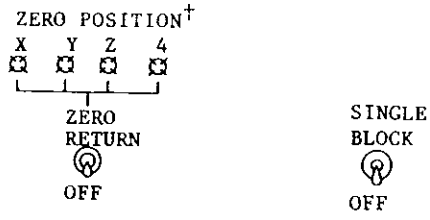


Fig. 6.1.14

6.1.15 SINGLE BLOCK SWITCH

Turning on this switch permits individual block-by-block operation. Turning on this switch after finishing the current block in the automatic operation mode, the machine stops. A block of data is executed each time the CYCLE START pushbutton is activated.

6.1.16 DISPLAY LOCK/MACHINE LOCK SWITCH

This switch functions to stop updating the position display, or to stop move command pulses to the servos. This switch cannot be set unless the machine is stopped at block end or temporarily stopped by FEED HOLD pushbutton.

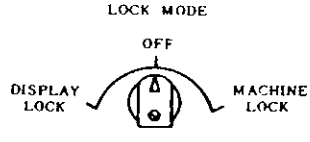


Fig. 6.1.16

"OFF"

Usual operation is made at "OFF" position in both manual and automatic operation. The machine and the position display operate according to the command by automatic operation or manual operation.

"DISPLAY LOCK"

This position is used to exclude the axis movement value from the position display. Current position display is not updated, though the machine moves. Universal display is not affected.

"MACHINE LOCK"

Setting the switch at MACHINE LOCK inhibits axis movement including Zero Return. The position display is updated. M-, S-, and T-functions are executed. This position is selected to preset the display or to check the tape data.

6.1.17 DRY RUN SWITCH

Turning on the DRY RUN switch in the TAPE, MDI or MEM mode causes the tool to move at the speed selected by the JOG FEEDRATE switch, ignoring all programmed F-functions. F commands can be displayed as they are programmed. This switch may be used to check the program.

Rapid traverse (G00) rate for dry run operation can be set by setting parameter No. "02."

Parameter No. "02"	Rapid Traverse at Dry Run Operation
"0"	Rapid traverse rate (Note)
"1"	Jog feedrate

Note: The tool moves at the traverse rate set by RAPID TRAVERSE RATE OVERRIDE switch† if provided.

NOTES:

- Switching the DRY RUN switch during automatic operation becomes effective on the current block. Switching it in mm/rev mode† becomes effective on the next block.

Fig. 6.1.17

6.1.18 OPTIONAL BLOCK SKIP SWITCH

This switch, when set on, ignores the block starting from optional block skip character "/" in the automatic operation mode (TAPE and MEM). To cancel this function, turn off the switch. The "/" character is placed before the sequence number.

Operation of this switch is not effective for the block stored in the buffer register and active register. During the automatic operation Block Skip is effective for the block to be entered in the buffer register.

6.1.19 OPTIONAL STOP SWITCH

This switch is to execute M01 command in automatic operation mode (TAPE, MEM or MDI).

When the switch is on, the program stops on completion of the block including M01 command, while CYCLE START pushbutton remains illuminated. When the control catches FIN signal, the light is extinguished. To restart the program, depress the CYCLE START button. When the switch is off, M01 command is ignored.

Operation of the switch is not effective for the block being executed. During the automatic operation, the switch acts for the next block.

6.1.20 MANUAL ABSOLUTE SWITCH

- When MANUAL ABSOLUTE switch is on.

- When automatic operation is restarted after interrupted by manual operation, the tool performs the rest of the command in the interrupted block from the end point of manual operation. The tool moves in parallel with the path specified by the program.

When the command of the next block is G00 or G01, the tool moves automatically to the target coordinate specified by the program. Then the operation is performed according to block of data.

When the command of the next block is G02 or G03 (circular interpolation), the interpolation is performed in parallel with programmed command. The tool automatically returns to the target coordinate when G00 or G01 is commanded after the circular interpolation.

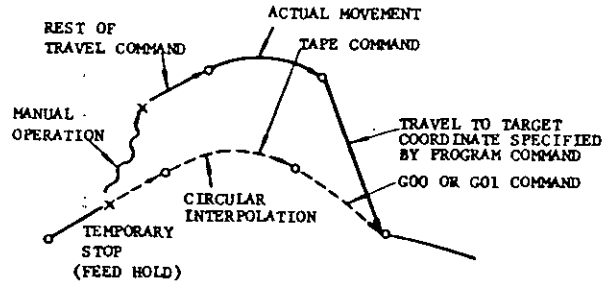
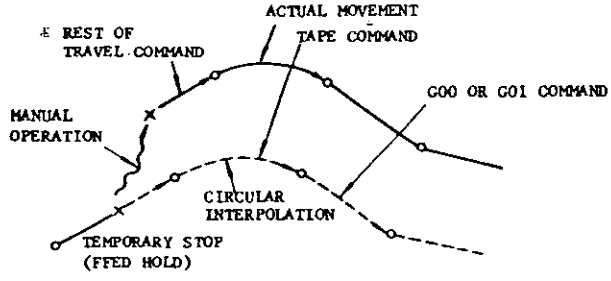


Fig. 6.1.20.1

· When MANUAL ABSOLUTE switch is off.

After the automatic operation is interrupted by manual operation, the coordinate system is shifted, and the tool performs the rest of the travel commands in parallel with programmed moves.



The parallel shift is reset by executing Reference Zero Return manually, automatically by G28, or operating the RESET key. The command value is forced to change to the current position. Thus the shift value is reset.

Fig. 6.1.20.2 Tool Movement with MANUAL ABSOLUTE Switch Off

6.1.21 M-FUNCTION LOCK SWITCH (MISCELLANEOUS FUNCTION LOCK)

· When the M-FUNCTION LOCK switch is on, it ignores the M, S, T, and B† commands. To check the tape data, the operation by the switch is used in combination with MACHINE LOCK function.

Fig. 6.1.21

· The following M codes are executed even if the switch is set on.

- M00, M01, M02, M30
- Both its decoded signals and its BCD codes are sent out to the machine.
- M90 to M99
  BCD code is not sent out.

· Turning on the M-FUNCTION LOCK switch during automatic operation becomes effective on the block after the next block of the current block.

6.1.22 Z-AXIS FEED NEGLECT SWITCH

The switch is used for dry run operation or drawing-check operation on the X-Y plane. Turning on the switch causes the Z axis in MACHINE LOCK condition. The Z axis movement is inhibited, though the position display is updated.

Operate the switch when the machine is stopped. That is, the switch does not function except when the machine is stopped at the block end by SINGLE BLOCK switch or temporarily stopped by FEED HOLD pushbutton.

6.1.23 FEED CODE LOCK SWITCH (F-CODE LOCK)

Turning on the FEED CODE LOCK switch causes the program execution to ignore all F-code commanding feed. If the switch is turned on during automatic operation, all the F-codes are ignored in the block stored in buffer register after switching operation.

Feedrate override is effective on the current F code even if locked by the switch, though read-in operation does not proceed.

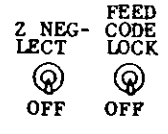


Fig. 6.1.23

6.1.24 BLOCK RETURN SWITCH AND BLOCK RETURN END LAMP†

BLOCK RETURN switch is used, when a trouble occurs during executing a block, to return the tool to the block starting position to restart machining. Completion of the tool returning to the start position is checked by the lamp. For detailed operation, refer to 6.2.2 Block Return.

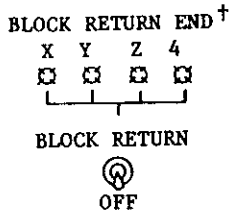


Fig. 6.1.24

6.1.25 MIRROR IMAGE AXIS SELECTOR SWITCH†

MIRROR IMAGE AXIS switch selects the axis whose motion is reversed for programmed operations.

To select the mirror image axis with this switch instead of parameter writing, set the data of parameter No. "91" to 0.

Turn on the MIRROR IMAGE AXIS switch of the axis to which Mirror Image function is assigned. The motion of the selected mirror image axis is set up at M95 command is given until M94 is commanded. For details, see 2.7.2.

NOTE: During the M95 (Mirror Image ON) mode, never operate the MIRROR IMAGE AXIS switch.

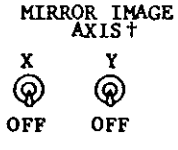


Fig. 6.1.25

6.1.26 ERROR DETECT ON/OFF SWITCH†

The ERROR DETECT ON/OFF switch is used to eliminate the corner-round-finish caused by the servo-lag.

· OFF position

After completion of distributing pulses for linear and circular interpolation, the control ignores servo-lag pulses and distributes pulses of the next block. Operation of a block to the next block is smoothly processed.

· ON position

After completion of distributing pulses for linear and circular interpolation, the control executes the next block after checking that servo-lag pulses are within the allowable value. The function is used to prevent the round finish of the corner.

G00 (positioning) command allows the control to execute the next block after servo-lag pulses reach the allowable value independently of the switch.

Turning on the switch during automatic operation activates at the end of the current block.

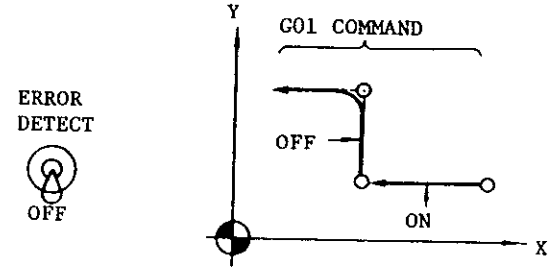


Fig. 6.1.26

6.1.27 TOOL OFFSET AUTO-STORE PUSHBUTTON AND LAMP†

Use the TOOL OFFSET AUTO-STORE button to automatically store the amount of Z-axis move manually made between "home-position" and "base-position" directly in the tool offset memory. For operating procedure, refer to 6.2.3 Automatic Tool-offset Store.

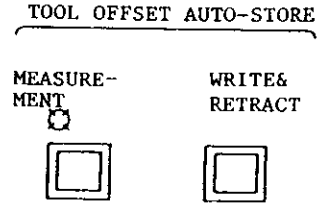


Fig. 6.1.27

6.1.28 EDIT LOCK SWITCH†

Turning on the EDIT LOCK switch prevents the function of ERS, INS, ALT, and EOB keys, and storing from NC tape.

6.1.29 START LOCK INPUT (OR SWITCH)†

When the START LOCK is on, CYCLE START pushbutton does not function. Use the START LOCK input to prevent operating the machine in abnormal condition during automatic operation. The input may be used as on/off switch on control station for machine.

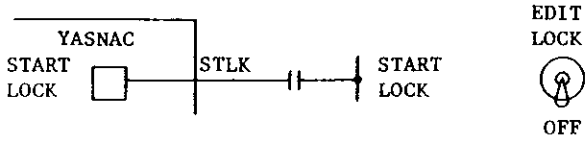


Fig. 6.1.29

6.1.30 AXIS INTERLOCK INPUT

The control is provided with AXIS INTERLOCK input for each axis to prevent axis motion.

Interlocking an axis in feed motion causes the axis to slows down to a stop. When the interlock is released, the axis motion finishes the interrupted block and proceeds to the next.

Interlocking one of the two or three axes being simultaneously interpolated disables the interpolation.

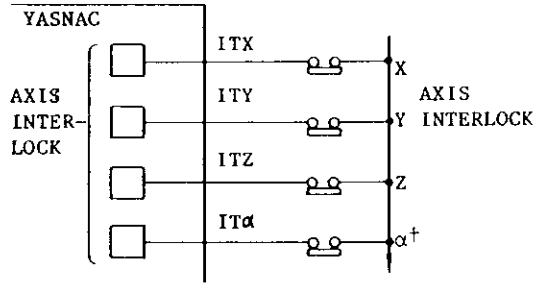


Fig. 6.1.30

· For cutting feed command (in mm/min mode)

The machine feedrate is decreased to the speed set by parameter No. 93 while the TRAVERSE RATE CLAMP limit switch is active.

Where the rate given by the F command is lower than the traverse rate clamp speed set by parameter, the machinemoves at the rate commanded by F-code.

6.1.31 TRAVERSE RATE CLAMP INPUT†

The machine is provided with traverse rate clamp limit switches so as to prevent the trouble caused by the motion according to faulty move command.

· For positioning by G00 and manual operation

The machine feedrate is decreased from the switch operating position after detected by the limit switch, and the motion is kept at the speed set by parameter No. 92. The machine moves at the programmed speed in the releasing direction of the limit switch.

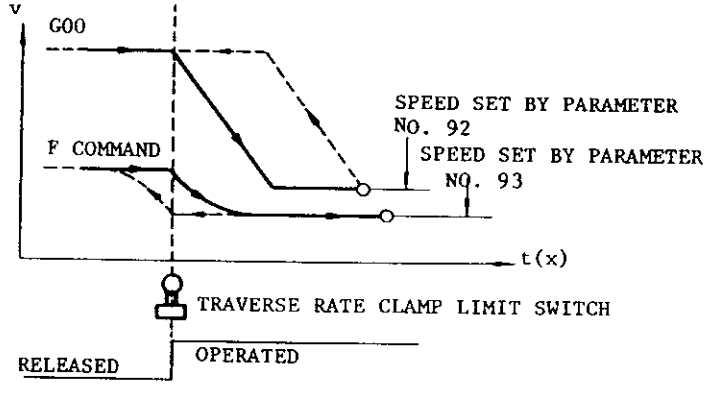


Fig. 6.1.31

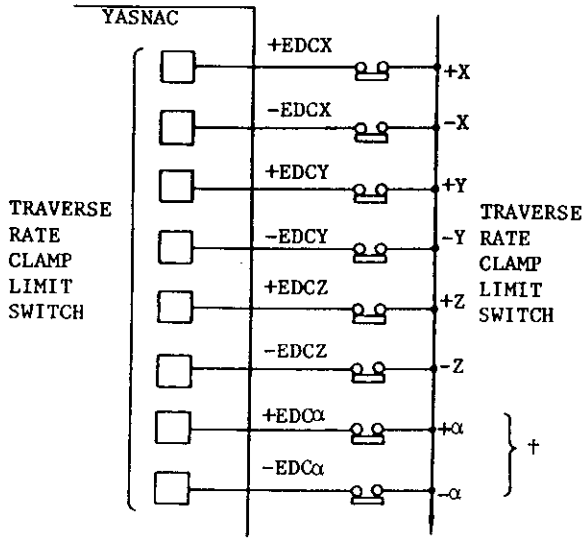


Fig. 6.1.31 (Cont'd)

NOTES:

- Traverse Rate Clamp function is uneffective for the feed command in mm/rev^{+} mode.
- Machine motion by HANDLE (Manual Pulse Generator) is not affected by the TRAVERSE RATE CLAMP limit switch.

6.2 OPERATION FOR MACHINING

6.2.1 MANUAL RETURN TO REFERENCE ZERO

To return the tool to the reference zero point manually, proceed as follows.

1. Set the MODE SELECT switch to RAPID or JOG.
2. Move the tool out of deceleration range manually. If the tool is in the deceleration range, Reference Zero Return is not executed(alarm code "22" is displayed).
3. Turn on ZERO RETURN switch.
4. Hold in the JOG button of the axis to be returned to the reference zero.

   The tool is moved in the same manner as manual feed operation, slowed down after deceleration starting point, and automatically stopped at the reference zero.
5. ZERO POSITION lamp (APX or APZ) indicates the axis which has reached the reference zero point.

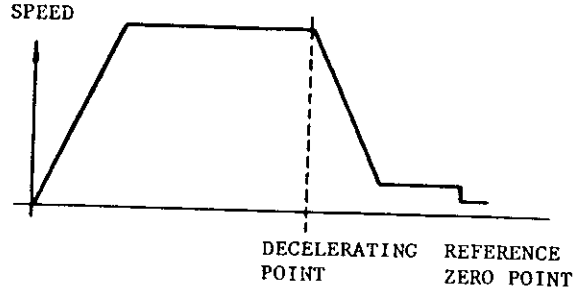


Fig. 6.2.1.1

NOTES:

- The axis which has been returned to the reference zero cannot be advanced manually unless the ZERO RETURN switch is turned off. Retracting the axis from the reference zero is possible.
- The axis in the deceleration range cannot be returned to the reference zero by the reference zero operation except for first zero return operation after power on.
- Turning on the MACHINE LOCK switch disables the reference zero return of the axis.
- Do not return the axis to reference zero manually when the block data read ahead is in the buffer register, for the data will be deleted.

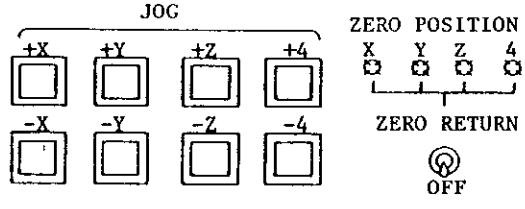


Fig. 6.2.1.2

6.2.2 BLOCK RETURN†

Block Return is used to restart the operation, if a trouble is encountered in a block during automatic operation, from the beginning of the block.

1. Stop the machine by depressing the FEED HOLD pushbutton at the time when the trouble is encountered in machine operation.

2. Set the MODE SELECT switch to manual operation mode (RAPID, JOG, HANDLE, STEP), and move the tool out of the position where the trouble occured.

3. Take a correct action for the trouble.

4. Turn on the BLOCK RETURN switch.

5. Move the tool manually on an axis to the starting position of the block in which a trouble is found. The axis stops automatically when it reaches the block starting position and BLOCK RETURN END lamp for the axis lights up.

6. Repeat the step 5 to complete all the axes motions to the block starting position.

7. Set back the MODE SELECT switch to the automatic operation mode (TAPE, MEM).

8. Depress the CYCLE START pushbutton to resume the operation. If BLOCK RETURN END lamps for all the axes do not light up, the CYCLE START does not function. Operation restarts from the block starting position.

9. After the machine starts by CYCLE START pushbutton, turn off the BLOCK RETURN switch.

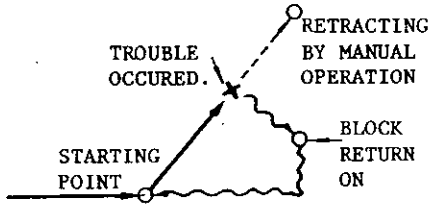


Fig. 6.2.2.1

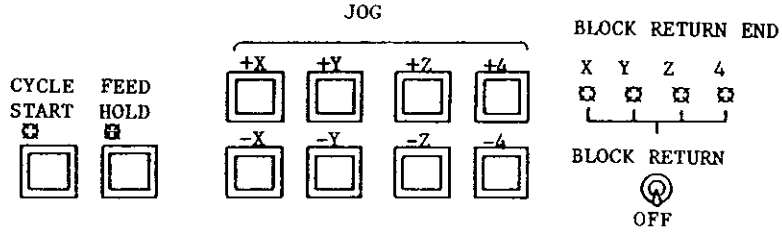


Fig. 6.2.2.2

NOTES:

- Be careful to follow the steps 8 and 9 orderly. Depress the CYCLE START pushbutton after turning off the BLOCK RETURN switch causes the program execution to process the remains of the block at the time when trouble occurs.
- Depressing the RESET key during Block Return cancels the Block Return function.
- It is not possible to change the data in the block which is to be executed by Block Return, because the data has been stored in active register.

6.2.3 AUTOMATIC TOOL-OFFSET STORING†

Automatic Tool-offset Storing is to automatically store the tool offset value of Z-axis, which is measured manually, in the tool offset memory. This function is effective for Z-axis only. Operation procedure is as follows.

1. Mount the tool on the spindle, and position it at the point to be set as "home-position" on Z-axis. To simplify changing tool, it is recommended to set home-position at the tool change position.

2. To display the offset value to be automatically stored, reset the display of Z-axis to zero. For the detailed operation, see NOTES. Writing offset value is correctly made without resetting the display.

3. Set the MODE SELECT switch to manual operation mode (RAPID, JOG, STEP or HANDLE).

4. Depress the OFS key and select the address H or D.

5. Set the desired tool offset number with NUMBER keys.

6. Depress the MEASUREMENT pushbutton. The MEASUREMENT lamp lights up and the current position of Z-axis is set as "home position."

   The MEASUREMENT pushbutton is effective with COM, POS, or OFS key selected in the manual operation mode. Depress the MEASUREMENT button while the machine stops.

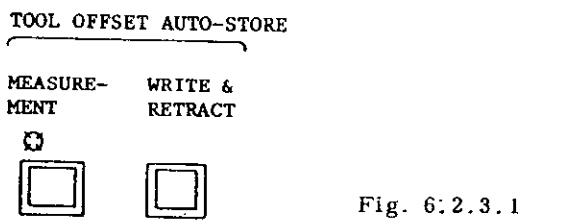


Fig. 6.2.3.1

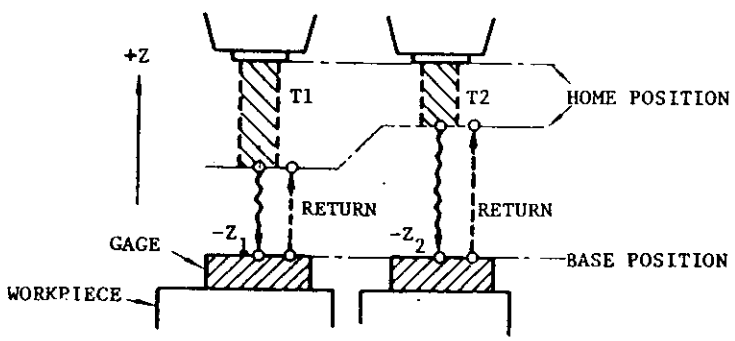


Fig. 6.2.3.2

7. Set the tool tip to the "base position" by moving the Z-axis manually using JOG pushbutton. "Base position" can be freely set according to the workpiece to be machined.

   If the resetting the display of Z-axis to zero, the display shows move amount of the tool between home position and base position. The move amount in Fig. 6.2.3.2 is shown by $-Z_1$ or $-Z_2$.

8. Depress the WRITE & RETRACT pushbutton. The control executes the following operation.

   · The control stores the move amount of the tool between home position and base level ($-Z_1$ or $-Z_2$) in the tool offset memory specified by the NUMBER keys.

   · One is automatically added to the tool offset number specified by NUMBER keys on the NUMBER display. This is a preparation for storing the tool offset to be written next. (NUMBER 99 is replaced with 01.) At this time Z-axis automatically returns to home position at rapid traverse rate. The home position of the tool is displayed by zero.

9. By manual operation or MDI operation, change the tool with a desired one.

   Even if the mode selector switch is turned to MDI for changing the tool, MEASUREMENT lamp is kept illuminated. On completion of changing the tool, set the MODE selector back to manual operation mode.

10. Repeat steps 7 to 9 until the required number of tool offsets are stored.

11. Depress the MEASUREMENT button. MEASUREMENT lamp extinguishes and automatic tool-offset storing function is cancelled.

NOTES:

· To set the display of home position at zero, take one of the following three methods.

  a. Depress the Z-axis RESET button if the current position display unit is provided. The display for Z-axis shows zero.

  b. When parameter No. 79 is "0," select the address Z and depress the POS key and the CAN key simultaneously. The universal display shows zero.

  c. When parameter No. 79 is "1," execute G92 Z0 * in the MDI mode. The universal display shows zero.

- The measured distance between home position and base position is stored in memory as absolute value in Tool-offset Auto-store mode.

- If the home-position is set at other than tool change position, the new tool changed can be moved to the base position directly from the tool change position without stopping at the home-position. Once the home-position is fixed by depressing MEASUREMENT button, positioning of each tool at home position is not required.

- Depressing the WRITE & RETRACT pushbutton when NUMBER is set at "00" makes the NUMBER display "01," and permits no tool offset to be written nor the tool to be returned to the home position. Depress the WRITE & RETRACT pushbutton again to store the new tool offset in the tool offset memory specified by NUMBER key.

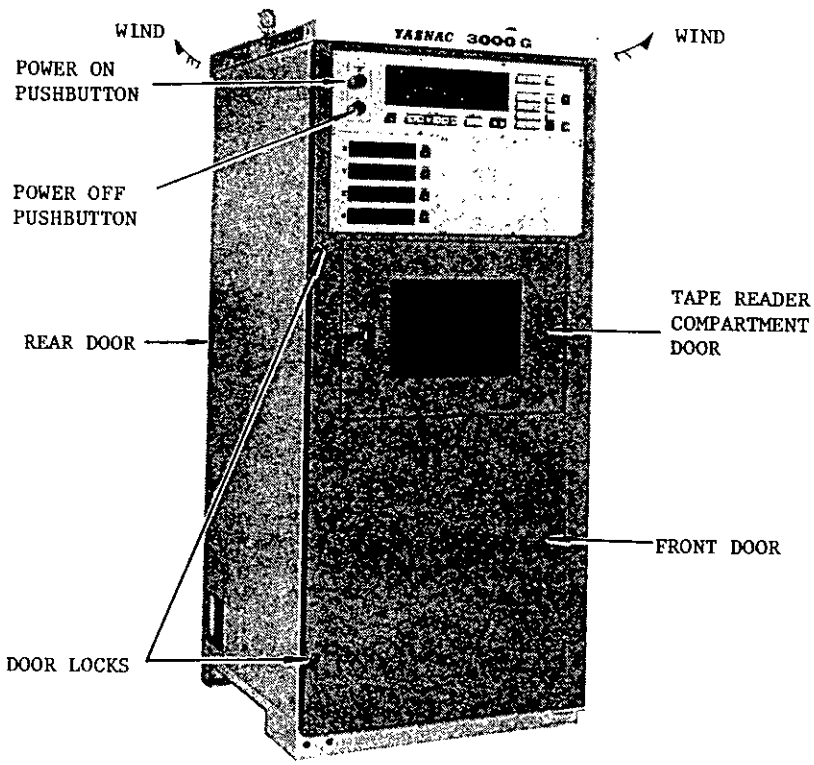

WIND
YASNAC 3000G
WIND
POWER ON
PUSHBUTTON
POWER OFF
PUSHBUTTON
TAPE READER
COMPARTMENT
DOOR
REAR DOOR
FRONT DOOR
DOOR LOCKS

Fig. 7.1.1

7. OPERATION PROCEDURE

7.1 INSPECTION BEFORE TURNING ON POWER

Make sure that the front and rear doors of the control are firmly closed. The control employs totally-enclosed, dustproof enclosure to shut out surrounding air. If the door is open, lock it closely by turning two door lockers with a large screwdriver (minus). In addition, inspect the machine referring to the machine tool buider's manual.

7.2 TURNING ON POWER

1. Check to see that the main power is supplied for the control.
2. Depress the POWER ON pushbutton on the operator's panel, and the control power is supplied and then the cooling fans will start running. Make sure that wind blows out from the exhaust ports of the upper side of the control.
3. Depress the POWER ON pushbutton again to turn on the servo power supply. When the machine is ready to operate, READY lamp lights.
4. If READY lamp does not light, detect and eliminate the cause according to the alarm code displayed. Refer to 4.3.12 Alarm and Status Code Display.

7.3 MANUAL OPERATION*

When the MODE SELECT switch on the control station for machine is set to RAPID, JOG, STEP or HANDLE position, the machine can be operated manually.

Operation in RAPID Mode

1. Set MODE SELECT switch to RAPID.
2. Select the speed using RAPID TRAVERSE RATE OVERRIDE switch†.

   Speed setting range: 100% - 50% - 25% - 0%
3. Push JOG button for the axis and direction to be moved. The machine moves at the specified speed while the JOG button depressed.

Manual operation is defined as the operation in RAPID, JOG, STEP, or HANDLE.

Operation in JOG Mode

1. Set MODE SELECT switch to JOG.
2. Adjust the feedrate to the desired setting with JOG FEEDRATE switch (Up to 32 steps).
3. Push JOG button for the axis and direction to be moved. The machine moves at the specified speed while the JOG button depressed.

Operation in STEP Mode

1. Set MODE SELECT switch to STEP.
2. Select the move amount per step using MANUAL PULSE MULTIPLY switch.

   (Move amount setting range)

   Metric: 0.001 - 0.01 - 0.1 - 1.0 - 10.0 mm/step

   Inch: 0.0001 - 0.001 - 0.01 - 0.1 - 1.0 inch/step
3. Depress JOG button for the axis and direction to be moved. The machine moves by the move amount per step each time the button is depressed.

Operation in HANDLE Mode†

The control with HANDLE dial† can permit the operation described below.

1. Set MODE SELECT switch to HANDLE.
2. Select the axis with HANDLE AXIS switch.
3. Select the move amount of the machine corresponding to one scale of HANDLE dial using MANUAL PULSE MULTIPLY switch.

   Metric: 0.001 - 0.01 - 0.1 (mm per graduation)

   Inch: 0.0001 - 0.001 - 0.01 (inch per graduation)

   NOTE: "X1000" or X10000" is regarded as "X100."
4. Rotate HANDLE dial.

   Turning the dial clockwise:
   The machine moves in the positive direction.

   Turning the dial counterclockwise:
   The machine moves in the negative direction.

7.4 PREPARATION FOR LEADSCREW ERROR COMPENSATION AND STROKE CHECK †

1. Return to reference zero

   The control with Leadscrew Error Compensation option or Stroke Check option requires any of the following operations for returning the machine to the reference zero point, because the data for operation of both options are referenced from this point.

   a. Return the tool to the reference zero point by manual operation. (Refer to 6.2.1 MANUAL ZERO RETURN)

   b. Execute "G91 G28 X0 Y0 Z0 *" in MDI mode.

2. Check of parameter No. 03 or No. 05

   After return to reference zero,

   when the contents of parameter No. 03 are "1," Stroke Check is in effect.

   when the contents of parameter No. 05 are "1," Leadscrew Error Compensation is in effect.

   when the contents of each parameter are "0," each option is effect.

3. Check of parameter No. 06

   When the contents of parameter No. 06 are "1" (Return to reference zero is required before Cycle Start), if the CYCLE START button is depressed without return to reference zero after power on, the alarm code "23" will be displayed. (Alarm code "23": Return to reference zero is unready). It is recommendable to set the contents of parameter No. 06 to "1," when the control is provided with Leadscrew Error Compensation option or Stroke Check option because these options require return to reference zero point before operation. (Leadscrew Error Compensation will not be performed if CYCLE START button is depressed without return to reference zero when the contents of parameter No. 06 are "0" even if the contents of parameter No. 05 are "1." The same can be said for the Stroke Check option.

7.5 PREPARATION FOR AUTOMATIC OPERATION

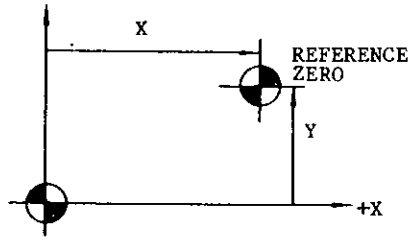
The machine must be positioned properly according to the part program prior to the start of automatic operation. After positioning the absolute coordinate system for the machining must be set properly by manual operation or programming.

1. When G92 is not programmed in a tape or memory.

   · Return the machine manually to the reference zero point.
   (Refer to 6.2.1 MANUAL ZERO RETURN.)

   · The G92 command according to the part program should be executed by MDI.

   G92 X... Y... *



ABSOLUTE ZERO POINT (0, 0)

Fig. 7.5.1

If "G92 X0 Y0 Z0 *" setting is required the coordinate of each axis can be set to "0" easily using ORG key.
Refer to 4.1.13 ORG KEY.

EXAMPLE

```
EOR *
N1 G00 X... Y... Z... *
 .
 .
 .
```

Fig. 7.5.2

2. When G92 is programmed in a tape or memory .

   When the program contains G92 to be executed at the reference zero, return the machine to the reference zero point by manual return to reference zero.

   EXAMPLE:

```
N1 G92 X... Y... Z... *
 .
 .
 .
```

3. When G28 and G92 are programmed.

   When the program is initiated with G28 and G92, move the machine manually into the area where return to reference zero can be performed.

   EXAMPLE:

```
EOB *
N1 G28 X... Y... Z... *
N2 G92 X... Y... Z... *
 .
 .
 .
```

7.6 OPERATION IN TAPE AND MEMORY MODE

1. Make sure that NC ALARM lamp is not illuminated. If illuminated, detect and eliminate the cause by the indication of alarm code. Refer to 4.3.12 Alarm Code Display.

2. Check and correct the stored offset values, and then put the machine in the correct start point.

3. Set the switches on the control station for machine to the proper positions.
   - MODE SELECT switch
   - SINGLE BLOCK toggle switch
   - RAPID TRAVERSE RATE OVERRIDE switch
   - MANUAL ABSOLUTE toggle switch
   - OPTIONAL BLOCK SKIP toggle switch
   - OPTIONAL STOP (M01) toggle switch
   - DRY RUN toggle switch
   - FEEDRATE OVERRIDE & JOG FEEDRATE switch

4. Set the punched tape onto the tape reader. In MEM mode, this operation is not required.

5. Depress RESET key on the operator's panel. Then LABEL SKIP lamp will be illuminated and the memory will be rewound.

6. Depress CYCLE START button to give a Cycle Start to the system.

7. When the Feed Hold is required for the machine during the system operation, depress FEED HOLD button.

8. If the unexpected event occurs in the system, immediately depress EMERGENCY STOP pushbutton.

7.7 MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION

1. Stop the automatic operation temporarily by depressing FEED HOLD pushbutton or by setting SINGLE BLOCK switch to ON position.

2. Record the current positions of each axis on a paper using the current position display operation. Refer to 4.3.3 Current Position Display.

3. Set MODE SELECT switch to manual operation mode (HANDLE, JOG or RAPID), and the machine will be manually operated.

4. Return the machine manually to the recorded positions.

5. Set MODE SELECT switch to the interrupted automatic-mode (TAPE, MDI or MEM).

6. Depress CYCLE START pushbutton, and the machine will resume the automatic operation.

NOTES:

- Where MODE SELECT switch is changed without depressing FEED HOLD pushbutton.
  a. When the automatic-mode (TAPE, MDI or MEM) is changed to the manual-mode (HANDLE, JOG or RAPID), the machine rapidly slows down and stops.
  b. When the automatic-modes are changed the machine is stopped at the block end.
- Where the machine is restarted by depressing CYCLE START button, the tool path shifted due to manual operation will be changed by ON-OFF operation of MANUAL ABSOLUTE switch. Refer to 6.1.18 MANUAL ABSOLUTE Switch.

· In RAPID and JOG modes, when the CYCLE START button is depressed after writing F, M, S, T or B code by use of the same procedure as that of MDI operation, the command becomes effective as soon as written, and BCD signal is fed.
This procedure is used to add a new data to an active buffer. However, M00, M01, M02, M30 and M91 to M99 cannot be written.

7.8 AUTOMATIC OPERATION IN MDI MODE

1. Set MODE SELECT switch to MDI position.
2. Write one block data by MDI operation, and BUFFER lamp on operator's panel lights. Refer to 4.3.2 Writing Command Data by MDI.
3. Depress CYCLE START button, and automatic operation can be executed in MDI mode. BUFFER lamp will be off.

7.9 MDI OPERATION INTERRUPTING AUTOMATIC OPERATION

To modify the block data after interrupting operation in TAPE or MEM mode, the following operation should be done after interrupting the operation.

1. Turn on SINGLE BLOCK switch, and the operation is stopped temporarily after the completion of the block being executed. At the same time, the next block data is stored in the buffer register.
2. Display the data on UNIVERSAL DISPLAY according to 4.3.1 Display of Command Data, and check it.
3. Set MODE SELECT switch to MDI operation.
4. Modify the data referring to 4.3.2 Writing Command Data by MDI.
5. After modifying the data, set back MODE SELECT switch to the interrupted automatic mode (TAPE or MEM).
6. Return SINGLE BLOCK switch to OFF position.
7. Depress CYCLE START button, and TAPE or MEM operation can be continued by the modified data.

NOTES:

· Writing data by MDI cannot be executed in tool radius compensation modes (G41, G42) because two or three blocks are read ahead.
However, F, M, S, T or B† code can be modified. The modification becomes effective in the next block.

· Writing data by MDI should not be performed in canned cycle modes (G73, G76, G81 to G89). The machine may not operate properly.

· Excepting in tool radius compensation and canned cycle modes, MDI operation is possible. Although, the next block of data is usually read ahead in the buffer register, additional or altered data can be written into the buffer register by MDI operation.

7.10 PREPARATION FOR TURNING OFF POWER

1. Make sure that the machine is at standby and CYCLE START lamp is extinguished.
2. Check to see that NC ALARM and INPUT ERROR lamps are not illuminated. If illuminated, detect the causes by displaying alarm code and eliminate them. Refer to 4.3.12 Alarm and Status Code Display.
3. Inspect the machine referring to the machine tool builder's manual.

7.11 TURNING OFF POWER

1. Depress EMERGENCY STOP pushbutton to turn off the servo power supply.
2. Depress POWER OFF pushbutton on the operator's panel to turn off the control power supply.
3. Cut off the main power supply from the control.

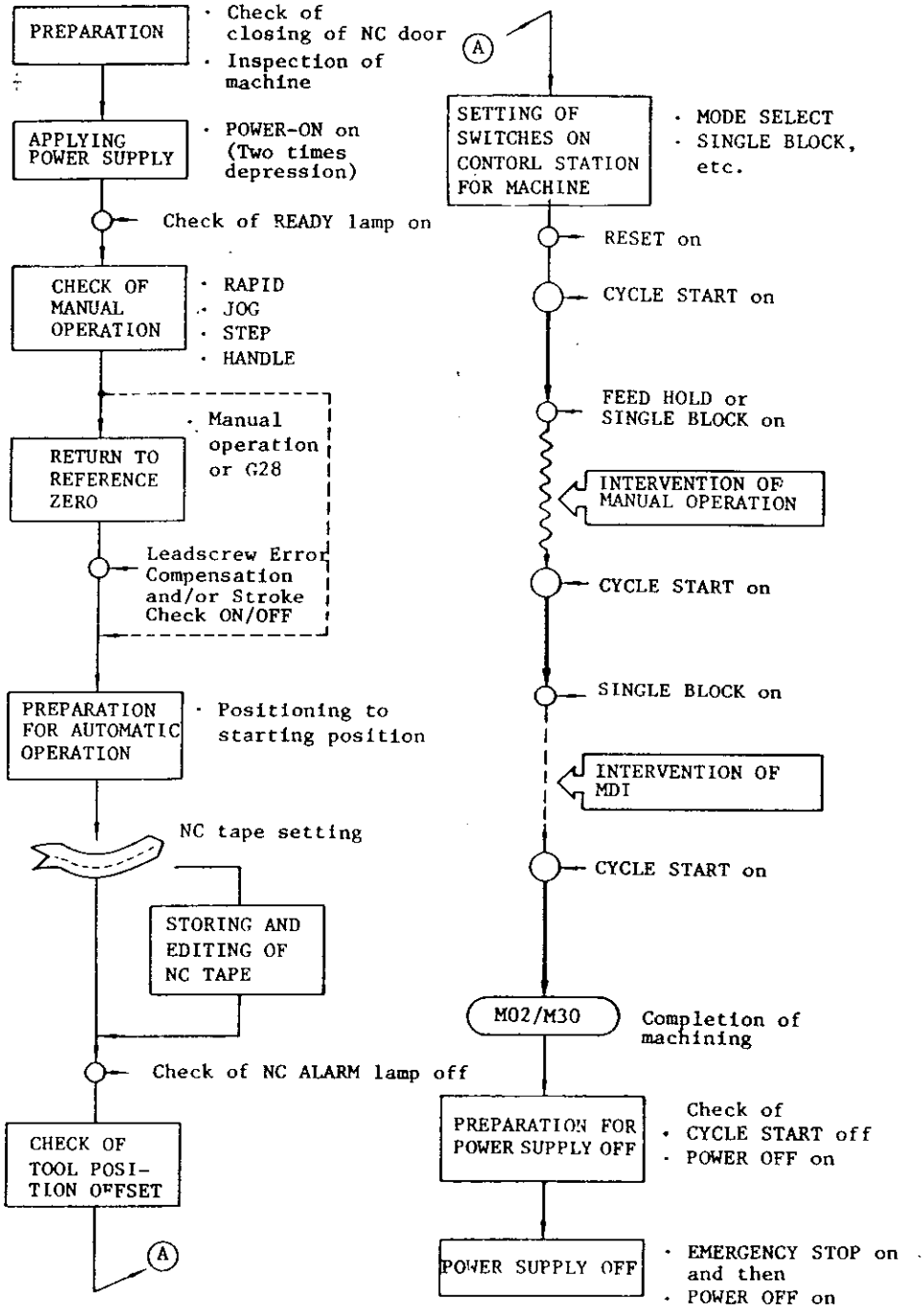


Fig. 7.11.1 Operation Procedure

8. MAINTENANCE

8.1 ROUTINE INSPECTION SCHEDULE

The following table shows the minimum requirements to be observed for maintenance according to time in order to keep the equipment optimum condition for extended period.

Table 8.1.0 Inspection Schedule

Items		Time Interval	With the system-off	With the system-on	Remarks
Tape reader	Cleaning of reading head	Daily	○		Including light source part.
	Cleaning of tape tumble box	Weekly	○		
	Lubricating of tension arm shaft end	As required	○		
Control panel	Tight close of the doors	Daily	○		
	Checking of loose fit and gap of side plates and worn door gaskets	Monthly	○		
Servomotor and DC motor for spindle	Vibration and noise	Daily		○	Feel by hand, and do the audible inspection.
	Motor contamination and breakage	Daily or as required	○	○	Inspect visually.
	Clearance of ventilation openings		○	○	Inspect mainly spindle DC motor.
	Burned spots, cracks, wear, and pressure of brushes	Every three months	○		Check the length of brushes.
	Roughened commutator surface		○		Check dark bar, threading and grooving of commutator.
	Dirt of motor interior		○		Clean with compressed air.
Battery		Daily	○	○	See if BATTERY lamp on operator's panel lights.

The details of inspecting operation for each component are as follows.

8.1.1 TAPE READER

1. Cleaning of head of tape reader

A. Remove tape rubbish and dust on the glass with a blower brush. If the glass is stained with oil or oily dust, wipe it using a gauze or soft cloth with absolute alcohol. Also clean the tape guide and the tape retainer.

B. Remove the dust, if any, on LED (light source) with a blower brush.

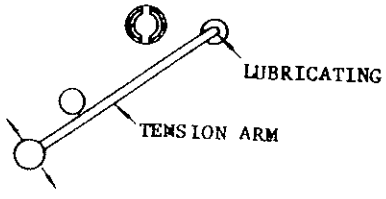
2. Cleaning of tape tumble box

A. Clean the polyester leading tape with a clean, soft cloth.

B. Remove the tape outlet cover (See Fig. 5.1.3) by loosening two mounting screws and clean the bottom of the tape tumble box with cloth or brush.

3. Lubricating of tension arm shaft†

For the control with 6 inches or 8 inches diameter reels, lubricate the shaft end of tension arm, when the tension arm does not move smoothly.



(In the case of 8 inches diameter reel)

Fig. 8.1.1

NOTE: When trouble occurs in feeding or winding tape with 8 inches diameter reels, open the front door and brush away dust around the photo-coupler by using a blower brush.

8.1.2 CONTROL PANEL

The control panel is dustproof, sheet-steel enclosure with gasketed doors.

A. Front and rear doors of the control should be always shut tightly, even if the control is not operating.

B. Where inspecting the control with the door open, after it is over, lock the door position by turning two door locks with a large screwdriver (minus).

Turning direction of door locks is as follows.

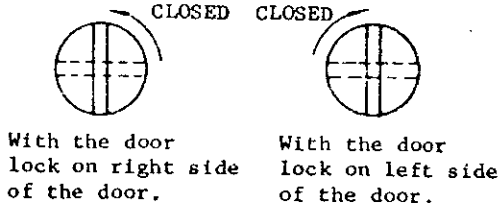


Fig. 8.1.2

The vertical groove on the door lock means the door locked, and the horizontal, the door unlocked.

NOTE: With the door interlocking switch as an option, opening the door shuts off the main power supply and stops all operations.

C. Check gaskets on the brims of front and rear doors.

D. See if the inside of enclosure is dusty. Clean it, if necessary.

E. Check for any opening in the door base with the doors shut tightly.

8.1.3 SERVOMOTOR AND DC MOTOR FOR SPINDLE

1. Vibration and noise.

Vibration can be checked by hand resting on the motors, and to hear the noise using a listening stick is recommended. If any abnormality is found, contact maintenance personnel immediately.

2. Motor contamination and impairment.

Check the motor exterior visually. If dirt or damage should be observed, inspect the motor by removing the machine cover. Refer to the machine tool builder's manual.

3. Clearance of ventilation window blockage

Check the ventilation window of DC spindle motor. If it is clogged with dust or dirt, inspect DC spindle motor removing the machine cover. Refer to the machine tool builder's manual.

Inspection of commutators and brushes is essential for keeping the excellent performance of the control. Inspection work to be executed is described in the following three items. For detailed instructions, refer to YASNAC MAINTENANCE MANUAL.

Three-Month Inspection of Commutators and Brushes

The carbon dust from brushes, scattered around the commutator inside the motor, may cause motor troubles such as the layer short of armature and the flashover of commutator. In the worst case, it may lead the control to a fatal damage. To avoid this, be sure to give an inspection on the commutators and brushes at least every three months.

4. Carbon brushes

A. Under normal operating conditions, brush wears by 2 to 4 mm per 1000 operating hours. If wear is excessive, check to see if oil contaminates armature surface, or abnormal overcurrent flows motor circuit.

B. When brush length becomes shorter than those shown below, replace the brush with new one.

Minertia motor junior series: 6 mm or below

DC motor for spindle: 17 mm or below

C. If either of brush or pigtail is broken, brush assembly must be replaced as a whole unit with new one.

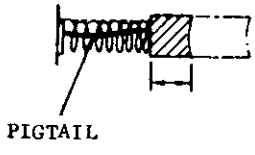


Fig. 8.1.3

NOTE: When replacing the brush assembly, consult the company.

5. Commutator surface

A. Check visually surface roughness of the commutator throughout inspecting window.

The commutator should take on a polished light brown or chocolate color after 100-200 operating hours in the process the motor ideal development of the commutator film. Such commutator needs no attention other than to be kept clean.

B. See if blackened bar, threading (or grooving) is on the commutator. If any of them is detected, investigate the cause of trouble.

Threading or grooving on the commutator surface may be due to too small motor load. Blackened bar is responsible to carbon dust in commutator slots, or incidentally produced sparkings. If it appears that the carbon dust is a cause of blackened bar, finish the commutator with a clean dry cloth to smooth the surface. If the cause seems to be responsible for sparking, consult the company.

6. Motor inside (dirty)

A. Check visually the motor interior through inspection window.

Though the dried carbon dust is not virtually harmful to the correct motor running, clean the inner parts such as commutator, brushholders and brushes with compressed air (air pressure: 2-4 kg/cm2).

B. Where oily carbon dust exists inside the motor due to poor oil seal or defective enclosure, consult the company.

7. Servomotor with oil seal

As the life expectancy of oil seal and brush is 5000 hours (about five years), the inspection and maintenance by the company should be done every 5 years as well as the control. If possible, yearly inspection taking less than 8 hours is recommended.

8.1.4 BATTERY

Make sure that the battery lamp on operator's panel is turned off. If it is turning on, consult maintenance personnel. The battery must be replaced with new one within a month.

8.2 REPLACEMENT OF BATTERY

While power is off, the batteries are used as power source for memory in order to prevent programming data stored in memory such as parameter, tool offset and part program from erasing.

When the battery is going to be discharged after a long period of use, BATTERY lamp on the operator's panel lights to give warning for replacement. In such occasion, consult the company. The battery must be changed with new one before a month passes. Where replacing, never remove the old battery, with power off, otherwise the data stored in memory may be cleared away.

Replacing Procedure

1. Depress POWER OFF pushbutton on the operator's panel.

2. Open the front door of the control. The battery of the memory (printed circuit) board can be seen on the CPU module which is mounted on rear of the front door. If the memory (printed circuit) board is mounted behind the CPU module, loosen two CPU module set screws, and the CPU is hinged open.

3. Depress POWER ON pushbutton. Where the control is equipped with a door interlock switch, draw it out with the movable section by hand. The power can be turned on, with the door open.

4. Check if LEDs (1LED and 2LED) on memory board are lighting. Fig. 8.2.1 shows the arrangement of LEDs and the battery.

1LED:	If lighting, replace the large battery "660S" as memory power of part program with new one.
2LED:	If lighting, replace the small battery "400S" as memory power of parameter and tool offset with new one.

660S . . . Type: JZNC-GBA02

400S . . . Type: JZNC-GBA01

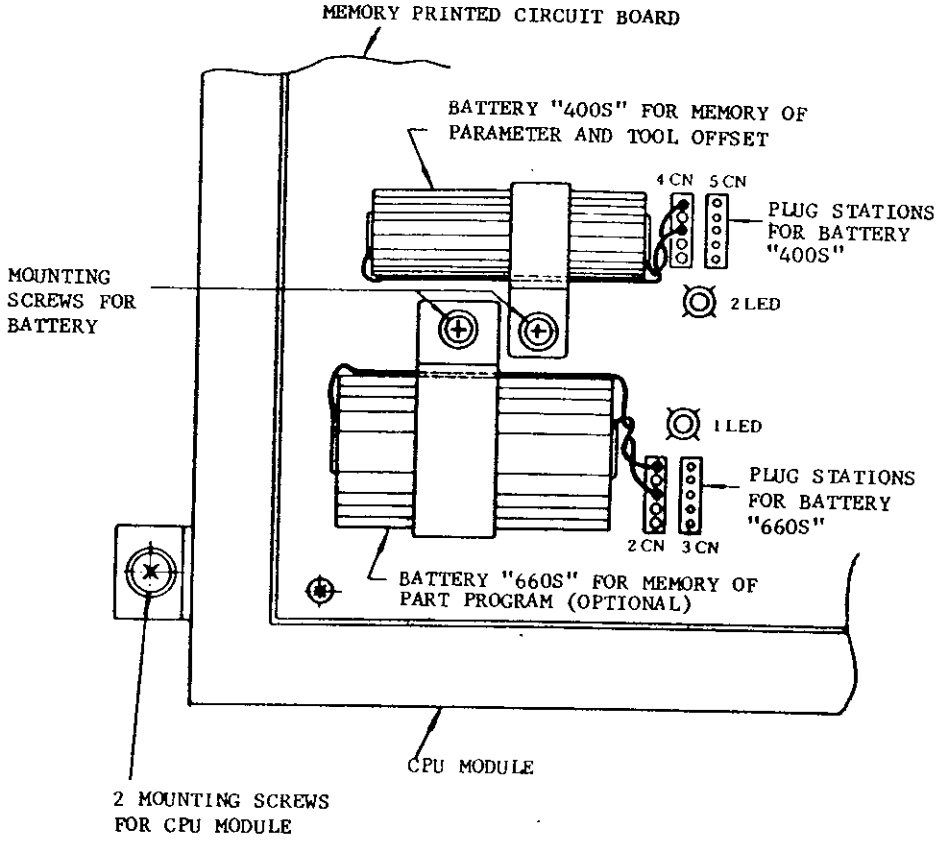


Fig. 8.2.1 Arrangement of LED and Battery

5. With the control power turned on, connect the receptacles of the new battery into the plugs (3CN or 5CN) on memory circuit board, and LED will be turned off. See Fig. 8.2.1. If LED is still lighting, it is due to the wrong insertion of battery connectors, or poor battery charge.

   IMPORTANT: Two plug stations 4CN and 5CN (or 2CN and 3CN) are connected together with common leads. Where an old battery is replaced with a new one, connect the new battery first to the plug station not occupied, then remove the receptacles of the old battery.

6. Depress POWER OFF pushbutton.

7. Remove the mounting screw of old battery, and then replace the battery with new one. In this case, pass the battery lead through gaps between the battery and the battery clamp, and use care not to contact the lead with memory circuit board.

   Where the control is provided with a door interlock switch, push it back in place with the movable section, and power cannot be turned on with the door open.

   Where CPU module is open, fasten it with two mounting screws.

8. Close tightly the front door.

9. Depress POWER ON pushbutton.

10. Make sure that BATTERY lamp on the operator's panel is turned off.

NOTES:

1. While battery is being replaced, exercise utmost care to prevent the oil mist from coming into the control, and to accomplish the work as quickly as possible.

2. Use special care so that no water drop, no oil or dust sticks to the devices (printed circuit board, connectors, cables, etc.) inside the control.

3. Never leave any screws or washers in the control. If left, take them out.

8.3 POWER FUSES

The power fuses are provided for the safety of the control; five in the composite control power supply unit and three in the servo unit for X-, Y- and Z-axis. If they are blown off, consult maintenance personnel.

8.3.1 FUSES OF COMPOSITE CONTROL POWER SUPPLY UNIT

If any of the fuses are blown off, all power supplies are turned off. Reset the control according to the following procedure.

1. Remove the blown-off fuse that is indicated with white mark coming on the fuse casing window.

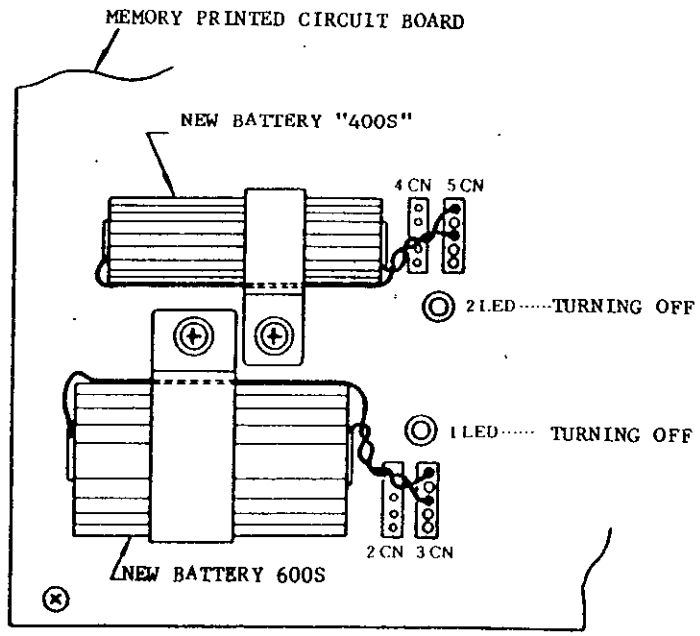


Fig. 8.2.2 New Battery Mounted

2. Inspect for the cause of the control power supply, I/O interface and servo unit, and remove, if any.

3. Replace the blown-off fuse with new one.

4. Depress POWER OFF pushbutton on the operator's panel to reset the control and then turn on the power.

5. If the fuse is blown off again, consult the company.

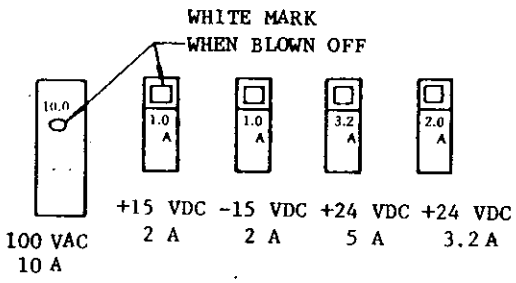


Fig. 8.3.1 Fuses of Composite Control Power Supply Unit

8.3.2 FUSES OF SERVO UNIT

Two plug-in fuses (1FU and 2FU) are mounted on the lower part of the each servo unit. Small capacity servo unit is equipped with only 1FU.

If the fuses of servo unit of any axis are blown off, the system becomes overloaded (shown by alarm code "35"), and servo power supply is turned off. Reset the system using the following procedure.

1. Check for a blown-off fuse which has white mark at the enclosure window, and remove it, if found.

2. Remove the cause of overcurrent.

3. Replace the blown-off fuse with new one.

4. Depress RESET key on the operator's panel to release the alarm status.

5. Depress POWER ON pushbutton to retrieve servo power supply.

6. If any of the fuses are blown off again, consult the company.

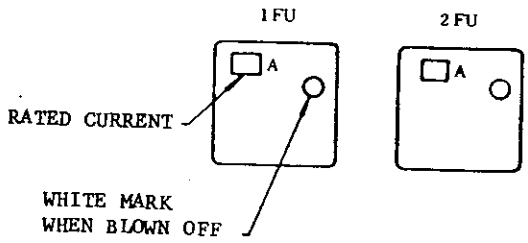


Fig. 8.3.2 Fuses of Servo Unit

8.4 THERMAL OVERLOAD RELAY OF SERVO UNIT

Servo units for each axis are provided with thermal overload relays. Thermal overload relay will be tripped in the following cases.

- When excessively heavy cutting is commanded.
- When excessively frequent commands with acceleration and deceleration are given.
- When friction is produced excessively in the machine system.

Correct the program or check the machine when it occured. Resetting operation of the thermal overload relays is as follows.

1. When the thermal overload relay of servo unit of any axis is tripped, the power supply for servo is turned off and the state of alarm code 35 (overload) occurs.

2. Wait until the thermal overload relay is cooled so that it can be reset. (Approx. 5 minutes) In this period, check fuses of each servo unit if there is any blown off fuses.

3. When the thermal overload relay has been tripped, it will click by depressing the RESET button (red). However, the click is usually killed by the environmental noise, so that take the procedure described below.

4. Set the FUNCTION select key to ALM and check to see that the alarm code "35" is displayed.

5. Depress the RESET button (red) on the thermal overload relay of X axis servo unit.

6. Depress RESET key on the NC operator's panel. If the alarm code "35" is diminished and the alarm code "31" (servo power unready) is displayed, it means that the thermal overload relay for X axis is tripped.

7. When alarm code 35 is not diminished take the same procedure with the item 5 and item 6 for the other axes.

8. Turning on the power after removing the cause from the tripped axis.

9. If the thermal overload relay is blown off again, contact the company.

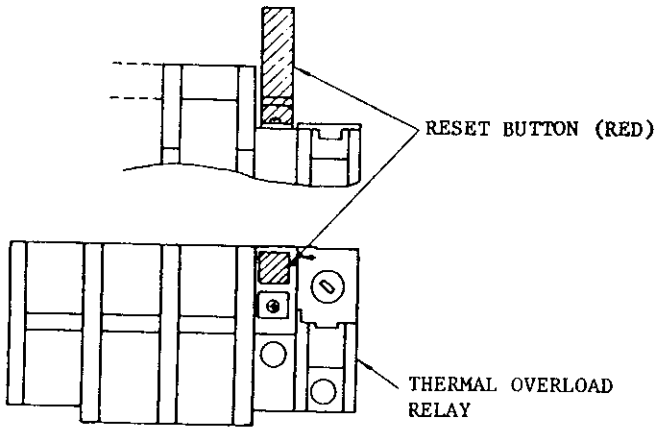


Fig. 8.4

8.5 OTHERS

8.5.1 MOLDED-CASE CIRCUIT BREAKERS (MCB)

When the molded-case circuit breaker (1MCB) is turned off in the NC control panel, power is not supplied to the circuit even if POWER ON push-button on the operator's panel is depressed. Be sure to keep the breaker turned on except it is turned off at inspection and maintenance.

NOTE: The NC control may sometimes not be furnished with MCB. Or some types of the control allow to operate the MCB from outside the enclosure as the all optional machine interface is housed in the NC control enclosure. For operation instructions on these panels, refer to the machine tool builder's manual.

8.5.2 POWER RECEPTACLES FOR MAINTENANCE TOOLS

- The receptacle for powering the devices for maintenance is provided in the enclosure. It can be used for maintenance devices of ratings shown below.

  100 VAC, 1A or below

  Note that the receptacle is not provided with the fuse.

- When the breaker is turned on, the receptacle becomes live.

8.6 TROUBLE CAUSES AND REMEDIES

8.6.1 ON-LINE DIAGNOSTICS

On-line diagnostics are implemented to locate a trouble quickly and protect the machine against malfunctions. Shown below are the displaying functions executed by the control being on-line and machining.

- Display of four-digit alarm code including a code showing an axis in error.
- Display of four-digit status code including a function code.
- Input/output signal display

These displays can be made at any time while the machine is in automatic operation or at standby.

8.6.2 ALARM CODES AND REMEDIES

If the machine stops with INPUT ERROR lamp or NC ALARM lamp illuminated, depress the ALM key. Then alarm and status codes will be displayed on the universal display. Alarm codes "81" and "82" are displayed as soon as the corresponding error occurs.

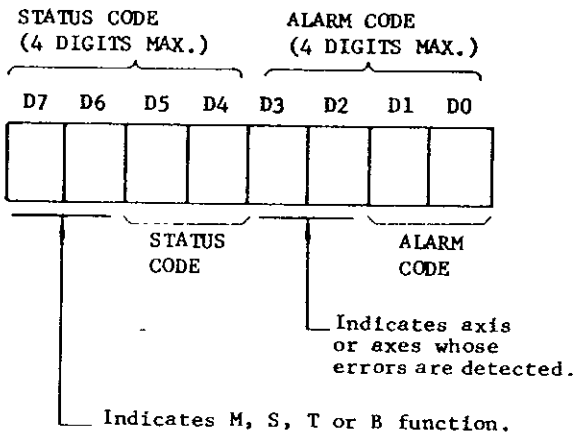


Fig. 8.6.1

For the remedies for trouble causes represented by alarm codes, see Table 8.6.1 Alarm Codes and Remedies on the last part of this manual.

8.6.3 INPUT/OUTPUT SIGNALS

To clear up the causes indicated by alarm and status codes, check the input/output signals on the universal display.

Input/output signals are divided into standard and custom-built ones, and displayed by specifying corresponding diagnostic number.

Standard signals are included in every type of YASNAC 3000G. Custom-built signals are provided for optional machine interface equipped with some types of YASNAC.

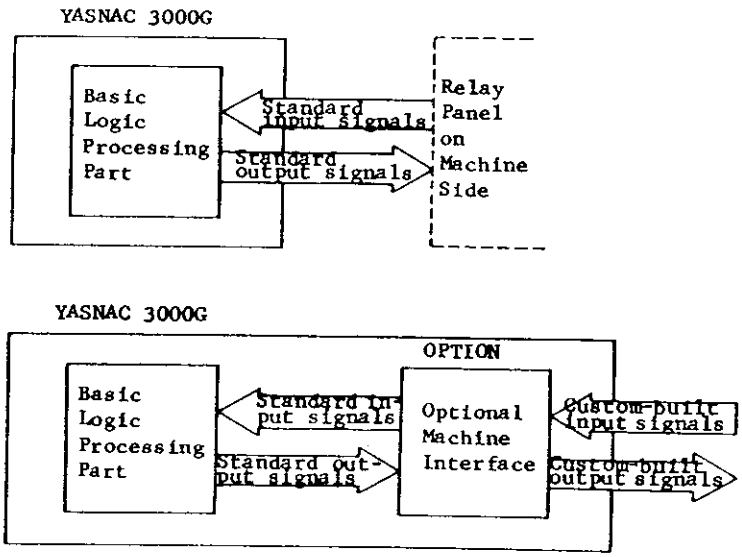


Fig. 8.6.3

To display input/output signals, proceed as follows.

1. Depress the ALM key.

   Alarm and status codes will be shown on the universal display.

2. Set the diagnostic number of signal to be displayed using ADDRESS key and NUMBER key.

Then I/O signal "1" or "0" replaces the displayed alarm code and status code on the universal display. ("1": contact close, "0": contact open). Operation of NUMBER key makes I/O signal display mode automatically

I/O signal display mode is cancelled by depressing the RESET key or FUNCTION key.

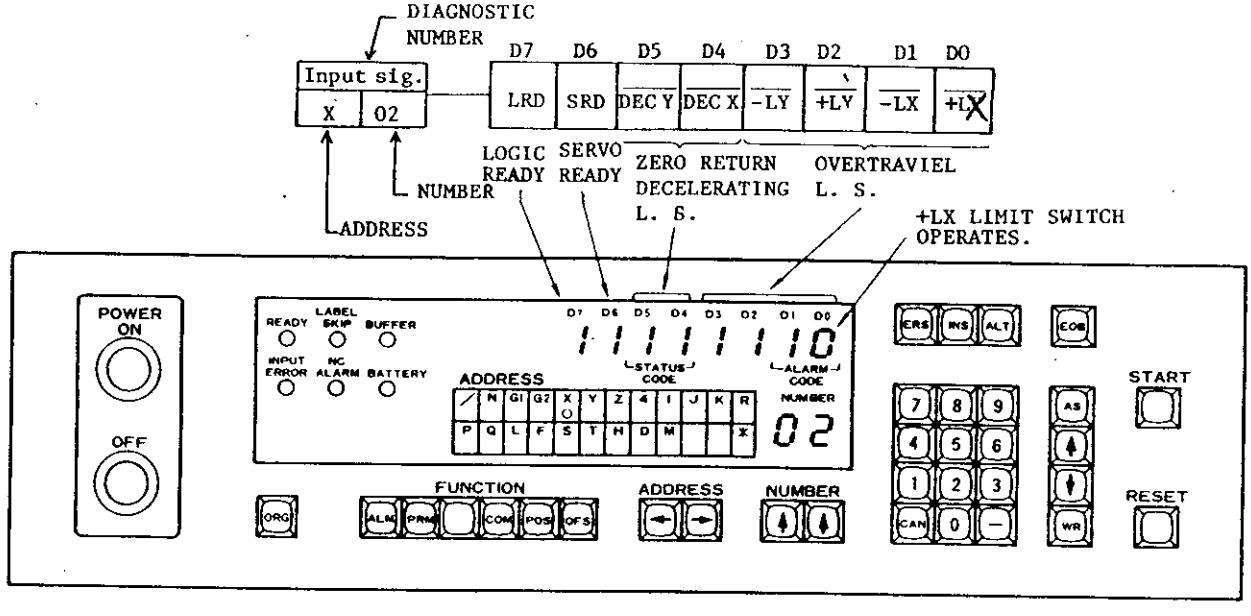


Fig. 8.6.4

Table 8.6.2 shows standard input/output signals. (For custom-built signals depending on the system, refer to the LIST OF I/O SIGNALS provided for the system.)

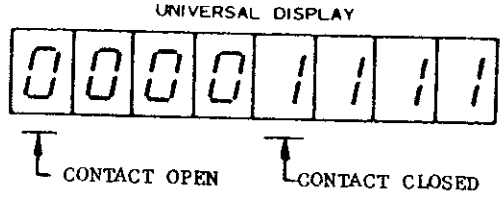


Fig. 8.6.5 Status Display of Input/Output Signals

Table 8.6.2 List of Standard Input/Output Signals

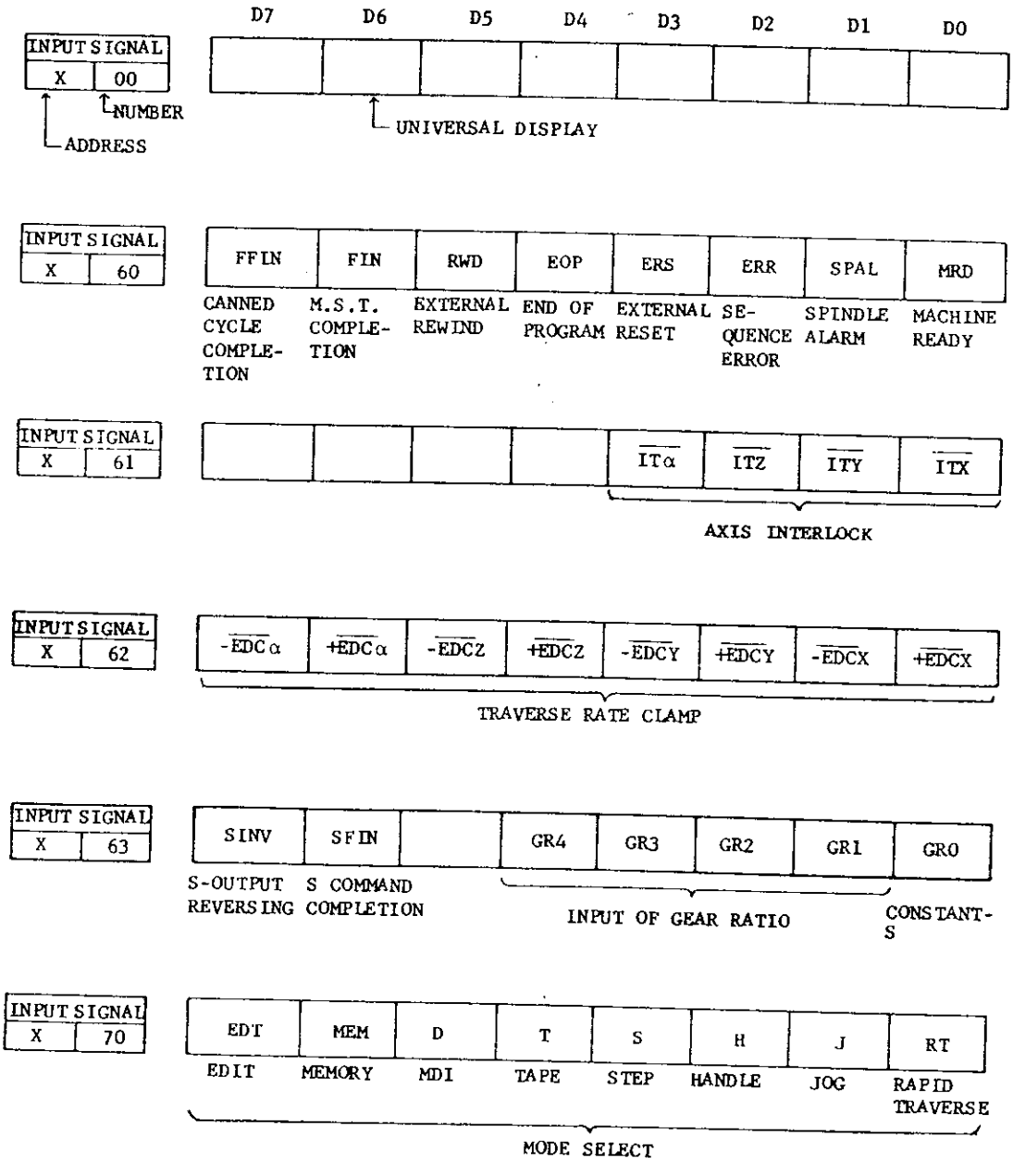


Table 8.6.2 List of Standard Input/Output Signals (Cont'd)

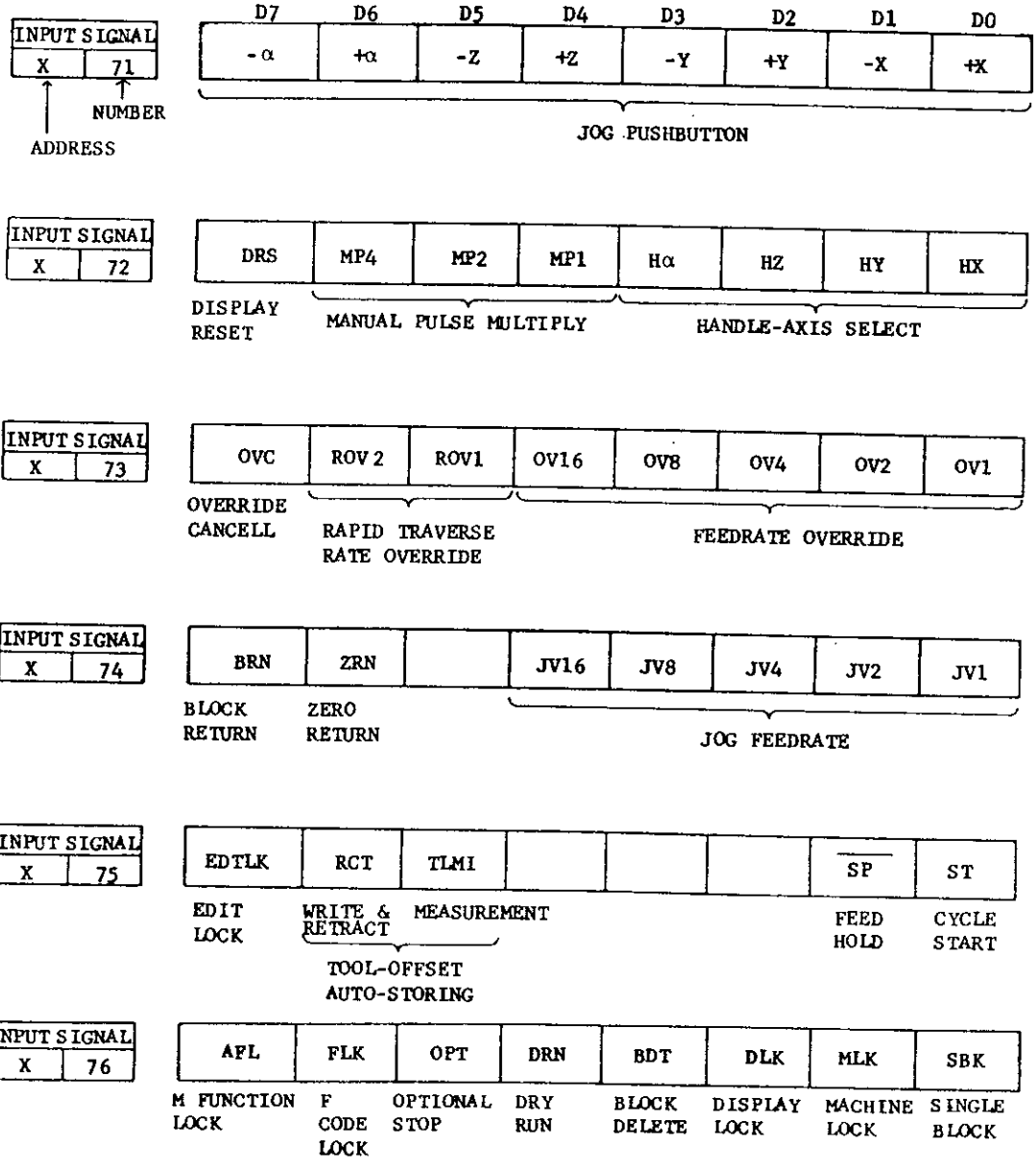


Table 8.6.2 List of Standard Input/Output Signals (Cont'd)

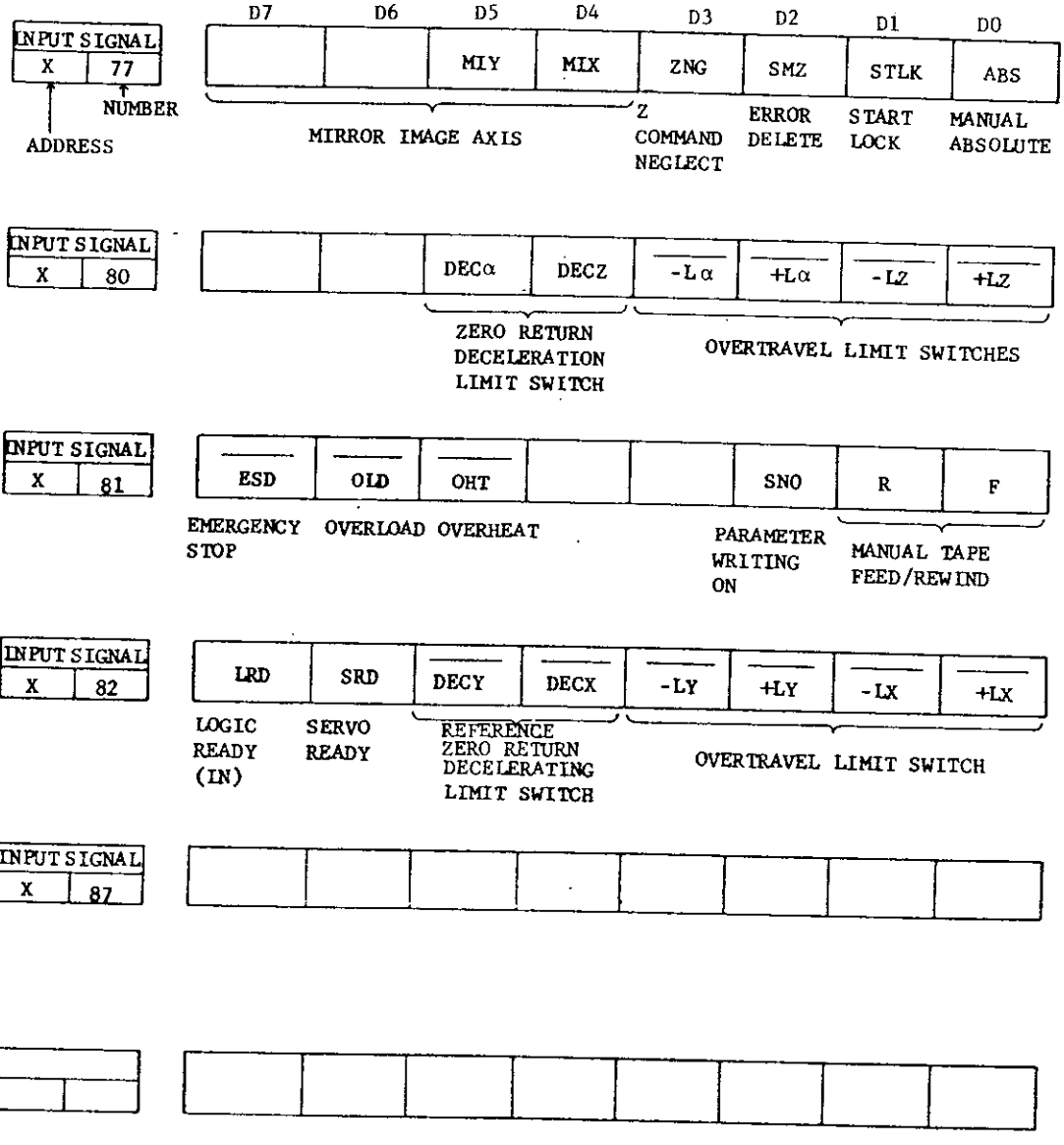


Table 8.6.2 List of Standard Input/Output Signals (Cont'd)

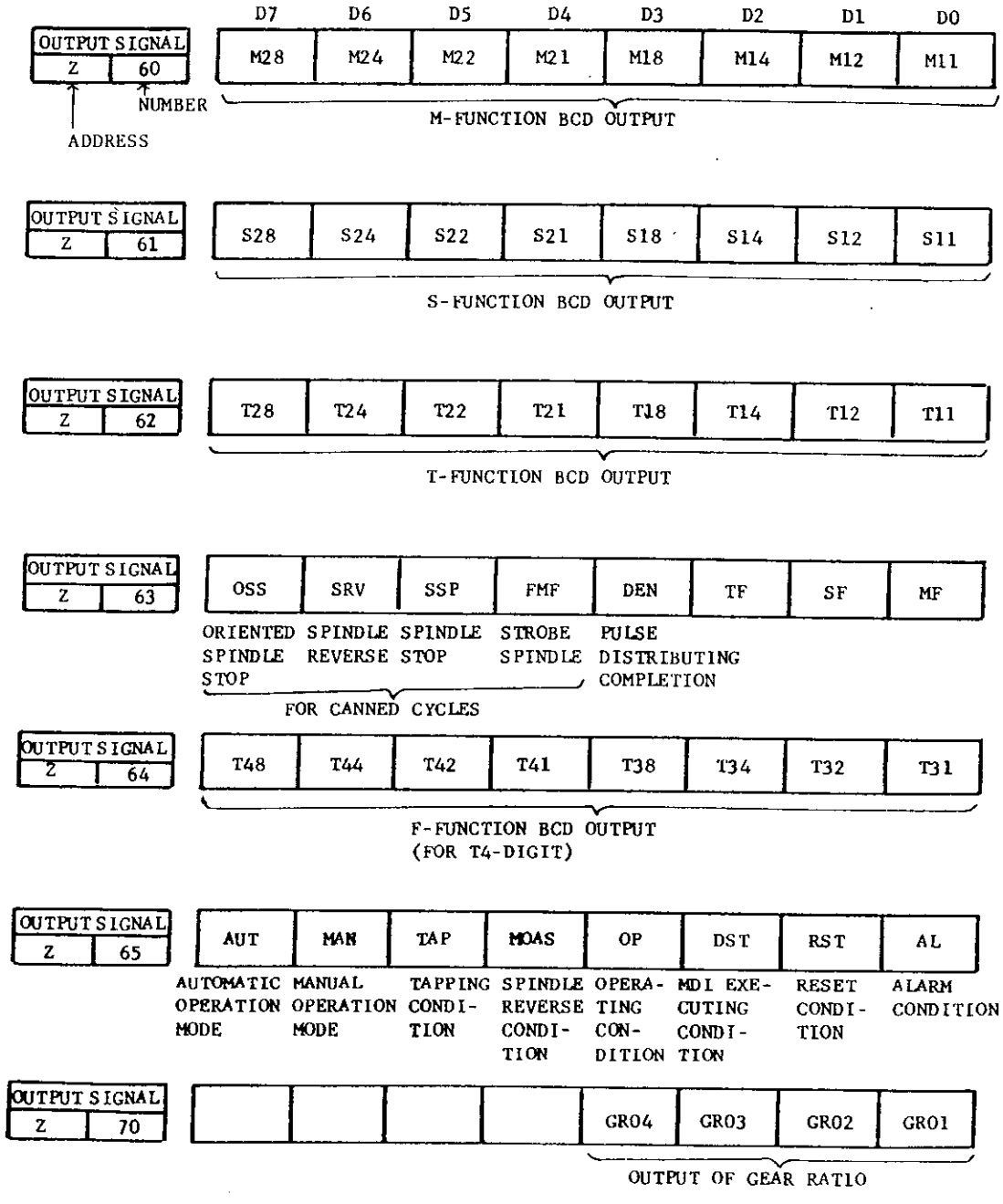


Table 8.6.2 List of Standard Input/Output Signals (Cont'd)

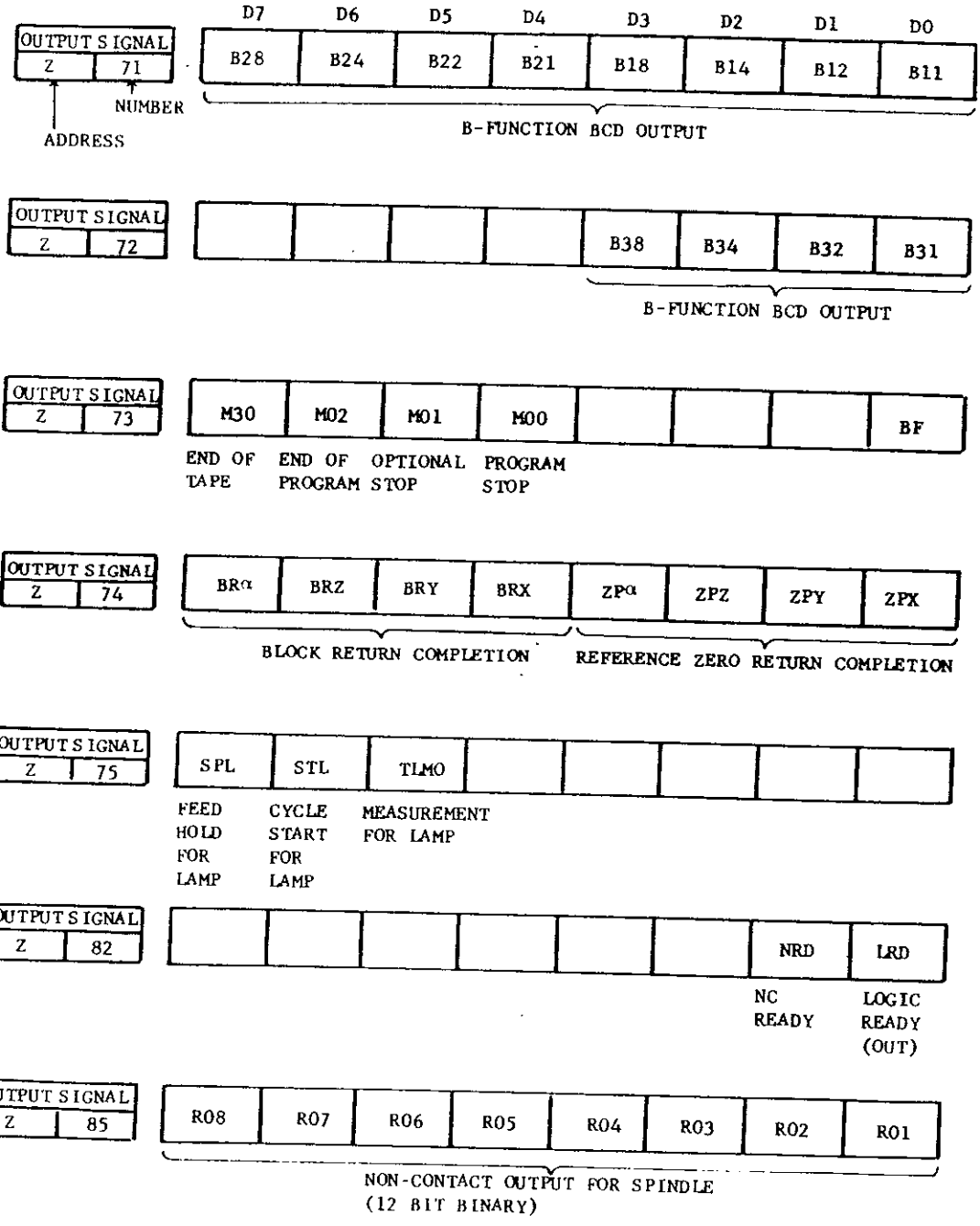
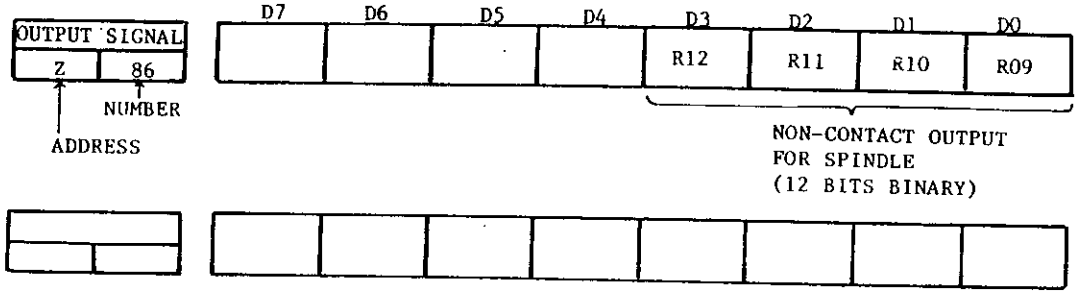


Table 8.6.2 List of Standard Input/Output Signals (Cont'd)



8.6.4 BEFORE MAINTENANCE CALL

If the cause of trouble cannot be found by using alarm codes or I/O signals (described in 8.6.1 to 8.6.3), or correct action for the trouble cannot be taken, record the following items, and notify the company as immediately as possible.

- Alarm code and status code and the accompanies data with them.
- The state and phenomenon of the trouble.
- The operational procedures just before the trouble have occured and number of applied tape.
- Whether the trouble occurs again or not in each time, the operation is repeated after depressing the RESET key.
- Date and time when the trouble occured.
- Name of the discover of the trouble and the operator.

While your are contacting the company, maintain the control in the condition when the trouble has occured as same as possible. Avoid turning off control power by depressing POWER OFF button, if the situation allows. Because the company may ask you to inspect some parts of the control.

APPENDIX-1

INTERFACE FOR M-, S-, T-, AND B-CODE

1. M CODE

YASNAC 3000G equipped with a standard interface sends M-BCD output and M- decoded output signals to the machine. In addition, DEN output and FIN input signals for the common signal of M-, S-, and T- code are also fed.

- DEN: Completion of tool move
- FIN: Completion of M-, S-, and T-function

NOTE: Where the control is equipped with optional machine interface†, the above signals are processed by the sequence control section of the interface, and then are transmitted to the machine. Signals for S- and T- code are also processed by the same way.

- M-BCD Output

  a. When the block containing M code is executed, M-BCD output signal is sent to the machine. But the command to read the code, MF is sent after "t" msec. This delay time, "t" msec is set by parameter No. 98X.

  b. Sending of M-BCD signal is stopped when FIN signal is fed back to the control. The reset or mode select operation also stops sending of M-BCD signal.

  c. To execute M-function after completion of a move command in the same block, use DEN and MF signal as the command to read the code.

- M-Decoded Output (M00, M01, M02 and M30)

  a. When a move command and M command are given in the same block, M-decoded signal is fed to the machine after the completion of tool move.

  b. If a move command is not given in the block, M-decoded signal and M-BCD signals are fed simultaneously.

  c. Sending of M-decoded signal is stopped by the Cycle Start or reset operation. But feedback of FIN signal does not stop M-decoded signal.

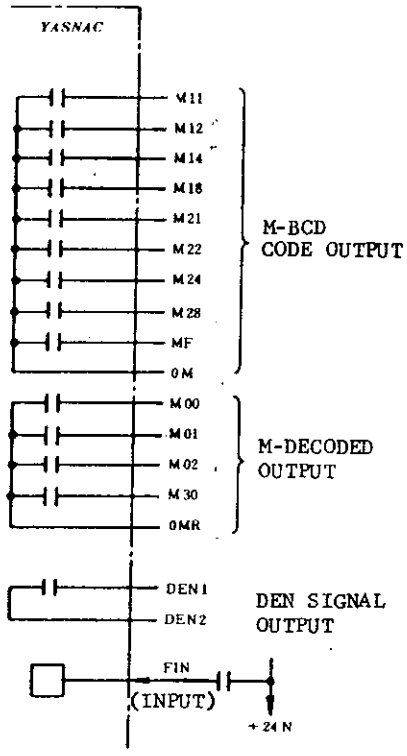


Fig. 1.1

· DEN Output Signal for Completion of Move

a. When a move command and M-, S-, or T- code are given in the same block, DEN signal is fed to the machine after completion of move. If FIN signal is fed back to the control during a movement of tool, DEN signal is not sent out.

b. DEN signal is fed to the machine together with BCD output when M-, S-, or T- code is commanded without move command.

c. Sending of DEN signal is stopped by the feedback of FIN signal, the reset operation or the mode select operation.

Example of M00

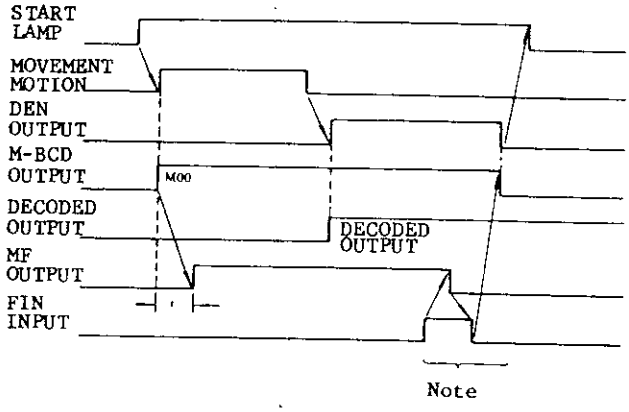


Fig. 1.2

Treat the FIN signal as follows. The control turns off MF signal when FIN input signal is turned on.

NOTE: Turns off FIN signal when MF signal is turned off. M-BCD and DEN signals are reset and the block is completed at the falling edge of FIN input signal.

Example of M30 (Where including the reset or rewinding operation)

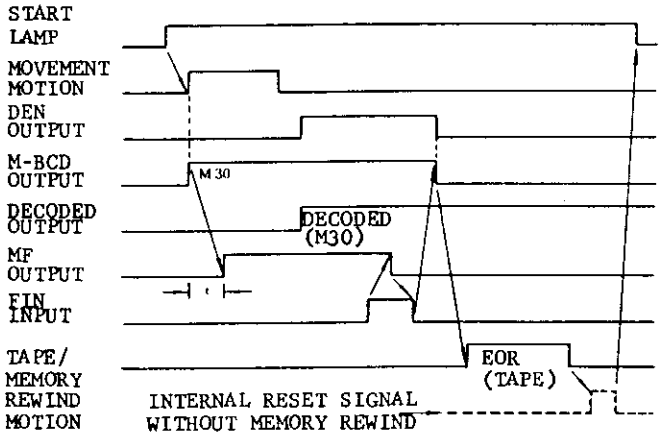


Fig. 1.3

It is determined in accordance with the following condition whether reset or rewind is executed or not by M02 or M30 command.

At the rising edge of FIN input signal,

The input signal EOP has been turned on: The internal reset is executed without a memory rewind, with LABEL SKIP lamp turned off.

The input signal RWD has been turned on: With MODE SELECT switch set to TAPE or MEM mode, the memory rewind is executed and LABEL SKIP lamp lights. In TAPE mode, the tape rewind is executed.

Fig. 1.4 shows an example of sequence control which uses M02 and M30.

M02: For resetting
M30: For resetting and rewinding

By means of the delay timer, EOP and RWD input signals should be set to ON or OFF at least 100 msec before the control receives FIN signal.

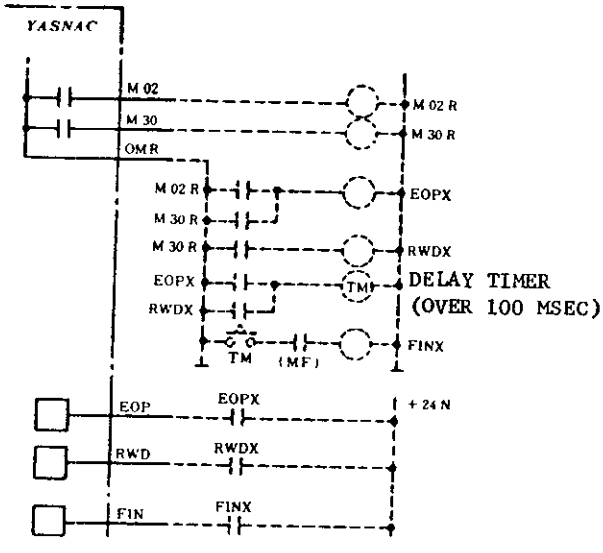


Fig. 1.4

· Precaution

Since M-codes shown below are processed internally by the control, these BCD and decoded output are not sent to the machine.

M90 to M99

2. S CODE

· S2-Digit Command

S2-digit BCD output signal is sent to the machine at the equal timing as that of M-BCD output except the following description. Sending of the BCD signal continues until a new S2-digit command is given, even if the control receives FIN signal.

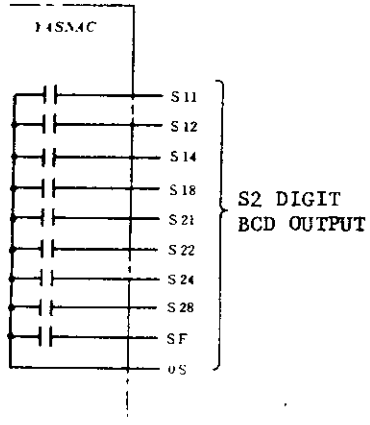


Fig. 1.5

· S4-Digit Command+

Where S4-digit command is used, 12 bits binary non-contact or spindle D/A output signal is sent to the machine. The case where the gear ratio is automatically changed is described below.

a. Interface and parameters

Fig. 1.6 shows the relation between S command output and spindle-speed ranges selected by gear ratio.

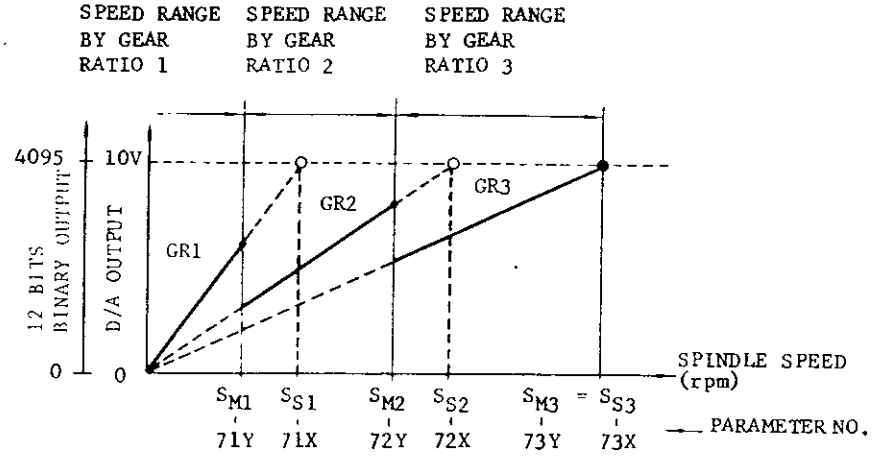


Fig. 1.6

S_{S1} to S_{S3}: Spindle D/A saturation speed for each gear ratio. The grade of D/A output for each gear ratio is determined with the contents of the corresponding parameter No. (71X, 72X...) which are written in rpm.

S_{M1} to S_{M3}: Maximum spindle speed for each gear ratio. Maximum spindle speed for each gear ratio is determined with the contents of parameter No. (71Y, 72Y...) which are written in rpm.

- 12 Bits binary output: Sn

Sn is obtained as follows.

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

{ S: Spindle speed (rpm) programmed by S
S_{Sn}: Spindle D/A saturation speed (rpm) for each gear ratio

Note: Some controls may have output with 1 bit for sign and 11 bits. In this case, Sn is obtain ed as follows.

$$Sn' = + \frac{2047}{S_{Sn}} \times S$$

However, when SINV input is on, negative binary output is sent.

- Spindle D/A output: Vn

$$Vn = \pm\frac{10}{S_{Sn}} \times S$$

{ S: Spindle speed (rpm) programmed by S
S_{Sn}: Spindle D/A max. speed for each gear ratio

However, when SINV input is on, negative voltage is output.

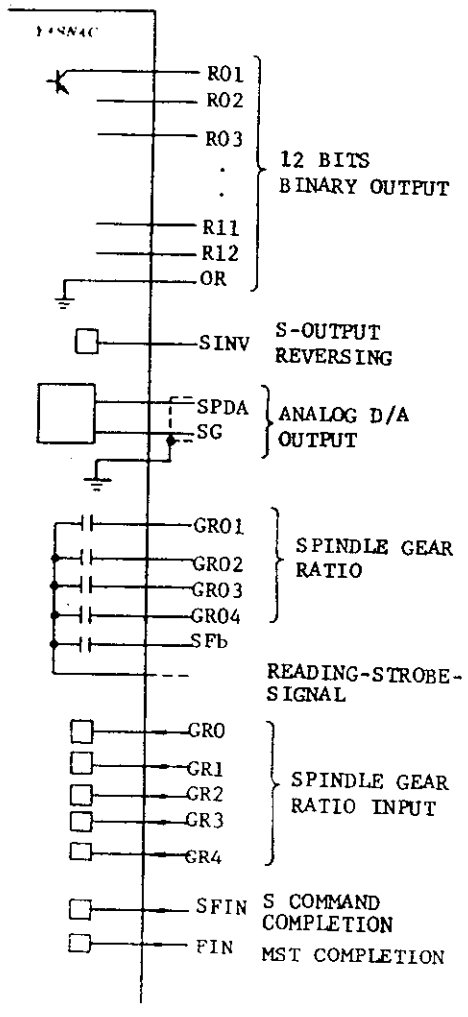


Fig. 1.7

· Spindle gear ratio input (GR1 to GR4) and S_{Sn}

When spindle output Sn or Vn described above is computed in the control, spindle gear ratio input (GR1 to GR4) determines. S_{Sn} to be used which are set according to each gear ratio. For example, if GR1 input is on at the timing of rising edge of SFIN signal (S command completion input), calculation is made using S_{S1} (= the contents of parameter 71X).

b. Timing of S command and output

Shown below is ON-OFF timing of signals, in the case of that the control judges whether gear ratio must be changed or not and performs necessary changing of gear ratio automatically to obtain the described spindle speed.

When

S1000 (rpm) M03* ··· Gear ratio in the speed range of GR2.

is programmed in the previous block and

S2000 (rpm) * ··· Gear ratio in the speed range of GR03.

is newly commanded.

· By the contents of parameter in Fig. 1.6, the control judges that GR2 must be changed to GR3 and gives output GR03.

· Read GR03 by turning on of Reading-strobe-signal SFb delayed by t and change the gear ratio.

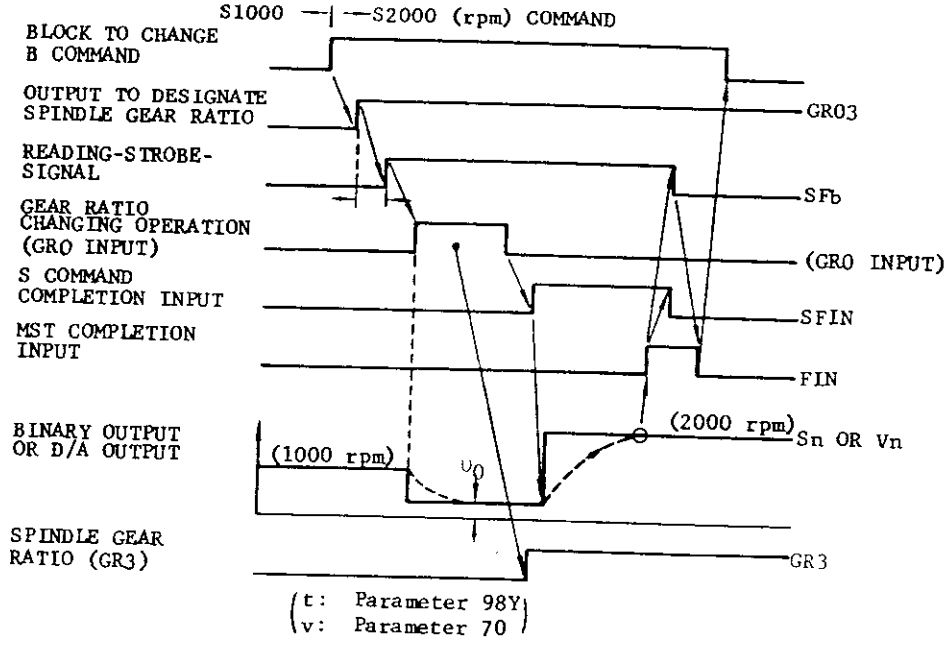


Fig. 1.8

Turn on GR0 input when spindle motor must be turned with a certain constant speed to change gear ratio. The spindle motor runs at the speed of V_0 which is specified with the parameter No. 70.

· Turn on GR3 input after gear ratio is changed to GR3 and then, turn on S command completion input (SFIN).

· The new S command (Sn or Vn) is computed and sent at the timing of rising edge of SFIN. The new S command is retained until the next S command is given.

· Turn on MST completion input FIN at the time when the spindle speed reaches the commanded speed (2000 rpm).

NOTE

· When GR0 to GR4 inputs are on at the same time, the priority is given by the following order in sending signal.

GR0 > GR4 > GR3 > GR2 > GR1

Priority ←——

· At the rising edge, FIN signal has the same function with SFIN which sends Sn or Vn.

· Spindle output specified with the parameter No. 70 is sent out of relation to SFIN immediately after GR0 input is turned on and is retained until SFIN signal is turned on.

· When GR0 input is not turned on, S command output of the previous block (1000 rpm in Fig. 1.8) is retained until SFIN signal is turned on.

3. T CODE

T2-digit BCD output signal for T2-digit command or T4-digit BCD output signal for T4-digit command is fed for tool selection with the equal timing as that of BCD output for M code except a next description. This BCD signal continues until a new T command is given, even if FIN signal is fed back to the control.

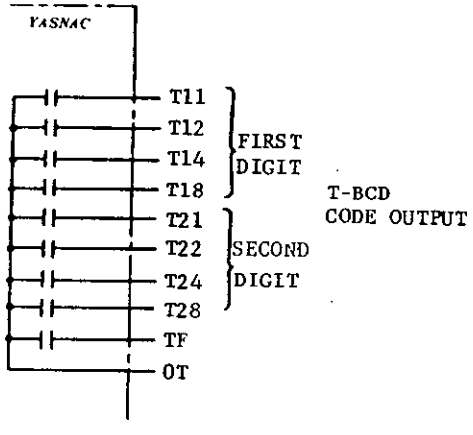


Fig. 1.9

Table 1.1 List of M-, S-, and T-code Output

Output Signal	DEN	M-decoded	M-BCD	S-BCD	S12 Bits or D/A Output	T-BCD
Command	M-, S-, and T-code	M00, M01 M02, M30	M code Excluding internally processed code	S2-digit command code	S4-digit command code	T-code for selecting the tool
Start Timing of Signal	After executing the tool move command in the same block.		At the start of the execution of commanded block.			
By FIN Signal	Reset	Held	Reset	Held		
By RESET Operation	Reset	Reset	Reset	Held		
By MODE SELECT Operation	Reset	Held	Reset	Held		
By CYCLE START Operation		Reset				
By Emergency Stop	Reset			Held		

4. B CODE

B codes are sent in 3-digit BCD output. The timing of sending and the conditions to be reset and held are the same with T codes.

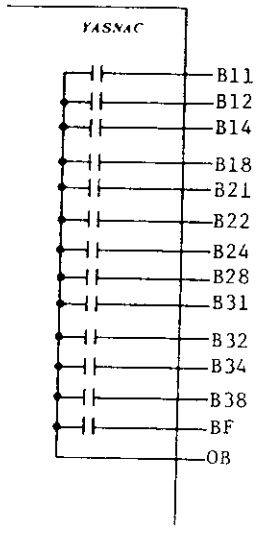


Fig. 1.10

APPENDIX-2

SPINDLE CONTROL IN CANNED CYCLES

In some canned cycles, the signals for spindle to command stop, reverse or start of running at the hole bottom or after retraction are fed. The signal is sent by M code or exclusion signal which depends on the control.

The following canned cycles require spindle control.

Table 2.1

G Codes	At the Hole Bottom	After Retraction
G84	Spindle reversing after dwell	Spindle reversing again
G86	Spindle stop	Spindle start
G87	Spindle stop	Spindle start
G88	Spindle stop after dwell	Spindle start
G76	Spindle indexing	Spindle start after shift

1. METHOD USING M CODE

M codes are sent as follows.

Table 2.2

G Codes	At the Hole Bottom	After Retraction
G84	M05* ⟶ M04 (M03)	M05** ⟶ M03 (M04)
G86	M05	M03 (M04)
G87	M05	M03 (M04)
G88	M05	M03 (M04)
G76	M05 ⟶ M19	M03 (M04)

* Shown above is the case that M03 was commanded before entering canned cycle. When M04 was commanded, M code in parentheses is sent. If neither of them is not commanded.

** Sending of M05 at G84 may be omitted, depending on the control.

The timing of sending of MF, DEN, and FIN signals accompanied by M codes are shown below.

a. In the case of G84 (Dwell time = 0)

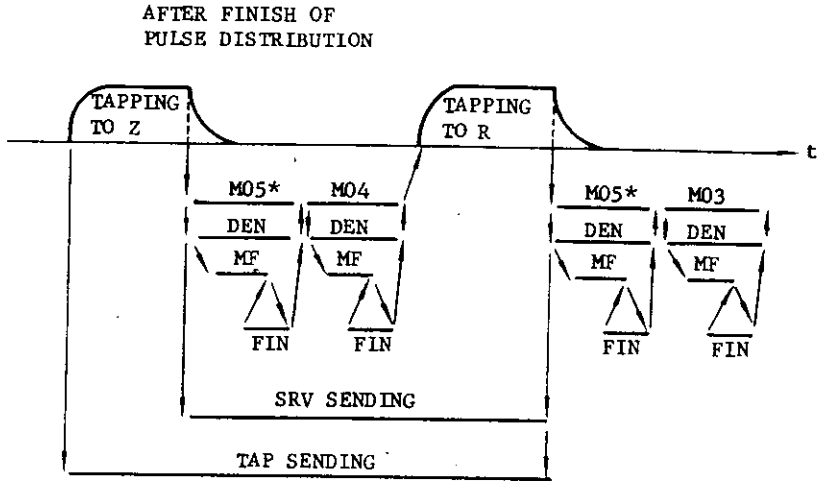


Fig. 2.1

* Sending of M05 at G84 mav be omitted, depending on the control.

b. In the case of G86, G87 and G88 (Dwell time = 0)

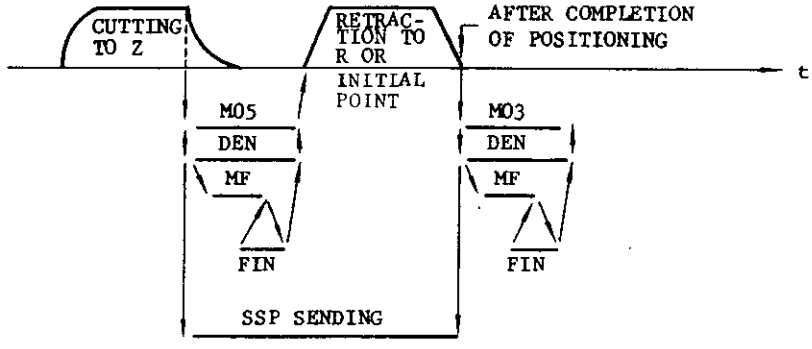


Fig. 2.2

c. In the case of G76

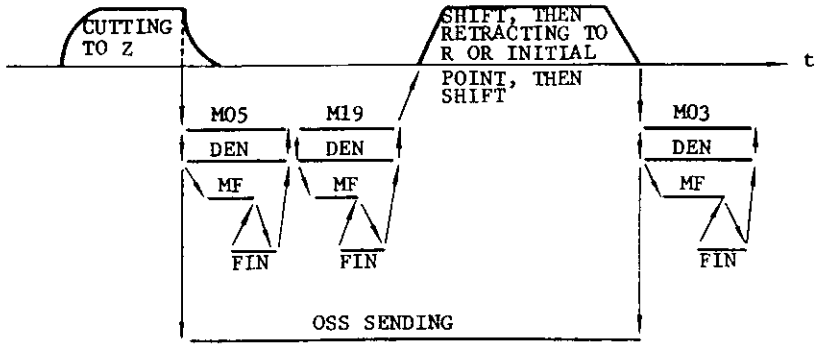


Fig. 2.3

2. METHOD USING EXCLUSIVE SIGNAL

The exclusive signals for spindle control for canned cycles shown below are used.

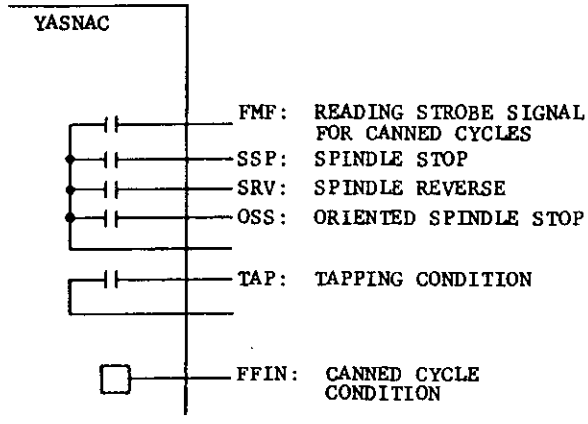


Fig. 2.4

Exclusive signals are sent as follows.

Table 4.3

G Codes	At the Hole Bottom	After Retraction
G84	SRV, FMF — ON	SRV — OFF
G86	SSP, FMF — ON	SSP — OFF
G87	SSP, FMF — ON	SSP — OFF
G88	SSP, FMF — ON	SSP — OFF
G76	OSS, FMF — ON	OSS — OFF

- FFIN is used as completion output for FMF output.
- The status code "58" is displayed until FFIN is given.
- After retracting motion, the spindle should be turned on and the auxiliary motion at the time when SRV, SSP or OSS is turned off.

Timing of sending of exclusive signal is shown below.

a. In the case of G84 (Dwell time = 0)

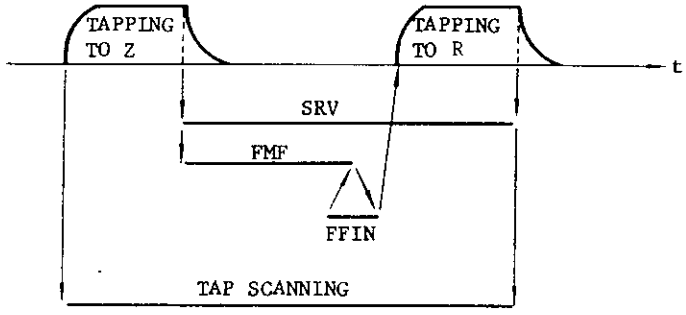


Fig. 2.5

b. In the case of G86, G87 and G88 (Dwell time = 0)

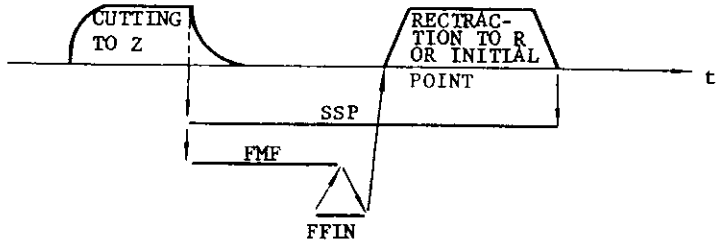


Fig. 2.6

c. In the case of G76

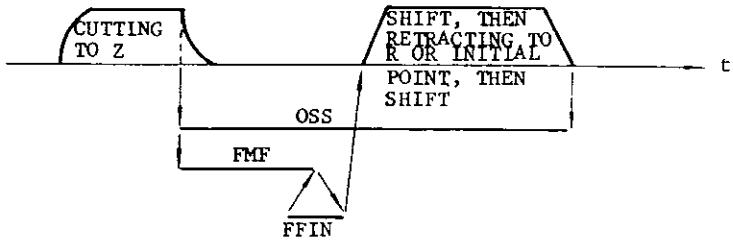


Fig. 2.7

APPENDIX-3

LIST OF PARAMETERS

付 3·1 表　パ ラ メ ー タ ー 覧 (1/10)

PARAMETER TABLE (1/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意　味 MEANING	初 期 値 (例) INITIAL VALUE (EXAMPLE)
00		稼動時間表示 Operation Time Display ○○○○, ○○, ○○ 時(H),分(M),秒(S)	
01		未使用 Not Used	0
02		ドライラン時早送り速度 Rapid Traverse Rate for Dry Run 〝0〟= RAPID, 〝1〟= JOG	0
03†		ストローク チェック入切 Stroke Check ON-OFF 〝0〟= OFF, 〝1〟= ON	0
04†		マシンロック時, ストローク チェック入切 Stroke Check ON-OFF for Machine Lock 〝0〟= OFF 〝1〟= ON	0
05		ピッチエラー補正入切 Leadscrew Error Compen ON-OFF 〝0〟= OFF, 〝1〟= ON	0
06†		サイクル スタート前の原点復帰 Reference Zero Return before Cycle Start 〝0〟= 不 要, 〝1〟= 要 Not Required Required	0
07		未使用 Not Used	0
08		未使用 Not Used	0
09		RESET時のAグループGコード状態 A Group G Code When Resetting 〝0〟= G00, 〝1〟= 保存. Held	0

パラメーター一覧 (2/10)
PARAMETER TABLE (2/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 (例) INITIAL VALUE (EXAMPLE)
10†	X	ピッチエラー補正量 Leadscrew Error Compen. Values '1' = 0.001 mm	0
	Y		0
	Z		0
	4		0
11†	X		0
	Y		0
	Z		0
	4		0
12†	X		0
	Y		0
	Z		0
	4		0
13†	X		0
	Y		0
	Z		0
	4		0
14†	X		0
	Y		0
	Z		0
	4		0
15†	X		0
	Y		0
	Z		0
	4		0
16†	X		0
	Y		0
	Z		0
	4		0
17†	X		0
	Y		0
	Z		0
	4		0
18†	X		0
	Y		0
	Z		0
	4		0
19†	X		0
	Y		0
	Z		0
	4		0

パ ラ メ ー タ ー 覧 (3/10)
PARAMETER TABLE (3/10)

パラメータ番号 RARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 (例) INITIAL VALUE (EXAMPLE)
20†	X	ピッチエラー補正量 Leadscrew Error Compen. Vales "1" = 0.001 mm	0
	Y		0
	Z		0
	4		0
21†	X		0
	Y		0
	Z		0
	4		0
22†	X		0
	Y		0
	Z		0
	4		0
23†	X		0
	Y		0
	Z		0
	4		0
24†	X		0
	Y		0
	Z		0
	4		0
25†	X		0
	Y		0
	Z		0
	4		0
26†	X		0
	Y		0
	Z		0
	4		0
27†	X		0
	Y		0
	Z		0
	4		0
28†	X		0
	Y		0
	Z		0
	4		0
29†	X		0
	Y		0
	Z		0
	4		0

パラメーター一覧 (4/10)
PARAMETER TABLE (4/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 (例) INITIAL VALUE (EXAMPLE)
30†	X	ピッチエラー補正量倍率 Multiply for Leadscrew Error Compen. 〝1〟＝1倍	0
	Y		0
	Z		0
	4		0
31†	X	ピッチエラー補正原点番号 Refference Point No. for L. E. C 〝20〟＝ Parameter No. 20	0
	Y		0
	Z		0
	4		0
32†	X	ピッチエラー補正の補正ピッチ Compensational Pitch for L. E. C 〝1〟＝ 0.001 mm	0
	Y		0
	Z		0
	4		0
33†	X	第1ストロークチェック座標（最大値） First Stroke Check Point (Max.) 〝1〟＝ 0.001 mm	0
	Y		0
	Z		0
	4		0
34†	X	第1ストロークチェック座標（最小値） First Stroke Check Point (Min.) 〝1〟＝ 0.001 mm	0
	Y		0
	Z		0
	4		0
35†	X	第2ストロークチェック座標（最大値） Second Stroke Check Point (Max.) 〝1〟＝ 0.001 mm	0
	Y		0
	Z		0
	4		0
36†	X	第2ストロークチェック座標（最小値） Second Stroke Check Point (Min.) 〝1〟＝ 0.001 mm	0
	Y		0
	Z		0
	4		0
37	X	未使用 Not Used	0
	Y		0
	Z		0
	4		0
38	X	未使用 Not Used	0
	Y		0
	Z		0
	4		0
39†	X	ストロークチェック領域指定 Area Select for Stroke Check （第4-11項「領域チェック機能」を参照 Refer to 4-11 Strok Check）	0
	Y		0
	Z		0
	4		0

パラメーター一覧 (5/10)
PARAMETER TABLE (5/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 (例) INITIAL VALUE (EXAMPLE)
40†		シーケンス制御用パラメータ Parameters for Optional Machine Interface† '0' = OFF, '1' = ON	0
41†			0
42†			0
43†			0
44†			0
45†			0
46†			0
47†			0
48†	X		0
	Y		0
	Z		0
	4		0
49†	X	システム番号 System Number	0
	Y	未使用 Not Used	0
	Z		0
	4		0

パラメーター一覧 (6/10)
PARAMETER TABLE (6/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 (例) INITIAL VALUE (EXAMPLE)
50†		※ シーケンス制御用タイマ定数 Timer Constant for Optional Machine Interface † $^{S}N^{S}$ = (16n − 16) msec	0
51†			0
52†			0
53†			0
54†			0
55†			0
56†			0
57†			0
58†	X	未使用 Not Used	0
	Y		0
	Z		0
	4		0
59†	X	未使用 Not Used	0
	Y		0
	Z		0
	4		0

パラメーター一覧 (7/10)
PARAMETER TABLE (7/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 (例) INITIAL VALUE (EXAMPLE)
60	X	バックラッシュ補正開始方向 Backlash Comp. Starting Direction '0' = 正から from (+)　'1' = 負から from (−)	0
	Y		0
	Z		0
	4		0
61†	X	原点復帰方向 Reference Zero Return Direction '0' = 正方向 to (+)　'1' = 負方向 to (−)	0
	Y		0
	Z		0
	4		0
62†	X	原点復帰最終速度 Reference Zero Return Final Speed '1' = 7.5 mm/min	40
	Y		40
	Z		40
	4		40
63	X	ポジションエラー領域 Position Error Zone '1' = 0.001 mm	32
	Y		32
	Z		32
	4		32
64	X	サーボエラー領域 Servo Error Zone '1' = 2^8 パルス (Pulses) または 2^7 パルス (Pulses) (装置による) Depend on the Control	16
	Y		16
	Z		16
	4		16
65	X	仕様指定用パラメータ (バイナリー) Designation of Optional Features (Binary)	0
	Y		0
	Z		0
	4		0
※ 66†		G12～G15 R指定区間送り速度 R Zone Feedrate '1' = 1/16 mm/min	48000
※ 67†	X	手動ハンドル送り最大速度 Max. Feedrate for Handle '1' = 7.5 m/min	1600
	Y		1600
	Z		1600
	4		800
68		切削送り時定数 Time Constant for Feed 'N' = (32n + 32) msec	0
69		切削送り速度バイアス Velocity Bias for Feed '1' = 2 Kpps	10

パラメーター一覧 (8/10)
PARAMETER TABLE (8/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 (例) INITIAL VALUE (EXAMPLE)
70†		GR0 主軸D/A出力値 Spindle DA Output 〝2047〟＝10V	0
71†	X	GR1 主軸DA飽和回転数	0
	Y	GR1′ 主軸最大回転数	0
	Z	Spindle Max. Speed	0
	4	〝1〟＝1RPM	0
72†	X	GR2 主軸DA飽和回転数	0
	Y	GR2′ 主軸最大回転数	0
	Z	Spindle Max. Speed	0
	4	〝1〟＝1RPM	0
73†	X	GR3 主軸DA飽和回転数	0
	Y	GR3′ 主軸最大回転数	0
	Z	Spindle Max. Speed	0
	4	〝1〟＝1RPM	0
74†	X	GR4 主軸DA飽和回転数	0
	Y	GR4′ 主軸最大回転数	0
	Z	Spindle Max. Speed	0
	4	〝1〟＝1RPM	0
75	X	切削送り最高速度 Max. Feedrate 〝1〟＝1/16mm/min	48000
	4		24000
76†		G76 シフト方向 Shifting Direction 〝1〟＝X+, 〝2〟＝X−, 〝4〟＝Y+, 〝8〟＝Y−	1
77†		補正Cコーナ円弧廻りスキップ値 〝1〟＝最小設定単位 (1倍) Least Input Increment(×1)	5
78		G90/G91 イニシャルセット Initial Set 〝0〟＝G90, 〝1〟＝G91	0
79		G92 表示プリセット入切 Display-Reset ON-OFF 〝0〟＝OFF, 〝1〟＝ON	0

パラメーター一覧 (9/10)
PARAMETER TABLE (9/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値 (例) INITIAL VALUE (EXAMPLE)
80		EIA/ISOの自動判別入切 - Auto Select '0' = OFF, '1' = ON	1
81		TVチェック入切 TV Check ON-OFF '0' = OFF, '1' = ON	0
82		EIA/ISOコード指定 Code Disignation '0' = EIA, '1' = ISO	0
83†		MM/INCH指定 Designation '0' = MM, '1' = INCH	0
84		タッチブザー入切 Touch Buzzer ON-OFF '0' = OFF, '1' = ON	1
85		未使用 Not Used	0
86		高速原点復帰入切 Rapid Reference Return ON-OFF '0' = OFF, '1' = ON	0
87		送り指令単位 Feedrate Designation '0' = F4.0, '15' = F4.1 mm/min	0
88		入力指令10倍入切 Command Data × 10 ON-OFF '0' = OFF, '1' = ON	0
89		未使用 Not Used	0

パラメーター一覧（10/10）
PARAMETER TABLE (10/10)

パラメータ番号 PARAMETER No.	アドレス ADDRESS	意味 MEANING	初期値（例） INITIAL VALUE (EXAMPLE)
90	X	バックラッシュ補正量 Backlash Value 1 = 0.001 mm	0
	Y		0
	Z		0
	4		0
91	X	ミラーイメージ軸指定 Miror Image Axis ON-OFF '0' = OFF, '1' = ON	0
	Y		0
	Z		0
	4		0
92†	X	外部減速速度（RPG） Traverse Rate Clamp Speed (RPG) '1' = 7.5 mm/min	0
	Y		0
	Z		0
	4		0
93†	X	外部減速速度（FG） Traverse Rote Clamp Speed (FG) '1' = 1/16 mm/min	0
	4		0
94	X	早送り速度 Rapid Traverse Rate '1' = 7.5 mm/min	1600
	Y		1600
	Z		1600
	4		800
95	X	早送り加減速定数 Accel./Decel. Time Const. for RT '1' = 125/8 mm/sec²	60
	Y		60
	Z		60
	4		30
96†	X	原点復帰クリープ速度 Reference Zero Return Approaching Speed '1' = 7.5 mm/min	40
	Y		40
	Z		40
	4		20
97†	X	原点復帰最終距離 Reference Zero Return Final Stroke '1' = 0.001 mm	5000
	Y		5000
	Z		5000
	4		2500
98	X	MF, SF, TF, BF 送出遅れ時間 SFb 送出遅れ時間 Delay Time for MF, SF, TF, BF and SFb '1' = 0.001 sec	120
	Y		120
	Z		0
	4		0
99†		メモリポインタ表示 Memory Pointer Display '1' = 1 ch	

備考 1
Notice 1

備　考
NOTICE

1）No. 00 および 99 は表示専用であり，書込みはできません。
表示は常時可能です。

No. 00 & 99 are for display only.
These cannot be written, but can be displayed anytime.

2）No. 01～98 は SYSTEM No. スイッチによってインタロックされており，同スイッチを〝1〟の位置にして書込みを行ないます。

No. 01～98 are interlocked by SYSTEM No. Swich, and can be changed only when the position of SYSTEM No. switch is 〝1〟.

3）† 印の付されたパラメータはオプションです。

Parameters with † Mark are optional.

4）ADDRESS 部が  のパラメータは軸指定を要しません。

Parameters with  mark in ADDRESS does not require the selection of Axis.

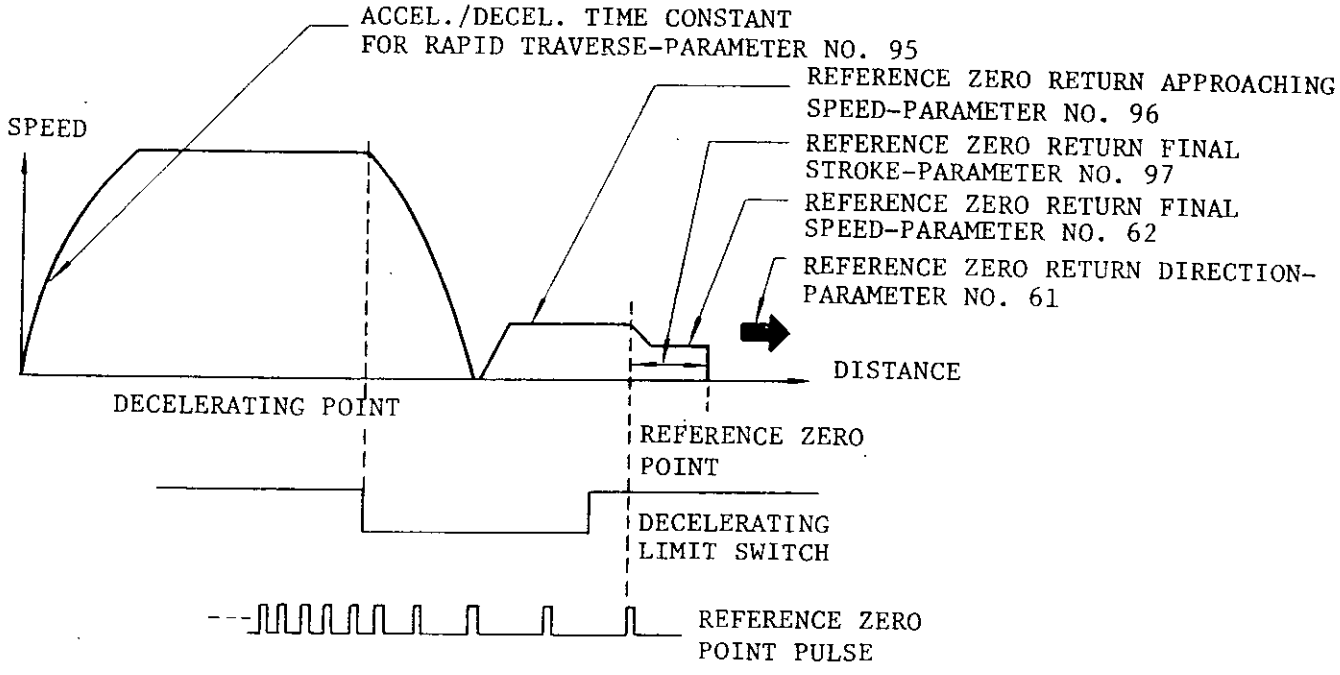

ACCEL./DECEL. TIME CONSTANT
FOR RAPID TRAVERSE-PARAMETER NO. 95
REFERENCE ZERO RETURN APPROACHING
SPEED-PARAMETER NO. 96
REFERENCE ZERO RETURN FINAL
STROKE-PARAMETER NO. 97
REFERENCE ZERO RETURN FINAL
SPEED-PARAMETER NO. 62
REFERENCE ZERO RETURN DIRECTION-
PARAMETER NO. 61
SPEED
DISTANCE
DECELERATING POINT
REFERENCE ZERO
POINT
DECELERATING
LIMIT SWITCH
REFERENCE ZERO
POINT PULSE

Fig. 3.1

APPENDIX-4

TROUBLE CAUSES AND REMEDIES

Table 4.1 Alarm Codes and Remedies

Alarm code	Causes	What to do
11 Excessive temperature rise in the panel.	· Ambient temperature exceeding 45°C (113°F). · Failure in cooling system such as fan.	· Depress RESET button after lowering ambient temperature, and alarm status is released. · Contact maintenance personnel, if necessary.
12 Tape/Memory horizontal parity error.	· Mispunched or dusty tape. · Reading error of tape reader. · Even holes in EIA code. · Blank in EIA code. · Odd holes in ISO code. · Memory error due in the poor battery, in MEM mode. (3rd and 4th digits: IC number)	· Depress RESET button, and alarm status is released. · Correct the tape. · Clean the head of tape reader. · Where the error occures in MEM mode or in EDT mode, inspect the control, especially, battery.
13 Tape vertical parity error.	· Odd number of significant characters in one block when TV check is on (parameter No. 81 = "1"). · Dusty tape. · Reading error of tape reader.	· Depress RESET button, and alarm status is released. · Correct the tape. · Clean the head of tape reader. · If TV check is not required, set the contents of parameter No. 81 to "0."
14 Format error.	· Illegal characters are used. · Illegal G code is used.	· Depress RESET button, and alarm status is released. · Clean the head of tape reader. · Correct the program. For the detailed description of alarm code 14, see the end of this table.
15 Data error.	· G code is not programmed correctly. · Circular arc is not programmed correctly. · Calculation error occurs.	· Depress RESET button, and alarm status is released. · Correct the program. For the detailed description of alarm code 15, see the end of this table.
16 Offset error.	· Tool offset values are destroyed due to poor battery.	· Make sure that BATTERY lamp is off. If lighting, contact maintenance personnel.

Table 4.1 Alarm Codes and Remedies (Continued)

Alarm code	Causes	What to do
17 Parameter error.	· Contents of parameter are destroyed due to poor battery.	· If BATTERY lamp is lighting, contact maintenance personnel.
18 Tape memory error.	· Disagreement between tape and memory at NC tape collating. · Memory capacity over.	· Depress RESET button, and alarm status is released.
21 Overtravel.	· The machine is at the end of the stroke. · The third and fourth digits of ALARM CODE indicate the axis.	· Move it back in stroke by manual operation and depress RESET button.
22 Reference Zero Return area error.	· Zero Return action is commanded in the area where decelerating switch is effective. · The third and fourth digits of ALARM CODE indicate the axis.	· Depress RESET button, and alarm status is released. · Put the machine out of the area by manual operation.
23 Reference Zero Return unready.	· CYCLE START pushbutton is depressed before performing Zero Return with the control interlocked for Zero Return. (parameter No. 06 = "1") · The third and fourth digits of ALARM CODE indicate the axis.	· Depress RESET button, and alarm status is released. · Perform Zero Return. · If interlocking is unnecessary, set the contents of parameter No. 06 to "0."
24 Reference Zero Return position error.	· Disagreement between reference zero and programmed point at G27. · In manual Reference Zero Return or G28 for automatic Reference Zero Return, positioning error. · The third and fourth digits of ALARM CODE indicate the axis.	· Depress RESET button, and alarm status is released. · Check the program. · For the error in automatic operation, contact maintenance personnel.
25 Sequence error.	· The program which is prohibited by the machine interface is executed. For example, when the spindle is running, the reverse running of the spindle is commanded.	· Depress RESET button, and alarm status is released. · Correct the program.
26 Spindle error.	· Spindle drive is affected due to failure in fuse or thermal relay, and ambient temperature rise.	· After eliminating the cause, depress RESET botton. · Contact maintenance personnel.

Table 4.1 Alarm Codes and Remedies (Continued)

Alarm code	Causes	What to do
27 Positioning error.	· Failure in mechanical or servo system.	· Check the machine and servo systems. Then depress RESET button. · Contact maintenance personnel.
28 Machine unready.	· MRD (machine ready) signal is not received after power is turned on. · MRD signal is temporarily eliminated on the way because of abnormal mechanical sequence.	· If MRD signal comes when turning on power, the alarm is soon released automatically. · On the other causes besides above, check the machine and machine sequence. · If temporarily eliminated signal, reset operation is available. In addition, contact maintenance personnel.
31 Servo power unsupplied.	· Servo power is not supplied.	· Depress the POWER ON button according to procedure turning on power, and the alarm is released automatically.
32 Control unit unready.	· For diagnostic of control system, the control is not yet ready for turning on servo power supply. · Servo system is not sufficiently adjusted.	· When the control is ready, alarm code turning to "31" automatically. · Contact maintenance personnel if alarm code "32" remains on the display for several seconds and above.
33 Emergency stop.	· Emergency stop button is depressed.	· Reset the system, and a alarm code turns to "31." · Depress POWER ON button for turning on servo power supply. · Servo power can be turned on before reset operation. But the alarm code remains on the display until performing reset operation.
34 Servo error.	· Excessive cutting command. · Excessive servo lag. · Defective servo system and mechanism. · The third and fourth digits of ALARM CODE indicate the axis.	· Correct the program. · Reset the system, and alarm code turns to "31" automatically. Servo power can be turned on. · Check servo system and mechanism.
35 Overload.	· Excessive cutting command. · Thermal overload relay tripped due to motor overloaded.	· Check the thermal overload relay and fuses in drive unit. · After cooling, reset the relay or replace the fuse with new one.

Table 4.1 Alarm Codes and Remedies (Continued)

Alarm code	Causes	What to do
35 Overload. (Continued)	· Blown out fuses. · Temperature in servo control unit rises excessively. · Defective servo system or mechanism. · The reset operation is executed during rapid traverse operation.	· Eliminate the cause and then reset the system, and alarm code turns to "31." Servo power supply can be turned on. · Correct the program. · Inspect servo system and mechanism.
36 Feedback error.	· Poor feedback encoder for X-, Y- or Z-axis. · Cable for encoder is disconnected. · The third and fourth digits of ALARM CODE indicate the axis.	· Contact maintenance personnel.
37 Hardware error. (FG error)	· Failure in package of type GCP03 or GSR02, printed circuit board in the control.	· Contact maintenance personnel.
38 Hardware error. (RPG error)	· Failure in package of type GCP03 or GSR02, printed circuit board in the control.	· Contact maintenance personnel.
81 CPU error.	· Impossible to perform the operation due to CPU trouble.	· Make the memorandum of contents indicated on UNIVERSAL DISPLAY, and contact maintenance personnel.
82 Memory collating error.	· System program stored in the memory is destroyed.	· Contact maintenance personnel. · The first and second digits show the number of poor ROM.
91 Contents disagreement of tape and memory. (For off-line only.)	· Contents of system memory and source tape are not equal. SYSTEM NO. "2": CHECK	· Replace or repair ROM or the package. · The first and second digits show the contents of memory. · The numbers displayed in NUMBER mean the tape contents.
92 Tape reading error (For off-line only.)	· Source tape is misread. SYSTEM NO. "2": CHECK SYSTEM NO. "3": LOAD	· Check the tape. · Clean the head of tape reader.

Detailed Display of Alarm Codes 14 and 15

Alarm codes 14 (format error) and 15 (data error) can provide the detailed description of the error by displaying the third and fourth digits.

Table 4.2

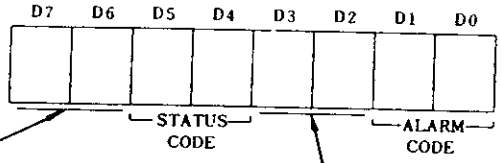
Format Error	Causes
0114 Buffer register overflow	· More than 96 characters are programmed in a block.
0214 Commanding of unusable G code	· An unusable G code is programmed.
0314 Address search error	· Address data to be searched by Address Search operation do not exist.
0414 Commanding of unusable characters	· Unusable characters or function characters are programmed.
0514 Tool offset tape input error	· Characters other than H, D and X are punched on the tape for tool offset value.
1514 Reset unready	· After editing in EDT mode, CYCLE START button is depressed in automatic operation mode without depression of RESET key.
0115 F code unspecified	· After turning on power supply, movement other than positioning is commanded without specifying F command.
0215 Erroneous programming of G code	· G43 or G44 is programmed in the modes other than G00 G01. · G45, G46, G47 or G48 is programmed in the modes other than G00, G01, G02 and G03. · Unusable G codes are programmed in G41 and G42 modes (G92, G28, G29, G81 to G89 etc.). · Unusable G codes are programmed in canned cycle mode (G92, G28, G29, G41, G42 etc.). · G codes of Δ group (except G40 to G49) and A group or D group are programmed in the same block.
0315 No data for operation	· In MEM operation mode, the program pointer comes into memory area where data for operation are not stored, caused by missing M02, M30 or the like.
0415 Subprogram error	· G25 is programmed in canned cycle mode. · M98 or M99 is programmed in the block containing G25. · Addresses P and Q are not programmed in the block containing G25. · Sequence numbers specified with P and Q are not in memory.

Table 4.2 (Continued)

Format Error	Causes
0415 Subprogram error	· G04 is programmed in the block containing M98. · P is not programmed in the block containing M98. · Sequence number specified with P is not in memory. · Sequence number specified with M99 P... * is not in memory. · G25 is programmed in the end block of program copyed by G25. · More than eight levels are commanded by G25, M98 or using then jointly in subprogram.
0515 Circular arc programming error (G02/G03)	· I, J, K and R are mixedly commanded in programming circular arc. · Radius value is zero or not specified though the end point of circular arc is designated. · The plane for circular interpolation is not defined caused by erroneous designation. (For example, circular arc is designated in three axes.) · In programming R designating circular arc, R value from which the center point cannot be obtained is commanded: $R < \frac{\text{Distance between start point and end point}}{2}$ The above R value causes error.
0615 Tool radius compensation C error	· Changing command of the plane for compensation. (G17 to G19) is made in G41 or G42 mode. · G41 or G42 is programmed in G02 or G03 mode. · G40 is programmed in G02 or G03 mode. · Point of intersection does not exist on the offset locus. · Reversing move command or command close to reversing is programmed in M97 mode. · Circular arc out of the plane specified for compensation is programmed.
0715 Computing error	· Calculation cannot be performed, caused by overflow or the like while intersection point is computed.
0815 Axis or R designation error	· Axis of X, Y, Z or α is programmed in the block containing G12, G13, G14, G15 or G04. · Designated R value with G12 or G13 is not in the following range. $\|R\| < \|I\|$ · Designated R value with G14 or G15 is not in the following range. $\|R\| < \|K\|$
0915 M, S or T code designation error	· M, S, T or B† code is programmed in the block containing canned cycle command in effect. · M, S, T or B† code is programmed in the block containing G12, G13, G14 or G15.
1015 Tape memory capacity error	· Operation data overflow the tape memory capacity caused by editing.

Table 4.2 (Continued)

Format Error	Causes
1115 Area check error	· Movement to the point out of prohibit area is commanded.
1215 G28 or G29 programming error	· G29 is programmed without execution of G28 after power on. · G28 or G29 is programmed in M95 (mirror image on) mode.



	M	S	T	B*
01	○			
02		○		
03	○	○		
04			○	
05	○		○	
06		○	○	
07	○	○	○	
08				
09				
10				○
11	○			○

	X	Y	Z	α*
01	○			
02		○		
03	○	○		
04			○	
05	○		○	
06		○	○	
07	○	○	○	
08				○
09	○			○
10		○		○
11	○	○		○

Table 4.3

Alarm Code	Lamp: INPUT ERROR	Lamp: NC ALARM	With the Condition of Tool Moving
11		○	Stop at block end
12	○		
13	○		
14	○		
15	○		
16	○		
17	○		
18	○		
21		○	Immediately decelerate and stop
22		○	
23		○	
24		○	
25		○	

Table 4.3 (Continued)

Alarm Code	Lamp: INPUT ERROR	Lamp: NC ALARM	With the Condition of Tool Moving
26		○	Immediately decelerate and stop
27		○	
28		○	
31		○	Emergency stop by servo power supply off
32		○	
33		○	
34		○	
35		○	
36		○	
37		○	
38		○	
01		○	• Servo power supply off • Emergency stop • CPU stop
02		○	

Table 4.4 List of Alarm Codes and Status Codes

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

Status Code	Status	Remarks
□□51	Performing M-, S-, T-, and/or B^{+}-function.	-
52	Distributing pulses. Dwelling.	-
□□53	Performing M-, S-, T-, and/or B^{+}-function and distributing pulses.	51 + 52
54	Reading tape.	-
□□55	Performing M-, S-, T-, and/or B^{+}-function and reading tape.	51 + 54
56	Distributing pulses and reading tape.	52 + 54
□□57	Performing M-, S-, T-, and/or B^{+}-function distributing pulses and reading tape.	51 + 52 + 54
58	Waiting for canned cycle's FIN signal.	-

Notes:

1. □□ of alarm code is filled with a digit indicating the axis whose error is detected. 1: X-axis, 2: Y-axis, 4: Z-axis, (8: α^{+}-axis)
   Where the error is detected in more than one axis, total of axis codes is shown.
2. For alarm code "12," additional two digits indicate the memory IC number.
3. □□ of alarm code "14," "15" shows the detailed error causes.
4. □□ of status code is filled with a digit indicating M, S, T, or B function which is being executed.
   1: M, 2: S, 4: T, (10: B^{+})
   Where the error is detected in more than one function, total of function codes is shown.

YASKAWA Electric Mfg. Co., Ltd.

TOKYO OFFICE Ohtemachi Bldg., Chiyoda-ku, Tokyo, 100 Japan
Phone (03) 217-4111 Cable address YASKAWAMOTOR TOKYO
Telex 222-2273YASKWA J

Printed in Japan 80-2 0.3 TA