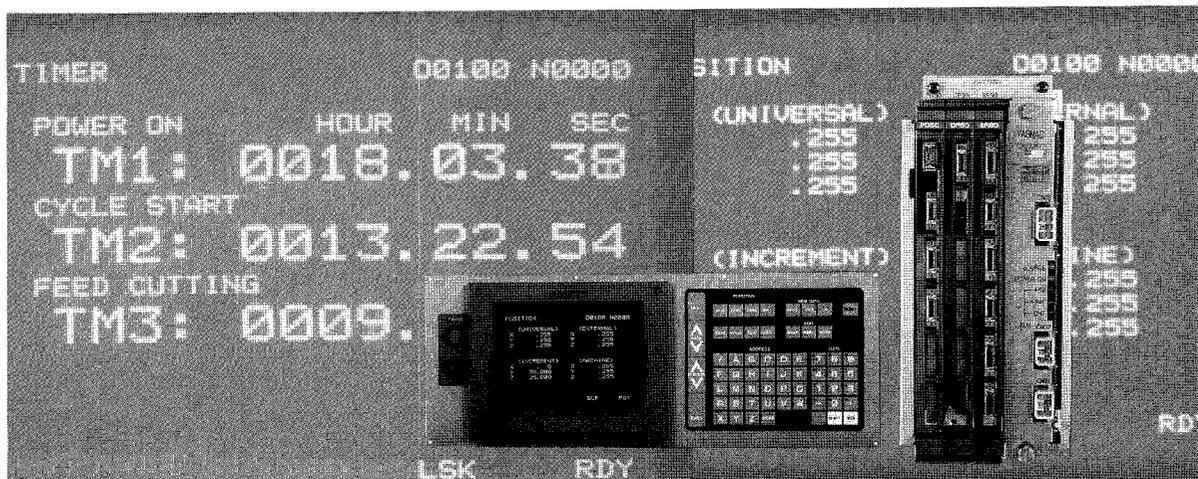


YASNAC J50M INSTRUCTIONS

CNC SYSTEM FOR MACHINING CENTERS



Before initial operation, read these instructions thoroughly, and retain for future reference

CONTENTS

	Page
1. PREFACE	1
2. PROGRAM MIN G	1
2.1 INPUT FORMAT	1
2.2 PROGRAM NUMBER AND SEQUENCE NUMBER	5
2.3 COORDINATE WORD	6
2.4 TRAVERSE AND FEED FUNCTIONS	9
2.5 SPINDLE-SPEED FUNCTION (S-FUNCTION)	11
2.6 TOOL FUNCTION (T-FUNCTION)	12
2.7 TOOL COMPENSATION	13
2.8 MISCELLANEOUS FUNCTIONS (M-FUNCTION)	14
2.9 PREPARATORY FUNCTION (G-FUNCTION)	19
2.10 USER MESSAGE DISPLAY †	93
2.11 uSERMACRO (G65, G66, G67)	94
2.12 SOLID TAP FUNCTION †	119
2.13 AUTOMATIC CORNER OVERRIDE	125
2.14 HIGH-SPEED CONTOURING FUNCTION*	127
3. PART PROGRAM TAPE CODING	130
3.1 TAPE CODE	130
3.2 PROGRAMMING	132
3.3 PART PROGRAM TAPE PUNCHING	133
3.4 PART PROGRAM TAPE HANDLING	133
4. NC OPERATOR'S PANEL WITH 9" CRT CHARACTER DISPLAY	134
4.1 PUSHBUTTONS, LAMPS AND KEYS	134
4.2 POWER ON/OFF OPERATION	137
4.3 DISPLAY AND WRITING OPERATION	138
4.4 TAPE INPUT/OUTPUT OPERATIONS OF NC DATA	151
4.5 LOADING PART PROGRAMS INTO MEMORY	154
4.6 EDIT OF PART PROGRAM	156
4.7 SUPPLEMENT TO DATA INPUT/OUTPUT INTERFACE †	159
4.8 TAPE VERIFYING	161
5. MACHINE CONTROL STATION	164
5.1 SWITCHING UNITS ON THE CONTROL STATION	164
5.2 OPERATION PROCEDURE	174
6. OPERATION Procedure	191
6.1 INSPECTION BEFORE TURNING ON POWER	191
6.2 TURNING ON POWER	191
6.3 MANUAL OPERATION	191
6.4 PREPARATION FOR STORED LEADSCREW ERROR COMPENSATION AND STORED STROKE LIMIT †	192
6.5 PREPARATION FOR AUTOMATIC OPERATION	192
6.6 OPERATION IN TAPE AND MEMORY MODE	193
6.7 MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION	193
6.8 AUTOMATIC OPERATION IN MDI MODE	193
6.9 MDI OPERATION INTERRUPTING AUTOMATIC OPERATION	193
6.10 PREPARATION FOR TURNING OFF POWER	194
6.11 TURNING OFF POWER	194
7. MAINTENANCE	196
7.1 ROUTINE INSPECTION SCHEDULE	196
7.2 BATTERY REPLACEMENT	196
7.3 POWER SUPPLY	197
7.4 THERMAL OVERLOAD RELAY OF SERVO UNIT	198
7.5 MOLDED-CASE CIRCUIT BREAKERS (MCCB)	199
7.6 TROUBLE CAUSES AND REMEDIES	199
APPENDIX-1 LIST OF SETTING NUMBERS	202
APPENDIX-2 LIST OF PARAMETER NUMBERS	213
APPENDIX-3 STORED LEADSCREW ERROR COMPENSATION LIST OF STANDARD INPUT/OUTPUT SIGNALS	239
APPENDIX-4	243
APPENDIX-5 LIST OF ALARM CODES	254
APPENDIX-6 LIST OF ADDRESS CHARACTERS	282

INDEX

Subject	Chapter	Par.	Page
A ABSOLUTE/INCREMENTAL PROGRAMMING (G90,G91)	2	2.9.30	85
ADDING PART PROGRAM (INSRT KEY)	4	4.6.4	158
ADDRESS AND FUNCTION CHARACTERS	2	2.1.2	3
ADDRESS KEYS	4	4.1.4	135
ADDRESS SEARCH	4	4.3.10	150
ALARM CODE DISPLAY	4	4.3.11	150
ALARM CODES AND REMEDIES	7	7.6.2	199
ALARM MESSAGE DISPLAY BY #8000 COMMAND	2	2.10.1	93
ALARM NUMBER OF USER MACROS	2	2.11.11	115
ARGUMENT DESIGNATION	2	2.11.3	96
AUTO MODE HANDLE OFFSET †	2	2.9.35	92
AUTOMATIC ACCELERATION AND DECELERATION	2	2.4.5	11
AUTOMATIC COORDINATE SYSTEM SETTING †	5	5.2.2	176
AUTOMATIC CORNER OVERRIDE	2	2.13	125
AUTOMATIC OPERATION IN MDI MODE	6	6.8	193
AUTOMATIC RETURN TO REFERENCE POINT (G28) †	2	2.9.15	33
AXIS INTERLOCK INPUT	5	5.1.29	172
B BATTERY	7	7.1.3	196
BATTERY REPLACEMENT	7	7.2	196
BEFORE MAINTENANCE CALL	7	7.6.5	201
BUFFER REGISTER	2	2.1.5	5
C CABLE CONNECTOR SPECIFICATIONS	4	4.7.4	160
CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99) †	2	2.9.28	68
CAUTIONS IN PROGRAMMING	3	3.2.3	133
CIRCLE CUTTING (G12, G13) †	2	2.9.9	26
CIRCUIT PROTECTOR TRIP (ALARM NO. 331 TO 334)	7	7.4.1	198
CIRCULAR INTERPOLATION (G02, G03)	2	2.9.4	22
CIRCULAR PATH MODE ON/OFF ON TOOL RADIUS COMPENSATION C (M97,M96) †	2	2.8.6	17
CIRCULAR PROJECTION COMPENSATION	2	2.14.3	129
COMMAND DATA DISPLAY	4	4.3.2	140
CONDITIONS FOR ADDING SOLID TAP FUNCTIONS	2	2.12.1	120
CONSIDERATIONS AND REMARKS FOR USER MACROS	2	2.11.10	113
CONSTANT DISPLAY	4	4.3.1	138
CONTROL COMMANDS	2	2.11.7	109
CONTROL PANEL	7	7.1.1	196
COORDINATE WORD	2	2.3	6
COORDINATE WORD	2	2.3.1	6
CRT CHARACTER DISPLAY	4	4.1.2	135
CURRENT VALUE NUMERICAL SETTING FUNCTION †	5	5.2.9	187
CURSOR KEYS	4	4.1.8	136
CYCLE START PUSHBUTTON AND LAMP	5	5.1.2	165
D DATA KEYS	4	4.1.5	135
DECIMAL POINT PROGRAMMING	2	2.1.3	5
DELETING PART PROGRAM BLOCK (ERASE KEY)	4	4.6.2	157
DESCRIPTION ON PROGRAMMING	2	2.12.2	120
DISPLAY AND WRITE OF LOCAL VARIABLES AND COMMON VARIABLES	2	2.11.9	112
DISPLAY AND WRITING OPERATION	4	4.3	138
DISPLAY IN THE  (SETTING) FUNCTION	4	4.3.6	146
DISPLAY LOCK/MACHINE LOCK SWITCH	5	5.1.21	170
DISPLAY OF REGISTERED PROGRAM NUMBER	4	4.5.4	156
DISPLAYING AND RESETTING CURRENT POSITION	4	4.3.4	143
DISPLAYING AND WRITING PARAMETERS	4	4.3.7	148
DISPLAYING AND WRITING TOOL OFFSET AND WORK OFFSET	4	4.3.5	144
DISPLAYING ON/OFF INPUT/OUTPUT SIGNALS	4	4.3.12	151
DRY RUN SWITCH	5	5.1.20	169
DWELL (G04)	2	2.9.6	25

INDEX (Cent'd)

Subject	Chapter	Par.	Page
M M CODES FOR INTERNAL PROCESSING (M90 TO M199)	2	2.8.2	14
M CODES FOR STOP (M00, M01, M02, M30)	2	2.8.1	14
M-FUNCTION LOCK SWITCH (AUXILIARY FUNCTION LOCK)	5	5.1.23	170
MACHINE CONTROL STATION	5		164
MACHINING INTERRUPTION POINT RETURN FUNCTION †	5	5.2.8	187
MAINTENANCE	7		196
MAKING ADDITION TO A PART PROGRAM	4	4.5.2	155
MANUAL ABSOLUTE SWITCH	5	5.1.24	171
MANUAL CENTERING FUNCTION †	5	5.2.11	188
MANUAL OPERATION	6	6.3	191
MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION	6	6.7	193
MANUAL PULSE MULTIPLY SELECT SWITCH †	5	5.1.7	165
MANUAL REFERENCE POINT RETURN SWITCH †	5	5.1.15	169
MANUAL SKIP FUNCTION †	5	5.2.6	184
MAXIMUM PROGRAMMABLE DIMENSIONS	2	2.3.6	8
MDI OPERATION INTERRUPTING AUTOMATIC OPERATION	6	6.9	193
MEMDATA (MEMORY DATA) KEYS	4	4.1.11	137
MESSAGE DISPLAY BY CONTROL-OUT AND CONTROL-IN	2	2.10.2	93
MIRROR IMAGE AXIS SELECTOR SWITCH	5	5.1.25	171
MIRROR IMAGE ON/OFF (M95, M94)	2	2.8.5	15
MISCELLANEOUS FUNCTIONS (M-FUNCTION)	2	2.8	14
MODE SELECT SWITCH	5	5.1.1	164
MODIFYING PART PROGRAM BLOCK (ALTER KEY)	4	4.6.3	157
MOLDED-CASE CIRCUIT BREAKERS (MCCB)	7	7.5	199
MULTI-ACTIVE REGISTERS ON/OFF (M93, M92) †	2	2.8.4	15
MULTI-ACTIVE REGISTERS †	2	2.1.6	5
N NC OPERATORS PANEL WITH 9" CRT CHARACTER DISPLAY	4		134
NC TAPE CHECK	3	3.3.3	133
NC TAPE PUNCH	3	3.3.2	133
NEXT KEY	4	4.1.6	136
O "OHT"LED(RED)IT	7	7.3.4	198
ON-LINE SELF-DIAGNOSTICS	7	7.6.1	199
OPERATION COMMANDS	2	2.11.6	108
OPERATION IN TAPE AND MEMORY MODE	6	6.6	193
OPERATION PROCEDURE	5	5.2	174
OPERATION PROCEDURE	6		191
OPERATION TIME DISPLAY	4	4.3.9	149
OPERATIONS USING DATA INPUT/OUTPUT INTERFACE	4	4.7.5	161
OPTIONAL BLOCK SKIP (/1 - /9) †	2	2.2.3	6
OPTIONAL BLOCK SKIP SWITCH	5	5.1.19	169
OPTIONAL STOP SWITCH	5	5.1.18	169
ORG (ORIGIN) KEY	4	4.1.9	136
OTHER M CODES	2	2.8.8	19
OUTLINE OF TOOL COMPENSATION	2	2.7.1	13
OUTPUTTING PART PROGRAM TO PAPER TAPE	4	4.4.5	152
OUTPUTTING SETTING DATA AND PARAMETER DATA TO PAPER TAPE	4	4.4.4	152
OUTPUTTING TOOL OFFSETS TO PAPER TAPE	4	4.4.3	152
OVERLOAD (ALARM NO, 351 TO 354)	7	7.4.2	198
OVERVIEW	2	2.11.1	94
OVERVIEW	2	2.14.1	127
OVERVIEW OF USER MACRO BODY	2	2.11.4	97
P PAGE KEYS	4	4.1.7	136
PAPER TAPE SELECT	3	3.3.1	133
PART PROGRAM TAPE CODING	3		130
PART PROGRAM TAPE HANDLING	3	3.4	133
PART PROGRAM TAPE PUNCHING	3	3.3	133

INDEX (Cent'd)

Subject	Chapter	Par.	Page
P "+5 V" LED (RED) LIGHT	7	7.3.2	197
"+24 V" LED (RED) LIGHT	7	7.3.3	197
PLANE DESIGNATION (G17, G18, G19)	2	2.9.10	28
POSITIONING (G00, G06)	2	2.9.2	21
POWER ON/OFF OPERATION	4	4.2	137
POWER ON/OFF PUSHBUTTONS	4	4.1.1	134
POWER SUPPLY	7	7.3	197
PREFACE	1		1
PREPARATION FOR AUTOMATIC OPERATION	6	6.5	192
PREPARATION FOR STORED LEADSCREW ERROR COMPENSATION AND STORED STROKE LIMIT	6	6.4	192
PREPARATION FOR TURNING OFF POWER	6	6.10	194
PREPARATORY FUNCTION (G-FUNCTION)	2	2.9	19
PROCESS SHEET	3	3.2.1	132
PROGRAM COPY † (G25)	2	2.9.13	31
PROGRAM INTERRUPTION ON/OFF (M91, M90) †	2	2.8.3 ²	15
PROGRAM NUMBER	2	2.2.1	5
PROGRAM NUMBER AND SEQUENCE NUMBER	2	2.2	5
PROGRAM RESTART	5	5.2.4	178
PROGRAMMING	2		1
PROGRAMMING	3	3.2	132
PROGRAMMING OF ABSOLUTE ZERO POINT (G92)	2	2.3	86
PUSHBUTTONS, LAMPS AND KEYS	4	4.1	134
R RAPID TRAVERSE RATE OVERRIDE	2	2.4.1	9
REFERENCE POINT CHECK (G27) † SWITCH	5	5.1.10	168
REFERENCE POINT LAMPS †	5	5.1.16	169
REGISTRATION OF USER MACROS	2	2.11.8	112
REMOTE POWER ON/OFF PUSHBUTTONS	4	4.2.3	138
RESET KEY	4	4.1.12	137
RETURN TO 2ND, 3RD AND 4TH REFERENCE POINT (G30) †	2	2.9.16	34
ROTATION OF COORDINATES (G68, G69) †	2	2.9.17	35
ROUTINE INSPECTION SCHEDULE	7	7.1	196
S 2-DIGIT PROGRAMMING	2	2.5.1	11
S 5-DIGIT PROGRAMMING	2	2.5.2	12
SCALING FUNCTION (G50, G51) †	2	2.9.22	57
2ND MISCELLANEOUS FUNCTION (B-FUNCTION) †	2	2.8.9	19
SEQUENCE NUMBER	2	2.2.2	6
SERVO ALARM (ALARM NO. 391 TO 394)	7	7.4.3	198
SETTING AND PARAMETER TAPE VERIFYING	7	7.1.2	196
SETTING OF BAUD RATE AND OTHERS OF SERIAL INTERFACE	4	4.7.3	160
SETTING OF DATA INPUT/OUTPUT TO BE USED	4	4.7.2	159
SETTING OF LOCAL COORDINATE SYSTEM (G52) †	2	2.9.34	9
SIMULTANEOUSLY CONTROLLABLE AXES OF FOUR-AXIS CONTROL †	2	2.3.3	7
SIMULTANEOUSLY CONTROLLABLE AXES OF THREE-AXIS CONTROL	2	2.3.2	7
SINGLE BLOCK SWITCH	5	5.1.17	169
SKIP FUNCTION (G31) †	2	2.9.18	36
SOLID TAP FUNCTION †	2	2.12	119
SOLID TAP 1/0 AND ITS RELATION WITH SPINDLE CONTROL 1/0	2	2.12.4	122
SOLID TAP RELATED FUNCTION	2	2.12.3	122
SOLID TAP RELATED PARAMETER	2	2.12.5	123
"SOURCE" LED [GREEN] UNLIT	7	7.3.1	197

INDEX (Cont'd)

Subject	Chapter	Par.	Page
S SPINDLE INDEXING FUNCTION †	5	5.2.12	188
SPINDLE SPEED OVERRIDE SWITCH †	5	5.1.14	168
SPINDLE-SPEED FUNCTION (S-FUNCTION)		2.5	11
SPlicing NC TAPE		3.4.1	133
START LOCK SWITCH †	5	5.1.27	172
STORed LEADSCREW ERROR COMPENSATION	APPENDIX-3		239
STORed PART PROGRAM DISPLAY	4	4.6.1	156
STORed STROKE LIMIT (G22, G23) †	2	2.9.12	29
SUBROUTINE PROGRAM (M98, M99)	2	2.8.7	18
SUMMARY OF EDITING OPERATION	4	4.8.4	163
supplementarY UNITS ON THE INPUT/OUTPUT Interface T	4	4.7	159
SWITCHING CONTROL STATION	5	5.1	164
T T 2-DIGIT PROGRAMMING	2	2.6.1	12
T 4-DIGIT PROGRAMMING	2	2.6.2	12
TAPE CODE	3	3.1	130
TAPE CODE	3	3.1.1	130
TAPE INPUT/OUTPUT OPERATIONS OF NC DATA	4	4.4	151
TAPE VERIFYING	4	4.8	161
THERMAL OVERLOAD RELAY OF SERVO UNIT	7	7.4	198
3RD TO 5TH STORed STROKE †	5	5.1.31	173
TOOL COMPENSATION	2	2.7	13
TOOL FUNCTION (T-FUNCTION)	2	2.6	12
TOOL LENGTH COMPENSATION (G43, G44, G49) †	2	2.9.20	5°
TOOL LENGTH MEASUREMENT PUSHBUTTON AND LAMP †	5	5.2.3	176
TOOL LIFE CONTROL (G122, G123)	2	2.9.32	86
TOOL OFFSET MEMORY	2	2.7.2	13
TOOL OFFSET VALUE DESIGNATION (G10)	2	2.9.8	26
TOOL OFFSET VALUE TAPE VERIFYING	4	4.8.2	161
TOOL POSITION OFFSET (G45 TO G48)	2	2.9.21	51
TOOL RADIUS COMPENSATION C (G40, G41, G42) †	2	2.9.19	37
TRAVERSE AND FEED FUNCTIONS	2	2.4	9
TROUBLE CAUSES AND REMEDIES	7	7.6	199
TURNING OFF POWER	4	4.2.2	138
TURNING OFF POWER	6	6.11	194
TURNING ON POWER	4	4.2.1	137
TURNING ON POWER	6	6.2	191
TYPEs AND FUNCTIONS OF INTERFACE-	4	4.7.1	159
u UNIDIRECTIONAL APPROACH (G60) †	2	2.9.26	66
UPGRADING THE CANNED CYCLE (G73, G74, G76, G77, G80 TO G89, G98, G99, G181, G182, G185, G186, G187, G189) †	2	2.9.29	79
USER MACRO (G65, G66, G67)	2	2.11	94
USER MACRO CALL COMMANDS	2	2.11.2	94
USER MESSAGE DISPLAY †	2	2.10	93
V VARIABLES	2	2.11.5	98
VERIFYING PART PROGRAM TAPE	4	4.8.3	161
W WORK COORDINATE SYSTEM SETTING A (G52 TO G59) †	2	2.9.23	58
WORK COORDINATE SYSTEM SETTING B (G52 TO G59) †	2	2.9.24	60
WORK COORDINATE SYSTEM SETTING C (G52 TO G59) †	2	2.9.25	63
WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI	4	4.3.3	142
z Z-MIS COMMAND NEGLECT SWITCH	5	5.1.22A	170

1. PREFACE

When reading this manual keep in mind that the information contained herein does not cover every possible contingency which might be met during the operation. Any operation not described in this manual should not be attempted with the control.

The functions and performance as NC machine are determined by a combination of machine and the NC control. For operation of your NC machine, the machine tool builder's manual shall take priority over this manual.

The illustration of machine control station should be used for your reference in understanding the function. For detailed array of operator's devices and names, refer to machine tool builder's manual.

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

Absolute Zero Point: 

Reference Zero Point
(Return to reference zero by manual and automatic return) : 

Dimensions : in mm

2. PROGRAMMING

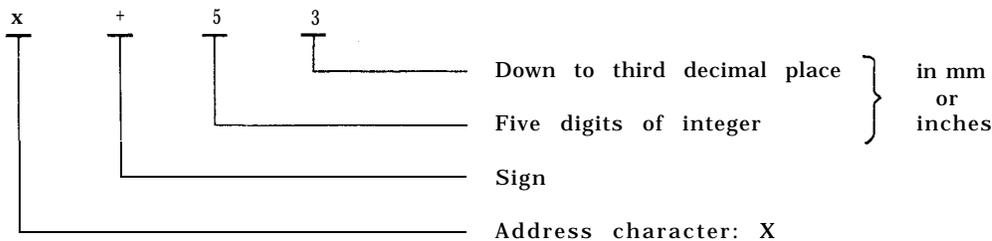
2.1 INPUT FORMAT

2.1.1 INPUT FORMAT

A variable block format conforming to JIS[#]B 6313 is used for YASNAC J50M.

Table Z. 1 shows the input format. Numerals following the address characters in Table 2.1 indicate the programmable number of digits.

EXAMPLE



Note:

A decimal point should be omitted in actual programming, when you make a program including decimal points, refer to 2.1.3, "DECIMAL POINT PROGRAMMING" on page 5.

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 (end of block) code in a program example is represented by a semicolon (;). In actual programming, CR (EIA code) or LF/NL (ISO code) should be used instead of the semicolon (;).

Japanese Industrial Standard

. Metric input format

O N4 G3 a+53 F5 S2 T2 M3 D(H)2 B3;

. Inch input format

0 N4 G3 a+44 F31 S2 T2 M3 D(H)2 B3;

Notes:

. "a" represents x, y, Z, I, J or K.

. P, Q, R and L are omitted in the above format because they are used for various meanings.

2.1.1 INPUT FORMAT (Cent'd)

Table 2.1 Input Format

No.	Address		Metric Output		Inch Input		B: Basic O: Optional
			Metric Input	Inch Input	Metric Input	Inch Input	
1	Program No.		04		04		B
2	Sequence No.		N4		N4		B
3	G function		G3		G3		B
4	Cordinate Word	Linear axis	a + 53	a + 44	a + 53	a + 44	B
		Rotary axis	b + 53	b + 53	b + 53	b + 53	o
5	Feed /rein		F50	F31	F50	F41	B
6	Feed/rein 1/10		F51	F32	F51	F42	B
7	S-function,		S2		S2		B
			S5		S5		o
8	T- function		T2		T2		B
			T4		T4		o
9	M-f unction		M3		M3		B
10	Tool Offset No.		H20r D2		H2 or D2		B
11	B-f unction		B3		B3		o
12	Dwell		P53		P53		B
13	Program No. designation		P4		P4		B
14	Sequence No. designation		P4		P4		B
15	No. of repetitions		L8		L8		B

2.1.2 ADDRESS AND FUNCTION CHARACTERS

Address characters and their meanings are shown in Table 2.2.

Table 2.2 Address Characters

Address Characters	Meanings	B: Basic O: Optional
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	B, O
E	User macro character	o
F	Feedrate	B
G	Preparatory function	B, O
H	Tool length offset number	B
I	X-coordinate of arc center Radius for circle cutting	B o
J	Y-coordinate of arc center Cutting depth for circle cutting	B, O
K	Z-coordinate arc center	B
L	Number of repetitions	B, O
M	Miscellaneous functions	B
N	Sequence number	B
O	Program number	B
P	Dwell time, Program No. and sequence No. designation in subprogram	B o
Q	Depth of cut, shift of canned cycles	o
R	Point R for canned cycles Radius designation of a circular arc	O, B
s	Spindle-speed 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

2.1.2 ADDRESS AND FUNCTION CHARACTERS (Cent'd)

Table 2.3 Function Characters

EIA Code	ISO Code	Meanings	Remarks
Blank	Nul	Error in significant data area in EIA Disregarded in ISO	
BS	BS	Disregarded	
Tab	HT	Disregarded	
CR	LF /NL	End of Block (EOB)	
/	CR	Disregarded	
SP	SP	Space	
ER	%	Rewind stop	
UC	/	Upper shift	
LC	/	Lower shift	
2-4-5 bits	(Control out (Comment start)	EIA : Special code
2-4-7 bits)	Control in (Comment end)	
+	+	Disregarded, User macro operator	
	-	Minus sign, User macro operator	
0 to 9	0 to 9	Numerals	
a to z	A to Z	Address characters, User macro operator	
/	/	Optional block skip	
Del	DEL	Disregarded (Including All Mark)	
.		Decimal point	
Parameter setting	#	Sharp (Variable)	EIA: Special code
*	*	Astrisk (Multiplication operator)	
=	=	Equal mark	
[[Left bracket	
]]	Right bracket	
o		User macro operator	
\$	\$	User macro operator	
@	@	User macro operator	
?	?	User macro operator	

Notes:

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

2.1.3 DECIMAL POINT PROGRAMMING

Numerals containing a decimal point may be used as the dimensional data of addresses related to coordinates (distance), time and speed.

Decimal points can be used in the following address words.

Coordinate words: X, Y, Z, I, J, K, A, B, C, U, V, W, Q, R

Time word: P

Feed rate word: F

EXAMPLE

	[mm]		[inch]
X15. ———	X15.000 mm	or	x15.0000 inches
Y20.5 —	Y20. 500 mm	or	Y20. 5000 inches
(G94)F25.6 - F25. 0	mm/min	or	F25.6 inches/rein
	(for F4.0)		(for F3.1)
G04P1. —	Dwell 1.000 sec		

Normally, when data without a decimal point is input, the control regards "1" as 0.001 mm (or 0.0001 inches, or 0.001 deg.).

2.1.4 LABEL SKIP FUNCTION

In the cases named below, the label skip function becomes effective, and LSK is displayed on the CRT.

- . When the power supply is turned on.
- . When the RESET key is pushed.

While the label skip function is effective, all data on the punched tape up to the first EOB code are neglected. When LSK is displayed on the CRT in the MEM (memory) or EDIT (editing) mode, it indicates the presence of a pointer at the leading end of the part program.

2.1.5 BUFFER REGISTER

During normal operation, one block of data is read in advance and compensation computing is made for the follow-on operation.

In the tool radius compensation C⁺ mode, two blocks of data or up to 4 blocks of data are read in advance and compensation computing required for the next operation is executed. One block can contain up to 128 characters including EOB.

2.1.6 MULTI-ACTIVE REGISTERS

For the portion of part programs sandwiched in between M93 and M92, up to 4 blocks of data are read in advance.

M code	Meaning
M 92	Multi-active register off
M93	Multi-active register on

Note: When power is applied or the control is reset, the control is in the state of M code marked with .

Inter-block stoppage can be eliminated when the program is so made that the automatic operation time of advance reading of 4 blocks is longer than processing time of advance reading of next 4 blocks of data.

Note :

Advance reading is not made for every 4 blocks but is always ready to be made up to 4 blocks in M93 mode.

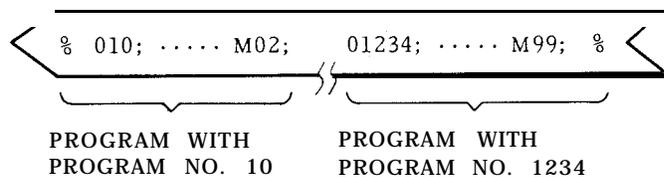
2.2 PROGRAM NUMBER AND SEQUENCE NUMBER

2.2.1 PROGRAM NUMBER

Program numbers may be prefixed to programs for the purpose of program identification.

Up to 4 digits may be written after an address character "0" as program numbers. up to 99 program numbers can be registered in the control, and up to 199 or 999 can be registered employing an option.

One program begins with a program number, and ends with M02, M30 or M99. M02 and M30 are placed at the ends of main programs, and M99 is placed at the ends of subprograms.



ER (or % at ISO code) is punched on the tape at the top and end of the program.

Notes :

The blocks for optional block skip such as /M02;, /M30;, /M99; are not regarded as ends of programs.

- . To make the reading of M02, M30, and M99 ineffective as a program end, and to make the succeeding ER (EIA) or % (ISO) as a sign of program ends is possible with a parameter change. (#6021 D0)

2.2.2 SEQUENCE NUMBER

Integers consisting of up to 4 digits may be written following an address character N as sequence numbers.

Sequence numbers are reference numbers for blocks, and do not have any influence on the meaning and sequence of machining processes. Therefore, they may be sequential, non-sequential, and duplicated numbers, and also not using any sequence number is also possible. Generally, sequential numbers are convenient as sequence numbers.

When searching for sequence numbers, be sure to search or specify program numbers beforehand.

Notes :

- . When 5 or more digits are written as a sequence number, only the digits up to the 4th from the trailing end are effective.
- . When two or more blocks have the same sequence number, only one is retrieved and read, and no more searching is performed.

Blocks without sequence numbers can also be searched for with respect to the address data contained in the blocks.

2.2.3 OPTIONAL BLOCK SKIP (/1 - /9) †

Those blocks in which “ /n” (n = 1 - 9) is included are neglected between In and the end of that block, when the external optional block skip switch for that number “n” is on.

EXAMPLE

```
/2 N1234 G01 X100 /3 Y200;
```

When the switch for /2 is on , the entire block is neglected, and when the switch for /3 is on , this block is read as if

```
N 1234 G01 X100; .
```

With “ 1” , “1” may be omitted.

Notes :

The optional block skipping process is executed while the blocks are read into the buffer resistor. If the blocks have been read , subsequent switching on is ineffective to skip the blocks.

While reading or punching out programs, this function is ineffective.

The optional block skip /2 - /9 is an option function.

2.3 COORDINATE WORD

Generally, commands for movements in axis directions and commands for setting coordinate systems are called coordinate words, and coordinate words consist of address characters for desired axes and numerals representing dimensions of directions.

2.3.1 COORDINATE WORD

Table 2.4 Coordinate Words

Address		Description
Main axes	X, Y, z	Position or distance in X, Y or Z coordinate direction.
4th axis †	A, B, C or u, v, W	These coordinate words are treated as commands in the directions of the 4th axis. A, B and C are used for rotary motion, and U, V and W are used for parallel motion
Circular interpolation auxiliary data	Q	Circular arc increment in circle cutting (G12, G13)
	R	Generally, radius values of circles.
	I, J, K	Generally, distances from start point to arc center (in X, Y and Z components) .

2.3.2 SIMULTANEOUSLY CONTROLLABLE AXES OF THREE-AXIS CONTROL

Table 2. 5 shows simultaneously controllable axes.

Table 2.5 Simultaneously Controllable Axes of Three-axis Control

	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 (see Note.)
Circle cutting+ G12, G13	Two axes: X and Y
Helical interpolation† G02, G03	Circle in XY-plane and linear feed in Z-axis direction . Refer to 2. 9.5 HELICAL INTERPOLATION .
Manual control	X, Y and Z

Note:

Circular arc plane is determined according to the currently effective G codes for plane designation. (G17 to G19)

For details, refer to 2.9.4, " CIRCULAR INTERPOLATION (G02,G03)" on page 22.

2.3.3 SIMULTANEOUSLY CONTROLLABLE AXES OF FOUR-AXIS CONTROL†

Table 2. 6 shows simultaneously controllable axes.

Table 2.6 Simultaneously Controllable Axes of Four-axis Control

	Simultaneously controllable axes
Positioning G00	X, Y, Z, and $\alpha^{(1)}$ axes
Linear interpolation G01	X, Y, Z, and $\alpha^{(1)}$ axes
Circular interpolation G02, G03	Two axes, ^{(2),(3)} XY, Yz, ZX, $X\alpha^{(1)}$, $Y\alpha^{(1)}$, or $Z\alpha^{(1)}$
Circular cutting† G12, G13	Two axes: X and Y
Helical interpolation† G02, G03	Three axes : circle in XY - plane and linear feed in Z-axis direction. Refer to 2. 9.5 HELICAL INTERPOLATION on page 27.
Manual control	Four axes, X, Y, Z, or $\alpha^{(1)}$,

- (1) The α axis represents any one of axes A, B, C, U, V or W, selected as the 4th axis.
- (2) Circular arc plane is determined according to the currently effective G codes for plane designation (G17 to G19) . For details, refer to 2.9.4, " CIRCULAR INTERPOLATION (G02,G03)" on page 22.
- (3) For circular interpolation axis α , any one of linear axes U, V, and W should be designated.

2.3.4 4TH AXIS CONTROL†

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. Address. is specified by parameter #6023.

2.3.4.1 ROTARY AXIS (A, B OR C AXIS)

The rotary axis is defined as follows.

Table 2.7 Rotary Axes for 4th Axis Control Table

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 B-axis .

The unit of output increment and input increment for B-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 B- axis remains "deg. " unit. The control does not convert B-axis coordinate commands. However, feedrate command F is converted. (Refer to 2.9.3. " LINEAR INTERPOLATION")

2.3.4.2 LINEAR AXIS (U, V OR W AXIS)

The linear axes are defined as follows.

Table 2.8 Linear Axes

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 c-axis.

2.3.4.2 LINEAR AXIS (U, V OR W AXIS) (Cont'd)

The unit output increment and input increment for C-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 C-axis.

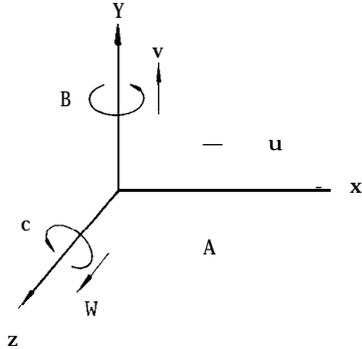


Fig. 2.1 4th Axis in Right-hand Coordinate System

2.3.5 LEAST INPUT INCREMENT AND LEAST OUTPUT INCREMENT

2.3.5.1 LEAST INPUT INCREMENT

The minimum input units that can be commanded by punched tape or MDI are shown in Table 2.9.

Table 2.9 Least Input Increment (#6006D5 = "O")

	Linear Axis	Rotary Axis [†]
Metric input	0.001 mm	0.001 deg
Inch input	0.0001 in	0.001 deg

Least input increment times ten can be set by setting parameter #6006D5 at "1."

Input Increment X 10 (#6006D5 = "1")

	Linear Axis	Rotary Axis [†]
Metric input	0.01 mm	0.01 deg
Inch input	0.001 in.	0.01 deg

Note : Selection of metric system or inch system is made by setting (#6001D0).

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.

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

Notes :

If NC programs set 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.

2.3.5.2 LEAST OUTPUT INCREMENT

Least output increment is the minimum unit of tool motion. Selection of metric or inch output is made by parameter (#6007D3) setting.

Table 2.10 Least Output Increment

	Linear axis	Rotary axis [†]
Metric output	0.001 mm	0.001 deg
Inch output	0.0001 in.	0.001 deg

2.3.6 MAXIMUM PROGRAMMABLE DIMENSIONS

Maximum programmable dimensions of move command are shown below.

Table 2.11 Maximum Programmable Dimensions

		Linear axis	Rotary axis [†]
Metric output	Metric input	±99999.999 mm	*99999.999 deg
	Inch input	±3937.0078 in.	±99999.999 deg
Inch output	Metric input	±99999.999 mm	±99999.999 deg
	Inch input	*9999.9999 in.	±99999.999 deg

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

In absolute programming , move amount of each axis 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.

The accumulative value must not exceed the maximum accumulative values shown below.

Table 2.12 Maximum Cumulative values

	Linear axis	Rotary axis [†]
Metric input	± 99999.999 mm	± 99999.999 deg
Inch input	± 9999.9999 in.	± 99999.999 deg

Listed input values do not depend on metric/inch output system.

2.4 TRAVERSE AND FEED FUNCTIONS

2.4.1 RAPID TRAVERSE RATE

2.4.1.1 RAPID TRAVERSE RATE

The rapid traverse motion is used for the motion for the Positioning (G00) and for the motion for the Manual Rapid Traverse (RAPID). The traverse rates differ among the axes since they are dependent on the machine specification and are determined by the machine tool builders. The rapid traverse rates determined by the machine are set by parameters in advance for individual axes. When the tool is moved in rapid traverse in two or three axial directions simultaneously, motions in these axial directions are independent of each other, and the end points are reached at different times among these motions. Therefore, motion paths are normally not straight.

For override rapid traverse rates, Fo, 25%, 50% and 100% of the basic rapid traverse rates, are available. Fo is a constant feed rate set by a parameter (#6231).

2.4.1.2 SETTING RANGE OF RAPID TRAVERSE RATE

For each axis, rapid traverse rates can be set at some suitable multiple of 0.001 mm/min (or deg/min).

The maximum programmable rapid traverse rate is 30,000 mm/min. However, respective machine tools have their own optimum rapid traverse rates. Refer to the manual provided by the machine tool builder.

2.4.2 FEEDRATE (F-FUNCTION)

With five digits following an address character F, tool feedrates per minute (mm/rein) are programmed.

The programmable range of feedrates is as follows.

Table 2.13 Programmable Range of Feedrate

		Format	Feedrate (Feed/rein) range
Metric output	Metric input	F50	F1.-F30000.0 mm/min
	Inch input	F31	FO.1-F1181.10 in./rein
Inch output	Metric input	F50	F1.-F76200. mm/min
	Inch input	F31	FO. 1-3000.00 in./rein

The maximum feedrate is subject to the performance of the servo system and the machine system. When the maximum feedrate set by the servo or machine system is below the maximum programmable feedrate given above, the former is set by a parameter (#6228), and whenever feedrates above the set maximum limit are commanded, the feedrate is clamped at the set maximum value.

F commands for linear and circular interpolations involving motions in simultaneously controlled two axial directions specify feedrates in the direction tangential to the motion path.

EXAMPLE G 91 (incremental)

G01 X40. Y30. F500

With this command,

$$F = 500 = \sqrt{300^2 + 400^2}$$

(mm/min)

X component

Y component

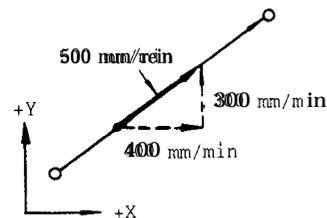


Fig. 2.2

G03 X... Y... I... F200

With this command,

$$F = 200 = \sqrt{fX^2 + fY^2}$$

(mm/min)

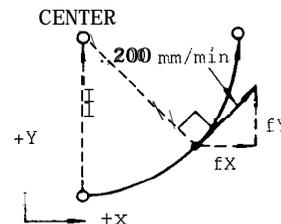


Fig. 2.3

2.4.2 FEEDRATE (F-FUNCTION) (Cent'd)

F commands for linear interpolations involving motions in simultaneously controlled three axial directions specify feedrates also in the direction tangential to the motion path.

EXAMPLE

With GO1 X... Y... Z... F400 ;

$$F = 400 = \sqrt{fX^2 + fY^2 + fZ^2}$$

(mm/min)

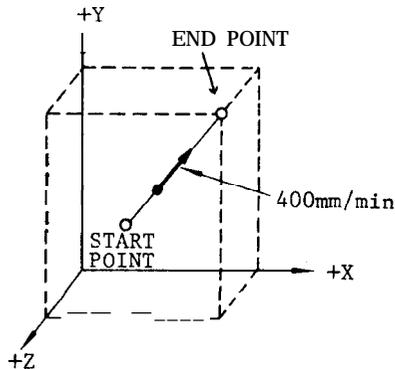


Fig. 2.4

F commands for linear interpolations involving motions in simultaneously controlled four axial directions specify feedrates also in the direction tangential to the motion path.

$$F \text{ (mm/min)} = \sqrt{fX^2 + fY^2 + fZ^2 + fA^2}$$

Notes :

- . If FO is programmed, it is regarded as a data error. (alarm code "030)
- . Do not program F commands with minus numerals, otherwise correct operation is not guaranteed.

EXAMPLE

F-250 ; wrong

2.4.3 FEEDRATE 1/10

The feedrate programmed by F commands can be converted to 1/10-th value with a parameter setting as follows.

- . When parameter #6020 D0 or D1 is set to "1," the feedrates range becomes as shown below.

Table 2.14 Programmable Range of 1/10 Feedrate

		Format	Feedrate (Feed/rein) range
Metric output	Metric input	F51	FO.1-F3000.0 mm/min
	Inch input	F32	F0.01-F1181.10 in./min
Inch output	Metric input	F51	FO.1-F76200.0 mm/min
	Inch input	F32	F0.01-F3000.00 in. /rein

- . When parameter #6020 DO or D1 is set to "0," the feedrate range returns to normal.

2.4.4 F 1-DIGIT PROGRAMMING †

- (1) Specification of a value 1 to 9 that follows F selects the corresponding preset feedrate.
- (2) Set the feedrate of each of F1 to F9 to the setting number shown in Table 2.15 (a).
- (3) By operating the manual pulse generator when F1-DIGIT switch is on, the feedrate of F1 - digit command currently specified may be increased or decreased. Set the increment or decrement value per pulse (F1-digit multiply) to the parameters listed in Table 2.15 (b).

As a result of this operation, the contents of the setting number of the F1-digit feedrate are changed.

(4) Upper Limit of Feedrate

Set the maximum feedrate of F1-digit designation to the following parameter. If a value greater than the usual maximum feedrate (the contents of #6228) is set, it is governed by the contents of #6228.

Table 2.15 (a) F Command and Setting No.

F command	Setting No. for F1-digit speed
F1	#6561
F2	#6562
F3	#6563
F4	#6564
F5	#6565
F6	#6566
F7	#6567
FE	#6568
F9	#6569

Setting "1" = 0.1 in. /rein or 0.01 in. /rein

Table 2.15 (b) F Command and Parameter No.

F command	Parameter No. for F1-digit multiply
F1	#6141
F2	#6142
F3	#6143
F4	#6144
F5	#6145
F6	#6146
F7	#6147
F8	#6148
F9	#6149

Setting "1" = 0.1 mm/min/pulse

Table 2.15 (c) Parameter No. for Maximum Feedrate

Parameter No.	Meaning
#6226	Max speed of F1 to F4
#6227	Max speed of F5 to F9

Notes :

- When this feature is installed, the specifying 1 to 9 mm /rein by the usual F function is not allowed. Specifying 10 mm /rein or more is allowed usually.
- If F0 is specified, error " 030" will be caused.
- When DRY RUN switch is on, the rate of dry run is assumed.
- For F1-digit specification, the feedrate override feature is invalid.
- The feedrate stored in memory is retained after the power is turned off.
- For the variable command of micro-program F 1-digit command is possible.

2.4.5 AUTOMATIC ACCELERATION AND DECELERATION

Acceleration and deceleration for rapid traverse and cutting feed are automatically performed.

2.4.5.1 ACCELERATION AND DECELERATION OF RAPID TRAVERSE AND MANUAL FEED

In the following operation , the pattern of automatic acceleration and deceleration is linear .

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

The 2-step linear acceleration/deceleration can be specified shown in Fig. 2.5.

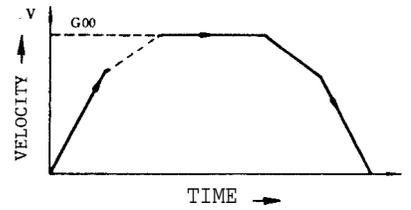


Fig. 2.5

Rapid traverse rate and acceleration deceleration constant of rapid traverse rate can be set by parameter. (#6280 to #6301)

2.4.5.2 ACCELERATION /DECELERATION OF FEEDRATE

Automatic acceleration and deceleration of feed motion (G01 - G03) are in the exponential mode.

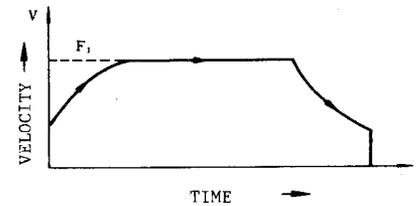


Fig. 2.6 Exponential acceleration deceleration

Feedrate time constants and feedrate bias are set by parameters. During tapping, another time constants and bias other than for usual feedrate can be set by parameters (#6406 -#6434) .

Note:

The automatic acceleration /deceleration parameters are set to the optimum values for the respective machines. Do not change the setting unless this is required for special purposes.

2.5 SPINDLE-SPEED FUNCTION (S-FUNCTION)

2.5.1 S 2-DIGIT PROGRAMMING

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

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

When a move command and an S code are issued in the same 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.

S codes are modal, remaining effective, when once commanded, until next S code is commanded. If the spindle is stopped by M05 (spindle stop) command, the S command in the control is kept.

2.5.1 S 2-DIGIT PROGRAMMING (Cent'd)

EXAMPLE

```

COO S11 M03 ;
      . . . S command
      Spindle CW
      X... Y... Z... ;
G01 Z... F... ;

G00 x... Y... Z... M05 ; Spindle stop
      M03 ;
      x... Y... Z... ;
G01 Z... F... ;
      S22 ;
      x... Y... F... ;
  
```

} S11: Effective
 } S11: Effective
 } S22: Effective

Note : The two-digit BCD output is sent 10 the machine when S two-digit command is issued.

2.5.2 S 5-DIGIT PROGRAMMING

With five digits written after an address character S(S) , spindle speeds (rein-l) are directly commanded.

The programmed speeds become effective upon the inputting of an S-command-comple tion-input-signal (SFIN) ,

When an S command is programmed in the same block with M03 (spindle forward run) or M04 (spindle reverse run) , the execution of the next block starts only after the spindle speed reaches the level specified by the S command, in most cases. However, for exact behavior of the machine tool under consideration, refer to the machine tool builder's manual.

The S commands are modal, and when it is programmed once, it remains effective until another command is programmed. Even when the spindle is stopped by M05, the S command remains effective. Therefore, when the spindle starts again with an M03 (or M04) , the spindle runs at the speed specified by the S command.

When the spindle speed is to be changed by a new S command after it is started with an M03 or M04, attention must be paid to the selected spindle speed range.

Notes :

- The lower limit of programmable S commands (SO and other S commands near O) is determined by the spindle motor of the machine tool. Refer to the machine tool builder's manual. Do not program minus values as S commands.

, When the control is equipped with the S 5-digit command function , spindle speed overriding is possible. That is, override speeds between 50 and 120% of the commanded spindle speed can be obtained at intervals of 10%.

EXAMPLE

S 1000 M03 ;

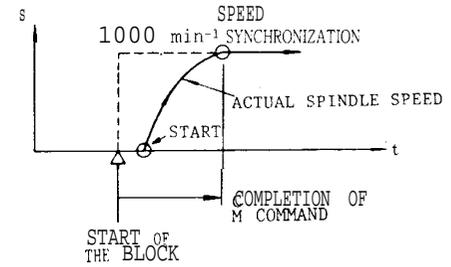
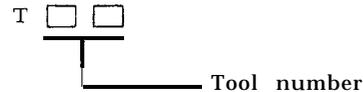


Fig. 2.7

2.6 TOOL FUNCTION (T-FUNCTION)

2.6.1 T 2-DIGIT PROGRAMMING

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.

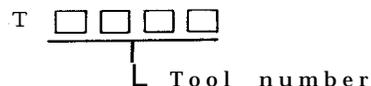
T codes are modal, and therefore, once they are given , they remain effective until another T command is given.

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.

2.6.2 T 4-DIGIT PROGRAMMING

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.

2.7 TOOL COMPENSATION

2.7.1 OUTLINE OF TOOL COMPENSATION

The tool compensation function is in the following three modes.

. Tool length compensation

This function is for compensating the differences in tool length, and is effective in the Z axis direction. Specified length compensation becomes effective from the block in which G43 or G44 is programmed together with an H code. It is cancelled with H00 or G49.

. Tool position offset (for simple compensation for tool radius)

This function is for compensating for errors in machined dimensions to be introduced by the radius of tools. It is effective in the X, Y, and Z (4th t) axis directions. It is effective only for the block in which G45-G48 is programmed together.

. Tool radius compensation C[±] (for compensating for tool radius effects with complicated machining contours)

This function is for compensating for the tool radius effect with any given machining contours. It is effective in X-Y , Y-Z, and Z-X planes. It becomes effective from the moment G41 , or G42 is commanded together with a D code, and is cancelled by G40.

Note : For details of these compensations functions, refer to 2.9. "PREPARATORY FUNCTION (G-FUNCTION)".

2.7.2 TOOL OFFSET MEMORY

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

The following number of offset values can be stored in the tool offset memory.

	Offset Value Storage	
Basic	1	99
t Optional		1199

The setting range of offset values is as follows.

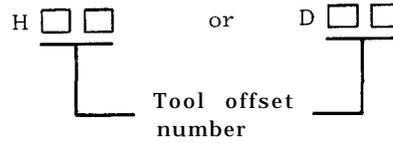
	Linear axis	Rotary axis [†]
Metric input	0 - ± 999.999 mm	0- ± 999.999 deg
Inch input	0 - ± 99. 9999 inch	0- ± 999.999 deg

Listed input values do not depend on metric/ input output system.

For the procedures of storing values into memory, refer to 4.3.5. "DISPLAYING AND WRITING OF TOOL OFFSET AND WORK OFFSET" on page 144.

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

Two or three† 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 g 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.

H- and D-codes must be used properly according to their functions.

Code	Function
H code	Tool length offset
D code	Tool position offset, Tool radius compensation

The tool offset numbers 01 through 99 can be used freely in combination with the both H and D codes.

- M90t : Program interrupt off
- M91†: Program interrupt on
- M92†: Multi-active register off
- M93†: Multi-active register on
- M94: Mirror image off
- M95: Mirror image on
- M96†: Tool radius compensation C : circular path mode
- M97†: Tool radius compensation C : intersection computing mode
- M98: Subroutine program call
- M99: Subroutine program end
- M100 to 199: Used for enhanced codes

2.8.3 PROGRAM INTERRUPTION ON/OFF (M91, M90) t

The following M codes are used for the program interruption function .

M code	Meaning
M90	Program interrupt function OFF
M91	Program interrupt function ON

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◀.

· M91 P. ;

During the time from this command to an M90 command, whenever a program interruption signal is received, the program under execution is interrupted (if the machine is in motion, it is stopped after deceleration) , and the a jump is made to the program the number of which is written after the P .

. M90 ;

With this command, the program interrupt function is cancelled.

2.8.4 MULTI-ACTIVE REGISTERS ON/OFF (M93, M92) t

M code	Meaning
M92	Multi-active register OFF
M93	Multi-active register ON

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◀.

M93:

During the time from this command to M92, the control assumes the 4 blocks-advance-reading mode. Namely, up to 4 blocks of data are read in advance for the following operation.

Inter-block stoppage can be eliminated when the program is so made that the operation time of advance reading of 4 blocks is longer than processing time of advance reading of next 4 blocks of data.

M92:

This command cancels 4 blocks-advance -reading mode.

Note : In tool radius compensation C mode, the blocks without move command can be contained(up to two blocks) . Under this condition, 6 blocks, including the two blocks, may be read in advance.

2.8.5 MIRROR IMAGE ON/OFF (M95, M94)

M code	Meaning
M94	Mirror image OFF
M95	Mirror image ON

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◀.

With these codes, mirror image operation can be started and stopped at any desired point in 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 image is to be effected is specified by setting #6000 Do to D3 (or mirror image axis designation switch). For this procedure, refer to 5.1.25, "MIRROR IMAGE AXIS SELECTOR SWITCH" on page 171.

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.

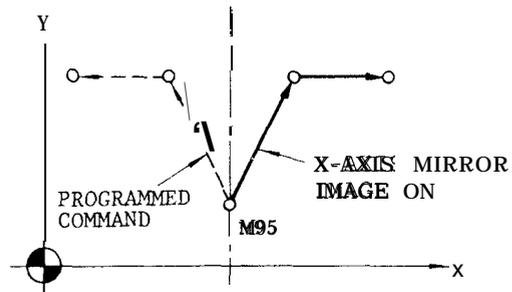


Fig. 2.8

2.8.5 MIRROR IMAGE ON/OFF (M95, M94) (Cont'd)

With both the absolute and increment move commands, the same mirror image effect will be obtained. The block including M95 command 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 G30 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 G30 is given, an error "058" occurs.

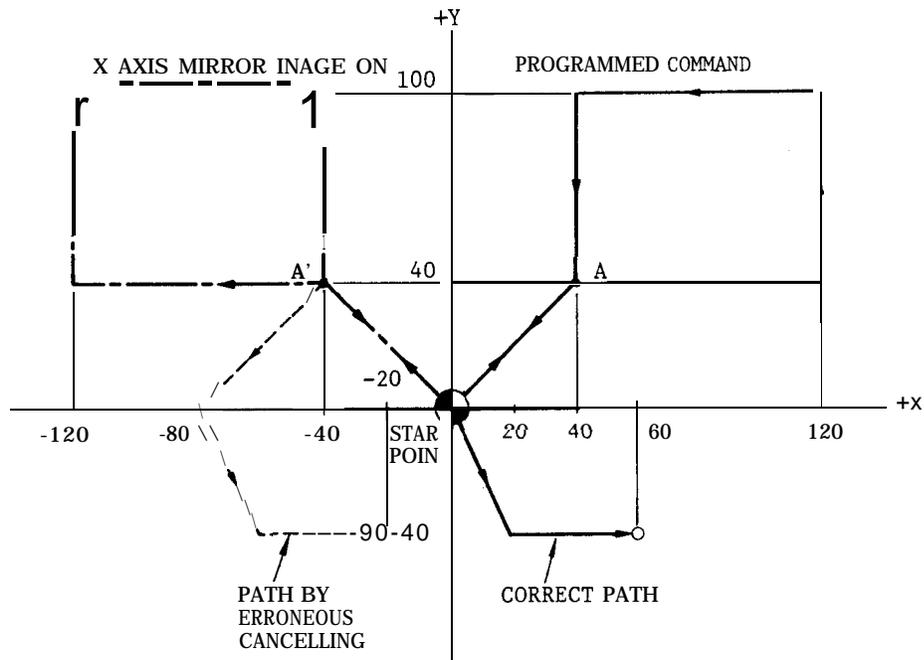
The mirror image effect is not effective on the offset movement resulting from the tool length offset function.

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,

- 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 canceling mirror image will be shifted by the difference between both positions.

EXAMPLE



```

NO1 G92 XO YO ;
NO2 M95 ; ..... Mirror image on.
NO3 G90 G01 X40. Y40. F300 ;
NO4 X120. ;
NO5 Y100. ;
NO6 X40. ;
NO7 Y40. ;
NO8 Xo Yo ;
NO9' M94 ; .... Correct mirror image off.
NO10 X20. Y-40. ;
NO11 x60. ;
    
```

← If "M94 ;" is programmed here, the tool moves on the dotted line.

Fig. 2.9

. Mirror image external input function

(a) Overview

In addition to the conventional mirror image function, the mirror image execution mode can also be set when power is turned on, or reset, by setting the corresponding parameter. When the mode is the mirror image execution mode (power on or reset,) the parameter can be set to select the command mirror image at the G28 intermediate point or not.

(b) How to use the function

(i) Upon power ON or upon reset

#6005, D2 0: M94 mode (mirror image off)
1: M95 mode (mirror image on)

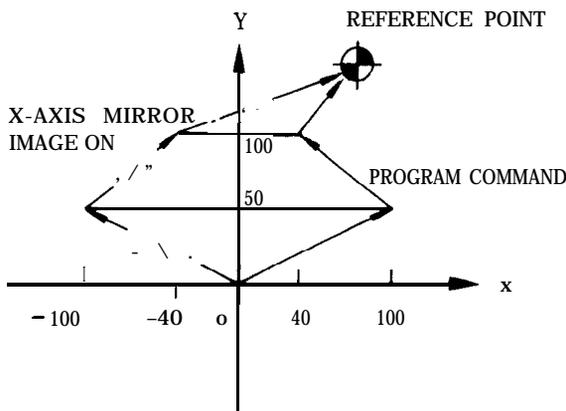
(ii) Upon power ON, when it is M95 mode (#6005, D2 is "1.")

#6005, D1 0: Commands mirror image at the G28 intermediate point
1: Does not command mirror image at the G28 intermediate point

Note : The specifications are the same as the conventional specifications, when #6005, D2 is "0." Therefore, turn off mirror image by M94, when commanding G28, or G29 under this mode. Error "058" occurs if not turned off.

(c) Program example

† Example of commanding mirror image on the G28 intermediate point
Program example (mirror image of X-axis only is on)



Note : When commanding axis designation under the mirror image mode (M95) by m code, stop the look-ahead by parameter (#61 16) of the set/reset M codes.

2.8.6 CIRCULAR PATH MODE ON/OFF ON
TOOL RADIUS COMPENSATION C (M97, M96) t

M code	Meaning
M 96	Tool radius compensation circular path ON.
M 97	Tool radius compensation circular path OFF. (Execution of intersection point)

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◀.

In the G41 or G42 tool 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.

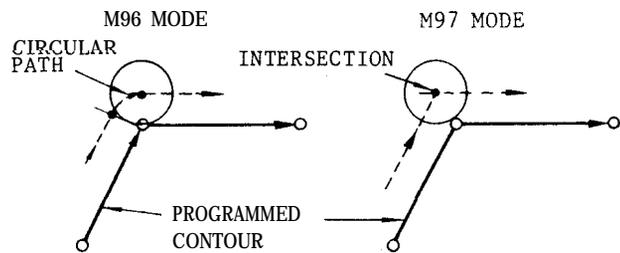


Fig. 2.10

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

M 96 and M97 are effective on the following move command blocks.

G01 X... Y... ;
(G01) X.. . Y.. . M96 ; } Effective from the corner of these 2 blocks

G01 X... Y... F... ;
M96 (or M97) ;
(G01) X... Y... ; } Effective from the corner of these 2 blocks

2.8.7 SUBROUTINE PROGRAM (M98, M99)

With this function, call of subroutine programs which have been numbered and stored in advance is made and executed as many times as desired.

The following M codes are used for this function.

M code	Meaning
M98	Call of subroutine program
M99	Subroutine program end

Call of subroutine program (M98)

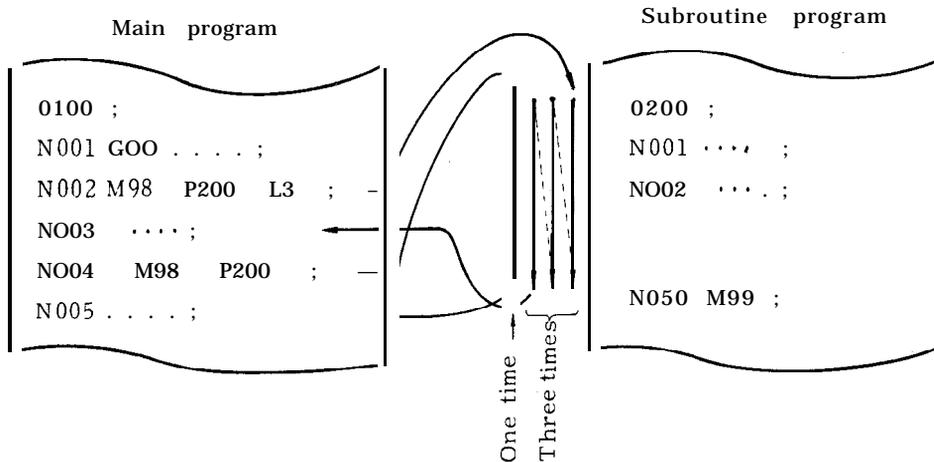
M98 P... L... ;

With this command, call of the subroutine program with the number specified after P is made and is executed number of times specified after L. When no L code is programmed, the subroutine is executed once.

Subroutine programs can be nested up to 4 times.

EXAMPLE

Call of subroutine program and execution of it are made in the sequence shown below.



Special use of M99

M99 P. . . ;

With this command, the main program does not return to the block following the M 98 block after executing the subroutine program, but returns to the block with a sequence No. specified by the P code.

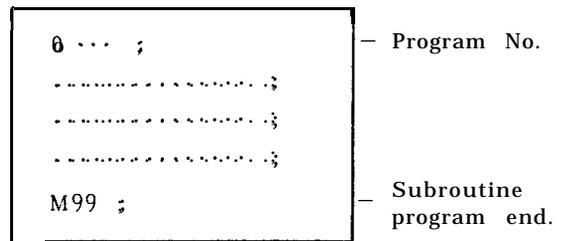
Notes :

If the program number specified by the P code is not found, this is regarded as an error "041. "

While a subroutine program is repeated L times, the number of remaining repetitions may be

Format of subroutine program (M99)

Subroutine programs are written in the following g format, and are stored in the part program memory in advance.



Automatic return command from subroutine program

M99 ;

At the end of subroutine programs, M99 is written in a block of its own. When M99 is commanded in the subroutine program which has been called by M98, the execution of the main program is automatically restarted at the block immediately following the M98 block.

displayed. For details, refer to 4.3 DISPLAY AND WRITING OPERATION.

- This function is usable when subroutine programs are stored in the part program memory. The main program can either be commanded from NC tape or the part program memory.

When the nesting of subroutine programs is attempted more than 4 times, an error state is caused.

Commanding M99; in main program will return the execution of the program to the head of the main program and control endless operation.

2.8.8 OTHER M CODES

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

Table 2.17 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 the same 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.

2.8.9 2ND MISCELLANEOUS FUNCTION (B-FUNCTION)

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 .

• With MDI operation on NC panel, "B" is used to specify address for B codes. Therefore, when the control has B-function, the 4th axis control cannot be added.

2.9 PREPARATORY FUNCTION (G-FUNCTION)

2.9.1 LIST OF G CODES AND GROUPS

An address character G and up to 3 digits following it specify the operation of the block.

Ordinary G codes are either non-modal G codes marked with * or modal G codes belonging to groups 01 through 15. Those G codes belonging to the division B are included in the basic specifications:

The G codes belonging to groups 01 through 15 are modal, remaining effective when once commanded until other G codes in the same group will be commanded.

The G codes in the * group are non-modal, and are effective only for the block in which they are commanded.

The G codes belonging to groups 01 through 15 may be programmed twice or more in the same block. However, when different G codes in the same group are programmed, the last appearing G code is effective.

When the RESET key is depressed during the execution of a tool compensation C (G41 , or G42) or a canned cycle, they become respectively G40 or G80 which cancels the programmed commands.

G43, G44, G49 and G45 through G48 belonging to the * group can be programmed together with the following G codes in the 01 group in the same block.

If * group other than those G codes and G codes of 08 group can be programmed in the same block as G code of 08 group alarm will be caused.

	Combination G code
G43, G44, G49	G00, G01, G60
G45 - G48	G00, G01, G02, G03, G60

The transition state of the following G codes can be changed by setting of parameters.

The G codes belonging to the following groups can be specified as to the state immediately after the application of supply power. (#6005D0 - D4)

Group	Initial State	Parameter
01	G00 or G01	#6005D6
03	G90 or G91	#6005D0
08	G43, G44 or G49	#6005D3, D4

G codes in the 01 group may selectively be changed to G00 or kept unchanged after resetting. (#6005D6)

2.9.1 LIST OF G CODES AND GROUPS (Cent'd)

During the execution of G 92, Display may selectively be made. (#6005D5)

G code in the 02 or 03 group at reset can be set by parameters.

Group	Timing	Parameter OFF	Parameter ON
01	Upon power ON	G17	G17
	Upon reset	Stores the G code commanded immediately before this command	G17
03	Upon power ON	PM6005 D0 0...G90 1...G91	Same as on the left
	Upon reset	Stores the G code commanded immediately before this command	PM6005 D0 0...G90 1...G91

Table 2.18 List of G Codes

G code	Group	Function	B: Basic O: Optional
G00		Positioning	B
G01		Linear interpolation	B
G02	01	Circular interpolation CW, Helical interpolation CW	B, O
G03		Circular interpolation CCW, Helical interpolation CCW	B, O
G04		Dwell	B
G06		Positioning in error detect off mode	B
G09		Exact stop	B
G10	*	Tool offset value and work coordinate, Shift-value modification	B, O
G12		Circle cutting CW	n
G13		Circle cutting CCW	o
G17		XY plane designation	B
G18	02	ZX plane designation	B
G19		YZ plane designation	B
G20	'6	Inch input designation	O
G21		Metric input designation	O
G22	04	Stored stroke limit ON	o
G23		Stored stroke limit OFF	n
G25	*	Program copy	o
G27		Reference point check	O
G28		Automatic return to reference point	o
G29	*	Return from reference point	o
G30		Return to 2nd, 3rd, 4th reference point	o
G31		Skip function	O
G40		Tool radius compensation cancel	O
G41	07	Tool radius compensation, left	O
G42		Tool radius compensation, right	O
G43	08	Tool length compensation, plus direction	B
G44		Tool length compensation, minus direction	B
G49		Tool length compensation, cancel	B
G45	*	Tool position offset, extension	B
G46		Tool position offset, retraction	B
G47		Tool position offset, double extension	B
G48		Tool position offset, double retraction	B
G50	15	Scaling OFF	O
G51		Scaling ON	O
G52	12	Return to base coordinate system	o
G53	*	Temporary shift to machine coordinate system	o
G54	12	Shift to work coordinate system 1	0
G55		Shift to work coordinate system 2	0
G56		Shift to work coordinate system 3	0
G57		Shift to work coordinate system 4	0
G58		Shift to work coordinate system 5	0
G59		Shift to work coordinate system 6	0

Table 2.18 List of G Codes (Cent'd)

G code	Group	Function	B, Basic O: Optional	
G60	01	Unidirectional approach	O	
G61	13	Exact stop mode	B	
G64		Exact stop mode cancel	B	
G65	*	Non-modal call of user macro	O	
G66	14	Modal call of user macro	O	
G67		Modal call of user macro cancel	O	
G68	18	Coordinate rotation ON	O	
G69		Coordinate rotation OFF	O	
G70	*	Bolt hole circle	O	
G71		Arc	O	
G72		Line-at-angle	O	
G73		Canned cycle 10	O	
G74		Canned cycle 11	O	
G76		Canned cycle 12	O	
G77		Canned cycle 13	O	
G80		09	Canned cycle cancel	O
G81		Canned cycle 1, Output for external motion	O	
G82		Canned cycle 2	O	
G83		Canned cycle 3	O	
G84		Canned cycle 4	O	
G85	Canned cycle 5	O		
G86	09	Canned cycle 6	O	
G87		Canned cycle 7	O	
G88		Canned cycle 8	O	
G89	Canned cycle 9	O		
G90	03	Absolute command designation	B	
G91		Incremental command designation	O	
G92	*	Programming of absolute zero point	B	
G93	05	Solid tap mode ON	O	
G94		Solid tap mode OFF	O	
G98	10	Return to initial point for canned cycles	O	
G99		Return to point R for canned cycles	O	
G106	*	Automatic corner override	O	
G122	17	Tool register start	O	
G123		Tool register end	O	
G124	*	Tool register cancel	O	
G198	.	Form offset mode ON	O	
G199	.	Form offset mode OFF	1 O	

Notes:

1. The G codes in the * group are non-modal, and are effective only for the block in which they are commanded. They cannot be programmed twice or more in a block. They must be programmed only once in a block of its own.
2. The codes marked with \surd are automatically selected at power on or reset.
3. The code marked \surd is automatically selected upon power ON.

2.9.2 POSITIONING (G00, G06)

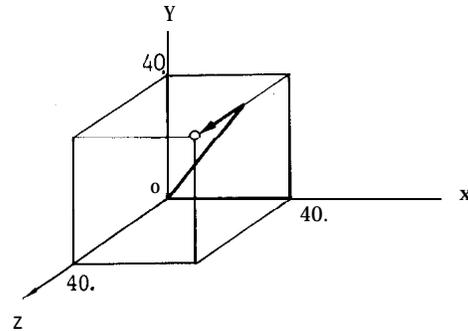
G00 X... Y... Z... (a t...);
(where a = A, B, C, U, V, or W)

With this command, the tool is sent to the specified position in rapid traverse motions along the 3 axes (5 axes) simultaneously. If any of the coordinate positions is not specified, the machine does not move along that coordinate axis

The rapid traverse rate for the respective axes are inherent to the machine tool. Refer to the machine tool builder's manual.

EXAMPLE

G00 X40. Y 40. Z40. ;



Rapid traverse rate

X axis: 8 m/rein
Y axis: 8 m/min
Z axis: 4 m/min

Fig. 2.11

G00 is a modal G code belonging to the 01 group.

Error detect OFF positioning (G06)

G06 X... Y... Z... (a t...);

With this command, the same positioning motions are initiated as with a G00 command, with the following exceptions.

After the completion of the positioning motion with G06 block, the program advances to the next block in the ERROR DETECT OFF mode (Note). Therefore, the tool path at the corner is rounded.

G06 is a non modal G code belonging to the * group, and therefore, it is effective only in which it is programmed.

Notes:

G00 commands position the tool in the ERROR DETECT ON mode, which means that the program advances to the next block only after the servo lag pulses are decreased below the permissible level, and this is detected by the control. With this command, therefore the corner of the workpiece is machined sharp.

2.9.2 POSITIONING (G00, G06) (Cent'd)

. With the ERROR DETECT OFF mode commanded by G06, the program advances to the next block immediately after the completion of pulse distribution.

2.9.3 LINEAR INTERPOLATION (G01)

G01 X.. .Y. .. Z... (a t..) F... ;

where $\alpha = A, B, C, U, V, \text{ or } W$

With this command, the tool is moved simultaneously in the three (four t) 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.

$$F = \sqrt{F_x^2 + F_y^2 + F_z^2 + F_\alpha^2}$$

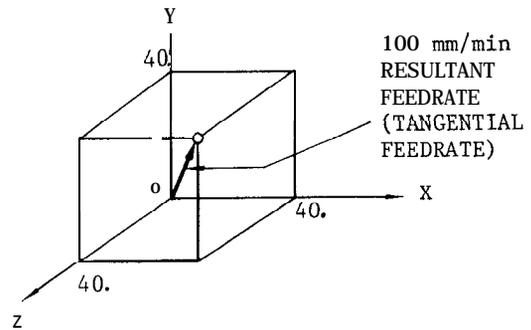
(where F_x, F_y, \dots are feedrate in the X, Y ... directions.)

The end point can be programmed either in absolute coordinates or in incremental values with G90 or G91 respectively. (Refer to 2.9.30, "ABSOLUTE /INCREMENTAL PROGRAMMING (G90, G91)").

If no F code is given in the block containing the G01 or in preceding blocks, the block constitutes an error " 030. "

EXAMPLE

G01 X40. Y40. Z40. F100 ;



Ftg. 2.12

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 indicated.

Table 2.19 Minimum F Command Unit

F-function			In minimum F command unit	
			Feedrate of basic three axes	Feedrate of rotary axes
Metric output	Metric input	F50	1 mm /rein	1 deg /rein
	Inch inp Ut	F31	0.1 in. /rein	2.54 deg /min
Inch output	Metric input	F50	1 mm/min	0. 3937 deg /rein
	Inch input	F 31	0, 1 in. /rein	1 deg/min

Note: Feedrate of linear 4th axis as the same as that of basic three axes.

2.9.4 CIRCULAR INTERPOLATION (G02, G03)

With the following commands, the tool is controlled along the specified circular pathes on the XV-, 2X-, or

YZ- plane, at a tangential speed specified by the F code.

XY-plane G17 { G02 } X... Y.. { R... I... J... } F.. . ;
 ZX-plane G18 { G02 } Z... X... { R... K... I... } F... ;
 YZ-plane G19 { G02 } .. . Z... J... { R... K... } F... ;

The moving direction of the tool along the circle is as follows.

- G02: Clockwise
- G03: Counter-clockwise

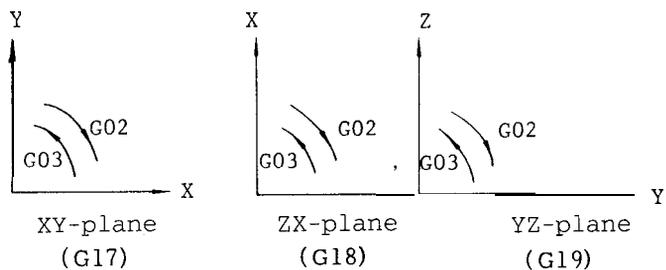
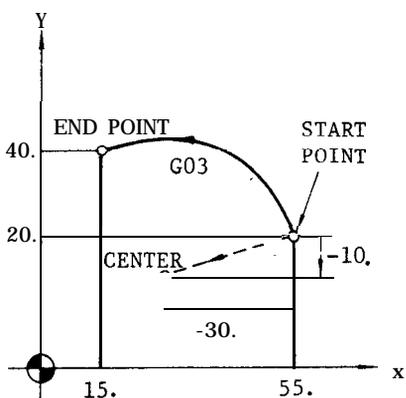


Fig. 2.13

When circular interpolation (G02,G03) is to be programmed, usually, the plane of interpolation should be specified in advance with G17, G18 or G19.

EXAMPLE

```
G17 G90 G03 X15, Y40, I-30, J-10, F150 ;
(a) Absolute command with (G90)
```



- G17 : XY-plane or X4-plane†
 - G18 : ZX-plane or Z4-plane†
 - G19: YZ-plane or Y4-plane†
- (when the 4th axis is linear)

In addition to the plane of circular interpolation, these G codes specify planes for tool radius compensation (G41, G42) . If no selection is made to the contrary, XY plane (G17) is selected automatically immediately after the switching of the power supply.

The end point of the circular arc may be specified by G90 or G91 respectively in absolute or incremental values. However, the center of the circle is always programmed in incremental values from the start point, irrespective of G90 or G91.

```
G17 G91 G03 X-40, Y20, I-30, J-10, F150;
(b) Incremental command
```

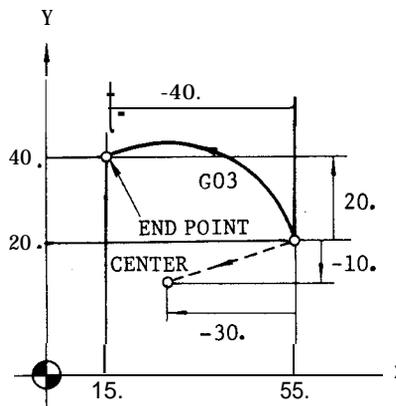


Fig. 2.14

Instead of the coordinates I, J, and K of the center of the circle, the radius can be directly specified with an R command. This is called circular interpolation with radius R designation mode.

In this case,

- when $R > 0$, a circular arc with the center angle less than 180° , and
- when $R < 0$, a circular arc with the center angle larger than 180° are specified.

```
G17 G02 X... Y... R±... F... ;
```

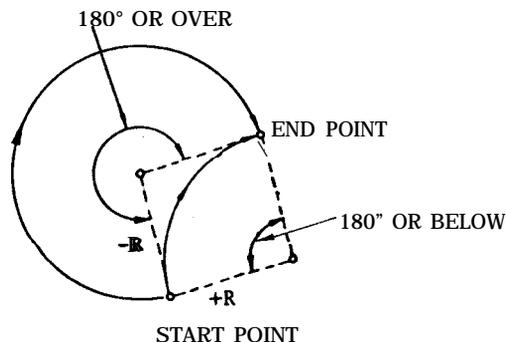


Fig. 2.15

2.9.4 CIRCULAR INTERPOLATION (G02, G03) (Cent'd)

G17 G02(G03) I.. . J.. . F.. . Ln ;

With this command, complete circular interpolations are repeated n times. Without an L designation, the interpolation is executed only once.

When a linear 4th axis option is used, circular interpolation is possible in the X α -, Z α -, and Y α - planes in addition to the XY-, YX-, and ZY- planes (where α = U, V, or W).

X α -plane	G17	$\left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\}$	X... α... R... I... J... F...;
Z α -plane	G18	$\left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\}$	Z... α... R... K... I... F...;
Y α -plane	G19	$\left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\}$	Y... α... R... J... K... F...;

Note :

G17 G02 X... $\left\{ \begin{array}{l} R... \\ I... J... \end{array} \right\}$ F... ;

Where address characters for the 4th axis is missing as in the above command, the XY- plane is automatically selected. Circular interpolation cannot be performed on the axes including rotary 4th axis.

Circular paths covering two or more quadrants can be programmed in a single block. A complete closed circle can also be programmed.

EXAMPLE

G00 XO YO ;
G02 XO YO 110. JO F100 ;
... complete circle

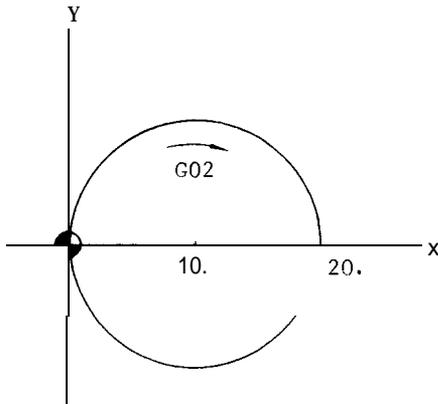


Fig. 2.16

When the coordinate values of the end point of a circular path is not exactly on the correct circular path due to calculation errors, etc., correction is made as shown below. Points O are commanded as end point. (See Fig. 2.17).

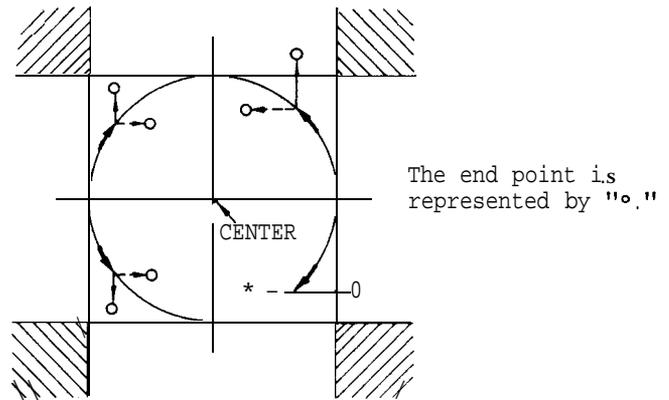


Fig. 2.17

When the end point is programmed in the hatched areas shown above, no alarm state is created, but the tool will keep on rotating. Especially when tool compensation is applied, coordinate values of the point and the center must be programmed accurately.

When radius is specified as 0 (I, J = 0 on G17 plane) in specifying circular arc, alarm 102 (CAL ERROR = DIVISION) is triggered.

When $\left| \frac{\ell}{2} \right| > R$ (ℓ : distance from the start point to the end point) at R command, the error is compensated for in the range of $\left| \frac{\ell}{2} \right| - R \leq \#6649$.

2.9.5 HELICAL INTERPOLATION (G02, G03) †

A circular interpolation on a certain plane, and a linear interpolation along an axis not included in that plane can be executed in synchronization, and this combined interpolation is called helical interpolation.

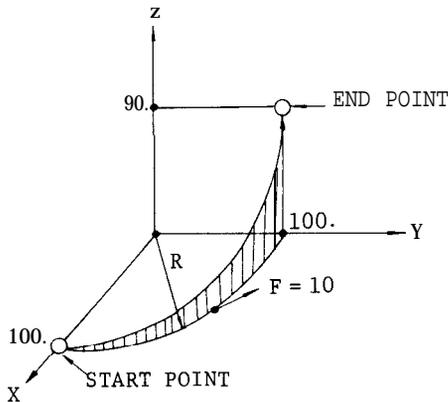
Command format

- (a) For XY-plane $G17 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X \dots Y \dots \left\{ \begin{matrix} R \dots \\ I0 \dots J \dots \end{matrix} \right\} Z(\alpha) \dots F \dots ;$
- (b) For ZX-plane $G18 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Z \dots X \dots \left\{ \begin{matrix} R \dots \\ K \dots I \dots \end{matrix} \right\} Y(\alpha) \dots F \dots ;$
- (c) For YZ-plane $G19 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y \dots Z \dots \left\{ \begin{matrix} R \dots \\ J \dots K \dots \end{matrix} \right\} X(\alpha) \dots F \dots ;$
- (d) For X α -plane $G17 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X \dots \alpha \dots \left\{ \begin{matrix} R \dots \\ I \dots J \dots \end{matrix} \right\} Z \dots F \dots ;$
- (e) For Z α -plane $G18 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Z \dots \alpha \dots \left\{ \begin{matrix} R \dots \\ K \dots I \dots \end{matrix} \right\} Y \dots F \dots ;$
- (f) For Y α -plane $G19 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y \dots \alpha \dots \left\{ \begin{matrix} R \dots \\ J \dots K \dots \end{matrix} \right\} X \dots F \dots ;$

Where α is one of the linear 4th axes U, V, or w. If no 4th axis is programmed in (d), (e), and (f), they are regarded as equal to (a), (b) and (c).

Format	Dwell time (P programmable range)
P53	0 - 99999.999 sec

EXAMPLE



G17 G03 X0 Y100. R100. Z90.F10. ;

Fig. 2.18

Notes :

- The circular arc should be within 360°.
- As long as above note (a) is satisfied, the start and end points can be taken at any time.
- The feedrate F means the tangential speed on the plane of circular interpolation.
- Tool radius compensation C† can be applied only to the circular path on the plane of circular interpolation.

2.9.6 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.

The value does not depend on metric /inch input or metric /inch output.

EXAMPLE

G04 P2500 ;
Dwell time: 2.5 sec.

Two types of dwell can be selected by parameter:

Dwell when the specified value in the command block before the dwell block is identified by lag pulses of servo, or dwell on completion of pulse distribution.

2.9.7 EXACT STOP (G09, G61, G64)

- Exact stop (G09)

When a block containing G09 is executed, the program advances to the next block after completing a block in the Error Detect On mode (Note a). This function is used when sharp corners are desired. G09 is non-modal, and is effective only in the block in which it is contained.

- Exact stop mode (G61)

When once G61 is commanded, all the following blocks will be completed in the Error Detect On mode before proceeding to the next block.

- Exact stop mode cancel (G64)

This G command is for canceling the effect of G61.

2.9.7 EXACT STOP (G09, G61, G64) (Cent'd)

Notes :

. In the Error Detect On mode, the program proceeds to the next block only after the number of servo delay pulses is found to have decreased below a permissible limit following the complete distribution of circular interpolation command pulses.

In the G09 and G61. off modes, the program proceeds to the next block immediately after the complete distribution of the pulses of ordinary linear and circular interpolations, and therefore, because of the servo delay, tool paths are rounded at the corner. This mode is called "Error Detect Off" mode

. For rapid traverse, the Error Detect On and Off modes are controlled only by G00 and G06, and not by the above G codes.

2.9.8 TOOL OFFSET VALUE DESIGNATION (G10)

With a G10 command, correction of tool offset values and work coordinate system can be made as follows.

(1) Designation of tool offset value (G10)

Normally, tool offset values are written in by MDI. On the other hand, with a program G10 P... R... ; , (where P = tool offset number and R = tool offset value) , any programmed offset values can be replaced by a designated value.

When G10 is commanded in the G90 mode, R is stored as it is.

When G10 is commanded in the G91 mode, R is added in the previous tool offset value.

(2) Changing work coordinate system[†]

(a) Corresponding to G54 through G59, separate work coordinate systems are set up as setting data in advance.

G10 Q2 Pm X... Y.. ,Z. .. (a...)

(where Q2 is used to discriminate from tool offset value designation and a means to set up a work coordinate system. Pm (m = 1 to 6) corresponds to the work coordinate system m to be set up.)

For P1... G54

For P2... G55

For P6... G59 corresponds.

With the above command, data of any desired work coordinate system can be changed.

(b) For the work coordinate system setting B specification, change the setting of the work coordinate system by specifying G10 Q2 Pm Jn X... Y.. . z.. . (a...); . (Jn specifies J1 to J5. The meaning of Q2 and Pm is the same as mentioned in (a).)

2.9.9 CIRCLE CUTTING (G12, G13)[†]

This is a canned cycle which includes a complete series of movements for machining a circle in a single block. It includes the following functions.

. Format

G12(G13) I... D... F... ;

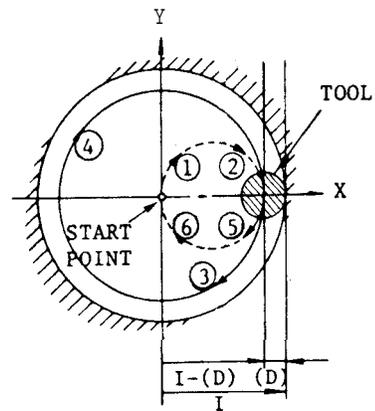


Fig. 2.19

. Designation of rapid traverse section R

G12 (G13) I... R... D... F... ;

With this command , a circular bore is machined as shown below. Numerals following an address character R specifies rapid traverse sections.

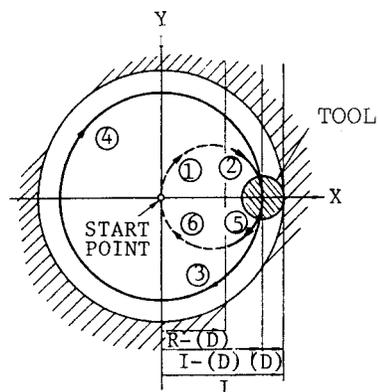


Fig. 2.20

Commanding repeated circle designation

G12(G13) I... D... L... F... ;

With this command, the circular bore surface can be executed L times.

Commanding spiral circle Q, K

G12(G13) I... D... K... Q... F... ;

With this command, the tool is moved along a spiral before finally finishing a circular hole, as shown below. For the sake of simplification, the diagram shows the tool path with a zero radius (D = 0) . Q (radius increment) must be programmed without sign.

Tool path



(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

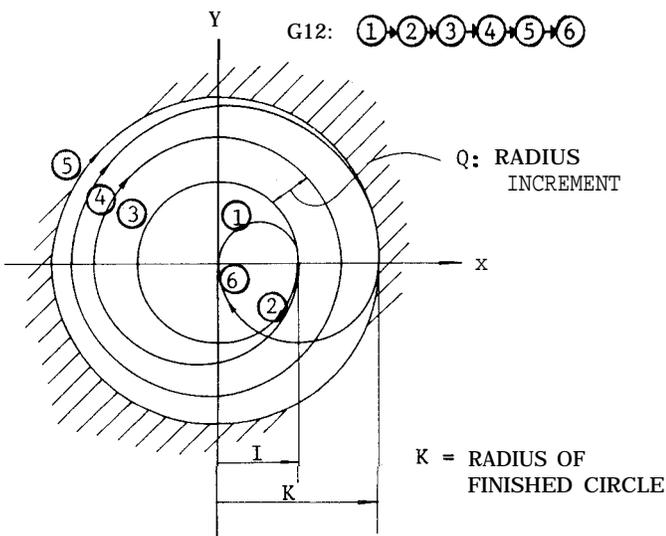


Fig. 2.21

Combined designation of rapid traverse section, repeated circle and spiral circle.

Rapid traverse section, repeated circle designation and spiral circle can be commanded in combination as shown below.

G12(G13) I... D... R... (or J...) K... Q... L... F... ;

Notes :

Circle cutting is possible only on the XY plane.

The tool speed in the rapid traverse section is set by parameter #6225.

Feedrate override cannot be applied to rapid traverse rate. While dry run switch is ON, the tool speed conforms to dry run speed.

With a circle cutting command (G12,G13) , the tool is offset for its radius compensation without the use of G41 or G42 (tool radius compensation) . When using G12 or G13, cancel tool radius compensation with G40.

In the explanation above, only motions in the + direction of X-axis is considered. With proper use of signs for I, K and D codes, motions in the - direction of X-axis (symmetrical with respect to Y-axis) can be commanded. In Fig. 2.23, signs of I, R and (D) are minus. However, cutting in the Y-axis direction is impossible.

I, J, K, R, Q and L codes in circle cutting command are effective only in the block containing them.

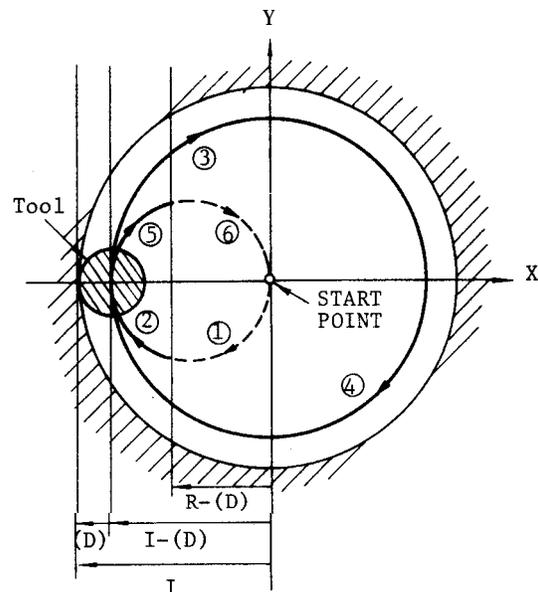
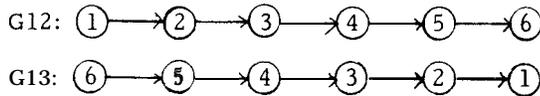


Fig. 2.22

2.9.9 CIRCLE CUTTING (G12, G13) (Cont'd)

Tool path



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

The radius I of finished circle and the rapid traverse section R are subject to the following restriction. When values not in conformity with the restriction are programmed, this is regarded as an error.

$$|R - d| < |I - d|$$

EXAMPLE

G00 Z-40. ;
 G12 150. R40. D10 F300 ; D10 = 10.0 mm
 G00 Z40. ;

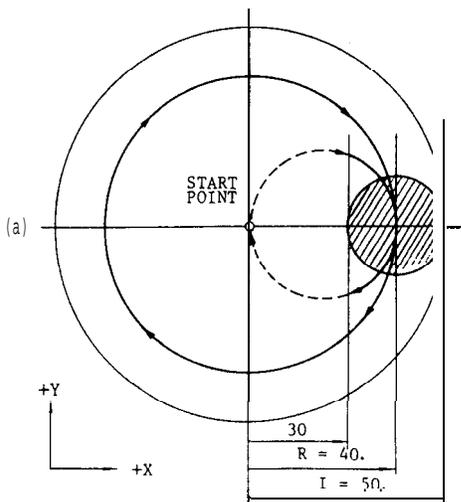


Fig. 2.23

G00 Z- 40. ;
 G13 I-50. J7. D15 F300; D15 = -8.0 mrn
 G00 Z40. ;

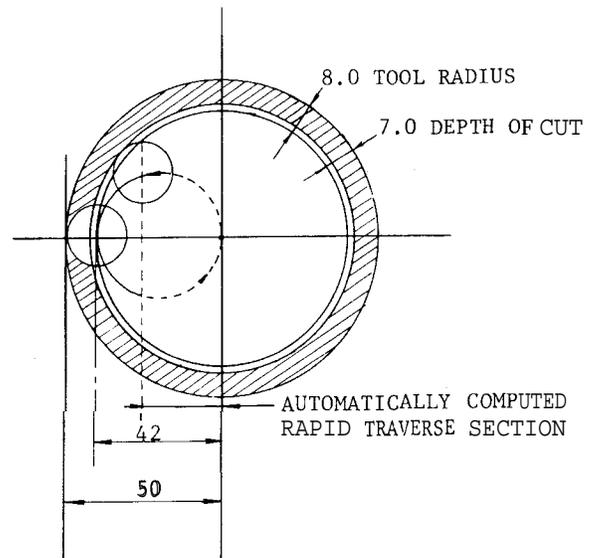


Fig. 2.24

When G12 or G13 is commanded, the following two conditions must be satisfied :

Condition 1 : $D \geq J$

Condition: $I - D - J > 0$

Unless otherwise, section R is invalidated and the machine moves at F-feed speed.

2.9.10 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

When the 4th axis[†] of linear axis is selected, the following planes are newly added.

G17 : XY-plane or X α -plane
 G18 : ZX-plane or Z α -plane
 G19 : YZ-plane or Y α -plane

α means U, V or W axis.

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 4th axis of rotary axis.

The XY-plane (G 17) is selected when the power is turned on.

2.9.11 INCH/METRIC DESIGNATION BY G CODE (G20, G21)†

Unit of input data are selectively specified by the following G codes between metric and inch.

G code	Input unit
G20	Inch
G21	Metric

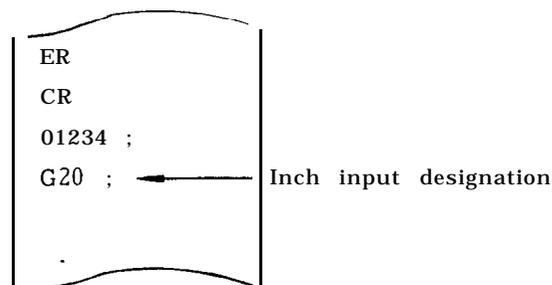
These G codes are programmed at the leading end of a block of its own. If one of these G codes are commanded, the units of all the following motions are changed afterwards.

- subsequent programs
- tool offset values
- part of setting parameters
- part of manual movements
- displays

Notes :

When G20 or G21 is commanded, the setting of inch/metric selection is changed. Therefore, the state of G20/G21 at the time of power application depends on the setting by parameter #6001 DO. "

EXAMPLE



When G20/G21 selection is commanded in the program, take the following procedure beforehand.

- A. When work coordinate system (G54 to G59) is used, return it to base coordinate system.
- B . Cancel all tool compensation command. (G41 to G48)

Take the following procedure after the command of G20 /G21 selection.

- A. Program absolute zero point (G92) for all axes before move command.
- B. In principle, make the display reset operation when current position display (external) is used.

The tool offset values are processed differently in the G20 mode and the G21 mode. G20/G21 must be commanded after modifying the tool offset values.

Stored off-set values	Processing in G20 (Inch)	Processing in G21 (Metric)
15000	1.5000	15.000 mm

2.9.12 STORED STROKE LIMIT (G22, G23) †

This function is for checking the current tool position during manual or automatic operation for entry into the prohibited area specified by parameters or by G22. If the tool enters a prohibited area, machine operation is stopped and an error sign is displayed.

- 1st prohibited area (stored stroke limit 1)

The area outside the area specified by a parameter is a prohibited area. Generally, this can be used as a substitute of overtravel checking function. Upper limit point A₁ and lower limit point B₁ are specified by parameters.

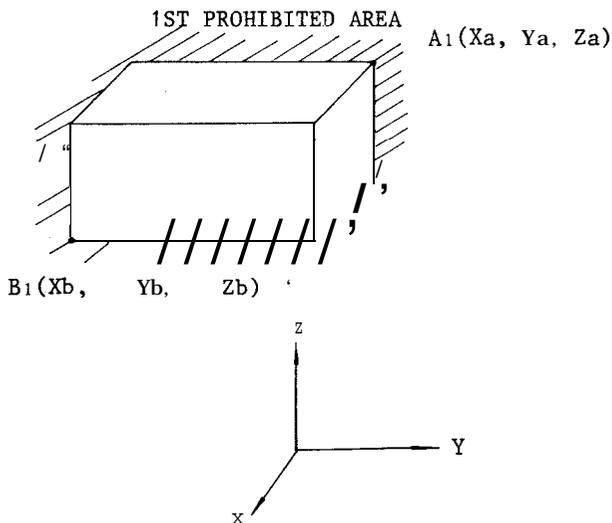


Fig. 2.25

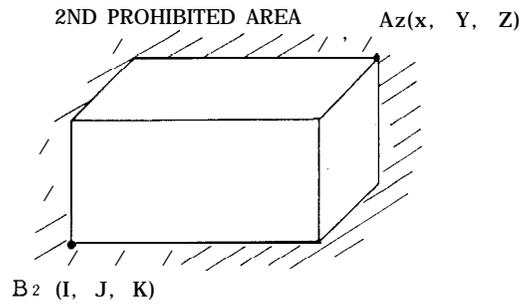
2.9.12 STORED STROKE LIMIT (G22, G23) † (Cont'd)

2nd prohibited area (Stored stroke limit 2)

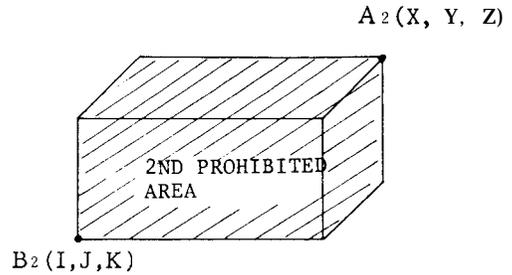
The boundary of the 2nd prohibited area is specified by a parameter setting or by G22. The inside or the outside of the boundary can selectively be made a prohibited area by means of parameter setting.

G 2 2 X... Y... Z... I... J... K... ;
 └──────────┬──────────┘
 C point D point
 Upper limit Lower limit

With this command, the checking of the 2nd prohibited area is started, and with G23 ;, the check function is cleared.



Where 2nd prohibited area is outside.



Where 2nd prohibited area is inside.

Fig. 2.26

Table 2.20 Setting of Stored Stroke Limit for Each Axis

		x	Y	z	Division
1st prohibited area	Point A ₁	#6600	#6601	#6602	Parameter
	Point B ₁	#6606	#6607	#6608	
2nd prohibited area	point A ₂	#6510	#6511	#6512	Setting
	Point B ₂	#6513	#6514	#6515	

Note: Point A sets plus value of boundary line on the machine coordinate system and point B sets minus value .

The parameters for specifying the inside and the outside of the 2nd prohibited area are as follows .

#6007D0	Meaning
"0"	Inside prohibition
"1"	outside prohibition

The 2nd prohibited area checking function can also be turned on and off with the following setting number.

#6001D1	Meaning
"0"	2nd prohibited area check; off
"1"	2nd prohibited area check; on

Notes :

The 1st and the 2nd prohibited area can be specified overlapping each other.

Boundary lines on the minus side are included in the prohibited areas. Boundary lines on the plus side are not included in the prohibited areas.

All the prohibited areas become effective after a manual return to reference point or a return to reference point by G28 after turning on the power supply.

If the tool is in the prohibited area at the time when the prohibited area becomes effective, this is immediately regarded as an error. In this case, turn off the 2nd prohibited area by the setting of the setting number, and either rewrite the data or move the tool out of the prohibited area manually.

If an alarm state is created by the entry of the tool into the prohibited area, the tool can move only in the returning direction.

The stored stroke limit cannot be checked during a machine lock operation.

2.9.13 PROGRAM COPY† (G25)

(1) The command G25 P $\overset{p1}{\cdot\cdot\cdot\cdot}$ $\overset{p2}{\cdot\cdot\cdot\cdot}$ Q $\overset{q1}{\cdot\cdot\cdot\cdot}$ $\overset{q2}{\cdot\cdot\cdot\cdot}$ L
 executes L times any program from sequence p2 of program p 1 to sequence q2 of program q1.

G25: Program copy G code

P: p1; First O number

p2; First sequence number (N)

Q: q1; Last O (p1=q1)

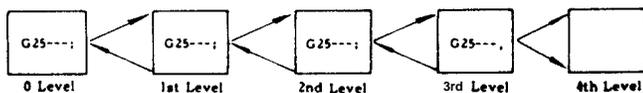
q2; Last sequence

Each will be commanded with four digits. The leading zeros of p2 and q2 cannot be omitted.

L: Number of repetitions (One time if omitted)

(Z) Although the G25 command can be given by tape, MDI or memory operation, it will be necessary to store the program to be copied in the memory. When sequence numbers are overlapped, the first sequence number will become effective.

(3) The G25 command may also be used again in the program being copied. This is called multiple program copying and is possible up to a maximum of four levels. However, error " 042" will occur if four levels are exceeded.



(4) Program copying can be commanded in a fixed cycle (G70 to G72, G73, G76, G81 to G89).

(5) Simple jump command "M99" can be used in the program being copied by the G25 command. It is the same as the M99 function commanded in the main program.

Notes :

(a) Error "042" will occur if the total number of levels called by G25 and M98 exceeds four when used in a subprogram.

(b) If G25 is used during a user macro call (G65, G66), Up to four separate levels are possible.

(c) Since the sequence number specified by address p2 and q2 will be searched from the first number of each program, care will be required not to duplicate the sequence numbers. If duplicated, the first sequence number will become effective.

(d) In command G25 P (pi) (p2) Q (q1)(q2) L ;, the following errors will occur.

Error "070 when p 1 cannot be searched

Error "04 1" when p2 cannot be searched

Error "040 when p 1 and q 1 are not the same

(e) If a reset operation for errors or an M30/M02 reset occurs when executing command G25, execution will return to the beginning of the called program of the O level.

(f) If a P or Q command is less than four digits, it will be considered a sequence number and the program sequence number will be searched.

(g) The leading zeros in both commands P and Q can be omitted. However, for five digits or more, the lower four digits will be considered the sequence number.

(h) If M98 or M99 is commanded in the G25 block, error "040" will occur.

(i) Since M99 will have priority if it is in the block (End block of G25) specified by q2 of G25, the command will return to the beginning of the program being executed. Do not command M99 in the block specified by q2.

(j) The O (q1) at the end of the Q command can be omitted. When omitted, it will automatically be considered to be the same as the leading O of the P command.

(k) A program copy command in a fixed cycle cannot be commanded in the same block with the fixed cycle command. If commanded, G code error "021" will occur.

(l) When the block indicated by G25 q2 (end block of G25) is a G25 block, q2 is disregarded and the next block is proceeded.

Therefore, create a program as shown page 32 and 33.

2.9.13 PROGRAM COPY '(G25) (Cent'd)

Example

G25 P7 Q8 L1;

:

N2 G73 Z-1, R-2. Q1. F100;

N3 G80;

N4 GO X1 O.;

N7 G25 P2 Q4 L9;

N8 G25 P2 Q3;

N9 G90 X-90. Y-10.;

AFTER MODIFICATION

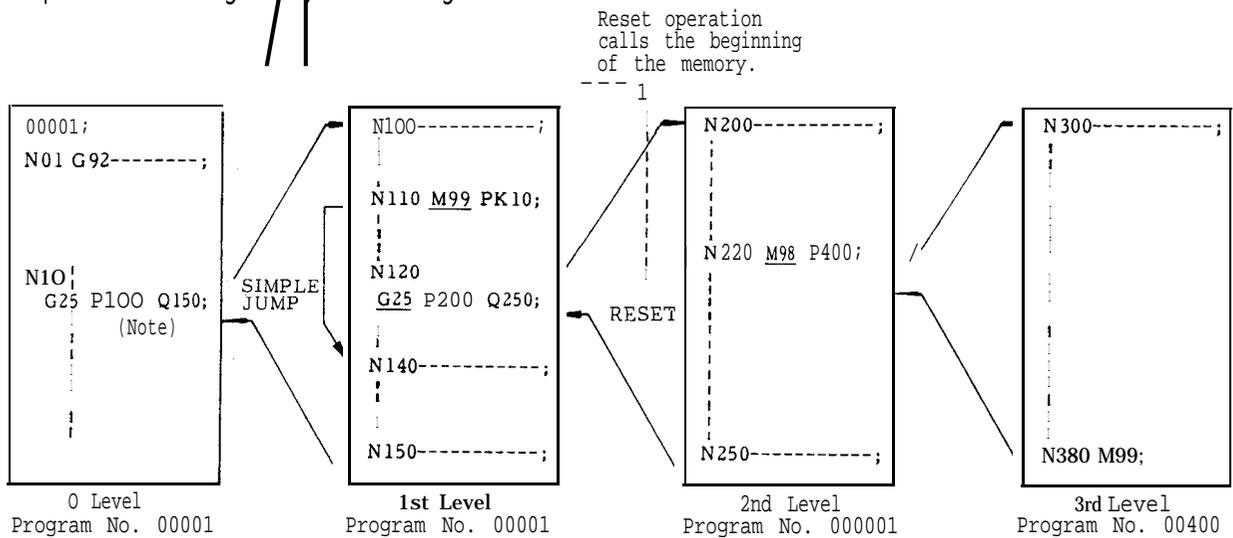
N7 G25 P2 Q4 L9;

G25 P2 Q3;

N8;

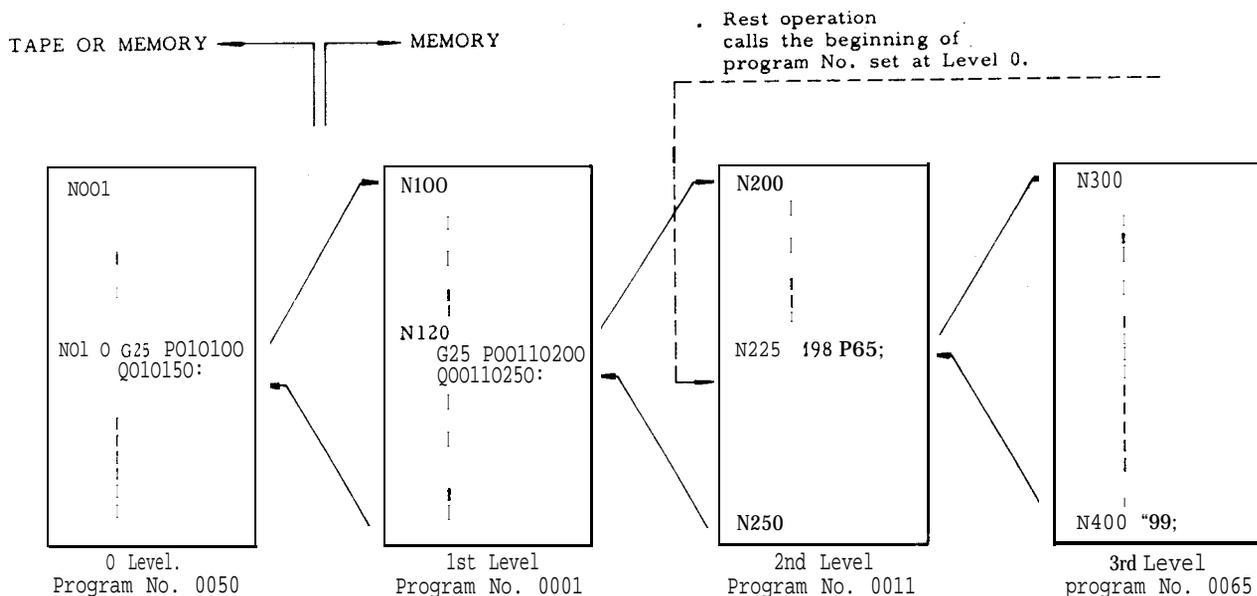
N9 G90 X-90. Y-10. ;

ApEORMEMORY 7r MEMORY



Note: Always use the program number in the P command when commanding from a tape.

Example: N1OG25 P010100Q1550; Program number



Notes :

1. M98 can be used in a program copied with G25. Four levels may not be exceeded even when using G25 with M98.
2. Care should be taken when jumping to a different L level with M99 since execution will become endless with no means of escape.

2.9.14 REFERENCE POINT CHECK (G27)[†]

This function is for checking the correct return of the tool to the reference point after performing a cycle of operation in accordance with a program which starts at the reference point and ends at the reference point.

G 27 X... Y... Z... (α^+ ...);

With this command, the tool moves towards the specified position along the three axes (4 axes[†]) simultaneously but independently, and after the arrival at the specified point, the point is checked for the conformity to the reference point. If any of the axes is omitted in the command, the tool does not move in that axis and no check is made in that axis.

If the point is in conformity with the reference point, the reference point return lamp lights. If the tool is correctly in the reference point in all the axes, automatic operation is performed further, but if the tool is not in the reference point even in one axis, this is regarded as an error (alarm 241 - 244 display), and the automatic operation is interrupted. (Cycle start lamp goes off.)

If G27 is commanded in the tool offset mode, the tool return point is also offset. Cancel the tool offset mode when commanding G27.

Reference point as meant here is a freed point relative to the machine to which the tool returns by the manual reference point return motion or by G28 automatic reference point return motion. Refer to 5.2.1, "MANUAL RETURN TO REFERENCE POINT" on page 174. The mirror image function can be applied to the G27 command. To avoid non-conformity errors, clear the mirror image mode with M94 (Mirror image off) before commanding G27.

2.9.15 AUTOMATIC RETURN TO REFERENCE POINT (G28)[†]

G 28 X... Y... Z... (α^+ ...);

With this command, the tool is sent back to the reference point. The tool moves towards the specified points in rapid traverse, and automatically stops at the reference point.

The tool moves simultaneously in up to 3 axes (4 axes[†]). However, the tool will not move in the direction of the axis for which a coordinate instruction is omitted.

2.9.15 AUTOMATIC RETURN TO REFERENCE POINT (G28) † (Cont'd)

EXAMPLE

```
G28 X... Y... Z... ;
```

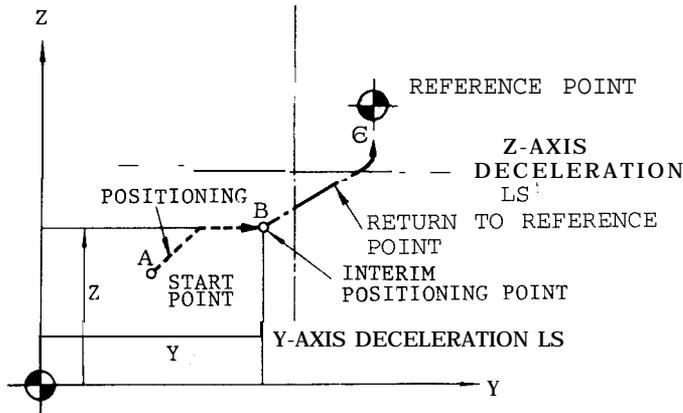


Fig. 2.27

“Return to reference point” involves the same series of motions as the manual return to reference point.

Notes :

If G28 is commanded in the tool radius compensation mode (G41, G42) or in a canned cycle, this is regarded as an input error “024.”

If G28 is commanded in the Mirror Image mode (M95), this constitutes an input error “058.”

The tool position offset command is not cancelled by G28. Be sure to cancel it before commanding G28. If G28 is given in the tool position offset mode, the tool motion by the succeeding program becomes as described below. Care should be taken.

- A. When the succeeding program is made in the incremental mode:
Tool moves by the amount of incremental value from the reference point. The tool offset is not effective.
- B. When the succeeding program is made in the absolute mode:
Tool moves to the position which is specified by absolute value and tool offset value.
- C. When G29 is given immediately after the G28:
By G29 command, the tool moves to the offset interim positioning point and the succeeding motion is made according to the item A and B.

When returning the tool to the reference point for the first time after turning on the power supply, pay attention to the tool position. Refer to 5.2.1 MANUAL RETURN TO REFERENCE POINT †.

Return to reference point in rapid traverse

In addition to the above “Automatic Return to Reference Point,” “Rapid Traverse Return to Reference Point” function may be incorporated in the control. With this function, the motion sequence is as follows.

- After positioning at an interim positioning point B, the tool directly moves to the reference point in rapid traverse. The returning time is shorter than that with the ordinary return to reference in which deceleration LSS are used in all the axes.
- With the “Rapid Traverse Return to Reference Point,” point B may not necessarily be within the reference point return possible area.
- The rapid traverse return to reference point becomes possible only after the tool has been returned once to the reference point in all the axes by manual operation or by G28, following the turning on of the power supply.
- Rapid traverse return to reference point is effective only with G28. Manual return motions are not changed by it.
- Where a 4th axis is used, when no command is given for the 4th axis in a G28 command, and when the tool has been returned to the reference point in the X-, Y-, and Z-axis, the tool moves to the reference point in the rapid traverse return mode. If a command for the 4th axis is included in the command, the tool returns to the reference point in the ordinary return mode, unless the return motions in all the 4-axes have been completed.
- For return to reference point in rapid traverse, tool cannot be moved in RAPID or JOG mode unless REFERENCE POINT RETURN switch turns off after completion of reference point return.

2.9.16 RETURN FROM REFERENCE POINT (G29) †

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

```
G28 Y... Z... ;      Point A → B → C
      └──────────┘      (Reference point)
           Point B

G29 Y... Z... ;      Point C → B → D
      └──────────┘
           Point D
```

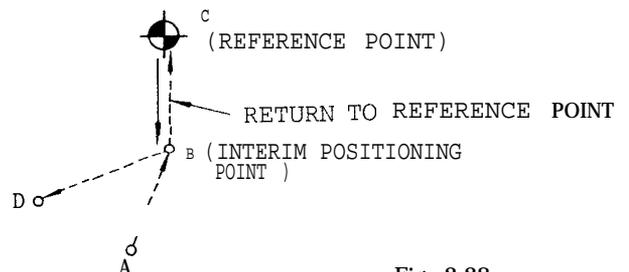


Fig. 2.28

When G29 is programmed, it is not necessary to consider the distance between point B and C in the program. Particularly when an incremental command 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 t) by G29. However, in an axis for which a coordinate command was omitted, the tool will not move.

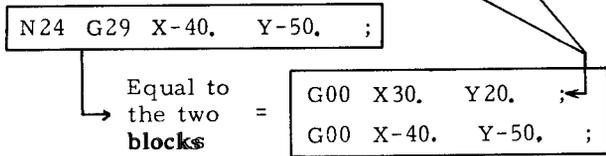
If G28 is programmed a number of times, the final coordinates of point B which the last G28 creates is effective for the move of G29.

EXAMPLE 1 (In the case of absolute input)

```

N21 G90 ;
N22 G28 Z10. Y20. ; → (0, 20, 10.)
N23 G28 X30. ; → (30, 20, 10.)

```



EXAMPLE 2

```

N31 G91 ;
N32 G28 Z... ;
N33 G28 X20. Y40.
N34 M06 ;
N35 G29 X40. Y-40.

```

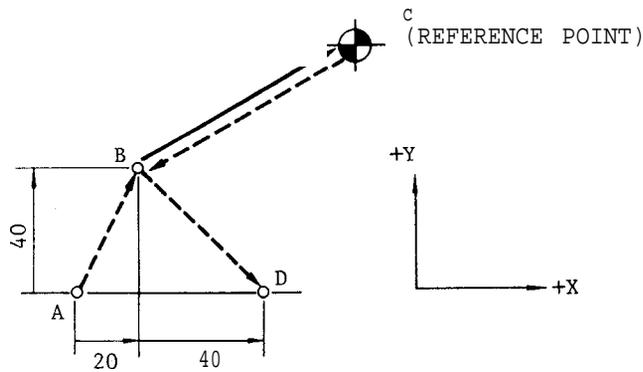


Fig. 2.29

Notes :

An input error "024" occurs if G29 is programmed in tool radius compensation mode (G41, G42) or during canned cycle mode (G73, G74, G76, G77, G81 to G89) .

An input error "059" occurs if G29 is given without execution of G28 after the control is turned on.

In principle, cancel tool offset before programming G28 or G29. If they are programmed when offset is also effective, interim positioning point B will also be offset, and the tool passes point B' .

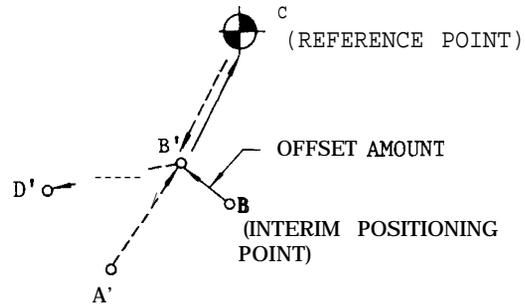


Fig. 2.30

An input error "058" occurs if G29 is given during mirror image (M95) .

The following command or operation must not be taken because interim 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 coordinate system (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.

2.9.17 RETURN TO 2ND, 3RD AND 4TH REFERENCE POINT (G30)

```
G30 Pn X... Y... Z... (a t...);
```

(where Pn = P2, p3, p4)

With this command, the tool first moves to an interim positioning point, and then, moves to the 2nd, 3rd or 4th reference point.

- | | |
|-------------------------|---|
| P2: 2nd reference point | } When P is omitted, the tool moves to the 2nd reference point. |
| P3: 3rd reference point | |
| P4: 4th reference point | |

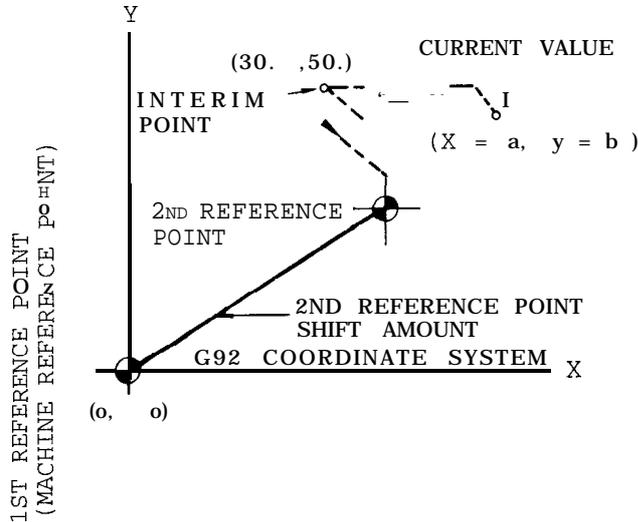
If any axis of the coordinate instruction is omitted, the tool remains motionless in the direction of that axis.

Each reference point is specified by the parameters (#6612 to #6629) before hand.

2.9.17 RETURN TO 2ND, 3RD AND 4TH REFERENCE POINT (G30)† (Cent'd)

EXAMPLE

G30 P2 x30. Y50. : The tool returns to the 2nd reference point moving in the X and Y directions.



(a) First, positioning is made at the interim point (X = 30. Y= 50.). The interim point is a coordinate value of the G92 coordinate system or work coordinate system (G54 to G59).

(b) Then positioning is executed at the second reference point as follows :

- X-axis 2nd reference point position = X-axis machine reference point + (#6612)
- Y-axis 2nd reference point position = Y-axis machine reference point + (#6613)

In the above example, when ;

- X-axis machine reference point = 0,
- Y-axis machine reference point = 0,
- #6612 = 10.000 and
- #6613 = 20.000 in the G92 coordinate system, the machine moves to the following positions by rapid traverse.
- X-axis 2nd reference point position = X-axis machine reference point + (#6612) = 0 + 10.000 = 10.000
- Y-axis 2nd reference point position = Y-axis machine reference point + (#6613) = 0 + 20.000 = 20.000

(c) Output during stay in the 2nd reference point (#12014, #12015) is provided in the range of 3 micron ± 2nd reference point.

Notes :

Three items except the last one in NOTES of 2.9.15 AUTOMATIC RETURN TO REFERENCE POINT (G28)† on page 33, apply to G30 in the same manner.

When G29 is commanded after G30, the tool moves to the designated point by G29 by way of interim positioning point designated by G 30. However the interim positioning point is renewed on the axis designated by G 30.

2.9.18 SKIP FUNCTION (G31)†

G31 X... Y... Z... (a t...) F... ;

With this command, a special linear interpolation is commanded. During the interpolation movement under the command of this program, whenever a skip signal is inputted, the interpolation is interrupted immediately, and the program advances to the next block. From the moment that a skip signal is inputted to the time the control start to process the signal, delay time is less than 0.5m sec. G31 is non-modal.

EXAMPLE

N100 G90 G31 X100. Y50. ;
N200 GO1 x80. Y15. ;

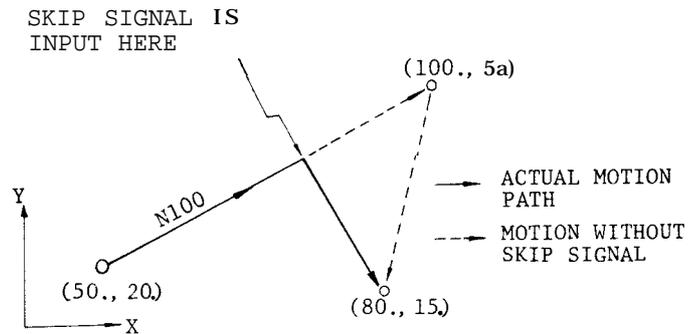


Fig. 2.31

When 'G31 block is executed without a skip signal being inputted, the machine stops at the end of the block, and the alarm code "087" is displayed.

Feedrate of the tool is set for G 31 blocks selectively by one of the following two methods as specified by parameter #6019D4.

To be specified by F similar to ordinary programs.

To be set in advance by parameter #6232.

When a skip signal is inputted, the coordinate values at the moment are automatically stored as parameter data.

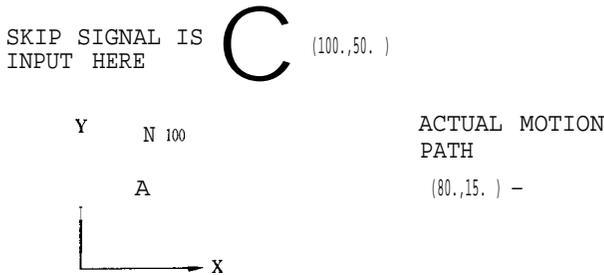
- #6552 ... storing X coordinate value
- #6553 ... storing Y coordinate value
- #6554 ... storing Z coordinate value
- #6555 ... storing 4th coordinate value

These coordinate values indicate the positions when skip signal is ON and not the position when the tool is stopped.

These data can be treated as coordinate data in user macros.

When a skip signal is not given in spite the execution of G31 by setting (#6004DO), the program moves on to the next block automatically.

Note : When actual skip stop detecting position is near the skip final reference point, the dead zone exists in the skip signal processing in the range shown below. At this time, alarm "87 occurs when #6004 Do = 0. When #6004 Do = 1, the final point value of the commanded axis is stored in #6552 to #6555.



(Metric input)

$$|a| < 1000 \times F \left(\frac{2}{7500} + \frac{1}{60} \frac{2}{XKP \cdot 1000 \times 60} \right)$$

(Inch input)

$$|a| < 10000 \times F \left(\frac{2}{7500} + \frac{1}{60} \frac{2}{XKP \cdot 1000 \times 60} \right)$$

KP : Position loop gain constant (1/S)

F: Reference speed (metric input : mm/min, inch input : inch/rein)

2.9.19 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 C 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.

Table 2.21 G codes of Tool Radius Compensation C

G code	Group	Meaning
G40	07	Cancellation of tool radius compensation C
G41	07	Tool radius compensation C, left
G42	07	Tool radius compensation C, right

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

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 DOO is commanded, tool radius will be regarded as "0."

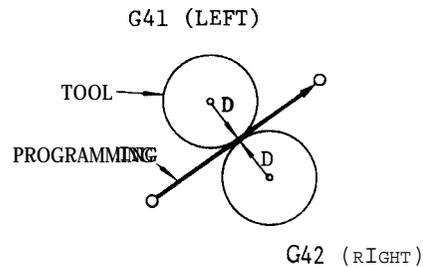


Fig. 2.32

Switching between G41 and G42 can be made in compensation mode. Details will be given in item 5 on page 40.

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 02 group. The XY-plane (G17) is in effect at the time power is turned on.

Table 2.22 G Codes for Designation of Planes

G code	Group	Meaning
G17	02	XY-plane
G18	02	ZX-plane
G19	02	YZ-plane

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

2.9.19 TOOL RADIUS COMPENSATION C
(G40, G41, G42) † (Cent'd)

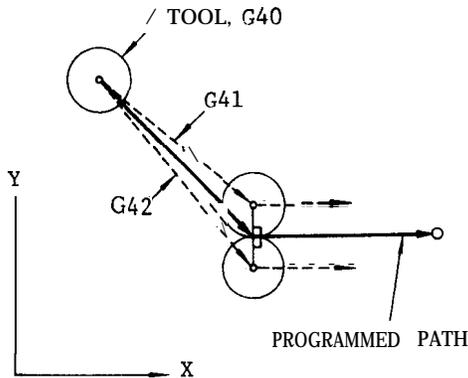
Make sure to designate a G code for plane designation in the same block as that of G41, G42 or in a preceding block. Plane designation cannot be made in a compensation mode. It is not possible to apply tool radius compensation in a plane including the 4th axis†.

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 "026) 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. . . ;
```

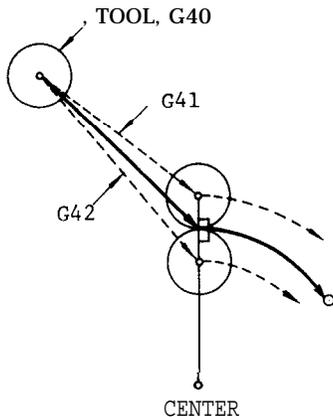
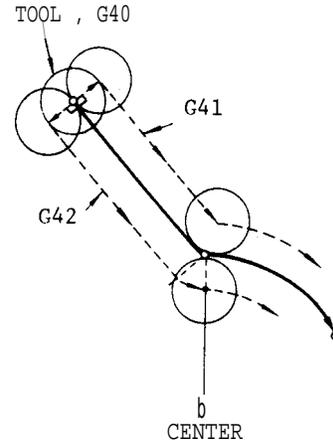


Fig. 2.33

EXAMPLE B

```
(C) G17 G01 F. . . ;
    G41(G42) D. . . ;
    x. . . Y. . .
    G02 X. . . Y. . . J. . . ;
```



```
(d) G17 G01 F. . . ;
    G41(G42) D. . . ;
    G02 X. . . Y. . . J. . . ;
```

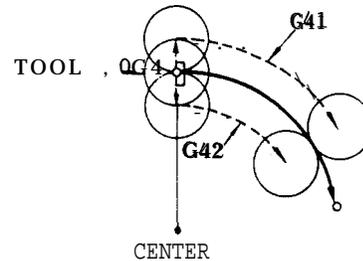


Fig. 2.34

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 (d) 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. The blocks without move command can be programmed continuously up to two. Input error occurs if move commands on the compensation plane are not programmed in more than three blocks.

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 workpiece.

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

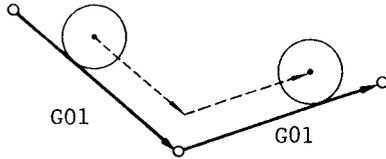


Fig. 2.35

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

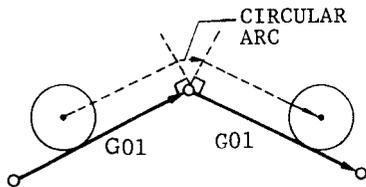


Fig. 2.36

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 computation, depending on the work. For details, refer to 2.8.6, "CIRCULAR PATH MODE ON/OFF ON TOOL RADIUS COMPENSATION C (M97, M96)" on page 17.

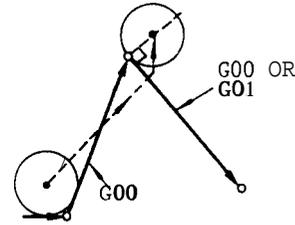
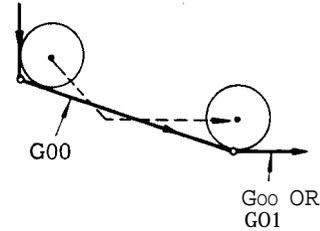
M96 . . . Tool radius compensation circular path ON

M97 . . . Tool radius compensation circular path OFF (execution of intersection computation)

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

c. 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. 2.37

2.9.19 TOOL RADIUS COMPENSATION C
(G40, G41, G42) † (Cont'd)

D. Shape requiring care

Do not program a wedge shape having an acute angle.

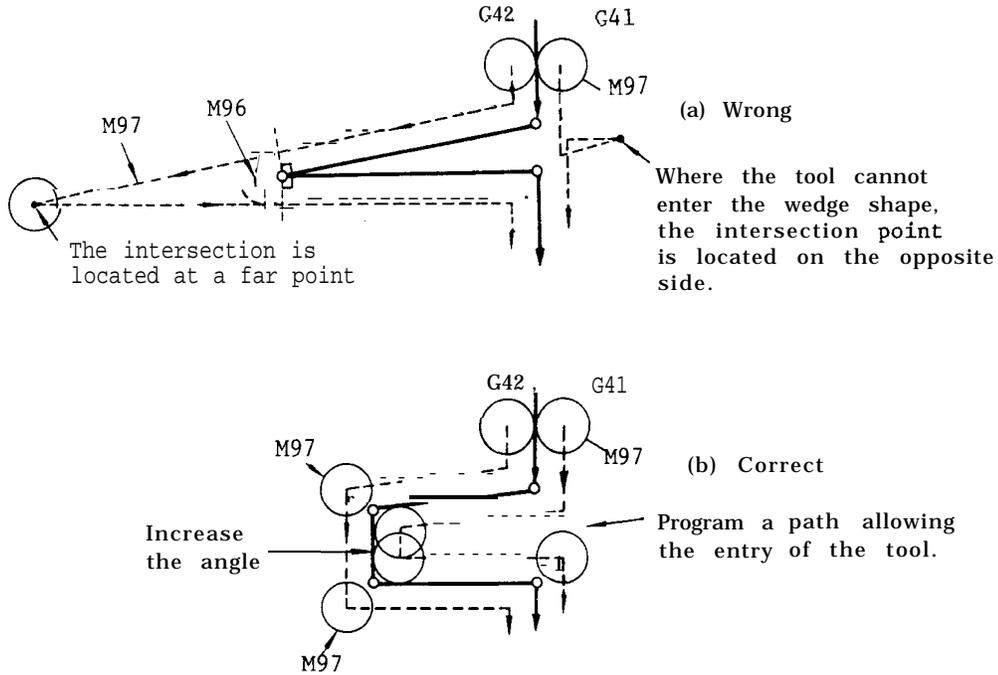


Fig. 2.38

5. Command involving no movement in compensation mode

The control normally reads in advance two blocks during tool radius compensation mode and computes 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 computing.

The blocks with no coordinate instructions can be processed continuously up to two blocks. When coordinate instructions are missing in three 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, three or more blocks without movement command plane will not follow.

EXAMPLE

```
G17 G01 G41 X... Y... D... F... ;
X... Y... ;
```

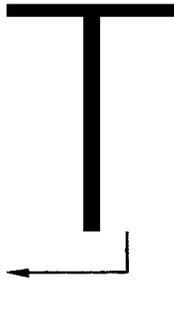
```
x... y... ;
```

```
G04 P1000 ;
x... y... ;
```

```
X... Y... ;
z...
z...
x... y...
```

```
X... Y... ;
```

```
G40 X... Y... ;
```



Blocks without movement
in compensation plane.
(When these blocks are within
two, machining is made smoothly.)

If no move command is programmed in three continuous blocks, offset in the block immediate before them is made on the normal line at the end point. Where movement in the compensation plane cannot be programmed in three 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... ;
```

```
N010 X... Y... ;
```



```
N012 Z... ;
```

```
N019 Z... ;
```



```
N029 X... Y... ;
```

```
N030 G40 X... Y... ;
```

XY-plane

Z axis
(3 blocks
or above)

XY-plane

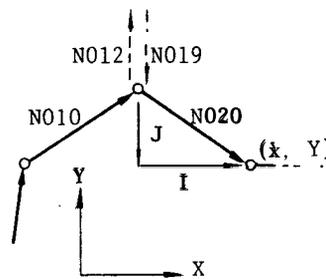


Fig. 2.39

2.9.19 TOOL RADIUS COMPENSATION C (G40, G41, G42) † (Cent'd)

The dummy block is not programmed for actual movement but it only provides data required for tool radius compensation computation. 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 and J. I, J and K are used as the addresses of this dummy instruction, and they correspond to X-, Y-, Z-axis 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 GO1 X... Y... ;

N051 GO1 I(b) J(-a) ; — Dummy block

N052 Z... ;
 N053 Z... ;
 N059 Z... ;

} z-axis

N060 G03 X... Y... I(a) J(b) ; — Circular interpolation

N061 GO1 X... Y... ;

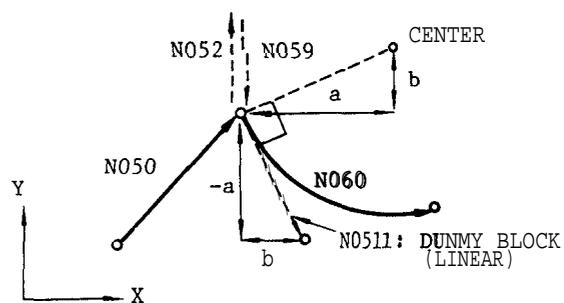


Fig. 2.40

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

EXAMPLE

N10 G17 G01 F... ;
 N11 G41(G42) D... ;

N20 GO1 X... Y... F... ;

N21 G42(G41) X... Y... ;	— Block of switching
--------------------------	----------------------

N22 x...

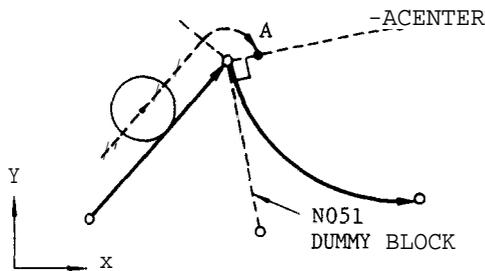
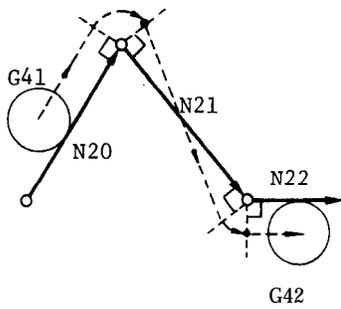


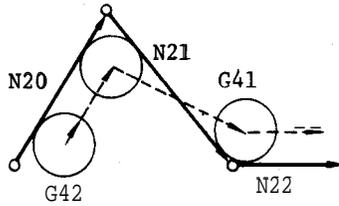
Fig. 2.41

6. Switching between G41 and G42 in compensation mode

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



(a) G41 → G42
(M96 mode)



(b) G42 → G41

Fig. 2.42

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

```

G42 (G41) ;
  X... Y... ;

```

7. Change of tool radius value in compensation mode

New D code commanded in compensation mode is effective in the block next to the commanded block.

8. Method of cancellation of compensation

G40 is the command for canceling tool radius compensation C 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.

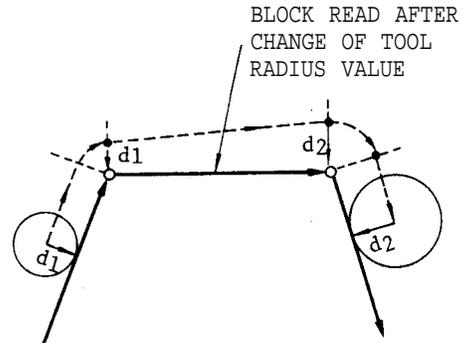


Fig. 2.43

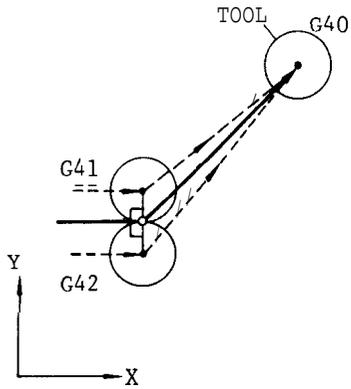
Therefore, no portion will be left unmachined even when a cancellation with sharp angle is programmed. Because G40 accompanies cancelling movement, program it in the G00 or G01 mode like G41, G42. An input error "027" occurs if group A other than G00, G02 is used.

2.9.19 TOOL RADIUS COMPENSATION C
(G40, G41, G42) (Cent'd)

EXAMPLE A

(a) G41 (G42)

```
G01 X... F... ;
G40 X... Y... ;
```



(b) G41 (G42)

```
G02 X... Y... I... J... ;
G01 G40 X... Y... ;
```

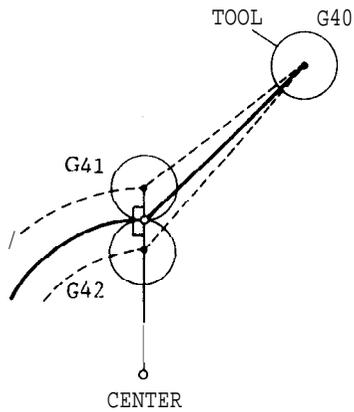
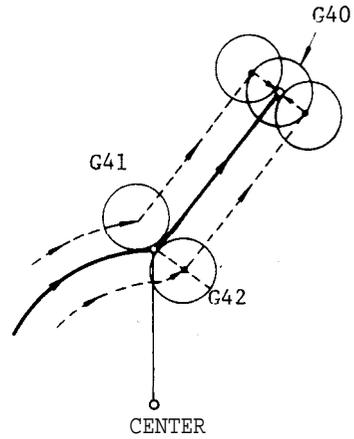


Fig. 2.44

EXAMPLE B

(c) G41 (G42)

```
G02 X... Y... I... J... ;
G01 X... Y... ;
G40 ;
```



(d) G41 (G42)

```
G02 X... Y... I... J... ;
G01 G40 ;
```

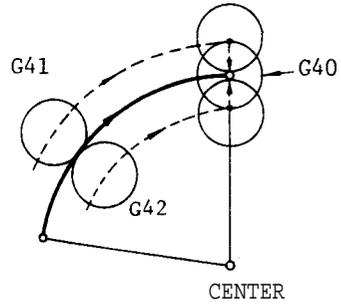


Fig. 2.45

In all cases (a) through (d) 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.

f. Cautions and remarks in tool radius compensation C

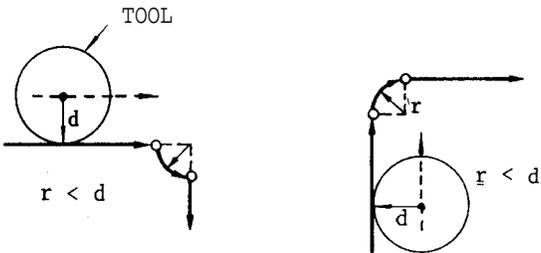
A. Maximum programmable dimensions (Refer to Table 2. 14) is not changed even in tool radius compensation C.

B. Programmed shapes that produce input errors

Input error " 045" occurs with the following programmed shapes.

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

Programmed arc radius $r + 5$ (setting unit)
 \leq tool radius d
 5: 0.001 (in mm)
 : 0.0001 (in inches)



(a) Outside compensation error (b) Inside compensation error

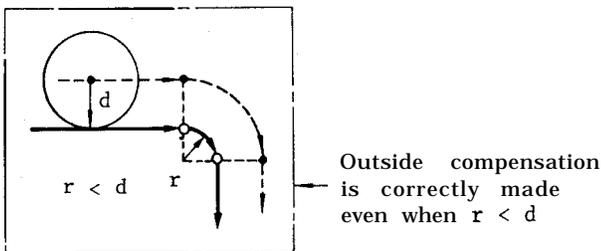
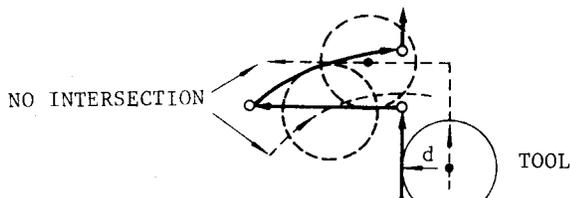


Fig. 2.46

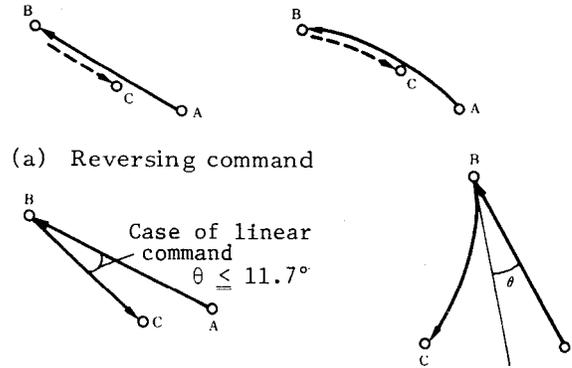
(2) When no intersection point exists on the locus of the offset tool center.



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

Fig. 2.47

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



(a) Reversing command

(b) Command close to reversing
 Case of linear command $\theta \leq 11.7^\circ$

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

Fig. 2.48

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

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

	Prohibited G codes
G codes causing input errors	G12, G13 (G17 to G19) G28, G29, G31 G73, G74, G76, G77 G81 to G89 G92

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

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

2.9.19 TOOL RADIUS COMPENSATION C
(G40, G41, G42) (Cent'd)

- 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), and canned cycles (G73, G74, G76, G77, G80 to G89) in the tool radius compensation cancel mode. Circle cutting and helical cutting incorporate tool radius compensation functions in themselves. Input error "024" occurs when they are programmed in compensation mode.
- H. Tool radius compensation C is also possible on circular interpolation by radius designation.
- I. Subprogram (M98, M99) can be programmed in 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 maximum) is programmed in compensation mode.

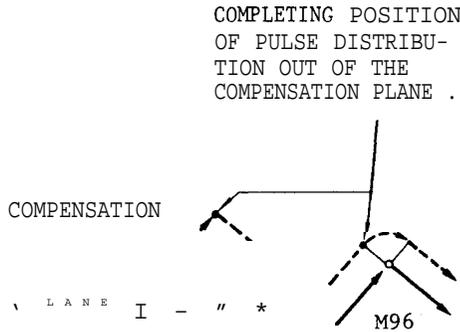


Fig. 2.49

- K. Input error "046" occurs when circular interpolation is programmed out of the plane designated by G17, G18 or G19.
- L. Offset position may be temporarily modified by programming a dummy block using addresses I, J, K.

(G42)

```
N100 GO1 X... Y... ;
N101 I... J... ;
N102 X...
```

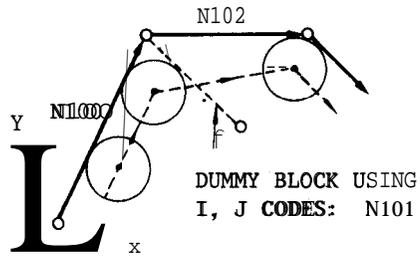
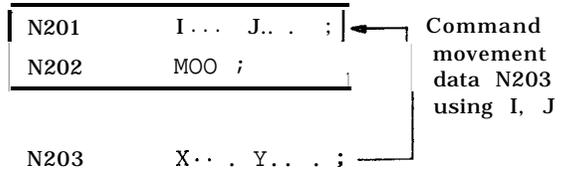


Fig. 2.50

- M. Advance reading of blocks is prohibited when MOO, 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 MOO, M01 to avoid interruption.

(G41)

```
N200 GO1 X... Y... ;
```



- N. Up to 99 radius values can be stored in the offset memory in total for the tool radius compensation, together with the values for other compensation. Make designation by a D code. The maximum programmable value of tool radius compensation is ±999.999 mm (or ±99.9999 inch).
- 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 G07 mode, it is better than overcut w

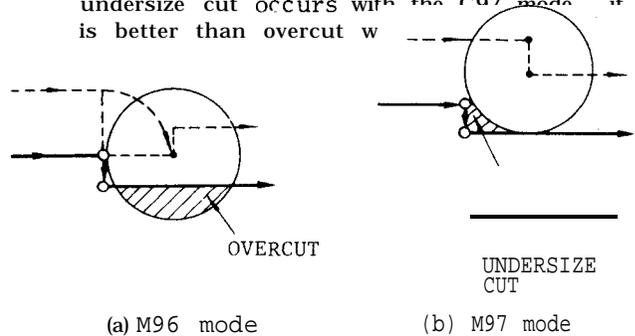
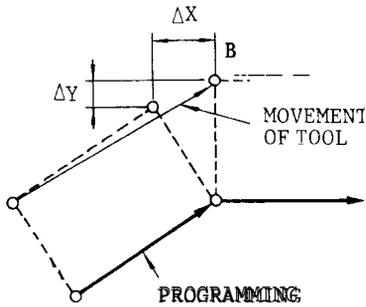


Fig. 2.51

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 (#6230).



$$\left. \begin{array}{l} \Delta X \leq Y \\ \Delta Y \leq Y \end{array} \right\}$$

Y y: Set value of parameter #6230



Fig. 2.52

10. Intervention of MD I operation in compensation mode

MDI operation can not be intervened in compensation mode.

11. Intervention to active buffer in compensation mode

The data given below can be programmed in 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 commands of the active buffer just executed

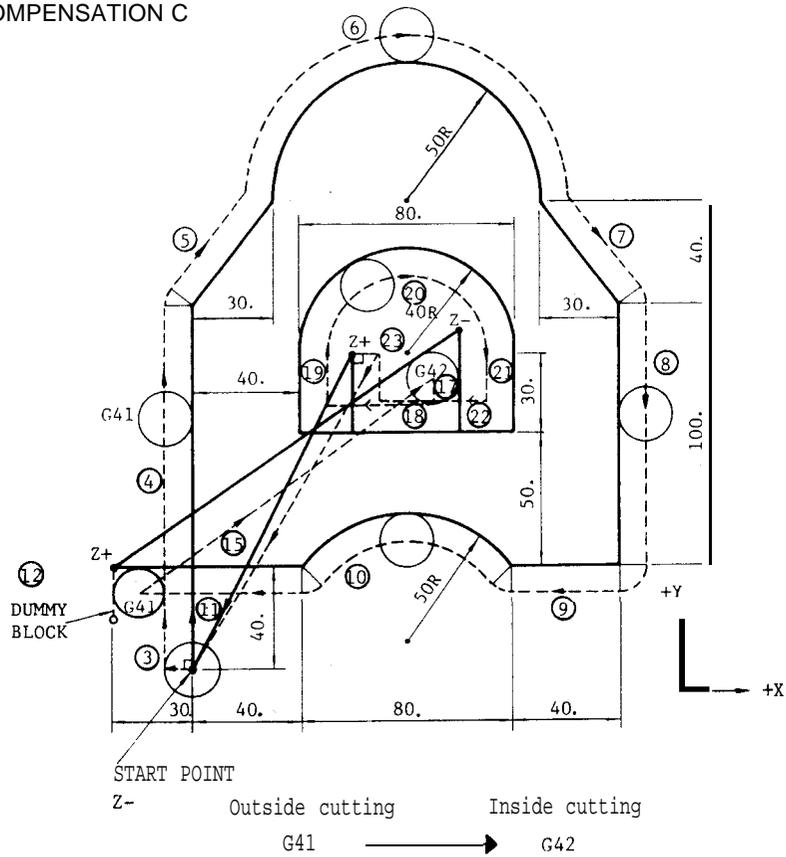
When the CYCLE START button is pushed in RAPID or JOG mode after programming, the commands are immediately executed and signals such as BCD output are sent out. Automatic operation can be resumed when CYCLE START is made after returning to the original automatic operation mode.

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

MOO, M01, M02, M30, M90 to M199

2.9.19 TOOL RADIUS COMPENSATION C
(G40, G41, G42) † (Cont'd)

EXAMPLE A

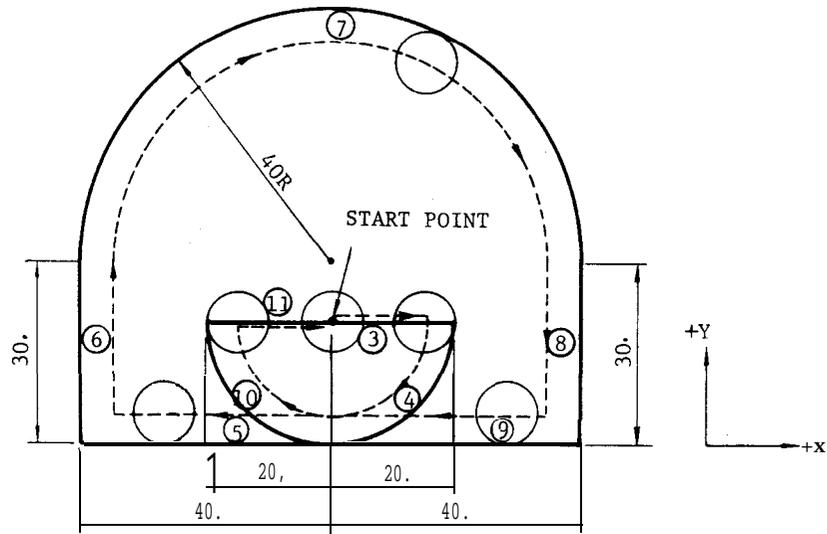


- | | | | |
|--------------------------|--|-----------------------------------|---|
| ① G91 G01 Z-25. ; F150 ; | - Incremental; Z axis lowering | ⑫ J-20. ; | - Dummy block (for modifying offset position) |
| ② G17 (G01) F300 ; | - XY plane designation, feed command | ⑬ Z25. ; | - Z axis elevation |
| ③ G41 D21 ; | - Tool radius compensation start command with tool offset No. 21 | ⑭ MO1 ; | - Optional stop |
| ④ Y140. ; | - Offset to a point on the normal line of start point of this block. | ⑮ G42 (G01) X130. Y90. F2000 ; | - Switching of direction of compensation (left + right) |
| ⑤ X30. Y40. ; | | ⑯ Z-25. F150. ; | - Z axis lowering |
| ⑥ G02 X100. 150. ; | | ⑰ Y-40. F300 ; | |
| ⑦ G01 X30. Y-40. ; | Outer Cutting | ⑱ X-60. ; | |
| ⑧ Y-100. ; | | ⑲ Y30. ; | |
| ⑨ X-40. ; | | ⑳ G02 X80. 140. ; | Inner Cutting |
| ⑩ G03 X-80. R50. ; | -R designation circular arc | ㉑ G01 Y-30. ; | |
| ⑪ G01 X-70. ; | | ㉒ X-60. ; | |
| | | ㉓ Y30. ; | - Offset to a point on the normal line of end point of this block |
| | | ⑳ 225. ; | - Z axis elevation |
| | | ㉔ G40 (G01) X-60. Y-120. FF2000 ; | 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. 2.53

EXAMPLE B



(G40)

- ① G91 G01 z-25. F150 ;
- ② G17 F300 ;
- ③ G42 D20 X20. ;
- ④ G02 X-20. Y-20. I-20. ;
- ⑤ G01 X-40. ;
- ⑥ Y30. ;
- ⑦ G02 x80. 140. ;
- ⑧ G01 Y-30. ;
- ⑨ X-40. ;
- ⑩ G02 X-20. Y20. J20. ;
- ⑪ G40 G01 X20. ;
- ⑫ Z25. ;

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

Fig. 2.54

2.9.20 TOOL LENGTH COMPENSATION
(G43, G44, G49)†

The tool length compensation function is for adding or subtracting the stored tool offset values to the Z-axis coordinate instruction values for the purpose of compensating for the deviations in tool length,

G codes for tool length compensation

G code	Group	Meaning
G43	08	(+) direction
G44	08	(-) direction
G49	08	cancel

G43 and G44 are modal functions, remaining effective when once commanded until cancellation by G49.

G49 cancels tool length compensation effects.

H00 also cancels tool length compensation effects.

The tool length compensation function is programmed in the following format.

A. (G01)

G43(G44) Z... H... ;

With this command, the tool moves towards the Z- coordinate position which is the sum of (or difference between) the H value and the Z value. As the result, the tool point is displaced from the specified Z-coordinate position by the distance specified by the H code.

B. (G01) Z... ;

G43(G44) H... ;

With this command, the tool is shifted by the distance specified by the G code.

c. G43(G44) Z... H... ;

H... i... (2)

With the command (2) , the tool is shifted by the difference between the previous tool offset value and the new tool offset value.

When G43, G44 and G49 are to be commanded, the accompanying 01 group G codes must be G00, G01 or G60. When G02 or G03 is used, this is regarded as an error.

Direction of shift

The direction of tool shift is determined by the sign of tool offset value as programmed in the H code and by the G code used.

	Sign of tool offset value	
	Positive	Negative
G43	Plus direction	Minus direction
G44	Minus direction	Plus direction

EXAMPLE

```

H10 . . . Offset value: -3.0
H11 . . . Offset value: 4.0

          CRT display including offset value
          (Z-direction only)

N101 G92 Zo ;                0.000
N102 G90 GOO X1. O Y2. O ;   0.000
N103 G43 Z-20. H10 ;         -23.000
N104 GO1 Z-30. F1000 ;      -33.000
N105 GOO ZO H00 ;           0.000

N201 GOO X-2. O Y-2. O ;
N202 G44 Z-30. H11 ;         -34.000
N203 GO1 Z-40. F1000 ;      -44.000
N204 GOO ZO H00 ;           0.000
    
```

Notes :

When the tool offset value is changed by the MDI function while programs in the offset mode is in execution, the change is effective from the block containing D code.

The tool position offset function or the tool radius compensation function is effective on the tool which is already offset by the tool length compensation function.

G43, G44 and G49 can be programmed in canned cycles. If they are programmed, this is regarded as an input error.

When a G92 command involving the Z- axis is given during the execution of a program in the tool length compensation mode, the tool length compensation is canceled. In principle, when G92 is to be programmed, the existing tool length compensation mode should first be canceled.

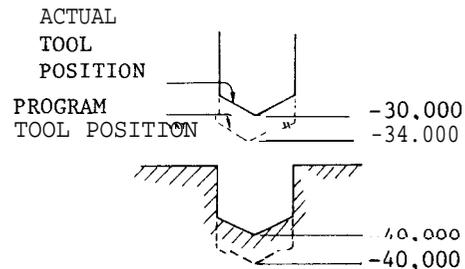
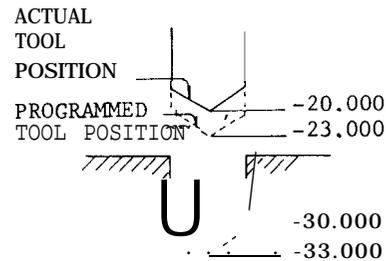
During the automatic execution of a program in the tool length compensation mode, the number of the effective tool compensation memory (H code number) can be displayed. For details, refer to 4.3.2, "COMMAND DATA DISPLAY " G43, G44, and G49 commands must be given in GOO or GO1 mode. When G43 is commanded in G02 or G03 mode, no alarm is triggered but correct motion may not be made.

2.9.21 TOOL POSITION OFFSET (G45 TO G48)

Tool position offset 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

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

2.9.21 TOOL POSITION OFFSET (G45 TO G48) (Cent'd)

3. Make program command by incremental designation (G91) for 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 GOO G46 X...Y...D01 ; . . . Reduction
② GO1 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 GOO G46 X...Y... (D01) ; . . . Reduction
    
```

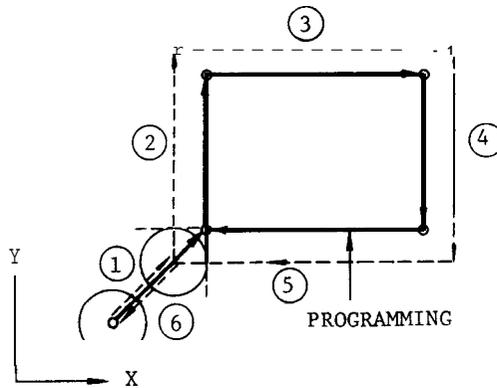


Fig. 2.55

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.

	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 GOO G47 x60. D10 ; D10 = 20.
```

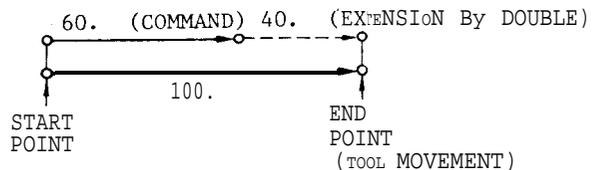


Fig. 2.56

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 a command same as that described above :

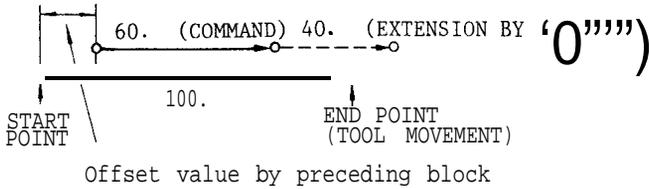


Fig. 2.57

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 X10. D10 ; D10 = 20.
```

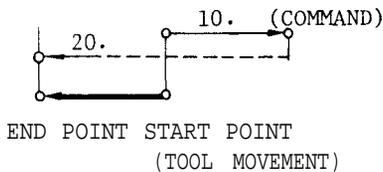


Fig. 2.58

7. The above applies to X- and Y-axis, but G45 to G48 may also be programmed to Z-axis in the same manner.

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, tool radius compensation is possible with 1/4 circle, 3/4 or full circle.

```
G91
```

```
G45 G02 X50. Y50. 150. D10 ;  
D10 = 20.
```

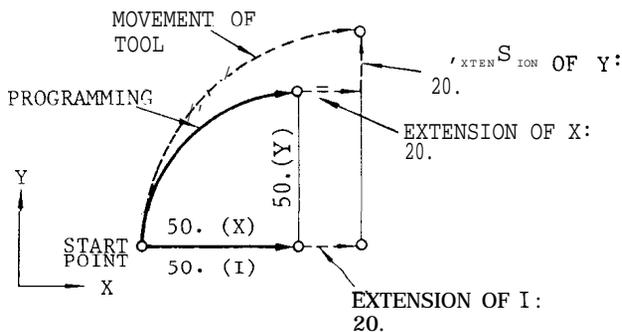
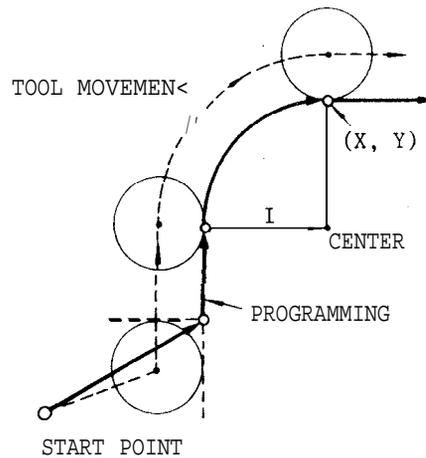


Fig. 2.59

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... ;
```



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

Fig. 2.60

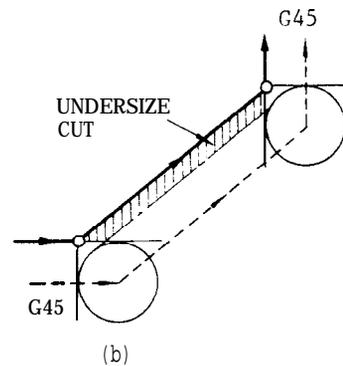
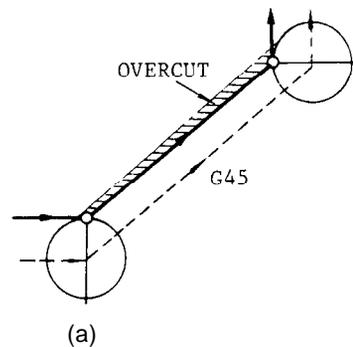


Fig. 2.61

2.9.21 TOOL POSITION OFFSET (G45 TO G48) (Cent'd)

9. When programming G45 to G48, the G code of 01 group can be given together in the same block. An input error occurs if command is given with other G codes.
10. When only movement by offset in the incremental designation (G91) is required, program "O" as the axis move command.

```
G91 G01 G45 XO 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 XO 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 "O. "

11. H code or D code can be selected by parameter setting of the tool position offset (G45 to G48), offset number command.

H code or D code can be selected for offset number command, by parameter setting.

Setting parameter

```
#6073 D2  
= 1 : H code only is effective  
= 0 : D code only is effective
```

Notes :

When G45 to G48 are programmed as the simultaneous move command along two axes, extension or reduction is made in the two axes. Overcut or under-size cut will occur if this is applied to cutting. Keep this in mind.

Even when the offset value is changed by MDI, the offset command previously programmed will not be affected. It becomes operable when G45 to G48 are programmed thereafter.

This tool position offset can be applied in addition to the tool length offset.

Mirror image can be applied to tool position offset. That is, it is possible to perform symmetrical cutting with this offset applied.

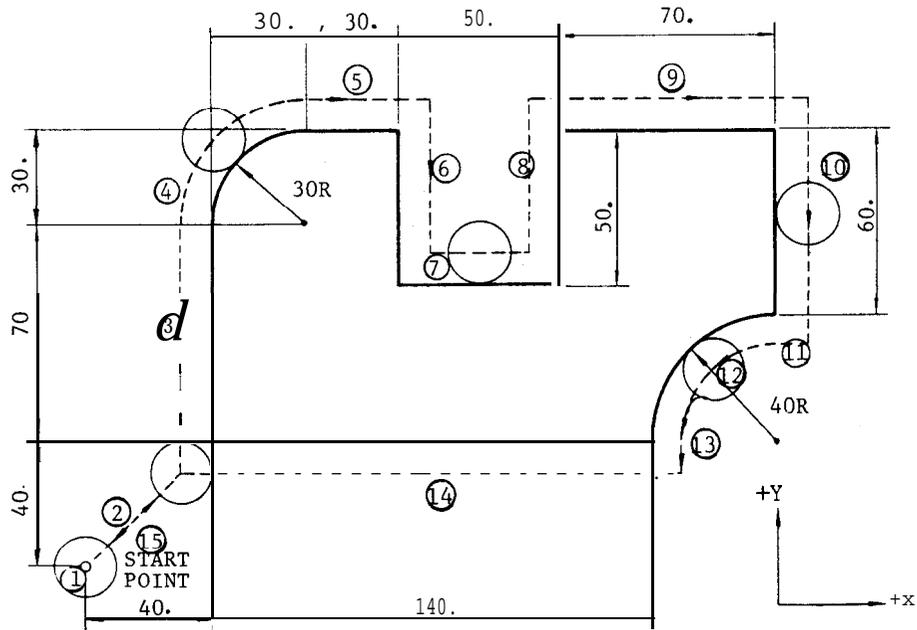
Tool position offset is independent of G codes (G17/G18/G19) of plane designation.

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 coordinate system is made after the offset value is canceled from the designated axes. 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 can be displayed. Refer to 4.3.2.3, "DISPLAY OF TOOL OFFSET STATE: COMMAND (OFFSET) ."

EXAMPLE A

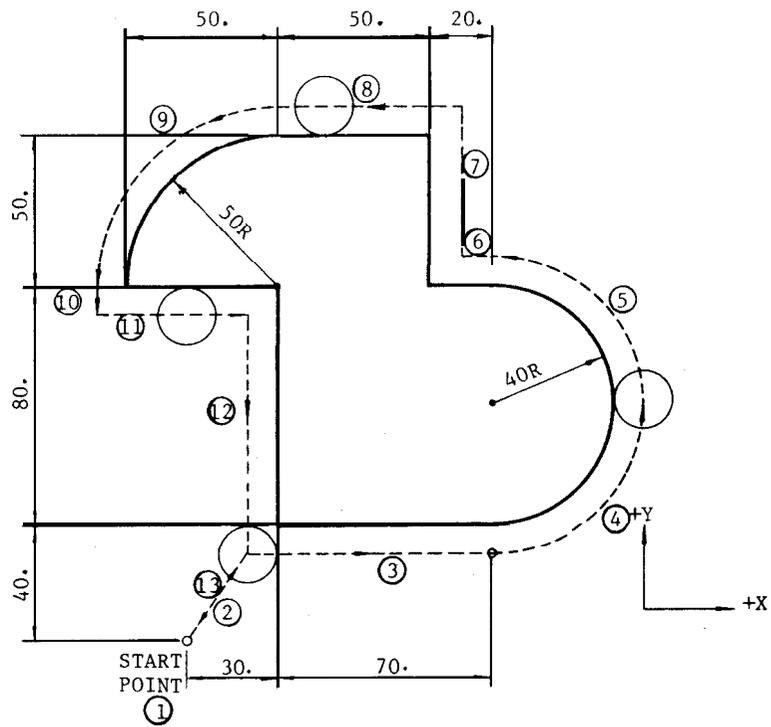


- ① G91 G01 Z-25. F150 ;
- ② G46 x40. Y40. D10 F300 ;
- ③ G45 Y70. ;
- ④ G45 G02 X30. Y30. 130. ;
- ⑤ G45 G01 X30. ;
- ⑥ Y-50. ;
- ⑦ G48 x50. ;
- ⑧ Y50. ;
- ⑨ G47 X70. ;
- ⑩ G47 Y-60. ;
- ⑪ G46 XO ;
- ⑫ G46 G03 x-40. Y-40. J-40. ;
- ⑬ G46 G01 YO ;
- ⑭ G47 X-140. ;
- ⑮ G46 x-40. Y-40. ;
- ⑯ z25. ;

Fig. 2.62

2.9.21 TOOL POSITION OFFSET (G45 TO G48) (Cent'd)

EXAMPLE B



- ① G91 G01 F300 ;
- ② G46 X 30. Y40. D15 ;
- ③ G45 X70.
- ④ G45 G03 X40. Y40. J40. ;
- ⑤ G45 G03 X-40. Y40. I-40. ;
- ⑥ G46 G01 X-20. ;
- ⑦ Y50. ;
- ⑧ G45 X-50. ;
- ⑨ G45 G03 X-50. Y-50. J-50. ;
- ⑩ G46 G01 Y0 ;
- ⑪ X50. ;
- ⑫ Y-80. ;
- ⑬ G46 x-30. Y-40. ;

Fig. 2.63

2.9.22 SCALING FUNCTION (G50, G51) †

With this function, workpiece contours programmed by part programs can be enlarged or reduced at any desired scale.

The following G codes are used for this function.

G code	Group	Meaning
G50	15	Scaling OFF
G51	15	Scaling ON

Note: When power is applied or the control is reset, the control is in the state of G code marked with ▽.

G51 I.. . J.. . K.. . P.. . ;

With this command, the program is executed on an enlarged or reduced scale with the scale ratio specified by P, and the center of scaling specified by I, J, and K.

G50; command cancels the scaling mode.

The enlarging and reducing scales can be selected within the following range.

Enlarging and reducing range	0.000001-99.9999991
------------------------------	---------------------

Reference unit for P is : 1 = 0.000001.

When P command includes decimal number, numbers after decimal point are regarded as six-digit numbers.

Example

```
P0. 999999      0.999999 time
P2.0            2 times
P2              0.000002 time
```

When P (designating multiplication) is omitted, multiplication is determined by setting #6500 and #6501.

$$\text{Multiplication} = \frac{\#6500}{\#6501}$$

Example

Where setting #6500 = 3, #6501 = 100

$$\text{Multiplication factor} = \frac{3}{100} = 0.03 \text{ times}$$

Multiplication should not exceed the enlarging and reducing range.

When I, J, or K is programmed in the G51 command, scaling functions on the axis designated: I... X-axis, J... Y-axis, K... Z-axis.

Scaling will work only on the axis selected by I, J, or K.

Example

G51 I100 JO P0.8

With this command, scaling will work on X- and Y-axis and not on Z-axis.

Where the work coordinate system is specified, I, J, and K in the G51 block designates the distance between coordinate system origin and scaling center.

Example

```
N1 G92 XO YO ;
N2 G00 G90 X200. Y100. ;
N3 G51 I200. J200. ;
```

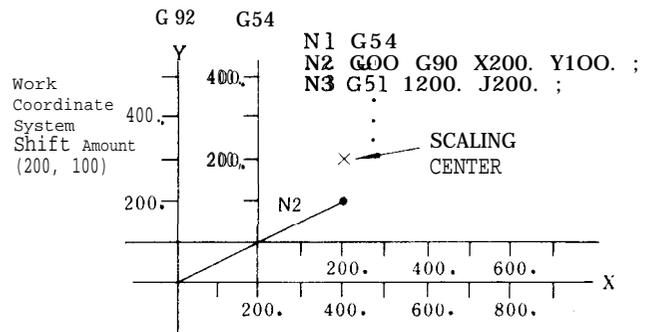
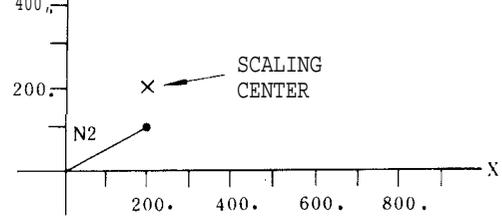


Fig. 2.64

Notes :

Scaling is turned on when approaching for usual machining and off after retraction on completion of approaching. Turning off and on scaling during machining will not form the correct contour.

Scaling is executed on the two axes on machining plane. If scaling is executed on a single axis, an alarm occurs at circular command because scaling cannot work according to circular command.

Block commands G51I...J...K...P...; and G50; should be programmed independently. If X, Y and Z commands coincide in the same block, an alarm will occur.

When the scale ratio of one or more is programmed, the resultant command value should not exceed the maximum.

Scale ratio 0 cannot be commanded. If commanded, an alarm will occur.

Scaling is not effective on compensation value.

Canned cycles cannot be executed with scaling commanded on Z-axis. If scaling is commanded on Z-axis during canned cycle execution, an alarm will occur.

When operation is reset (reset pushbutton, M02, M30 command), scaling is turned off (G50).

2.9.22 SCALING FUNCTION (G50, G51) † (Cont'd)

Display of command and position will show the values of command and position after scaling is finished.

The following G codes cannot be commanded during scaling. If commanded, an alarm will occur.

G28, G29, G30, G31, G36, G37, G38, G53, G92

Scaling (G51) command cannot be given during tool radius compensation C.

Alarm codes for scaling are listed below.

Table 2.23 Alarm Codes for Scaling

Alarm Code	Causes
050	G50 and G51 blocks not commanded independently.
051	<ul style="list-style-type: none"> • Multiplication factor set as 0. • Unusable G code commanded during scaling Scaling function working on Z-axis during canned cycle. • Scaling (G51) commanded during compensation C.

- Scaling should always be commanded by G51 and G50 as a pair. Commanding G 51 during scaling mode is ignored.

The setting number for setting the shift amounts for G codes from G54 to G59 are as follows.

Table 2.24 Work Coordinate System Setting A

G code	Coordinate system	X	Y	Z	α
G54	Work coordinate system 1	#6516	#6517	#6518	#6519
G55	Work coordinate system 2	#6522	#6523	#6524	#6525
G56	Work coordinate system 3	#6528	#6529	#6530	#6531
G57	Work coordinate system 4	#6534	#6535	#6536	#6537
G58	Work coordinate system 5	#6540	#6541	#6542	#6543
G59	Work coordinate system 6	#6546	#6547	#6548	#6549

Setting up work coordinate systems (G54 to G59)

G54 (G55, G56, G57, G58 or G59);

When this command is given, from that time on, the tool will be controlled by the work coordinate system.

2.9.23 WORK COORDINATE SYSTEM SETTING A (G52 TO G59) †

Six types of work coordinate systems corresponding to six G codes, G54 through G59, are available for selective use.

There are three types of coordinate systems as follows .

A. Basic coordinate system

This is the basic coordinate system to be set up by G92, by the ORIGIN key, or by the automatic coordinate system setting function. When the power supply is turned on, until any of these actions will be made, the tool position at the time of turning on is treated as the temporary coordinate origin.

B . Work coordinate system

When any of the G codes G54 through G59 is commanded, a coordinate system with the origin shifted by the amount set by the setting numbers corresponding to that G code is set up. The coordinate systems set up by these G codes are referred to as work coordinate systems, and when a work coordinate system is set up, the tool will be controlled to it. Since there are six G codes for work coordinate systems, up to six work coordinate systems can be used.

C. Machine coordinate system

This is a coordinate system which is fixed to the machine, and is set up when the tool is returned to the reference point. This coordinate system has its (0, 0, 0) point at the reference point.

Returning to basic coordinate system (G52)

G52 ;

With this command, the currently effective work coordinate system is cancelled, and the basic coordinate system becomes effective again .

EXAMPLE

```
N1 G90 X100. Y200. ;
N2 G54 ;
N3 X100. Y300. ;
N4 X300. Y200. ;
N5 G52 ;
N6 Xo Yo ;
```

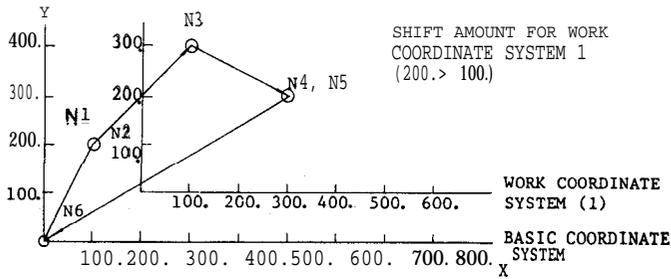


Fig. 2.65

Temporary shift to positions on machine coordinate system

```
(G90) G53 X... Y... Z... ;
```

With this command, the tool is shifted to the position (X, Y, Z) on the machine coordinate system only in this block. G53 is a non-modal G code.

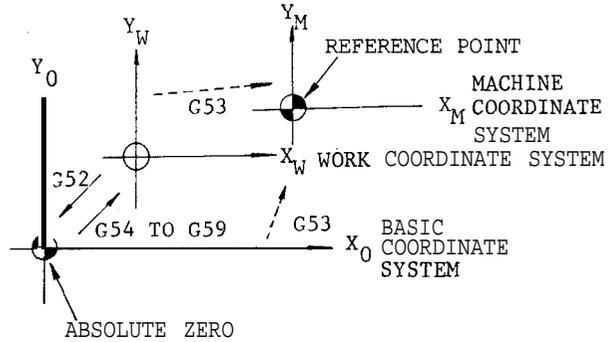


Fig. 2.66

EXAMPLE (Reference program)

```
N1 G92 X200. Y100. ;
N3 G54 G90 X100. Y200. ;
N4 G53 X300. Y100. ;
N5 X300. Yo ;
N6 G52 ;
N7 Xo Yo ;
```

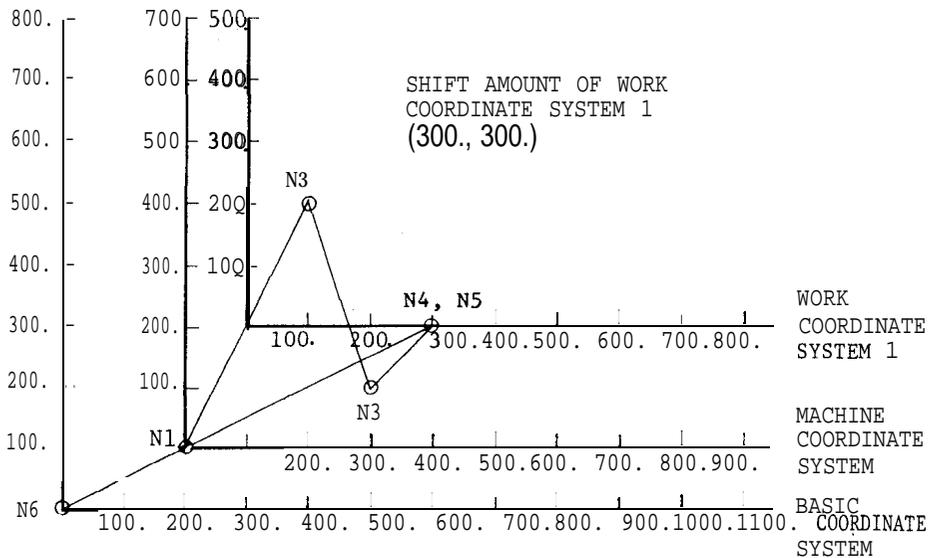


Fig. 2.67

2.9.23 WORK COORDINATE SYSTEM SETTING A (G52 TO G59) '(Cent'd)

Notes :

The shift amounts for work coordinate systems can be specified by programs with G10 commands, in addition to the MD I writing. For details, refer to 2.9.8. "TOOL OFFSET VALUE DESIGNATION (G10)."

Work coordinate systems set up by G54 through G59 are canceled by the G52 command, and the basic coordinate system becomes effective again.

When a work coordinate system has been set up by any of the commands G54 through G59, the selected shift amounts can not be changed even when they are rewritten.

The rewritten shift amounts will become effective when a new work coordinate system command is executed.

G53 commands should only be given under the following conditions. If these conditions are not satisfied, the commands are regarded as errors.

- (1) The mirror image function is not used.
- (2) No canned cycle is in use and no tool compensation C is in use.
- (3) If a 01 group G code is used, it is G00, G01 or G60 and nothing else.

If a G53 command is executed with the machine lock function on, the current value displayed changes sequentially until the command value corresponding to the machine lock function off state will be displayed. If the machine lock function is switched on and off during the execution of G53 blocks, correct positioning can not be achieved.

However, when a complete G53 block is executed with the machine lock function off, correct positioning is achieved as programmed, even when the machine lock function is switched on and off before that block.

G53 commands should be given in the G 90 mode. If they are given in the G91 mode, the command values are regarded as G 90 mode values.

When work coordinate systems are to be changed with any one of the G54 through G59 commands, the program should be written so that a new coordinate system will be set up in the G90 mode and the basic coordinate system will be reset in the G90 mode.

If a G53 command is given while the tool length compensation or tool position offset function is on, the tool offset value is deleted temporarily. Generally, when giving a G53 command, the tool length compensation and tool position offset commands should be canceled in advance.

If a G 53 command is given while the tool position offset command is ON, subsequent programs will be as follows :

- (1) If subsequent programs are given in incremental mode, it will cause the tool to move by incremental amount given from reference point as a command. (Offset amount not considered.)
- (2) If subsequent programs are given in absolute mode, it will cause the tool to move by "absolute position plus offset amount" given as a command.

When any one of the commands G54 through G59 is given while the tool length compensation or tool position, the compensation remains effective. Generally, when any one of the commands G54 through G59 is to be given, the tool length compensation or tool position offset command should be canceled in advance.

```

G43 Z0 H01 ; ... COM Z100.
G54 ;
G90 Z1000 ; ... COM # 100,

```

Shift amount for Z axis is 1300.

G54 shift: Z = 300.

Offset: H01 = 100.

- If G92 is given during execution on the work coordinate system set up by G54 through G59, G54 through G59 or the basic coordinate system is shifted so that the current position is to be a shifted position by G92, G92 should not be used in G54 to G59 modes in general.

2.9.24 WORK COORDINATE SYSTEM SETTING B (G52 TO G59)'

- (1) Outline of work coordinate system-setting expansion

Up to 30 types of work coordinate systems can be set with the expansion of the work coordinate systems of specification A (6 types) by using commands J1 to J5 at the same time as G54 to G59. The expanded area will be set by #6700 -#6771.

The four axes X, Y, Z, α , β , can be set since G54 to G59 and G54 (J 1) to G59 (J 1) are the same. In commands J2 to J5, only three axes X, Y, Z can be set but the 4th axis cannot be set.

- (2) Setting numbers for specifying work coordinate system shift

Table 2.25 Work Coordinate System Setting B

	x	Y	z	α
G54(J1)	#6516	#6517	#6518	#6519
J2	#6700	# 6701	# 6702	
J3	#6718	#6719	#6720	
J4	#6736	#6737	# 6738	
J5	#6754	#6755	#6756	
G55(J1)	#6522	#6523	# 6524	#6525
J2	#6703	# 6704	#6705	
J3	#6721	# 6722	#6723	
J4	#6739	# 6740	# 6741	
J5	#6757	#6758	# 6759	
G56(J1)	#6528	#6529	#6530	#6531
J2	#6706	#6707	#6708	
J3	# 6724	#6725	#6726	
J4	# 6742	#6743	# 6744	
J5	#6760	# 6761	#6762	
G57(J1)	#6534	#6535	# 6536	#6537
J2	# 6709	#6710	#6711	
J3	#6727	#6728	# 6729	
J4	#6745	#6746	# 6747	
J5	#6763	#6764	# 6765	
G58(J1)	#6540	# 6541	# 6542	#6543
J2	#6712	#6713	#6714	
J3	#6730	#6731	#6732	
J4	# 6748	#6749	#6750	
J5	#6766	#6767	# 6768	
G59(J1)	#6546	#6547	#6548	#6549
J2	#6715	#6716	#6717	
J3	#6733	#6734	#6735	
J4	#6751	#6752	#6753	
J5	#6769	#6770	#6771	

(3) Setting comment display

For work coordinate system setting, comments will be displayed as shown below. This display will be possible if the 4th axis of G54 (J 1) to G59 (J1) is provided.

Setting Comment Display

SETTING	01234	N1234
#6520	-12345678	(G54 β)
#6521	o	
#6522	o	(G55 X)
#6523	o	(G55 Y)
#6524	0	(G55 Z)
#6525	0	(G55 α)
#6526	0	
#6527	0	
#6528	0	(G56 X)
#6529	12345678	(G56 Y)
		LSK RDY

Selection of 4th axis displays any one of A, B, C, U, V, or W axis.

(4) Work coordinate system setting

(a) G54 Jn; (n=1, 2, 3, 4, 5)

With this command, subsequent programs will cause the tool to move along the work coordinate system. G 54 to G59 are modal commands.

(b) The G54; command and G54 J1; command are equivalent.

(c) If a command is a numerical value above J6, alarm " 12911 will occur.

(d) Durtng the J2 to J5 commands, the amount of shift on the 4th axis will be 0 and the basic coordinate system will become effective again.

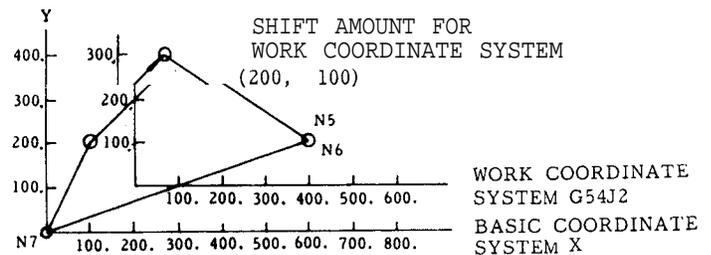
(5) Returning to the basic coordinate system (G52)

(a) G52;

With this command it will be possible to cancel the currently selected work coordinate and return to the basic work coordinate system.

(b) G52 is a modal command.

(6) EXAMPLE



```

N1 G92 XO YO;
N2 G90 G00 X100. Y200. ;
N3 G54J2;
N4 X200. Y300;
N5 x400. Y100;
N6 G52;
N7 XO YO;
    
```

Work Coordinate Svstem Shift Amount	
#6700	200.
#6701	100.

(7) Temporary shift on the machine coordinate system (G53)

Similar to Specification A, G53 can command a temporary shift on the machine coordinate system with the G 53 command. G53 is a non-modal G code.

Example: G53(G90)G00 X.. .Y. .. Z.. .(a ...);

(8) Changing work coordinate systems with G 10

(a) Similar to Specification A, G 10 command corrects the work coordinate system from the program .

G10 Q2 Pm Jn X... Y...Z ... (a...);

This command corrects the specified work coordinate system to be corrected can be specified by the combination Pm Jn.

2.9.24 WORK COORDINATE SYSTEM SETTING B
(G52 TO G59)† (Cent'd)

Pm selects commands from G54 to G59.

P1 = G54
to to
P6 = G59

Jn selects any of J1 to J5.

Example: P2J3, G55J3

P4J2 G57J2

(b) Omission of J or specification of JO will be considered equivalent to J1.

(c) Error "038" will occur if a wrong numerical value is specified for m or n.

(d) Error "038" will occur if the 4th axis is commanded when executing commands J2 to J5.

(9) Work coordinate system shifting in user macro

(a) The amount of work coordinate shifting (and external work coordinate system corrections) can be read by using a system variable for the right element of an arithmetic expression.

(b) The values can be changed by using the above system variable as the left element.

(c) The correspondence between the system variable and the amount of work coordinate shift will appear as listed in the following table. Y-axis of #2600's, the Z-axis of #2700's the α axes of #2800's have the same correspondence as the X-axis. Command G54 to G59 (equivalent to J 1) only in the case of α axis.

Table 2.26 (a)

System Variables	Work Coordinate System Shift Amount	Axis
#2500	External work coordinate system correction amount	x
#2501	G54 (J1)	
#2506	G59 (J1)	
#2511	G54 J2	
#2516	G59 J2	
#2521	G54 J3	
#2526	G59 J3	
#2531	G54 J4	
#2536	G54 J4	
#2541	G54 J5	
#2546	G59 J5	

Table 2.26 (b)

System Variables	Work Coordinate System Shift Amount	Axis
#2600	External work coordinate system correction amount	Y
#2601	G54 (J1)	
#2606	G59 (J1)	
#2611	G54 J2	
#2616	G59 J2	
#2621	G54 J3	
#2626	G59 J3	
#2631	G54 J4	
#2636	G54 J5	
#2641	G54 J5	
#2646	G59 J5	

Table 2.26 (c)

System Variables	Work Coordinate Shift Amount System	Axis
#2700	External work coordinate system correction amount	z
#2701	G54 (J1)	
#2706	G59 (J1)	
#2711	G54 J2	
#2716	G59 J2	
#2721	G54 J3	
#2726	G59 J3	
#2731	G54 J4	
#2736	G59 J4	
#2741	G54 J5	
#2746	G59 J5	

Table 2.26 (d)

System Variables	Work Coordinate Shift Amount System	Axis
#2800	External work coordinate system correction amount	α
S2801	G54 (J1)	
#2806	G59 (J1)	

(d) Program examples

(i) #116 = #2501 ;

Command G54 (J 1) assigns the amount of work coordinate system shift of X-axis as the common variable #116.

(ii) #2511 = #4

Command G54 (J2) cancels the amount of work coordinate system shift of X-axis and sets the contents of local variable #4.

(10) Correction of the work coordinate system shift by an external input

(a) The amount of work coordinate system shift can be corrected by an external data input signal.

(b) The new amount of shift can be obtained by inputting the amount of axis correction from the outside and adding this to all of the shift values from G54 (J1) to G59 (J5).

For example, inputting the amount of shift for the X-axis will correct X-axis shift amounts for all commands from G54 (J1) to G59 (J5).

(11) Notes and remarks

(a) The notes in Specification A are also applicable to Specification B.

(b) Use the G54 to G59 ; commands in the GOO or GO1 mode. Alarm "129" will occur if used in any other mode.

Table 2.26 (e)

	x	Y	z	α	R
G54 (J1)	#6516	#6517	#6518	#6519	#6521
J2		#670 1	#6702	#6703	#6705
J3	#6736	#6737	#6738	#6739	#6741
J4	#6772	#6773	#6774	#6775	#6777
J5	#6808	#6809	#6810	#6811	#6813
G55 (J1)	#6522	#6523	#6524	#6525	#6527
J2	#6706	#6707	#6708	#6709	#67 11
J3	#6742	#6743	#6744	#6745	#6747
J4	#6778	#6779	#6780	#6781	#6783
J5	#6814	#6815	#6816	#6817	#6819
G56 (J1)	#6528	#6529	#6530	#6531	#6533
J2	#6712	#6713	#6714	#6715	#6717
J3	#6748	#6749	#6750	#6751	#6753
J4		#6785	#6786	#6787	#6789
J5	#6820	#682 1	#6822	#6823	#6825
G57 (J1)		#6535	#6536	#6537	#6539
J2	#6718	#6719	#6720	#672 1	#6723
J3	#6754	#6755	#6756	#6757	#6759
J4		#6791	#6792	#6793	#6795
J5	#6826	#6827	#6828	#6829	#6831
G58 (J1)	#6540	#654 1	#6542	#6543	#6545
J2	#6724	#6725	#6726	#6727	#6729
J3	#6760	#676 1	#6762	#6763	#6765
J4	#6796	#6797	#6798	#6799	#680 1
J5	#6832	#6833	#6834	#6835	#6837
G59 (J1)	#6546	#6547	#6548	#6549	#6551
J2	#6730	#673 1	#6732	#6733	#6735
J3	#6766	#6767	#6768	#6769	#677 1
J4	#6802	#6803	#6804	#6805	#6807
J5	#6838	#6839	#6840	#6841	

2.9.25 WORK COORDINATE SYSTEM SETTING C (G52 TO G59) †

(1) Outline of work coordinate system setting expansion function

The rotary angle can be set in the work coordinate system setting when commanding G54 to G59, to perform the coordinate system rotation at the same time. The J2 to J5 commands of G54 to G59 are also expanded to allow setting of the 4th axis.

(2) The item on the rotary angle is added to the work coordinate system setting. The 4th axis can be set to all the work coordinate systems at the same time. See Table 2.26 (e) for the setting numbers.

(3) Setting command display

SETTING		
	0 1234 N 1234	
#6520	-12345678	(G54 β)
#6521	3000	(G54 R)
# 6522	0	(G 55 X)
# 6523	0	(G 55 Y)
#6524	0	(G 55 Z)
#6525	0	(G 55 α)
#6526	0	
#6527	0	
#6528	0	(G 56 X)
#6529	12345678	(G 56 Y)
		LSK RDY

A, B, C, U, V or W is displayed by the 4th axis specification.

Fig. 2.67

2.9.25 WORK COORDINATE SYSTEM SETTING C (G52 TO G59) (Cont'd)

(4) Work coordinate system setting

Setting is performed in the same way as for work coordinate system setting B. Rotation is added to move the work coordinate system.

(a) G54 Jn; (n = 1, 2, to 5)

With this command, all later programs move on the specified work coordinate system. G54 to G59 are modal commands.

(b) The G54; command and G54 J1; are equivalent.

(c) Commanding numerals exceeding J6 causes alarm "129."

(d) When J2 to J5 are commanded, the shift amount of the 4th axis becomes 0 and returns to the basic coordinate system.

(5) Returning to the basic coordinate system (G52)

(a) G52;

With this command, the currently selected work coordinate system is canceled, to return to the basic coordinate system.

(b) G52 is modal.

(6) Program example

G92 XO YO ZO;

- Nothing changes under G92.
-
-
-

G54;

- (G68 XO YO R (value of #6521))
- Shifts the work coordinate system by G54, and rotates the coordinate system for R, centering the work coordinate system (O, O).

G55;

- (G69)
- (G68 XO YO R (value of #6527))
- Shifts the work coordinate system by G54 and cancels the rotation, to create a new coordinate system by the G55 work coordinate system shift amount and rotary angle.

G52;

- (G69)
- Returns to the G92 coordinate system by G52. The rotation is also canceled.
-

M30;

() shows what the program will be like when actually programmed.

(7) Temporary move in the machine coordinate system (G53)

As in the A-specification, the move on the machine coordinate system can be temporarily commanded by the G53 command. G53 is a non-modal G code.

(Example) G53 (G90) GOO X...Y...Z... (a...) ;

(8) Work coordinate system alteration by G10

As is the work coordinate system A- and B-specifications, the work correction of the coordinate system can be commanded from the program by the G10 command.

(a) G10 Q2 Pm Jn X... Z... a... R... ;

Command as above, to correct the specified work coordinate system.

The combination of Pm and Jn specifies the coordinate system to be corrected.

Select G54 to G59 by Pm...

P1 = G44

to

P6 = G59

Select J1 to J5 by Jn...

Example P2 J3 ... G55 J3

P4 J2 ... G57 J3

(b) When J is omitted or when JO is specified, it is regarded as J1.

(c) When a wrong numeral is commanded for m or n, error "038" occurs.

(d) The 4th axis command is enabled when J2 to J5 are commanded.

(This is added to the work coordinate system shift B-specifications).

(e) R... corrects the rotary angle.

(9) Work coordinate system shift amount in the macro program

(a) The following system variables are added to the macro, by the addition of the work coordinate system shift amount.

(b) The notes and how to use the system variables are the same as the A- and B-specifications.

Table 2.26 (f)

System Variables	Work Coordinate System Shift Amount	Axis
#2500	External work coordinate system correction amount	
#2501	G54 (J1)	x
⋮	⋮	
#2506	G59 (J1)	
#2511	G54 J2	
⋮	⋮	
#2516	G59 J2	
#2521	G54 J3	
⋮	⋮	
#2526	G59 J3	
#2531	G54 J4	
⋮	⋮	
#2536	G59 J4	
#2541	G54 J5	
⋮	⋮	
#2546	G59 J5	

Table 2.26 (g)

System Variables	Work Coordinate System Shift Amount	Axis
#2600	External work coordinate system correction amount	Y
#2601	G54 (J1)	
⋮	⋮	
#2606	G59 (J1)	
#2611	G54 J2	
⋮	⋮	
#2616	G59 J2	
#2621	G54 J3	
⋮	⋮	
#2626	G59 J3	
#2631	G54 J4	
⋮	⋮	
#2636	G59 J4	
#2641	G54 J5	
⋮	⋮	
#2646	G59 J5	

Table 2.26 (i)

System Variables	Work Coordinate System Shift Amount	Axis
#2800	External work coordinate system correction amount	α
#2801	G54 (J1)	
⋮	⋮	
#2806	G59 (J1)	
#2811	G54 J2	
⋮	⋮	
#2816	G59 J2	
#2821	G54 J3	
⋮	⋮	
#2826	G59 J3	
#2831	G54 J4	
⋮	⋮	
#2836	G59 J4	
#2841	G54 J5	
⋮	⋮	
#2846	G59 J5	

Table 2.26 (h)

System Variables	Work Coordinate System Shift Amount	Axis
#2700	External work coordinate system correction amount	Z
#2701	G54 (J1)	
⋮	⋮	
#2706	G59 (J1)	
#2711	G54 J2	
⋮	⋮	
#2716	G59 J2	
#2721	G54 J3	
⋮	⋮	
#2726	G59 J3	
#2731	G54 J4	
⋮	⋮	
#2736	G59 J4	
#2741	G54 J5	
⋮	⋮	
#2746	G59 J5	

(c) Assign the following system variables to the rotary angle.

Table 2.26 (j)

System Variables	Work Coordinate System Rotary Angle	Address
#2951	G54 J1	R
⋮	⋮	
#2956	G59 J1	
#2961	G54 J2	
⋮	⋮	
#2966	G59 J2	
#2971	G54 J3	
⋮	⋮	
#2976	G59 J3	
#2981	G54 J4	
⋮	⋮	
#2986	G59 J4	
#2991	G54 J5	
⋮	⋮	
#2996	G59 J5	

2.9.25 WORK COORDINATE SYSTEM SETTING C (G52 TO G59) '(Cent'd)

(10) Correction of the work coordinate system shift amount by external input

(a) The work coordinate system shift amount can be corrected by external data input signals.

(b) The externally input axis correction amount is added to all the G54 (J1) to G59 (J5) shift amounts to make new shift amounts. The setting of the work coordinate system shift amount is not directly corrected. The work coordinate system is corrected for external work coordinate system correction amount. Shift amount on actually moving work coordinate system = External work coordinate system correction amount + work coordinate system shift amount setting.

(c) For rotary angle, the direct setting is corrected. Therefore, there is no external work coordinate system correction amount.

(11) J-number monitor during execution of work coordinate system shift

The pair addition number J can be monitored by #6195, during execution of the work coordinate system shift (G54 to G59).

No J	
J1 #6195	= 1
J2 . ? " "	= 2
J3	= 3
J4	= 4
J5	= 5

(12) Notes and remarks

(a) All the notes for the B-specifications apply to the C-specifications.

(b) Command the G54 to G59 commands under the GOO to GO1 mode. If commanded under other modes, alarm "129" occurs.

(c) G68 and G69 cannot be used in this specification. Executing G68 and G69 will cause "alarm."

(d) The coordinate system rotation by the G54 command is canceled by the G52 command or by setting O in the work coordinate system setting.

(e) The G28 /G30 commands during coordinate system rotation under the G54 command, can perform rotation at the intermediate positioning point, but not on the reference point. Coordinate system rotation is also not performed for G53.

(f) The coordinate rotation plane of this specification is fixed to the G17 plane.

2.9.26 UNIDIRECTIONAL APPROACH (G60)'

This function is effective to position the tool at high accuracy .

G60 x . . . Y . . . Z . . . (a . . .) ;

With this command, the tool moves and stops at the specified position. If the tool approaches the stop position in the direction specified by the parameter (#6014),

it overtravels the stop position by the amount specified by parameters (#6436 - #6439), and then returns to the specified position to stop.

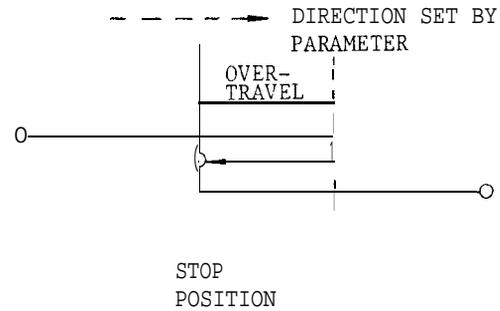


Fig. 2.68

Notes :

- . Unidirectional approach is effective in the canned cycle mode. Shift by G76 includes unidirectional approach. Cancel this function before executing G76 for safety purposes.
- . Unidirectional positioning is performed at the interim point by G28 command.
- . When GOO is commanded in a G60 block, the latter commanded G code is validated.

2.9.27 HOLE PATTERN CYCLES (G70, G71 , G72) '

With this function , when a radius and a center angle are specified, the corresponding rectangular coordinate positions are computed automatically and the tool is brought to the required positions. This function is used in conjunction with one of the canned cycles G81 through G89, G73, G74, G76 and G77. With this function, the bolt hole cycle , the arc cycle, and the line at angle cycle are programmed. The tool moves to the position specified by a radius and an angle in rapid traverse (G00) .

. Bolt hole cycle (G70)

G70 X . . . Y . . . I . . . J . . . L . . . ;

With this command, the tool is positioned successively at equally spaced L points on a circle with the center at X, Y and the radius of 1, starting at a point located on a line forming J degree with X-axis. In the command,

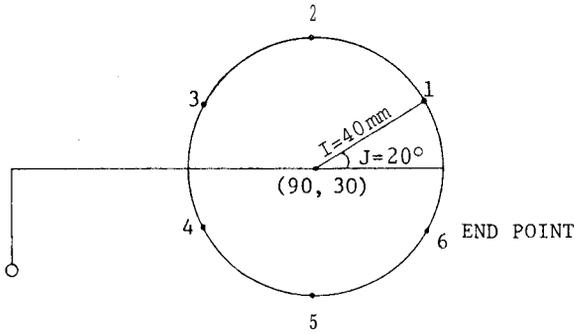
X, Y : Coordinate of the bolt hole cycle, defined either in G90 or G91 mode.

I: Radius of the bolt hole circle, programmed in a positive number and programmed with an accuracy of the least input increment.

J : Angular position of the first hole, programmed in degrees with an accuracy of 0.001 degree. CCW direction is regarded positive.

L : Number of division of the circumference. For the counter-clockwise sequence, positive numbers are programmed, and vice versa.

EXAMPLE



```
G81 G98 G90 Z-50. R-20. F20 L0 ;
G70 X90. Y30. 140. J20. L6 ;
G80 G00 XO YO ;
```

Fig. 2.69

Arc cycle (G71)

With this function, when the following command is given, the tool is successively positioned to L points located on a circular arc with the center located at X, Y and with the radius of I, at a center angle of K degrees, starting from the point lying on a line intersecting the X axis at J degrees.

```
G71 X... Y... I... J... K... L... ;
```

X, Y: Coordinates of the arc center, defined either in G90 or G91 mode

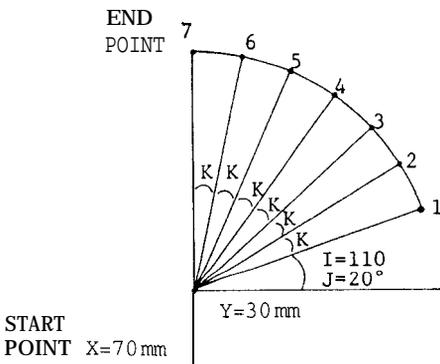
I: Radius of the arc programmed with an accuracy of the least programmable increment, and in positive numbers

J: Angular position of the first hole, programmed in 0.001 degrees. Positive values are used to command counter-clockwise direction.

K: Angular spacing in degrees with an accuracy of 0.001 degrees. Positive values are used to command counter-clockwise direction.

L: Number of holes, to be set in positive numbers.

EXAMPLE



```
G81 G98 G90 Z-50. R-20. F20 L0 ;
G71 X70. Y30. 110. J20. K15.2 L7 ;
G80 G00 XO YO ;
```

Fig. 2.70

Line at angle cycle (G72)

When the following command is given, the tool is positioned successively at L points lying on a line forming J degrees with X-axis, with a uniform interval of I, starting at X, Y.

```
G72 X... Y... I... J... L... ;
```

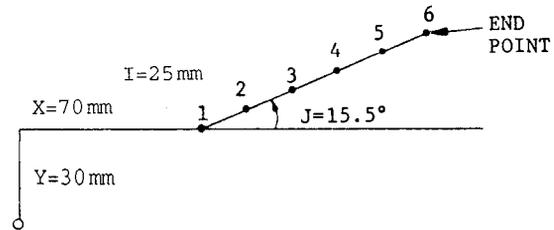
X, Y: Coordinates of the starting point, either in G90 or G91 mode

I: Interval is programmed in degrees, at an accuracy of the least input increment. When I is negative, the holes will be located on the line in the negative direction.

J: Angles programmed with an accuracy of 0.001 degrees. Positive values are used for CCW direction.

L: Number of holes programmed in positive numbers.

EXAMPLES



```
G81 G98 G90 Z-50. R-20. F20, L0 ;
G72 x70. Y 30, 125. J15.5 L6 ;
G80 G00 XO YO ;
```

Fig. 2.71

Notes :

When the hole pattern cycles are to be programmed with G70, G71 or G72, in principle, a canned cycle G73, G74, G76, G77, G81 to G89 should be programmed with L = 0, in the preceding block. Since L = 0, the canned cycle is not executed, but drilling data will be registered.

G70, G71 or G72 can be programmed in the same block with a canned cycle G code.

However G73 and G83 which involve I, J, and K can not be programmed with G70, G71 or G72 in the same block. When G73 or G83 is to be used, either Q is to be used or I, J, and K should be programmed in the preceding block.

2.9.27 HOLE PATTERN CYCLES (G70, G71, G72) '(Cent'd)

The machining control of G70, G71 or G72 ends with the drilling of the last hole, and to move the tool to the next position, the G90 (absolute) mode is more convenient than the G91 (incremental) mode, because the latter involves complicated calculations.

Immediately after the completion of the machining process as commanded by G70, G71 or G72, the canned cycle is still effective, and care must be taken in programming the subsequent block. Be sure to cancel the canned cycle by G80.

If G70, G71 or G72 command is given in the tool radius compensation mode (G41 or G42) , the ALARM code will be displayed.

G70, G71 and G72 are non-modal G codes.

Give G70, G71 or G72 command in a canned cycle mode only. If they are given without programming any canned cycle, the ALARM code will be displayed.

2.9.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99) '

Canned cycles (G73, G74, G76, G77, G80 to G89, G98, G 99) are simplified programs that contain specific movements over a number of blocks in one block. 14 types of cycles are available, and G80 code is commanded for canceling them .

Table 2.27

G code	Plunging	At hole bottom	Retraction	Application
G73	Wood pecker feed		Rapid traverse	High speed deep hole drilling
G74	Feed	Spindle forward running after dwell	Spindle reversing after feed	Reverse tapping
G76	Feed	Spindle indexing → shift	Rapid traverse → shift, spindle start	Boring
G77	Spindle indexing → shift → rapid traverse → shift → spindle start → feed	Dwell	Rapid traverse → spindle indexing → shift → rapid traverse → shift, spindle start	Back boring
G80				Cancel
G81	Feed		Rapid traverse	Drilling
G82	Feed	Dwell	Rapid traverse	Spot, facing
G83	Wood pecker feed		Rapid traverse	Deep hole drilling
G84	Feed	Spindle reversing after dwell	Spindle forward running after feed	Tapping
G85	Feed		Feed	Boring
G86	Feed	Spindle stop	Rapid traverse → Spindle start	Boring
G87	Feed	Spindle stop	Manual retraction → Spindle start	Boring
G88	Feed	Spindle stop after dwell	Manual retraction → Spindle start	Boring
G89	Feed	Dwell	Feed	Boring

Command format:

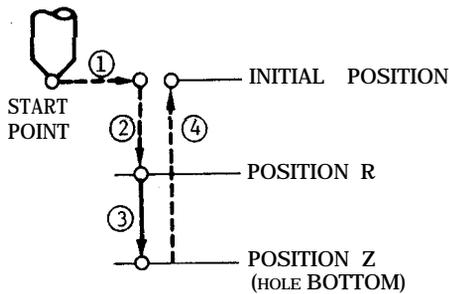
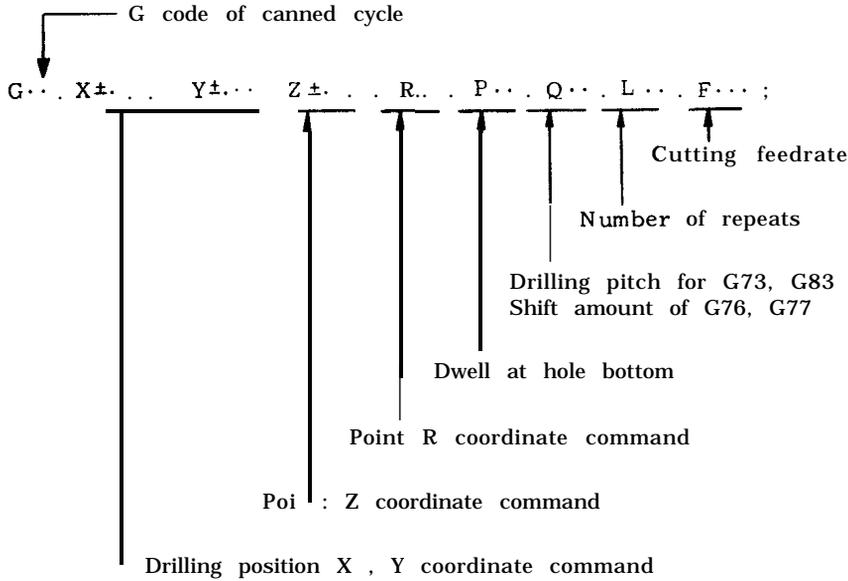


Fig. 2.72

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 Return to R point or to initial point

Number of repeats is specified by the address L. Where L is not given, number of repeats is regarded as "1."
 If 0 is given for L, only positioning to (X, Y) is made. Shift direction of shift of G76, G77 can be made at the intended angle specified by the parameter. (#6019D0)
 Z- axis returning position at the end of canned cycle can be designated by the following G code.

G code	Meaning
G98	Initial level return
G99	Position R level return

Note: When power is applied or the control is reset, the control is in the state of G code marked with ▽.

Where parameter #6019D0 is set to 0, the shift direction is made in the specified direction. The direction is set as listed below.

Bit Direction	Bit	
	D2	D1
X(+)	0	0
X(-)	0	1
Y(+)	1	0
Y(-)	1	1

Where parameter #6019D0 is set to 1, the shift direction is made at the intended angle. The shifting angle is set by setting #6506 (1 = 0.001 deg) .

2.9.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99) '(Cent'd)

Table 2.28 Canned Cycle

	With G99 (return to R)	With G98 (return to initial point)
<p>G73 (Fixed pitch)</p> <p>High speed deep hole drilling</p>	<p>G73 X... Y... Z... R... Q... L... F... ;</p> <p>δ: Setting data (#6211)</p>	<p>δ: Setting data</p>
<p>G73 (Variable)</p> <p>High speed deep hole drilling</p>	<p>G73 X... 2... 2... R... L... J... K... L... F... ;</p> <p>I: Initial value J: Reducing value K: Final value</p> <p>s: Setting data (#6211)</p>	<p>δ: Setting data</p>
<p>G74</p> <p>Reverse tapping</p>	<p>G74 X... Y... Z... R... L... L... F... ;</p> <p>SPINDLE REVERSE</p> <p>SPINDLE FORWARD AFTER DWELL</p>	<p>INITIAL POINT</p> <p>SPINDLE REVERSE</p> <p>SPINDLE FORWARD AFTER DWELL</p>

Table 2.28 Canned Cycle (Cont'd)

	With G99 (return to R)	With G98 (return to initial point)
<p>G76</p> <p>Boring</p>	<p>G76 X... Y... Z... R... Q... L... F... ;</p>	
<p>G77</p> <p>Back boring</p>	<p>G77 X... Y... Z... R... Q... L... F... ;</p> <p>NOT USED</p>	
<p>G80</p> <p>Cancel</p>	<p>G80 ;</p>	
<p>G81</p> <p>Drilling</p>	<p>G81 X... Y... Z... R... L... F... ;</p>	

2.9.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99) † (Cent'd)

Table 2.28 Canned Cycle (Cent'd)

	With G99 (return to R)	With G98 (return to initial point)
<p>G82</p> <p>spot facing</p>	<p>G82 X... Y... Z... R... P... L... F... ;</p> <p>DWELL (P)</p>	<p>INITIAL POINT</p> <p>DWELL (P)</p>
<p>G83 (Fixed pitch)</p> <p>Deep hole drilling</p>	<p>G83 X... Y... Z... R... Q... L... F... ;</p> <p>δ: Setting data (#6213)</p>	<p>s: Setting data</p>
<p>G83 (Variable pitch)</p> <p>Deep hole drilling</p>	<p>G83 X... Y... Z... R... I... J... K... L... F... ;</p> <p>I: Initial value J: Reducing value K: Final value</p> <p>δ: Setting data (#6213)</p>	<p>δ: Setting data</p>

Table 2.28 Canned Cycle (Cent'd)

	With G99 (return to R)	With G98 (return to initial point)
G84	G84 X... Y... Z... R... P... L... F... ;	
Tapping	<p>SPINDLE FORWARD</p> <p>R</p> <p>Z</p> <p>SPINDLE REVERSE AFTER DWELL</p>	<p>INITIAL POINT</p> <p>SPINDLE FORWARD</p> <p>R</p> <p>Z</p> <p>SPINDLE REVERSE AFTER DWELL</p>
G85	G85 X... Y... Z... R... L... J... ;	
Boring	<p>(x, Y)</p> <p>R</p> <p>Z</p>	<p>INITIAL POINT</p> <p>R</p> <p>Z</p>
G86	G86 X... Y... Z... R... L... F... ;	
Boring	<p>(X, Y)</p> <p>SPINDLE START</p> <p>R</p> <p>Z</p> <p>SPINDLE STOP</p>	<p>SPINDLE START</p> <p>INITIAL POINT</p> <p>R</p> <p>Z</p> <p>SPINDLE STOP</p>

Table 2.28 Canned Cycle (Cent'd)

	With G99 (return to R)	With G98 (return to initial point)
G87	G87 X... Y... Z... R... L... F... ;	
Boring		
G88	G88 X... Y... Z... R... P... L... F... ;	
Boring		
G89	G89 X... Y... z... R... P... L... F... ;	
Boring		

EXAMPLE

- A. G98 G90 G81 X... Y... Z-70. R-40. F... ;
 ... Return to initial point, absolute
- B. G99 G91 G81 X... Y... Z-70. R-40. F... ;
 ... Return to point R, incremental

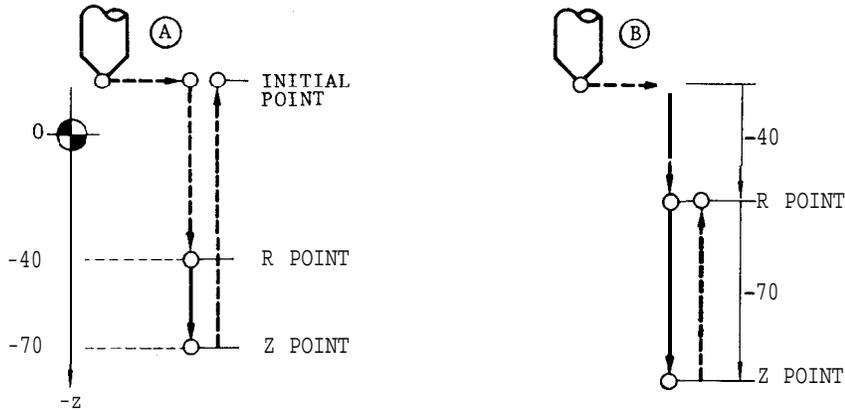


Fig. 2.73

```

c. N1 G92 XO YO Z0 ;
   N2 G98 G90 G81 X10. Y10.
     Z-50. R-20. F100 ;
   N3 G91 X20. R-30. ;
   N4 Z-50. ;
   N5 G99 Z-40. R-50. ;
   N6 G80 G00 Z50. ;
    
```

Absolute position		Remarks
Z =	R =	
-50.	-20.	Canned cycle start
-50.	-30.	Change in X, R
-80.	-30.	Change in Z
-90.	-50.	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 → N 3 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.

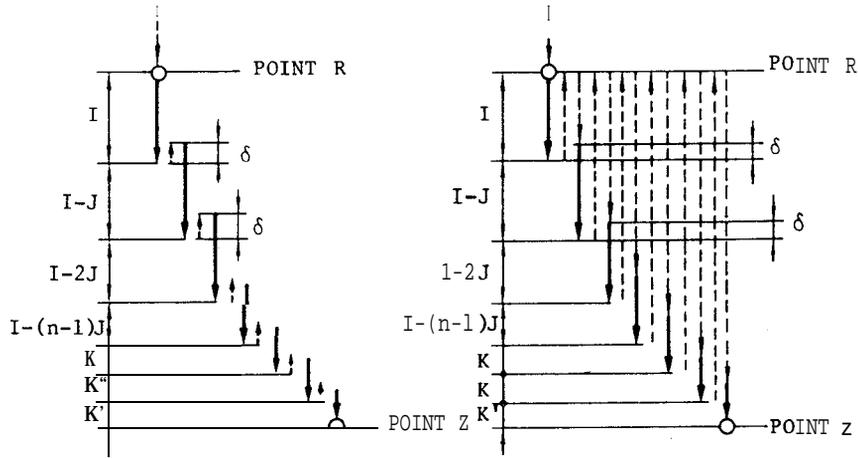
2.9.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99) (Cent'd)

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 point Z
 $K' \leq K$

(a) With G73

(b) With G83

Fig. 2.74

The value of δ is given by setting (#6211 for G73, #6213 for G83).

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 commands Q, I, J, K are given simultaneously, drilling cycle is executed with variable drilling pitch with Q as the initial value. Q0 must be commanded in the block including modal G code before programming variable pitch with I, J, and K.

EXAMPLE

```
G91 G73 X... Y... R-30. Z-55. I10. ....
      J1. K4. F... ;
```

Drilling pitch		
1st plunge . . .	10 mm	← 110.
2nd plunge ..0	9 mm	
3rd plunge .0.	8 mm	
4th plunge . . .	7 mm	
5th plunge . . .	6 mm	
6th plunge . . .	5 mm	
7th plunge . . .	4 mm	← K4.
8th plunge . . .	4 mm	
9th plunge . . .	2 mm	← K'
Total		55.00 mm ← z-55.

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 HOLD lamp lights up.

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

The single block stop after the completion of canned cycles is normal, and the FEED HOLD lamp does not light up.

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

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

X, Y, Z, α, R

When M, S, T or B† code is given in the canned cycle , M , S , T signals are sent at the first positioning in the block. In general, M, S, T should be commanded in their own block.

An program error "021" occurs when any one of the following G codes is programmed in the canned cycle mode.

G codes of * group except for G04
G70, G71, and G72
G codes of 07 group (G41, G42)

When programming G92, G27, G28 etc., make sure to cancel the canned cycles in advance.

Cancellation is performed when a G code of 01 group is programmed during canned cycles.

A single-block of DWELL (G04) can be inserted in the canned cycle mode. DWELL is executed normally.

An program error "024" 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. The address P (program number of the first block of subprogram) with M98 command destroys temporary the contents of address P for designation of dwell time, but after the jumping to subprogram, it resumes the contents.

Notes :

Programming consideration of M98 in the canned cycle mode is the same as those of other than canned cycle modes. (e. g. Restriction of execution to no more than four levels, M98 command from punched tape and the like.)

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 X10. R-20. Z-30. F100 ;

L3 ; . . . The canned cycle is not executed because X, Y, Z, u, or R is not designated in this block. The L3 is retained.

X20. ; . . . The canned cycle G81 is executed 3 times using the retained L3. After the execution , the L3 is erased.

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

Changing of R point and Z point

When R is commanded instead of Z during the execution of canned cycle in G91 mode, Z becomes incremental value from the new R point. Care should be taken.

G92 XO YO ZO
G91 X. . Y.. . R-5. O

	Point R	Point Z
Z-10. OF ;	-5.0	-10.0
X... R-7.0 ;	-7.0	-12.0
X... Z-3.0 ;	-7.0	-10.0
R-4.OZ-11.0 ;	-4.0	-15.0

G74/G84 are commands for reverse tapping/tapping but they are not distinguished. That is, when inputting by M03, M04 is output at the bottom of the hole ; when inputting by M04, M03 is output at the bottom of the hole. When G74/G84 are commanded without spindle revolution, M03 is output at the bottom of the hole.

2.9.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99) '(Cont'd)

EXAMPLE

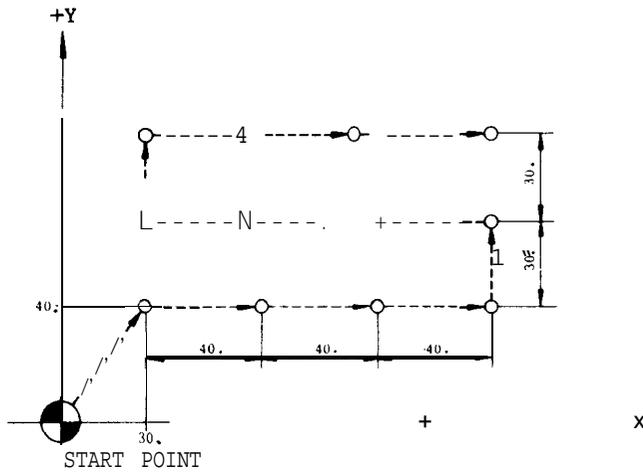


Fig. 2.75

```

N10 G92 XO YO 20 ;
N11 G90 G98 ;           . . . Return to initial point, Absolute
N12 G81 X30. Y40. R-20 2-30. F200 ; . . . Drilling cycle
N13 M98 P100 ;         . . . Jump to subprogram
N14 GOO XO YO ;
N15 T05 ;             . . . Tapper selection
N16 M06 ;             . . . Tool change
N17 G84 X30. Y40. R-20 2-30. F2000 ; . . . Tapping cycle
N18 M98 P400 ;         . . . Jump to subprogram (Note)
N19 GOO XO YO ;

```

0400

<pre> N100 G91 X40. L3 ; N101 Y30. ; N102 X-40. L3 ; N103 Y30. ; N104 X40. L3 N105 G90 G80 ; N106 M99 : </pre>	<p>Subprogram for drilling position pattern .</p>
--	---

2.9.29 UPGRADING THE CANNED CYCLE
(G73, G74, G76, G77, G80 TO G89, G98, G99,
G181, G182, G185, G186, G187, G189)†

The following functions are added to the conventional YASNAC canned cycles.

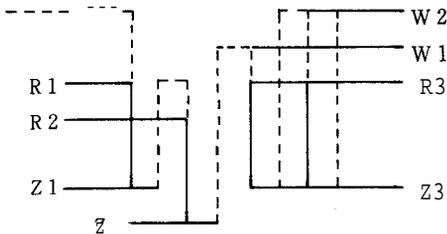
- Initial point alteration
- Improved command procedure for the high-speed deep hole drilling cycle
- 2-step feed
- Reciprocal feed
- Improved command procedure for boring and back boring
- 2-step drilling cycle
- Improved command procedure for the tapping cycle

2.9.29.1 VARIOUS FUNCTIONS

(1) Initial point alteration

Canceling the canned cycle before changing the initial point in the canned cycle is not required. Address W is used.

Example



```
G99G81 X...Y...R1...Z1...F...;
G98 X...Y...R2...Z2...W1...;
  X...Y...R3...Z3...W2...;
  X...Y...;
```

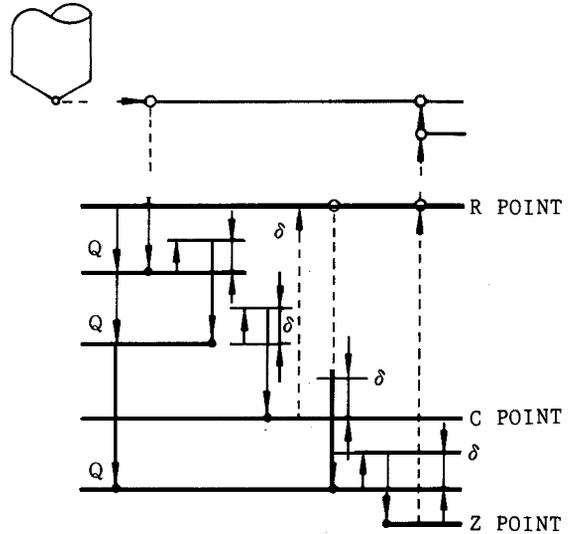
Notes:

1. The W point command under the G91 mode creates the R point and Z point based on the initial point of the previous block; not the W point of command block. Therefore, The old R point and Z point are stored to make the move, unless the R point and Z point are newly commanded.
2. When under the G98 mode (return to the initial level), single-block stop is not performed at the R point, but at the W point or the initial point.

2.9.29.2 IMPROVEMENT ON THE HIGH-SPEED DEEP HOLE DRILLING CYCLE

(1) In high-speed deep hole drilling, inserting some dwell at the bottom of the hole in each pitch can help lighten the load on the drill. Command the dwell time by setting #6212 or by address P.

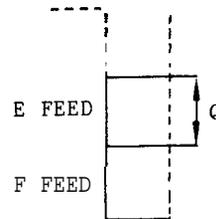
(2) The drill can be moved up to the R point during high-speed drilling. This allows the chips to be removed during the cycle.



● : Dwell
○ : Single-block stop

2.9.29.3 2-STEP FEED

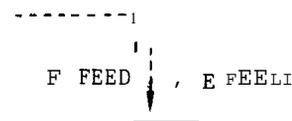
Delaying the cutting speed near the drilling start point can improve the precision without performing center drilling.



Address E: Speed of the 1st step
Address F: Distance of the 1st step speed (commanded without sign)

2.9.29.4 RECIPROCAL FEED

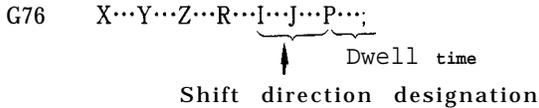
Tapping/boring, etc. :
The forward and return cutting speeds can be commanded separately. In tapping, the machining can also be ended before the taper is elongated. In boring, the machining time can be shortened. In reaming, the flaw made on the machining surface during the return cycle can be prevented.



Address: E Return speed

2.9.29.5 IMPROVED COMMAND PROCEDURE FOR BORING AND BACK BORING

The G76/ G77 shift direction can now not only be specified by setting, but can also be commanded in the program. The dwell time before shift can also be given by setting #6210 or by the program command,
This makes commanding of G76/G77 easier.



2.9.29.6 2-STEP DRILLING CYCLE

Two-step drilling cycle is provided to correspond with drilling cycles of hollow workplaces, such as castings. 100 is added to the canned cycles of the G80s, as the 2-step drilling command (G181, G182, G185, G186, G187, G189).

2.9.29.7 IMPROVED COMMAND PROCEDURE FOR THE TAPPING CYCLE

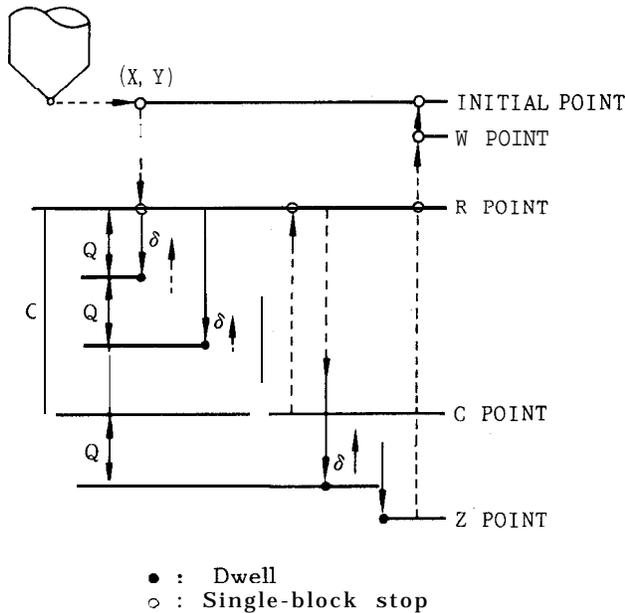
In the tapping cycle, dwell can be commanded upon R point return and initial point return, in addition to the hole-bottom dwell. This can absorb the elongation of the taper.

G 73 High-speed deep hole drilling

G73 X... Y... Z... R... Q... I... F... C... P...
w... ; (Fixed pitch)

G73 X... Y... Z... R... I... J... K... L... F...
c...P... w... ; (Variable pitch)

Example: Fixed pitch



c : Temporary R point return depth (increment)
P : Hole-bottom dwell

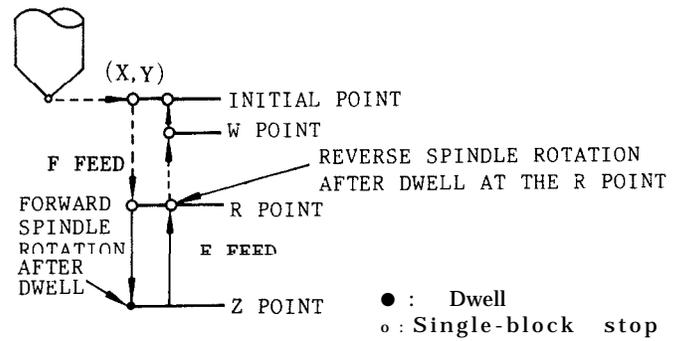
w: New initial point (absolute /increment)
delta : Setting data (#6211)

Notes:

1. When the pitch dwell is commanded by the program, it has priority over setting #6212. When program command is not given, dwell is performed by entering a numeral in the setting #6212 (1 = 1 ins). PO also follows #62 12.
2. The default of the hole-bottom dwell can also be set by #6212.
3. C and Q are incremental amounts without sign.

G 74 Reverse tapping

G74 X... Y... Z... R... P... L... F... E... Q...
w... ;



E Return feedrate (Z point -> R point)

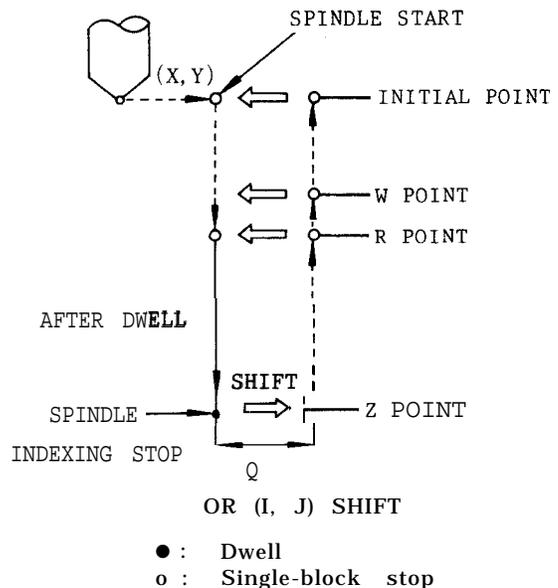
R point dwell

w: New initial point (Absolute /incremental)

G76 Boring

G76 x... Y... Z... R... Q... L... F... P...
w... ; (Fixed shift)

G76 X.Y... Z... R... I... J... L... F... P...
w... ; (Variable shift)



OR (I, J) SHIFT

● : Dwell
○ : Single-block stop

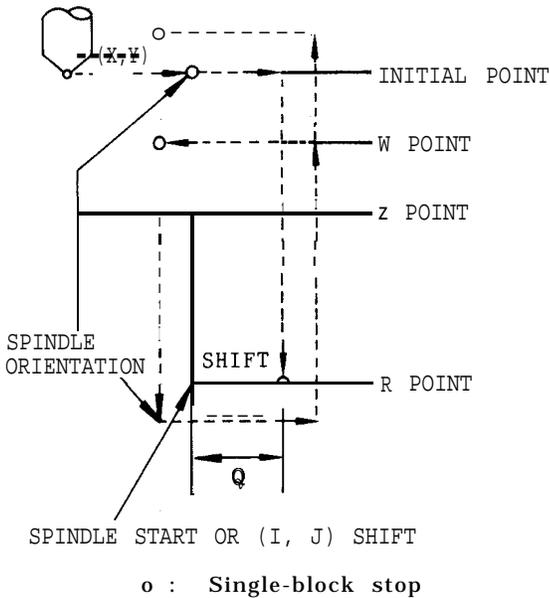
I : X-axis shift amount (incremental)
 J : Y-axis shift amount (incremental)
 P : Dwell
 W : New initial point

Notes:

1. Addresses I and J are disregarded when address Q and addresses I, and J are commanded at the same time.
2. In hole-bottom dwell, the program command has priority over the #6210 setting.
3. The shift speed is set in #6223. Note that this setting is effective only when #6019, bit DO is ON.
4. Address Q is modal, so the I or J after a Q command is disregarded. If the I or J is to be enabled, command Q80 to cancel the canned cycle before commanding I or J.
5. Setting #6210 is effective only when #6018, bit D7 is effective,

G77 Back boring

G77 X... Y... Z... R... Q... L... F... P...
 W... (Fixed shift)
 G77 X... Y... Z... R... I... J... L... F... P...
 w... ; (Variable shift)



I : X-axis shift amount (incremental)
 J : Y-axis shift amount (incremental)
 P : Dwell
 w : New initial point

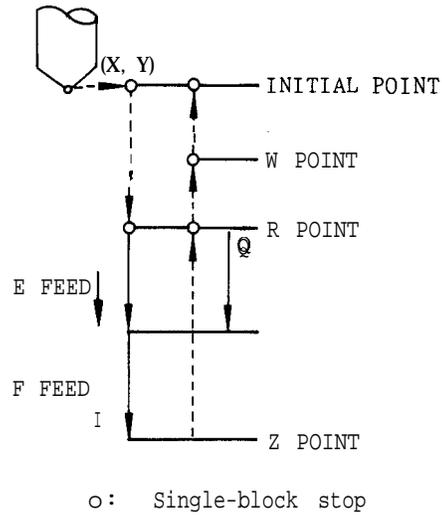
Notes:

1. Addresses I and J are disregarded when address Q and address I or J are commanded at the same time.

2. In the dwell after back boring, the program command has priority over the #62 10 setting.
3. The shift speed is set in #6223.

G81 Dwell

G81 X... Y... Z... R... L... F... E... Q...
 w... :

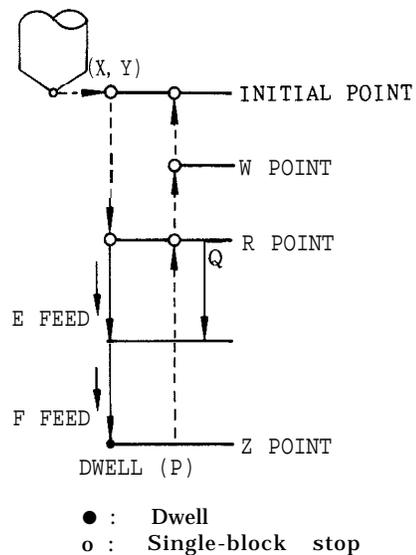


E : 2-step feedrate (1st step: E, 2nd step: F)
 Q : Speed switching point (incremental)
 w : New initial point (Absolute /incremental)

Note: Return to the R point or initial point after reaching the Z point is made by rapid feed.

G82 Spot facing

G82 X... y Z... R... P... L... F... E... Q...
 w... :



E : 2-step feedrate (1st step: E, 2nd step: F)
 Q : Speed switching point (incremental)
 w : New initial point (absolute/incremental)

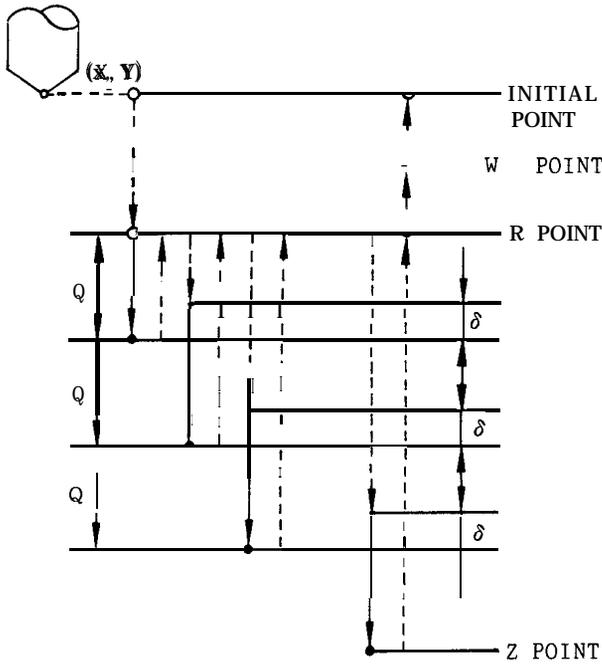
2.9.29.7 IMPROVED COMMAND PROCEDURE FOR THE TAPPING CYCLE (Cent'd)

Note: Return to the R point or initial point after reaching the Z point is made by rapid feed.

G83 Deep hole drilling

G83 X... Y... Z... R... Q... L... F... P... ;
w... ; (Fixed pitch)

G83 X... Y... Z... R... I... J... K... L... F... ;
P... w... ; (Variable pitch)



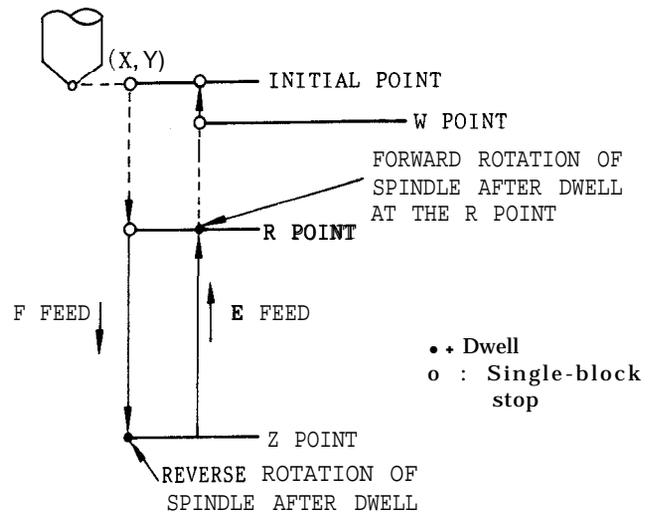
● : Dwell (P)
δ : Setting data
o : Single-block stop

P : Hole-bottom dwell
W : New initial point (absolute/incremental)
δ : Setting data (#6213)

Note: When the pitch dwell is commanded by the program, it has priority over setting #6212. When the P command is not given, dwell is performed by entering a numeral in the setting #6212 (1 = 1 ms).

G84 Tapping

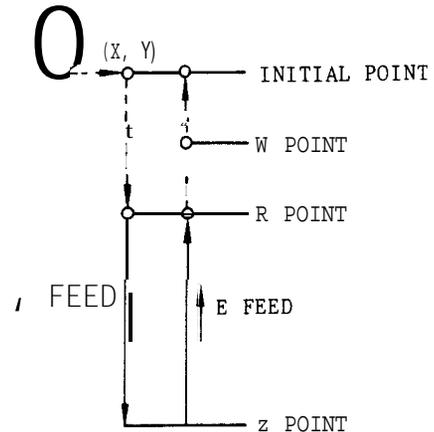
G84 X... Y... Z... R... P... L... F... E... Q... ;
w... ;



E : Return feedrate (Z point → R point)
Q : R point dwell
w : New initial point (absolute/incremental)

G85 Boring

G85 X... Y... Z... R... L... F... E... W... ;

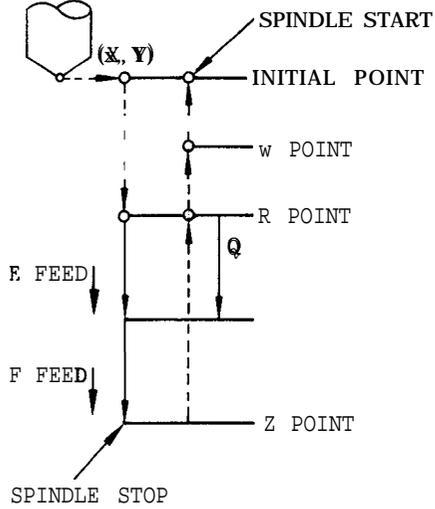


o : Single-block stop

E : Return cutting feedrate
W : New initial point (Absolute/incremental)

G86 Boring

G86 X... Y... Z... R... L... F... E... Q...
W... ;



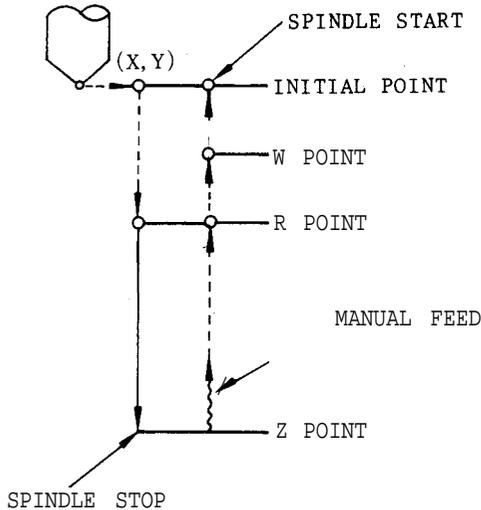
o : Single-block stop

E : 2-step feedrate (1st step: E, 2nd step: F)
Q : Speed switching point (incremental)
w : New initial point (absolute/incremental)

Note: The return to the R point or initial point after reaching the Z point is made by rapid feed.

G87 Boring

G87 X... Y... Z... R... L... F... W... ;

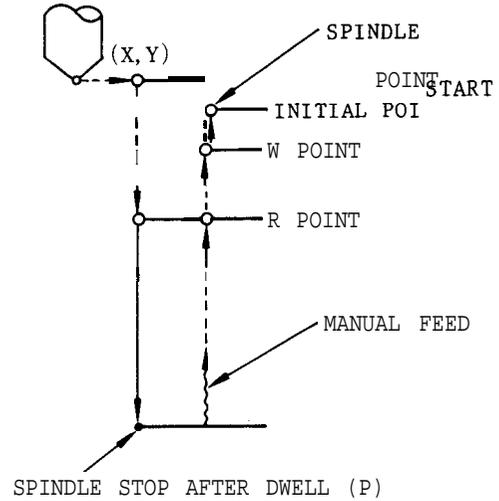


o : Single-block stop

W : New initial point (absolute/incremental)

G88 Boring

G88 X... Y... Z... R... P... L... F... W... ;

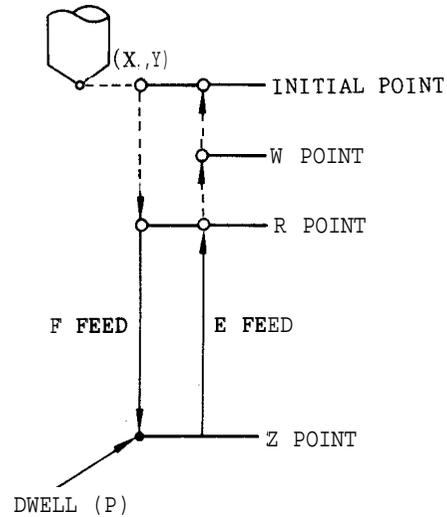


●+ Dwell
o : Single-block stop

w : New initial point (absolute/incremental)

G89 Boring

G89 X... Y... Z... R... P... L... F... E...
W... ;



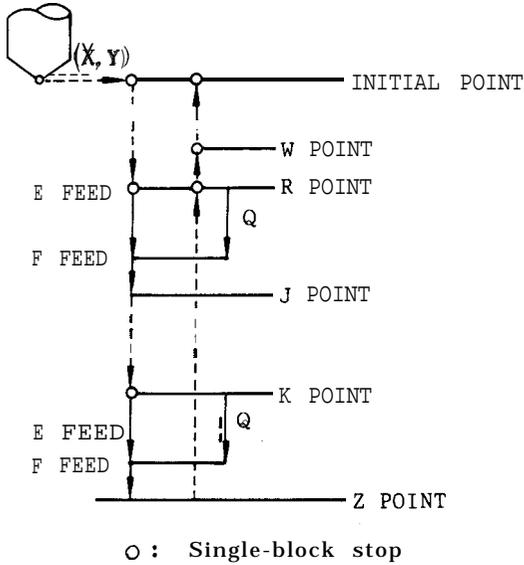
●+ Dwell
o : Single-block stop

E : Return cutting feedrate
W : New initial point

2.9.29.7 IMPROVED COMMAND PROCEDURE FOR THE TAPPING CYCLE (Cont'd)

G181 2-step drilling

G181 X.. . Y.. . Z.. . R.. . L.. . F... J... K...
E..Q., . W... ;

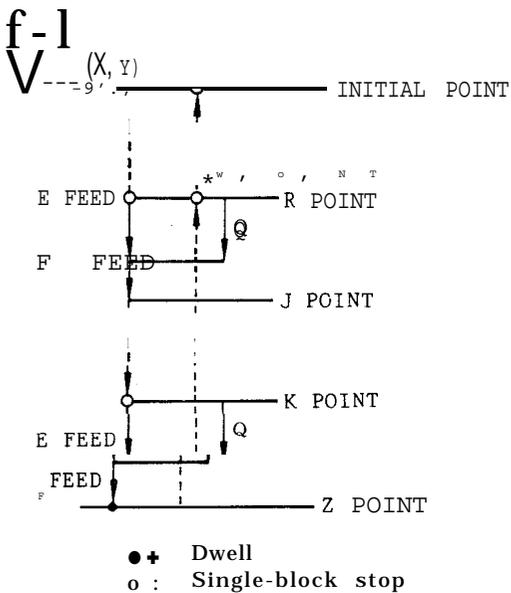


J : 2-step hole rapid feed start point (absolute/incremental)
K : 2-step hole cutting feed start point (absolute/incremental)
E : 2-step feedrate (1st step: E, 2nd step: F)
Q : Speed switching point (incremental)
w : New initial point (absolute /incremental)

Note: The return to the R point or initial point after reaching the Z point is made by rapid feed.

G182 2-step spot facing

G182 X., . Y... Z., . R... P... L.. . F.. . J...
K.E.. . Q.. . w.. . ;



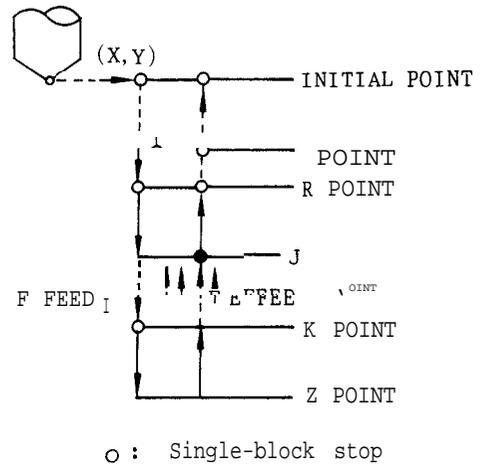
●+ Dwell
○ : Single-block stop

J : 2-step hole rapid feed start point (absolute/incremental)
K : 2-step hole cutting feed start point (absolute/incremental)
E : 2-step feedrate (1st step: E, 2nd step: F)
Q : Speed switching point (incremental)
w : New initial point (absolute/incremental)

Note: The return to the R point or initial point after reaching the Z point is made by rapid feed.

G185

G185 X.. . Y.. . Z.. . R.. . L.. . F.. . Jo. . κ...
E..W.. . ;

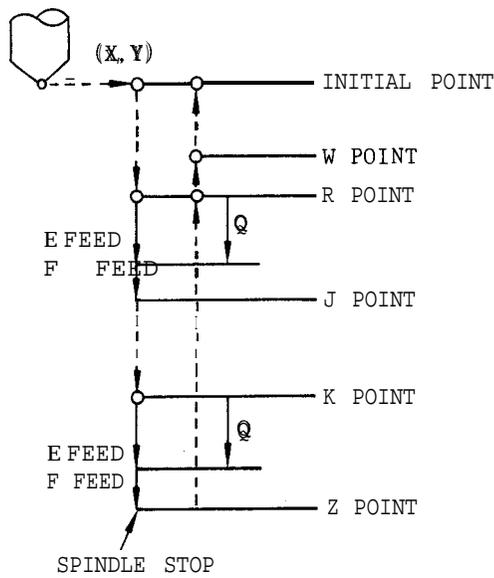


J : 2-step hole rapid feed start point (return: cutting feed start point) (absolute/incremental)
K : 2-step hole cutting feed start point (return: rapid feed start point) (absolute/incremental)
E : Return feedrate
w : New initial point (absolute /incremental)

Note: The return to the R point or initial point after reaching the Z point is accomplished by switching the cutting feed and rapid feed at points J and K.

G186 2-step boring

G186 x.. , Y., . Z.. . R.. . L.. . F.. . J.. . K...
E...Q.. W... ;



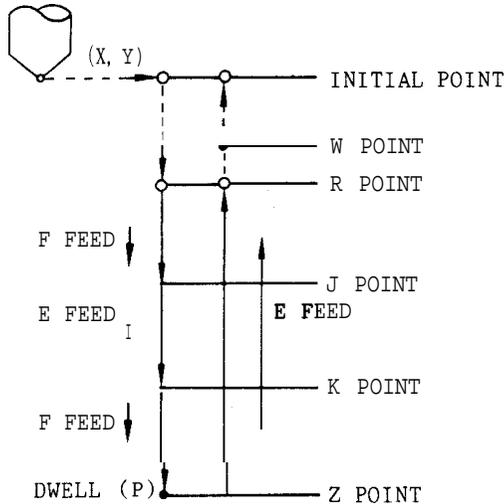
o : Single-block stop

- E : 2-step feedrate (1st step: E, 2nd step: F)
- Q : Speed switching point (incremental)
- J : 2-step hole rapid feed start point (absolute/incremental)
- K : 2-step hole cutting feed start point (absolute/incremental)
- w : New initial point (absolute/incremental)

Note: The return to the R point or initial point after reaching the Z point is made by rapid feed.

G187 2-step boring

G187 X... Y... Z... R... L... F... p... J...
K...E... W... ;



●+ Dwell
o : Single-block stop

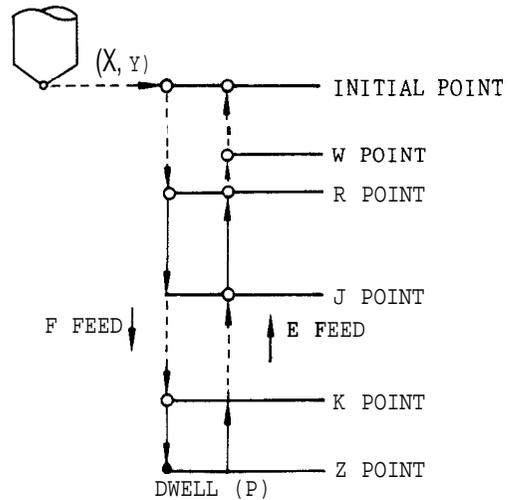
- P : Z point dwell
- J : 2-step hole rapid feed start point (return: cutting feed start point) (absolute/incremental)

- K : 2-step hole cutting feed start point (return: rapid feed start point) (absolute/incremental)
- E : Feedrate between J and K, and return cutting feedrate
- w : New initial point (absolute/incremental)

Note: The return to the R point or initial point after reaching the Z point is accomplished by switching the cutting feed and rapid feed at points J and K.

G189 2-step boring

G189 X... Y... Z... R... P... L., . F... E., .
J...K... W... ;



●+ Dwell
o : Single-block stop

- E : Return cutting feedrate
- J : 2-step hole rapid feed start point (return: cutting feed start point) (absolute/incremental)
- K : 2-step hole cutting feed start point (return: rapid feed start point) (absolute/incremental)

Note: The return to the R point or initial point after reaching the Z point is accomplished by switching the cutting feed and rapid feed at points J and K.

2.9.30 ABSOLUTE/INCREMENTAL PROGRAMMING (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, (α^\dagger) are regarded as absolute values.

G90 GOO X.. . Y.. . Z.. . ;

... Absolute designation

2.9.30 ABSOLUTE/INCREMENTAL PROGRAMMING
(G90, G91) (Cent'd)

• G91 . . . Incremental designation

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

G91 G01 X.. . Y...z.. . ;
... Incremental designation

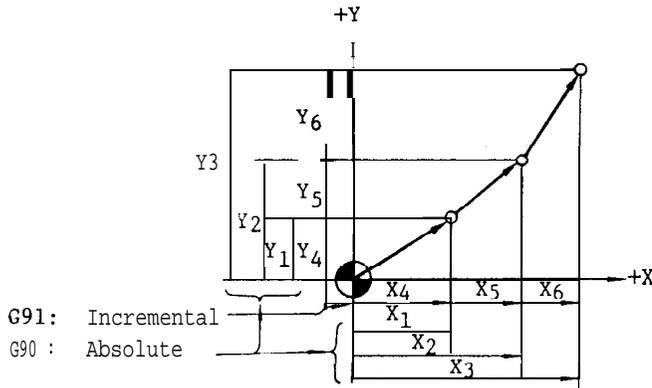


Fig. 2.76

- G90, G91 are modal G codes of 03 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 #6005D0.

Parameter(#6005D0)	Initial state
"0"	G 90
"1"	G 91

2.9.31 PROGRAMMING OF ABSOLUTE ZERO POINT (G92)

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

• G92 X.. . Y.. . Z... (α†...);

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

EXAMPLE

G92 X500. Y300. Z400. ;

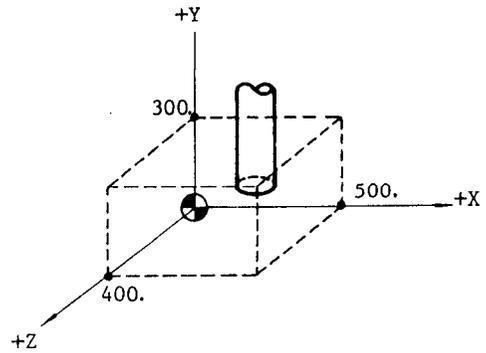


Fig. 2.77

- 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 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 G 92 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. 9)
2. Write G92 XO YO ZO α†0β0; in MDI mode, and then execute . "
3. Turn the power off and on again

2.9.32 TOOL LIFE CONTROL (G1 22, G123)

2.9.32.1 TOOL LIFE CONTROL

The tools are classified into groups and tool life (usage time, total usages or usage distance) is set for each group. This is a function to give commands for tool groups from the part program and to select the next tool in the same group, which has been sequentially arranged, when the fixed life expires.

(1) Maximum number of tools to be controlled256
Of the tool numbers from T01 to T9998, 256 can be registered as tools for tool life control. If T code commands are given with two digits, tool numbers from T01 to T99 can be used.

(2) Number of groups that can be registered and the number of tools that can be registered per group.
Maximum number of groups 128
Maximum number of tools per group 16
The maximum controllable number of tools is 256.

Note : Different number of tools can be set for each group such as 12 in group 1, 8 in group 2.

(3) Setting and displaying tool life control data:
 The tool numbers used in each tool group and the tool compensation number, life, total usages, etc. of each tool can be entered directly from the operator's panel or can be input from the part-program. The tool life control data can also be displayed with the **[OFS]** function.

TOOL LIFE CONTROL 01234 N1234															
001	002	003	004	005	007	009	010								
012	014	015	017	018	022	025	030								
031	032	033	034	035	042	043	048								
051	066	067	068	069	070	072	073								
074	075	077	078	079	080	081	082								
084	084	085	086	087	088	089	090								
091	092	093	085	099	100	101	102								
103	104	105	111	122	128	*	*								
REGISTERED GROUP NO.															
RDY															

GROUP NO.		LIFE CLASSIFICATION					
TOOL LIFE CONTROL 01234 N1234				GROUP 123-1 MINUTES			
T-NO	H-NO	D-NO	LIFE	USED	STS		
T0001	001	002	300	302	OVR		
T0011	011	012	400	400	OVR		
TO024	100	101	500	153	SKP		
T9001	212	214	1000	25			
TOO05	023	022	9000	00			
T1278	*	*	500	0			
TO054	099	00	200	0			
*	*	*	*	*	*		
TOOL NO.	TOOL RADIUS COMPENSATION NO.		TOTAL USAGES	UNREGISTERED MARK			
TOOL LENGTH COMPENSATION NO.	TOOL LIFE		TOOL LIFE OVR-LIFE EXPIRED		SKP-SKIPPED TOOL		

2.9.32.3 TOOL LIFE CONTROL DATA SETTING

(1) Setting by key input

Writing and correcting of tool life control data can be performed from the operator's panel.

(a) Sequence of operation

(i) Press the **[OFS]** key to display the tool life control screen.

(ii) Use the **[PAGE]** key or group search and search the group to be rewritten.

(iii) Since the cursor will be at the first T-NO, move the cursor to the T-NO by pressing the **[CURSOR]** key. (The cursor moves horizontally.)

(iv) Key-in the numerical value.

(v) Press the **[WR]** key. The data at the cursor position will then be rewritten.

(b) Writing T-NO

If the cursor is placed under a registered T-NO and a new T-NO is entered, the other data will be initialized and LIFE = 9999, H-NO and D-NO will be unregistered (* mark), and USED = 0 STS will be blank. Key-in a numerical value between 0 - 9998 for the T-NO.

(c) Writing H-NO and D-NO

(i) When a command such as H999 or D999 is given from the part program, the corrected numbers stored here will be searched.

(ii) As long as nothing in particular is written in H-NO and D-NO, they will be considered unregistered (* mark) .

(iii) Unregistered and H-NO = 0 are different.

Although H-NO = 0 and D-NO = 0 are corrections cancelled, if H (D) 999 is executed in unregistered state, an error will occur.

(iv) Key-in numerical values from 0 to 299 for H-NO and D-NO.

(d) Writing LIFE

(i) The numerical value 9999 will be automatically written when T-NO is written.

2.9.32.2 TOOL LIFE CONTROL DATA DISPLAY

With the function set to **[OFS]**, tool life control data are displayed by depressing **[OFS]** once again. The **[OFS]** key is a two stage switch for offset display and tool life control display.

(1) List of tool life control group registrations

Pages 1 and 2 of the tool life control display shows a list of registered tool groups. Check the information on these pages for any group to be indexed.

(2) Tool life control data display

Page 3 and subsequent pages of the tool life control display are the tool life control data display. Each tool group composes a 2-page data display screen. Using "group search" will be convenient to display the desired page.

<Operating sequence of "group search">

(a) Press the **[OFS]** key to display the tool life control screen.

(b) Press the **[PAGE]** key to display the tool life control data screen.

(c) Key-in the numerical value of the object group.

(d) Press the **[CURSOR]** key. Data display of the object group will then appear.

2.9.32.3 TOOL LIFE CONTROL DATA SETTING (Cont'd)

(ii) Key-in LIFE with a numerical value between 1 and 9999.

(e) Writing USED

(i) When T-NO is written as the value of USED, the numerical value 0 will also be automatically written.

(ii) In general, start USED from 0.

(f) Writing life classification

(i) Press the **PAGE** key to move the cursor to the initial T-NO position and press **CURSOR↑**. The cursor will move to the life classification position,

(ii) Key-in **[]** and **[WR]** to change life classification will change to MINUTES.

Key-in **[1]** and **[WR]** to change life classification will change to COUNTS. Key-in **[2]** and **[WR]** to change life classification will change to METERS or FEET.

Key-in **[3]** **[WR]** to change the life classification to drilling count (HOLES).

(iii) Writing of life classification is effective for the group written. Although one group has two pages, both pages display the same in life classification so either page can be written.

(iv) For life classification, key-in the numerical values "0 to "3".

(g) Data erasure

Data can be erased by using the **[ORG]** key.

A list of data erasure methods is shown in the table below.

Table 2.29 Data Erasure Method

Erasure Type	Page	Cursor Position	Key-in Data
Erasure of all groups	All pages	Possible at any place.	[] [-] [9] [9] [9] [9] [ORG]
Erasure of one group	All pages	Possible at any place.	[G] xxx [ORG] Group number to be erased.
Erasure of T-NO	Page 3 and subsequent pages	T-NO with data.	[T] [ORG]
Erasure of STS	Page 3 and subsequent pages	STS with data.	[S] [ORG]

(h) Notes :

(i) There are mutual relations in data modification of LIFE, USED and STS.

When LIFE ≤ USED, OVR is generally lit. If STS is cleared, USED will also become "O".

(ii) Data correction is possible by key operation only when the edit lock is "open" and not in automatic operation,

(iii) If one group or all groups are erased, the life classification of the groups concerned will be initialized to MINUTES.

(2) Setting with the part program

Tool life control data can be set by executing a program with the format shown in the table below.

(a) Tool registration G codes (G122/G123) commands are given in the following form.

G122;, Starts tool registration

G123; Ends tool registration

Give commands on tool information to be set in the section of G122;~G123;.

(b) Data commanding format

Table 2.30 Tool Life Control Data Format

Tape Format	Meaning
G122;	Tool registration start.
P#### 1A ;	Commands group No. after P. Commands life classification after 1.
T####H X D### Lxxx;	Specifies tool No. after T Specifies tool length compensation No. after H.
T####H(X) D### Lxxx;	Specifies tool radius compensation No. after D. Specifies tool life(1-9999) after L.
P#### 1Δ ;	Sets new group data.
T####H OUC D### Lxxx;	
G123	Tool registration end

(c) G code (G 124) for clearing tool registration
The tool life control data can be cleared by entering the G124; command. If a command for P is given in the same block as G124, one group only will be erased. (P = 1 to 128)

In other words, by entering the command G124P. ..., it will be possible to erase only the group specified by P.

Notes :

(i) All previously registered data will be cancelled by executing G 124 ;. If command G 122 is entered without executing G 124, the following will occur.

(a) If tool numbers have already been registered previous data will be erased and replaced with new data. If H and D are omitted in the block in which T exists, T will change to unregistered (* mark).

(b) If tool numbers are unregistered, new data will be registered in simple form. If H and D are omitted in the block in which T exists, T will change to unregistered (* mark) . If L is omitted, L9999 will be stored.

(ii) Settings are possible in which "registration will start after canceling all previously registered data if command G122; is executed" .

(iii) The H and D codes can be omitted when not being used.

(iv) Tool life L will be modal between G 122 and G 123 so these commands can be omitted when setting to the same tool life.

(v) The following life classifications can be set by specifying I.

- IO Controlling by time (1 to 9999 minutes)
- 11 Controlling by counts (1 to 9999 counts)
- 12 Controlling by distance (1 to 9999 mm, 1 to 9999 feet)
- 13 Controlling by the drilling count (1 to 9999 counts)

(vi) Each of the addresses T, H, D, L, P, I has the following commendable range. Any other command values will cause in an error.

$$0 \leq T \leq 9998, 0 \leq H, D \leq 299, 0 \leq L \leq 9999, 1 \leq P \leq 128 > 0 < I \leq 2$$

(vii) Commands G122 and G123 can be used in independent blocks.

(viii) Do not enter commands (such as a shift command) other than tool information in the G 122 ; . . . G123 ; section. (Alarm will occur if such a command is entered.)

(ix) Do not switch over the mode to execute a manual MDI command while executing G 122 ;. (Alarm will occur if this is attempted.)

(x) If I is omitted, the life classification originally specified in that group will remain unchanged.
A cleared group will generally become in the "minutes" state.

2.9.32.4 EXECUTION OF TOOL LIFE CONTROL

(1) Commands with the part program

Tool life control can be performed with commands from the part program by using the T and H codes shown in the table below.

Table 2.31 Tool Life Control by Part Program

Tape Format	Meaning
T10001;	Tool life control ON mode will be set with the M06 command of "tool group number + 10000. "
①M06;	The tool will be mounted on the main shaft with the M06 command and time and distance count will normally start thereafter.
H999;	The H-NO registered on the tools will be selected with the H999 command.
D999;	The D-NO registered on the tools will be selected with the H999 command.
D0;	Cancel D
H0;	Cancel H
TΔΔΔΔ;	Returns to normal T command
②M06;	The tool on which life control is being carried out with the M06 command will be removed from the main shaft and counting of time and distance will end.
M30	Machining ends.

Notes :

1. The T code command of the numerical value of "tool life group number + 10000" with T10001 to T 10128 controls tool life and outputs the T codes registered in that group.
2. The H and D codes of the tool being selected will be selected by commands H999 and D999 with the tool life control function.

(2) Tool life count

(a) One of the following can be selected with parameter #6020 on the tool on which life count is to be performed.

- . Counting of a tool in the latest T command group of the M06 command.
- . Counting of a tool in the T command group just prior to the M06 command.

(b) Tool life count will not be performed by time, distance or counts when the "tool life count disregard input" is in ON state.

(c) When tool life is set by time

The control will count every second in which the tool is actually used in the section between ① and ② of the table in para. (1) and, in the cutting mode. Life count will be increased by 1 each time one minute is exceeded. However, the time required for single block stop, feed hold, fast feed and dual operation will not be counted.

2.9.32.4 EXECUTION OF TOOL LIFE CONTROL (Cont'd)

(d) When tool life is set by distance

The cutting distance is calculated by the interval (every second) outlined in para. (c) and life count will be increased by "1" each time the distance exceeds 1 inch or 1 foot.

(e) When tool life is set by count

The count will increase by "L" when the command T9999L $\Delta\Delta\Delta$; in the section between ① and ② of the table in section (1) is executed. If L is omitted, L1 will be equivalent.

(f) When the tool life is set by the drilling count

The repeat count "L" of the drilling command (canned cycle <G73, G74, G76., G77 and G81 to G89>, drilling pattern cycle <G70, G71, G72>) counts the life within the commanded tool life count range. If no "L" is specified, one drilling is counted, when "L" = 0, 0 drilling is counted.

The life count is performed in the look-ahead process. Therefore, if the life ends during execution of a single block, the life will be over before executing that block.

(g) The count can be increased by "1" only during the M02/M30 command. However, the count cannot be increased by incorporating T9999L $\Delta\Delta\Delta$ and M02/M30 in the same program. Selection of whether to count with T9999L $\Delta\Delta\Delta$ or M02/M30 must be made with parameter #6020.

(h) The maximum value for USED is 9999 so the count will not go above 9999.

2.9.32.5 PARAMETERS AND SETTINGS FOR TOOL LIFE CONTROL

(1) Settings

#6004 (D6)

Clears life data registered prior to the G122; command

O: Does not clear 1: Clears

#6204

Specifies group number when resetting for tool change. However, this is only effective when #6020 (D5) is "O". Setting 1 to 128

(2) Parameters

#6020 (D7)

Specifies group number for tool change skipping (TL SKP)

O: Currently specified group

1: External signal (TL1 to TL64)

#6020 (D6)

Selection of a T command group during an M06 command.

O: Latest T command

1: Immediately Prior T command

#6020 (D5)

Group number command for tool change reset (TLRST)

O: Setting #6204 1: External input (TL1 to TL64)

#6020 (D4)

T/M selection when tool life control is by count

O: Counts with T9999L $\Delta\Delta\Delta$

1: Counts with M02/M30

2.9.32.6 ALARM CODE DETAILS ON TOOL LIFE CONTROL

(1) Format errors of alarm 125: G122/G123/G124

(a) A different address command was made in the G122 or G123 block.

(b) P command was neglected and T command was made in G122.

(c) A numerical value other than $1 \leq P \leq 128$, $0 \leq I \leq 2$, $0 \leq T \leq 9998$, $0 \leq L \leq 9999$, $0 \leq H \leq 299$, and $0 \leq D \leq 299$ was commanded in G122.

(d) Intervened in G122 with a manual MDI.

(2) Alarm 126:

This is an error when there is an overflow during execution of a life control data command.

(a) A command was entered for 257 or more tools.

(b) A command was entered for 17 or more tools in one group.

(3) Alarm 127:

This is an error in a T5 digit command and a T9999 command function

(a) When a T5 digit command was executed, the object group was not registered.

(b) The T9999L $\Delta\Delta\Delta$ command was executed although the parameter was for a count with M02/M30.

(c) The T9999L $\Delta\Delta\Delta$ command was executed although it was not a life control classification count.

(4) Alarm 128:

This is an error in the T5 digit command and the H(D) 999 command function.

(a) All of the groups were in SKP status when the T5 digit command was executed.

(b) Tool numbers H-NO and D-NO were found unregistered (* mark) when executing command H(D)999.

Notes :

(1) Do not rewrite the life control data when executing life control.

(2) Even if M06 is executed in manual MDI, life control will not be executed. Do not execute commands T10001 through T10128 in manual MDI.

(3) If OVR and SKP occur at the same time, T-NO will display SKP on a priority basis.

2.9.33 ROTATION OF COORDINATES (G68, G69) †

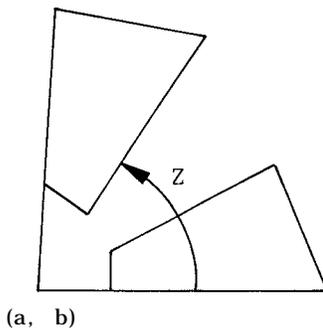
$$(1) \left\{ \begin{array}{l} G17 \\ G18 \\ G19 \end{array} \right\} \quad G68 \quad a \dots b \dots R \dots$$

Where,

a, b: Values of coordinates of the center of rotation

R: Angle of rotation (+ for counterclockwise direction, and absolute value is to be used for command)

By commanding with the above, the commands thereafter are rotated by the value of the angle assigned by R around the center commanded by point a, b. The angle of rotation must be commanded in 0.001 degree units.



(2) Plane of rotation is determined by the plane (G17, G18, G19) that has been selected.

G 17 XY-plane or X4-plane
 G 18 ZX-plane or Z4-plane (But the 4th axis is limited to linear axis.)
 G 19 YZ-plane or Y4-plane

(3) G69;

When this is commanded, the coordinate rotation mode is cancelled.

(4) Note:

(a) If a and b are omitted, the position where the command of G68 was made becomes the center of rotation. Omission of R is not permitted.

(b) Blocks of G68 a R; and G69; must be commanded by a single block. Be sure to use G68 G69 in pairs for the rotation of coordinates.

(c) Position display shows the position where the rotation of coordinates was commanded.

(d) When using together with scaling functions, the rotation of the coordinate system is made after turning-on scaling, and thus the command should be made in the following order:

G51; (Scaling on)
 G68 (Coordinate rotation on)

G69; (Coordinate rotation off)

G50; (Scaling off)

(e) G68, G69 are modal G codes of group 18.

(f) G69 is automatically selected during power turn-on or reset operation.

(g) Commanding the rotation of coordinates (G68) is not possible during tool radius compensation C. It causes an alarm.

(h) The following G codes cannot be commanded during the rotation of coordinates: They activate alarms.

G29, G31, G36, G37, G38, G53 and G92

(i) The rotation of coordinates turns on during the approach of ordinary machining and turns off upon completion of machining. It should be noted that no proper machining shape is obtained if the rotation of coordinates is turned on during machining.

If the rotation of coordinates (G68) is commanded during the canned cycles, the canned cycle mode is canceled.

(j) Alarm codes related to the rotation of coordinates are as follows:

Alarm No.	Contents
150	G code not to be used in G68 mode has been commanded. G68 is commanded again when a G68 command is executed.
151	Format of G68, G69 command blocks is not correct.

2.9.34 SETTING OF LOCAL COORDINATE SYSTEM (G52) †

(1) G52 Q2 X . . . Z . . . Z . . . a . . . ;
 (α indicates the 4th axis.)

When the above command is issued, a coordinate system shifted by the corresponding commanded value from the work coordinate system is set.

This coordinate system is called the local coordinate system, and thereafter the tool moves on this assigned local coordinate system.

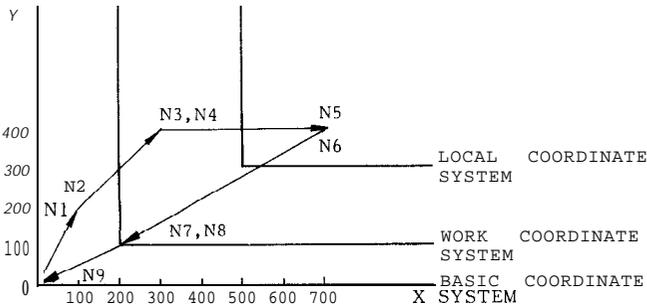
2.9.34 SETTING OF LOCAL COORDINATE SYSTEM (G52)(Cent'd)

Programming Example

```

N1 G90 G01 X100 Y200 F100;
N2 G54;
N3 X100 Y300;
N4 G52 Q2 X300 Y200;
N5 X200 Y100;
N6 G52 Q2 X0 Y0;
N7 X0 Y0;
N8 G52;
N9 X0 Y0;
    
```

- Work Coordinate System Shift Amount (200, 100)



(2) G52 Q2 XO YO ZO (α 0) ;

When this command is issued, the local coordinate system is cancelled and return to the work coordinate system occurs.

(3) G52;

By this single block command, the return to reference coordinate system occurs.

(4) Remarks

(a) G52 Q2 command is effective only when the work coordinate system has been set. Alarm "043" occurs if the command is made under the state of the work coordinate system setting.

(b) Setting of coordinate system by G92 command or ORG key is not permitted under the state of setting work coordinate system and local coordinate system.

(c) Precautions with the setting functions of work coordinate system are also applicable here.

(d) It should be noted that G52 performs the operation of canceling the work coordinate system (G54 to G59) if the above option has not been added.

2.9.35 AUTO MODE HANDLE OFFSET†

This is the function of synchronizing the movement by manual pulse generator with the movement by auto operation during auto operation (tape operation, MDI operation, memory operation). Deviation due to the mounting of work can be offset by this function.

(1) Operating procedures are as follows :

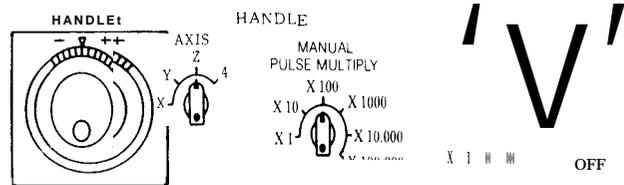
(a) Turn on the auto mode handle offset switch.

(b) Select the axis to be moved by means of the handles axis select switch. However, if "the manual pulse generator of simultaneously controllable axes of three-axis control" has been added, the movement with simultaneous 3 axes can be performed.

(c) Select the distance traveled per graduation of handle by means of manual pulse multiply switch. The distance traveled per graduation can be switched to 1, 10 or 100 pulses.

(d) If the handle is turned during the auto operation of interpolation block, the distance traveled by handle is synchronized with the distance traveled by auto operation on the axis selected by Step (b) .

Clockwise direction: To positive direction
 Counterclockwise direction: To negative direction



(e) Turn off the auto mode handle offset switch.

(f) After that the movement is made with the shift corresponding to the offset made by the handle. However, for the setup command (such as G92) of coordinate system thereafter, the offset portion by the handle is not added, and the setup only by the commanded values is performed.

(2) Remarks

(a) Movement of auto mode handle offset is effective only during interpolation. in auto operation. It is invalid during rapid traverse or single stop.

(b) Under an alarm state, movement by the auto mode handle offset is not possible.

(c) When the axis interlock input (IT) is on, movement by auto mode handle offset is not possible.

(d) By means of parameter setting, it is possible to invalidate the movement by auto mode handle offset of each axis.

Axis	Parameter		Valid/Invalid
	No.	Setting	
x-axis	#6010D5 (HOF SX)	1	Valid
		0	Invalid
Y-axis	#6010D6 (HOF SY)	1	Valid
		0	Invalid
Z-axis	#6010D7 (HOF SZ)	1	Valid
		0	Invalid
4th-axis	#6011D5 (HOF S4)	1	Valid
		0	Invalid

- (e) The auto mode handle offset switch is #13207 (HOFS).
- (f) For the function stated above, manual absolute function is invalid.

2.10 USER MESSAGE DISPLAY †

2.10.1 ALARM MESSAGE DISPLAY BY #8000 COMMAND

The user can display any alarm message on the CRT screen. By specifying the following command in the part program, 3-digit alarm number n and the alarm message (less than 32 characters) may be designated:

#8000 = n (<alarm message>) ;

The alarm number should be 3 digits and one that is not used by the control. For the alarm message, alphanumeric characters may be used. When this #8000 command is executed, the alarm message is displayed and "ALM" flashes at the bottom of the CRT screen.

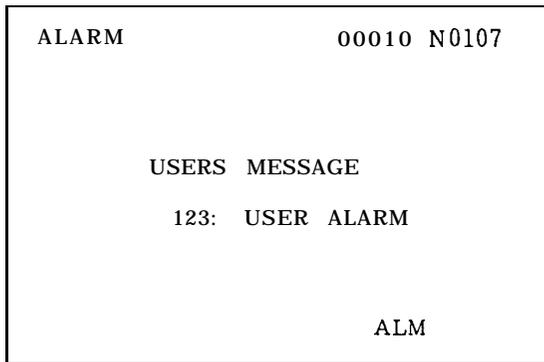


Fig. 2.78 Sample Display of Alarm Message by #8000 Command

Note : If a command with a wrong alarm message is specified, a format error will occur.

2.10.2 MESSAGE DISPLAY BY CONTROL-OUT AND CONTROL-IN

(1) Programming and Display by Control-out and Control-in

The user can program any message between the information enclosed with control-in and control-out in the part program in order to display the message on the CRT screen. The enclosed information is assumed to be insignificant.

(2) How to Edit Control-out and Control-in

The control-in and control-out part may be edited by the usual edit operation.

- a. Depress key and **SHIFT** key, and character "(" is entered.
- b. Depress key and **SHIFT** key, and character ")" is entered.

Notes:

1. The characters which are indicated in the thick-lettered keys shown below may be enclosed in control-out and control-in.
2. The number of characters that may be enclosed in control-out and control-in less than 32.
3. Nesting of the control-out and control-in is not allowed.

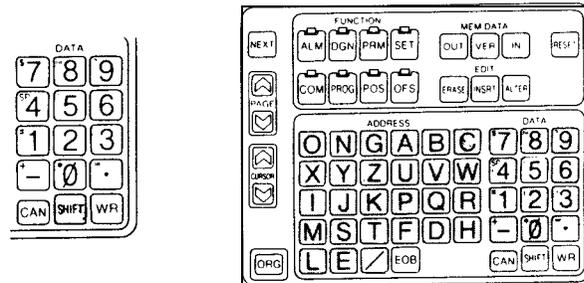
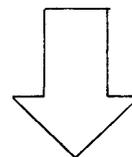
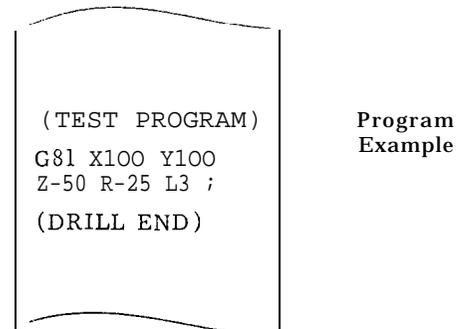
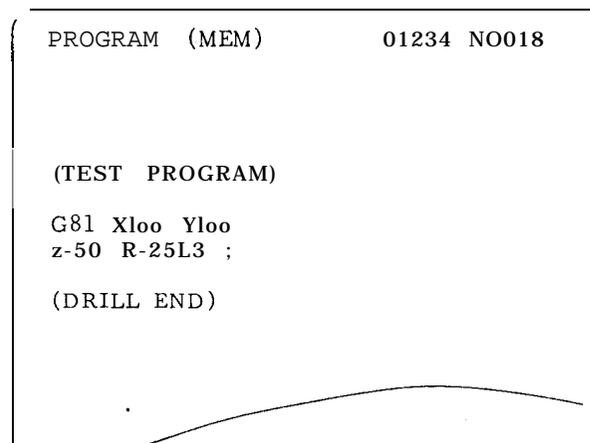


Fig. 2.79

Characters Enclosed in Control-out and Control-in (Thick-Lettered Keys) for 9 CRT



Message Display by Control-out and Control-in

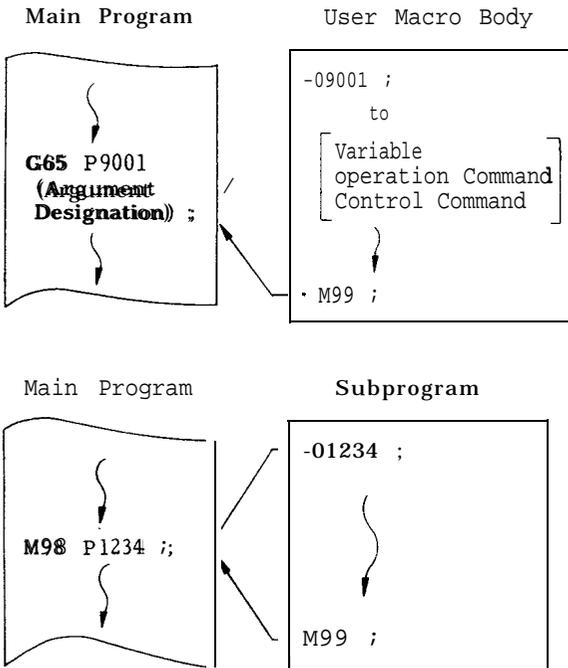


2.11 USER MACRO (G65, G66, G67)

2.11.1 OVERVIEW

Special programs written by the machine builder or user by the use of a group of commands are registered in the part program memory. These programs can be called by the use of G65 or G66 command to execute them.

These special programs are referred to as the user macro body, which can be written and stored in the same format as a subprogram.



However, unlike a subprogram, a user macro allows:

- (1) Use of variables.
- (2) Computation between variable or between constants.
- (3) Use of control commands such as conditional branch.

These features enable the user macro body to provide a generalized program that requires complicated computations and decisions.

The "argument designation" in calling a user macro body from the main program makes it possible to assign the real numbers to the variables in the body. This enables this user macro to run as a series of specific program that provide tool movements.

In this manual, the user macro body is sometimes referred to as "macro program" or, simply, "macro."

2.11.2 USER MACRO CALL COMMANDS

A user macro body may be called in the following five manners:

No.	Type of Call	Code	Remarks
1	Simple call	G65	
2	Modal call	G66	G67: For cancel.
3	Call by arbitrary G code	Gxx	
4	Call by M code	Mxx	
5	Call by T code	Txxxx	4 digits max.

(1) Simple Call (G65)

G65 P... L... (argument designation) ;

The macro program whose program number was specified by P is called and is executed L times. The default value of L is 1. When the designation of an argument to the user macro is desired, specify it in (argument designation). "Argument designation" is the assignment of real numbers to the "local variables" used in the user macro. For details, see 2.11.3 ARGUMENT DESIGNATION.

(2) Modal Call (G66, G67)

G66 P... L... (argument designation) ;

This command provides the mode to call the macro program of the program number specified by P. Each time a move command is executed, the specified macro is run L times.

G67 ;

This command cancels the modal call mode.

(3) Macro call by Arbitrary G Code

Gxx (argument designation) ;

This provides the command which is equivalent to G65 P (argument designation) ;. For Gxx, ten sets of G codes of G01 through G199 except for those designated by NC machine builder may be set for the parameter. The macro program numbers which correspond to these G codes are as follows :

#6120 ... Sets G code which calls the macro of program number 09010.

#6121 ... Sets G code which calls the macro of program number 09011.

to

#6129 ... Sets G code which calls the macro of program number 09019.

Note : Macro call by arbitrary G code permits only single nesting. Namely, the macro which was called by the macro call using arbitrary G code or the macro call with M code or T code does not permit another macro call by arbitrary G code.

(4) Macro Call by M Code

G... X... Y... Mxx ;

This command may call macros. In this case, the macro is executed after the execution of the move command in the same block. MF and M codes are not transmitted. For Mxx, four sets of M codes of except for MOO, MO1, MO2, M30, M90 through M99 may be set for the parameter.

#6130... Sets M code which calls the macro of program number 09001.

#6131... Sets M code which calls the macro of program number 09002.

#6132... Sets M code which calls the macro of program number 09003.

#6133... Sets M code which calls the macro of program number 09004.

For macro call by arbitrary M code macro, argument designation is available.

#6073 D₄ 1 : Argument designation applicable
 O : Argument designation not applicable

	M code
#6580	Program No. 09005 macro calling M code
#6581	Program No. 09006 macro calling M code
#6582	Program No. 09007 macro calling M code
#6583	Program No. 09008 macro calling M code
#6584	Program No. 09009 macro calling M code
#6585	Program No. 09020 macro calling M code
#6586	Program No. 09021 macro calling M code
#6587	Program No. 09022 macro calling M code
#6588	Program No. 09023 macro calling M code
#6589	Program No. 09024 macro calling M code
#6590	Program No. 09025 macro calling M code
#6591	Program No. 09026 macro calling M code
#6592	Program No. 09027 macro calling M code
#6593	Program No. 09028 macro calling M code
#6594	Program No. 09029 macro calling M code
#6595	Program No. 09030 macro calling M code
#6596	Program No. 09031 macro calling M code
#6597	Program No. 09032 macro calling M code
#6598	Program No. 09033 macro calling M code
#6599	Program No. 09034 macro calling M code

Notes:

1. The current M code macro specifications and the M code macro specification that allows argument designation can also be switched by parameter setting.

2. No argument designation can be made to the M code in the M code macro. If commanded, error "104" [PROG ERROR (DOUBLE ADD)] will occur.

3. When arguments can be specified, addresses X, Y, Z, A, B, C, U, V and W are also regarded as arguments.

When these M codes are specified in the macros called by the macro call using arbitrary G code or by the macro call using M code or T code, the macro call is not executed but is handled as a usual M code.

(5) Macro Call by T Code

All the T code commands provide a macro call command.

G... X... Y... Txxxx ;

With this command, the macro of program number 09000 is executed after the move command in the same block is executed:

Transmission of T code and TF signal is not performed. Whether a T code is to be used as a macro call command may be specified by the following parameter:

Parameter No.

#6134

o : T code designation is handled as a T code.

1 : T code designation is handled as macro call command to call the macro of program number 09000.

When a T code is specified as a macro call command, the value designated by T "xxxx" (up to decimal 4 digits) becomes the argument of common variable #149. Note that the argument designation other than this is not permitted. When a T code is designated in the macro called by the macro call by arbitrary G code or by the macro call by M code or T code, the macro call is not executed but is handled as a usual T code.

Priority of G. M. T codes specified in the same block is :
 G > M > T.

Example

If specified as G** M**T**, only G** is valid and M and T are ignored.
 (In this case, no alarm.)
 T**M**G** results in the same. That is, specifying order does not make matter ; only G** is valid.

When the tool life control option is provided, T5-digit command is also called by T-code macro.

At this time, T number of the group called by T5-digit command is input to #149.

2.11.2 USER MACRO CALL COMMANDS (Cent'd)

(6) Multiple Call

A. G65 Simple Call and G66 Modal Call

As a subprogram is called from another subprogram, a user macro may be called from another user macro. Quadruple nesting is permitted for simple call and modal call combined.

Program example	
061 ;	062 ;
DO1 ;	DO1 ;
D02 ;	M98P63 ;
M98P62 ; - 1	GOT020 ;
GOTO10 ;	END1 ;
END2 ;	N20M99 ;
END1 ;	
N10M30 ;	

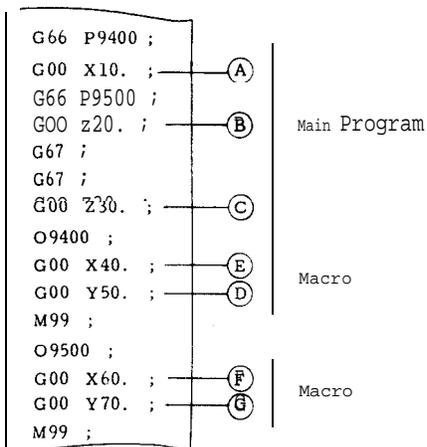
M98 or /G65 can be specified at the above "-1."

Besides quadruple nesting is permitted for subprogram. Multiple call is disabled for the macro call by arbitrary G code, or M code or T code.

B. Multiple Call by G66 Modal Call

In modal call, each time a move command is executed, the designated macro is run. This is also valid for the move command in the macro called by multiple call. The macros are sequentially executed from the one designated latest.

Sample Program



The above sample program is executed in the following order :

(A) → (E) → (D) → (B) → (F) → (E) → (D) → (G) → (E) → (D) → (C)

2.11.3 ARGUMENT DESIGNATION

Argument is the real value to be assigned to a variable used in the user macro body. Argument designation, therefore, is the act of assigning real values to variables. Argument designation is of type I and type II, which can be selected as required.

(1) Argument Designation I

Argument may be designated in any address except for G, L, N, O, and P. The relationships between the argument designation addresses and the variables are as shown below.

Table 2.32 Argument Designation I

Address of Argument Designation :	Variable in User Macro Body
A	#1
B	#2
c	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
s	#19
T	#20
u	#21
v	#22
W	#23
x	#24
Y	#25
z	#26

For the address in which no argument need be designated, the command can be omitted.

Sample Program

	#1	#2	#24	#26
	↑	↑	↑	t
G65 P9100 L3 A30 B60 x34.8 Z50.2 ;				
			Argument Designation Part	

(2) Argument Designation II

A, B and C arguments and 10 sets of I, J and K arguments can be designated. I, J and K must be designated in this order. The relationships between the argument designation addresses and the variables are as shown in Table 2.33.

Table 2.33 Argument Designation II

Address of Argument Designation II	Variables in User Macro Body
A	#1
B	#2
C	#3
11	#4
J1	#5
K1	#6
12	#7
J2	#8
K2	#9
13	#10
J3	#11
K3	#12
14	#13
J4	#14
K4	#15
15	#16
J5	#17
K5	#18
16	#19
J6	#20
K6	#21
17	#22
J7	#23
K7	#24
18	#25
J8	#26
K8	#27
19	#28
J9	#29
K9	#30
I 10	#31
J10	#32
K10	#33

The suffixes 1 through 10 to I, J and K are determined by the order of the designated I, J and K combination.

For the address in which no argument need be designated, the command can be omitted.

Sample Program

```

#4 #5 #6 #7 #9
  ↑  ↑  ↑  ↑  ↑
G65 P9005 A... B... C... I... J... K... I... K... ;

```

Argument Designation Part

(3) Position of Decimal Point Argument

An argument may generally be designated with a sign and decimal point. For the designation without decimal point, the position of decimal point is as shown in Table 2.34.

Table 2.34 Position of Decimal Point Argument

Address in Argument Designation	Metric Input	Inch Input
A, C	3 (2)	3 (2)
B (Without B 3-digit option)	3 (2)	3 (2)
B (With B 3-digit option)	0	0
D, H	0	0
E, F	0 (1)	1 (2)
I, J, K	3 (2)	4 (3)
M	0	4
S, T	0	0
Q, R	3 (2)	4 (3)
u, v, w	3 (2)	4 (3)
X, Y, Z	3 (2)	4 (3)

The value shows the position of decimal point as counted from the least significant digit. The value in parentheses indicates the number of digits that follows decimal point as designated by parameter #6020D 0 = 1, D 2 = 1 for addresses E and F, and parameter #6006D 5 = 1 for the other addresses.

(4) Considerations in Argument Designation

A. Argument designation types I and II can be used concurrently. If the same variable has been duplicated, the last one is validated.

B. For both types I and II, addresses 1, J, and K should be designated in this order. The other addresses can be designated in any order.

C. In the argument designation part, negative sign and decimal point can be used regardless of the address.

D. In G65 and G66 blocks, G65 and G66 should always be specified before each argument designation. This holds true with the macro call by G code.

(e) M cannot be used for the argument designation address when using 24 pairs, or alarm will occur.

2.11.4 OVERVIEW OF USER MACRO BODY

A user macro body is programmed using the combination of the following commands.

(1) Variables

- A. Local variable (#1 through #33)
- B. Common variable (#100 through #559)
- C. System variable (#1000 through #5104)

2.11.4 OVERVIEW OF USER MACRO BODY (Cent'd)

(2) operation Commands

- A. Arithmetical operations (+, -, *, / , " ")
- B. Functional operations (SIN, COS, ROUND, . . .)

(3) Control Commands

- A. Branch command (IF <qualification > GOTO n)
- B. Repeat command (WHILE <qualification > DO m)

Using these commands, a program which requires complicated operations and conditional judgments can be written in the general format. The feature of user macro is to enable the programming of a wide range of NC functions from a simple machining cycle which is rather a subprogram to a special, complicated canned cycle, and the storing of these cycles in the machine. Described below are details of the commands mentioned above.

2.11.5 VARIABLES

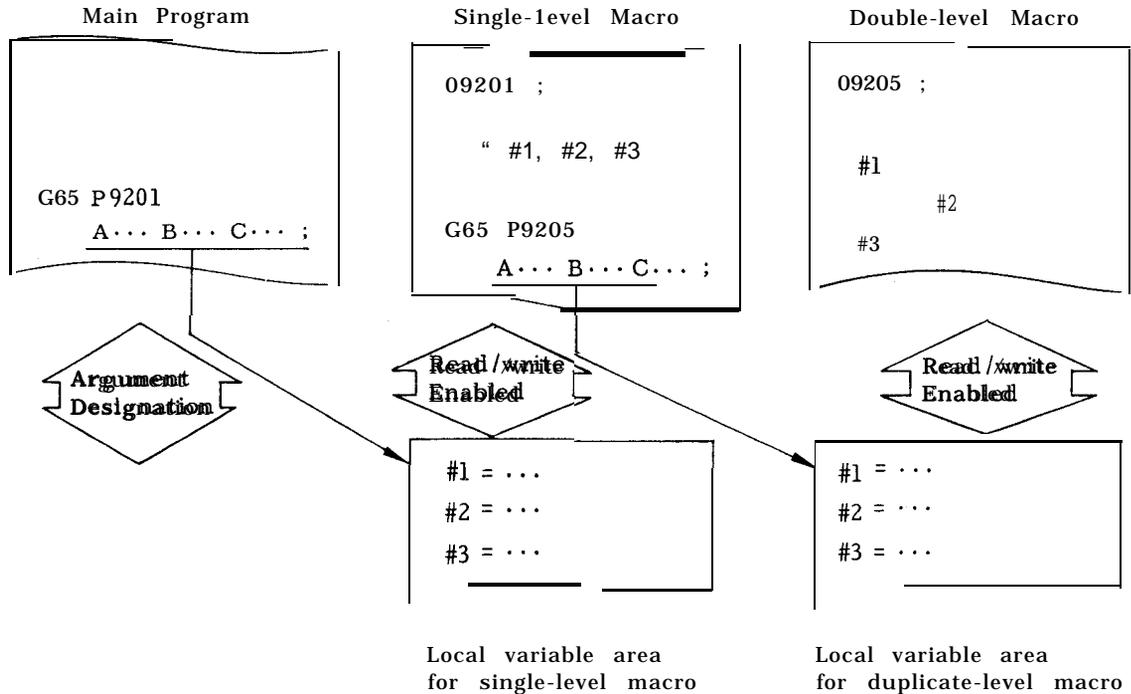
Instead of directly assigning a value to an address in a user macro body, the address can be designated by a variable. When this variable is called during execution, the corresponding value is searched from the variable area to provide the address value.

There are three types of variables: local variable, common variable, and system variable. Each is identifiable by a variable number.

To the local variables, real numbers can be assigned using the argument designation part of macro call command by G65 or G66.

(1) Local Variables (#1 through #33)

A local variable is the one that is used for each macro locally. That is, when the local variable is used, the variable area (#1 through #33) is independently allocated for each macro call, certain values are stored by argument designation, and the results of operations in macro are retained.



The variables #1, #2, #3, . . . of the same macro assume different values each time it is called. Each local variable is reset for each macro call and is registered by argument designation. The variable not designated becomes "blank." Each local variable is set to "blank" at the time of power-on and reset operations.

The local variables can also be used in the main program, as in macro of other levels.

Example

```

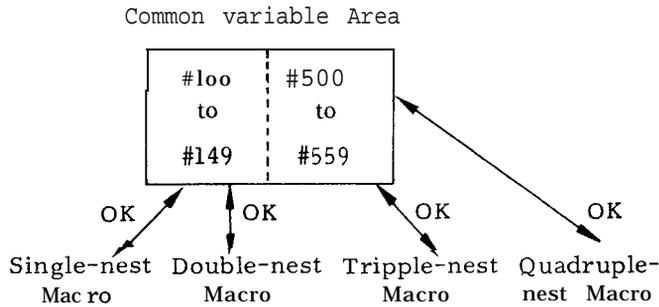
01000;           O 8000;
M98 P 8000 ;    # 1 = 1 ;
T # 1 ;        # 2 = 2 ;
T # 2 ;

M99;

M30;
    
```

(2) Common Variables (#1 00 through #1 49, #500 through #559)

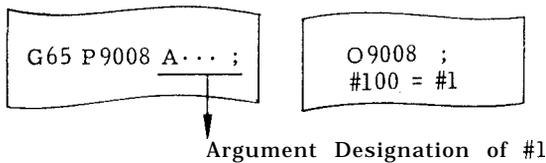
A common variable may be shared by all macros and through all macros of all nesting levels. That is, the common variable enables a macro to refer to the results obtained by another macro.



Common variables are divided into the following two types depending on clear conditions:

- A. #100 through #149: These common variables are cleared at the time of power-on and reset operations and are set to "blank." They are not cleared by reset operation in some controls if parameter #6008 D 1 is set to "1."
- B. #500 through 559 These common variables are not cleared at the time of power-on and reset operations.

The common variables are available to the user without restrictions. They cannot be designated by arguments. Indirectly, however, they can be designated as follows:



(3) System Variables

A system variable is the one whose use is unique to the system. There are following types of system variables :

- A. Interface input signals . . . #1000 through #1015, #1032†
- B. Interface output signals . . . #1100 through #1115, #1132†
- c. Tool offset amount and work coordinate-system^{sh} ft amount . . . #2001 through #2099 (or #2299†) , #2500 through #2906†

- D. Alarm message display . . . #3000
- E. Clock . . . #3001, #3002
- F. Single-block stop and auxiliary-function completion wait control . . . #3003
- G. Feed-hold, feedrate-override, and exact-stop control . . . #3004
- H. RS232C data output . . . #3100 (print out feature) .
- I. Modal information . . . #4001 through #4120
- J. Position information . . . #5001 through #5105

Note: The interface input and output signals of (A) and (B) may not be installed. Follow the specifications of the machine tool builder.

The following paragraphs describe the details of the variables mentioned above.

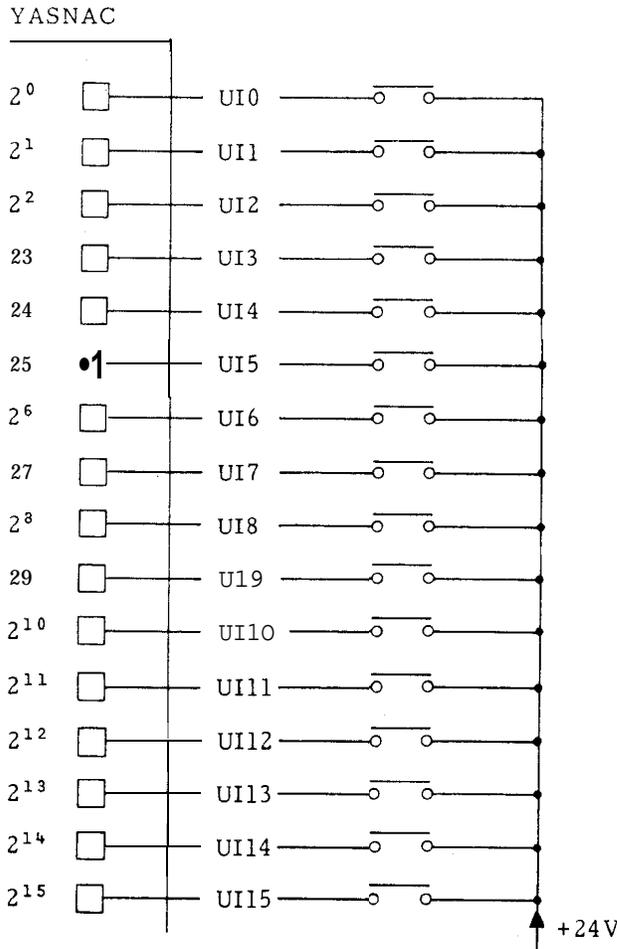
A. Interface Input Signals (#1000 through #1015, #1032) †

a. When one of system variables #1000 through #1015 is specified to the right-hand of an operational expression, the on /off state of each of user-macro-dedicated 16-point input signals is read. The relationships between the input signals and the system variables are as shown below.

#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000
U17 2 ⁷	U16 2 ⁶	U15 2 ⁵	U14 2 ⁴	U13 2 ³	U12 2 ²	U11 2 ¹	U10 2 ⁰
#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008
U115 2 ¹⁵	U114 2 ¹⁴	U113 2 ¹³	U112 2 ¹²	U111 2 ¹¹	U110 2 ¹⁰	U109 2 ⁹	U108 2 ⁸

Variable Value	Input Signal
1	Contact Closed
0	Contact Open

2.11.5 VARIABLES (Cent'd)



Each read variable is 1. 0 or 0. 0 when the associated contact is "closed" or "open" respectively, regardless of the unit system of the machine.

b. When system variable #1032 is designated, the input signals (UI0 through UI15) that consist of 16 points (16 bits) are collectively read as a decimal positive value.

$$\#1032 = \sum_{i=0}^{15} \# [1000 + i] * 2^i$$

Sample Program

```
IF [#1015 EQ 0] GOTO 100 ;
```

Bit 2¹⁵ (UI15) is read and, if it is "0," a branch is made to sequence number N100.

```
#130 = #1032 AND 255
```

Bits 2⁰ through 2⁷ (UI0 through UI7) are collectively read to be stored in common variable #130 as a decimal positive value.

Note: System variables #1000 through #1032 cannot be placed to the left-hand of operational expressions.

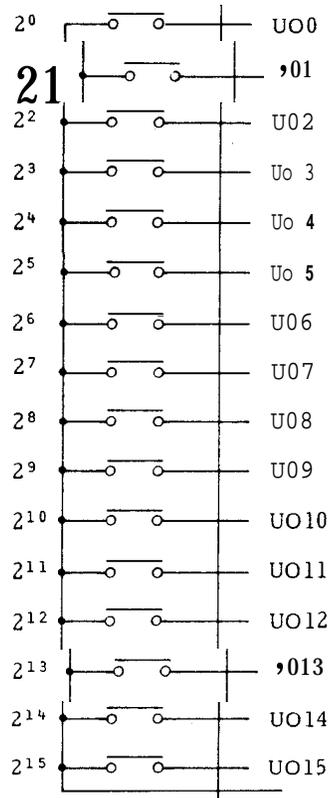
B. Interface Output Signals (#1100 through #11 15, #1132)7

a. When one of system variables #1100 through #1115 is specified to the left-hand of an operational expression, an on/off signal can be sent to each of user-macro-dedicated 16-point output signals. The relationships between the output signals and the system variables are as shown below :

#1107	#1 106	#1105	#1104	#1103	#1102	#1101	#1100
U07 2 ⁷	U06 2 ⁶	U05 2 ⁵	U04 2 ⁴	U03 2 ³	U02 2 ²	U0 1 2 ¹	U00 2 ⁰
#1115	#1114	#1113	#1112	#1111	#1110	#1 109	#1108
U15 2 ¹⁵	U14 2 ¹⁴	U13 2 ¹³	U12 2 ¹²	U11 2 ¹¹	U10 2 ¹⁰	U09 2 ⁹	U08 2 ⁸

Variable Value	Output Signal
1	Contact Closed
0	Contact Open

YASNAC



When 1.0 or 0.0 are substituted in any of #1100 through #1115, the associated output contact is output in the "closed" or "open" state.

b. When system variable #1132 is specified, the output signals (U00 through U015) that consist of 16 points (16 bits) are collectively output. At this time, the decimal positive value substituted in #1 132 is output in the form of binary 16-bit value.

$$\#1132 = \sum_{i=0}^{15} \#[1100 + i] * 2^i$$

c. With system variables # 1100 through # 1132, the value sent last is retained. When one of them is written to the right-hand of an operational expression, its value is read.

d. Considerations

When any values other than 1.0 or 0.0 are substituted into one of #1 100 through #1 115, the values are handled as follows:

“Blank” is assumed to be “O. ”

Values other than “blank” and O are assumed to be “1.”

Sample Program

#1107 = #lo ; (#lo = 1.5)

The output signal of bit 2⁷(U07) is output in the contact (closed) state.

#1132 = (#1132 AND 240) OR (#8 AND 15;)

The value of the last sending data is stored in system variables #1100 to #1132. Therefore, when these system variables are written at the right side of the equation, the values are read in.

The output signal Of bits 2⁴ through 27 (uo4 through U07) are output without change and the contents of local variable #8 are output to the output signals of bits 2⁰ through 23 (U00 through U03) .

(Decimal 240) = 11110000

(Decimal 15) = 00001111

C. Tool Offset Amount and Work Coordinate System Shift Amount (#2001 through #2299, #2500 through #2906)

a. When one of system variables #2001 through #2299 is specified to the right-hand of an operational expression, the tool offset amount can be read.

b. When one of system variables #2500 through #2906 is specified to the right-hand of an operational expression, the work coordinate system shift amount (and the external work coordinate system correction amount) can be read.

c. The relationships between the tool offset numbers and the system variables are as shown below :

Table 2.35 System Variables and Tool offset No.

System Variable	Tool Offset No.
#2001	01
#2002	02
#2098	98
#2099	99
#2100	100
#2299	299

2.11.5 VARIABLES (Cont'd)

Table 2.36 System Variables and Work Coordinate System Shift Amount

System Variable	Work Coordinate System Shift Amount	Spindle
#2500 #2501	External work coordinate system correction amount G54 (J1)	X
#2506	G59 (J1)	
#2600 #2601	External work coordinate system correction amount G54 (J1)	Y
#2606	G59 (J1)	
#2700 #2701	External work coordinate system correction amount G54 (J1)	Z
#2706	G59 (J1)	
#2800 #2801	External work coordinate system correction amount G54 (J1)	A
#2806	G59 (J1)	

Note : When "work coordinate system setting B" feature is provided, system variables listed in Table on page 105 can be applied.

(d) When one of the above system variables is specified to the left-hand of an operational expression, its value can be changed.

Sample Programs

a. #116 = #2016 ;

The contents of tool offset number 16 are substituted for common variable #116.

b. #2506 = #4 ;

The work coordinate system shift amount of G59 X-axis is erased and the contents of local variable #4 are set.

System Variables	Work coordinate System Shift Amount	Axis
# 2511	G54 J2	X
	⋮	
# 2516	G59 J2	Y
# 2611	G54 J2	
	⋮	
# 2616	G59 J2	Z
# 2711	G54 J2	
	⋮	
# 2716	G59 J2	X
# 2521	G54 J3	
# 2526	G59 J3	Y
# 2621	G54 J3	
# 2626	G59 J3	Z
# 2721	G54 J3	
# 2726	G59 J3	X
# 2531	G54 J4	
# 2536	G59 J4	Y
# 2631	G54 J4	
# 2636	G59 J4	Z
# 2731	G54 J4	
# 2736	G59 J4	X
# 2541	G54 J5	
	⋮	
# 2546	G59 J5	Y
# 2641	G54 J5	
# 2646	G59 J5	Z
# 2741	G54 J5	
# 2746	G59 J5	

D. Alarm Message Display (#3000)

When a condition to be alarmed occurs in a user macro program, specify the system variable #3000 to put the machine in the alarm state.

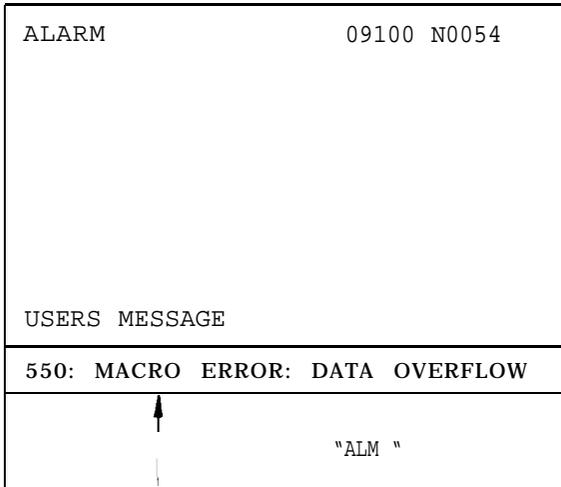
#3000 = n (<alarm message>);

Using this command, specify the alarm message (less than 32 characters) enclosed by 3-digit alarm number n and control-in and control-out. The alarm number should be three digits and be the one that is not used by the machine.

When this #3000 command is executed, "ALM" or "A/B" is displayed on the bottom of CRT screen regardless of the mode and function. Its message can be seen by the following operation :

a. Depress **ALM** function key.

The alarm number and message are displayed on the bottom of CRT screen.



Message display area and sample display

b. When **RESET** key is pressed after removal of the cause of alarm, the message display and the alarm state can be cleared.

Sample Program

```
#3000 = 550 (MACRO ERROR: DATA OVERFLOW)
```

E. Clock (#3001 , #3002)

When system variable #3001 or #3002 for clock is specified, the clock can be read.

System Variable	Type	Unit	At Power-On	Count Condition
#3001	Clock 1	1 ms	Reset to " O "	Always
#3002	Clock 2	1 s	Same as power-off time	When STL signal is on

To preset the clock, substitute the value with this system variable put at the left-hand of the expression.

Sample Program

```
#3001 = 0 ; . . . The clock is preset to value " O . "
```

Restrictions

- a. The accuracy of clock 1 is 8 ms. When 4294968000 msec has been reached, an overflow occurs, setting the clock to " O . "
- b. The accuracy of clock 2 is 8 ms. When 429496800 sec has been reached, an overflow occurs , setting the clock to "0. "

Sample Program

```
Main Program [ Argument designation of variable #1
G65 P9351 A5. ; "" Macro to wait 'or 5 seconds
```

Macro Program

```
09351 ;
#3002 = 0 ;
WHILE [#3002 NE #1] DO1 ;
END 1 ;
M99 ;
```

F. Single Block Stop and Auxiliary Function Completion Wait Control (#3003)

when the value listed in the following table is substituted in system variable #3003, the single block switch can be disabled or the next block can be entered without waiting for the checking of the finish signal (FIN) of the auxiliary function (MST).

When the finish signal is not waited for, the distribution end signal (DEN) is not transmitted. In this case, the FIN is waited for in the block with the check skip cleared. Hence, when the FIN is not waited for, don't specify the next auxiliary function.

#3003	Single Block Switch	FJN Signal
0	Valid	Waited
1	Invalid	Waited
2	Valid	Not waited
3	Invalid	Not waited

G. Feed-Hold, Feedrate-Override, and Exact-Stop Control (#3004)

When the value listed in the following table is substituted in system variable #3004, feed hold, feedrate override, and exact stop can be made valid or invalid.

2.11.5 VARIABLES (Cont'd)

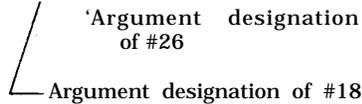
#3004	Feed Hold	Feedrate Override	Exact Stop
0	Valid	Valid	Valid
1	Invalid	Valid	Valid
2	Valid	Invalid	Valid
3	Invalid	Invalid	Valid
4	Valid	Valid	Invalid
5	Invalid	Valid	Invalid
6	Valid	Invalid	Invalid
7	Invalid	Invalid	Invalid

Sample Program

Tapping Cycle (for Incremental command)

Main Program

```
G91 ;
G65 P9601 L... R... Z... ;
```



Macro Program

```
09601 ;
#3003 = 1 ; ——— Single block switch
GOO Z#18 ; ——— invalidated
#3004 = 7 ——— Feed hold invalidated
G01 Z#26 ; ——— Feedrate override invalidated
M05 ; ——— Exact stop invalidated
M04 ;
#3004 = 0 ; ———
GOO Z-#18
M05 ;
M03 ;
#3003 = 0 ; ———
M99 ;
```

H. RS232C Data Output (#3100)

When system variable #3100 is specified, messages and NC internal data can be output to external equipment via RS232C data input/output interface. If the external equipment is a printer, the above information is printed.

a. Output of Messages

#3100 = (<Message>)

When this command is specified, the message enclosed by control-in and control-out is output via RS232C interface.

Each output message is followed by CR/LF (Carriage Return/Line Feed). Hence, when #3100 = () is specified, only CR/LF is output, which is useful in tabulating the punched data. To output message, enclose it by control-in and control-out.

b. Output of Data

#3100 = <variable >

When this command is specified, the value of the local variable, common variable, or system variable at the right-hand is output via RS232C interface as sign and decimal 9-digit data (4 after and 5 before decimal point).

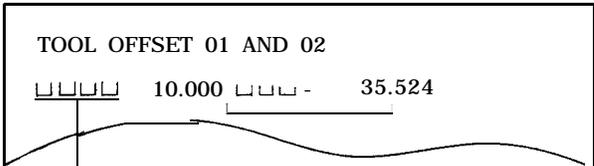
The value is rounded out to the 5th digit after the decimal point. When the value is of b digits or more before the decimal point, the asterisk is output.

c. The above output is performed when system variable #3100 is executed in the macro program. It is required, therefore, to previously attach the external equipment such as a printer via RS 232c interface and preset the parameters that use the interface.

Sample Program

```
#3100 = ( ) ; . . . Carriage return /line feed
#3100 = (TOOL oFFsET 01 AND 02):
#3100 = #2501 ; . . . = 10.000 mm
#3100 = #2502 ; .0. = -35.524 mm
#3100 = ( ) ;
```

Printout Data



Four spaces precede signed decimal.

I. Modal Information (#4001 through #4120)

When one of system variables #4001 through #4120 is specified, the modal commands that are specified up to the immediately preceding block can be known. These modal commands are sometimes called the current values of modal information commands.

System Variable	Modal Information
#400 1	G code (group 01)
to	to
#402 1	G code (group 21)
#4102	B code
#4107	D code
#4109	F code
#4111	H code
#4114	Sequence number
#4115	Program number
#4119	S code
#4120	T code

#4001 through #4120 cannot be placed to the left-hand of the operation expression.

Sample Program

Main Program

```
G65 P9602 <Argument Designation> ;
```

Macro Program

```
09602 ;
#1 = #4001 ; _____ - G codes (G00 through
GO(I X...Y... ;      G03) of 01 group are
GO1 Z...F... ;      retained.
G03 X...Y... R... ;
G00 Z... ;
G#1 ; _____ - G codes of 01 group
M99 ;               are restored.
```

J. Positional Information (#5001 through #5104)

When system variables #5001 through #5104 are specified, various positional information can be obtained. The unit of the information is millimeters or inches.

	Unit
Metric input	0.001 millimeter
Inch input	0.0001 inch
Deg input†	0.001 degree

In the user macro body, the "input unit x 10" feature is invalid.

System Variable	Positional Information	Read During Move
#5001	X-axis block end position (ABSIO)	Enabled
#5002	Y-axis block end position (ABSIO)	
#5003	Z-axis block end position (ABSIO)	
#5004	4th block end position (ABSIO)	
#5021	X-axis current position (ABSMT)	Enabled
#5022	Y-axis current position (ABSMT)	
#5023	Z-axis current position (ABSMT)	
#5024	4th current position (ABSMT)	
#5041	X-axis current position (ABSOT)	Enabled *
#5042	Y-axis current position (ABSOT)	
#5043	Z-axis current position (ABSOT)	
#5044	4th current position (ABSOT)	
#5061	X-axis skip signal position (ABS KP)	Enabled *
#5062	Y-axis skip signal position (ABS KP)	
#5063	Z-axis skip signal position (ABS KP)	
#5064	4th skip signal position (ABS KP)	
#5083	Tool length correction amount	Enabled
#5101	X-axis servo position deflection amount	Enabled *
#5102	Y-axis servo position deflection amount	
#5103	Z-axis servo position deflection amount	
#5104	4th servo position deflection amount	

* Reading of #5021 to #5024, #5041 to #5044, and #5101 to #5104, when commanded during movement, will be performed after completion of the movement of the preceding block. Therefore radius compensation is not performed.

Variable No.	Meaning
#5001 to #5004	End position of immediately preceding block (for each axis) .
#5021 to #5024	Current position of machine coordinate system (for each axis).
#5041 to #5044	Current position of POS. UNIVERSAL (for each axis) .
#5061 to #5064	Position at which G31 skip signal is turned on (for each axis) .
#5083	Valid tool length compensation amount.
#5101 to #5104	Servo position deflection amount (for each axis)

K. Variable Representation

Each variable is represented in a variable number that follows #.

a. How to designate a number direct] y:

#i (i = 1, 2, 3, 4, ...)

Sample #10
#130
#2000

b. How to designate an expression as a variable number:

[<expression>]

Sample # [#100]
[#500 + 1]
[#20/2]

In the following description, variable #i may be replaced with variable # [< expression >].

L. Variable Reference

a. The value that follows an address may be replaced by a variable.

When <address> #i or <address> -#i is specified, the value of the variable or its negative value (complement, more exactly) is made the specified value of the address.

Sample #30 = 1.0 ;
#101 = 100 ;
#102 = 200 ;
#103 = 300 ;
#140 = 250 ,
G#30 X#101 Y#102 Z-#103 F#140 ;

The above specification is equivalent to the specification below:

G01 X100. Y200. Z-300. F250 ;

b. When the F1-digit command option is provided, the F2-digit command and decimal point command are regarded as linear commands.

Program Example	System
G01X100. Y20. F1; G01X100. Y20. F1. 0;	F1-digit command Direct command Direct command Direct command
G01X100. Y20. F01; G01X100. Y20. F#1; (F5. 0 specification) #1 = 0.001 to 0.499 #1 = 1.0 to 9.0 Integers. (2.0, 3.0, 4.0...) #1 = 0.50 to 9.499 (Digits below decimal point are rounded off) #1 = 9.500 or more	No lower digit error Lower 1-digit command Lower 1-digit command Direct command
(F5. 1 specification) #1 = 0.001 to 0.499 #1 = 0.1 to 0.9 (0.2, 0.3, 0.4...) #1 = 0.050 to 0.949 (Digits of 0.01 or below are rounded off) #1 = 0.950 or more	No lower digit error Lower 1-digit command Lower 1-digit command Direct command

Notes :

(1) Address /, O and N can not refer to variables.

Sample /#8, N#100... Error.

(2) A variable number can not be replaced with a variable.

Sample ##20 ... Error.
#[#20] ... Correct.

(3) When a variable is used as address data, the values below the least significant digit are rounded .

Sample

- (i) When #1 = 45.2346
X#1... = x45.235 mm (for mm input)
- (ii) When #2 = 350.85
F#2... F351 (mm/min)
- (iii) When #3 = 5.37672
G04 P#3... = G04 P5.377 (sec)
- (iv) When #4 = 2.7236
M#4 ... = M03
G#4 ... = G03

(4) Value for each address should not exceed the maximum programmable value.

(5) The value that follows an address may be replaced with < expression > .

(6) The constant without decimal point enclosed in brackets [] is assumed to have a decimal point at its end.

M. Undefined Variable

The value of an undefined variable is assumed to be "blank." An undefined variable occurs in the following situations:

a. The local variable for which argument designation was not performed in macro call command.

b. Common variables #100 through #149 at the time of power-on and reset operations.

c. The local variables and common variables for which the values were not written from MDI panel.

2.11.5 VARIABLES (Cent'd)

Item	Meaning of (Blank)
1	Concept of "#0" . #0 defined as variables of blank . Commanding #0 at the left-hand side of the equation causes alarm.
2	Variable <blank> is commanded in the replacement equation. . Where #2 is blank command #3=#2; means #3= <blank>.
3	(1) Variable <blank> is commanded in the part program (2) On F commands only . Where #2 is <blank> command GOO X #2: is equivalent to COO (Address is ignored.) F figure 1, F figure 2: commands Numeral 1 and numeral 2 include Indirect commands. #1 #2 Blank Full - Blank Blank Blank - Numeral 2 Full Blank - Numeral 1 Full Full - Numeral 2.. . Note
4	Variable <blank> is commanded in the condition of EQ and NE. . Where #2 is <blank> #3 is 0 ① Condition "IF #3 NE #2" is not established. ② Condition "IF #3 EQ #2" is established,
5	Others #3=#[#0+#0] . . . #3 is blank. #3=#2 * #0; } #3=#0 + #0; } In these com- #3=#0/#0; } mands, #3=0, #3=5*#0 ; } #3=2-#0; means #3=2 #3=5/ #0; causes alarm. "Blank" in the replacement other than described above is treated as "0." . Condition IF#3 GE#2 is established when #2 and #3 are <blank>, or #2 is 0 and #3 is <blank>. . Condition IF #3LT #2 is not established when #2 and #3 are <blank>, or #2 is <blank>, and #3=0. Blanks in conditions except EQ and NE are treated as 0.

Note: Parameter setting can be made to select whether to cause double address error or to enable numeral 2, when both numeral 1 and numeral 2 are full.
 Setting parameter #6073 D3 1: Numeral 2 is effective
 0: Double address error

2.11.6 OPERATION COMMANDS

Various operations can be performed between variables and between variables and constants. The operation expression is represented in the form of #i □ <expression>, in which <expression> is a general arithmetic operational expression produced by combining variables and constants with operators and functions. The available operators and functions are as follows. Instead of #j and #k, constants may be used.

Note: <expression> must be commanded at the right-hand side of the equation,

(1) Variable Definition and Replacement

#i = #j . . . Definition, replacement.
 #i = #[#j + #k] . . . Indirect specification

(2) Add-Type Operations

#i = #j + #k . . . Sum.
 #i = #j - #k . . . Difference.
 #i = #j OR #k . . . Logical sum (for each of 32 bits) .
 #i = #j XOR #k . . . Exclusive logical sum (for each of 32 bits) .

(3) Multiply-Type Operations

#i = #j * #k . . . Product.
 #i □ #j / #k . . . Quotient.
 #i □ #j AND #k . . . Logical product (for each of 32 bits) .

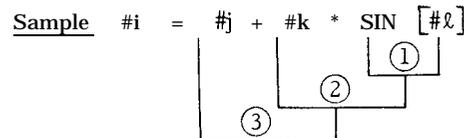
Note: In OR, XOR, or AND operation, the variable value (or constant) is converted into the binary 32-bit equivalent and the operation is performed on each bit.

(4) Functions

#i □ ,SIN [#j] . . . Sine (in degrees) .
 #i = COS [#j] . . . Cosine (in degrees) .
 #i = TAN [#j] . . . Tangent (in degrees) .
 #i □ ATAN [#j] / [#k] . . . Arctangent
 #i = SQRT [#j] . . . Square root.
 #i = ABS [#j] . . . Absolute value.
 #i = BIN [#j] . . . Convert from BCD .
 #i - BCD [#j] . . . Convert into BCD.
 #i □ ROUND [#j] . . . Produce integer by rounding
 #i = FIX [#j] . . . Truncate the fractions.
 #i = FUP [#j] . . . Raise the fractions to a unit.

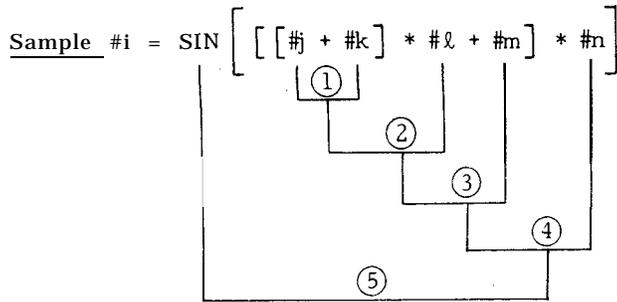
(5) Combinations of operations

The above operations and functions may be used in combinations. A functional operation is performed first. Then, a multiply-type operation is performed. An add-type operation is performed last.



(6) Change of operational order by []

Priority can be given to an operation by enclosing it in brackets []. Up to quintuple (five-fold) nesting of brackets is permitted including those of functional operations.



(7) Considerations for Operational Commands

A. The constant without decimal point used in <expression> is assumed to have a decimal point at its end?

B. When used in conditional expression IF or WHILE, function ROUND truncates the fractions.

c. When used in address data, function ROUND rounds the part below the least significant digit.

Sample (a)

#10 = 12.3758

When the least significant digit of address X is 0.001 mm, the following command

GOO x [ROUND [#10]];
means

GOO x12.376 ;
because 8 of 12.3758 is rounded.

This command is also equivalent to

GOO X#10 ;

Usually, ROUND is not used as mentioned above; it is used as shown below:

Sample (b)

When ROUND is used as follows, the program returns to the start point correctly:

- ① #10 = 12.3758 ; START POINT
- ② #11 = 13.1236 ; ○ #10 ○ #11 ○
- ③ G91 GOO X#10 ; └──────────┘
- ④ G01 X#11 F... ; RETRACTION
- ⑤ GOO x- [ROUND [#10] + ROUND [#n]];

This is because the data of #10 and #11 in ③ and ④ blocks are substantially rounded before being executed.

If ⑤ block is

⑤ GOO X- [#10 + #11] ;

then, the movement is made by the following amount:

$$\begin{aligned}
 X - [\#10 + \#n] &= X - [12.3758 + 13.1236] \\
 &= X - [25.4994] \\
 &= X - [25.499]
 \end{aligned}$$

On the other hand, block movement of

③+④ is

$$\begin{aligned}
 X\#10 + X\#11 &= x12.376 + x13.124 \\
 &= x25.500
 \end{aligned}$$

The program of ⑤ is not correct.

(8) Operational Errors

The data format and the operational errors in the user macros are as follows:

A. Data Format

The numeric data handled in user macros are of the floating point format.

$$M * 2E$$

where, M is sign + data 52-bit binary,
E is sign + data 10-bit binary.

B. operational Errors

Each time an operation is performed, the following error is caused and is accumulated. The number of significant digits is 15 to 16, which compensates the error sufficiently.

2.11.7 CONTROL COMMANDS

The commands which control the flow of micro-program are of the following two types:

A. Branch Command . . . IF [<conditional expression>] GOTO n ;

B. Repeat Command . . . WHITE [<conditional expression>] DO m ;

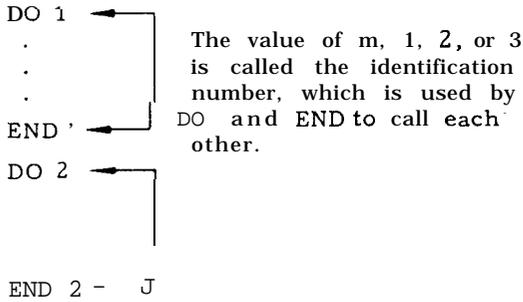
(1) Branch Command

A. IF [<conditional expression>] GOTO n ;

If <conditional expression> of this command is established, a branch is made to the block of sequence number n within the same program. When a variable or an expression is used for n, the branch destination can be changed. If the condition is not satisfied, the program proceeds to the next block.

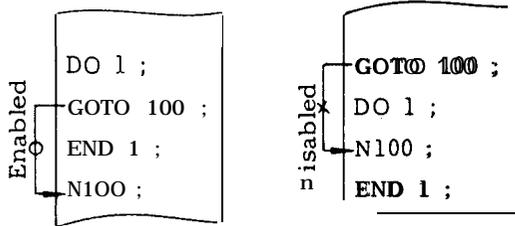
Notes :

- DO m should be specified before END m.
- m of DO m and END m should have the same value. However, only 1, 2, or 3 can be specified in m.

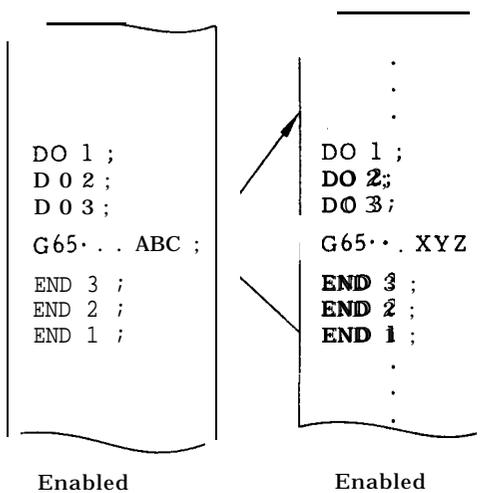


3. The same identification number can be used repeatedly except where repeat ranges overlap.

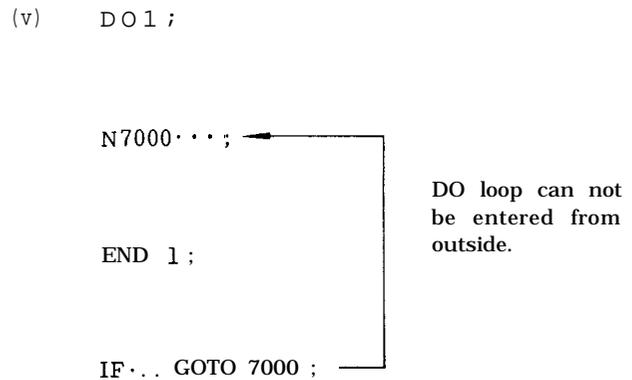
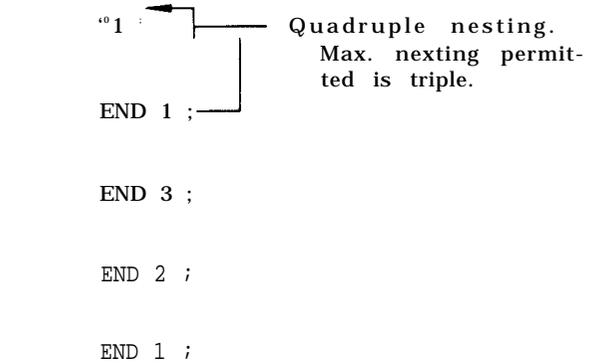
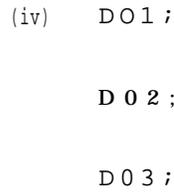
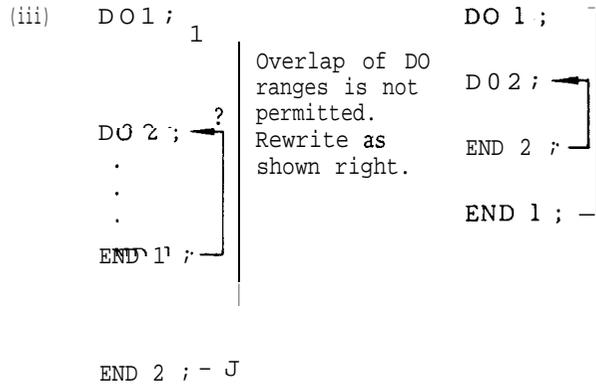
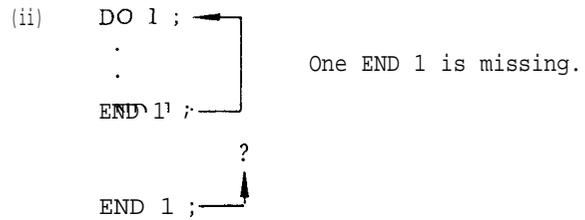
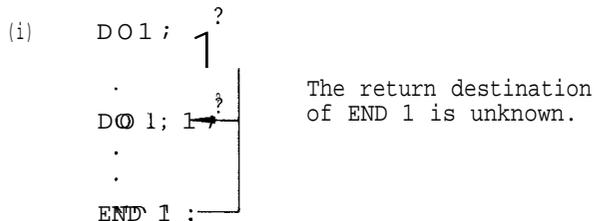
4. GOTO n enables to get out of DO loop but it does not enable to get into DO loop as shown below :



5. Triple DO-loop nesting is permitted for each micro program.



6. The codings shown below cause an error:



2.11.8 REGISTRATION OF USER MACROS

(1) How to Make Registration of User Macros

The registration and edit of user macro bodies (macro programs) are performed in the same manner as usual part programs and subprograms. Hence, there is no program size restriction that applies to the user macro body only. Part programs, subprograms, and macro programs can be stored together in the part program memory to its full capacity.

(2) Classification of Program Numbers

The program numbers are classified into the following:

Program No.	Classification
01 to 07999	These programs can be registered, erased, or edited without restrictions.
08000 to 08999	When D4 of #6004 is set to 1, the registration, erase, and edit of programs are disabled.
09000 to 09999	When D7 of #6021 is set to 1, the registration, erase, and edit of programs are disabled.

2.11.9 DISPLAY AND WRITE OF LOCAL VARIABLES AND COMMON VARIABLES

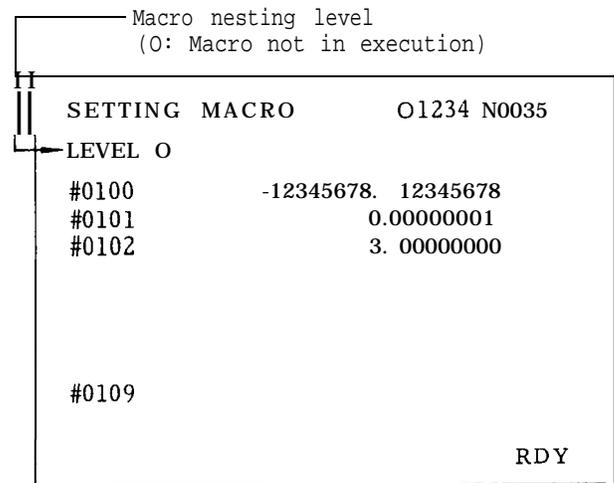
Local variables (#1 through #33) and common variables (#100 through #149, #500 through #559) can be displayed and written by the following operations:

(1) Display Operations

A. Display of Variables

a. Depress **SET** function key. Mode select position can be provided anywhere.

b. Key-in the variable number and press **m**
 ‘ey’ **r** **CURSOR** ‘ey’ However, # need not be keyed-in. Ten sets of variable numbers including the specified variable number and their data are displayed. The data is displayed in the signed 8-digit integer part and the 8-digit fraction part.



Sample Display of Common Variables

c. Depress **PAGE** key or **PAGE** key, and the display can be scrolled up or down.

B. Remarks

a. Common variables can always be displayed for review.

b. For local variables, those of the macro currently executed are displayed. Consequently, when a macro of a nesting level is in execution, the local variables of the macros of the other nesting levels cannot be seen. The local variables after completion of execution are all reset to "blank."

(2) Write Operations

A. Write of Values to Variables

a. Depress **SET** function key. Mode select position may be provided anywhere.

b. Key-in the variable number to be written. Press **CURSOR** key. However, # need not be keyed-in. The keyed-in variable number is specified and the cursor is positioned to it.

c. Key-in the value to be written. Press WR key. The keyed-in value is stored as the data of the variable number with the cursor positioned.

d. Depress **CURSOR** key **CURSOR** key
PAGE key or **PAGE** key to move the cursor.

e. Repeat operations in c. and d. to write the values to the desired variables.

B. Remarks

- a. Common variables can always be changed.
- b. Local variables can not be written at any time other than when a macro is in execution. Any attempt to do so is invalidated. However, rewriting of local variables during macro execution may cause an unexpected failure. Before attempting the rewriting, stop the machine operation by single stop function and check to see if it is safe to rewrite variables.
- c. The written local variables and common variables #100 through # 149 are reset to "blank" by the reset operation or the power-on operation.

2.11.10 CONSIDERATIONS AND REMARKS FOR USER MACROS

(1) Summary of Restrictions

A. Available Variables

- #1 through #33 ... Local variables.
- #100 through #149 } Common variables.
- #500 through #559 }
- System variables

B. Available Variable Values

Maximum value ... $\pm 10^{+308}$
 Minimum value . . . $\cdot 10^{-308}$

C. Constant Values Usable in < Expression >

\pm (8 digits above decimal point) . (7 digits below decimal points) .

Sample Maximum value 399999999.9999999
 Minimum value ± 0.0000001

D. Operational Accuracy

Decimal 15 digits significant.

E. Macro Call Maximum Nesting Level

Quadruple (four-hold) .

F. Maximum Nesting Level of Repeat Command

Triple (three-hold) for each macro.

G. Repeat Command (DO) Identifier m

m = 1, 2, and 3.

H. Maximum Nesting Level of Brackets

Quintuple (five-hold) .

(2) Differences Between User Macro and Subprogram

A. User macros G65 and G66 allow argument designation but the subprogram (M98) does not.

B. The user macro directly branches to the user macro body without executing any command that was specified in G65 or G66 block and has no relationship with the macro. With the subprogram, however, a branch is performed after the execution of the command (if any) other than P and L in M98 block.

C. The maximum nesting level of user macro is quadruple including G 65 and G 66 calls. That of subprograms is also quadruple but separately.

D. If user macros are specified via the MDI during automatic operation, the maximum nesting level is restricted to quadruple. With subprograms, up to four levels of nesting are permitted in tape mode or memory mode, or separately in MDI mode.

(3) Relationship with MD I Operation

A. MDI writing permits the macro call and the execution of the called macro.

B. MDI writing does not permit or execute macro body commands such as operational commands and control commands.

C. When a macro program being executed is stopped by the single block stop function, any MDI writing command not related to the macro can be specified and executed.

(4) Relationship with Address Search

The address search function is not permitted to search for the sequence numbers in the user macro body.

(5) Relationship with single Block Switch

A. The operational command and control command blocks do not single-block stop if the single block switch is turned on. This switch is enabled for the other macro program blocks.

B. However, when setting number #6004D 1 = 1, the single block switch is enabled for the operational command and control command.

C. System variable #3003 (for the control of single block stop, see para. 2.11.5, F on page 103) and setting #6004 D 1 mentioned above operate as shown below :

2.11.10 CONSIDERATIONS AND REMARKS FOR USER MACROS (Cent'd)

Setting #6004	System Variable #3003	When Single Block Switch is on
D1 = 0	= 1 or 3	Operational command, control command, and general commands do not stop.
D1 = 0	= 0 or 2	Operational command and control command do not stop. General commands stop.
D1 = 1	= 1 or 3	Operational command, control command, and general commands do not stop.
D1 = 1	= 0 or 2	Operational command, control command, and general commands stop.

(6) Relationship with operational **Block'Skip**

The slash "/" character used in the right-hand of an operational expression or in brackets is assumed to be the operator for quotient. It does not mean the optional skip.

(7) Setting and Parameter of Program Number Classification

A. Disabling of Program Registration, Erase, and Edit

The following setting is permitted to protect the registered user macros and subprograms from inadvertent destruction:

Setting Number

#6004

D4 = 1 ... The programs of program numbers #8000 through #8999 are disabled for registration, erase, and edit.

D4 = 0 ... Registration, erase, and edit are enabled.

Parameter Number

#6021

D7 = 1 ... The programs of program numbers #9000 through #9999 are disabled for registration, erase, and edit.

D7 = 0 ... Registration, erase, and edit are enabled.

(8) Effects of Reset Operation

A. A reset operation resets all local variables (#1 through #33) and part of common variables (#100 through #149) to "blank."

B. A reset operation resets the user-macro multiple call state and the multiple DO loop state, making the program pointer return to the program head.

(9) Special Codes Usable in User Macro Body

A. The special codes listed below may be used in the user macro body:

Code	Use	EIA Code					ISO Code										
		8	7	6	5	4	3	2	1	8	7	6	5	4	3	2	1
SP	For comment																
*	(
*)																
+	Add																
-	Subtract																
:	For comment																
/	Divide																
#	Variable																
*	Multiply																
*	=																
*	[
*]																
\$	For comment																
@																	
?																	
.	Decimal point																

Notes:

1. For the hole pattern of EIA code of the character attached with an asterisk, the pattern shown above is standard. However, other patterns may be specified by using the following parameters :

#6110 ... [} Hole pattern setting parameter
#6111 ... 1	
#6112 ... *	
#6113 ... =	
#6114 ... (
#6115 ...)	

Read the desired hole pattern in the binary value, convert it into the decimal equivalent, and set the parameter. For example, the hole pattern shown below is set as "152":

8	7	6	5	4	0	3	2	1
○			○	○	○			

When the value of the parameter is "0," the hole pattern listed in the above table is provided.

2. When the codes shown below are output from the NC unit for punch-out or other purposes, the upper code (UC) or lower code (LC) is output immediately before.

- a. Codes preceded by UC . . . #, +, \$, ?, *, =
- b. Code preceded by LC . . . @.
- c. Codes preceded by UC only at parameter designation . . . (,).

2.11.11 ALARM NUMBER OF USER MACROS

Shown below are the user-macro-associated alarms and their causes.

Code	Cause
105	<p>MACRO ERROR (CONSTANT)</p> <p>The number of constants is in excess of the specified range.</p>
106	<p>MACRO ERROR</p> <p>There are too many G67 cancel codes.</p>
107	<p>MACRO ERROR (FORMAT)</p> <p>A format other than expression has an error.</p>
108	<p>MACRO ERROR (UNDEFIN #NO)</p> <p>The value not defined as a variable number is designated.</p>
109	<p>MACRO ERROR (#NO NOT LEFT)</p> <p>The variable of assignment statement is the one that is disabled for assignment.</p>
110	<p>MACRO ERROR ([] 5 LIMIT)</p> <p>The bracket nesting level is in excess of the upper limit (5) .</p>
111	<p>MACRO ERROR (MOVE G66 - M99)</p> <p>A move command is specified in the macro end command M99 called by G66.</p>

Code	Cause
112	<p>MACRO ERROR (5)</p> <p>The macro call nesting level is in excess of the upper limit (4) .</p>
113	
114	<p>MACRO ERROR (DO FORMAT)</p> <p>DO and END are not paired.</p>
115	<p>MACRO ERROR ([] UNMATCH)</p> <p>The format of <expression> has an error.</p>
116	<p>MACRO ERROR (DO - END NO.)</p> <p>DO m is not in the range of $1 \leq m \leq 3$.</p>
117	
118	<p>MACRO ERROR (GOTO N)</p> <p>GOTO n is not in the range of $0 \leq n < 9999$.</p>

2.11.12 EXERCISES OF USER MACRO

(1) Canned-cycle G82

T (Teacher) : We have discussed many complicated rules you have to understand to write user macros. Now, let's create some user macros as exercises. Let's take G82 Spot Facing Cycle of canned cycles for example, because it is a simple operation.

S (Student) : Where shall we start?

T: An example of usual G82 command takes the following format:

```

(Pi)
G91 ; (... Incremental Designation)
G82 X100. Y50. R-80. Z-40. P3. O F250 ;

```

This command is divided into the following and executed within the NC unit:

2.11.12 EXERCISES OF USER MACRO (Cent'd)

```

G91 ;
① GOO X100. Y50 ;
② GOO Z-80. ;
③ GO1 Z-40 F250 ;
④ G04 P3. ; (Dwell)
⑤ G00 Z-((-80) + (-40)) ;
        
```

First , these moving distances may all be converted into variables .

s: They are local variables #1 through #33, aren't they? But which type of local variable?

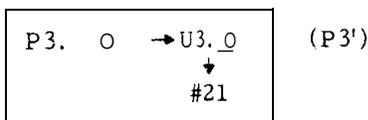
T: Type 1 for small number of variables. This type allows the use of X, Y and Z and therefore makes the argument designation easier to understand.

S: OK. When type 1 is used, we have the following g variables.

X100.	Y50.	R-80.	Z-40.	P3.	F250	(P3)
↓	↓	↓	↓	↓	↓	
#24	#25	#18	#26	?	#9	

Address P of dwell time cannot be used for argument designation, can it?

T: No. Use some other address. Then, write the address U instead of P



Using these variables, rewrite the former program (P2) .

s: OK.

```

G91 ;
① GOO X#24 Y#25 ;
② GOO Z#18 ;
③ GO1 Z#26 F#9 ;
④ G04 P#21 ;
⑤ GOO Z- [#18 + #26] ;
        
```

Is this all right?

T: Sorry, it isn't right. You have forgotten to specify something in ⑤, haven't you?

s: Oh, I should have specified " ROUND. " (P5)

```
⑤ GOO Z- [ ROUND [#18] + ROUND [#26];
```

Is this good?

T: Yes. Write as follows after the above, and we have a complete user macro body.

```
⑥ M99 ;
```

s: That's easy.

T: Then, using G65, create this macro call and the user macro body in the complete formats.

s: Let me try it.

The macro call command is as follows:

```

G91 ;
G65 P9082 X100. Y50. R-80. z-40.
U3.0 F250.;
        
```

The user macro body is as follows:

```

09082 ;
GOO X#24 Y#25 ;
COO Z#18 ;
GO1 Z#26 F#9 ;
G04 P#21 ;
GOO Z- [ROUND [#18] + ROUND [#26] ] ;
M99 ;
        
```

Program number "09082" of the user macro is arbitrary.

T: That looks OK .

s: I think something is wrong. With this program, I have to specify points R and Z every time !

T: That's true. With a usual canned cycle, when points R and Z have been specified once , their values are retained.

s: Do you have any trick to overcome this inconvenience ?

T: I do. In such a case, common variables help. Using common variables, write the macro to designate the position of points R and Z. u and F may also be used for the same purpose.

s: I've got it! Now, I divide the macro body into two parts as follows:

(P8)

```
09000 ;
#lo = #18 ;
#101 = #26 ;
#lo2 = #21 ;
#lo3 = #9 ;
M99 ;
```

(P9)

```
09082 ;
GOO X#24 Y#25 ;
GOO Z#100 ;
GO1 Z#101 F103 ;
G04 P#102
G01 z- [ROUND [#100] + ROUND [#101]] ;
M99 ;
```

and I write macro call as follows:

(P10)

```
G91 ;
G65 P9000 R-80. z-40. U3. O F250. ;
G65 P9082 x100. Y50. ;
G65 P9082 X...Y... ;
```

T: Very good.

s: Wait a minute! This canned cycle always returns to the initial point.

T: I also overlooked it. A program would be useless if it is not able to designate the initial point return (G98) and point-R return (G99) . To solve this problem, use the system variable called " current value of modal information command" to know which state, G98 or G99, is provided, and change the specification of the tool return destination,

s: G98 and G99 belong to group "IO. " So, I have to use system variable #4010. Is it right ?

T: Yes. Then, using "IF . . . GO TO . . ." command, change the specification of the tool return destination.

There is one more point to be improved. Write the program so that the group " 01" G codes before execution may be retained after the execution of this macro. Group "01" G codes include GOO through G03.

s: It looks difficult . . .

What would you say to the following program?

T: Well done !

(P11)

```
Macro Call Program

G91 G99;
G65 P9000 R-80. Z-40. U3. O F250;
G65 P9082 X100. Y50. ;
G65P9082X.. .Y. .;

G98;
G65P9082 X.. .Y. .;
```

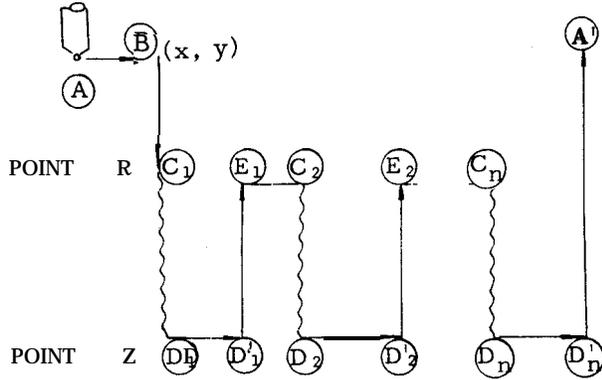
User Macro Body

(P12)

```
09000;
#100 = #18;
#101 = #26;
#lo2 = #21;
#lo3 = #9;
#lo4 = 0
-----

09082;
#lo4 = #lo4 + 1
#1 = #4001; . . . G0 to G3
#2 = #4010; . . . G98/G99
GOO X#24 Y#25;
IF[ #104 NE1] GO TO 1;
IF[ #2 EQ 99] GO TO 2;
GOO Z#100;
N1 GO1 Z#101 F#103;
G04 P#102;
IF[ #2 EQ 98] GO TO 2;
GOO 7-[ #101]
GOTO 3;
N2 GOO Z-[ ROUND [ #100] + ROUND [ #101]] ;
N3 G#1; . . . Restore G Code
M99;
```

2.11.12 EXERCISES OF USER MACRO (Cent'd)



```

09000 (G82 INITIAL SET)
#100 = #18 . . . POINT R
#101 = #26 . . . POINT Z
#102 = #21 . . . u (DWELL)
#103 = #9 . . . F CUTTING
#104 = 0
    
```

```

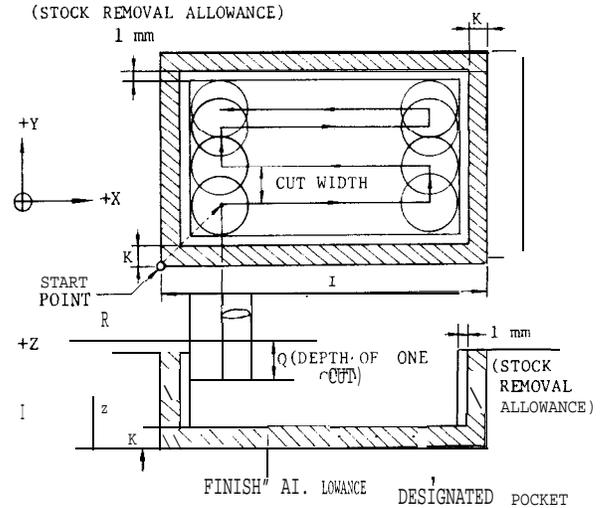
O9082
CANNED CYCLES BY G82
    
```

```

graph TD
    Start([Start]) --> GCode[G CODE READING SET  
#104 = #104 + 1  
#1 = #4001 ..... G0 - G3  
#2 = #4010 ..... G98/G99]
    GCode --> PosB[POSITIONNING TO B, C2, ... Cn  
G00 X #24 Y #25]
    PosB --> Dec1{#104 ≠ 1}
    Dec1 -- Yes --> PosC[POSITIONING TO POINT C  
G00 Z #100]
    Dec1 -- No --> PosC
    PosC --> Cut[CUTTING, DWELL AT POINT C → D  
G01 Z #101 F #103  
G04 P #102]
    Cut --> G98{G98/G99}
    G98 -- G98 --> PosA[POSITIONING TO POINT A'  
G00 Z - [ROUND (#100) + ROUND (#101)]]
    G98 -- G99 --> PosE[POSITIONING TO POINT E  
G00 Z - (#101)]
    PosA --> Restore[RESTORE OF G CODE  
G #1]
    PosE --> Restore
    Restore --> Jump[JUMP TO THE MAIN PROGRAM  
M99]
    
```

(2) Pocket Mill

A. The cycle for the pocket machining shown below is created by user macros as follow:



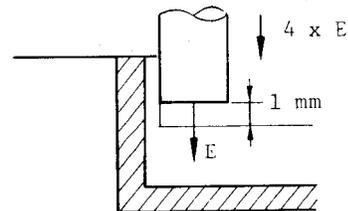
B. Macro Call Command

```

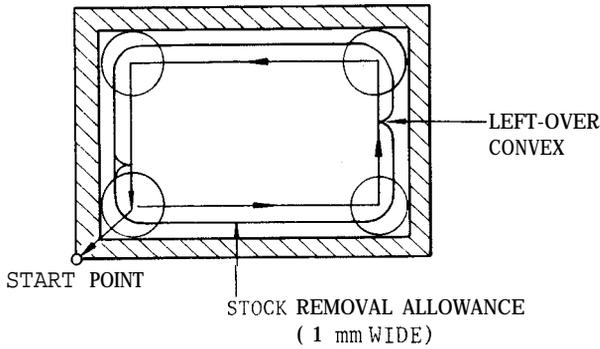
G65 P9061 X...Y... Z... R... I... J... K...
      T... Q... D... F... E... ;
    
```

where,

- X, Y: The absolute coordinate values of the start point (the lower left position of the pocket) .
- Z: The absolute position of the bottom of the pocket.
- R: The absolute position of rapid traverse tool return.
- I, J: X- and Y-axis lengths of the pocket (unsigned).
- K: Finish allowance (left-over allowance, unsigned) . Default value is 0.
- T: cut width rate (designated in %) .
Cut width = tool radius x T/100
- Q: Z-axis cut depth for each time (unsigned) .
- D: Tool offset number.
- F: Feedrate on XY-plane.
- E: Feedrate at Z-axis cut.
(Tool is fed 4 times as fast as E up to the point 1 mm to the preceding cut bottom.)



The stock removal allowance (1 mm) inside the finish allowance is all cut by a single operation in the final process as shown below. Then, the tool returns to the start point, completing the cycle.



C. User Macro Body

```

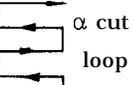
09061 ;
#10 = # [2000 + #7]; ... Tool radius
#11 = #6 + 1.0 + #10 ;
#12 = #5 - 2 * #11 ;
#13 = 2 * #10 * #20 / 100 ; ... Cut width
#14 = FUP [#12 / #13] ; ... X-axis cut count: -1
#27 = #24 + #11 ; ... X, Y coordinates of
#28 = #25 + #11 ; ... } machining start point
#29 = #26 + #6 ; ... Z-axis coordinates of
                cut bottom
#30 = #24 + #4 - #11 ;
#15 = #4003 ; ... Read of G90/G91
G90 ; ... Absolute command
GOO X#27 Y#28 ;
GOO Z#18 ;
#32 = #18 ; ... #32: Cut bottom in execution
DO 1 ;
#32 = #32 - #17 ;
IF [#32 GT #29] GOTO 1 ;
#32 = #29 ;
N1 GO1 Z#32 F#8 ;
GO1 X#30 F#9 ;
#33 = 1 ;

```

```

WHILE [#33 LE #14] DO 2 ;
IF [#33 EQ #14] GOTO 2 ;
GO1 Y [#28 + #33 * #13] F#9 ;
GOTO 3:
N2 GO1 Y [#25 + #5 - #n] ;
N3 IF [#33 AND 1 EQ 0] GOTO 4 ;
GO1 X#27 ;
GOTO 5 ;
N4 GO1 X#30 ;
N5 #33 = #33 + 1 ;
END 2 ;
GOO Z#18 ;
IF [#32 LE #29] GOTO 6 ;
GOO X#27 Y#28 ;
GO1 Z [#32 + 1.0] F [4 * #8] ;
END 1 ;
N6 #11 = #11 - 1.0 ;
#27 = #27 - 1.0 ;
#28 = #28 - 1.0 ;
#30 = #30 + 1.0 ;
#31 = #25 + #5 - #11
GOO X#27 Y#28 ;
GO1 Z#32 F#8 ;
GO1 X#30 F#9 ;
    Y#31 ;
    X#27 ;
    Y#28 ;
GOO Z#18 ;
GOO X#24 Y#25 ; ... Return to start point
G#15 ; ... Restore of G90/G91
M99 ;

```



Stock removal cycle

2.12 SOLID TAP FUNCTION †

Solid tap functions can be optionally added to YASNAC J50M.

These functions are performed by completely synchronizing the spindle revolution and Z-axis feed for tapping. They eliminate the need for a floating chuck and ensure tapping operation at high speed and high accuracy. Conventional tapping can also be specified by the program.

2.12.1 CONDITIONS FOR ADDING SOLID TAP FUNCTIONS

The following conditions are required for adding solid tap functions to YASNAC J50M.

- (1) S5-digit analog output is provided as an option for YASNAC J50M.
- (2) Yaskawa VS-626VM3 is used for spindle drives.
- (3) PG is provided to the spindle or spindle motor and the gear ratio used for solid tap has been fixed (best performance is ensured if the spindle and motor are directly connected) .

2.12.2 DESCRIPTION ON PROGRAMMING

2.12.2.1 G CODE TO SPECIFY SOLID TAP MODE

The following G codes are specified to classify tapping by solid tap function or conventional tapping.

(1) G93—Solid Tap Mode

The tap cycle (G84/G74 commands) , as mentioned below, performs the solid tap. It is fed by command for feedrate per revolution. This mode is only for solid tap.

(2) G94—Solid Tap Mode Cancel

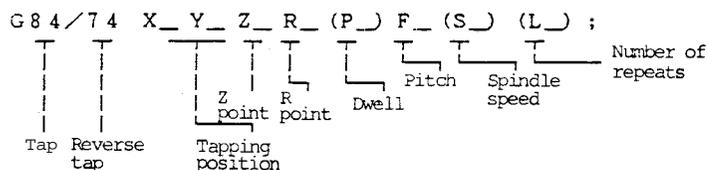
This cancels the solid tap mode and changes to the conventional mode.

The following tap cycle performs conventional tapping. It is fed by command for feedrate per minute.

Note: G93/G94 is modal with G code of 05 group. G94 is selected at the time of power-on/resetting.

2.12 .2.2 SOLID TAP CYCLE COMMAND

The solid tap can be made by specifying as below after G93 specification.



The format and contents other than F are the same as conventional tap. Solid tap is specified by pitch of Z-axis per spindle revolution (mm/rev. , inch/rev.).

Program Unit and Programmable Range of F

Input (mm) l=0.0001 mm/rev.
Range 0.0001 to 200.0000 mm/rev.

Input (in.) l=0.000001 inch/rev.
Range 0.000001 to 7.874015 inch/rev.

Programmable Range of S

l=1 rev.
Range 1 to 4500 rev.

Note: FX S cannot specify a program exceeding 24000.0 mm/min., 944.8 inch/rein.

2.12.2.3 OPERATION

(1) G93 Command

If G93 command is executed, the spindle stops, the position control loop is formed to the spindle and the solid tap mode is selected.

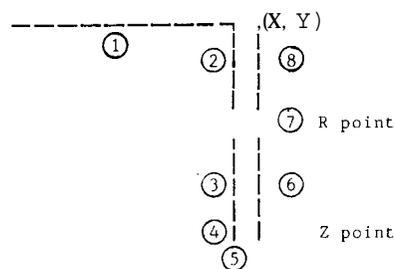
The solid tap mode is also selected by another way, namely, by spindle indexing and positioning to the home position, after spindle stop by parameter (#6055 D1) . However, the spindle can be indexed only at the time when feedback pulse for one spindle revolution is 4096 pulses (#6061 * #6449=4096) .

This mode is reset by G94.

(2) G84/74 Command

a. G84

- ① Positioning to the position specified by X and Y.
- ② Positioning to R point.
- ③ Spindle runs forward and is fed to Z point. At the start of this block, it checks that error pulse of spindle and Z-axis enters within specification-Error Detect ON.
- ④ Spindle stops; deviation pulse of spindle and Z-axis is checked here.
- ⑤ Dwell, if there is P command.
- ⑥ Spindle runs reverse and is fed to R point.
- ⑦ Spindle stops.
- ⑧ Positioning to the initial point (in case of G98) .



Linear form acceleration /deceleration is automatically applied to the spindle movement (the constant set by #6291) .

b. G74

The same operation as G84 except for the following points:

- ③ in G84 makes spindle reverse running, and ⑥ in G84 makes spindle forward running in case of G74.

(3) G94 Command

If G94 command is executed, the solid tap mode in G93 is cancelled. This mode selects the conventional tap operation if specifying G84/G74.

2.12 .2.4 PROGRAMMING EXAMPLE

```
G93; --- Solid tap mode
G84X100, Y100. Z-50. R-20. F1. S3000 ;
--- Solid tap to position (100. , 100. )
X200. Y200. ; --- Solid tap to position (200. , 200. )
X10. Y10. ; --- Solid tap to position (10. , 10. )
G80; --- Canned cycle cancel
G94; --- Solid tap mode cancel
```

2.12.2.5 PRECAUTIONS

(1) G93

- ① The codes that can be specified to G93 block are G93, S, F and N codes only. If the another code is specified, it causes an alarm (090).
- ② Specify G code in 01 group at GO(I or GO1 status to specify G93. If it is specified by a mode other than above, it causes an alarm (090).
- ③ In G93 mode (solid tap mode), S code is regarded as S command for solid tap.
- ④ In G93 mode, the G codes other than below cannot be specified, or it causes an alarm (090).

Specifiable G Codes

GOO, 01, 04, 70, 71, 72, 74, 80, 84, 90, 91, 98, 99

In case of G01, however, although G code can be specified, movement command cannot be specified.

- ⑤ Once G93 mode is specified, the spindle does not run (as a result of processing machine sequence). even by canceling the mode by G94, unless M03 and M04 are specified again. S command is left. Since gear selection signal is not output during G93 mode, specify S command again at the time of M03 and M04 specification.
- ⑥ If there is more than one gear, select the gear to be used for solid tap before specifying G93 and, then, execute G93.

(2) G84/G74 and G94 for Solid Tap

- ① Precautions for conventional tap program are all applicable to the precautions on specifying R and Z point commands, G98/G99, G90/G91 and number of repeats L.
- ② Output voltage of spindle forward run and reverse run by G84/G74 takes a different sign depending on the parameter. At #6006 D6=0, "++" is output by forward run and at D6=1, "--" is output by forward run. The spindle is automatically reversed at the controller side without using SINV input.

- ③ Be sure to cancel the canned cycle with G80 at the time of specifying G94 after completing solid tap.
- ④ When G94 is specified after completing solid tap, F command changes to O. After specifying G94, be sure to specify F so as to specify the cutting feed program.

(3) Solid Tap and Its Relation to Various Operations

① Dry Run

G93 becomes invalid even by executing G93 with Dry Run SW ON, and G84/G74 command for solid tap mode are handled as conventional type G84/G74. But, feedrate in G84/G74 is determined by Jog SW in feedrate per minute. Once solid tap mode is selected by Dry Run SW ON, Dry Run SW is handled as ON in connection with G84/G74 tap feed, even if Dry Run SW is changed over before completion.

At G93 block, Dry Run is designated if ON at the time of reading code of G93. Normally, it is designated during execution of block at a front of G93 block.

When actuating G93 with Dry Run ON for program check, keep Dry Run SW ON from the first and do not reset it before completion.

② Auxiliary Function Lock

G93 becomes invalid even by executing G93 with Auxiliary Function Lock SW ON and G84/G74 command for solid tap is handled as the one which changed only the feed to feed per revolution. The spindle position is not controlled.

At G93 block, it is designated if auxiliary function lock is ON or not at the time of reading code of G93.

When checking the program by Auxiliary Function Lock SW ON, keep Dry Run SW ON from the first and do not reset it before completion.

③ Machine Lock, Z-Axis Neglect

If solid tap is executed with Machine Lock or Z-Axis Neglect ON, the spindle revolves, but Z-axis is actuated only for display.

④ Override, Spindle Override

Feedrate override during solid tap is fixed completely. But, rapid traverse rate override is valid.

Validation /invalidation of spindle override input during solid tap mode is selectable by parameter #6007 D2 as in tapping of conventional type.

⑤ Feed Hold

Feed hold during cutting is invalid with G84/G74 for solid tap.

⑥ Mode Change

Mode change during cutting is invalid with G84/G74 for solid tap.

2.12.2.5 PRECAUTIONS (Cent'd)

⑦ Program Restart

G93 cannot be executed for program restart to the block during solid tap mode. G93 has to be modified by MDI and be executed.

2.12.3 SOLID TAP RELATED FUNCTION

The following functions are added or modified by the optional addition of solid tap function.

(1) Servo Positioning Error Display

In the solid tap mode, servo positioning error of the spindle is displayed at X-axis and the number of synchronous error pulses of the spindle and Z-axis is displayed at Y-axis, respectively, of servo positioning error display screen. (Z-axis displays servo positioning error of Z-axis.)

However, pulse display at Y-axis is not precise in continuous mode (single block OFF).

If parameter #6065 D1 is 1, peak value of synchronous error pulses can be displayed at X- and Z-axis.

(X-axis : Plus peak value,
z-axis : Minus peak value)

(2) Error Detect in Solid Tap

Rapid traverse command in solid tap (X, Y positioning, Z-axis positioning) can be made to Error Detect OFF mode by setting parameter #6065 D0 to 1.

Cycle time can be shortened by turning this parameter ON. Be careful of the program, since it proceeds to Z-axis block just after completing pulse distribution for positioning X and Y.

(3) Alarm Code

The following alarm codes are added.

① Alarm 090 (PROG ERROR (G93))

(Contents) . There is a command other than G93, S, F and N in G93 block. G code at 01 group is other than 00 or 01 when G93 is specified., G code that should not be specified was specified in G93 mode.

- Pitch F of solid tap is beyond the allowable range (200 mm/rev.).

② Alarm 091 (TAP ERROR)

(Contents) . When solid tap is executed, position control loop is not formed to the spindle. . SLPC turned OFF before completing index at the time of indexing with G93.

③ Higher return speed function upon solid tap

The return solid tap cutting feedrate (cutting from Z point → R point) can be controlled at n times ($0.1 \leq n \leq 25.5$) the forward speed (cutting from R point → Z point), by setting parameter #6470.

Note that the commanded S x magnification of the spindle speed is clamped at the maximum solid tap speed of #6471.

When #6470 = 0, it is regarded as n = 1.

#6470 Specification range 0 to 255
Unit 1 = 0.1 times
When 0 is specified, the magnification is 1 time.

2.12.4 SOLID TAP I/O AND ITS RELATION WITH SPINDLE CONTROL I/O

(1) I/O Used for Solid Tap

a. OUT Signal

① G93M Output (#12070): G93 Solid Tap Mode Output Signal

The signal is output when G93 is specified. The machine side receives this signal, turns ON the spindle forward signal and returns SLPC (spindle loop command input signal) after confirming the spindle zero speed.

Notes :

1. Cancel the soft start function at the same time SLPC turns on, if the spindle is equipped with soft start circuit (so that NC unit can control acceleration/deceleration).

2. Turn off the spindle reverse input (SINV), gear shifting input (GR0) and spindle orientation input (SOR).

② SLPS Output (#12071): Spindle Loop Status Output Signal

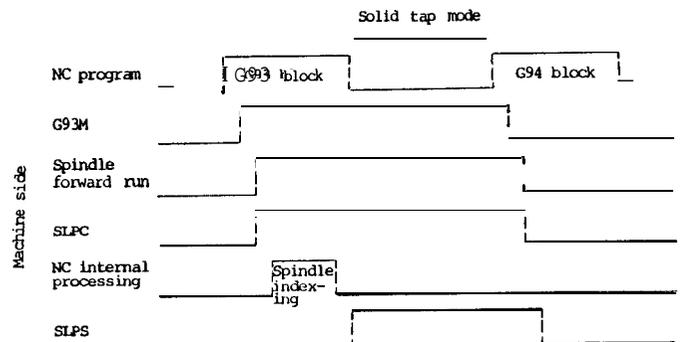
The signal is output at the time when NC unit forms position control loop to the spindle for solid tap. G93 block ends at the time this signal is output.

b. IN Signal

SLPC Input (#13133): Spindle Loop Command Input Signal

The signal is to form the spindle position control signal to the NC unit. The NC unit returns SLPS by forming the position control loop to the spindle after ensuring the spindle stop (following the number of revolutions at #6473) when this input turns ON.

c. Time Chart



Additional Explanation

1. G93M turns on when G93 block is executed with Dry Run OFF and Auxiliary Function Lock OFF.
2. G93M turns off when G94 block is executed or reset.
3. G93 block completes as SLPS turns on and job proceeds to next block.
4. G94 block completes as SLPS turns off and job proceeds to next block.

(2) Solid Tap and Its Relation with Spindle 1/0

During solid tap mode, previous output of gear select output and SF output 'is held. Gear select input, gear shifting input and spindle orientation input are disregarded.

If there is more than one gear, specify and execute G93 after selecting the gear to be used for solid tap.

(3) Connection to Spindle Pulse Generator

In the solid tap, the spindle revolution is controlled by using the spindle pulse generator (hereinafter called "spindle PG"). Consequently, both revolving direction of spindle motor and feedback signal direction from spindle PG have to be synchronized.

For this purpose, if the spindle PG is not mounted to the motor and revolving direction of spindle motor and spindle PG are different, change the connection to phases A and B as shown in the Connecting Manual.

If both revolving directions are the same, connect them as shown in the Connecting Manual.

2.12.5 SOLID TAP RELATED PARAMETER

"*" mark below shows the need for power on/off after setting parameter.

* (1) Spindle Override in G84

- #6007 D2 O: Valid
1: Invalid (completely fixed)

Note :

Override is valid at the time of reading G84/G74 and it cannot be changed over during tapping.

* (2) Operation at G93

- #6055 D1 O: Spindle not indexing
1: Spindle indexing

(3) GOO Error Detect in Solid Tap

- #6065 DO O: Error Detect on
1: Error Detect off

* (4) Display of Simultaneous Error Peak in Solid Tap

- #6065 D10: Not display the peak of simultaneous error for spindle and Z-axis
1: Display the peak of simultaneous error for spindle and Z-axis

During solid tap, plus (+) peak of simultaneous error is displayed to X-axis error pulse display area and minus (-) peak of simultaneous error is displayed to Z-axis error pulse display area.

* (5) Spindle PG Mounting Position

- #6065 D7 O: Spindle side
1: Motor side

Note: This parameter is used when a gear ratio exists between the spindle and motor. If the ratio is 1:1, set 0 even if it is at the motor side.

* (6) Gear Ratio between Spindle and Spindle Motor when Using Solid Tap

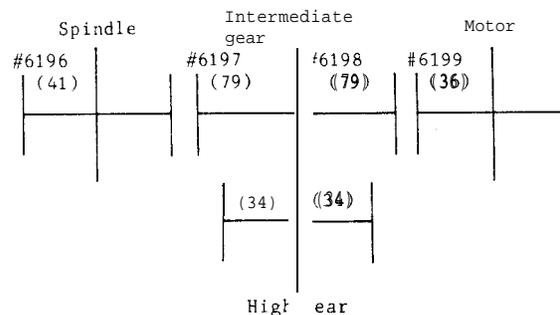
- #6198 Range : 0 to 127
Number of teeth at spindle side :
n, unit : 1
#6199 Range : 0 to 127
Number of teeth at motor side :
m, unit : 1

Notes:

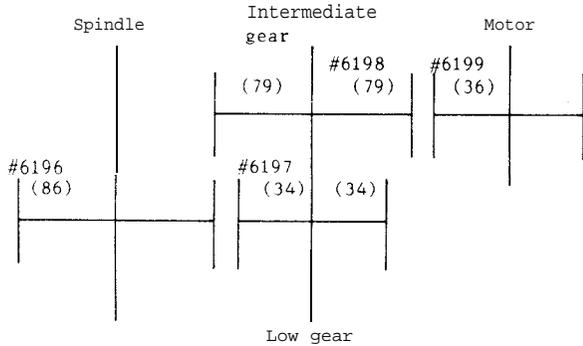
1. It can be set with #6169 or #6197 at the time of using intermediate gear. Set 0 to #6196 and #6197 for the machine not using an intermediate gear (Fig. 1 shows setting sample).

- #6196 Range : 0 to 127
Number of teeth at spindle side
#6197 Range : 0 to 127
Number of intermediate gear teeth at spindle side
#6198 Range : 0 to 127
Number of intermediate gear teeth at motor side
#6199 Range : 0 to 127
Number of teeth at motor side

2. It sets the gear tooth ratio if there is a gear between the spindle and motor. If spindle: motor=n: m, set the value of n and m to #6198 and #6199. If it is 1:1, set 0 both to #6198 and #6199 (it can be as 1, 1) .



2.12.5 SOLID TAP RELATED PARAMETER (Cont'd)



* (7) Spindle Feedback Pulse Multiplication

#6061

D3	D2	D1	D0	Multiplication
0	0	0	1	×1
0	0	1	0	×2
0	1	0	0	×4
1	0	0	0	×8

Standard setting: ×4

* (8) Number of Feedback Pulses per Revolution of Spindle PG

#6449 Range 1 to 32767
Unit: Pulse /rev.
Standard setting: 1024

Set the value before applying multiplication for this parameter.

(9) Spindle Speed to Command 10 V at Solid Tap Using Gear

#6472 Range : 1 to 32767
Unit 1 = 1 rein-l
Standard setting : 6000

(10) Maximum Spindle Speed at Solid Tap

#6471 Range : 1 to 32767
Unit 1 = 1 rein-l
Standard setting : 4500

Set up carefully to maximum spindle motor speed at solid tap.

(11) Spindle Command Unit Position Loop Gain

#6411 Range : 1 to 32767
Unit 1 = 0.01 s⁻¹
Standard setting : 2500

(12) Spindle Acceleration /Deceleration Constant

#6291 Range 1 to 32767
Unit 1 = 1 ms
Standard setting: 2000

It sets the acceleration time from the stop status to number revolutions specified by #6471. It is determined by motor torque and load. Refer to Adjustment Procedure for Solid Tap Function for YASNAC J50M (DE8408877) for setting.

(13) Magnification of Spindle Position Error Area

#6155 Range 1 to 200
Unit 1 = 1%
Standard setting: 120

It sets the area to detect excessive error of the spindle at solid tap mode by the rate of load to maximum spindle speed (#6471) at the time of solid tap. When the load corresponding to 1.2, times maximum spindle speed is applied at #6155=120, it causes alarm 346 SERVO ERROR(S) .

(14) Spindle Error Detect ON Area

#6161 Area : 0 to 255
Unit 1 = 1 pulse
Standard setting : 50

It specifies the error detect area at R point and error detect area at hole bottom (Z point) at the start of solid tapping.

(15) Z-axis Pull-in Area

#6476 Area : 0 to 65535
Unit 1 = 1 pulse
Standard setting : 1000

It specifies pull-in area of air cut area near R point after solid tapping. After completing distribution, when residual movement of Z-axis enters into this area, it cancels simultaneous feed of the spindle and Z-axis and pulls Z-axis into R point.

(16) Fine Adjustment of Spindle/ Z-axis Simultaneous Compensation Parameter (K1)

#6468 Range : -32768 to 32767
Standard setting : 0

This is the parameter for compensating for the simultaneous operation of the spindle and Z-axis at the steady-state cutting feed. Refer to Adjustment Procedure for Solid Tap Function for YASNAC J50M (DE8408877) for adjustment.

(17) Spindle/Z-axis Simultaneous Compensation Parameter (K2)

#6469 Range : 0 to 32767
Standard setting : 2000

This is the parameter for compensating for the simultaneous operation of the spindle and Z-axis at the time of acceleration/deceleration feed.

Refer to Adjustment Procedure for Solid Tap Function for YASNAC J50M (DE8408877) for adjustment.

(18) Revolving Speed Command for Spindle Indexing

#6462 Range : 1 to 32767
Unit 1 = 500 pulses/s
Standard setting : 20

However, #6472, maximum spindle speed at the gear using solid tap, is the upper limit (to be clamped).

(19) Creep Speed Command for Spindle Indexing

#6463 Range : 1 to 32767
 Unit 1 = 500 pulses/s
 Standard setting : 10

However, #6472, maximum spindle speed at the gear using solid tap, is upper limit (to be clamped).

(20) Spindle Stop Confirmation Timer for Spindle Indexing

#6464 Range : 0 to 255
 Unit 1 = 8 ms
 Standard setting : 1

(21) Original Position for Spindle Indexing

#6465 Range : 0 to 4095
 Unit 1 = 1 pulse
 Standard setting : 0

(22) Revolving Speed to Confirm Spindle Stop

#6473 Range : 0 to 30
 Unit 1 = 1 rein-l
 Standard setting : 30

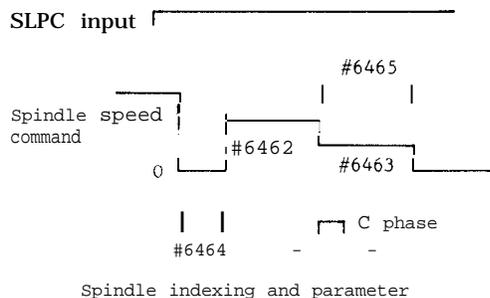
It sets the revolving speed to confirm spindle stop at the time of forming position loop of spindle with G93. Position loop of spindle is formed after G93 is specified and the spindle speed decreases to a value lower than this parameter. Set the value lower than the parameter to confirm zero speed at the spindle drive side.

(23) Return speed in solid tap

#6470 Range 0 to 255
 Unit 1 = 0.1 times

When 0 is specified, the magnification is 1 time.

Items (18) to (23) cover the parameter about spindle indexing at the time of specifying G93.



Note :

In case of setting original position O, it stops on C phase after one revolution at creep speed after detecting C phase.

2.13 AUTOMATIC CORNER OVERRIDE

The load at the corner increases when machining the inner surface.

In conventional machines, the cutting speed was reduced by the override switch, or the speed was commanded by dividing the corner rounding block in several blocks.

Corner override is a function where the override automatically changes in a single-block command.

(1) Command format

(a) Non-modal G code G106 is used.

G106G01 X... Y... I... (or J...)

F...

G106 : Non-modal G code

I : Cutting margin (X-axis direction)

J : Cutting margin (Y-axis direction)

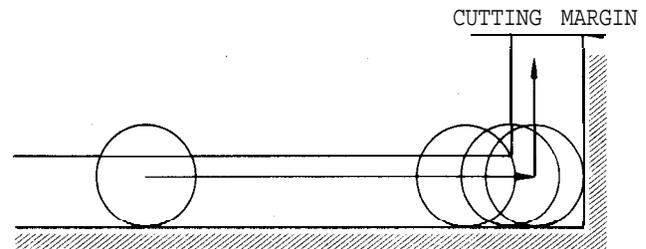


Fig. 2.80

(b) Override command

#6216 : F1: 1st step F feedrate override % (1 to 100)

#6217 : F2: 2nd step F feedrate override % (1 to 100)

When the cutting margin is larger than the tool radius, #6216 is always calculated as 100%. (Refer to the Note.)

(2) Operation

This function automatically calculates the position in the corner where the load on the tool starts to change, from the tool radius and cutting margin. The position where the load is applied differs according to the size of the tool radius and cutting margin. This is why the following be considered.

(a) When the cutting margin is larger than the tool radius

(b) When the tool radius is larger than or equal to the cutting margin

(c) When the tool radius is larger than the cutting margin

2.13 AUTOMATIC CORNER OVERRIDE (Cent'd)

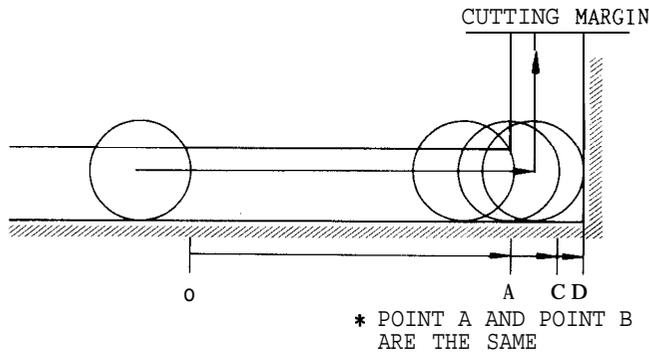


Fig. 2.81

(i) The tool moves in sequence of O → A(B) → C → D.

(ii) The speed between O → A is as commanded (F command * override)

A → C is the variable speed zone.

C → D is the clamped speed zone (n% the commanded speed) .

(iii) Speed variation curve

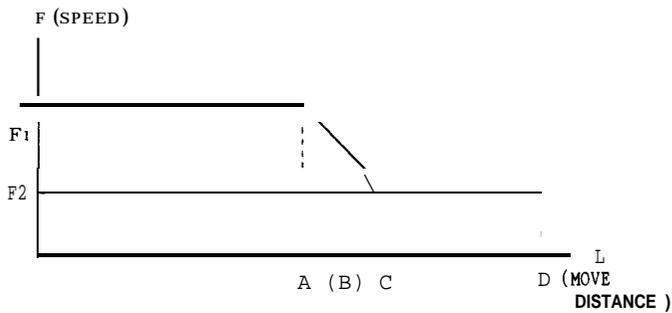


Fig. 2.82

$$F_o = (\text{Commanded feedrate}) * (\text{override } \%)$$

$$F_1 = F_o$$

$$F_2 = F_o * (\#6217) \dots (\text{given as } n\% \text{ of } F_o)$$

Note : The operator is not required to compare the sizes of the tool radius and the cutting margin. In this case, the override of F1 is regarded as 100%, regardless of the setting in #6216.

(iv) Speed change positions

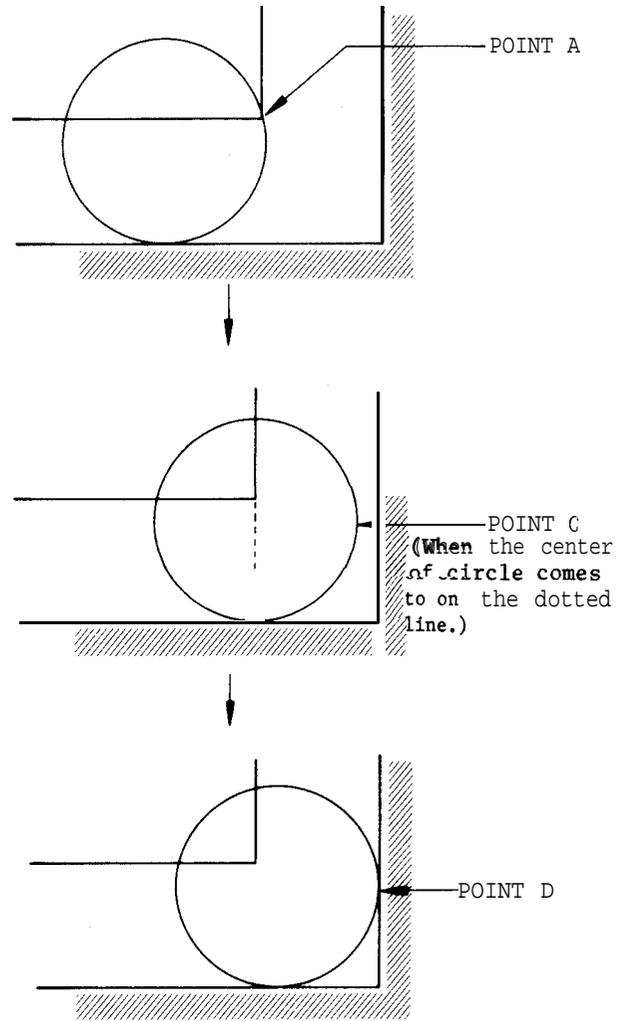


Fig. 2.83

(b) When the tool radius is larger than or equal to the cutting margin

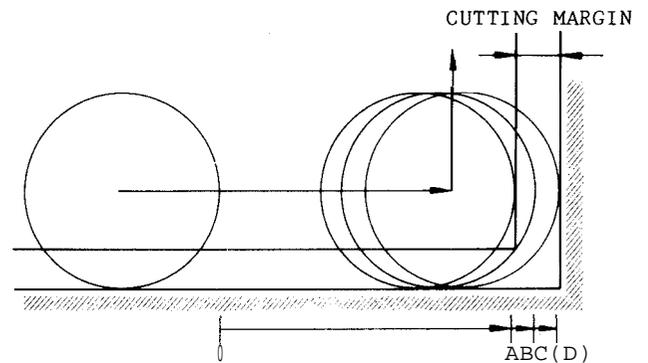


Fig. 2.84

- (i) The tool moves in sequence of O → A → B + C (D) .
- (ii) The speed between O → A is as commanded (F command * override)
A → C is the variable speed zone.
- (iii) Speed variation curve

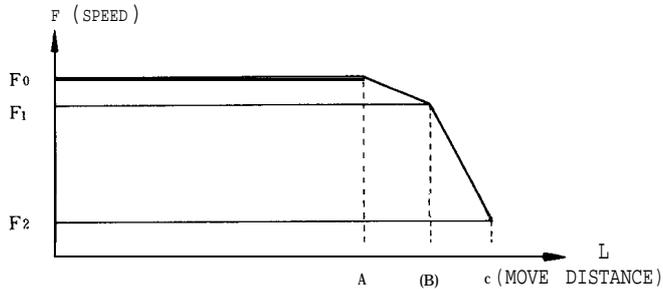


Fig. 2.85

$F_0 = (\text{Commanded feedrate}) * (\text{Override \%})$
 $F_1 = F_0 * (\#6216) \dots (\text{given as } n\% \text{ of } F_0)$
 $F_2 = F_0 * (\#6217) \dots (\text{given as } n\% \text{ of } F_0)$

- (iv) Speed change positions

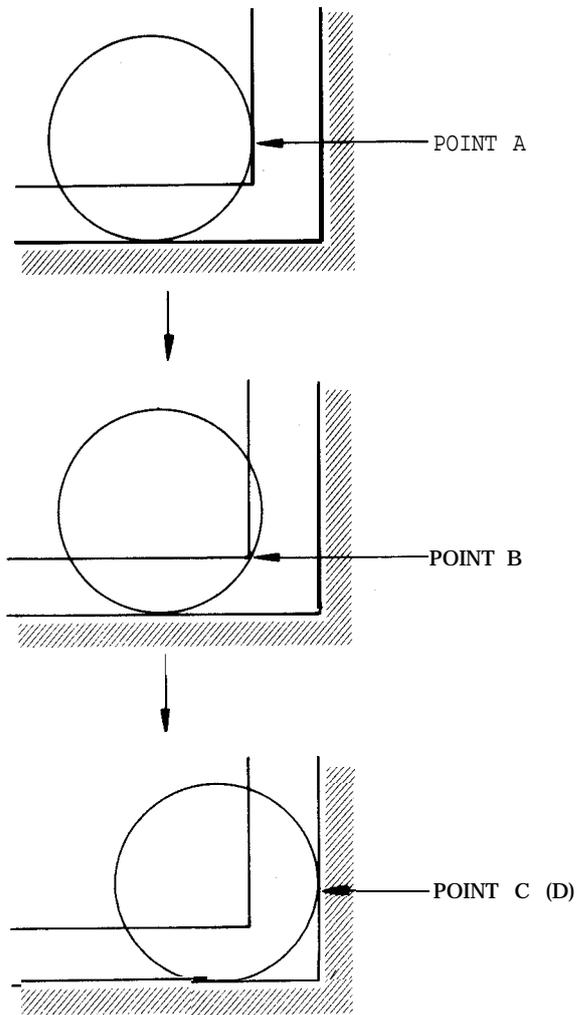


Fig. 2.86

- (3) Notes

- (a) Corner override can be calculated to a single position. If corner override is commanded in a 2-axis simultaneous command, the override automatically changes to the I or J direction at the commanded position.
- (b) Command the G codes of group 01 when the G106 command is given. If not, alarm (029: G106) occurs.
- (c) Commanding I and J simultaneously by the G106 command causes an alarm (039: G106 X, Y, I, J).
- (d) When the tool radius is larger than the cutting margin, and a mistake in setting is made to be $F_0 \leq F_1 \leq F_2$, the speed is clamped at the value immediately before (F_1 against F_2). For example, if F_1 is set at 80% and F_2 at 90% by mistake, calculation is made with F_2 as 80%. There will thus be no automatic corner override between A and B, and the speed is clamped at F_1 .

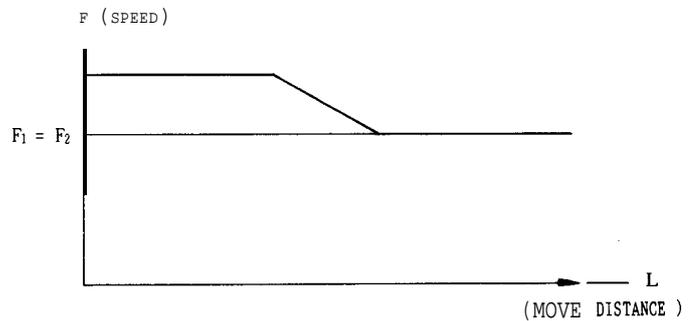


Fig. 2.87

- (e) Set #6216, #6217 between 1 and 100. (Do not set "O.")
- (f) Commanding G106 in a canned cycle causes alarm (029: G106) .
- (g) Alarm (039: G106X, Y, I, J) occurs when no I or J command is found against the move axis in the G 106 block.
- (h) No override is performed by the G106 command, if the tool radius is 1 / 2 or less the cutting margin.
- (i) This command is valid during the radius compensation.

2.14 HIGH-SPEED CONTOURING FUNCTION*

2.14.1 OVERVIEW

Form compensation function

Generally, when contour cutting is commanded, the orbit shifts (sagging or shrinking arc radius at the corner) because of the time delay of the exponential function accel I decel, or by the influence of the servo follow-up delay.

So the linear interpolation with linear acceleration (G 198) and linear interpolation with linear deceleration (G 199) are developed and added to the preparatory y functions. This limits the acceleration in case the speed variation is too radical since it may shock the machine.

2.14.1 OVERVIEW (Cent'd)

G198 is also the start and G199 the end of form compensation.

Circular projection compensation

Another problem is the projection remaining in circular cutting, at the point where the quadrant is switched.

This projection can be made smaller by using the circular projection compensation function.

The above two functions are called high-speed contouring functions.

Note :

When using form compensation, set "0" in all the exponential accel/decel parameters (#6418 to #6420) . Otherwise, correct movement cannot be guaranteed.

2.14.2 FORM COMPENSATION FUNCTION

Programming

The form compensation mode turns on by commanding G198 in the G01 mode. This mode compensates for the delay of the servo system, and performs interpolation. This mode turns off by commanding G 199 in the single-block, feed hold or G01 modes.

(a) Command format

(i) G01 G198 X... Y... Z... ;
(Start form compensation mode)

Command G198 under the G01 mode after positioning, to turn on the form compensation mode. Then perform linear acceleration to the currently commanded cutting speed in the block. The move command must be large enough for acceleration.

(ii) G01 G199 X... Y... Z... ;
(End form compensation mode)

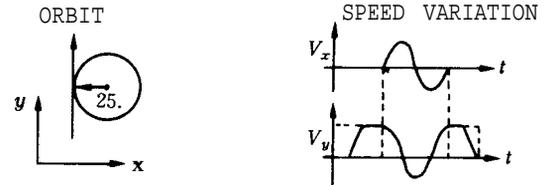
Command G 199 under the GO 1 mode in the form compensation mode, to perform linear deceleration and position at the commanded point. The form compensation mode turns off after positioning. The move command must be large enough for deceleration.

(b) Program example

(i) How to command linear accel / decel in a block with discontinuous command speed

EXAMPLE (C I R C U L A R C U T W I T H F O R M C O M P O) ;

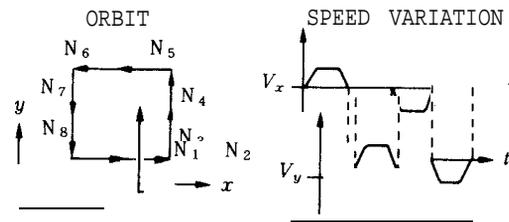
```
G01 G91 F10000 ;
N1 Y25. G198 ;
N2 G02 125;
N3 G01 Y25. G199 ;
M30 ;
```



(ii) When commanding a form with angle, divide each line before and after the angle in two, and command accel/decel before and after the angle.

EXAMPLE (S Q U A R E C U T W I T H F O R M C O M P .) ;

```
G01 G91 F6000 ;
N1 X25. G198 ;
N2 X25. G199 ;
N3 Y25. G198 ;
N4 Y25. G199 ;
N5 X-25. G198 ;
N6 X-25. G199 ;
N7 Y-25. G198 ;
N8 Y-25. G199 ;
M30 ;
```



(2) Notes

(a) The movement has no delay against the interpolation command, under the form compensation mode. Thus, if disconnected speed is commanded, it can shock the machine, or cause servo error.

(b) Giving commands other than the interpolation command under the form compensation mode can shock the machine, or cause servo error.

(c) Performing single-block or feed hold under the form compensation mode causes the form compensation mode to turn off. The form compensation mode will not turn on until G198 is commanded under modes other than the single-block mode.

(d) The override cannot be changed while executing the block commanded with G 199.

(e) When interpolating a line shorter than the accel/decel distance, divide the line into two equal parts, and command G198 and G199. Smooth accel/decel cannot be gained unless this division is precise.

(f) G198/G199 are non-modal group G codes. Do not use two or more G codes of the same group in an single block.

(3) How to set the acceleration for linear accel/ decel

Set the maximum acceleration of a single axis in the parameter.

Setting: "1" = 1/64 (m/s²) = 60/64 [(m/min) /s]

Setting example How to accelerate to F10000 within 0.1 s
 10/0.1 = 100 [(m/min)/s]
 100/(60/64) = 106.7

Set 107 as the maximum acceleration parameter.

(4) How to set the form compensation parameter

Set the servo follow-up coefficient Kx, Ky . . . K4 under the form compensation mode, by the parameter. The larger this coefficient, the better the servo follow-up characteristics, and the better the form precision. But, too large a setting can cause overshoot. The following value can be considered as guidance upon setting.

Exponential accel/decel time constant: Te (ins)
 Servo follow-up coefficient: kx

$$kx \cdot \frac{1}{\left[1 - \exp\left(-\frac{8}{Te}\right)\right]} \cdot 8192$$

Set the above value to gain orbit precision equivalent to the precision when zero is set as the exponential accel/decel time constant. However, the servo delay and machine system delay cannot completely be disregarded by this value, and the kx must be adjusted while measuring the form precision by using orbit measuring devices if the form precision is still to be improved.

(5) Parameters related to the form compensation function

Single-axis maximum acceleration -#6488 (1 to 32767) ³⁶
 Servo follow-up coefficient kx (X-axis) - #6690 (0 to 65535) | 2630
 Servo follow-up coefficient ky (Y-axis) - #6691 (0 to 65535), 2630 (Symbol)
 Servo follow-up coefficient kz (Z-axis) - #6692 (0 to 65535)
 Servo follow-up coefficient k4 (4th axis) - #6693 (0 to 65535)

2.14.3 CIRCULAR PROJECTION COMPENSATION

(1) Overview

When a circle is cut by the machining center, a convex projection can be left at the switch point of the quadrants.

This projection occurs because the lost motion affects the machine to disturb immediate movement when the move direction changed.

The servo speed loop gain can be increased to prevent the projection to some degree. However, increasing vibration and other causes will not allow the speed loop gain to be increased.

This function is to automatically output the speed impulse preset as the parameter, when the move direction of the machine changes, so that the projection can be "hammered down."

(2) Parameters related to the circular projection compensation function

(a) Circular projection compensation ON/OFF

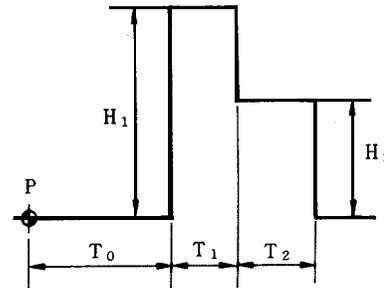
The following parameters must be set for each axis.

x-axis : #6056 D7 D6 1.0
 Y-axis : #6057 D7 D6 1.0
 z-axis : #6058 D7 D6

0 0: Circular projection compensation off
 1 0: Circular projection compensation on

Always turn the NC power off and then on when change is made in this parameter.

(b) Offset amount and timing



- P : Start point of the circular projection compensation (Actually when the sign of the command is reversed.)
- H1: Height of the 1st offset amount
- H2: Height of the 2nd offset amount
- TO: Wait time from the start point to the first offset
- T1: Width of the 1st offset
- T2: Width of the 2nd offset

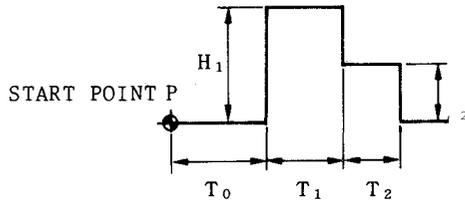
(Param No.)	(Description)	(Setting range)	(Unit)
H1X #6071	(X-axis 1st height)	0 to 255	1 = 500 [pps]
H1Y #6078	(Y-axis 1st height)	0 to 255	1 = 500 [pps]
H1Z #6079	(Z-axis 1st height)	0 to 255	1 = 500 [pp.]
H2X #6080	(X-axis 2nd height)	0 to 255	1 = 500 [pps]
H2Y #6081	(Y-axis 2nd height)	0 to 255	1 = 500 [pps]
H2Z #6082	(Z-axis 2nd height)	0 to 255	1 = 500 [pps]
T0X #6083	(X-axis comp. start wait time)	0 to 255	1 = 2 [ms]
T0Y #6084	(Y-axis comp. start wait time)	0 to 255	1 = 2 [ms]
T0Z #6085	(Z-axis comp. start wait time)	0 to 255	1 = 2 [ms]
T1X #6086	(X-axis 1st width)	0 to 255	1 = 2 [ms]
T1Y #6087	(Y-axis 1st width)	0 to 255	1 = 2 [m.]
T1Z #6088	(Z-axis 1st width)	0 to 255	1 = 2 [m.]
T2X #6089	(X-axis 2nd width)	0 to 255	1 = 2 [m.]
T2Y #6090	(Y-axis 2nd width)	0 to 255	1 = 2 [m.]
T2Z #6091	(Z-axis 2nd width)	0 to 255	1 = 2 [m.]

Notes:
 1. Any change made in parameters #6056 to #6058 becomes effective only after turning the power off and then on again. DG of #6056 to #6058 must always be set o.
 2. Any change made in parameters #6077 to #6091 becomes effective-by NC RE.SET.

2.14.3 CIRCULAR PROJECTION COMPENSATION (Cont'd)

(3) Adjustment

The following are standard setting and adjustment procedures.



EXAMPLE

```
G92 XO YO ZO
G91 G01 F3000
G01 G198 Y25.
G03 I-25.
G01 G199 Y25.
G00 Y-50.
M30
```

(a) Adjustment procedure

(i) The start point P is fixed at the position where the sign of the (segment) changes.

(ii) The delay time T_0 is calculated as follows.

$$T_0 = 1/k_p \times 0.7 \text{ [s]}$$

(Half of the above value is set in the parameter, as $1 = 2$ ms.)

(iii) Increase the height H_1 from the temporary setting $H_1 = 20$ by 5 ($T_1 = 10$ when $H_2 = 0$, $T_2 = 0$), and observe the effect.

(iv) Feed H_1 while checking the effect by actual cutting, orbit analyzer, plan, DBB, etc.

(b) Notes

(i) Set the soft hammer parameter after setting the form compensation parameter.

(ii) If normal adjustment is not effective enough, try adjustment by changing T_0 , H_2 and T_2 .

3. PART PROGRAM TAPE CODING

3.1 TAPE CODE

3.1.1 TAPE CODE

With this control, both the EIA and the ISO codes can be used.

EIA code: EIA RS-244-A

ISO code: ISO 84

Table 3.1 shows the EIA and ISO punched tape formats.

Before starting to program any machining operation, a decision must be made as to the code to be used.

3.1.2 EIA/ISO AUTO-SELECT

Before starting to use part program tapes, the control must be switched to the same code as the tapes, in accordance with the procedure for writing-setting under 4.3.6, "DISPLAY AND WRITING OF SETTING DATA."

Despite the content of #6001D7, the control is automatically adapted to the code used for the part program tape. The control recognizes the code used when it reads the first EOB code in the label skip mode, and all the subsequent data can be read automatically in that code.

For punching tapes, the code must be selected by the setting of #6000D7.

When "O" is set with #6000D7 .0. EIA code

When "I" is set with #6000D7 . . . ISO code

Table 3.1 Tape Code

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

Notes:

1. For characters from #to ?, EIA codes have not been agreed upon. In the present system, for the time being, the above provisional codes are used.
2. EIA code of character #can be designated by the parameter #6017.

3.2 PROGRAMMING

3.2.1 PROCESS SHEET

Programs are first drafted on process sheets.

Process sheets should be easy to read and to make corrections, and should be designed and prepared by the user in conformity with the specifications of the NC.

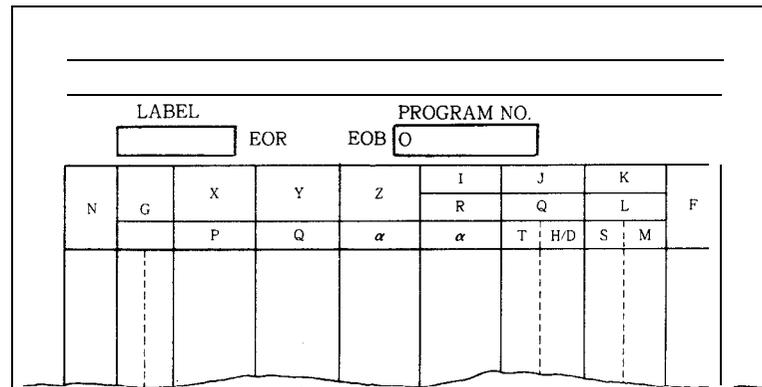


Fig. 3.1 Example of Process Sheet

3.2.2 GENERAL PART PROGRAM FORM

Part programs are generally written on tapes in the following formats.

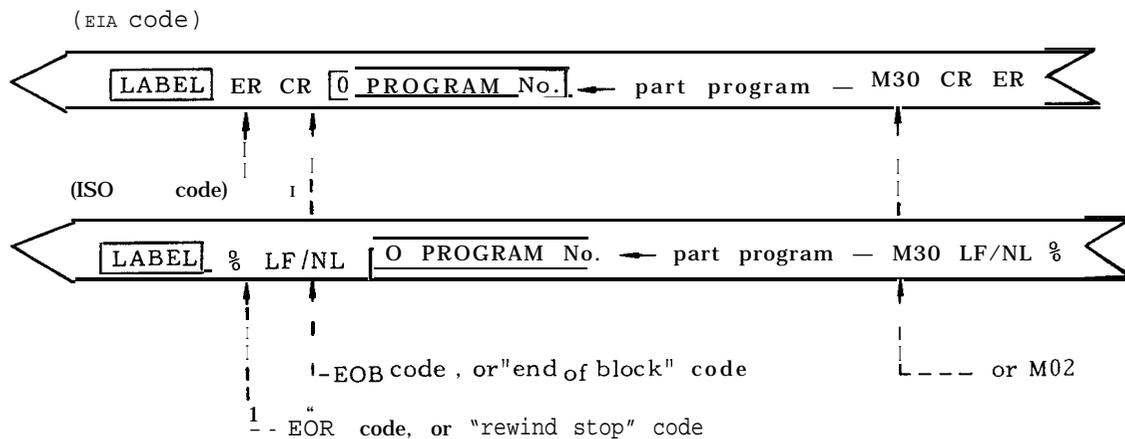


Fig. 3.2

To facilitate classification and handling of tapes, any identifying labels can be written at the leading end of all part program tapes. Since all data appearing before the first EOB code are skipped by the NC reading these tapes using the label skip function, even addresses and function codes not specified can be written here, and also out-of-parity codes can be written.

The EOR (rewind stop) code following the label is the stopping point of rewinding motion which is initiated by a rewind command.

When programs on a part program tape is transferred into the memory, the data between the first EOB code, following label skip, and the next EOR code is transferred. Therefore, the EOR code at the end should not be omitted.

3.2.3 CAUTIONS IN PROGRAMMING

One block ends with an EOB code. The EOB code is CR in EIA, and LF/NL in ISO. However, in the example programs in this manual, “;” is used to represent EOB codes to make recognition easy.

One part program ends with a block containing M02 (end of program) or M30 (end of tape) .

When an M02 or M30 code is read , the NC re-sets itself or rewind the tape (or the memory) , depending on the design of the machine control. For this, refer to the manual prepared by the machine tool builder.

Do not use any characters other than the address characters and the function code characters specified in 2. 1.2 ADDRESS AND FUNCTION CHARACTERS .

The maximum number of characters that can be written in a block is 128, the parameter can also be switched to allow setting of a maximum of 256 characters, but not counting disregard characters such as BS, Tab, SP, UC, LC and Del.

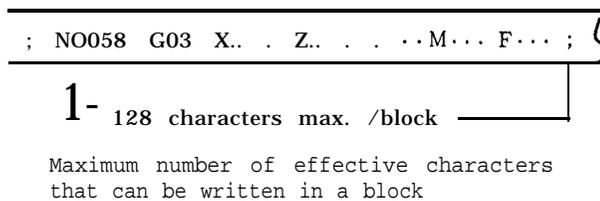


Fig. 3.3

3.3 PART PROGRAM TAPE PUNCHING

3.3.1 PAPER TAPE SELECT

For part program tapes , eight-channel paper tapes for computers conforming to JIS C6243 (width: 25.4 ±0.08 mm, thickness: 0.108 mm) are used.

The color should be black or gray. Tapes with high transparency tend to cause reading errors , and should not be used.

3.3.2 NC TAPE PUNCH

Part programs written on process sheets are punched in EIA or ISO codes in paper tape with a tape puncher .

A part program tape should be provided with a proper length of feed holes at the leading and the trailing ends. For a tape reader using 6" reels, the feed hole length should be at least 70 cm , and for a tape reader using 8" reels , it should be at least 1 m ,

3.3.3 NC TAPE CHECK

Punched part program tapes can be checked by an NC with the following functions.

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

3.4 PART PROGRAM TAPE HANDLING

3.4.1 SPLICING NC TAPE

To join part program tapes, the two ends should be placed end to end without overlapping and without a space, and a proper length of splice tape should be pasted on one side. (approx. 0. 08 mm in thickness) Tape splices are available in the fully perforated type and in the type with which only the feed holes are punched, but the former is more convenient. After splicing, the tape should be checked for correct alignment of the feed holes before use. Do not use rigid industrial adhesive, and do not make the joint too thick, as these conditions are conducive to jamming troubles.

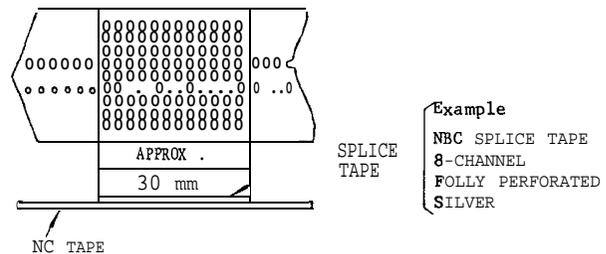


Fig. 3.4 Splicing Part Program Tape

3.4.2 KEEPING NC TAPE

Part program tapes should be stored in a clean area, free of contaminants and humidity. Do not handle part program tapes wearing gloves contaminated with oil or cutting fluid. Generally, properly maintained part program tapes can last at least 300 cycles, with one cycle consisting of one reading and one rewinding pass.

4. NC OPERATOR'S PANEL WITH 9" CRT CHARACTER DISPLAY

4.1 PUSHBUTTONS, LAMPS AND KEYS

Fig. 4.1 shows an overall view of NC operator's panel with 9" CRT display. The names and functions of operator devices are as follows.

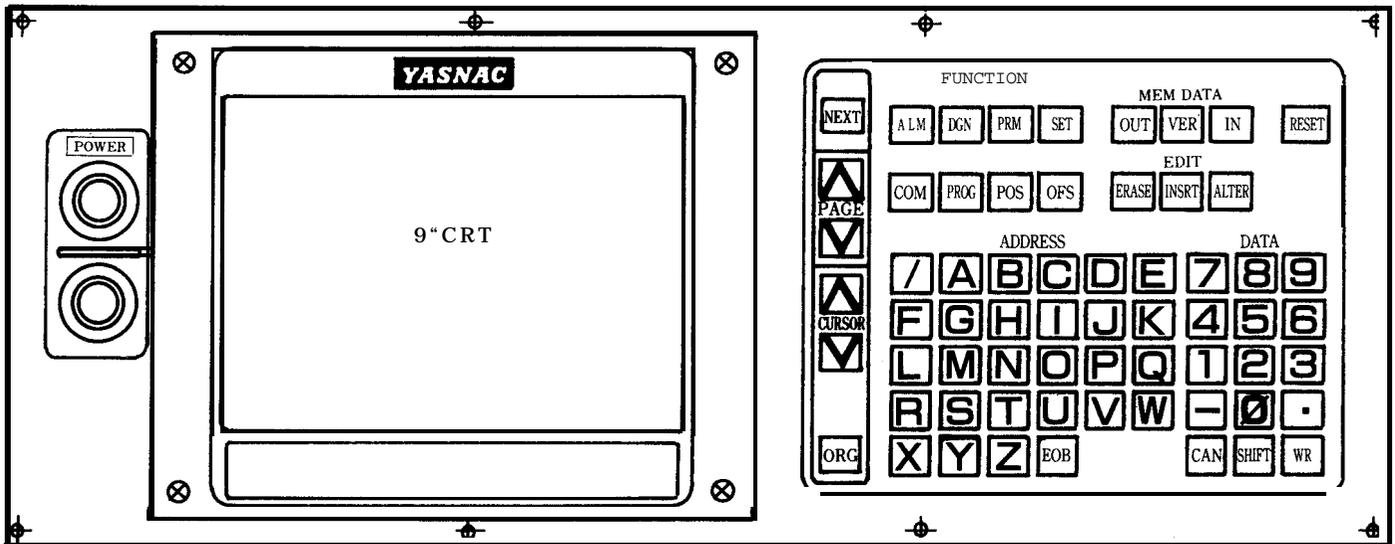


Fig. 4.1 Standard NC Operator's Panel with 9" CRT Character Display
(Keyboard on Right Side of CRT)

4.1.1 POWER ON/OFF PUSHBUTTONS

. POWER ON pushbutton

To turn on the power for the control: Depress the pushbutton once to turn on the control power and depress it again to turn on the servo power. Push this button to recover the servo power after an emergency stop.

. POWER OFF pushbutton

To turn off the power for the control: Depress POWER OFF pushbutton to turn off both the servo and control powers.

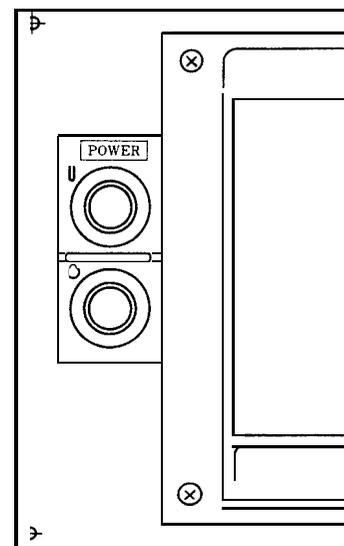


Fig. 4.2

4.1.2 CRT CHARACTER DISPLAY

According to each operation, this display indicates the alpha-numerical data in a regular size (1x1) and, triple-size (3x3) of the regular size.

Braun tube size: 9 inches

Maximum number of characters:
 32 characters x 16 lines
 512 characters (at regular size)

Indicating characters:

N umerals - □ through □, □, □
 Alphabetic characters - □ through □
 Special code - □ (EOB), □ (slash), etc.

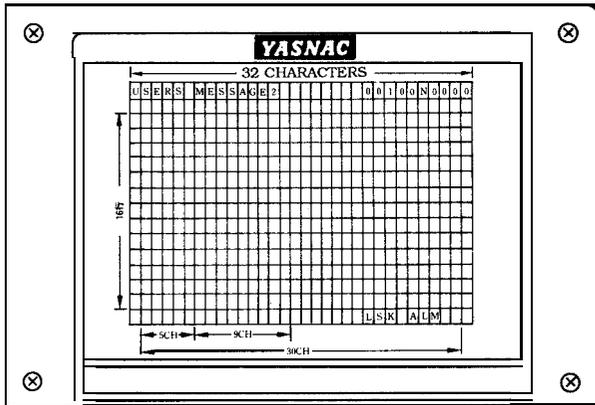


Fig. 4.3 Braun Tube

4.1.3 FUNCTION KEYS

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

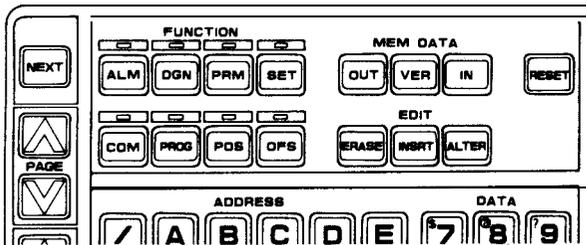


Fig. 4.4

ALM (Alarm) key:

Select this key for display of alarm and status codes. The function becomes effective when the power is turned on.

DGN (Diagnosis) key:

Select this key for display of input /output signal status.

PRM (Parameter) key:

Select this key for display or writing-in of parameters.

SET (Setting) key:

Select this key for display or writing-in of setting data.

COM (Command) key:

Select this key for display or writing-in (MDI) of the command data for automatic operation.

PROG (program) key:

Select this key for display or writing-in of a part program.

POS (Position) key:

Select this key for display of various current positions.

OFS (Offset) key:

Select this key for display or writing-in of tool offset values.

4.1.4 ADDRESS KEYS

These keys are to designate an address character when writing in various data.

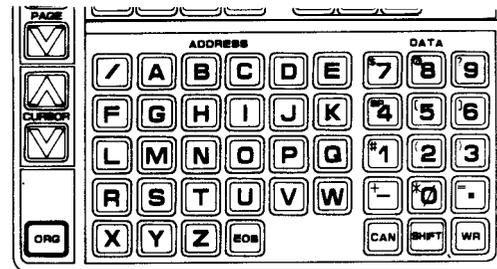


Fig. 4.5

Note: Special characters

/ (Slash) key: For an optional block skip command.

EOB (EOB) key: For the block end command. On the CRT display, ". " is displayed instead of "EOB "

4.1.5 DATA KEYS

These keys consist of 15 keys in total, such as □ through 9, □ (minus) □ **CAN** **SHIFT** **WR**, and can be used for writing-in of such all numeral values as tool offset value setting data, parameter data, and so on, in addition to command value.

4.1.5 DATA KEYS (Cent'd)

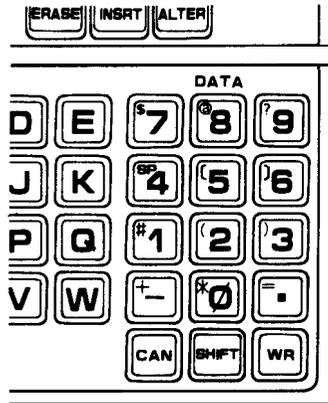


Fig. 4.6

Note:

- to □ key : For input of numerical data
- (minus) key : For input of numerical data
- (decimal point) key: For input of decimal point
- CAN** (cancellation) key:
For cancellation of the numeric value or address data erroneously keyed.
- WR** (write) key:
For storing address data by address keys and data keys into buffer storage.
- SHIFT** (shift) key:
Depressing **SHIFT** key after depressing 0 to 9, -, or □ key makes the display turn into □ to □, +, □ which are written on the upper left corner of the keys. These special characters are used in user macro.

4.1.6 NEXT KEY

The **NEXT** key is used for special purpose and expanding function in display or writing data.

- . Writing of optional in EDIT mode.
- . For other special purpose and expanding function.

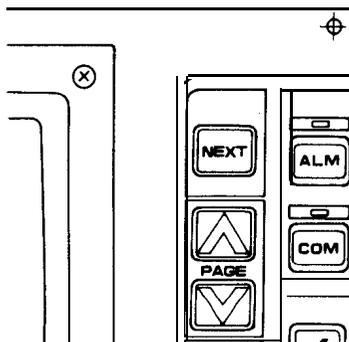


Fig. 4.7

4.1.7 PAGE KEYS

The **PAGE** key is used to display the next page or the previous page when CRT display is regarded as page.

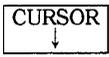
For example, when a group of tool offset values are displayed by OFS key, this key is pushed to display the next group of tool offset values, which just looks like opening the pages of a book.

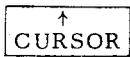
- Depressing  key displays the next page.
- . Depressing  key displays the previous page.

Keeping the **PAGE** key depressed makes the page step automatically forward or backward.

4.1.8 CURSOR KEYS

The **CURSOR** control key is used to move the cursor. For example, when a page of parameter data are displayed by PRM key.

Depressing  key moves the cursor backward.

Depressing  key moves the cursor forward.

Keeping the cursor control key depressed makes the cursor move automatically forward or backward.

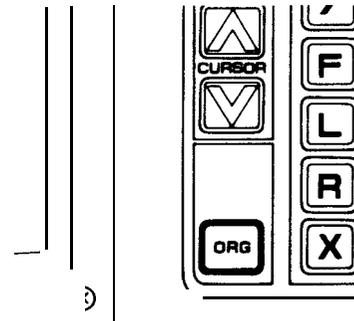


Fig. 4.8

4.1.9 ORG (ORIGIN) KEY

The **ORG** key is used to set the current position of the machine tool as the origin of the reference coordinate system.

The origin setting can be made for each axis. The reference coordinate system means the coordinate system which is set by G92 command or the automatic coordinate system setting.

ORG key is used for the following operation .

- Reset of current position (UNIVERSAL, EXTERNAL)
- . Reset of tool offset values
- . Reset of operation time

4.1.10 EDIT KEYS

These keys are for editing a stored part program.

- ERASE key: Used for erasure of data in storage.
- INSRT key: Used for insertion of data in memory
- ALTER key: Used for alteration of data in memory.

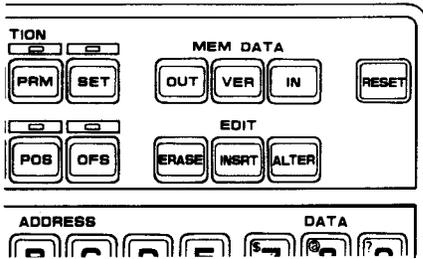


Fig. 4.9

4.1.11 MEM DATA (MEMORY DATA) KEYS

TAPE KEYS are to start the tape operation except in the automatic operation mode. They are effective only in the EDT mode.

- . OUT key
This key is to start outputting various data in memory through data I/O interface.
- IN key
This key is to start storing various data into memory through tape reader or data I/O interface.
- . VER key
This key is to start verifying between memory data and punched tape data.

4.1.12 RESET KEY

This key resets the control.

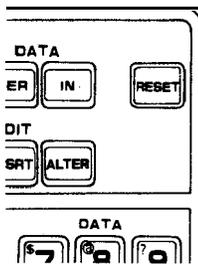


Fig. 4.10

Operations to be executed by this RESET key are:

- . Move command cancel
- . Buffer register clear
- . Alarm code release if the cause is eliminated
- . Tool offset cancel

- Auxiliary function cancel
- Label skip function ON
- Memory pointer rewind
- Sequence number reset
- RST signal transmission
- G code

Refer to 2.9.1, "LIST OF G CODES AND GROUPS."

The following will not be affected by operating the RESET key.

- . Current position values of each axis.
- . F commands
- . S, T and B commands
- . Tool offset values, setting data, parameter data

Note: Depressing the RESET key or the remote reset pushbutton is defined as "Reset operation" in this manual.

4.2 POWER ON/OFF OPERATION

4.2.1 TURNING ON POWER

Check the machine before turning on power, referring to the machine tool builder's manual for details. Operations after completion of preinspections are as follows.

Depress the POWER ON pushbutton to turn on the control power. The internal timer will be read in about two seconds. Then the servo power is ready for turning on, which is shown by alarm code "310."

Depress the POWER ON pushbutton again to turn on the servo power. The NRD (NC READY) signal is sent out when the NC power is normally supplied.

When the NRD signal turns on the machine power, and the MRD (MACHINE READY) signal returns back to the control, the READY lamp will be lit.

4.2.1 TURNING ON POWER (Cent'd)

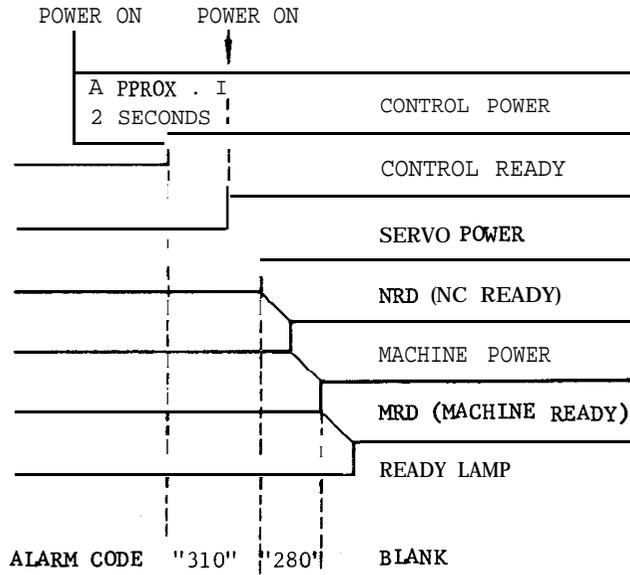


Fig. 4.11 Sequence of Turning on Operation

4.2.2 TURNING OFF POWER

Depressing the POWER OFF pushbutton causes both the servo and control powers to be turned off simultaneously. However, for more stable operation, use the following procedure.

First depress the EMERGENCY STOP pushbutton to cut off the servo power. The NRD (NC READY) signal is interrupted, which usually results in turning the machine power, too.

Depress the POWER OFF pushbutton to cut off the control power.

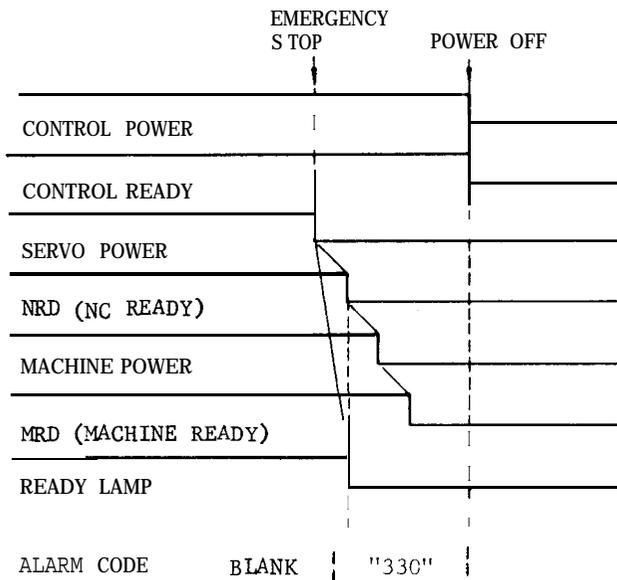


Fig. 4.12 Sequence of Turning off Operation

4.2.3 REMOTE POWER ON/OFF PUSHBUTTONS

Connect the power ON /OFF pushbuttons to EON, EOF and COM terminals on the control panel as shown below. Then the remote turning ON /OFF operation can be made exactly the same as with the POWER ON /OFF pushbuttons.

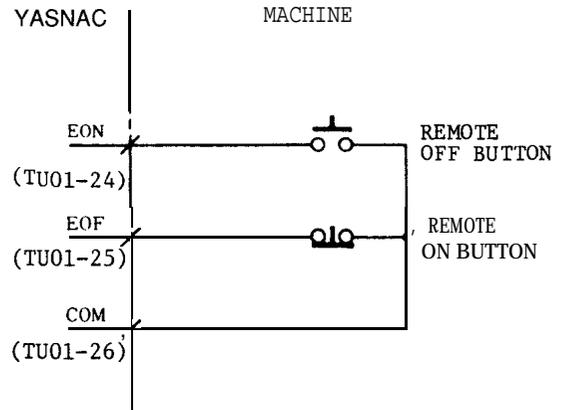


Fig. 4.13 Connections of Remote ON/OFF Pushbuttons

4.3 DISPLAY AND WRITING OPERATION

4.3.1 CONSTANT DISPLAY

The following display is made on both the top and bottom on the CRT, irrespective of the FUNCTION key currently selected.

. Function message

Any of the following eight function messages corresponding to the function key is displayed at the top of CRT display.

ALARM	PROGRAM
DIAGNOSIS	POSITION
PARAMETER	OFFSET
SETTING	
COMMAND	

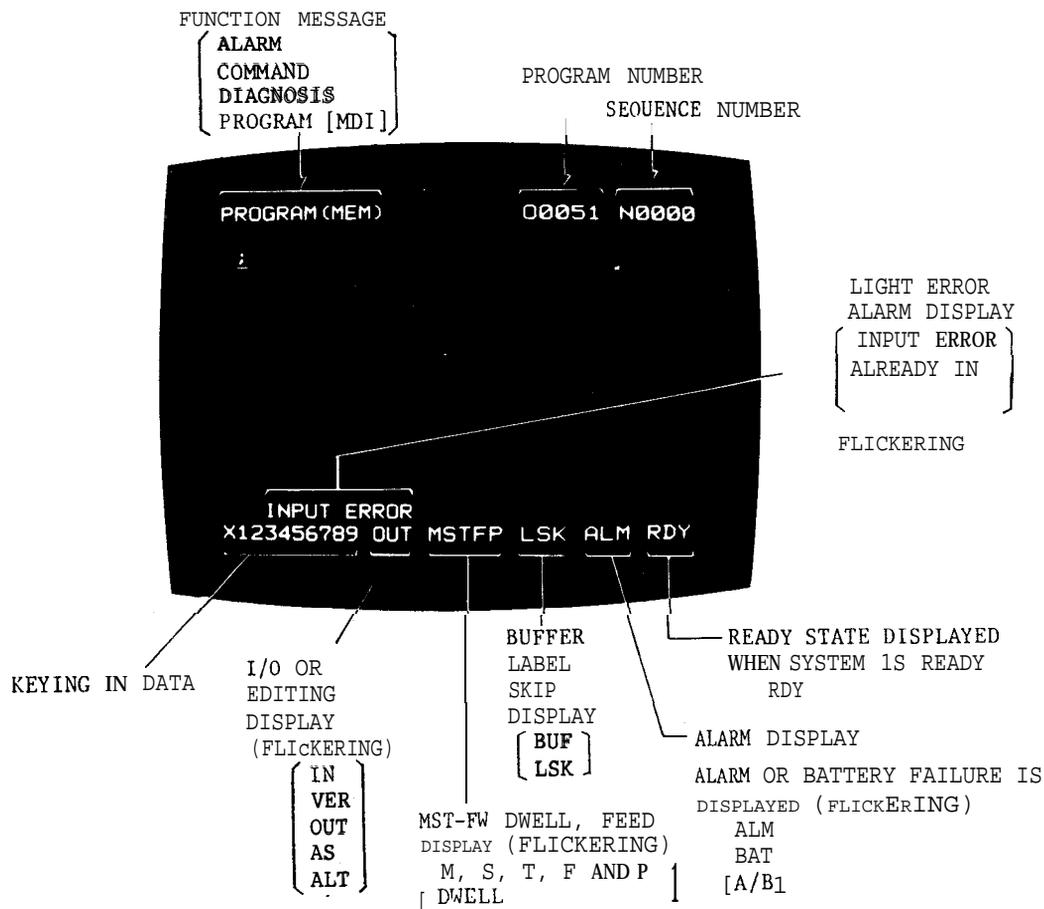


Fig. 4.14

Program No.

0 and 4 digits of the program No. under execution is constantly displayed at the top of CRT irrespective of function key.

Sequence No.

N and 4 digits of program No. under execution is constantly displayed at the top of CRT irrespective of function key.

Display of keying data

Up to 32 characters of keyed in data can be displayed at one time. The data is processed by using ERASE key, INSRT key, ALTER key, etc.

Display of I/O and editing (flashing)

The following messages are flashing on the screen during loading of punched tape, address search or editing.

- "IN" . . . loading tape
- "VER" . . . verifying tape
- "OUT" . . . punching tape out
- "AS" . . . searching address
- "ALT" . . . altering data in EDIT mode

"INS" . . . inserting data in EDIT mode

"ERS" . . . erasing data in EDIT mode

Display of MST-FIN signal waiting, dwelling and feeding

"M" . . . waiting FIN signal of M command

"S" . . . waiting FIN signal of S command

"T" . . . waiting FIN signal of T command

"F" . . . feeding
"R" is displayed at rapid traverse

"P" . . . loading tape

"DWELL" . dwelling

M, S, T, F and P are displayed independently of each other.

Display of the state of buffer full and label skip

"BUF" . . . displayed at completion of advanced reading

"LSK" . . . displayed at label skip on

Display of alarm (blinking)

Alarm continues to blink until the cause is removed and reset operation is made.

4.3.1 CONSTANT DISPLAY (Cont'd)

"ALM" . . . indicates alarm state occurring
 "BAT" . . . indicates battery alarm occurring
 "A/B" . . . indicates both of alarm and
 battery alarm occurring

. Display of ready state

"RDY" . . . indicates the system is normal
 and the control is operable

. Display of simple errors (flashing)

The messages shown below indicate simple errors which occur in keying or searching operation . Differing from the alarm codes , these error messages are cleared by depressing some key. (Generally CAN key)

"INPUT ERROR ! " . . . Format error of keyed-in data

"ALREADY IN ! " . . . The same number of part program has been stored already.

"EDIT LOCK ON ! " . . Editing operation is made with Edit Lock on.

"MEMO RY OVER ! " . . Part program to be stored is beyond memory capacity.

"PROGRAM OVER ! " . . . Registered number of part program is beyond 99 (basic) or 199 (option) .

"NOT FOUND! " . . . Desired data has not been located.

"BREAK POINT!" . . . Break point occurs

4.3.2 COMMAND DATA DISPLAY

1. Depress COM key.

Anyone of the following three screens appears.

A. Command data (COMMAND)

B. Repetition number of subprogram (SUB PROG. NESTING)

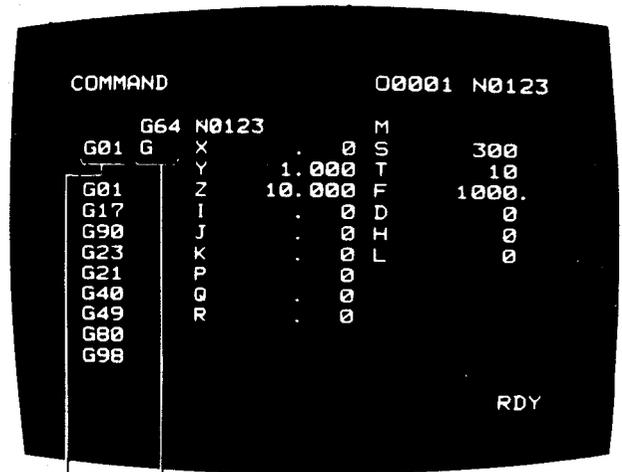
c. State Of tool offset (COMMAND [OFFSET])

2. The above display steps forward or backward by depressing PAGE ↓ or **m** key one by one.

4.3.2.1 COMMAND DATA DISPLAY

The display shows the block data under execution or just before execution in which compensation calculations have been completed. The conditions of the data to be displayed is as follows.

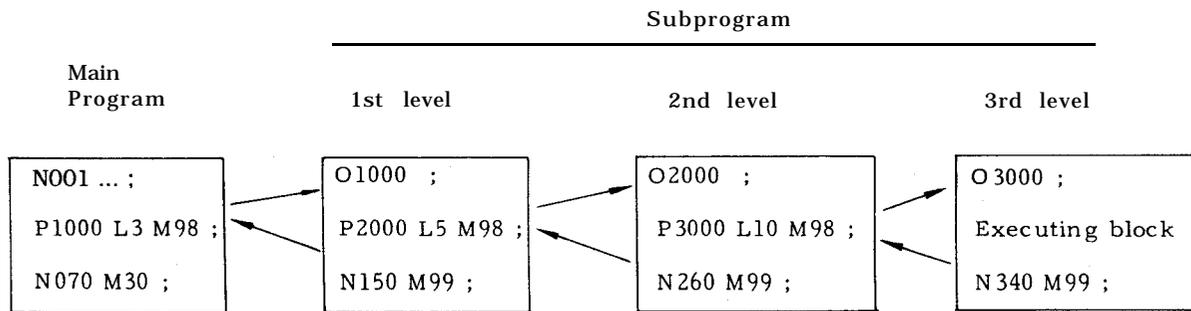
1. The data shows the contents of the active register during an automatic operation or a feed hold.
2. While the control is stopped at a block end, the contents of the buffer register are displayed. If the buffer register blank (BUF is not displayed) , the contents of the just executed block are displayed.
3. In the MD I operation , the current block data are displayed after cycle is indicated.



'Display of non-modal G code
 Display of G code under execution

4.3.2.2 DISPLAY OF REMAINING NUMBER OF REPETITIONS OF SUBPROGRAM (SUB PROG. NESTING)

The remaining number of repetitions of a subprogram is displayed.



The subprogram has executed the 3rd level twice and entered into the execution of 3rd time of the 3rd level.

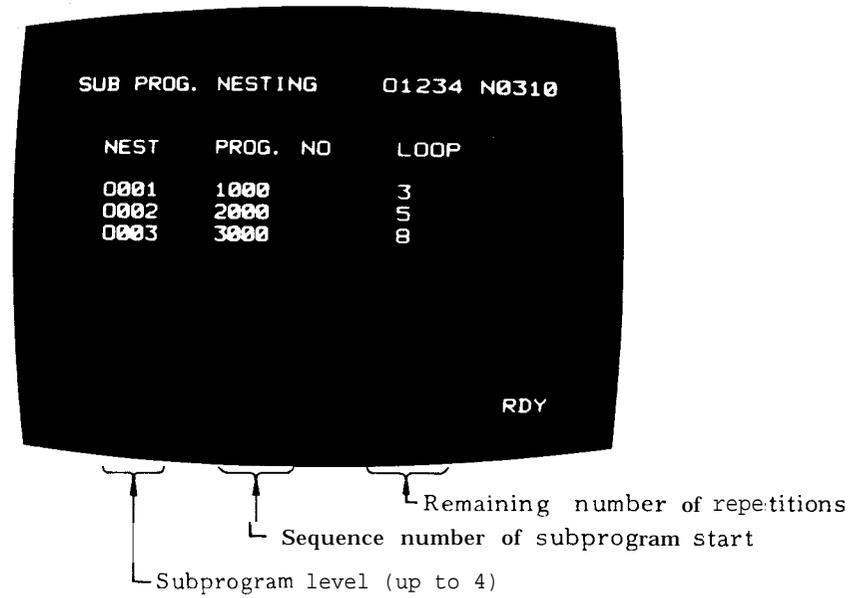
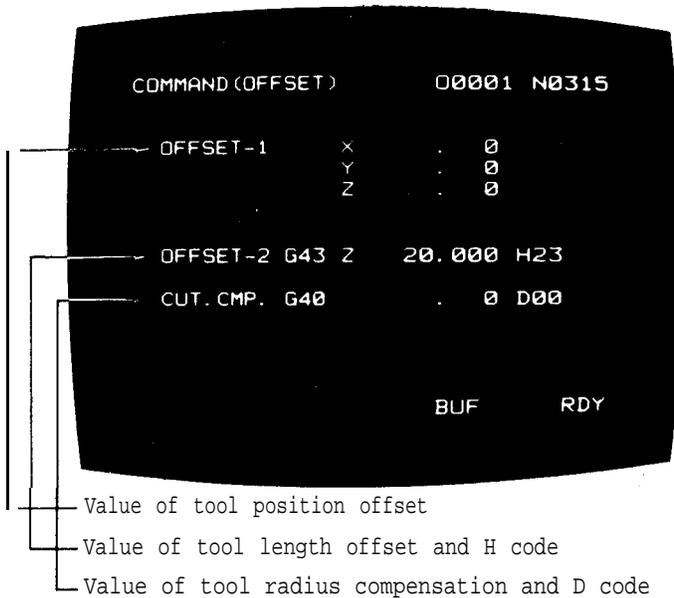


Fig. 4.15 Example of SUB PROG. NESTING

4.3,2.3 DISPLAY OF TOOL OFFSET STATE: COMMAND (OFFSET)

The current state of tool offset is displayed as shown below.



Note: Function COM is exclusively used for display. Data cannot be written under function COM. Select function PROG. to write block data.

Fig. 4.16 Example of display of tool offset state

4.3.3 WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI

In MDI, EDIT, and MEM modes, writing data into blocks by MDI and perform operation available. (MEM mode permits displaying only.) The following operations are available when function PROG. is selected.

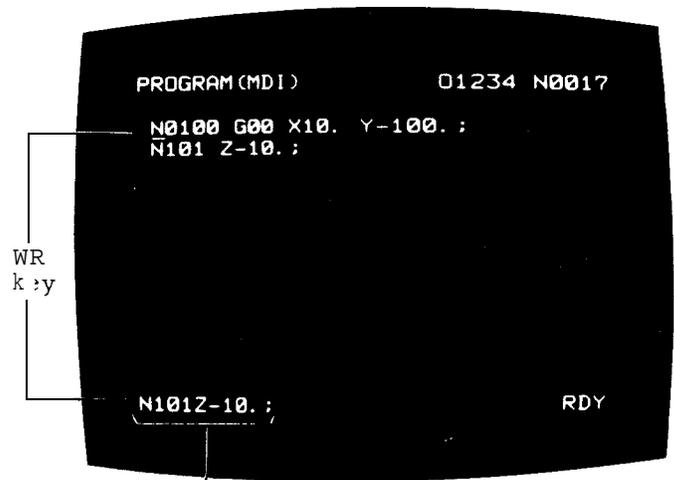
1. In MDI mode

A. Writing and displaying data

Data entered through the keyboard will appear on the bottom line of the CRT screen, from left to right. Up to 32 characters may be entered at a time.

Depress the WR key, then the data moves to the middle of the CRT and the bottom line becomes blank.

In MDI mode, data of up to 10 lines can be collected on the CRT screen.



Referred to as "the data which has just been entered."

Enter **[N]**, **[1]**, **[0]**, **[1]**, **[Z]**, **[-]**, **[1]**, **[0]**, **[.]**, **[EOB]** in this order.

Note: The depression of the EOB key appears ".,"

Fig. 4.17

B. Editing MDI data

The **[CURSOR]**, **[CURSOR]**, **[ERASE]**, **[INSRT]**,

and **[ALTER]** keys permit editing multi-block data written in. Address (word) pointed to by the cursor will be edited.

The **[CURSOR]** and **[CURSOR]** keys move the cursor forward and backward.

- (1) ERASE key: When this key has been depressed, the whole word designated is erased.
- (2) INSRT key: This key inserts the data which has just been entered to the location which is next to the word the cursor points to.
- (3) ALTER key: This key replaces the word which the cursor points to by the data which has just been entered.
- (4) WR key: This key appends the data which has just been entered at the end of the program displayed. In MDI mode, editing only the current page is available.

Note: Displaying and editing of various screens such as EDIT and MEM modes is not available.

C. Operation in MDI mode

Depress the Cycle Start button to let the program of blocks displayed on the CRT to run automatically.

At the end of operation, the multi-block program displayed is cleared from the CRT.

Programs written by MDI can be repeated by M99.

2. In EDIT mode

See 4.6 EDIT OF PART PROGRAM.

3. In MEM mode

This mode permits the display of the program which is running by memory operation. The cursor points to the top of the block which is currently being executed, and it moves to the next block as execution proceeds.

Up to 10 lines can be displayed at a time. When execution of the 9th has been completed, the next page appears with the 10th line of the last screen appearing at the top.

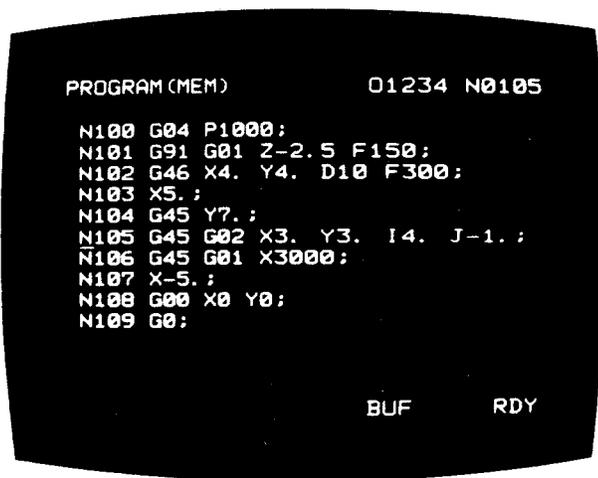


Fig. 4.18

4.3.4.4 DISPLAYING AND RESETTING CURRENT POSITION

Observing the current position is available in any mode. Operate as follows.

1. Depress the POS key. Any of the following screens will appear.

- A. Current position display-universal (pPOSITION [UNIVERSAL])
- B. Current position display-external (pPOSITION [EXTERNAL])
- c. Current position display-increment (pPOSITION [INCREMENT])
- D. Current position display-all (POSITION)
- E. Servo positioning error display (pPOSITION [ERROR])

2. Depress the

PAGE
↓

 or

PAGE
↑

 key, and one screen will change to the next.

4.3.4.1 CURRENT POSITION DISPLAY (uNIVERSAL) :

POSITION (uNIVERSAL)

The current tool position which is the sum of the parameters of move commands will be displayed. Depending on the value of Parameter #6005D5 (G92 display preset) , either of the following will appear.

- . When parameter #6005D5 = 1 (Position in the reference coordinate system)
 1. The tool position displayed is based on the coordinate system set up with G92.
 2. To reset this screen, depress the ORG key after designating an axis with the ADDRESS key. The current position will be reset to "0". This is possible only during a manual operation mode (RAPID, JOG, STEP, or HANDLE). The depression of the ORG key is invalid during normal operation and in the "buffer full" state.
 3. The coordinate system which is employed for this screen is called the "reference coordinate system". A work coordinate system (option) will be set up based on the reference coordinate system.

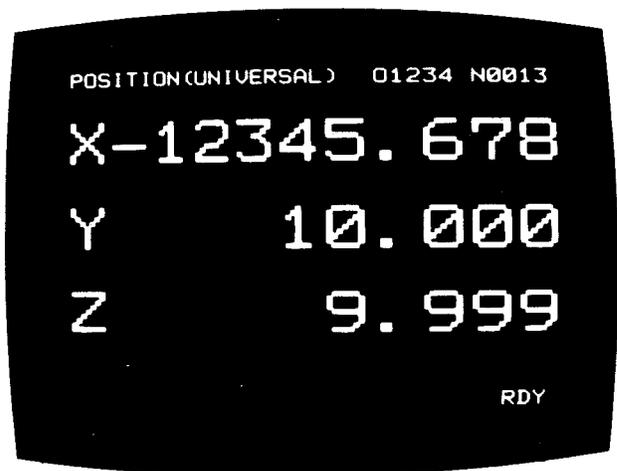


Fig. 4.19 Current Position Display (Universal) -Example

. When parameter #6005D5 = 0 (Position obtained by simple summation)

1. G92, even if issued, does not affect the display. Move commands will be added and displayed.
2. To reset this screen, depress the ORG key after designating an axis with the ADDRESS key. The current position along the designated axis will be reset to " 0. " Resetting operation is available in any modes and even during operation.

4.3.4.2 CURRENT POSITION DISPLAY (EXTERNAL): POSITION (EXTERNAL)

- Move commands will be summed and displayed. G92, if issued, does not affect the display.

To reset this screen, depress the ORG key after designating an axis with the ADDRESS key. The current position along the designated axis will be reset to "0". Resetting operation is available in any mode and even during operation.

These displaying and resetting operations are the same as with the case of POSITION(UNIVERSAL) #6005D5 = O (Position obtained by simple summation) . But the resetting operation is effective only to the displayed screen since there are independent position registers.

The data displayed in this mode are the same as those displayed on the " 3-axis/4-axis external position display" (option) . You may consider that the coordinate data of POSITION (EXTERNAL) are transmitted to the outside as they are unchanged.



Fig. 4.20 Current Position Display (External) -Example (with 4-axis control)

4.3.4.3 CURRENT POSITION DISPLAY (INCREMENT) : POSITION [INCREMENT]

Displayed in this mode are:

- . In automatic mode, distance to the end point of the block at every moment
- . In manual mode, distance to the position where manual operation is to start.

The increment display in manual mode will be cancelled in automatic mode. (Fig. 4.21)

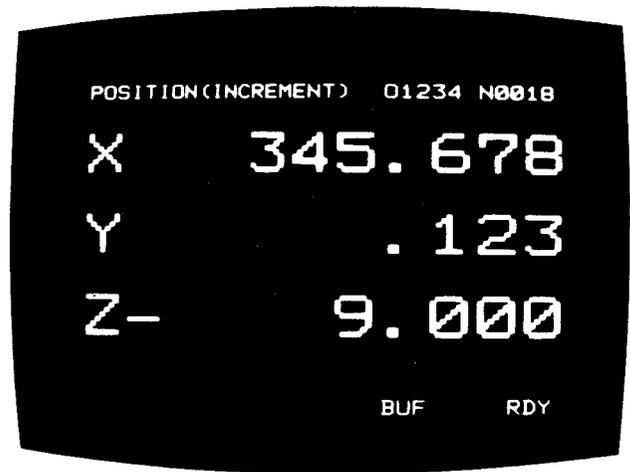


Fig. 4.21 Current Position Display (Increment) - Example

4. 3.4.4 CURRENT POSITION DISPLAY (ALL) : POSITION

- . All position data will be displayed.

< MACHINE > coordinates indicate the current position in the coordinate system whose origin is the reference point set up by resetting. Data for "stored stroke limit" and "pitch error compensation" functions are defined in this coordinate system . (Fig. 4.22)

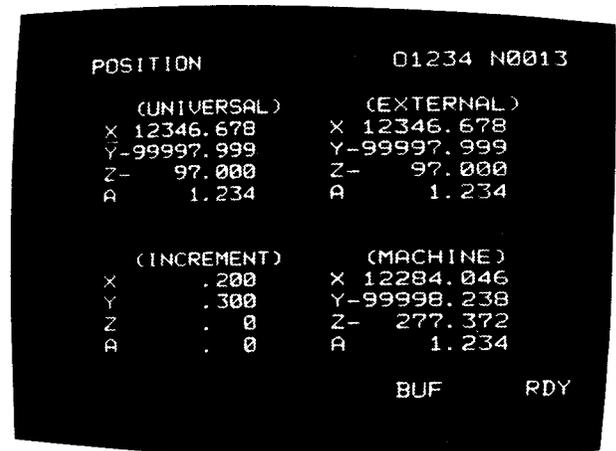


Fig. 4.22 Current Position Display (All) - Example

4. 3.4.5 SERVO POSITIONING ERROR DISPLAY: ERROR PULSE

This mode will be normally used during maintenance.

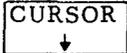
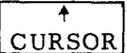
Servo positioning error means the difference between the command position and the current tool position. Error will be displayed in units of pulse.

4.3.5 DISPLAYING AND WRITING TOOL OFFSET AND WORK OFFSET

4.3.5.1 DISPLAYING AND WRITING TOOL OFFSET DATA

Tool offset data are stored in the memory of the control. These data can be displayed and rewritten in any mode and even during automatic operation.

. Displaying tool offset data.

1. Select the OFS function key.
2. Enter numerals, like 1 and 0, then depress the  or  key. Then ten pairs of tool offset number and tool offset, including the designated pair, will be displayed and the cursor positioned at the designated tool offset number.

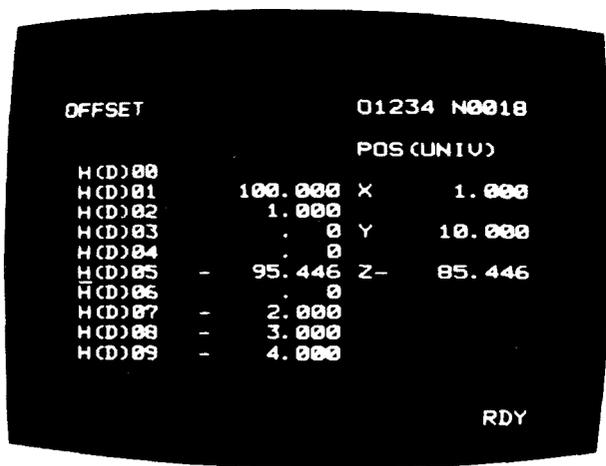
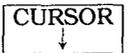
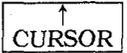


Fig. 4.23 (a)

3. Depress the  or  key to move to a smaller or larger tool offset number. If the cursor moves beyond the first or last tool offset number displayed in the current screen, the neighboring ten sets of tool offset number and tool offset will appear automatically.
4. The preceding or following page can be displayed by depressing the  or  key. The cursor will be positioned at the first tool offset number displayed on that page.
5. Tool offset will be displayed in units of 0.001 mm (or (0.0001") and up to 999.999 mm (or 99.9999").

. Writing tool offset data

To rewrite a tool offset data, specify an increment which is to add arithmetically to a tool offset data held in memory.

1. Position the cursor at the tool offset number whose offset data is to be changed.
2. Enter the increment which is to be added to the tool offset.
3. Depress the WR key. Then the specified increment will be added to the old tool offset.

Notes :

- . A new tool offset itself can be input instead of an increment. For this purpose, depress the ORG key first. The tool offset number pointed by the cursor will be reset to "0". Then enter a new tool offset.
- . Tool offset data held in the memory of the control are preserved even after power is turned off.
- . Rewriting and modifying tool offset data are available in any mode, even during automatic operation.
- . Tool offsets modified during automatic operation become effective when the system starts to read commands for a new block. The old tool offsets remain effective for the current block and the blocks whose data are already read in the buffer for advance reading.

4.3.5.2 DISPLAYING AND WRITING WORK OFFSET*

(1) Work offset display

(a) Select the  function key.

(b) Depress the  key until the screen shown in Fig. 4.23 (b) appears.

(2) Writing the work offset

(a) Depress the  key. Then enter the "work offset amount."

(b) Depress the  key to write the X-axis work offset data.

(c) Write the offset data of other axes in the same manner.

(d) Select G54J1 to J5, through G59J1 to J5, by the  key or  key.

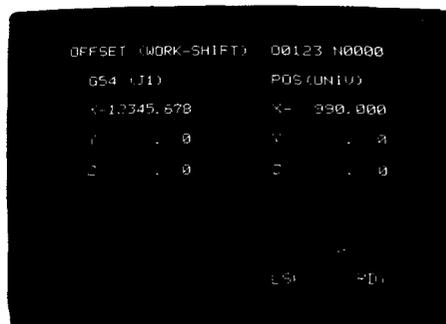


Fig. 4.23 (b) Example of Work Offset Display Screen

4.3.6 DISPLAY IN THE (SETTING) FUNCTION

4.3.6.1 DISPLAYING AND WRITING SETTING DATA

In this system, various setting data are held in the internal memory and permit to specify mirror image axes, TV check on/off, etc. For details, see Appendix 1, "LIST OF SETTING NUMBERS".

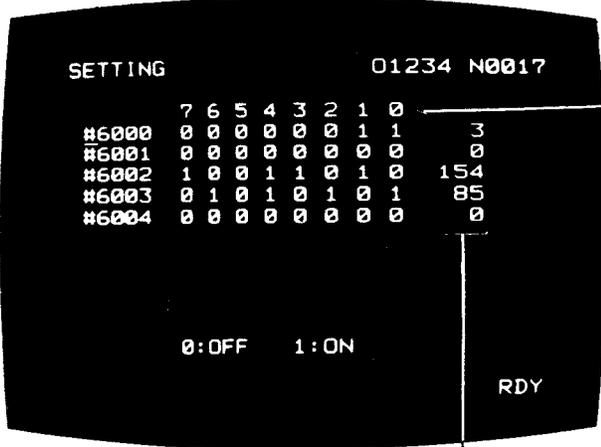
It is possible to display and write setting data at any time even during automatic operation.

1. Types of setting

Setting is made in binary mode or decimal mode.

A. Binary mode

Setting numbers #6000 -#6004 are associated with setting data of binary mode, that is, 8-bit information (D7-D0). Each bit indicates the on /off state of the associated function. The decimal value of each line is given at the rightmost column.



SETTING	7	6	5	4	3	2	1	0	DECIMAL VALUE
#6000	0	0	0	0	0	0	1	1	3
#6001	0	0	0	0	0	0	0	0	0
#6002	1	0	0	1	1	0	1	0	154
#6003	0	1	0	1	0	1	0	1	85
#6004	0	0	0	0	0	0	0	0	0

0:OFF 1:ON

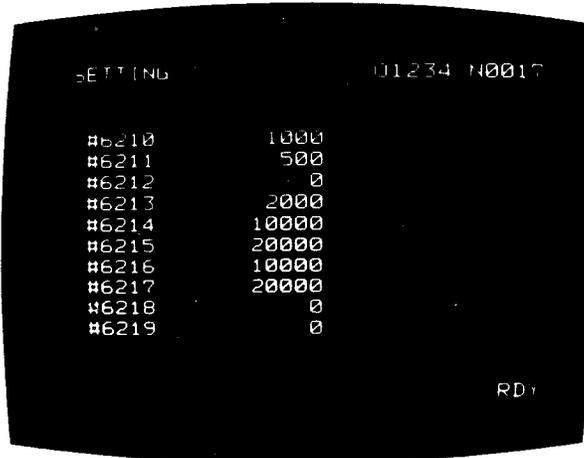
RDY

DECIMAL VALUE

Fig. 4.24 Setting (Decimal model) -Example

B. Decimal mode

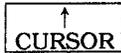
Setting numbers of #6200 -#6219 and #6500 - #6599 are associated with setting data of decimal mode.



#6210	1000
#6211	500
#6212	0
#6213	2000
#6214	10000
#6215	20000
#6216	10000
#6217	20000
#6218	0
#6219	0

RDY

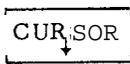
2.

Enter a setting number and then depress the   key. (“#” need not be entered.) Up to 10 groups of setting number and data will be displayed at a time.

Depress the CURSOR keys to change a setting number and the PAGE keys to change a screen,

3. Writing setting data

A. In binary mode

- (1) Designate a desired setting number.
- (2) Depress the INSRT key. The cursor moves to the bit data from a setting number. Designate the data of D7.
- (3) Depress the  key. Each time the key is depressed, the cursor moves one bit toward D0. Locate the cursor at a desired bit position.
- (4) Depress the WR key : The designated bit data reverses (0 to 1 or 1 to 0). If the WR key is depressed again, the bit data will reverse again. Normally, “ 1” designates on state and “0” off state.
- (5) To write data in decimal mode, locate the cursor at the rightmost column (decimal data) .

EXAMPLE : Writing in decimal mode

Entered data	7	6	5	4	3	2	1	0
0 WR 5	0	0	0	0	0	0	0	0
5 WR	1	1	1	1	1	1	1	255

(6) Repeat steps (2) through (5) to write desired data. If you keep the **m** or **CURSOR** key depressed, the cursor will move column by column in the screen automatically.

(7) When data has been written, depress the INSRT key. Normally, this sequence of operations begins and ends both with the depression of the INSRT key.

B. [n decimal mode

- (1) Designate a desired setting number.
- (2) Enter a data and depress the WR key. The data will be assigned to the setting number which the cursor points to.

(3) Depress the **CURSOR**, **CURSOR**, **PAGE** or **PAGE** key to change a setting number or the screen.

4.3.6.2 DISPLAY AND WRITING IN THE F1-DIGIT DATA INPUT SCREEN*

(1) Display of the F1-digit data

(a) When the F1-digit option is provided, display the screen shown in Fig. 4.25 (b) as follows.

(b) Select the **BET** function key.

(c) Depress the **NEXT** key, and the screen appears.

(2) Writing the F1-digit data

(a) Move the cursor in the F1-digit data input screen to where the data is to be written.

(b) Enter the desired numeral.

(c) Depress the **WR** key. The entered numeral is now written.



Fig. 4.25 (b) Example of F1-Digit Data Input Display Screen

< REMARKS >

. System No. setting (#6219)

Set system No. at "1" to write parameter number. System number can be set by setting the value of #6219 through the operator's panel.

Setting of #6219

"0" : SYSTEM

For normal operation. Writing parameters is prevented.

"1" : PARAMETER

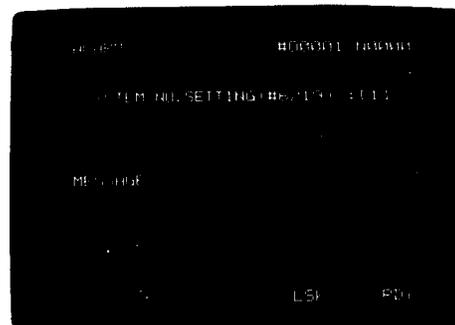
For writing parameters. At this position, cycle start is prevented.

1. Setting values other than those described above will prevent correct operation.
2. Setting can be effective only when the system number switch provided on the upper part of the unit is set at "0."

Alarm Display of System No. Setting (#6219)

When setting #6219 is set at "1" or "4, cycle start is prohibited or self-diagnostics are over looked, causing unexpected errors.

To prevent this, alarm status is displayed if #6219 is set at a value other than "0".



4.3.6.2 DISPLAY AND WRITING IN THE F1-DIGIT DATA INPUT SCREEN* (Cont'd)

System number #6219 should be set at "0" except for writing parameters.

System number can also be set by the rotary switch on the main printed circuit board. For details, refer to Maintenance Manual.

4.3.7 DISPLAYING AND WRITING PARAMETERS

In this system, various parameters are stored in the memory and they determine operating conditions such as tape code and feed rate. For details, see Appendix 2, "LIST OF PARAMETER NUMBERS". The parameters can be displayed at any time even during automatic operation.

1. Kinds of parameters

Parameters are displayed either in decimal mode or in binary mode.

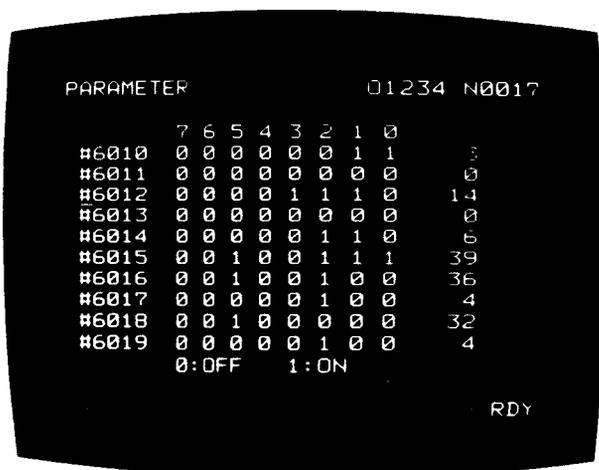


Fig. 4.26 Parameters (in binary mode) -Example

Parameter numbers #6005-#6099 are assigned to binary mode. Those of #6050 and up are assigned to decimal mode.

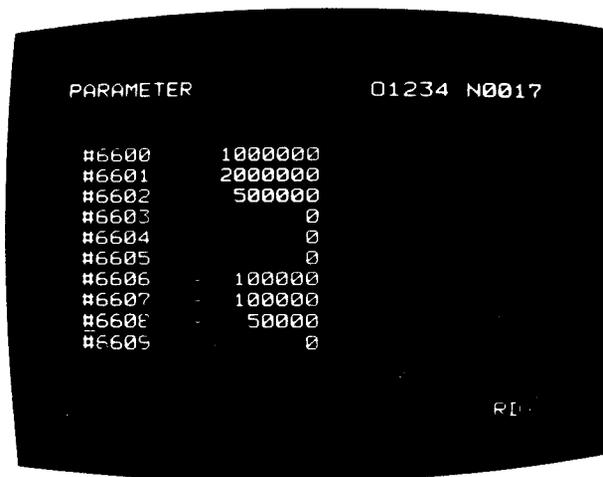


Fig. 4.27 Parameters (in decimal mode) -Example

2. Displaying parameters

Operation is the same as in displaying setting data except that PRM should be depressed instead of SET. See 4.3.6 "DISPLAY IN THE SETTING FUNCTION".

3. Writing parameters

The parameter values are preset according to the performance of the machine and purposes. Therefore, you should consult the machine tool builder if you want to change parameter settings.

The parameter are protected with a system No. switch provided on the top of control unit main package so that they should not be destroyed by wrong operation. Normally system No, 0 is selected and, at this time, the parameters cannot be rewritten by any operation.

A The operation of writing parameters is the same as of writing setting data but the parameters are protected. See 4.3.6 "DISPLAY IN THE SETTING FUNCTION".

B Parameters cannot be rewritten unless the system No. switch is set at "1. "

After rewriting parameters, be sure to re-set the system No. switch at "0. "

C. If the following parameters have been changed, be sure to turn off power then turn it on again. Otherwise the system might fail to operate properly.

#6050 to #6055 (Servo control)
 #6056 to #6061 (PG multiplication factor)
 #6062 (Skip)
 #6063 (Skip)
 #6156 to #6161 (Error detect area)
 #6280 to #6284 (Rapid traverse rate)
 #6286 to #6290 (Linear accel/decel time 1)
 #6292 to #6296 (2-step accel/decel switch speed)
 #6298 to #6302 (Linear accel/decel time 2)
 #6400 to #6404 (Backlash compensation amount)
 #6406 to #6411 (Move amount/ motor rotation)
 #6444 to #6448 (Minimum move amount)
 #6450 to #6454 (Servo input gain)
 #6456 to #6411 (Kp)
 . After reading-in of parameter tape

Setting data	Machine's switch	Result on/off
"0" = OFF	OFF	OFF
"0" = OFF	ON	ON
"1" = ON	OFF	ON
"1" = ON	ON	ON

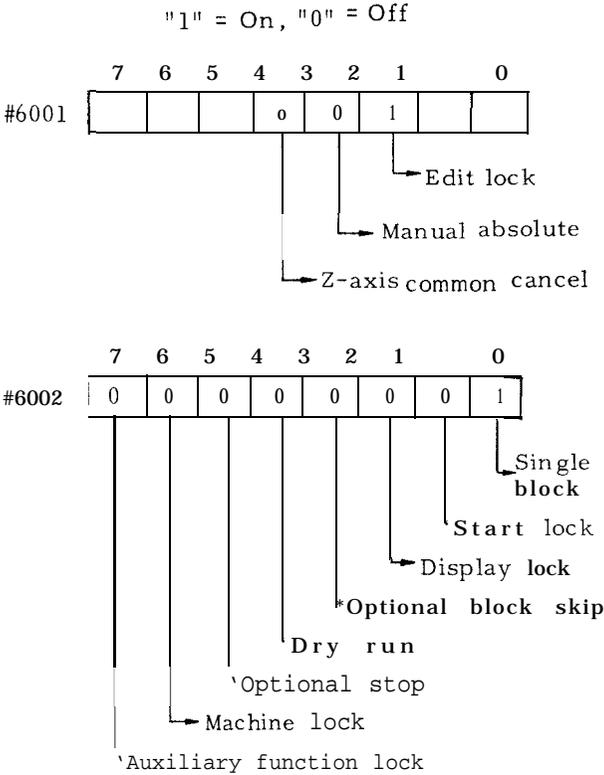
The functions of the internal toggle switches work only when parameter #6006D3 = 1 (internal toggle switch function on). If it is off, only the switches of the machine control panel work.

4.3.9 OPERATION TIME DISPLAY

The system counts the duration of automatic operation and it can be displayed. This function permits the display of the time it has been taken for a piece of work or the total operational time of the system.

4.3.8 INTERNALTOGGLE SWITCHES

The following switches maybe easily turned on and off on the NC operator's panel even when they cannot be operated on the machine control station. Setting numbers and their contents are as follows.



If the machine control station is provided with the switches that turn on and off the above machine functions, the state of a switch on the machine's control panel is ORed with that of the NC operator's panel will determine the machine condition.

1. Procedure of display

Depress the ALM key, then select a screen of running time with the PAGE key as shown below.



Three kinds of operation time will be displayed in hours, minutes, and seconds.

Top : Total operating time after POWER ON

Middle : Total operating time of CYCLE START

Bottom: Total operating time of FEED

2. Resetting display

Each operation time display can be reset independently by the following procedure.

When operating times are displayed :

- A. "1" "ORG" POWER ON time at the top will be reset
- B. "2" "ORG" CYCLE START time in the middle will be reset.
- c. "3" "ORG" FEED time at the bottom will be reset

The timers of operation time preserve data unless they are reset, even after power is turned off.

4.3.9 OPERATION TIME DISPLAY (Cont'd)



Fig. 4.28

4.3.10 ADDRESS SEARCH

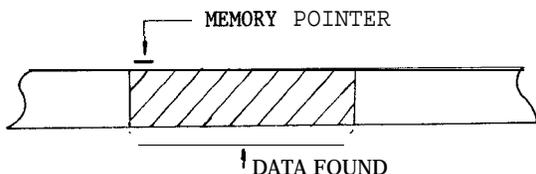
Search continues until the data (character string) held in the memory which coincides with the data (character string) entered through the NC operator's panel is found. The contents of the part program memory will be searched in MEM or EDIT mode.

1. Operation

- A. Select MEM, or EDIT mode,
- B. Depress the PROG function key.
- C. Depress the RESET key. "LSK" appears and the cursor returns to the top of the program number in MEM mode.
- D. Enter the data (string of within 10 characters headed by address) to be searched.
- E. Depress the  key to start searching.
"AS" blinks during search.

2. End of search

- A. "AS" disappears when search is completed.
 - (1) The pointer of the part program memory points to the top of the data found (pointed by the cursor). In all cases, only search will be performed but neither BUF display nor advance reading will be performed.



- B. "AS disappears and "NOT FOUND!" appears on the CRT if the desired data is not found. This message will disappear when a key (CAN normally) of the control panel is depressed.

3. Remarks

When #6022 D6 = 0, the "reading zero" cannot be omitted, and the search is performed in character string units. Searching N5 outputs N5/ N5?/N5??/ N5??? (? represents decimal point or numeral of 1 to 9). The data input from the key and the part program memory data are verified.

When #6022 D6 = 1, "reading zero" can be omitted, to search in address units. Searching N5 outputs N5/ N05/ NO05/ NOO05. When #6022 D6 = 1, the same search can be performed as when #6022 D6 = 0 by depressing the NEXT key. The "reading zero" can be omitted in registered "program number" search, regardless of the setting in #6022 D6.

Commands encountered during search will be ignored even if they are modal commands.

On Cycle Start after search, the data of a block' which the cursor points to will be read and executed.

Address search cannot be made in TAPE mode.

4. Search of program number

The address search function also permits the search of a part program which is stored in the memory.

- A Select MEM or EDIT mode.
- B Depress the PROG function key.
- C Depress the RESET key.
- D. Enter the program number "O□□□□".
- E. Depress the  key.

The designated program number will be searched. The result of search is as described in 2. In MEM mode, the CYCLE START button can be depressed immediately after completion of search to start automatic operation from the beginning of the program.

4.3.11 ALARM CODE DISPLAY

If an alarm status has happened, "ALM" or "A/B" (on battery alarm) blinks on the bottom line of the screen regardless of working mode and function. If this happens, the detailed information of the alarm status can be displayed by the following operation.

1. Depress the ALM key.

Then up to four pairs of alarm code and message will be displayed, with more serious one on a higher line.

Note : The alarm screen will appear during an alarm state and therefore, depressing the  key is not necessary.

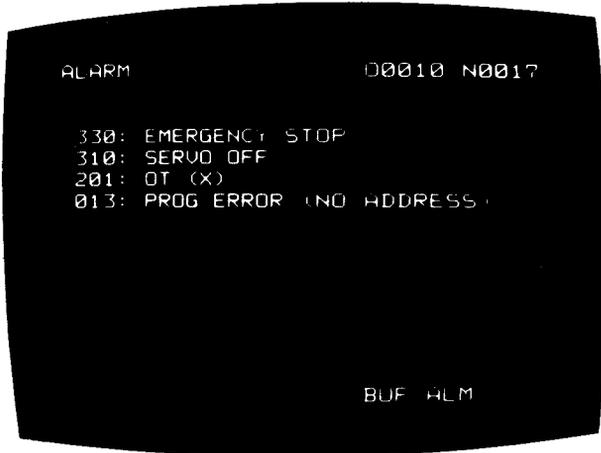


Fig. 4.29 Alarm Codes and Message Displayed-Example

To reset the alarm status and screen, remove the cause of alarm then depress the RESET key.

For details of alarm codes, see Appendix 5, "LIST OF ALARM CODES".

4.3.12 DISPLAYING ON/OFF INPUT/OUTPUT SIGNALS

Depress the DGN function key, and the state of every input/output signal will be displayed on the CRT. Displaying the state is available at any time even during automatic operation.

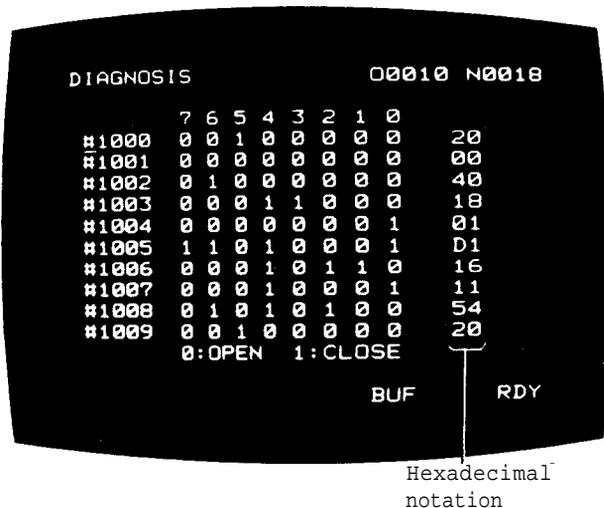


Fig. 4.30 State of I/O Signals Displayed-Example

For more detail of this operation, see 7.6.3, "INPUT/OUTPUT SIGNALS".

The state of the input/output signal is also given in the hexadecimal notation at the right-most column for the ease of maintenance work.

4.4 TAPE INPUT/OUTPUT OPERATIONS OF NC DATA

Such NC data as tool offsets, setting data, and parameter data can be read from and written onto tape. A tape reader can work to read data from tape. To write data onto tape, a data input/output interface (option) is needed.

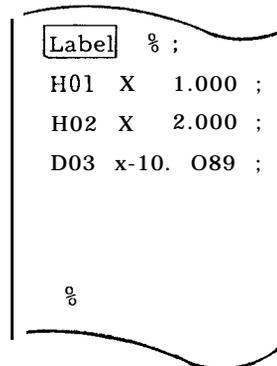
Here we assume that this option is incorporated.

See Par. 4.7.2 for how to set the type of input/output device (setting #6003) and baud rate (parameter #6026).

4.4.1 INPUTTING TOOL OFFSETS FROM TAPE

Though tool offsets are normally input by MDI operation, they can also be entered by means of paper tape.

1. The tape format of tool offsets is as follows.



Note :
Either H or D can be used for address.

2. The input operation is as follows

- A. Select EDIT mode.
- B. Depress the RESET key.
- C. Depress the OFS key.
- D. Set the tool offset data tape onto the tape reader.
- E. Depress the IN key.
The tape reader starts reading the tape.
"IN" blinks on the CRT while the data are read.
- F. The tape reader stops when it has read "%" (or ER) and "IN disappears from the CRT.
Now the tool offset data have been read into memory.

i. Depress the OUT key.

The part program of the keyed-in program number is output to the external equipment. If it is a tape puncher, tape punch is performed. When the output of the part program is completed, the tape puncher stops automatically. During the output of data, "OUT" is blinking.

j. To abort the punch out operation, depress the RESET key. However, the continued operation cannot be resumed.

Go back to f. and repeat the operations again.

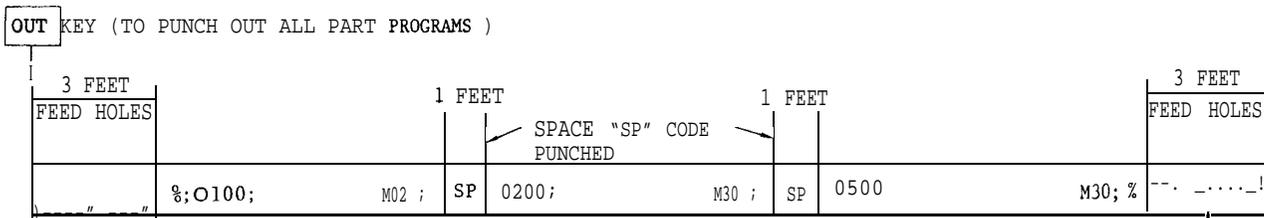
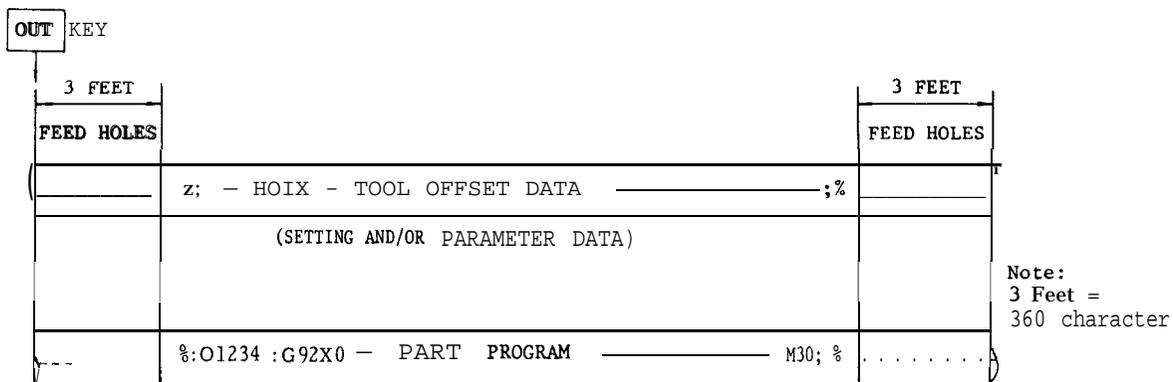
Note: When RESET, OUT are operated without keying in the program number, the part program of the currently displayed program number is outputted.

(2) Punch out of all part programs

All registered part programs should be output to the external equipment by the above operations except that O -9999 must be keyed in the operation of h.

All part programs stored in the memory are output (punched out, etc.) consecutively.

Note: The contents of program number 00000 are output only when #602 1 D₃ = 1.



DEPRESS THE CAN KEY, AND THE PAPER TAPE PUNCH IMMEDIATELY BEGINS TO PUNCH SIGNIFICANT DATA WHICH BEGIN WITH "%: . . ."

DEPRESS THE CAN KEY, AND THE PAPER TAPE PUNCH STOPS TO PUNCH SPROCKET PERFORATION.

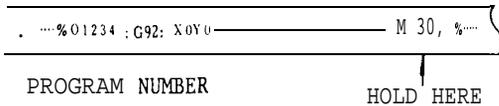
Fig. 4.31 Data and Program Formats on Paper Tape

4.5 LOADING PART PROGRAMS INTO MEMORY

4.5.1 LOADING PART PROGRAM TAPE INTO MEMORY

(1) Loading a part program which has a program number.

- a. Select EDIT mode.
- b. Depress the PROG key.
- c. Load the NC tape to the tape reader or an equivalent external equipment.



- d. Depress the RESET key.
- e. Depress the IN key.

Then the system starts reading the tape and enlists the program number punched on the tape as the first record. The system checks for a duplicate of the program number as in 1.

If the program number is duplicated, "ALREADY IN!" blinks on the CRT screen,

If this happens, delete the program number, then repeat steps a. through f.

While the tape is being read, "IN" blinks on the CRT.

- f. When the tape reader has read "M02;," "M30;," or "M99;," it stops and "TN disappears from the CRT. Now the part program has been stored in memory.

(2) Loading a part program which has no program number.

- a. Select EDIT mode.
- b. Depress the PROG keys.
- c. Load the NC tape to the tape reader or an equivalent external equipment.



The tape stops at this location when loading is completed.

- d. Depress the RESET key.
- e. Depress the "O" key and then enter the program number.
- f. Depress the IN key.

The system starts reading the tape. If the keyed-in program number coincides with the registered program number, "ALREADY IN" blinks on the CRT screen. If this happens, delete the program number, then repeat steps a. through f. While the tape is being read, "IN" blinks on the CRT.

- g. When the tape reader has read "M02 ; , " "M30 ; , " or "M99 ; , " it stops and "IN" disappears from the CRT. Now the part program has been stored in memory.

Notes :

1. program number "00000" is always in the registered state, so it cannot be erased. This program number should not be used.
2. The tape which has no program number can be stored as described before. However, write a program number to the head of the tape, in principle. The operation of "Oxxxx IN" described before causes only program number registration. It does not cause the storing of information of "OxXxx" into the part program memory. Only the program number on tape is stored into the memory. Assume that a tape having no program number is stored and then all part programs are punched out by depressing "O", "-", "9", "9", "9", "9" and "OUT" keys. Since this tape contains programs with no program number, the correct restoring of all part programs may not be performed by depressing "O", "-", "9", "9", "9", "9" and "IN" keys.
3. Consequently, when a tape having no Program number has been stored, write the program number to the head of part program by the EDIT operation.

EXAMPLE :

N1 G92 XO Z0 ;

When this is in the first block, move the cursor to N and key in as follows (in EDIT and PROG modes) :

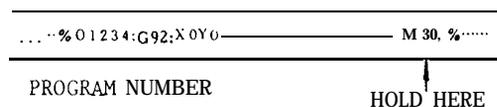
Oxxxx ; N1 **ALTER**

Move the cursor to an arbitrary word of the block to be deleted and depress the : and the ERASE keys. blocks from the cursor indicating word to the EOB (;) can be deleted.

(3) Storing a program with program numbers changed I

To register a program with a program number different from the one punched on tape, perform the following operations:

- a. Select the EDIT mode.
- b. Depress the PROG key.
- c. Set the NC tape to the tape reader or the external equivalent equipment.



- d. Depress the RESET key.
- e. Depress "O" key and enter the program number.

f. Depress the IN key.

The program number entered from the key is registered in preference to the program number punched on the tape. At this time, the program number on the tape is written to the part program memory simply as a label.

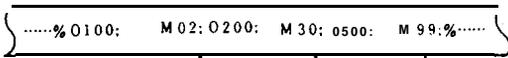
g. M02 ; , M30 ; or M99 ; is read and the storing operation is completed.

Note :

If a program is stored with a changed program number as described above, the program number punched on the tape is stored in the part program memory without change. Consequently, to avoid the confusion in the later handling, replace the program number in the part program memory with changed program number by the EDIT operation.

(4) Loading part programs from a tape

- a. Select EDIT mode.
- b. Depress the PROG key.
- c. Load the NC tape to the tape reader or an equivalent external equipment.



The tape stops to travel here. The tape stops to travel here. The tape stops to travel here.

- d. Depress the RESET key.
- e. Depress the IN key.

Then the system starts to read the tape and enlists the program number punched on the tape as the first record. The system checks for duplication of program number as described in (2) .

The tape reader stops each time it has read "M02 ; , " "M30 ; , " or "M99 ; ."

f. Depress the IN key again.

The tape reader resumes to read the tape. Repeat this operation until all programs are loaded.

(5) Storing a program with program numbers changed II

When "O" key is depressed and program number is keyed in before depressing IN key as described in (4) above, the keyed in program number is registered in preference to the program number punched on the NC tape.

(6) Loading programs continuously

programs existing on a tape as shown in (4) may be loaded continuously without interruption. For this purpose, depress "O, "-", "9", "9", "9" and "9" before the first depressing of the IN key. The tape reader stops at the position of "/0".

4.5.2 MAKING ADDITION TO A PART PROGRAM

Perform the following operation to add data to a part program which is already loaded.

- a. Select EDIT mode .
- b. Depress the PROG key
- c. Depress the "O" key and then enter the part program number and depress the CURSOR key.
The system searches the designated program.
- d. Load the tape of adding data to the tape reader.



- e. Depress the RESET key.
- f. Depress the NEXT and IN keys in this order.
The data will be read from the tape into the memory.

Notes :

- 1. Adding data to a program from the middle of it is valid. If necessary, delete the last part of the program by editing operation and perform this adding.
- 2. Resetting operation is required each time of NEXT IN continuous storing operation.

4.5.3 LOADING PART PROGRAMS BY MDI

Part programs can be loaded not through the tape reader but by MDI operation. Perform the following.

- a. Select EDIT mode.
- b. Depress the PROG key
- c. Depress the RESET key .
- d. Depress the "O" key and then enter the part program number and depress the WR key.

The designated program number has already been registered, "ALREADY IN blinks and, in this case, deleting the registered program number is required.

4.5.3 LOADING PART PROGRAMS By MDI (Cont'd)

- e. Write the part program by operating the address key and the data key. As shown in the figure below, the keyed in data is displayed on the bottom line from left to right sequentially. The maximum number of characters that can be written at a time is 32. Within this limit, data may be keyed in over multiple words or blocks. However, when the 10th character is keyed in, the normal display shown to the right of the line is blanked.
- f. Depress the INSRT key.
The keyed in data is stored in the part program memory.
- g. Repeat the operations of e. and f. above to write the part program. The program edit operation is enabled by the use of ERASE, INSRT, and ALTER keys during this program storing operation.

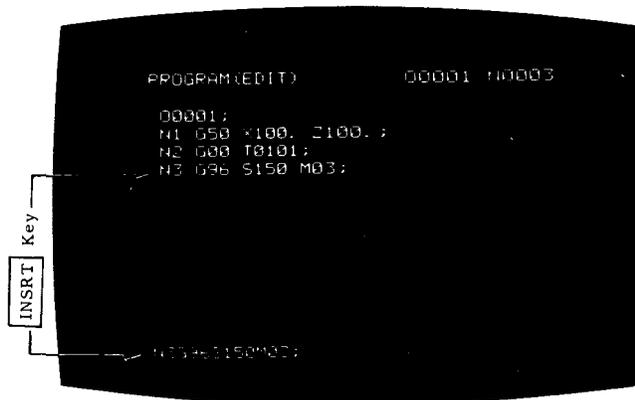


Fig. 4.32

- h. Key in M02 ; , M30 ; , or M99 ; and depress INSRT key. This completes the storing of the part program.

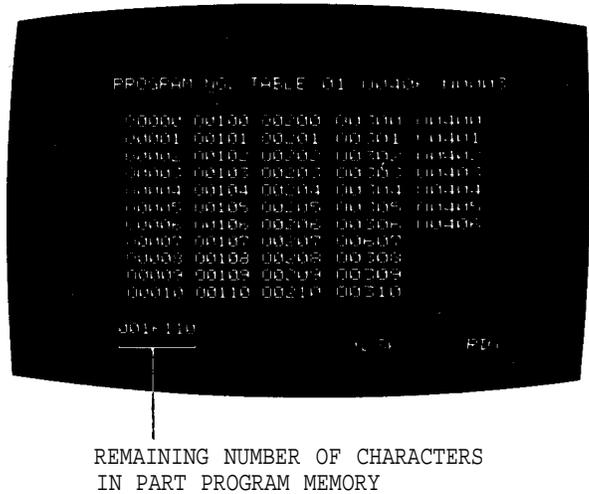
4.5.4 DISPLAY OF REGISTERED PROGRAM NUMBER

This screen displays all registered program numbers and the number of remaining characters in the part program memory.

- (1) The number of program numbers that can be registered depends on options.

No.	Max. Number of Programs	Type	Program No. Table
1	99	Basic	01 to 02
2	199	Option 1	01 to 04
3	999	Option 2	01 to 19

- (2) All program numbers already registered are displayed. By depressing **PAGE** or **PAGE** key, the page shown below may be obtained.



REMAINING NUMBER OF CHARACTERS IN PART PROGRAM MEMORY

Fig. 4.33

The remaining number of characters in part program memory is displayed in the lower left corner of the screen.

Notes "

1. This screen displays the registered program numbers or permits program number deletion. A program number is registered by depressing the PROG function key in EXIT mode.
2. When program number is increased by specifying options 1 and 2, part program area will be reduced.
199 program numbers: 800 characters decreased
999 program numbers: 7200 characters decreased
3. Program number will also be displayed with macro-interlock.

4.6 EDIT OF PART PROGRAM

4.6.1 STORED PART PROGRAM DISPLAY

Stored program contents can be displayed on the CRT, and checked by the operator.

Part Program Call

1. Select the EDIT mode.
2. Depress the RESET and PROG keys.
3. Depress the "O" key and enter the program number.

Then, depress the **CURSOR** key.

The specified program number will be searched and the data of 10 lines from the beginning of program will be displayed on the CRT . If the program number is not found by searching, "NOT FOUND" will flicker. The display will be reset by depressing the CAN key.

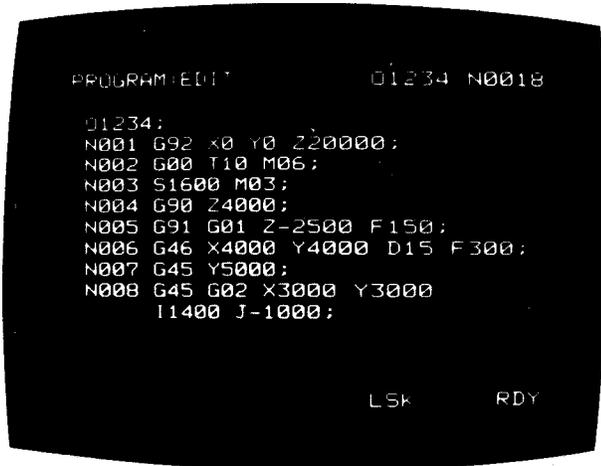


Fig. 4.34

Operation of PAGE and CURSOR key

1. Page keys  and  respectively advances and returns by one page.
2. CURSOR keys  and  respectively moves the CURSOR after and before a word.

Operation in the MEM mode

Searching can be performed in the MEM mode. However, page and cursor cannot be moved by PAGE and CURSOR keys.

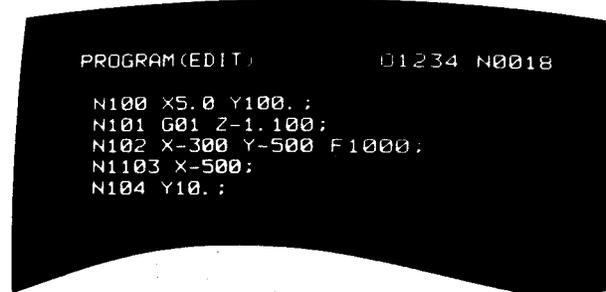
4.6.2 DELETING PART PROGRAM BLOCK (ERASE KEY)

Part programs can be deleted using PROG keys in the EDIT mode.

Deletion of Words

Move the CURSOR to the word to be deleted and depress the ERASE key. The CURSOR-indicated word will be deleted.

Before Deletion



After Deletion

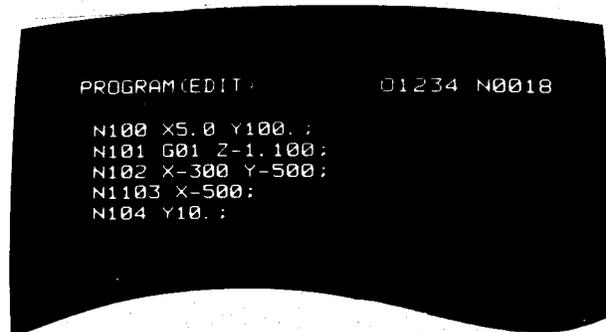


Fig. 4.35

Deletion of Blocks

Move the cursor to an arbitrary word of the block to be deleted and depress the ; and the ERASE keys, blocks from the cursor indicating word to the EOB (;) can be deleted.

Deletion of Program No.

Enter the program No. with address O and depress the ERASE key. The input program No. and its part program will be deleted.

Deletion of All the program numbers

Input o -, 9, 9, 9, 9 and depress the ERASE key. All the registered program numbers will be deleted. Program No. " O" is registered newly in the form of EOB .

Note: Specified program No. or all the program numbers can be deleted on the program number directory display by using ALM function key.

4.6.3 MODIFYING PART PROGRAM BLOCK (ALTER KEY)

Program modification is made using PROG key in the EDIT mode.

Word Modification

Specify the word to be altered with the CURSOR key in the new word, and depress the ALTER key. The new word will replace the CURSOR-indicated word.

4.6.3 MODIFYING PART PROGRAM BLOCK (ALTER KEY) (Cont'd)



Fig. 4.36

The input word less than 32 characters can be replaced as one group.

Program No. Modification

Specify the program number of the program head with the CURSOR, key in the new number, and depress the ALTER key. The new number will replace the CURSOR-indicated number and upper right number. When this function is used, set parameter #6008 D5 to "1".

4.6.4 ADDING PART PROGRAM (INSRT KEY)

Programs will be inserted using PROG key in the EDIT mode.

Insertion of words

Specify the word before the word to be added using CURSOR, key in the data to be added, and depress the INSRT key. The new data will be inserted immediately after the word specified by the CURSOR.

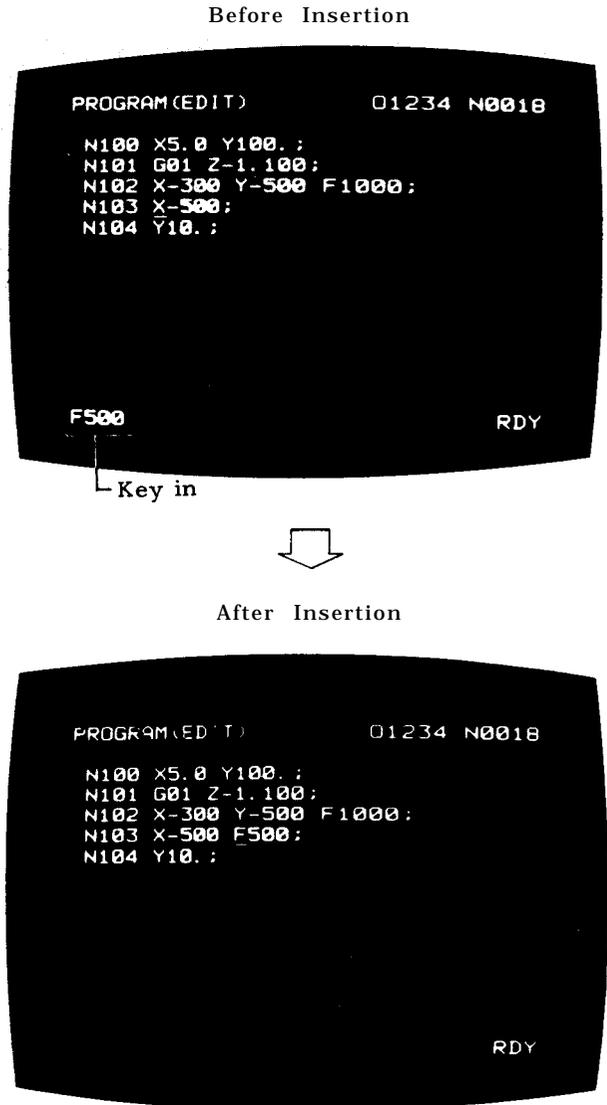


Fig. 4.37

After insertion, CURSOR indicates the last word. The input words less than 32 characters can be inserted as one group.

How to Edit Control-Out and Control-In

The control-in and control-out part may be edited by the usual edit operation.

- a. Depress **C2** key and **SHIFT** key, and character "(" may be entered.
- b. Depress **□** key and **SHIFT** key, and character ")" may be entered.

Notes:

1. The characters which are indicated in the thick-lettered keys shown below may be enclosed in control-out and control-in.
2. The number of characters that may be enclosed in control-out and control-in less than 32.
3. Nesting of the control-out and control-in is not allowed.

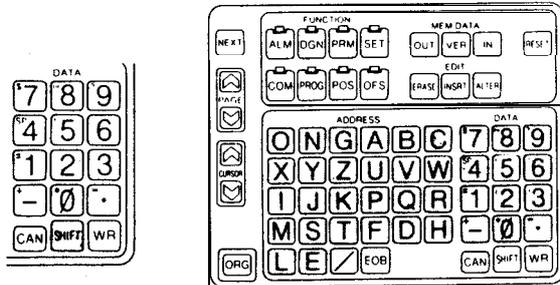


Fig. 4.38

Characters Enclosed in Control-Out and Control-In (Thick-Lettered Keys) for 9" CRT

4.7 SUPPLEMENT TO DATA INPUT/OUTPUT INTERFACE †

The external equipment having the designated input/output interface may be attached to the NC to input/output the following NC information:

- (1) Part Programs
- (2) Tool Offset Amount
- (3) Setting and Parameters

4.7.1 TYPES AND FUNCTIONS OF INTERFACE

Table 4.1

①	
Name of Interface	RS232C Interface No. 1
Type of Interface	Serial voltage interface
Parameter Setting	Parameter setting Input : #6026 Output : #6028
Connector (Note)	DB-25S
Max Cable Length	15 m
External Equipment	Equipment having RS232C interface
Functions	<ul style="list-style-type: none"> · Storing NC data into NC memory or collating them. · Outputting NC data to external equipment (punch out). · Automatic operation in the tape mode (for tape reader).

Note : The types of the connector on the NC side. For the mating connectors to this connector, use the following : MR-20F, DB-25P, DB-37P.

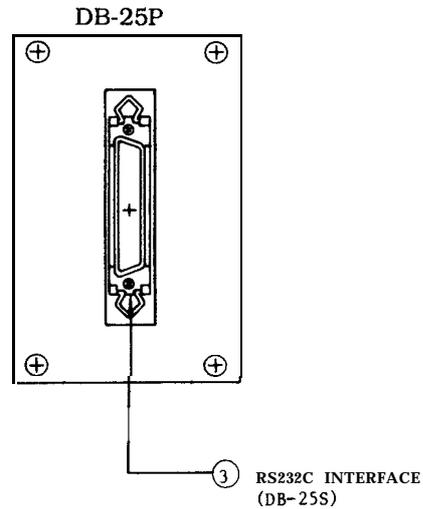


Fig. 4.39 Data I/O Interface Receptacles in Control Cabinet

4.7.2 SETTING OF DATA INPUT/OUTPUT INTERFACE TO BE USED

To use data input/output interface, it is necessary to set which interface is to be used. Make this setting as follows:

- (1) Setting of Data Input Interface to Be Used

IDVCE1 (#6003, D1)	IDVCE0 (#6003, DO)	Data Input Interface to be used
0	1	RS232C Interface

Note: PT R interface is for the standard tape reader. Usually, this interface is set.

4.7.2 SETTING OF DATA INPUT/OUTPUT INTERFACE TO BE USED (Cent'd)

(2) Setting of Data Output Interface to Be Used

ODVCE1 (#6003, D5)	ODVCE0 (#6003, D4)	Name of Interface
0	1	RS232C Interface

4.7.3 SETTING OF BAUD RATE AND OTHERS OF SERIAL INTERFACE

To use serial interface (RS232C), it is necessary to set the baud rate, stop bit length, and control code transmission specification to parameters.

a. Setting of Baud rate

	Input	#6026D3	#6026D2	#6026D1	#6026D0
	output	#6028D3	#6028D2	#6028D1	#6028D0
Baud Rate	50	0	0	0	0
	100	0	0	0	1
	110	0	0	1	0
	150	0	0	1	1
	200	0	1	0	0
	300	0	1	0	1
	600	0	1	1	0
	1200	0	1	1	1
	2400	1	0	0	0
	4800	1	0	0	1

b. Setting of stop bit length

Input	#6026D4	= 1: Stop bit as 2 bits
output	#6028D4	= 0: Stop bit as 1 bit

c. Setting of control code transmission designation

Input	#6026D5	= 1: Does not send out control code
output	#6028D5	= 0: Sends out control code

Notes:

- Set the baud rate and stop bit length according to the specifications of the input/output equipment to be used.
- The start and stop signals to be sent from the NC to the input/output equipment after pressing IN, VER, or OUT key are called "control codes." If the specifications of the input/output equipment do not allow the acceptance of the control codes, set the parameter for control code transmission designation to "1" (not send). In this case, it is necessary to press IN, VER, or OUT key on the NC side then start/stop the input/output equipment manually.

4.7.4 CABLE CONNECTOR SPECIFICATIONS

The specifications of the cable connectors for data input/output interface are as shown in Tables 4.2 and 4.3. These specifications depend on the external equipment to be used and are therefore listed in this publication for reference purpose only. Refer to the manual of the external equipment.

Table 4.2 RS 232C Interface Connecting Cable (A)

NC (DB-25P)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
FG	Frame grounding	1			FG
SD	Sending data	2			'D
RD	Receiving data	3			'D
RS	Request sending	4			RS
CS	Capable of sending	5			CS
	Not used	6			DR
SG	Signal grounding	7			SG
		8			IO BUSY
		25			ER (or IO ALARM)
	Not used				

Note: When the external equipment does not control the CS (Capable of Sending) signal given to NC, short-circuit pins RS and CS on both ends of the cable as shown in Table 4.3.

Table 4.3 RS 232C Interface Connecting Cable (B)

NC (DB-25P)			Connections	External Equipment	
Symbol	Signal Name	Pin No.		Pin No.	Symbol
FG	Frame grounding	1			FG
SD	Sending data	2			SD
RD	Receiving data	3			RD
RS	Request sending	4			RS
CS	Capable of sending	5			CS
	Not used	6			DR
SG	Signal grounding	7			SG
		8			
		25			ER (or IO ALARM)
	Not used				

4.7.5 OPERATIONS USING DATA INPUT/OUTPUT INTERFACE

The use of data input/output interface allows the following operations and runs:

(1) The input /output operations of tool offset amounts, setting data, and parameter data and the output operations (punch out) of part programs. For details, see 4.4, "TAPE INPUT/OUTPUT OPERATIONS OF NC DATA.

(2) The storing of part programs into memory. For details, see 4.5, "LOADING PART PROGRAMS INTO MEMORY'.

(3) Tape-verification of part programs, tool offset amount, setting data, and parameter data, For details, see 4.8, "TAPE VEFIRYING".

4.8 TAPE VERIFYING

The punched tape of the data shown below may be compared to the contents of the NC internal memory to check if they match.

- . Part program
- . Tool offset data
- . Setting data and parameter

The punched tape is entered through the NC tape reader or the data input/output interface (option). For the methods of setting the input/output equipment (setting #6003) and the baud rate (parameters #6026 through #6029), refer to 4.7 "SUPPLEMENT TO DATA INPUT/OUTPUT INTERFACE †". The following description is made assuming that this option is installed.

4.8.1 SETTING AND PARAMETER TAPE VERIFYING

1. Select " 1" of system No. switch.
2. Set the MODE SELECT switch to the EDIT.
3. Depress PRM function key.
4. Depress RESET key.
5. Load the NC tape via tape reader.
6. Depress VER key.
Tape starts, and the contents of tape and the contents of setting /parameter are verified. "VER" blinks on the CRT.
If disagreement with the tape data is detected, " INP UT ERROR" blinks on the CRT.
7. After completion of verifying without disagreement , tape reader stops and "VER" disappears.

4.8.2 TOOL OFFSET VALUE TAPE VERIFYING

1. Set the MODE SELECT switch to the EDIT.
2. Depress OFS function key.
3. Depress RESET key.
4. Load the source tape via tape reader .
5. Depress VER key
Tape starts , and the contents of tape and the contents of tool offset values are verified. If disagreement with the tape data is detected, "INPUT ERROR" blinks on the CRT .
6. After completion of verifying without disagreement, tape reader stops and "VER" disappears.

4.8.3 VERIFYING PART PROGRAM TAPE

- (1) Verifying a part program tape having program number
 - a. Select the EDIT mode.
 - b. Depress the PROG function key .
 - c. Set the part program tape to the tape reader.
 - d. Depress the RESET key.
 - e. Depress the VER key.
The tape is started to compare the contents of the part program memory to the contents of the part program tape. During this operation , "VER" blinks . If a mismatch is found , "INPUT ERROR" blinks on the CRT.
When a match is found and this operation is completed, the tape reader stops , upon which "VER" display is erased.

Note: By the operation of "RESET , VER, " the verifying feature verifies the data from the tape head to % code.

- (2) Verifying a part program tape having no program number
 - a. Select the EDIT mode .
 - b. Depress the PROG function key.
 - c. Set the part program to the tape reader.
 - d. Depress the RESET key .
 - e. Depress the "O"key and key in program number.

4.8.3 VERIFYING PART PROGRAM TAPE (Cont'd)

f. Depress VER key.

The tape starts to compare the contents of the part program memory to the contents of the part program tape. During this operation, "VER" keeps blinking. If a mismatch is found, "INPUT ERROR" blinks on the CRT. When a match is found and this operation is completed, the tape reader stops, upon which "VER" display is erased. If the keyed in program number is not found in the memory, "NOT FOUND! " blinks. In this case, depress the CAN key and start with the operation of d.

Notes :

1. The operations for verification with a program number different from the program number punched on the tape are the same as those of (2) above. The input program number is processed in preference to the punched program number.
2. Verification by the operation of "Oxxxx VER" regards the punched information as the information on the input program number.
When verifying a tape containing program numbers, no program number should be input.

(3) Verifying a tape containing multiple part programs

Multiple part programs punched in a single tape are continuously verified by the following operations:

- a. Select the EDIT mode.
- b. Depress the PROG function key.
- c. Set the part program tape to the tape reader.
- d. Depress the RESET key.
- e. Depress the VER key.

When M02 ;, M30 ; or M99 ; is read, the tape reader does not stop but all the part programs are continuously verified up to % code. When the verification is completed, the tape reader stops at the position of % code.

4.8.4 SUMMARY OF EDITING OPERATION

Operation		Edit Lock	System No. Switch	Mode	Function	Procedure			
Parameter	Storing from NC operator's panel keyboard		1	/		Parameter number → <input type="text" value="CURSOR"/> Data → <input type="text" value="WR"/>			
	Storing from tape (Note 1) (Note 2)		1	EDIT	PRM	<input type="text" value="RESET"/> → <input type="text" value="IN"/>			
	Punch out (Note 3)					<input type="text" value="RESET"/> → <input type="text" value="OUT"/>			
	Matching with tape (Note 1)					<input type="text" value="RESET"/> → <input type="text" value="VER"/>			
Storing from NC operator's panel keyboard			/			SET	Setting number → <input type="text" value="CURSOR"/> Data → <input type="text" value="WR"/>		
Offset	Storing from NC operator's panel keyboard			EDIT	OFS	Offset number → <input type="text" value="CURSOR"/> Address data → <input type="text" value="WR"/>			
	Storing from tape					<input type="text" value="RESET"/> → <input type="text" value="IN"/>			
	Punch out					<input type="text" value="RESET"/> → <input type="text" value="OUT"/>			
	Matching with tape					<input type="text" value="RESET"/> → <input type="text" value="VER"/>			
	Clear of all offsets					/		<input type="text" value="O"/> → -9999 → <input type="text" value="ORG"/>	
Part Program	Storing from tape	One part program Tape with number 0	OFF	EDIT	PROG	<input type="text" value="RESET"/> → <input type="text" value="IN"/>			
		part program Tape without number 0 (Note 4)	OFF			<input type="text" value="RESET"/> → <input type="text" value=" "/> , Program number → <input type="text" value="IN"/>			
		All part programs on tape	OFF			<input type="text" value="RESET"/> → <input type="text" value="O"/> → -9999 → <input type="text" value="IN"/>			
		Addition to registered part program	OFF			<input type="text" value="RESET"/> → <input type="text" value="NEXT"/> → <input type="text" value="IN"/>			
	Punch out	Designated part program					<input type="text" value="RESET"/> → <input type="text" value="O"/> → Program number → <input type="text" value="OUT"/>		
		All part programs					<input type="text" value="RESET"/> → <input type="text" value=" "/> , -9999, <input type="text" value="OUT"/>		
	Matching with tape	One part program Tape with number 0					<input type="text" value="RESET"/> → <input type="text" value="VER"/>		
		part program Tape without number 0 (Note 4)					<input type="text" value="RESET"/> → <input type="text" value="O"/> → Program number - <input type="text" value="VER"/>		
		All part programs on tape					<input type="text" value="RESET"/> → <input type="text" value="VER"/>		
	Edit	Modify of address data (Note 5)	OFF				<input type="text" value="CURSOR"/> (Set to address data to be modified) → Address data → <input type="text" value="ALTER"/>		
		Add of address data (Note 5)	OFF				<input type="text" value="CURSOR"/> (Set to address data just before addition) - <input type="text" value="INSRT"/>		
		Delete of one address data	OFF				<input type="text" value="CURSOR"/> (Set to address data to be deleted) → <input type="text" value="ERASE"/>		
		Delete of one block (Note 6)	OFF				<input type="text" value="CURSOR"/> (Set to address data at head of block to be deleted) → <input type="text" value="EOB"/> → <input type="text" value="ERASE"/>		
	Address search						MEM EDIT		Address data to be searched → <input type="text" value="CURSOR"/>
	Clear	Designated part program	OFF				EDIT	PROG OR ALM	<input type="text" value="O"/> → Program number to be searched → <input type="text" value="ERASE"/>
All part programs on tape		OFF		<input type="text" value=" "/> , -9999, <input type="text" value="ERASE"/>					

Notes :

1. If the tape contains setting information, it is also stored and matched at the same time.
2. When data has been stored from a parameter tape, turn the power on and off.
3. Setting is punched out at the same time.
4. Storing of a part program having a program number different from program number 0 on tape is performed by the same operation as for "tape without program number 0.
5. Within the limit of 32 characters, addition of multiple address data and the change to one address data are permitted.
6. When the cursor to the address data in the middle of a block and EOB and ERASE keys are depressed, the data following the cursor position is deleted.

5. MACHINE CONTROL STATION

5.1 SWITCHING UNITS ON THE CONTROL STATION

Fig. 5.1 shows the layout of switching unit on the control station. For details, refer to the machine tool builder's manual.

5.1.1 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 : Allows the tool to traverse rapidly or return to reference point 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: Automatically controls the system from NC tape.

MDI : Allows the operator to write reference value and execute the contents.

MEM: Automatically controls the system with the stored part program.

EDT : Stores the part program into memory and edit the part program.

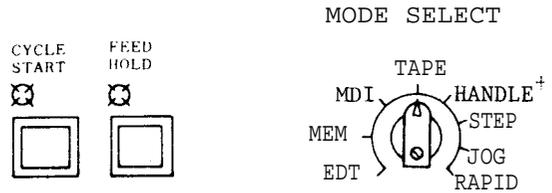


Fig. 5.2

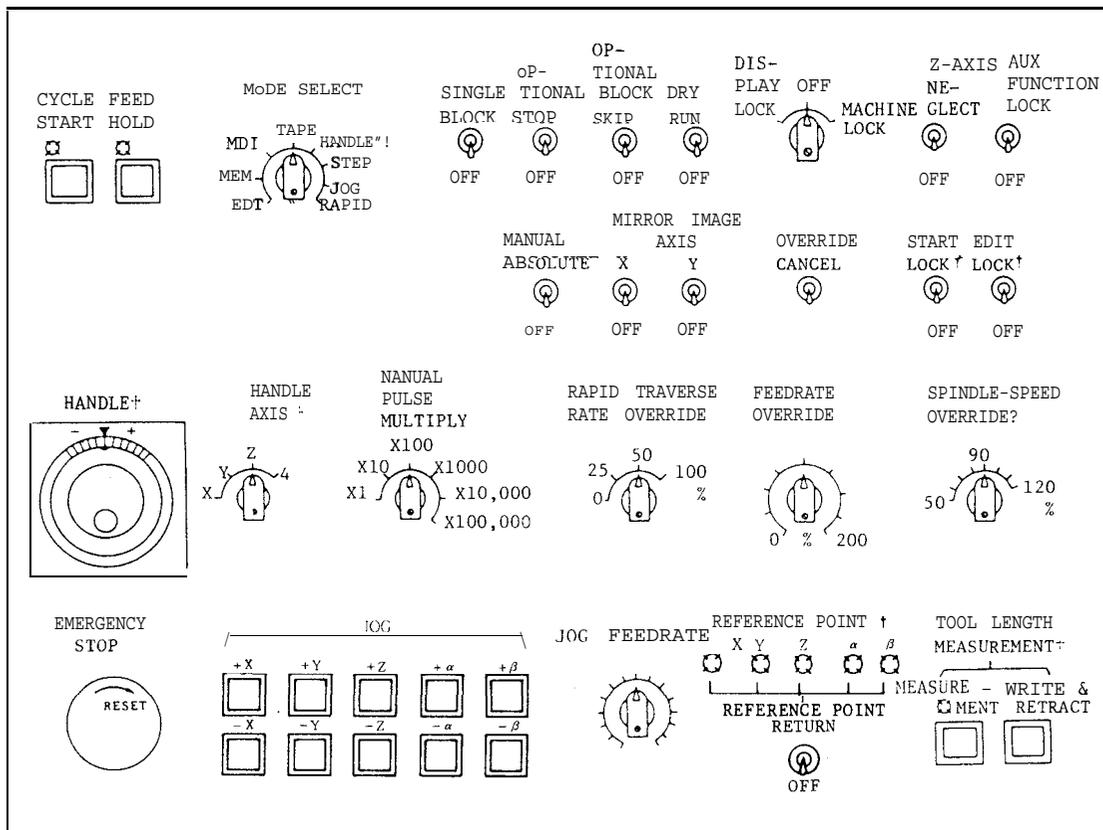


Fig. 5.1 Machine Control Station

5.1.2 CYCLE START PUSHBUTTON AND LAMP

Depress this pushbutton to start the system in the automatic operation mode (TAPE, MD I and MEM) . The CYCLE START indicating lamp lights when automatic operation starts. Depress it again to start the operation after temporary stop by pressing FEED HOLD pushbutton or MODE SELECT switch.

5.1.3 FEED HOLD PUSHBUTTON AND LAMP

When the FEED HOLD pushbutton 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.

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 continued 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, which indicates the operation is during canned cycle.

5.1.4 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 "330" is displayed.

To recover the system 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 "330" is changed to "31 0".
3. Turn on the servo power again by depressing POWER ON pushbutton. "ALM" on the CRT is extinguished and "RDY" lamp lights up.

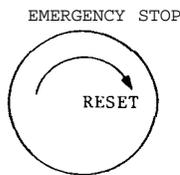


Fig. 5.3

5.1.5 HANDLE DIAL † (SIMULTANEOUS ONE-AXIS CONTROL 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. After selecting the HANDLE mode, the machine moves in the plus or minus direction according to the rotating direction by turning this HANDLE. Use the axis changing switch for simultaneous one-axis control. 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.

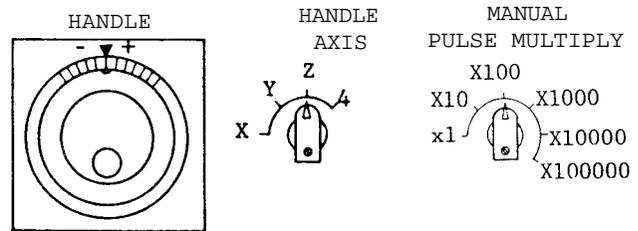


Fig. 5.4

5.1.6 HANDLE AXIS SELECT SWITCH †

This switch is used to select and axis to be operated. For operation of the switch, see Paragraph 5.1.5 HANDLE DIAL †

5.1.7 MANUAL PULSE MULTIPLY SELECT SWITCH †

This switch is used to:

- Select the value from Table 5.1, corresponding to a single graduation of the HANDLE dial in the HANDLE mode.
- Pulses x100 can be multiplied by any multiplication factor using parameter #6383.
- Select the move amount (1 step) from Table 5.2 corresponding to each depression of JOG pushbutton in the STEP mode.

5.1.7 MANUAL PULSE MULTIPLY SELECT SWITCH ' (Cent'd)

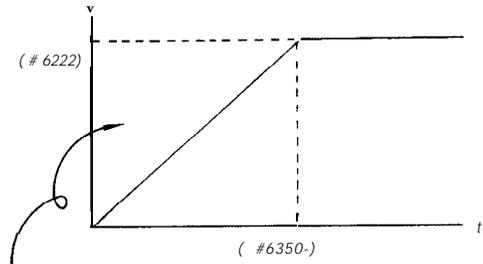
Notes :

1. Handle coasting can be prevented by #6677 (handle lag pulse clamp capacity). With this parameter installed, when handle is turned quickly, there will be a gap between quality of handle turning and that of movement.
2. Handle coasting can be prevented by #6676 (handle input pulse clamp capacity). The following describes an example of this parameter setting.
When #6677 is used as "handle lag pulse clamp capacity", a fraction is obtained in values at the lower 1st/2nd digits even by motion in the handle 10-time/ 100-times mode. To avoid this, set PRM#6677 = 0. Then verify the value of PRM#6222. Assuming that ;

(PRM #6222)

7.5 X (Manual pulse maximum magnification) *a

the value of a obtained by omitting under the decimal point, which is set to #6676. The setting range of #6676 is 0 to 127. When #6676 = 0, lag pulse clamp is not executed. The above input pulses are disregarded. Therefore, the maximum distance of the flowrate is as shown below, affected by the relation to the handle feeding acceleration/deceleration.



Move command value accumulated internally when handle input enters in a step.

This value is discharged as remaining moving amount after the handle input is interrupted.

This is equivalent to the JOG button, corresponding to deceleration to a stop after the command is turned off.

Table 5.1 Selection of Move Amount in the HANDLE Modet

	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 x 100000	0.1 mm/ graduation	0.01 inch/ graduation	0.1 deg/ graduation

Table 5.2 Selection of Move Amount in the STEP Mode

	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
x 100000	100.0 mm/step	10.0 inch/step	100.0 deg/step

5.1.8 JOG PUSHBUTTONS

This pushbutton is used to feed the tool manually.

With any of pushbuttons +X, -X, +Y, -Y, +Z, or -Z (+a, -a) held in the RAPID mode, the axis can be moved rapidly until the button is released.

These pushbuttons move 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 equal to step set by MANUAL PULSE MULTIPLY select switch. Maximum feedrate per step is determined by parameter # "6222."

Note : JOG pushbuttons work on all axes.

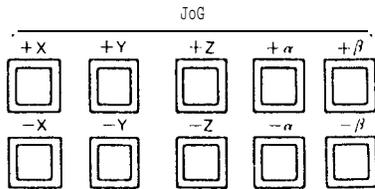


Fig. 5.5

5.1.9 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 5.3. The JOG feedrate can be preset by parameters #6233 to 6264.

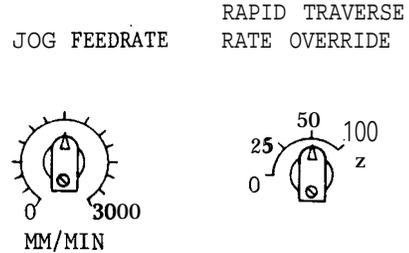


Fig. 5.6

Table 5.3 Jog Feedrate

Step	Parameter No.	mm /min	Step	Parameter No.	mm /min
0	#6233	0	16	#6249	100
1	#6234	1	17	#6250	120
2	#6235	2	18	#6251	150
3	#6236	4	19	#6252	200
4	#6237	6	20	#6253	250
5	#6238	8	21	#6254	300
6	#6239	10	22	#6255	400
7	#6240	12	23	#6256	500
8	#6241	15	24	#6257	600
9	#6242	20	25	#6258	800
10	#6243	25	26	#6259	1000
11	#6244	30	27	#6260	1200
12	#6245	40	28	#6261	1500
13	#6246	50	29	#6262	2000
14	#6247	60	30	#6263	2500
15	#6248	80	31	#6264	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 t, if provided, is shown by deg/min.

5.1.10 RAPID TRAVERSE RATE OVERRIDE SWITCH

This switch is used to adjust the traverse rate by FO, 25, 50 and 100%. 100% rate is the rapid traverse rate set by parameter #6280 to 6284.

The switch is effective both in automatic operation including GOO command and in manual operation (RAPID mode). FO is set by parameter #6231.

5.1.10.1 RAPID FEED OVERRIDE SELECTION SWITCH 2

(1) In the conventional function, override of the four steps of "100%, 50%, 25%, and F0" can be commanded against the rapid feedrate. This function is now expanded to allow override of the six steps of "100%, 50%, 25%, FO, F1, and F2."

(2) The F1 and F2 feedrates are the feedrates given by the parameter (#6466, #6467).

5.1.11 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. Feed during tapping by G74 follows F command. Where OVERRIDE CANCEL switch is set on, the tool will be moved at the programmed feedrate by F code regardless of switch setting.

Table 5.4 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		

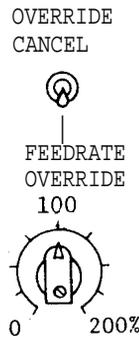


Fig. 5.7

5.1.12 JOG FEEDRATE OVERRIDE SWITCH †

1. This option permits overrides in 21 steps of 10% per step within a range of 0 to 200% in relation to the jog feedrate.

2. This switch also serves as the "FEEDRATE OVERRIDE" switch. When the JOG mode is selected, the Feedrate Override switch becomes the Jog Feedrate Override switch.



Fig. 5.8

3. Use this function as follows.

Although the Jog Feedrate switch can be switched over in a maximum of 32 steps, this is limited to the three steps of LOW - MEDIUM - HIGH and the intermediate speeds are compensated for with the "JOG FEEDRATE OVERRIDE" function.

5.1.13 FEEDRATE OVERRIDE CANCEL SWITCH

Turning on the OVERRIDE CANCEL switch prevents the function of FEEDRATE OVERRIDE switch.

5.1.14 SPINDLE SPEED OVERRIDE SWITCH †

1. With this switch, the current spindle speed can be changed to an override speed which is set at 10% intervals between 50 and 120% of the current spindle speed.

2. During the tapping cycle by G84 and G74, the spindle speed may selectively be made independent of this switch, remaining at the speed set by an S code. This selection is made with the parameter #6007D2.

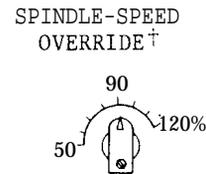


Fig. 5.9

5.1.14.1 SPINDLE SPEED OVERRIDE SWITCH 2

(1) Overview

The conventional function that allows 50% to 120% override against the S command is expanded to allow override of 10% to 200%.

5.1.15 MANUAL REFERENCE POINT RETURN SWITCH'

This switch is for bringing the tool back to the reference point manually.

For its operation method, refer to 5.2.1 "MANUAL RETURN TO REFERENCE POINT'.

5.1.16 REFERENCE POINT LAMPS '

These lamps indicate that the tool is positioned on the reference point. They light when the tool is brought to the reference point through the manual or automatic return to reference point (G28), or by the reference point return check (G27), and goes out as the tool moves away from the reference point by a subsequent operation.

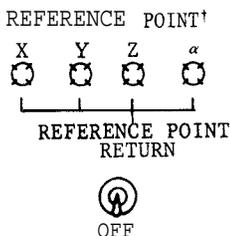


Fig. 5.10

5.1.17 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 push-button is activated.



Fig. 5.11

5.1.18 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.

5.1.19 OPTIONAL BLOCK SKIP SWITCH

This switch selects whether the data in blocks including a " /" is disregarded or not.

1. While the switch is on, all the commands in a block programmed after a "/" are neglected. However, block data appearing before the " /" remains effective.
2. While this switch is off, blocks including a "/" are executed along with other blocks. This switch is ineffective on the block under execution and blocks stored in the advance-reading buffer . When this switch is turned on during an automatic operation cycle , it works on the block read after the switching on has occurred.

Notes :

- a. The two commands " /" and " /1" are equivalent.
- b. With the control provided with the optional block skip B function "+, 8 independent blocks can be skipped with the switching of the switches corresponding to " /2" through " /9. "

5.1.20 DRY RUN SWITCH

Turning on the DRY RUN switch in the TAPE, MD I 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 # "6006D2. "

Table 5.5

Parameter # "6006D2"	Rapid Traverse at Dry Run Operation
"0"	Rapid traverse rate*
"1"	Jog feedrate

* 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+ or during tapping becomes effective on the next block.

During tapping , the set speed when tapping starts will be kept. It cannot be changed by JOG operation during dry run operation .



Fig. 5.12

5.1.21 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 FE ED HOLD pushbutton.

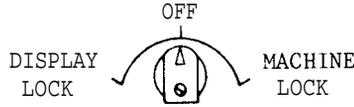


Fig. 5.13

"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. 3rd/4th axis external current position display is not updated, though the machine moves.

"MACHINE LOCK"

Setting the switch at MACHINE LOCK inhibits axis movement including Reference Point 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 by normal operation.

If MACHINE LOCK is on, reference point return is not executed. Do not operate this switch unless the machine is in the stopped state. That is, it is changed only during block stop or feed-hold.

Note :

When operation is restarted with MACHINE LOCK off after operation with MACHINE LOCK on, since the NC operates with the coordinate value obtained immediately before MACHINE LOCK off as reference, motion is made in which the coordinate system is shifted by MACHINE LOCK moving amount when an absolute command (G90) is provided. Therefore, set again to the proper coordinate value suitable for the machine position at MACHINE LOCK off.

5.1.22A Z-AXIS COMMAND 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.

Note :

When operation is restarted by CANCEL off after operation by Z-axis CANCEL on, since the NC operates with the coordinate value obtained immediately before CANCEL off as reference, motion is made in which the coordinate system is shifted by Z-axis CANCEL moving amount when an absolute command is provided.

Therefore, set Z-axis again to the proper coordinate value suitable for the machine position at CANCEL off.

5.1.22B 4TH AXIS NEGLECT INPUT

(1) When the POWER ON switch is pushed with a 4th axis neglect input on (contact is closed), the system is equivalent to one without the 4th axis control. If motion along the 4th axis is commanded in this state, the machine remains motionless along the 4th axis, but the position display indicates the suppressed motion. (Machine lock state).

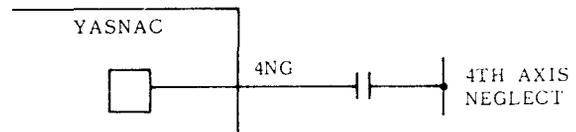


Fig. 5.14

(2) The ON /OFF switching of parameter #6023 D6 is equivalent to the ON /OFF switching of the 4th axis neglect input. The result of these two is determined by their logical OR.

Table 5.6

4NG Input	#6023 D 6	Logical OR (4GNC for 4th Axis Neglect)
ON	ON (1)	ON (Closed)
ON	OFF (0)	ON (Closed)
OFF	ON (1)	ON (Closed)
OFF	OFF (0)	OFF (Open)

Note : When 4NG input or #6023 D6 is switched over, be sure to switch on the power supply again. Otherwise, the switching is ineffective.

5.1.23 M-FUNCTION LOCK SWITCH (AUXILIARY 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.

AUX
FUNCTION
LOCK



OFF

Fig. 5.15

- The following M codes are executed even if the switch is set on.
 - MOO, MO1, 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.

5.1.24 MANUAL ABSOLUTE SWITCH

This switch specifies how manual moving amount is processed at restart of operation when manual operation is interrupted during automatic operation to move the tools.

- 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.

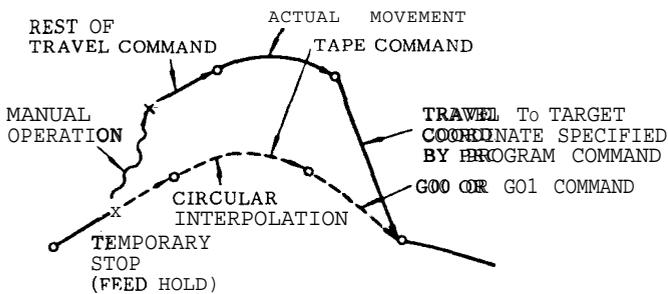


Fig. 5.16

Note: #6008 D3 can determine whether manual absolute is ON or not in G91 mode; "0" -- effective, "1" -- ineffective. If #6008 is set to "1" and G91 is ineffective, next G90 makes manual absolute effective.

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.

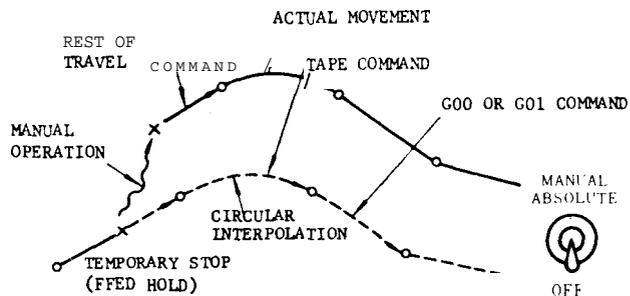


Fig. 5.17

The parallel shift is reset by executing Reference Point Return manually, automatically by G28, or operating the RESET key. The command value is forced to change to the current position.

- (a) Manual or automatic reference point return operation
- (b) RESET operation

5.1.25 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 as well as setting function, set the data of setting # "600 D0-D3" 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 2.8.5.

Note: During the M 95 (Mirror Image ON) mode, never operate the MIRROR IMAGE AXIS switch,

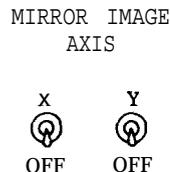


Fig. 5.18

5.1.26 TOOL LENGTH MEASUREMENT PUSHBUTTON AND LAMP†

Use the WRITE button, to automatically store the amount of Z-axis move manually made between "home-position" and "t base-position" directly in the tool offset memory. For operating procedure, refer to 5. 2.3 Automatic Tool Length Measurement† .

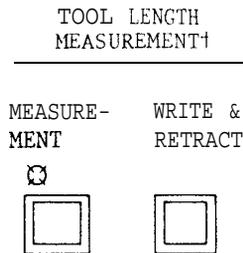


Fig. 5.19

5.1.27 START LOCK 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.

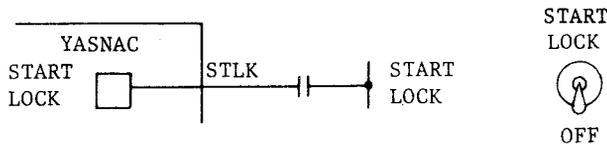


Fig. 5.20

Note : START LOCK switch is not effective during feed hold.

5.1.28 EDIT LOCK SWITCH †

Turning on the EDIT LOCK switch prevents the function of ERS, INS and ALT keys, and storing from NC tape. When editing is made with EDIT LOCK switch turned on, "EDIT LOCK" blinks on the CRT display.

5.1.29 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 slow 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

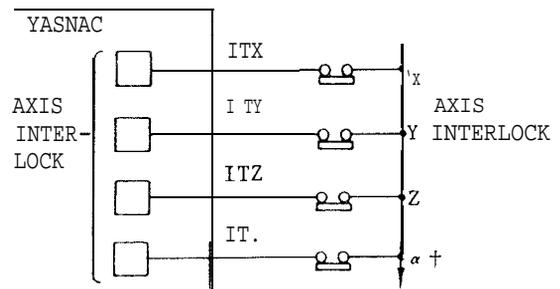


Fig. 5.21

5.1.30 EXTERNAL DECELERATION INPUT SIGNALS †

In order to eliminate the danger of high speed contact at speed end caused by erroneous motion commands , limit switches for originating external deceleration input signals may be installed at selected points.

1. During rapid traverse (G00) and manual operation

When the limit switch is tripped by the tool movement, the traverse speed is decelerated in the tripping direction to a level set by the parameter #6340. In the direction opposite to the tripping direction, the original speed remains unchanged.

2. During motion at feedrate (G 94)

While the limit switch is being tripped, the tool moves at a speed set by the parameter #6341. If the feedrate set by the F command is lower than the rate set by the parameter, the original feedrate remains unchanged.

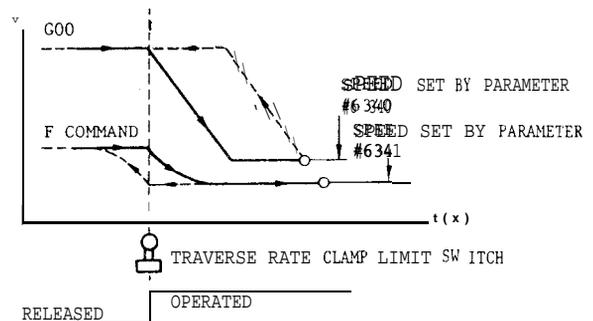


Fig. 5.22

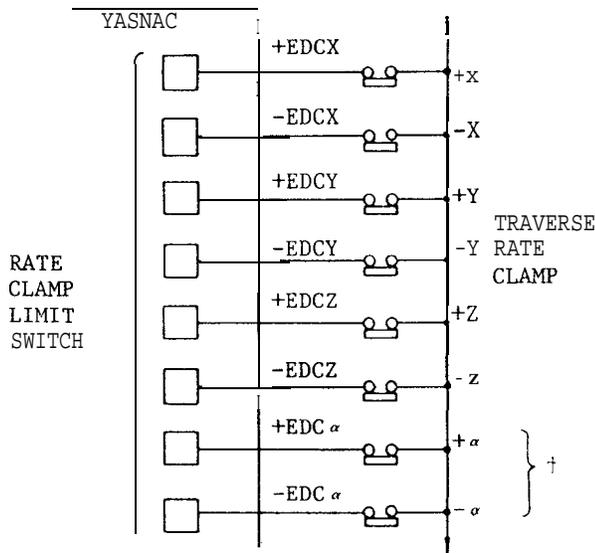


Fig. 5.23

Notes:

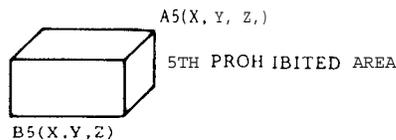
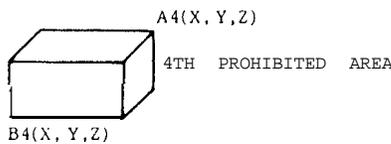
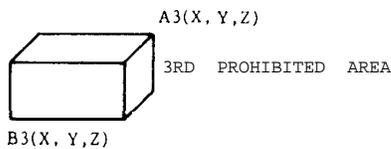
- i. The external deceleration function is ineffective on feedrate specified in mm per revolution of the spindle (mm/rev).
- ii. It is also ineffective on the HANDLE feed.

5.1.31 3RD TO 5TH STORED STROKE

By adding the stored stroke limit function (G22, G23), it is possible to set the 3rd, 4th and 5th prohibited areas simultaneously. However, selecting the ON/OFF state of this function must be by means on an external signal.

(1) 3rd, 4th and 5th prohibited areas

The 3rd, 4th, and 5th prohibited areas can be set by means of the parameters "Address Stroke Limit -3. -4. -5"



(2) The parameters for setting prohibited areas are as shown in the table below.

Table 5.7

		x	Y	z	Parameter
3rd	Point A 3	#6650	#6651	#6652	
	Point B 3	#6653	#6654	#6655	
4th	Point A 4	#6656	#6657	#6658	
	Point B 4	#6659	#6660	#6661	
5th	Point A 5	#6662	#6663	#6664	
	Point B 5	#6665	#6666	#6667	

Point A sets the boundary value on the positive (+) side of the machine coordinate system and point B sets the boundary value on the negative (-) side.

(3) Setting of the inside or outside of the 3rd, 4th and 5th prohibited areas as the prohibited area is performed simultaneously by the following parameters.

#6007 D1	Meaning
"0"	Inside Prohibited
"1"	Outside prohibited

(4) The following external signals are used to turn on area check of either the 3rd or 4th prohibited area.

Table 5.8

#1312		Meaning
D1	DO	
0	0	3rd to 5th prohibited area check OFF
0	1	3rd prohibited area check ON
1	0	4th prohibited area check ON
1	1	5th prohibited area check ON

0: Contact open 1: Contact closed

When this function is not used, set input signal #1312 D0 and D1 to "0."

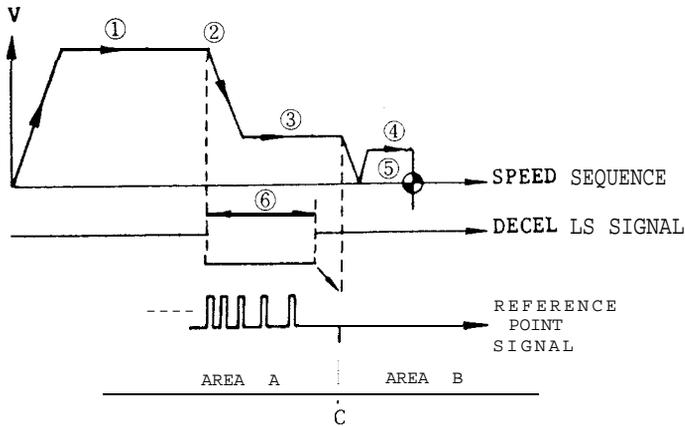
Note: Same as the notes for section 2.9.12 "STORED STROKE LIMIT".

5.2 OPERATION PROCEDURE

5.2.1.1 MANUAL RETURN TO REFERENCE POINT

With this function, the tool is returned to the reference point manually. The procedure is as follows .

1. Set the mode select switch to RAPID or JOG.
2. Manually move the tool to a position some distance away from the reference point. When the tool is within the range A shown below, it can be brought back to the reference point in the normal way, as described below.
3. Turn on the REFERENCE POINT RETURN switch.
4. Keep the JOG button for the return direction depressed. The tool starts to move as in the normal manual control, but the speed is decelerated at the deceleration point, and the motion stops automatically at the reference point.
5. Then, the reference point lamp for the relevant axis lights at reference point return completion.



- | | |
|--|--|
| ① RAPID TRAVERSE RATE
(#6280 - #6283) | ④ APPROACH SPEED 2
(#6316 - #6319) |
| ② DECELERATION POINT | ⑤ TRAVERSE DISTANCE
(#6304 - #6307) |
| ③ APPROACH SPEED 1
(#6310 - #6313) | ⑥ DOG WIDTH |

Fig. 5.24

Notes :

- a. As long as the power supply is turned on , either the manual or the automatic return to reference point can be initiated, regardless of the tool position , but the tool will not return to the reference point accurately if the tool is started from a point in the area B . Be sure to bring the tool into the area A before initiating a manual or automatic return motion.

- b. Once the tool is returned to the reference point, the point C is stored, and if the reference return motion is initiated from a point in the area B , this is regarded as an error. Start the reference return motion from a position in the area A.
- c. Once the tool is returned to the reference point, it can not be further moved in the same direction unless the REFERENCE POINT RETURN switch is turned off.
- d. While the MACHINE LOCK switch is on , the reference point return function is ineffective.
- e. Do not return the tool to the reference point by the manual reference point return function, while the buffer is loaded with blocks read in advance of execution, because the stored motion data will be erased by the reference point return motion.

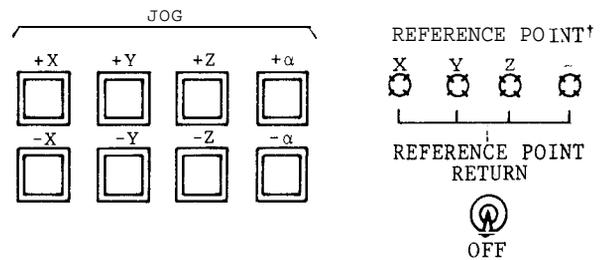


Fig. 5.25

5.2.1.2 2ND MANUAL REFERENCE POINT RETURN

This function is used to automatically position the machine at the 2nd reference point under manual mode. Positioning can be made without observing upon which side of the 2nd zero point the current position is located.

(1) Function

(a) Command ZRN2 (2nd reference point return request) and +X (or +Y, +Z) under JOG or RAPID mode, to position the X-axis (or Y-axis, Z-axis) at the 2nd reference point. The move speed for positioning is the JOG or RAPID feedrate.

(b) If ZRN2 turns off (= 0) while moving to the 2nd reference point, the move will stop before completion. If the ZRN2 is turned on (= 1) again, the move will restart.

(c) If +X (or +Y, +Z) turns off (=0) while moving to the 2nd reference point, the move will also stop. Turn on (= 1) +X (or +Y, +Z) again, to restart the move.

(2) Notes

(a) Inputting the 2nd reference point return mode when the 1st reference point is still not completed is not permitted.

(b) The -X (or -Y, -Z) inputs are also valid in the 2nd reference point return mode. Therefore, the 2nd reference point return will not stop before completion by turning on the -X (or -y, -Z) input.

(c) If the mode is switched to modes other than the JOG or RAPID modes during the 2nd reference point return, the ZRN2 input is regarded to be turned off.

(d) The 1st reference point return mode and the 2nd reference point return mode cannot be used at the same time. If the two inputs are turned on at the same time, both modes will become invalid.

(e) The -X (or -Y, -Z) input as well as the +X (or +Y, +Z) input is effective as the 2nd reference point return input.

5.2.1.3 EASY REFERENCE POINT RETURN t

When setting the machine reference point the position where the C-phase pulse is actually output to the dog position cannot be measured. Thus the dog position must be adjusted and tried several times to find the optimum position. This function eliminates the need of the uncertain dog adjustment; the optimum reference point position can be set rapidly and without failure, just by rewriting the internal data.

The new concept of the theoretical C-phase pulse is employed to embody this function. The C-phase pulse position output for a single pulse per motor rotation is mechanically fixed, and cannot be adjusted. Therefore, another theoretical C-phase pulse that generates at an optional position of a motor rotation is assumed. This theoretical C-phase pulse is represented by the shift distance from the C-phase pulse.

(1) Function

The reference point return operation in this function is completely the same as the conventional reference point return. The concept of a theoretical C-phase is used in this function to allow the positional relation between the dog and the C-phase pulse as well as the relation between the dog and the reference point position to be changed, can be accomplished just by rewriting the parameter, instead of adjusting the dog position. Therefore, latch mistakes can be avoided, even when the C-phase pulse is near the dog switching point, by increasing or decreasing the theoretical C-phase shift distance. The distance from the dog to the reference point can be kept constant, by adjusting the final run distance. In other words, the reference point position can be set at an optional distance from the dog, regardless of the C-phase pulse position.

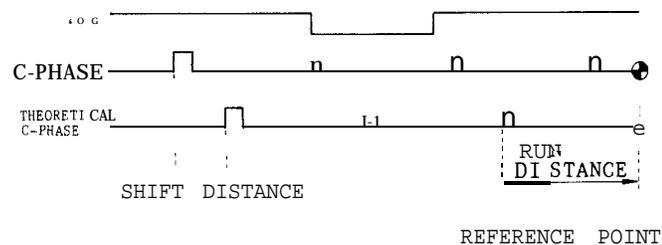


Fig. 5.26

(2) Related parameters

- #6010 DO to D3 : Reference point return direction of each axis
1: -/0 : +
- #6016 DO to D3 : Reference point return of each axis
1: Enabled/O : Disabled
- #6304 to #6307 : Run distance from the theoretical C-phase
Setting range : -32768 to 32767
- #6310 to #6313 : Reference point return approach speed 1
Setting range : 0 to 24000
- #6316 to #6319 : Reference point return approach speed 2
Setting range : 0 to 24000
- #6444 to #6447 : Move distance per motor rotation
Setting range : 1 to 65535
- #6480 to #6483 : Theoretical C-phase shift distance
Setting range : -32768 to 32767

Note :

Always turn off the power one time when change is made to the above parameters.

(3) Adjustment

(a) When the deceleration limit switch of the machine reference point is at an optional position

(i) Set the parameter. The temporary setting is run distance (#6304 to #6307) 0, theoretical C-phase shift distance (#6480 to #6483) 1000.

(ii) Perform reference point return.

(iii) If it stopped before the machine reference point, add the distance from that position to the machine reference point to the theoretical C-phase shift distance.

If it stopped beyond the machine reference point, subtract the distance from that position to the machine reference point from the theoretical C-phase shift distance.

(iv) Fine adjustment for the true machine reference point can be made by the theoretical C-phase shift distance or by the run distance.

(b) When the machine reference point and the deceleration limit switch are at 1/2 the distance of the ball screw pitch.

(i) Adjust the machine reference point assuming that the above machine reference point (iii) is at 1/2 the ball screw pitch. Fine adjustment for the true machine point can be made by the run distance.

(4) Notes

(a) Perform reference point return after returning from the deceleration limit switch position. If the distance to go beyond the deceleration limit switch is too short, alarms 231 to 234 occur.

(b) If the setting of the theoretical C-phase shift distance (#6480 to #6483) and the run distance (#6304 to #6307) are not appropriate, pull-back occurs, to cause alarms 261 to 264. If this occurs, set a larger value for the run distance, or make any necessary parameter adjustments.

(EXAMPLE) When the theoretical C-phase shift distance is set too large that the distance from the deceleration limit switch to the theoretical C-phase pulse is larger than the motor pitch.

5.2.1.3 EASY REFERENCE POINT RETURN t (Cent'd)

(EXAMPLE) When the setting value of the final run distance is negative and the distance exceeded the distance from the deceleration limit switch to the theoretical C-phase pulse.

5.2.2 AUTOMATIC COORDINATE SYSTEM SETTING†

With this function, a new coordinate system is set up automatically upon the return of the tool to the reference point by the manual reference point return function. The coordinates of the new origin are preset with the following parameters. The coordinate system set up by this function is equivalent to the ones set up by G92.

1. Parameters for metric input system

Parameter	Meaning
#6636	X coordinate
#6637	Y coordinate
#6638	Z coordinate
#6639	4th coordinate

2. Parameters for inch input system

Parameter	Meaning
#6630	X coordinate
#6631	Y coordinate
#6632	Z coordinate
#6633	4th coordinate

3. Axis can be selected by parameter #6015 for both metric and inch systems.

5.2.3 TOOL LENGTH Measurement

When a tool mounted on the spindle is manually brought to a position where the tool tip makes contact with the reference surface for Z-axis, and the WRITE & RETRACT button is pushed, the following operations are performed by the control.

- The distance between the set Z-axis home position and the reference surface is stored automatically in the memory having the currently specified correction number. The difference between tool touch position and base position can be set by parameter.
- Increase the correction number by 1, in preparation for the next writing.
- Return the tool to the Z-axis home position.

1. Measuring method (parameter #6039D4 = "0")

- Mount a tool on the spindle, and move it to a Z-axis position which is to be set as the reference point. Any position may be set as the reference point, but for facilitating tool changing process, the tool changing position may be set up as the reference point.
- Select the manual operation mode (RAPID, JOG, HANDLE or STEP) using MODE SELECT switch.



- Depress the function key

The offset number specified previously and related data are displayed.

- The page covering tool offset values specified the tool offset number keyed in will be shown. The specified number is shown by



- Depress the MEASUREMENT button when the motion stops. MEASUREMENT lamp lights and the current position of Z-axis will be set as home position.

(The button is effective only in the manual operation mode and while the



key is selected.)

TOOL LENGTH MEASUREMENT †

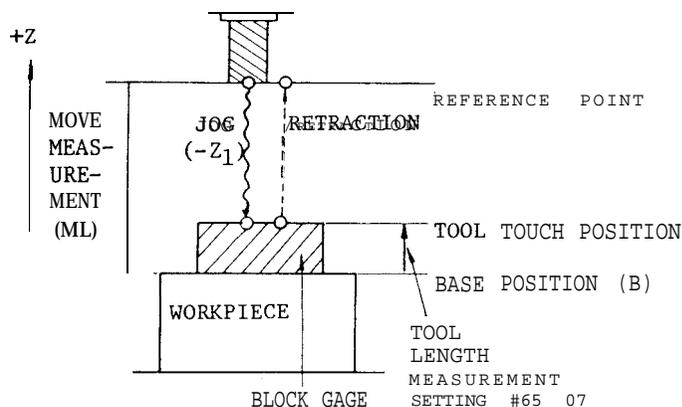
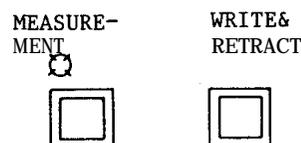


Fig. 5.27

Reference point is the point where measurement starts. Measurement is made with the point temporarily determined as coordinate O.

Note: To display this position as coordinate O, reset the POSITION [UNIVERSAL] referring to par. 4.3,4.1. However the proceeding can be made normally, regardless of this reset.

Return the function key .

f. Set the tool to the tool touch position by

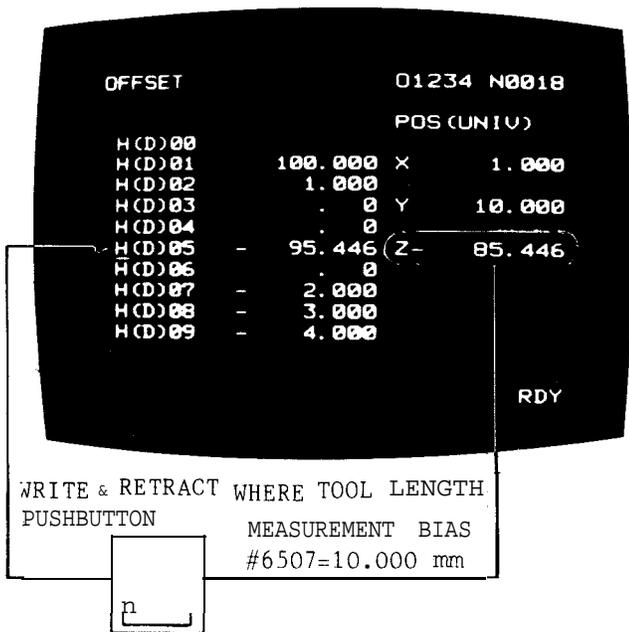


Fig. 5.28

g. Depress the WRITE and RETRACT buttons. The NC equipment executes the next operation

i. The distance from reference point to base point, that is, move measurement (M_L) is stored as the contents of offset no. specified.

$$M_L = (-Z_1) - (B)$$

ii The tool automatically returns in the Z-direction to the reference point in rapid traverse.

iii, The tool offset number is increased by one in preparation for next writing of offset value. (When it is H99, H01 is designated.)

- h Exchange the tool with a next tool by manual operation or by MD I operation. Even when the MDI mode is switched on, the MEASUREMENT lamp remains lighted. Return to the MANUAL mode afterwards.
- i. Repeat the processes f. through h. to store all the required offset values.
- j. Depress the MEASUREMENT key.

The MEASUREMENT lamp goes out, and the automatic writing function is turned off. The length measuring data also disappears from the CRT.

Notes :

- . In this automatic writing mode , the measured values are stored in the absolute values .
- when the reference point is different from the tool change position, the new tool may be brought to the reference surface directly without first returning to the reference point. Once the home position has been set up by the use of the MEASUREMENT key, repositioning to the reference point is not necessary.

When the WRITE & RETRACT key is depressed with H00 designated, offset amount will not be changed. Execute TLM operation by moving the CURSOR AFTER HO1 .

2. Remaining distance measurement (Pm. #6039D4= "1")

a. For the set of Pm. #6039D4 = "1", store the remaining distance R_L instead of move measurement M_L to tool offset memory.

The bottom level which is used as a base of calculating remaining distance can be set by setting #6508 (bottom level setting, 1 = input unit) . Writing operation is the same as described in step 1.

$$R_L = -(A - M_L)$$

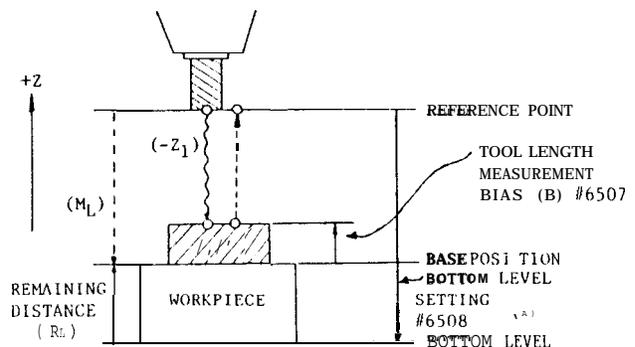


Fig. 5.29

5.2.3 TOOL LENGTH Measurement (Cont'd)

3. Measurement by NC control station

- a. Without using MEASUREMENT button , the same measuring method as referring to step 1 on page 188 can be performed with NC control station.
- b. Instead of the MEASUREMENT button, execute as follows.

- i. Depress the  and  keys to select the page of POSITION [U NIVERSALI].

- ii. Reset the display by depressing  and  keys. This means that home position is set to "0".

- iii. Return the function mode to offset by depressing  key.

- c. Instead of the WRITE & RETRACT key, execute as follows.

- i. Depress the  and  keys,

This results in the automatic writing of the same tool offset measured as written in step 1. The Z-axis, however, will not return to home position.

- d. In this measurement operation, parameter #6039D4 (move measurement value/remaining distance switching) is effective. Storing either one of the values is determined by parameter.

Program restart is of either type P or type Q, depending on whether the change of the coordinate system before or after the restart is permitted.

5.2.4.1 PROGRAM RESTART OF TYPE P

In type P, the program is restarted assuming that the change of coordinate system before and after the restart will not occur. Therefore, this type is used after the replacement of the tool broken during operation, for example.

Operation Procedure

When the tool is broken during machining operation, the automatic operation may be restarted in the following procedure:

- (1) Press FEED HOLD button to stop the operation and make the tool escape.

- (2) Replace the broken tool and perform the associated chores. Change the tool offset, if required.

- (3) Turn on PROGRAM RESTART switch on the machine control station.

- (4) Press PROG function key to display the part program being executed. Take note of the sequence number of the block immediately before the block on which the feed-hold operation was performed.

- (5) Set the head of the part program as follows:

Set memory mode, key-in the program number (Oxxxx) to be restarted, and press

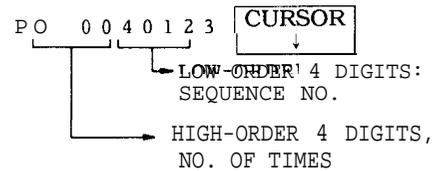
 key.

- (6) Key-in p, sequence number, and

m

in this order. The sequence number is the one that was taken note of in step (4).

While making the setups for program restart, the control searches for the block of the specified sequence number. If the same sequence number appears repeatedly, the nth sequence number may be specified.



For example, the nth sequence number may be called in a subprogram for which L times of execution is specified. For the sequence number that appears first, the high-order 4 digits may be omitted. In this case, the leading zero of the sequence number may also be omitted.

5.2.4 PROGRAM RESTART†

Machining may be restarted from the block that follows the one for which the sequence number was specified. This restart is useful when replacing the broken tool or taking over the machining operation from the last work shift.

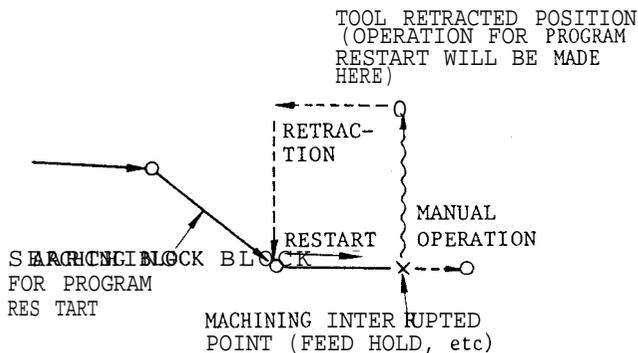


Fig. 5.30

(7) When the search is completed, the CRT screen automatically displays the program restart information,

```

PROGRAM RETURN      00001 N0125
(UNIVERSAL) M 03 33 10 06 45
X  100.000          63 ** ** ** **
Y- 200.000
Z  300.000
A   45.000

( INCREMENT ) M COUNT ( 0006)
X- 248.774
Y  483.123          T 0015 0025
Z- 182.347          S 0500
A   73.442          B 0000

PRN      RDY

```

Fig. 5.31 Display of Program Restart Information

- a. Position (UNIVERSAL) indicates the position of machining restart. Normally, this position is the start point of the block on which the feed-hold operation was performed.
- b. Position (INCREMENT) indicates the distance from the current tool position to the machining restart position.
- c. The M codes and the number of M codes designated (M COUNT) from the head of the program to the block of the specified sequence number are displayed. However, if the number of designated M codes exceeds 35, the 35 M codes as counted from the specified block are displayed.
- d. The two last T codes specified up to the designated block are displayed.
- e. The last S code specified up to the designated block is displayed.
- f. The last B code specified up to the designated block is displayed.

Note: The M codes and T codes are displayed in the order in which they were specified. The code displayed last is nearest the designated block .

(8) Turn off PROGRAM RESTART switch on the machine control station.

(9) Look at the displayed program restart information and specify the M, T, S, or B code necessary for the restart as shown below :

- a. Set MD I mode.
- b. Press PROG function key.
- c. Key-in the necessary M, T, S, or B code and depress WR key.

d. Press CYCLE START button.

e. Press POS function key and check the display of program restart information.

(10) Set the memory mode.

(11) Depress CYCLE START button.

The tool moves to the machining restart position, axis by axis ; that is, the 4th axis, X-axis, Y-axis and Z-axis, in this order. Then, the automatic operation restarts from the head of the block (on which the feed-hold operation was performed) that follows the block of the designated sequence number.

(12) The macro program by the T or M code upon program restart can be selected to be executed or prohibited, by the parameter. Among the macro programs by the T or M code when restarting the program, execution /prohibition of #6130 (M macro) and #6134 (T macro) can be selected by parameter setting (2 bits).

Setting parameters

- #6073 Do
= 1 : Prohibit execution of the M code macro
= 0 : Allow execution of the M code macro
#6073 D₁
. 1 : Prohibit execution of the T code macro
= 0 : Allow execution of the T code macro

5.2.4.2 PROGRAM RESTART OF TYPE Q

If the coordinate system is changed by any of the following operations performed after the interruption automatic operation, use the program restart of type Q :

- (1) The machine power was turned off.
- (2) G92 is specified by MDI operation.
- (3) The setting of work coordinate system is specified.
- (4) The automatic setting of work coordinate system was specified by reference point return.
- (5) ORG key was depressed.

Operation Procedure

The automatic operation interrupted by any of the above operations may be resumed using the following procedure:

- (1) When the machine power is turned on after the interruption of machining , perform the necessary operations such as reference-point return.
- (2) Manually move the tool to the start point (of machining) of the part program . Change the tool offset amount if necessary.
- (3) Turn on PROGRAM RESTART switch on the machine control station.

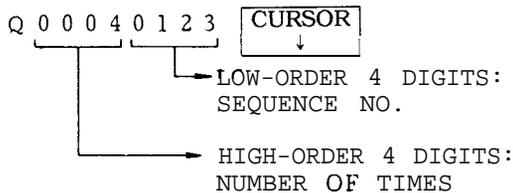
5.2.4.2 PROGRAM RESTART OF TYPE Q (Cont'd)

(4) Press PROG function key to display the part program (being executed) . Record the sequence number of the block immediately before the block to be restarted.

(5) Set the head of the part program as follows:

Set memory mode, key-in the program number (Oxxxx) to be restarted, and depress  key.

(6) Key-in Q, sequence number, and  in this order. The sequence number is the one that was taken note of in step (4). While making the setups for program restart, the machine searches the block of the specified sequence number. If the same sequence number appears repeatedly, the nth sequence number may be specified.



For example, the nth sequence number may be called in a subprogram for which L times of execution was specified. For the sequence number that appears first, the high-order 4 digits may be omitted. In this case, the leading zero of the sequence number may also be omitted.

(7) When the search is completed, the CRT screen automatically displays the program restart information.

(8) Turn off PROGRAM RESTART switch.

(9) Look at the displayed program restart information and specify the M, T, S, or B code required for the restart in MD I mode.

(10) Set the memory mode .

(11) Depress CYCLE START button.

The tool moves to the machining restart position, axis by axis ; that is, the 4th-axis, X-axis, Y-axis, and Z-axis, in this order. Then, the automatic operation restarts from the head of the block (on which the feed-hold operation was performed) that follows the block of the designated sequence number.

(12) When the canned cycle count is specified (L counts), it moves to the final machining cycle position.

(13) The macro program by the T or M code upon program restart can be selected to be executed or prohibited, by the parameter. Among the macro

programs by the T or M code when restarting the program, execution/prohibition of #6130 (M macro) and #6134 (T macro) can be selected by parameter setting (2 bits).

Setting parameters

#6073 D0
= 1: Prohibit execution of the M code macro
. 0: Allow execution of the M code macro
#6073 D1
= 1: Prohibit execution of the T code macro
= 0: Allow execution of the T code macro

CONSIDERATIONS AND REMARKS

(1) Before depressing CYCLE START button to restart the program execution, check to see if the axis-by-axis tool movement (the 4th-axis, X-axis, Y-axis, and Z-axis, in this order) interferes with the work or any part of the machine. If the tool is found interfering, correct the tool position manually.

(2) In both types P and Q, the tool which is moving to the machining restart position axis-by-axis may be single-block-stopped after the completion of the movement for each axis. Even if the incremental shift amount is "0," the single-block stop is performed when SINGLE BLOCK switch is on. When the single-block stop is performed, however, the intervention of MD I operation is not allowed. Manual intervention is possible. But, if a manual intervention is performed on the axis which has already returned, it will not return to the machining restart position again.

(3) During the search operation for program restart, set the switches on the machine control station to the state before commanding program restart. Otherwise, the former position cannot be reached.

(4) When a feed-hold operation was performed during the search operation for program restart, or a reset operation was performed during or after the search operation, perform the operations all over again.

(5) When PROGRAM RESTART switch is on, the operation of CYCLE START button is ignored.

(6) Whether it is before or after machining, each manual operation should be performed with MANUAL ABSOLUTE switch on and MACHINE LOCK switch off.

(7) In any of the following situations, the tool cannot return to the correct position:

a. A manual operation was performed with MANUAL ABSOLUTE switch off.

b. A manual operation was performed with MACHINE LOCK switch on.

c. The search operation for program restart was performed with MACHINE LOCK switch on and then this switch was turned off.

5.2.5.1 PLAYBACK FUNCTION t (Cent'd)

NOTE : The current value in POSITION (UNIVERSAL) is displayed by depressing the PROG key in the PLAYBACK mode. It is the same display as that shown in POSITION (UNIVERSAL) by depressing the POS key.

(8) Write-in the value, except for axis, by a block of data in the same way as writing in the EDIT mode. (See 4.6 EDIT OF PART PROGRAM.)

(9) Depress the EOB key.

(10) Depress the INSRT key.

A block of data is stored in the part program.

(11) Repeat steps (5) through (10) until the desired data are written.

(12) Key-in M02; or M30; at the end of the part program.

Notes :

1. The part program stored in the PLAYBACK operation can be edited in both PLAYBACK and EDIT modes. Position data stored in the PLAYBACK operation can be edited (deletion, insertion, alteration) .

2. The current value in POSITION (UNIVERSAL) can be written-in using the WR key. Therefore, start PLAYBACK operation after setting up the machining reference point by G92, etc.

3. Part programs made by PLAYBACK operation and the ones made by writing operation are the same. Maximum capacity of part program is the total of both programs.

4. EDIT operation in the EDIT mode is as usual.

5. When the parameter #6006 D5 (ten times the input) is " 1", data written through the keyboard will be multiplied by ten.

Ex. At the current position X1.000, keying-in X, I, O and depressing the WR key will store the value X1.1000.

6. MDI operation cannot be intervened during compensation C in the PLAYBACK mode. If the Cycle Start pushbutton is depressed in the PLAYBACK mode, message "PLAYBACK LOCK !", will be displayed. With PLAYBACK switch turned off, MDI operation can function.

Writing Operation

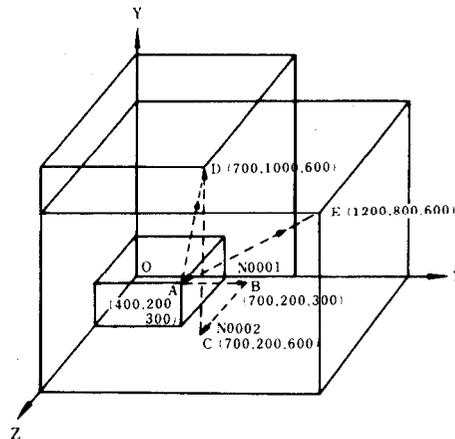


Fig. 5.34

Data of positioning at point A is stored in the part program. The block after point A is stored in the PLAYBACK mode.

A. Procedure of changing current position data using GOO when machine was moved from point A to point B. Sequence No. for this data block is N0001.

(1) Key-in N, O, 0, 0, 1.

Do not depress the WR key. If depressed, "INPUT ERROR !" will appear on the screen.

(2) Key-in G, O, 0.

(3) Depress WR key after X.

(4) Depress the EOB key.

(5) Depress the INSRT key.

Then, block of N0001 GOO XO. 700; will be stored in the part program.

B. Procedure for storing current position data using GO1, F02 when machine was moved from point B to point C. Sequence No. for this data block is N0002.

(1) Key-in N, O, 0, 0, 2.

(2) Key-in G, O, 1.

(3) Depress the WR key after Z.

(4) Key-in F, 2, 0.

(5) Depress the EOB key.

(6) Depress the INSRT key.

Then block of N0002 G01 Z0.600 F20; will be stored in the part program.

C. Procedure for storing current position data using GOO when machine was moved from point C to point D. Sequence No. for this data block is NOO03.

- (1) Key-in N, O, 0, 0, 3.
- (2) Key-in G, O, 0.
- (3) Depress the WR key after Y.
- (4) Depress the EOB key.
- (5) Depress the INSRT key.

Then, block of NOO03 GOO Y1.000; will be stored in the part program.

D. Procedure of storing current position data for three axes simultaneously when machine was moved from point A to point D with linear interpolation Sequence No. for the data block is NOO04.

- (1) Key-in N, O, 0, 0, 4.
- (2) Key-in G, O, 1.
- (3) Depress the WR key after X.
- (4) Depress the WR key after Y.
- (5) Depress the WR key after Z.
- (6) Key-in F, 1, 0, 0.
- (7) Depress the EOB key.
- (8) Depress the INSRT key.

Then block of NOO04 GO1 XO.700 Y1.000 ZO.600 F100; will be stored in the part program.

E. When the machine current position is at point D (700, 1000, 600) with start at A, follow the procedure to store the current position data using the position data of point D when the machine is moved from point A to point E (1200, 800, 600).

Key-in the incremental value for each axis when machine moves from point D to point E. Sequence No. for this data block is N 0005.

- (1) Key-in N, O, 0, 0, 5.
- (2) Key-in G, O, 1.
- (3) Depress the WR key after keying-in XO. 5.
- (4) Depress the WR key after keying-in Y-O. 2
- (5) Depress the WR key after keying-in Z.
- (6) Key-in F, 1, 0, 0.
- (7) Depress the EOB key.
- (8) Depress the INSRT key.

Then, block of NOO05 GO1 X1.200 YO. 800 ZO.600 FO. 600 F100; will be stored in the part program.

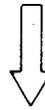
```

PROG (PLAYBACK)          00010N0000
00010;
G91 G28 XO YO ZO;
G92 X9 YO ZO;
G90 G00 X0.4 YO.2 ZO.3;
N0001 COO XO. 700;
NOO02 G01 ZO.600 F20;
NOO03 GOO Y1.000;
NOO04 GO1 XO. 700 Y1.000 ZO.600 F100;

Pos      x      .700      Y1.000
(uNIV)   z      .600

NOO05 G01X0.5                      RDY
  
```

Fig. 5.35



BY DEPRESSING WR KEY, POSITION X IS ADDED TO THE KEYED-IN VALUE. AFTER THAT, EACH TIME THE KEY IS DEPRESSED, POSITION X IS INCREASED BY MULTIPLES (X2, X4, X8 . ..).

```

PROG(PLAYBACK)          00010N0000
00010;
G91 G28 XO YO ZO;
G92 XO YO ZO;
G90 G00 XO.4 YO.2 ZO.3;
N0001 GOO XO. 700;
NOO02 G01 ZO.600 F20;
NOO03 GOO Y1.000;
NOO04 GO1 XO. 700 Y1.000 ZO.600 F100;

Pos      x      .700      Y1.000
(uNIV)   z      .600

NOO05 GO1 X1.200                      RDY
  
```

Fig. 5.36

5.2.5.2 PLAYBACK FUNCTION 2

In addition to the conventional playback function, the current value of each axis, as well as the cutting feedrate F, spindle speed S and tool command T execution value can be written by the external writing signal.

(1) Writing by key operation

The operation procedure is as usual. Refer to the Operator's manual (Par. 5.2.5.1, **PLAYBACK function**).

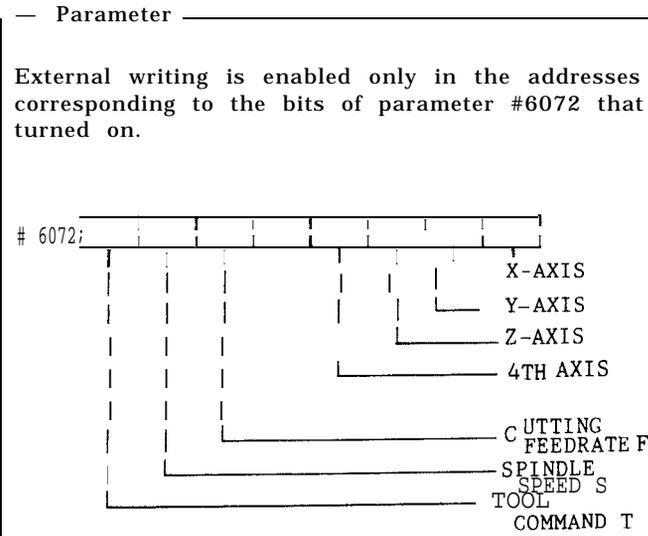
The cutting feedrate F, spindle speed S and tool command T are also written in the same way as the axis data. Note that if a numeral is typed on the keyboard before depressing the WR key, the numeral becomes the command value. If no numeral is input on the keyboard after typing F or S, over-ridden values of the current cutting feedrate command F or spindle speed S become the command value.

5.2.5.2 PLAYBACK FUNCTION 2 (Cent'd)

(2) Writing by the external writing signal

External writing signal #13094

When this signal is input, the current value of the axis specified by parameter #6072, as well as the cutting feedrate F, spindle speed S and tool command T values can be written as the command values.



(3) Operation procedure

- (a) Select manual mode (RAPID, JOG, STEP, HANDLE) .
- (b) Turn on the playback switch.
- (c) Set the function key at PRG.
- (d) Register the O number to be created.
- (e) Move to the position where the command value is to be written, according to the selected mode.
- (f) Turn ON the external writing signal (#13094). The current value of the axis specified in parameter #6072 as well as the F, S, and T values are compared with the old values; if any change is observed, the new value is written.
- (g) Repeat the above (e) and (f) , to edit the setting values.

* If the axis move is not completed when the writing signal is input, "INPUT ERROR" flashes on the CRT, to disable any insertion until this error is canceled by the cancel key.

Note :

The following are the command values of F, S, and T.

- F: Current command F x Override (%)
- s: Current command S x Override (%)
- T: Current command tool

5.2.6 MANUAL SKIP FUNCTION †

(1) Outline

The operation (feed) stops by turning ON the tactile sensor under the manual skip mode. The coordinate value where the tactile sensor turned on can also be read.

(2) Manual skip operation

- (a) Call up the manual mode.
- (b) Call up the manual skip mode, by turning on WSKANI (#13356) and WSH (#13091).

— The work coordinate system shift amount setting screen is automatically selected.

— The LED flashes.

(C) Contact the tactile sensor at an optional point (PI) on the datum plane, by manual handle operation.

The X, Y and Z axes positions (POS_MACHINE) where the tactile sensor signal "PST" turned on are stored in #6844: X1; #6845: Y1; and #6846: Z1.

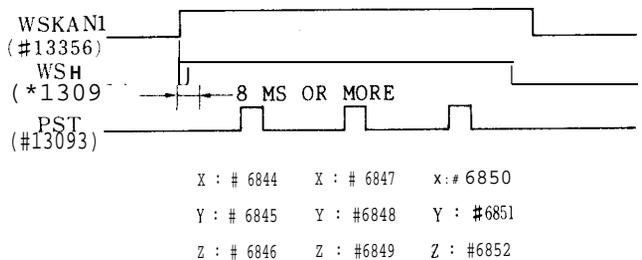
(d) Contact the tactile sensor at the second measurement point (P2) .

The X, Y, Z axes positions (POS_MACHINE) where the tactile sensor signal "PST" turned on are stored in #6847: X2; #6848: Y2; and #6849: Z2.

(e) Contact the tactile sensor at the third measurement point (P3) .

The X, Y, Z axes positions (POS_MACHINE) where the tactile sensor signal "PST" turned on are stored in #6850: X3; #6851: Y3; and #6852: Z3.

(f) After measurement of the three points is completed (turn WSKANI and WSH off) , and exit the manual skip mode.



Notes:

1. It is also possible to exit the manual skip mode after measuring the first point alone.
2. If a fourth point is measured, the result will be overwritten on the first point.

(3) Remarks

(a) POS [MACHINE] is used for the manual skip measurement. Do not forget the reference point return before the measurement.

(b) If writing is done without completing the reference point return, alarm occurs when the PST signal turns on. (ZR UNREADY (X)).

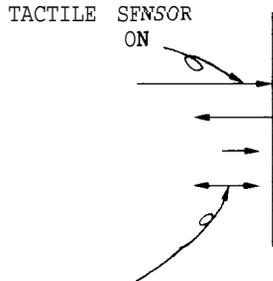
(c) The WSKAN1, WSH signals selected before selecting the manual mode are invalid.

It enters the manual skip mode ( LED blinks)

only when the WSKAN1 and WSH signals are selected after selecting the manual mode. The PST signal is also valid only when activated (turned on from off) under the writing mode.

(d) Be sure to touch (SKIP signal ON) the datum plane with the same axis.

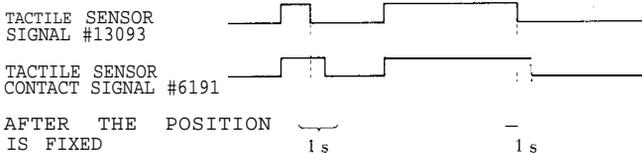
(e) When the tactile sensor turns on, it must be pulled back for a certain amount (parameter #6578) , or it cannot be moved in the same direction again.



PARAMETER #6578 1 = 1 PULSE (WITHOUT SIGN)

(f) Tactile sensor contact signal

The tactile sensor contact signal operates in the following timing, when the tactile sensor signal turns on.



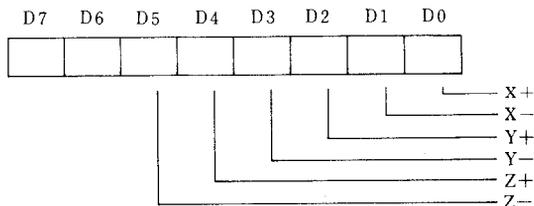
When the tactile sensor signal turns on, the tactile sensor contact signal turns on after the position is fixed, and when the tactile sensor signal turns off, the tactile sensor contact signal turns off after 1 second.

The tactile sensor contact signal (#6191) is "0" when off, and "1" when on.

(g) Tactile sensor contact direction monitor

When the tactile sensor contact signal (see above) turns ON, it can be made determined which axis contacted the sensor and from which direction.

Tactile sensor contact direction monitor (#6196)



(h) Manual skip warning monitor

The following warnings appear when a measurement mistake is made while using manual skip. If this warning appears, #6192 becomes "1." It becomes "0" by resetting the warning.

(EXAMPLE) SET P3 (Point ERROR)

(i) Manual skip measurement point monitor

Refer to parameter #6194 to find on which point the measurement is made.

The value changes as; 0, 1, 2, 0, 1, 2.

It returns to the initial state if reset during the setting (SET P2 or SET P3).

5.2.7 FEEDRATE, SPINDLE SPEED EDITING FUNCTION †

(1) Outline

The execution values of the feedrate and spindle speed during automatic operation are stored and feed back to the NC part program to create the optimized program.

The feedrate (F code) and spindle speed (S code) will be called FS in the following description.

(2) 1/0

(a) Input

(i) FSCM (#13134)

FS editing mode

Close this signal during FS editing, to prepare for FS storage and FS memory change.

(ii) FSMEM (#13135)

FS storage

Close this signal during automatic operation to store the operation feedrate and spindle revolution data into the internal memory.

(iii) FSCH (#13136)

FS memory change

Close this signal while the automatic operation is held, to reflect the data stored during automatic operation in the NC part program.

(iv) FSCLR (#13357)

FS data clear

Close this signal to clear all the stored FS data.

(b) output

(i) FSMD (#12090)

During FS editing mode

This signal indicates that the FSCM is closed and it is ready for FS storage and FS memory editing.

When this signal changes from closed-to open, the feedrate and spindle speed data are canceled.

(ii) FSCE (#12091)

End of FS memory change

This signal closes when the FSCH closes and the data are reflected in the NC part program. It opens again when the SFCH opens again.

5.2.7 FE EDRATE, SPINDLE SPEED EDITING FUNCTION ' (Cent'd)

(iii) FSCLRE (#12092)

End of FS data clear

This signal closes when the FSCLR closes and the stored FS data are cleared. This signal closes again when the FSCLR opens.

(3) Functions

The rough calculation of the FS is made in the NC part program to leave the optimization of the feed override and spindle override after the actual cutting.

This optimization data and the position of the actual cutting on the program are stored in the NC internal memory and reflected on the part program by closing the FS memory change input.

(a) Operation procedure

(i) Call up the operation mode.

(ii) Search the program to be operated under the memory mode.

(iii) Close the FSCM.

(iv) Check that FSMD is output, and start the memory operation.

(v) Change the feedrate or the spindle speed, for optimization.

(vi) Close the FSMEM when the optimization is completed.

(vii) Repeat (v) and (vi) sequentially.

(viii) Hold the operation (label skip status) .

(ix) Close FSCH.— Enter the execution value stored from FSMEM in the part program. (The cursor position does not change.)

(x) Open FSCH. — FSCE closes.

(xi) Open FSCM. — Data stored in (v), (vi) is canceled.

(EXAMPLE)

(Before FS editing)

```

o 1 2 3;
G28 X10. Y10. Z10. ;
G92 X100. Y100. Z100. ;
GOO G90 F100. ;
M03 S3000 ;
G01 X...;
  Y...; → FSMEM Closes when the override reaches 80%
        and S override 110%.
  Z...;
  X"" Y...;
  X... Y...; → OVERRIDE 60%, S OVERRIDE 100%
  X... Y...;
  X... Y...;
  X... Y...; → OVERRIDE 50%, S OVERRIDE 80%
  X... Y...;
  X... Y...;
  Z...;
GOO X...Y...Z...;
M30 ;

```

(After FS editing)

```

o 1 2 3;
G28 X10. Y10. Z10. ;
G92 X100. Y100. Z100. ;
GOO G90 F100. ;
M03 S3000 ;
G01 X...;
  Y... F80.S3300 ;
  Z...;
  X... Y...;
  X... Y... F60.S3000 ;
  X... Y...;
  X... Y...;
  X... Y... F50.S2400 ;
  X... Y...;
  X... Y...;
  Z...;
GOO X...Y...Z...;
M30 ;

```

(b) Conditions for the FS editing

(Conditions to store the FS data into the NC internal memory)

(i) The FSMD must be during output.

(ii) Must be during automatic operation (STL close) .

(iii) The effective values of the feedrate and spindle speed are stored upon rise of the FSMEM.

(iv) The data stored in the above (iii) are all canceled upon fall of the FSCM.

(v) Storage can be made for a maximum of 128 times (the times that FSMEM can be closed).

(Conditions to reflect the FS data on the part program.)

(i) The FSMD must be during output.

(ii) Must be during automatic operation (M02 /M30) Open to close the FSCH under the above conditions, and the F, S data are inserted into each block according to the stored data.

(4) Notes

(a) When the FSMEM is closed two or more times in a single block, the later feedrate becomes effective. However, the FSMEM count is not incremented.

(b) When the FSMEM is closed in the subprogram, the FS data are inserted in the subprogram itself. This needs care, as the subprogram may also be called up from another program.

(c) The cycle start button is disabled while the FSCH is closed.

(d) The FSMEM cannot effectively be closed during the tapping cycle.

(e) Close the FSMEM during cutting, to insert the F, S data,

(f) Close the FSMEM during rapid feed, to insert the S data.

(g) When the FSMEM is closed 128 times or more, a warning is output, to permit no further storage.

(h) The following system variable can be used to monitor the FSMEM.

FSMEM count #6180	128 for 128 times or more
Warning #6181	DO 128 over

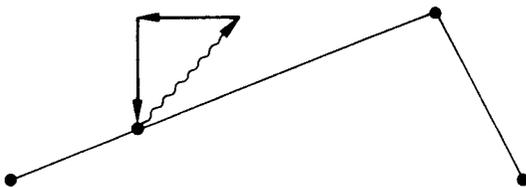
(i) The FSCM cannot be closed during automatic operation.

(j) The cycle start cannot be enabled while the FSMEM is closed.

(k) Warning occurs if editing is attempted with the FSMD closed, after the FS editing is completed.

5.2.8 MACHINING INTERRUPTION POINT RETURN FUNCTION †

This function is used to recover the automatic mode operation interruption point, after interrupting the automatic mode and moving the machine by manual operation for measuring the workpiece or removing the chips.



(1) Machining interruption point recovery procedure.

(a) Interrupt the automatic mode operation as follows .

- ① Turn on the single-block switch.
- ② Depress on the feed hold button.

(b) Call up the manual mode.

(c) Move the machine under manual mode to free the tool.

(d) Measure the workpiece or remove the chips.

Note :

The condition of the machine when the mode is switched from automatic mode to manual mode may differ in some models.

(e) Turn on the machining interruption point switch.

(f) Depress the manual feed selection button of the machining interruption point (the point where it changed from automatic mode to manual mode).

Move the machine at the speed selected for the machining interruption direction :

When reaching to the point, it stops automatically.

(g) Turn off the machining interruption point return switch.

(h) Return to the automatic mode, and depress the start button to restart the automatic operation.

(2) Notes

(a) The machining interruption return switch must be turned off if manual jogging is to be done, if it is during machining interruption point return.

(b) If the NC is reset after changing the mode from automatic mode to manual mode, no later machining interruption point return can be performed.

(c) When move is made from the automatic mode to manual mode, then returned to the automatic mode, and the move is performed again by switching the automatic mode to manual mode, the machining interruption point is the position where the mode is switched from automatic mode to manual mode the last time.

5.2.9 CURRENT VALUE NUMERICAL SETTING FUNCTION †

Numerical setting can be made in the same way as depressing the "ORG" key in the current position display "Universal" and "external" screens to display "O" .

(1) Call up the manual mode.

(2) Select the "universal" or "external" screen.

(3) Input the axis to be set-up and the data by the keys, and then input "ORG" for display set-up.

(4) Set "axis designation /ORG" to divide the current value by two.

5.2.10 FEED STOP FUNCTION[†] BY SENSOR SIGNAL

Turn on the sensor stop switch, to receive the sensor input signal necessary for centering, groove width detection step difference designation, and then to stop. When the sensor input is on, operation is not made in the approaching direction, but only in the leading direction.

Therefore, centering, groove width detection etc. can also be performed by using this function and the numerical setting function.

Notes :

(1) The axis that stopped by sensor input will not move unless once returned in the opposite direction.

Set this axis return distance in parameter #6578. Always set this parameter to prevent sensor chattering, (1 = 1 pulse)

(2) The sensor stop function is cancelled when the mode is changed to the automatic mode. For safety, it is recommended to keep the sensor away from the workpiece when changing the mode to automatic mode.

5.2.11 MANUAL CENTERING FUNCTION[†]

(1) Call up the manual mode.

(2) Select the current screen. "Universal, " " External. "

(3) Turn on the sensor stop mode switch.

(4) Specify the centering axis.

If X "0" or X "123.456" WR or the like is then specified, this value is temporarily stored , and displayed as follows.

x 1 2 3 . 4 5 6

(5) Move the X axis and place the sensor on the circumference of the circle. The feed stops when the sensor generates a signal, and the value stored in (4) is set up at that point.

(6) Move the axis in the opposite direction, and place the sensor on the circumference of the reverse side. The feed stops when the sensor generates a signal, and the center of the two points touched by the sensor is set up.

(7) Repeat (4) to (6) above also on the y axis.

(8) Turn off the sensor stop mode switch.

5.2.12 SPINDLE INDEXING FUNCTION[†]

The spindle indexing function is a function that stops the spindle at an optional position (a position with an optional rotary angle) .

5.2.12.1 ADDITIONAL CONDITIONS FOR THE SPINDLE INDEXING FUNCTION[†]

The spindle indexing function must be understood well, and the conditions consolidated before use.

(a) The S5-digit analog output option must be incorporated.

(b) Out VS-626 VM3 must be mounted on the spindle drive.

(c) PG must be mounted on the spindle or spindle motor, and the gear ratio to be used fixed at 1:1. (The best performance is gained when the spindle and the motor are direct-link type).

(d) The number of feedback pulse per spindle rotation must be 4096.

(e) The solid tap function option must be validated.

5.2.12.2 RELATED PARAMETERS

The following parameters must be preset to use the spindle indexing function. If any change is made to the parameters, be sure to turn off the power once.

(1) Spindle PG mounting position

#6065 D7 1: Motor side
0: Spindle side

Note :

This parameter is used to link the spindle and the motor with the gear. Set "0" to use the spindle indexing function.

(2) Gear ratio of spindle and spindle motor

#6198 Spindle side gears
#6199 Motor side gears
Setting range : 1 to 127

Note :

Set the gear ratio when linking the spindle and spindle motor with the gear. Set "0" both #6198 and #6199 when using the spindle indexing function.

(3) Spindle feedback magnification

#6061 D3 to D0

D3	D2	D1	D0	Setting Magnification
0	0	0	1	×1
0	0	1	0	×2
0	1	0	0	×4
1	0	0	0	×8

Standard setting magnification: × 4

(4) Feedback pulses per spindle PG rotation

#6449 Setting unit: 1 = 1 pulse
Setting range: 1 to 32767
Standard setting value: 1024

Notes :

1. Set the value before commanding the spindle feedback pulse magnification, in this parameter.
2. When performing spindle indexing, set the parameter so that the feedback pulses (#6061 x #6449) per spindle rotation become 4096 pulses.

(5) Spindle revolutions against commanded voltage 10 V in the spindle indexing execution gear

#6472 Setting unit: 1 = 1 rein-l
Setting range: 1 to 32767
Standard setting value: 6000

(6) Spindle designation unit position loop gain

#6486 Setting unit: 1 = 0.01 s⁻¹
Setting range: 1 to 32767
Standard setting value: 2500

(7) Spindle error detection ON area

#6174 Setting unit: 1 = 1 pulse
Setting range: 1 to 255
Standard setting value: 50

(8) Spindle indexing rotary speed reference

#6462 Setting unit: 1 = 500 pulse/s
Setting range: 1 to 32767
Standard setting value: 20

Note :

The spindle indexing rotary speed cannot exceed the maximum speed of the spindle (#6472) .

(9) Spindle indexing creep speed reference

#6463 Setting unit: 1 = 500 pulse/s
Setting range: 1 to 32767
Standard setting value: 10

Note :

The spindle indexing creep speed cannot exceed the maximum speed of the spindle (#6472) .

(10) Spindle indexing spindle stop acknowledgment timer

#6464 Setting unit: 1 = 8 ms
Setting range: 1 to 255
Standard setting value: 1

(11) Spindle indexing reference point position

#6465 Setting unit: 1 = 1 pulse
Setting range: 0 to 4096
Standard setting value: 0

(12) Spindle stop acknowledgment revolutions

#6473 Setting unit: 1 = rein-l
Setting range: 0 to 30
Standard setting value: 30

(13) Spindle revolutions upper limit for solid tap

#6471 Setting unit: 1 = 1 rein-l
Setting range: 1 to 32767
Standard setting value: 4500

5.2.12.3 1/0 SIGNALS

The 1/0 signals described here are used to transmit the operation request and the ending between the NC unit and the sequencer. Spindle indexing can normally be performed by the miscellaneous function command, without considering these I / O signals.

(1) Spindle indexing position command

#1341 D7 to DO
#1342 D3 to DO

Note :

A binary 12-bit (0 to 4096) input that specifies the spindle stop position. The sum of the data input to this external input signal and the data preset in the parameter (#6465) is the spindle stop position.

(2) Spindle indexing input

#1343 DO 1: Spindle indexing request ON
o: Spindle indexing request OFF

Note :

If this input turns on while the spindle is rotating, the revolution command for the spindle indexing operation is output. The spindle indexing operation is performed when the spindle stops. The spindle revolution command analog output is still the spindle positioning command even after the spindle indexing is completed, and continuous the spindle indexing operation until this input is turned off.

(3) Spindle indexing position incremental command

#1343 DI 1: Spindle indexing incremental request ON
o: Spindle indexing incremental request OFF

Note :

This input is to accept, the spindle indexing position command input prepared for multiple spindle indexing to different positions, as the incremental position command from the previous spindle indexing position. However, turning on this input for the first spindle indexing after rotating the spindle with an operation other than the spindle indexing operation, or the first spindle indexing operation after the power is supplied, will not make an incremental position command.

(4) Spindle indexing restart input

#1343 D2 1: Spindle indexing restart request ON
o: Spindle indexing restart request OFF

5.2.12.3 I/O SIGNALS (Cent'd)

Note :

If this input is turned on while the spindle indexing operation execution output is on, the spindle indexing operation is interrupted, and the spindle indexing operation execution output is turned off.

By turning off this input in such conditions, the spindle indexing operation restarts.

(5) Spindle indexing execution output

#1207 D2 1: Spindle indexing execution
 0: Not spindle indexing execution

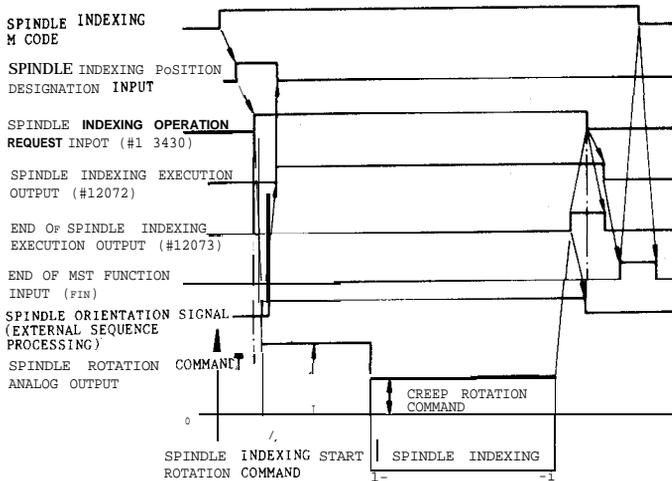
(6) End of spindle indexing execution output

#1207 D3 1: End of spindle indexing
 0: Spindle indexing not completed

Note :

The spindle indexing input signal must be turned on and the spindle drive unit soft start cancel input signal turned on by the sequencer, to execute the spindle indexing function.

5.2.12.4 SPINDLE INDEXING TIMING CHART [NC SIDE]



5.2.12.5 SPINDLE INDEXING PROCEDURE

Before using the spindle indexing function, it is important to consolidate the hardware resource, and to control the I/O signals by the sequencer. The following is a general operation procedure. For details, refer to the instruction manual of the machine tool manufacturer.

- (1) Command the M code specified by the machine tool manufacturer the position in the part program to perform the spindle indexing.
- (2) Execute the part program. The spindle indexing completes by executing the spindle indexing M code.
- (3) Then continue to execute the part program.

Note :

Spindle indexing cannot be performed under the solid tap mode. Command spindle indexing after canceling the solid tap mode.

6. OPERATION PROCEDURE

6.1 INSPECTION BEFORE TURNING ON POWER

Make sure that the front and rear doors of the control are firmly closed. The control employs a totally-enclosed, dustproof enclosure to shut out surrounding air. If the door is open, lock it by turning two door lockers with a large screwdriver. In addition, inspect the machine referring to the machine tool builder's manual.

6.2 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 air 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 DISPLAYING ALARM CODE.

6.3 MANUAL OPERATION

When the MODE SELECT switch on the machine control station 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% - F0
3. Push JOG button to select the axis and direction of movement. 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 to select the axis and direction of movement. 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 -100.0
mm /step
Inch: 0.0001 - 0.001 - 0.01 - 0.1 - 1.0-10.0
inch/step
3. Depress JOG button to select the axis and direction of movement. 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 "x1 0000" 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.

6.4 PREPARATION FOR STORED LEADSCREW ERROR COMPENSATION AND STORED STROKE LIMIT†

Return to Reference Point

With an NC equipped with the stored lead screw error compensation or the stored stroke limit functions, either of the following two reference point return motions must be performed after switching on the power supply and before starting automatic operation.

- a. Manual return to reference point (See 5.2.1)
- b. Execute G91 G28 XO YO ZO ; in the MDI mode.

This procedure is to teach the reference point to the control, since doing so is necessary because both pitch error compensation and stored stroke check are performed with reference to the reference point.

Checking Parameter #6006D₁

When the control is equipped with the pitch error compensation function or the stored stroke limit function, set this parameter to "1". With the parameter #6006D₁ set to "1", a return to the reference point is required before starting cycles, alarm codes (001 to 004 "reference point return incomplete") are displayed, if the CYCLE START key is pushed without making a reference point return immediately after turning on the power supply. Be sure to perform the operation for return to reference point.

Set this parameter to " 1" when pitch error compensation or stored stroke limit is provided.

6.5 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 point.
(Refer to 5.2.1.1 MANUAL RETURN TO REFERENCE POINT).
 - . The G92 command according to the part program should be executed by MDI .
G92 X... Y... Z... ;

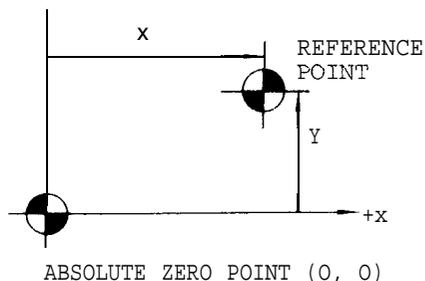


Fig. 6.1

If "G92 XO YO ZO ;" setting is required, the coordinate of each axis can be set to "0" easily using ORG key. Refer to 4.1.9 ORG KEY.

EXAMPLE

```
EOR ;
N1 GOO X... Y... Z... ;
```

Fig. 6.2

2. When G92 is programmed in a tape or memory.

When the program requires G92 to be executed at the reference point, return the machine to the reference point by manual return to reference point return.

EXAMPLE :

```
EOR ;
N1 G92 X... Y... Z... ;
```

Fig. 6.3

3. When G28 and G92 are programmed .

When the program begins with G28 and with G28 and G92, move the machine manually into the area where return to reference point can be performed.

EXAMPLE :

```
EOR ;
N1 G28 x... y... z... ;
N2 G92 X... Y... Z... ;
```

Fig. 6.4

6.6 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, "DISPLAYING ALARM CODE. "
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 of 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 control station . 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.

6.7 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 .
3. Set MODE SELECT switch to manual operation mode (HANDLE, JOG or RAPID) , and the machine can be manually operated.
4. Return the machine manually to the recorded positions .
5. Set MODE SELECT switch to the interrupted automatic-mode (TAPE, MD I 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 immediately 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 5.1.24, "MANUAL ABSOLUTE SWITCH".

In manual operation mode, 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 MD I operation, the command becomes effective and is executed as soon as written. This procedure is used to add new data to an active buffer. However, MOO, M01, M02, M30 and M90 to M99 cannot be written.

6.8 AUTOMATIC OPERATION IN MDI MODE

1. Set MODE SELECT switch to MDI position.
2. Write up to 10 blocks of data by MDI operation, and execute by pressing CYCLE START. Refer to 4.3.3 WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI.
3. Depress CYCLE START button, and automatic operation can be executed in MD I mode.

6.9 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 interrupted after the completion of the block being executed. At the same time, the next block of data may be read in advance. However, several blocks may sometimes be read ahead.
2. Verify the buffer data according to 4.3.2 COMMAND DATA DISPLAY.
3. Set MODE SELECT switch to MDI operation.

6.9 MDI OPERATION INTERRUPTING AUTOMATIC OPERATION (Cont'd)

4. Write the data referring to 4.3.3 WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI.
5. 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.

Notes :

. Writing data by MDI cannot be executed in tool radius compensation modes (G41, G42) because two to four blocks are read ahead. In principle, MDI operation cannot interrupt. However, F, M, S, T and B[†] codes can be output by the 6.7 MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION.

. Writing data by MDI should not be performed in canned cycle modes (G73, G74, G76, G77, G81 to G89) . The machine may not operate properly.

Tape mode MDI interrupting is available only after completion of advance reading stop block operation.

. Any other than described above are possible in MDI operation.

6.10 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 is not indicated on CRT. If alarm is displayed, detect the causes of displayed alarm code and eliminate them. Refer to 4.3.12, " DISPLAYING ALARM CODE " .
3. Inspect the machine referring to the machine tool builder's manual.

6.11 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.

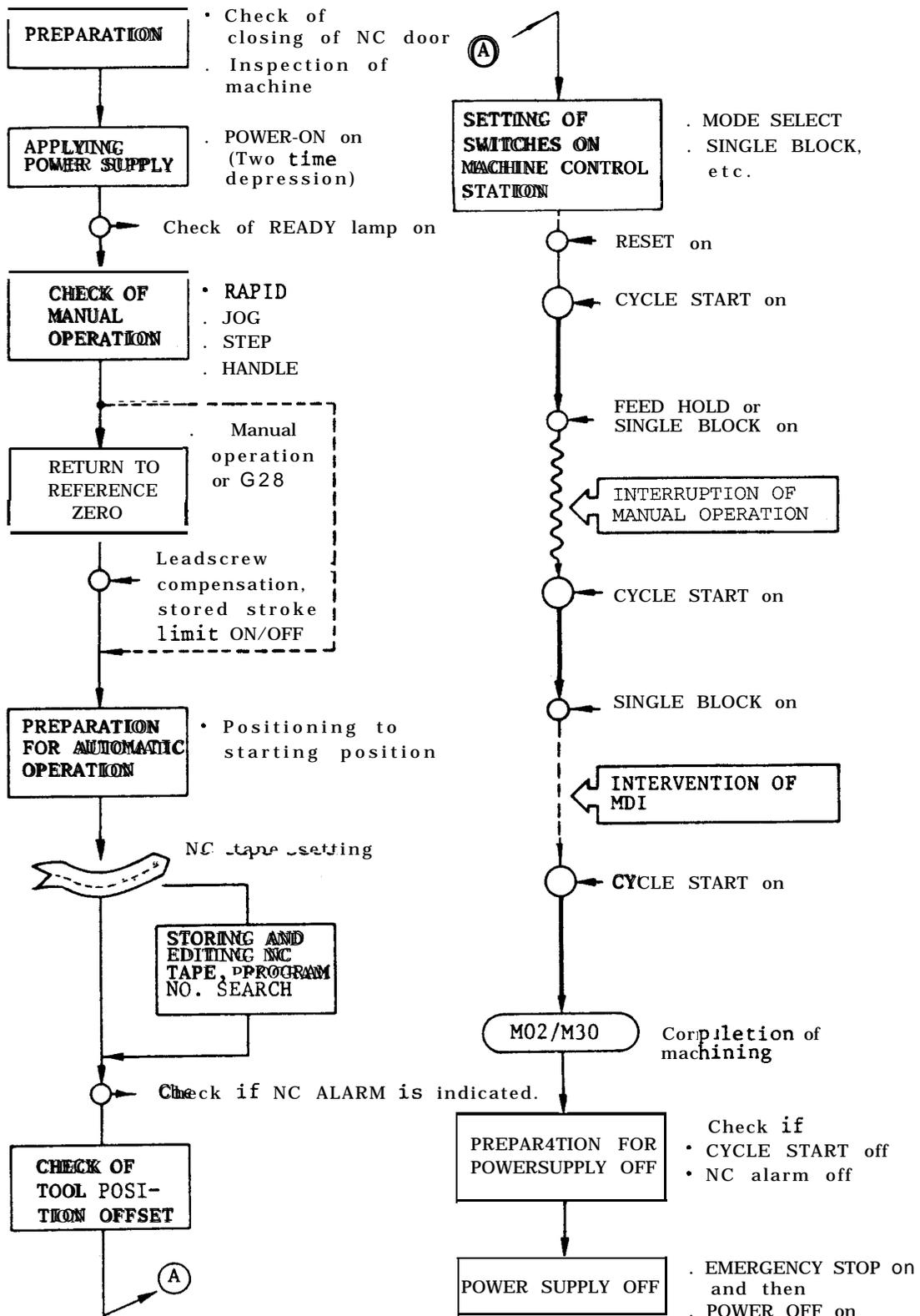


Fig. 6.5 Operating Procedure

7. MAINTENANCE

7.1 ROUTINE INSPECTION SCHEDULE

The following describes the requirements to be observed for maintenance according to time in order to keep the equipment in optimum condition for extended period.

Except for those checks which can be made with the NC energized, such as checks for external cleanliness for vibration and for noise, be sure to turn off the power supply to the NC before starting to undertake routine maintenance service.

For this, turning off the power supply by pushing the POWER OFF button on the NC operator's panel is not sufficient, because after this button is pushed, several areas in the housing are energized, and are potentially dangerous. If power supplied to the NC is cut off, all fans in the unit are stopped.

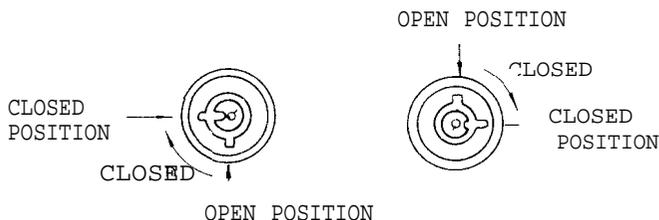
7.1.1 CONTROL PANEL

(1) Doors are completely closed. (Daily)

(a) The unit is of a totally-enclosed type and designed to prevent the outside air containing oil mist, etc. will not enter the panel. Be sure to close the doors of the control panels at any time other than during operation.

(b) When it is necessary to open/close the doors for maintenance, lock the doors by turning two door locks with the key (model : YEOO 1) attached to the control panel. When opening or closing the doors, insert the key and press and turn it until it stops.

The key can be removed in either position of open or closed.



With the door lock on right side of the door.

With the door lock on left side of the door.

Fig. 7.1

Note : If the optional door interlocking switch[†] is provided, opening the door shuts off the main power supply and stops all operations.

(2) Inspection on gaps and damages in sealed structure (monthly)

(a) Open each door to check that sealing packing around the door is not damaged.

(b) Check that there is no abnormal contamination inside of the unit. If there is, wipe it off after researching the cause.

(c) Check that there is no gap when the door is locked completely.

By executing the items described in this paragraph, the NC control panel can be operated stably for a longer period.

7.1.2 SERVOMOTOR AND DC MOTOR FOR SPINDLE

1. Vibration and noise. (Daily or when necessary)

Vibration can be checked by resting the hand on the motors, and for noise, using a listening stick is recommended. If any abnormality is found, contact maintenance personnel immediately.

2. Motor contamination and impairment. (Daily or when necessary)

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.

7.1.3 BATTERY

Make sure that "BAT" or "A/B" on the right-low position of CRT screen is not displayed. If it is displayed, inform maintenance personnel. The battery must be replaced with a new one within a month.

7.2 BATTERY REPLACEMENT

The battery is used as power source for memory in order to prevent programming g data stored in memory , such as parameter , tool offset and part program from erasing .

When the battery is discharged after a long period of use, "BAT" or "A/B" is blinked on CRT screen to give warning for replacement. On such occasions, the battery must be replaced within 30 days. When replacing , never remove the old battery with power off, otherwise the data stored in memory may be cleared.

Commercially sold batteries cannot be used ; contact your Yaskawa representative before replacement of batteries.

Replacing Procedure

1. Depress POWER OFF pushbutton.
2. When door interlock switch provided
Open the front door and pull out the door interlock

switch movable section by hand (l-layer snap action is provided).

Then the power supply can be turned on with the door open.

3. Open the door so that the CPU module (model : JZNC-JRK□□) mounted inside of the front door can be seen.

Note : Some models have the CPU module mounted not on the door but on the main unit.

4. Depress POWER ON pushbutton. Where the control is equipped with a door interlock switch, pull it out by hand. The power can be turned on, with the door open.

5. Verify that LED1 on the CPU module (JANCD-PC50) lights, which indicates that replacement of battery "ER6V3" is needed.

Fig. 7.2 shows the positions of the LED indicator lamp and the battery.

Check for the defective battery by the above procedure.

"ER6V3": YE model JANCD-BB51

6. Turn off the power supply. Remove the battery cover to remove the existing battery mounted on the holder.

7. Insert new battery into the battery holder and then plug it into the connector.

Note : The connector may be inserted even upside down.

8. When door interlock switch is provided, depress the movable section.

The switch returns to the previous position and the power supply cannot be turned on when the door is "open".

9. Be sure to close the front door.

10. Turn the power off.

11. Confirm that the battery alarm display on the CRT screen of NC operation panel and red display lamp on the front of JANCD-PC50 board have been turned off.

Notes :

(1) Since the work is performed with the front door open, it is necessary to work as quickly as possible (to prevent contamination by oil mist, dust, etc.).

(2) Contamination by water oil, dirt, etc. to printed boards, connectors, cables, etc. inside the equipment must be avoided.

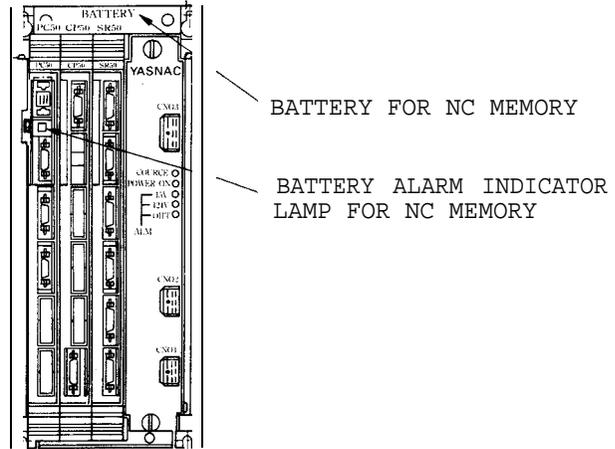


Fig. 7.2

7.3 POWER SUPPLY

Various kinds of protective functions are provided for compound power supply (CPS- 18 F).

However, in the event of a malfunction such as no power to NC, it is necessary to confirm the items stated below, immediately alert maintenance personnel and then rectify the causes.

7.3.1 "SOURCE" LED [GREEN] UNLIT

(1) Check if the main breaker of power supply has tripped or has an open phase.

(2) Check if the compound control power supply is properly operating.

7.3.2 "+5 V" LED (RED) LIT

(1) The red LED lights up when an overcurrent due to the short-circuit of output from +5 V or an overvoltage of +5 V is detected.

(2) If the trouble occurs due to overcurrent, re-closing is possible by means of POWER OFF→ON . Re-close after removing the cause of trouble.

(3) If the trouble is caused by the overvoltage of +5 V, turn off the main breaker once, confirm that " SOURCE" LED is out, and then re-close the main breaker.

7.3.3 "+24 V" LED (RED) LIT

(1) This red LED is lit when an overcurrent due to the short-circuit of output from +24 V is detected.

(2) Re-closing by POWER OFF→ON is possible. Thus, re-close after removing the cause.

7.3.4 "OHT" LED (RED) LIT

(1) This red LED lit when abnormal temperature rise of the internal parts is detected.

(2) When this is because of abnormal temperature rise, the power supply can be turned on after POWER OFF → ON.

7.4 THERMAL OVERLOAD RELAY OF SERVO UNIT

The servo control unit has the function of detecting the following alarm status.

	Alarm No.
Circuit protector trip	331(X) 332(Y) 333(Z) 334(4)
Overload	351(X) 352(Y) 353(Z) 354(4)
Servo alarm	390 for all axes

7.4.1 CIRCUIT PROTECTOR TRIP (ALARM NO. 331 TO 334)

When the main circuit of the servo control unit is shorted or when the servo control unit itself becomes faulty, the circuit protector in the unit will be tripped, and the following alarm Nos. will be displayed on the CRT.

331: FUSE(X) . . . for X-axis

332: FUSE(Y) . . . for Y-axis

333: FUSE(Z) . . . for Z-axis

334: FUSE(4) . . . for 4th axis

When a circuit protector is tripped, and the alarm No. 331 to 334 is displayed, do not attempt to take measures, but the user should immediately notify your Yaskawa representative.

7.4.2 OVERLOAD (ALARM NO. 351 TO 354)

The servo control unit is provided with electronic thermal relays respectively and independently for the X and Z axes, and they trip under the following conditions.

programs involving excessively heavy cuts are executed.

Programs involving excessively frequent speed changes are executed.

Frictions in the machine system become excessively large.

When the electronic thermal relay trips, the servo power supply is turned off, and the following alarm Nos. are displayed on the CRT .

351: OL(X) . . . X-axis overload

352: OL(Y) . . . Y-axis overload

353: OL(Z) . . . Z-axis overload

354: OL(4) . . . 4th axis overload

When this is the case, take the following measures.

- (1) Push the POWER OFF button to turn off the power supply, and then, stop the supply of power to NC.
- (2) Find the cause of the overloading . For example , the cause may be eliminated through modifications of the part program, or by the elimination of abnormally large load on the machine.
- (3) Supply power to the NC, and push the POWER ON button to turn on the power supply and make the system ready for operation. However, since the servo motor requires approximately 30 minutes to cool down after being overloaded to the extent of tripping the electronic thermal relay, wait at least 30 minutes before starting to operate NC .
- (4) If the electronic thermal relay trips, notify your Yaskawa representative.

7.4.3 SERVO ALARM (ALARM NO. 391 TO 394)

The servo control unit can detect the following alarm states.

- (1) Where "SERVO ALARM (SERVOPACK)" is displayed on the CRT, open the door of NC unit, confirm the LED display for each axis on the servo control unit, and then immediately notify your YASKAWA representative.
- (2) Release the door interlock switch before opening the door of NC unit.
- (3) Read the accompanying Maintenance Manual for details.

Table 7.1 Relation between Display Specifications for LEDs and Output Signals

LED Name	Indications (When Lit)
MP	Main circuit voltage (higher than 200 VDC) in Servopack is normal.
P	Control power supply (+5 V) in Servopack is normal.
IN	Speed command input (higher than 60 mV) has been applied.

Display	Indication (When Lit)	Relation With output Signals
<input type="checkbox"/>	Base breaking state	
<input type="checkbox"/>	(No current through motor)	
<input type="checkbox"/>	Overcurrent detected	
<input type="checkbox"/>	Circuit protector trip detected	• Base of power circuit in Servo-pack is interrupted. • Self-hold is made until reset.
<input type="checkbox"/>	Abnormal regeneration detected	
<input type="checkbox"/>	Overvoltage detected	
<input type="checkbox"/>	Overspeed detected	
<input type="checkbox"/>	Undervoltage detected	
<input type="checkbox"/>	Overload detected	
<input type="checkbox"/>	Heat sink overheat detected	
<input type="checkbox"/>	A/D error detected	
<input type="checkbox"/>	Open phase detected	
<input type="checkbox"/>	Runaway prevention activation detected	
<input type="checkbox"/>	Abnormal MPU (unlit)	Servo alarm output

7.5 MOLDED-CASE CIRCUIT BREAKERS (MCCB)

Those special housing type controls, with which all the power sequence control circuits are converted to the NC area, are sometimes provided with MCBS which can be turned on and off externally.

Generally, when these MC CBS are turned off, the power supply to the NC is stopped. For details, refer to the manual of the machine tool builders.

7.6 TROUBLE CAUSES AND REMEDIES

7.6.1 ON-LINE SELF-DIAGNOSTICS

On-line self-diagnostics are implemented to locate 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 three-digit alarm code including a code showing an axis in error.

Display of four-digit status code including a function code showing M, S, T, F, DWELL.

Input/output signal display

These displays can be made at any time, while the machine is in automatic operation, or at stand-by.

7.6.2 ALARM CODES AND REMEDIES

Where "ALM" or "A/B" on CRT screen is blinking and the machine stops, depress the ALM key. Then alarm code and message will be displayed on CRT screen. Alarm codes "800," "810," "820," "830" and "840" are displayed as soon as the corresponding error occurs.

For the remedies for trouble causes represented by alarm codes, see APPENDIX Table A5.2 "LIST OF ALARM CODES".

7.6.3 INPUT/OUTPUT SIGNALS

To clear up the causes indicated by alarm codes, check the input/output signals on the CRT screen.

Input/output signals are divided into standard and custom-built ones, and displayed by specifying the corresponding diagnostic number with keys on the operator's station.

Standard signals are included in every type of YASNAC J50M. Custom-built signals are provided for high-voltage sequence control panel equipped with some type of YASNAC controls.

7.6.3 INPUT/OUTPUT SIGNALS (Cent'd)

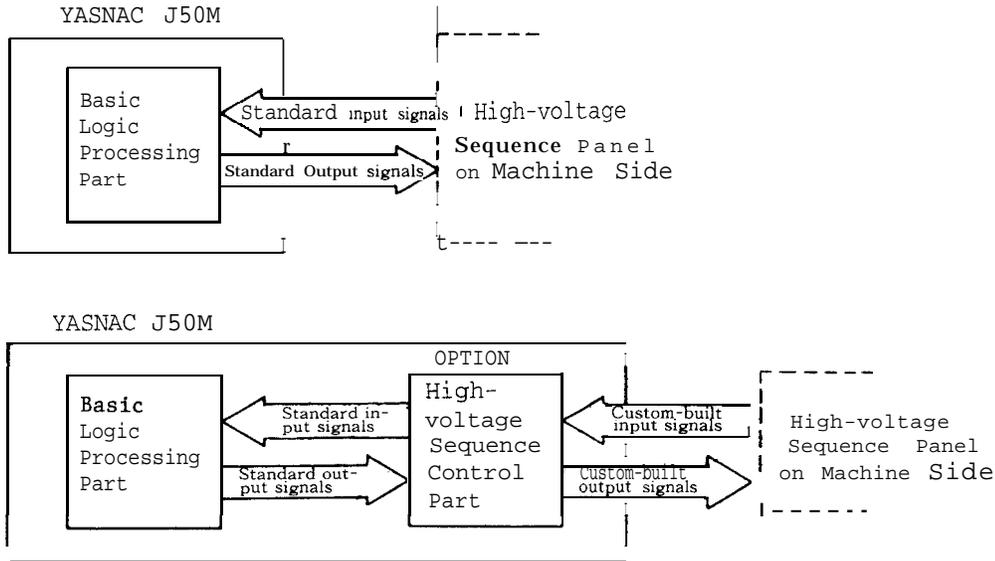


Fig. 7.3

7.6.4 IMPORTANT ALARM CODES

Alarm codes shown below are important ones for the system configuration. If these alarms occur, immediately notify your YASKAWA representative.

(1) Alarms by self-diagnosis between CPUS

325: SERVO CPU ERROR.... This occurs in the event of a MOTION CPU error.

329: PC CPU ERROR.... This occurs in the event of a PC CPU error.

381: AXIS ERROR (X) This occurs when READY from X-axis has not been received.

382: AXIS ERROR (Y) This occurs when READY from Y-axis has not been received.

383: AXIS ERROR (Z) This occurs when READY from Z-axis has not been received.

384: AXIS ERROR (4) This occurs when READY from 4th axis has not been received.

(2) Alarms that occur when the hardware related to optional items is changed:

327: UNFINISHED PROM GEN. . . . This occurs if no initialization has been made after changing the memory length of part program.

328: UNSUITABLE MM21 This occurs when the memory length of part program does not match the parameter setting.

373: SR-20 UNSUITABLE. . . . This occurs when the added hardware does not match the parameter setting.

To display input /output signals, proceed as follows.

1. Depress the DGN key.

A page containing the diagnostic number specified previously occurs on the CRT screen. The input /output signals are shown in "1," "0" and hexadecimal digit.

"1": contact close
"0": contact open

2. Key-in the diagnostic number to be displayed.
3. Depress the cursor or key to page the keyed-in diagnostic number on the screen.

By depressing the cursor key,

a cursor on the screen moves to the page containing the next diagnostic number.

"1": Contact is closed.
"0": Contact is open.

Data of each line are displayed in hexadecimal notation are displayed at the right side of the screen.

By depressing the cursor key, the cursor moves to the previous line. When up to the most upper line, the previous page is displayed.

By depressing the page key, the next page appears on the screen.

By depressing the page key, the previous page appears.

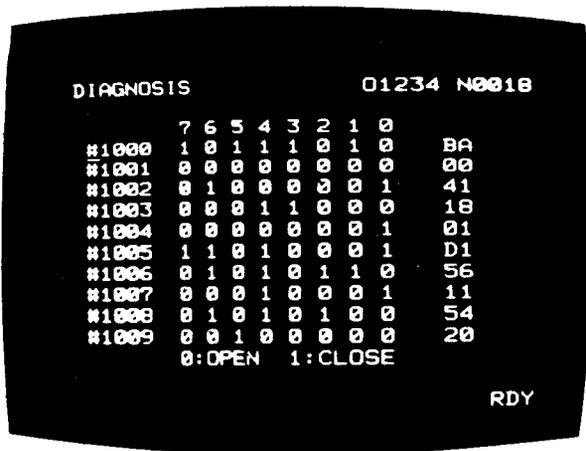


Fig. 7.4 Example of Input/Output Signal Display

7.6.5 BEFORE MAINTENANCE CALL

If the cause of trouble cannot be found by using alarm codes or I/O signals (described in 7.6.1 to 7.6.3), or corrective action for the trouble cannot be taken, record the following items, and notify the company as soon as possible.

Alarm codes and the accompanying data with them.

The types and characteristics of the troubles.

The operational procedures just before the trouble occurred and number of applied NC tape.

Whether the trouble recurs each time, the operation is repeated after depressing the RESET key.

Date and time when the trouble occurred.

Name of the discoverer of the trouble and the operator.

If trouble occurs, keep the control in the same condition until it can be checked by your Yaskawa representative. If the situation permits, avoid turning off control power, or depressing POWER OFF button.

APPENDIX-1 LIST OF SETTING NUMBERS

Setting numbers are classified in the following three groups:

Table Al. 1

Setting number	Group
#6000 to 6004	Setting by bit
#6200 to 6219	Setting by word
#6500 to 6599	Setting by double words
#6700 to 6771	Setting by double words
#6772 to 6899	Setting by double words

Appendix table 1-2 lists setting numbers and their functions.

SETTING NUMBERS AND THEIR CONTENTS

#6000	D7		D5		13	D2	D1	D0
-------	----	--	----	--	----	----	----	----

- 137** 1: Sets punch-out code to ISO
0: Sets punch-out code to EIA
- D5** 1: Sets playback ON
0: Sets playback ON /OFF with an external switch
- D3, D2, D1, D0**
Selects whether to make the mirror image of the 4th axis, Z-axis, Y-axis and X-axis effective in successive order or to make them external switches.
1: Mirror image effective
0: External switch

#6001	D7			D4	D3	D2	D1	D0
-------	----	--	--	----	----	----	----	----

- D7** 1: Emits operating panel buzzer sound
0: Does not emit operating panel buzzer sound
- D4** Z-axis canceling switch
1: ON
0: OFF
- D 3** Manual absolute switch
1: ON
0: OFF
- D2** Editing lock switch
1: ON
0: OFF

- D1** 1: Checks the 2nd prohibited area of stored stroke limit
0: Does not check the 2nd prohibited area of stored stroke limit
- D0** 1: Specifies input units as inch inputs
0: Specifies input units as metric inputs

#6002	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

- D7** Auxiliary function lock switch
1: ON
0: OFF
- D6** Machine lock switch
1: ON
0: OFF
- D5** Optional stop switch
1: ON
0: OFF
- D4** Dry run switch
1: ON
0: OFF.
- D3** Optional block skipping switch
1: ON
0: OFF
- D2** Display lock switch
1: ON
0: OFF
- D1** Start lock switch
1: ON
0: OFF
- D0** Single block switch
1: ON
0: OFF

#6003		D5	"4	"3		"1	D0
-------	--	----	----	----	--	----	----

- D5, D4:** Selects the output device of the data input interface
- D3** When outputting EOB in I SO code;
0: Output with LF.
1: Output with CRLF.
- D1, D0:** Selects the input device of the data input interface

Setting Code	Input Device	output Device	Parameter for Setting Baud Rate
01	RS232C	RS232C	#6026 (#6028) D3 to D0

#6004	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

- D7 The group G02 G code upon reset is;
O: G17.
1: The G code commanded immediately before is retained.
- D6 1: Cancels life data when executing command G 122
0: Does not cancel life data when executing command G 122
- D5 The group G03 G code upon reset ;
O : Follows #6005 DO.
1: The G code commanded immediately before is retained.
- D4 1: Programs of program numbers 8000 through 8999 cannot be registered, erased or edited.
0: Normal registration, erasure and editing can be performed.
- D3 1: Programs with program numbers 8000 through 8999 will not be displayed.
0: Programs with program numbers 8000 through 8999 will be displayed.
- D2 O-number automatic renaming function is ;
1: Validated.
O : Invalidated.
- D1 1: Single blocks become effective in relation to an operation command from the user program.
0: Single blocks become ineffective in relation to operation commands or control commands from the user program.
- D0 1: The next block will be executed if there is no skip signal input until shifting of the G31 block ends.
O: Causes alarm "087"

#6200	Break point '1
-------	----------------

#6201	Break point -2
-------	----------------

Break point sequence setting.
Setting range : 1 to 9999.
Will not stop at 0.

#6202	
-------	--

Specifies the program No. of a macro program to be called by B code.
Setting range : 1 to 9999

#6204	
-------	--

Group number specification during tool change and reset.
Setting range : 1 to 256

#6207	
-------	--

Program No. setting
Specifies program number of tape when inputting tape without a program number.
Setting range : 1 to 9999

Note: If O is set, input error may cause except when the number on the upper right of CRT is "O. "

#6210	
-------	--

Dwell time setting when executing G76 and G77
Setting : "1" = 1 ms
Setting range : 0 to 32767

#6211	
-------	--

Amount of δ setting when executing G73
Setting : "1" = 0.001 mm or 0.0001 inch
Setting range : 0 to 32767

#6212	
-------	--

Default dwell time setting for G73/G83
Setting : "1" = 1 ms
Setting range : 0 to 32767

#6213	
-------	--

Shift amount " δ " setting when executing G83
Setting : "1" = 0.001 mm or 0.0001 inch
Setting range : 0 to 32767

#6214	
-------	--

Automatic corner override A1 point
(n% of tool radius)
Setting : "1" = 1%

#6215	
-------	--

Automatic corner override A2 point
(n% of tool radius)
Setting : "1" = 1%

#6216	
-------	--

Automatic corner override F1
(n% of command feeding)
Setting : "1" = 1%

SETTING NUMBERS AND THEIR CONTENTS (Cont'd)

#6217

Automatic corner override F2
(n% of command feeding)
Setting : "1" = 1%

#6218

Automatic corner override F3
(n% of command feeding)
Setting : "1" = 1%

#6219

System No. switch setting
Setting range: 0, 1, 4

#6500

#6501

Scaling multiple setting
Multiple = #6500/#6501
Setting : "0" = Scale multiple = 1

#6506

Angle setting during execution of commands
G76 and G77
Setting : "1" = 0.001 deg

#6507

Tool length measuring bias setting
Setting : "1" = Least input increment

#6508

Tool length measuring bottom level setting
Setting : "1" = Least input increment

#6510 X-axis

#6511 Y-axis

#6512 Z-axis

Sets stored stroke limit of the X-, Y- and Z-axis
and the boundary value in the positive
direction of the 2nd prohibited area.
Setting : "1" = 1 pulse
Setting range : 0 to ±99999999

#6513 X-axis

#6514 Y-axis

#6515 Z-axis

Sets stored stroke limit of the X-, Y- and Z-axis
in successive order and also the negative
direction boundary value of the 2nd prohibited
area.

Setting : "1" = 1 pulse
Setting range : 0 to *99999999

#6516 X-axis

#6517 Y-axis

#6518 Z-axis

#6519 4th axis

Work coordinate system setting G54 (J1)
Setting : "1" = Least input increment
Setting range : 0 to *99999999

#6521

Rotary angle setting G54 (J1)
Setting : "1" = 0.001 deg
* Setting is valid in C specification.

#6522 X-axis

#6523 Y-axis

#6524 Z-axis

#6525 4th axis

Work coordinate system setting G55 (J1)
Setting : "1" = Least input increment
Setting range : 0 to *99999999

#6527

Rotary angle setting G55 (J1)
Setting : "1" = 0.001 deg

#6528 X-axis

#6529 Y-axis

#6530 Z-axis

#6531 4th axis

Work coordinate system setting G56 (J1)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6533

Rotary angle setting G56 (J1)
Setting : "1" = 0.001 deg
* Setting is valid in C specification.

#6534 X-axis

#6535 Y-axis

#6536 Z-axis

#6537 4th axis

Work coordinate system setting G57 (J1)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6539

Rotary angle setting G57 (J1)
Setting : " 1" = 0.001 deg
* Setting is valid in C specification.

#6540 X-axis

#6541 Y-axis

#6542 Z-axis

#6543 4th axis

Work coordinate system setting G58 (J1)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6545

Rotary angle setting G58 (J1)
Setting : " 1" = 0.001 deg
* Setting is valid in C specification.

#6546 X-axis

#6547 Y-axis

#6548 Z-axis

#6549 4th axis

Work coordinate system setting G59 (J1)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6551

Rotary angle setting G59 (J1)
Setting : "1" = 0.001 deg
* Setting is valid in C specification.

#6552 X-axis

#6553 Y-axis

#6554 Z-axis

#6555 4th axis

Coordinate value at skip signal ON
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6561 F1

#6562 F2

#6563 F3

#6564 F4

#6565 F5

#6566 F6

#6567 F7

#6568 F8

#6569 F9

F1 -digit speed setting
Setting : " 1" = 0.1 mm or 0.01 inch
Setting range : 0 to 300000

#6580

#6581

#6582

SETTING NUMBERS AND THEIR CONTENTS (Cont'd)

#6583	
#6584	
#6585	
#6586	
#6587	
#6588	
#6589	
#6590	
#6591	
#6592	
#6593	
#6594	
#6595	
#6596	
#6597	
#6598	
#6599	

Set the M codes to call up the macro program.
(Maximum 20 M codes)

WORK COORDINATE SYSTEM SETTING
SPECIFICATIONS B

#6700	X-axis
#6701	Y-axis
#6702	Z-axis

Work coordinate system setting G54 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6703	X-axis
#6704	Y-axis
#6705	Z-axis

Work coordinate system setting G55 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6706	X-axis
#6707	Y-axis
#6708	Z-axis

Work coordinate system setting G56 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6709	X-axis
#6710	Y-axis
#6711	Z-axis

Work coordinate system setting G57 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6712	X-axis
#6713	Y-axis
#6714	Z-axis

Work coordinate system setting G58 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6715	X-axis
#6716	Y-axis
#6717	Z-axis

Work coordinate system setting G59 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6718	X-axis
#6719	Y-axis

#6720 Z-axis

Work coordinate system setting G54 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6721 X-axis

#6722 Y-axis

#6723 Z-axis

Work coordinate system setting G55 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6724 X-axis

#6725 Y-axis

#6726 Z-axis

Work coordinate system setting G56 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6727 X-axis

#6728 Y-axis

#6729 Z-axis

Work coordinate system setting G57 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6730 X-axis

#6731 Y-axis

#6732 Z-axis

Work coordinate system setting G58 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6733 X-axis

#6734 Y-axis

#6735 Z-axis

Work coordinate system setting G59 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6736 X-axis

#6737 Y-axis

#6738 Z-axis

Work coordinate system setting G54 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6739 X-axis

#6740 Y-axis

#6741 Z-axis

Work coordinate system setting G55 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6742 X-axis

#6743 Y-axis

#6744 Z-axis

Work coordinate system setting G56 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6745 X-axis

#6746 Y-axis

#6747 Z-axis

Work coordinate system setting G57 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6748 X-axis

#6749 Y-axis

#6750 Z-axis

Work coordinate system setting G58 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

WORK COORDINATE SYSTEM SETTING
SPECIFICATIONS B (Cent'd)

#6751 X-axis

#6752 Y-axis

#6753 Z-axis

Work coordinate system setting G59 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6754 X-axis

#6755 Y-axis

#6756 Z-axis

Work coordinate system setting G54 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6757 X-axis

#6758 Y-axis

#6759 Z-axis

Work coordinate system setting G55 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6760 X-axis

#6761 Y-axis

#6762 Z-axis

Work coordinate system setting G56 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6763 X-axis

#6764 Y-axis

#6765 Z-axis

Work coordinate system setting G57 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6766 X-axis

#6767 Y-axis

#6768 Z-axis

Work coordinate system setting G58 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6769 X-axis

#6770 Y-axis

#6771 Z-axis

Work coordinate system setting G59 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

Note : #6772 to #6899 is unused area. Do not use this area.

WORK COORDINATE SYSTEM SETTING
SPECIFICATIONS C

#6700 X-axis

#6701 Y-axis

#6702 Z-axis

#6703 4th axis

Work coordinate system setting G54 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6705

Rotary angle setting G54 (J2)
Setting: "1" = 0.001 deg

#6706 X-axis

#6707 Y-axis

#6708 Z-axis

#6709 4th axis

Work coordinate system setting G55 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6711

Rotary angle setting G55 (J2)
Setting: "1" = 0.001 deg

#6712 X-axis

#6713 Y-axis

#6714 Z-axis

#6715 4th axis

Work coordinate system setting G56 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6717

Rotary angle setting G56 (J2)
Setting: "1" = 0.001 deg

#6718 X-axis

#6719 Y-axis

#6720 Z-axis

#6721 4th axis

Work coordinate system setting G57 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6723

Rotary angle setting G57 (J2)
Setting: "1" = 0.001 deg

#6724 X-axis

#6725 Y-axis

#6726 Z-axis

#6727 4th axis

Work coordinate system setting G58 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6729

Rotary angle setting G58 (J2)
Setting: "1" = 0.001 deg

#6730 X-axis

#6731 Y-axis

#6732 Z-axis

#6733 4th axis

Work coordinate system setting G59 (J2)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6735

Rotary angle setting G59 (J2)
Setting: "1" = 0.001 deg

#6736 X-axis

#6737 Y-axis

#6738 Z-axis

#6739 4th axis

Work coordinate system setting G54 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6741

Rotary angle setting G 54 (J3)
Setting: "1" = 0.001 deg

#6742 X-axis

#6743 Y-axis

#6744 Z-axis

#6745 4th axis

Work coordinate system setting G55 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

WORK COORDINATE SYSTEM SETTING
SPECIFICATIONS C (Cent'd)

#6747

Rotary angle setting G55 (J3)
Setting: "1" = 0.001 deg

#6748

#6749

#6750

#6751

Work coordinate system setting G56 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6753

Rotary angle setting G56 (J3)
Setting: "1" = 0.001 deg

#6754

#6755

#6756

#6757

Work coordinate system setting G57 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6759

Rotary angle setting G57 (J3)
Setting: "1" = 0,001 deg

#6760

#6761

#6762

#6763

Work coordinate system setting G58 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6765

Rotary angle setting G58 (J3)
Setting: "1" = 0.001 deg

#6766

#6767

#6768

#6769

Work coordinate system setting G59 (J3)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6771

Rotary angle setting G59 (J3)
Setting: "1" = 0.001 deg

#6772

#6773

#6774

#6775

Work coordinate system setting G54 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6777

Rotary angle setting G54 (J4)
Setting: "1" = 0.001 deg

#6778

#6779

#6780

#6781

Work coordinate system setting G55 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6783

Rotary angle setting G55 (J4)
Setting: "1" = 0.001 deg

#6784 X-axis

#6785 Y-axis

#6786 Z-axis

#6787 4th axis

Work coordinate system setting G56 (J4)
Setting : "1" = Least input increment
Setting range : 0 to *99999999

#6789

Rotary angle setting G 56 (J4)
Setting: "1" = 0.001 deg

#6790 X-axis

#6791 Y-axis

#6792 Z-axis

#6793 4th axis

Work coordinate system setting G57 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6795

Rotary angle setting G 57 (J4)
Setting: "1" = 0.001 deg

#6796 X-axis

#6797 Y-axis

#6798 Z-axis

#6799 4th axis

Work coordinate system setting G58 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6801

Rotary angle setting G58 (J4)
Setting: "1" = 0.001 deg

#6802 X-axis

#6803 Y-axis

#6804 Z-axis

#6805 4th axis

Work coordinate system setting G59 (J4)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6807

Rotary angle setting G59 (J4)
Setting: "1" = 0.001 deg

#6808 X-axis

#6809 Y-axis

#6810 Z-axis

#6811 4th axis

Work coordinate system setting G54 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6813

Rotary angle setting G54 (J5)
Setting: "1" = 0.001 deg

#6814 X-axis

#6815 Y-axis

#6816 Z-axis

#6817 4th axis

Work coordinate system setting G55 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

WORK COORDINATE SYSTEM SETTING
SPECIFICATIONS C (Cent'd)

#6819

Rotary angle setting G55 (J5)
Setting: "1" = 0.001 deg

#6820 X-axis

#6821 Y-axis

#6822 Z-axis

#6823 4th axis

Work coordinate system setting G56 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6825

Rotary angle setting G56 (J5)
Setting: "1" = 0.001 deg

#6826 X-axis

#6827 Y-axis

#6828 Z-axis

#6829 4th axis

Work coordinate system setting G57 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6831

Rotary angle setting G57 (J5)
Setting: "1" = 0.001 deg

#6832 X-axis

#6833 Y-axis

#6834 Z-axis

#6835 4th axis

Work coordinate system setting G58 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to ±99999999

#6837

Rotary angle setting G58 (J5)
Setting: "1" = 0.001 deg

#6838 X-axis

#6839 Y-axis

#6840 Z-axis

#6841 4th axis

Work coordinate system setting G59 (J5)
Setting : " 1" = Least input increment
Setting range : 0 to *99999999

#6843

Rotary angle setting G59 (J5)
Setting: "1" = 0.001 deg

APPENDIX-2 LIST OF PARAMETER NUMBERS

Parameter numbers are classified in the following five groups:

Table A2. 1

Parameter number	Group
#6005 to #6099	Setting by bit
#6100 to #6199	Setting by byte
#6220 to #6499	Setting by word
#6600 to #6699	Setting by double words
#7000 to #7099	Setting by byte for sequencer

Optimum data of parameters have been set according to machine performance and applications. For any modification of parameter data, consult the machine tool builder.

Data pertaining to parameters #6033 to #6049 must not be modified, for they have been incorporated as part of the system.

PARAMETER NUMBERS AND THEIR CONTENTS

#6005	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

D7 1: Stores the H code at G28 for reset.
 0: **Sets H code to H00 at G28 for reset.**

D6 1: Stores the G code in the 01 group for reset.
 0: Sets the G code in the 01 group to G00 for reset.

D5 1: Allows the current value display (universal) to be preset by the coordinate system setting command G92.
 At this time, the ORG key has coordinate system ZERO setup function only in manual mode.
 0: Keeps the current value display (universal) from being preset by the coordinate system setting command G92.
 At this time, the ORG key does not have coordinate system ZERO setup function while ZERO clear is valid in all modes.

D4, D3: Status of G codes at power on.

D4	D3	Initial status
1	0	Sets the G code in the 08 group to G44 at power on.
0	1	Sets the G code in the 08 group to G43 at power on.
0	0	Sets the G code in the 08 group to G49 at power on.

D2 1: M95 mode (mirror image ON) at power on or reset
 0: M94 mode (mirror image OFF) at power on or reset

D1 1: Does not apply mirror image in the G28 interim point in the G95 mode at power on or reset.
 0: Applies mirror image in the G28 interim point in the G95 mode at power on or reset.

D0 1: Sets the G code in the 05 group to G91 at power on.
 0: Sets the G code in the 05 group to G90 at power on.

#6006	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

D7, D6 Signs of S5-digit analog (SDA) output

D7	D6	Sign	
\	1	Minus	Minus (1)
1	0	Plus	Plus (1)
0	1	Minus	Minus (2)
0	0	Plus	Plus (2)
		at M03 output	at M04 output

Note: The functions of (1) and (2) are the same.

D5 1: Sets the least input increment x10
 0: Does not set the least input increment to x10

D4 1: Checks to see if the spindle speed match signal (SAGR) is off upon transition from a rapid traverse block to a cutting feed block.
 0: Provides no check on the spindle speed match signal (SAGR).

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

D3 1: Enables the internal toggle switches.
0: Disables the internal toggle switches.
See #6001, #6002

D2 1: Enables dry run in response to the rapid traverse command.
0: Disables dry run in response to the rapid traverse command.

D1 1: Causes an alarm ("001-004") upon cycle start when reference point return is not made after turning the power on.
0: Causes no alarm.

Note: Set "1" when pitch error compensation or stored stroke limit is provided.

When this parameter is set to "1," Cycle Start is not activated unless reference point return of all the axes have not be completed. Setting it to "0" determines whether reference point return is required or not for each axis.
Refer to parameters #6011 DO to D3.

D O 1: Enables automatic coordinate system setting
2: Disables automatic coordinate system setting.

Refer to #6630 to #6639.

#6007	D7	D6	D5	D4	D3	D2	D1	DO
-------	----	----	----	----	----	----	----	----

D7 1: Disables start interlock after edit.
0: Enables start interlock after edit.

D6 1: Employs the newly entered tool compensation value in place of the old value.
0: Adds the newly entered tool compensation value to the soterd value to establish another offset.

D5 1: Enables error detect ON mode at dwell.
0: Disables error detect On mode at dwell.

D4 1: Provides output during rewinding.
0: Provides no output during rewinding.

D3 1: Sets the least increment to 0.0001 in.
0: Sets the least increment to 0.001 mm.

D2 1: Makes the spindle override 100% during tapping by G84.
0: Does not make the spindle override 100% during tapping by G84.

D1 1: Establishes the prohibited area of the 3rd, 4th, and 5th stored stroke limit outside the boundary.

0: Establishes the prohibited area of the 3rd, 4th, and 5th stored stroke limit inside the boundary.

DO 1: Establishes the 2nd prohibited area of stored stroke limit outside the boundary.

0: Establishes the 2nd prohibited area of stored stroke limit inside the boundary.

#6008	D6	D5	D4	D3	D2	D1	DO
-------	----	----	----	----	----	----	----

D6 1: Rewinds at M30 command in Tape operation with YASNAC standard tape reader (1st RS-232C)

0: Does not rewind at M30 command in Tape operation with YASNAC standard tape reader (1st RS-232C)

D5 1: 0 number is changeable with ALT operation.

0: 0 number is not changeable with ALT operation.

D4 1: Feed hold block stop ineffective when tapping with G84

0: Feed hold block stop effective when tapping with G84

D3 1: Manual absolute effective in the G 91 mode.

0: Manual absolute ineffective in the G91 mode.

D2 1: Changes approach speed to jog speed when restarting program.

0: Changes approach speed to fast feed when restarting program.

D1 1: Does not clear the common variables of #100 through #149.

0: Clears the common variables of #100 through #149.

DO 1: Requires Edit interlock for Manual Zero Setup.

0: Does not require Edit interlock for Manual Zero Setup

Note: Some controls are not provided with the parameter #6008.

#6009	D7	D6	D5	D4	D3	D2		
-------	----	----	----	----	----	----	--	--

D7 1: Up to 256 characters can be set in 1 tape read block.

0: Up to 128 characters can be set in 1 tape read block.

D6 1: Feed is not provided when output from the NC.
 0: Feed is provided when output from the NC.

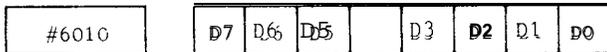
D5 1: Space code does not exist between programs at all program outputs (O-9999 OUT).
 0: Space code exists between programs at all program outputs (O-9999 OUT).

D4 1: RST is output when both RWD and EOP are input in M02/M30, or when only EOP is input.
 0: RST is not output when both RWD and EOP are input in M02/M30, or when only EOP is input.

D3 1: G68 is effective by the coordinate rotation command.
 0: G54 is effective by the coordinate rotation command.

* Valid only when the work coordinate system C (option) is set to "1".

D2 1: FH LT lights and MST is stored when the mode is changed from program execution mode to manual mode.
 0: FH LT does not light and MST is forced to FIN when the mode is changed from program execution mode to manual mode.



D7, D6, D5

Specifies whether or not the manual pulse is effective on the Z-, Y-, and X-axis, respectively.

1: Effective
 0: Ineffective
 (Automatic handle Offset)

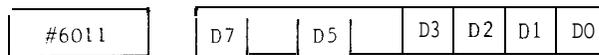
D3, D2, D1, D0

Specifies the direction of reference point return on the 4th-, Z-, Y- and X-axes, respectively.

1: Minus direction
 0: Plus direction

Note : The specification is effective for an axis with #6016 at "1".

Backlash is applied first when the tool is moved in the reverse direction of that specified by D0 to D3 after turning the power on.



D7 1: Convert the ":" into "O" upon tape in/out.
 0: Do not convert the ":" into "O" upon tape in/out.

D5

Specifies whether or not the manual pulse is effective on the 4th axis.

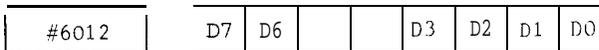
1: Effective
 0: Ineffective

D3, D2, D1, D0

Command reference-point interlock for 4th, Z-, Y-, and X-axis respectively, before cycle start.

1: Interlock enable
 0: Interlock disable

Note : The specification is effective when #6006 D1 is "0".



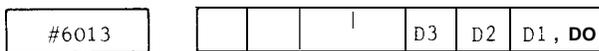
D7 } JSD data transmission method selection
 D6 }

D7	D6	Transmission Method
0	0	No procedure
0	1	DC code control
1	0	RS-CS control
1	1	No procedure

D3, D2, D1, D0

Specifies whether or not the plus-direction external deceleration signal is effective on the 4th-, Z-, Y- and X-axes, respectively.

1: Makes the plus-direction external deceleration signal effective.
 0: Makes the plus-direction external deceleration signal ineffective.



D3, D2, D1, D0

Specifies whether or not the minus-direction external deceleration signal is effective on the 4th-, Z-, Y- and X-axes, respectively.

1: Makes the minus direction external deceleration signal effective.
 0: Makes the minus direction external deceleration signal ineffective.

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)



- D7 1: PC CPU error occurs.
0: PC CPU error does not occur.
- D6 1: Motion CPU error occurs.
0: Motion CPU error does not occur.

D3, D2, D1, D0

Specifies the direction of the G60 uni-directional approach upper limit on the 4th-, Z-, Y- and X-axes, respectively.

- 1: Minus direction
0: Plus direction

Note : The approach upper limit is set with #6436 to #6439.



D3, D2, D1, D0

Specifies whether or not the automatic coordinate system setting is effective on the 4th-, Z-, Y- and X-axes, respectively.

- 1: Effective
0: Ineffective

Note : The automatic coordinate system is established with the following parameters :

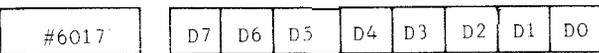
- Metric system ; #6636 to #6439
Inch system ; #6630 to #6633



D3, D2, D1, D0

Specifies whether or not reference point return is effective on the 4th-, Z-, Y- and X-axes, respectively.

- 1: Reference point return effective.
0: Reference point return ineffective.



D7 - D0

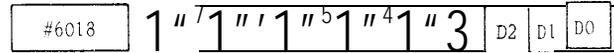
Specifies whether or not a hole is to be made on channels 8-1, respectively, in a code corresponding to symbol "#" (used with user macro) in the EIA code.

- 1: Hole
0: No hole

Example:
D7 - D0 = 01001001

Notes :

- The code with holes on channels 7, 4 and 1 is considered equivalent to symbol "#" in the EIA code. No code for use by the unit can be set.
- The specification of D7-D0 = 00000000 assumes that symbol "#" is not used in the EIA code.



- D7 1: Provides dwell at hole bottom in the canned cycles of G76 and G77.
0 : Does not provide dwell at hole bottom in the canned cycles of G76 and G77.

Note : The dwell time is set with #6210.

- D6 1: Establishes M03 for G74 and M04 for G84 as the M code for output at hole bottom in the canned cycles of G74 and G84.
0: Reverses the M code in effect before the canned cycles for output at hole bottom in the canned cycle of G74 and G84.

Before canned cycle	Hole bottom
M03	M04
M04	M03

Note: This specification is effective when D4 = 0 in #6018.

- D5 1: Rotates the spindle forward and in reverse, outputting M05 or SSP at hole bottom in the canned cycles of G74 and G84.

0: Rotates the spindle forward and in reverse, not outputting M05 at hole bottom in the canned cycles of G74 and G84.

Note : This specification is effective when D4 = 0 in #6018.

- D4 1: Outputs a read-only signal (SSP, SRV, OSS) in the canned cycles.
0: Outputs the M code in the canned cycles

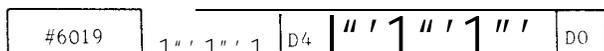
- D3 1: Does not wait for in-position at GOO and G01 to GOO commands.

0: Waits for in-position at GOO and GO 1 to GOO commands.

- D2 1: Needs power off at occurrence of an alarm related to reference point return.
 0: Does not need power off at occurrence of an alarm related to reference point return.

- D1 1: Outputs the FMF signal twice in a canned cycle.
 0: Outputs the FMF signal once in a canned cycle.

- Do 1: Outputs the external operation signal EF at the end of positioning by G81 (G81 being external operation function)
 0: Does not output the external operation signal EF at the end of positioning by G81 (G81 being canned cycle)



- D7 Stops the tape at the initial “ %” during tape operation.

- 1: Effective
 0: Ineffective

- D6 1: When parameter outputs, only #7000 to #7999 is output.
 0: When parameter outputs, full range is output.

- D4 1: Employs the feedrate set in parameter #6232 for the skip function command (G31).
 0: Employs the F code command as the feedrate for the skip function command (G31).

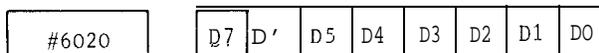
- D3 1: Reset OFF at Emergency Stop
 0: Reset ON at Emergency Stop

- D2 D1 Specifies the tool shift direction in the canned cycles of G76 and G77 (effective when #6019 DO= 0).

D2	D1	Shift direction
1	1	-Y
1	0	+Y
0	1	-x
0	0	+x

- D0 1: Allows the tool shift direction in the canned cycles of G76 and G77 to be specified in setting #6506 (specifiable in the +X direction in increments of 0.001; the shift being cutting feed).

- 0: Allows the tool shift direction in the canned cycles of G76 and G77 to be specified in D2 and D1 of parameter #6019 (specifiable only axially)



- D7 1: Assigns selection of group specification numbers to an external signal when the tool change skip signal is ON during life control.
 0: Assigns selection of group specification numbers to the currently specified group when the tool change skip signal is ON during life control.

- D6 1: Assigns selection of the T command group to the T command immediately before during the M06 command in life control.

- 0: Assigns selection of the T command to the latest T command during the M06 command in life control.

- D5 1: Assigns group number specification to an external signal during tool change reset in life control.
 0: Assigns group number specification to setting #6204 during tool change reset in life control.

- D4 1: Counts with M02/M30 when count is the type of life control.
 0: Counts with T9999L△△△ when count is the type of life control.

- D3 1: Sets F14 (in/rev) for the feed per minute in the inch system.

- 0: Sets F13 (in/rev) for the feed per minute in the inch system.

- D2 1: Sets F23 (mm/rev) for the feed per minute in the metric system.

- 0: Sets F22 (mm/rev) for the feed per minute in the metric system.

- D1 1: Sets F32 (in/min.) for the feed per minute in the inch system.

- 0: Sets F31 (in/min.) for the feed per minute in the inch system.

- D0 1: Sets F51 (mm/min.) for the feed per minute in the metric system

- 0: Sets F50 (mm/min.) for the feed per minute in the metric system.

PARAMETER NUMBERS AND THEIR CONTENTS
(Cent'd)



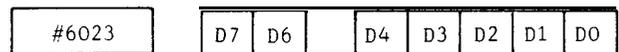
- D7 1: Makes editing interlock 09000 through 09999 effective and prohibit the external I/O operation (tape in/out).
0: Makes editing interlock 09000 through 09999 ineffective.
- D6 1: Erases and stores the previous O number when loading a tape provided with an O.
0: ALREADY IN will be displayed if the same O number exists when a tape with an O is loaded.
- D5 1: Executed ON/OFF control of RTS signal during RS-232C operation until loading ends.
0: Sets RTS signal to ON state during RS-232C operation until loading ends.
- D4 1: Refers to DR (Data set ready) during RS 232C operation.
0: Does not refer to DR (Data set ready) during RS 232C operation.
- D3 1: Outputs 00 with O through 9999 OUT.
0: Does not output 00 with O through 9999 OUT.
- D2 1: Displays the O number when the power supply is turned on/off.
0: Displays 00 when the power is turned on.
- D1 1: Employs the value following address O or N as the program number (speciable in one block) .
0: Employs the value following address O as the program number.
- D0 1: Considers M02, M30 and M99 as the program end when machining data is stored into memory.
EOB code is provided for the top of the O number at punch-out (O-9999 OUT).
0: Does not consider M02, M30 and M99 as the program end when machining data is stored into memory.
EOB code is not provided for the top of the O number at punch-out (O-9999 OUT).



- D7 1: Sets input unit of parameter setting to 10 rein-1 when specifying the S5 digits.

0: Sets input unit of parameter setting to 1 rein-1 when specifying the S5 digits.

- D6 1: Enables binary search of EDIT /MEM. Character search is also possible if the NEXT key is pressed.
0: Enables search of EDIT /MEM character only.
- D5 1: Makes editing display of 090000 through 09999 Interlock effective.
0: Makes editing display 09000 through 09999 Interlock ineffective.
- D4 1: Makes intermediate POT display effective.
0: Makes intermediate POT display ineffective.
- D3 1: Makes tool POT word display effective.
0: Makes tool POT byte display effective.
- D2 1: Enables writing word
0: Enables writing bytes in keep memory.
- D1 1: No parity bit when punching out an ISO tape
0: Parity bit exists when punching out an ISO tape
- D0 1: Enables read of ISO tape even without parity.
0: Disables read of ISO tape without parity.



D7 When 4th axis is rotary axis, stored stroke limit is

- 1: Effective
0: Ineffective

D6 Ignorance of additional axis (4NG)

- 1: ON
0: OFF

Note: When this parameter setting is changed, turn off the power. Setting the parameter except to "0," display shows additional axis is provided. However, this additional axis cannot be effective without changing hardware (SR50-**).

D4-D0

Set the address for punch-out and CRT display on the additional axis.

Address	D4	D3	D2	D1	D0
A	o	0	0	0	1
B	o	0	0	1	0
c	o	0	0	1	1
u	1	0	1	0	1
v	1	0	1	1	0
w	1	0	1	1	1

#6025									
-------	--	--	--	--	--	--	--	--	--

Parameter #6025 cannot be written at any time.

#6026			D5	D4	D3	D2	D1	D0
-------	--	--	----	----	----	----	----	----

D5 1: Does not allow the control code (DC1-DC4) to be used on the input device.

o: Allows the control code (DC1-DC4) to be used on the input device.

D4 1: Employs 2 stop bits on the input device.

o: Employs 1 stop bit on the input device.

D3 - D0

Baud rate setting

Baud rate	D3	D2	D1	D0
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1

Note : #6026 provides the setting on input device 1 (s10-1).

#6028			D5	D4	D3	D2	D1	D0
-------	--	--	----	----	----	----	----	----

D5 1: Does not use control codes (DC1-DC4).

0: Uses control codes (DC 1-DC4)

D4 1: Employs 2 stop bits on the output device,

0 : Employs 1 stop bit on the output device.

D3 to D0 Baud rate setting

Baud rate setting

Baud Rate	D3	D2	D1	D0
50	0	0	0	0
100	o	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	1	0	0
4800	1	0	0	1

Note : #6028 is for the setting on output device 1 (s10-1).

#6050	D7	D6	D5					DO
-------	----	----	----	--	--	--	--	----

#6051	D7	D6	D5					DO
-------	----	----	----	--	--	--	--	----

#6052	D7	D6	D5					DO
-------	----	----	----	--	--	--	--	----

#6053	D7	D6	D5					DO
-------	----	----	----	--	--	--	--	----

#6055	D7	D6	D5				D1	DO
-------	----	----	----	--	--	--	----	----

Shown in the order of X-axis, Y-axis, Z-axis, 4th axis and spindle.

D7, D6

Be sure to set the assignment of servo control form.

CNTL 2: 0

CNTL 1: 0

Spindle only: 0, 1

D5 1: No servo control is performed.

o: Servo control is performed.

D1 1: Perform spindle indexing when entering the solid tap mode by G93.

o: Do not perform spindle indexing when entering the solid tap mode by G93.

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

DO 1: Operation confirmation of each unit is not performed during system operation.
0: Operation confirmation of each unit is performed during system operation.

Notes :

1. Normal setting is "0".
2. When these parameters are changed, be sure to turn the power off and on.

#6056	D7	D6		D3	D2	D1	DO
-------	----	----	--	----	----	----	----

X-axis

D7, D6

X-axis Circular Projection Compensation

D7	D6	Compensation
0	0	None
1	0	Provided

D3 to D0

	D3	D2	D1	D0
x 1	0	0	0	1
x 2	0	0	1	0
x 4	0	1	0	0
x 8	1	0	0	0

Note : Be sure to turn the power on and off after changing the parameter.

#6057	D7	D6		D3	D2	D1	DO
-------	----	----	--	----	----	----	----

Y-axis

D7, D6

Y-axis Circular Projection Compensation

D7	D6	Compensation
0	0	None
1	0	Provided

D3 to D0

Set the magnification of the feedback pulse input to the NC unit.

	D3	D2	D1	D0
x 1	0	0	0	1
x 2	0	0	1	0
x 4	0	1	0	0
x 8	1	0	0	0

Note : Be sure to turn the power on and off after changing the parameter.

#6058	D7	D6		D3	D2	D1	DO
-------	----	----	--	----	----	----	----

Z-axis

D7, D6

Z-axis Circular Projection Compensation

D7	D6	Compensation
0	0	None
1	0	Provided

D3 to D0

Set the magnification of the feedback pulse input to the NC unit.

	D3	D2	D1	D0
x 1	0	0	0	1
x 2	0	0	1	0
x 4	0	1	0	0
x 8	1	0	0	0

Note : Be sure to turn the power on and off after changing the parameter.

#6059				D3	D2	D1	DO
-------	--	--	--	----	----	----	----

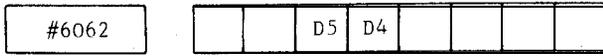
#6061				D3	D2	D1	DO
-------	--	--	--	----	----	----	----

4th axis spindle

D3 to D0 : Set the magnification of the feedback pulse input to the NC unit.

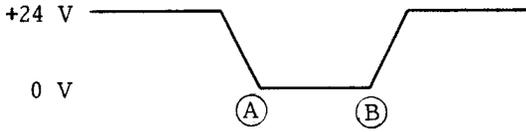
	D3	D2	D1	D0
x1	0	0	0	1
x2	0	0	1	0
x4	0	1	0	0
x8	1	0	0	0

Note: Be sure to turn the power on and off after changing the parameter.



D5 : Sets the signal state of processing start of "PINT" input for pint function.

D4 : This sets the signal state of processing start of "skip" input for skip function.

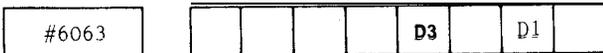


(A) : When this parameter is set to "0," the processing starts at the time when the change of 24V → 0V occurs.

(B) : When this parameter is set to "1," the processing starts at the time when the change of 0V → 24V occurs.

Notes :

1. Be sure to turn the power on and off after changing the parameter.
2. Be sure to set "0" except in D4 and D5.



D3 : This determines the ENABLE/DISABLE of control circuit of "PINT" input for pint function.

D1 : This determines the ENABLE/DISABLE of control circuit of "SKIP" input for skip function.

Notes :

1. Be sure to set "1" when using the skip function and program interruption function.
2. Be sure to turn the power on and off after changing the parameter to set "0" except in D1 and D3.



D3, D2, D1, D0

Shown in the order of 4th-axis, Z-axis, Y-axis and X-axis.

- 1: Follow-up processing is performed during servo-off input.
- 0: Follow-up processing is not performed during servo-off input.



D7 1: Spindle PG is attached to the motor (indicate the PG mount position).

- 0: The spindle PG is directly linked to the spindle.

Note : Be sure to turn the power on and off after setting this parameter.

D6 1: New radius compensation method

0: Old radius compensation method

D5 1: Type A compensation method

0: Type B compensation method

D4 1: Changes type A compensation method offset direction.

0: Changes type B compensation method offset direction.

D3 1: Changes offset value at the starting point.

0: Changes offset value at the end point.

Note : D5 to D3 are valid only when #6065 = 1.

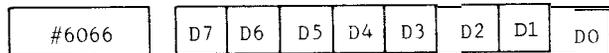
D1 1: The synchronous error peak value of the spindle and the z-axis during solid tap is displayed.

0: The synchronous error peak value of the spindle and the Z-axis during solid tap is not displayed.

(The plus peak value is displayed on the X-axis error pulse display section, and the minus peak value is displayed on the Z-axis error pulse display section).

DO 1: Do not wait for PSET during the tapping cycle by the solid tap function.

0: Waits for PSET during the tapping cycle by the solid tap function.



D7 1: JOG speed of 4th axis is 1/10 of basic 3 axes.

0: JOG speed of 4th axis is not 1/10 of 3 axes.

D6 1: At power on, M97.

0: At power on, M96.

D5 1: H/D changing 150/150 sets.

0: H/D changing 100/200 sets.

D4 1: With interference check function

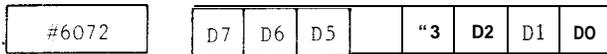
0: Without interference check function

D3 1: Cross point calculation automatic selection is performed

0: Cross point calculation automatic selection is not performed

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

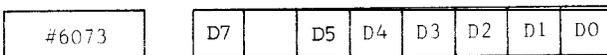
- D2 1: CMOS free area check function is invalid
 0: CMOS free area check function is valid
- D1 New method interference checking function
 1: Checks to cause an alarm.
 0: Checks to correct it automatically.
- D0 Workpiece setter, tool setter, touch sensor input signal selection
 1: Touch sensor signal hardware interruption
 0: Touch sensor signal 1/0



When the expanded playback external writing signal is on

- 0: Not written
 1: Written

D0	X-axis
D1	Y-axis
D2	z-axis
D3	4th-axis
D5	Cutting feedrate F
D6	Spindle speed S
D7	Tool command T



- D7 1: Does not convert "O" code to ":" at tape-out when #6011 D7 = 1.
 0: Execute 1/0 normally according to the #60 11 D7 setting.
- D5 1: Cancels compensation of tool position offset (G45 to G48) at reference point return.
 0: Does not cancel compensation of tool position offset (G45 to G48) at reference point return.
- D4 1: With M code macro call argument designation
 0: Without M code macro call argument designation

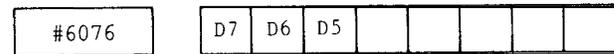
- D3 When there are two numbers for the same address
 1: The latter number is effective
 0: Double-address error occurs.
- D2 Tool position offset (G45 to G48) number command
 1: H code, D code are effective
 0: Only D code alone is effective

- D1 The TR
 1: Prohibits
 0: Performs

the T code macro call upon restarting the program

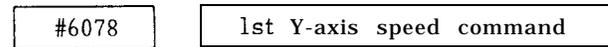
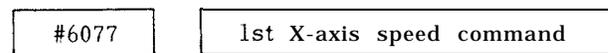
- D0 The TR
 1: Prohibits
 0: Performs

the M code macro call upon restarting the program



- D7 Reset function new specification and head calling function
 1: Valid
 0: Invalid
- D6 User time (TM4) function
 1: Valid
 0: Invalid
- D5 Selection of G code in 08 group at reset
 1: G code in 08 group is G49 when both #6005 D3 and D4 are "0".
 0: G code that is used at the previous time is stored.

#6077 to #6091 are parameters for circular arc projection compensation (soft hammer) and used as byte parameters.



#6079	1st Z-axis speed command
#6080	2nd X-axis speed command
#6081	2nd Y-axis speed command
#6082	2nd Z-axis speed command

Setting: "1" = 500 pps

Setting range: 0 to 255

#6083	X-axis compensation start waiting time
#6084	Y-axis compensation start waiting time
#6085	Z-axis compensation start waiting time
#6086	H1X effective time
#6087	H1Y effective time
#6088	H1Z effective time
#6089	H2X effective time
#6090	H2Y effective time
#6091	H2 Z effective time

Setting: "1" = 2 ms

Setting range: 0 to 255

#6092	D7		D5	D4	D3			
-------	----	--	----	----	----	--	--	--

- D7 1: Displays the macro call T code when restarting the program
- 0: Does not display the macro call T code when restarting the program
- D5 1: Invalidates cycle start while waiting for MOO/MO1 FIN.
- 0: Validates cycle start while waiting for MOO/MO1 FIN.
- D4 1: G28 or G30 in the G68 mode is to be alarm "150.
- 0: G28 or G30 in the G68 mode is validated.
- D3 1: 1: 1 deg/min in inch system.
- 0: 1: 2.54 deg/min in inch system.

#6107	
-------	--

Specifies the number of manual pulse generators.
 Setting: "1" = 1
 Setting range: 0 or 1

Note : Be sure to turn the power on and off after changing the parameter.

#6110	
#6111	(1)
#6112	(*)
#6113	(=)
#6114	(()
#6115	())

#6110 to #6115:

Specifies punches of the codes corresponding to the symbols of EIA codes used in the user macro body.

- 1: Punched
- 0: Not punched

#6116	
#6117	
#6118	
#6119	

Maximum 4 types of M codes to stop advance reading.

Setting range: 0 to 255

#6120	G-1
#6121	G-2
#6122	G-3
#6123	G-4

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

#6124		G-5
#6125		G-6
#6126		G-7
#6127		G-8
#6128		G-9
#6129		G-10

#6120 to #6129:

Specify up to 10 G codes for calling user macros.

Setting range: 0 to 199
(except for the G code specified by NC machine builder)

#6130		M-1
#6131		M-2
#6132		M-3
#6133		M-4

#6130 to #6133 :

Specifies up to 4 M codes for calling user macros.

Setting range : 0 to 255

Note : M00, M01, M02, M30 and M90 to M99 cannot be called by user macros.

#6134	
-------	--

1: Allows the T code to call a user macro.

0: Does not allow the T code to call a user macro.

#6141	
#6142	
#6143	
#6144	
#6145	

#6146	
#6147	
#6148	
#6149	

#6141 to #6149 :

Specifies the feedrate change for one increment on a manual pulse generator, for F1 to F9, respectively, of F1 -digit designation.

Setting : " 1 " = 0.1 mm/min/pulse

Setting range : 0 to 255

#6150	
#6151	
#6152	
#6153	

Specifies magnification for the deviation pulse at the most rapid traverse time of the X-axis, in the alarm zone in the successive order of X-axis, Y-axis, Z-axis and 4th axes.

Setting : " I " = 1%

Setting range : 1 to 200

#6155	
-------	--

Specifies the magnification against the deviation pulse at the maximum speed of the spindle in the spindle alarm zone.

Setting: " I " = 1%

Setting range: 1 to 200

Note: The error when " 0 " is set in #6150 to #6155 is " 120. " The parameter value is also rewritten to 120.

#6156	
#6157	
#6158	
#6159	

Specifies error detect on area for X-, Y-, Z- and 4th axes, respectively.

Setting : "1" = 1 pulse

Setting range : 0 to 255

Note : Be sure to turn the power on and off after changing the parameter.

#6161 []

Spindle error detect on area (for solid tap)

Setting: "1" = 1 pulse

Setting range: 0 to 255

Note : Be sure to turn the power on and off after changing the parameter #6150 to #6161.

#6162 []

#6163 []

#6164 []

#6165 []

#6166 []

#6167 []

M codes to stop advance reading setting expanded area.

#6168 []

#6169 []

#6170 []

#6171 []

Set pitch error compensation setting pulse multiplication factor for X-, Y-, Z- and 4th axes respectively.

Setting : "1" = 1 multiplication

Setting range: 0 to 3

#6174 []

Spindle error detect ON area (for spindle indexing)

Setting: "1" = 1 pulse

Setting range: 0 to 255

#6196 []

Number of teeth on the spindle side gear

Setting : "1" = 1

Setting range : 0 to 127

#6197 []

Number of teeth on the spindle intermediate gear

Setting : "1" = 1 tooth

Setting range : 0 to 127

#6198 []

Number of teeth on the motor intermediate gear

Setting : "1" = 1 tooth

Setting range : 0 to 127

#6199 []

Number of teeth on the motor side gear

Setting : "1" = 1 tooth

Setting range : 0 to 127

#6220 []

Specifies the interval from the time, M, S, T and B codes are transmitted until the time MF, SF, TF and BF are transmitted. This parameter must be a multiple of 8.

Setting : "1" = 1 ms

Setting range : 0 to 32767

#6221 []

Time from gear output (GRH, GRL) sending to SF sending at S. D/A or S12-bit output.

Setting : "1" = 1 ms

Setting range : 0 to 32767

#6222 []

Specifies the maximum handle feedrate, which is common to the linear axes (X, Y, Z, U, V, W).

Setting : "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 30000

Note : The settings for the rotary axes (A, B, C) are made with #6348.

#6223 []

Specifies the tool shift speed for canned cycles of G76 and G77.

Setting : "1" = 1 mm/min

Setting range : 0 to 30000

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

Note: This specifications effective when #6019 DO= 1.
If #6019 DO = 0, rapid traverse is effective regardless of this parameter specification.

#6224

Specifies the delay time for checking the spindle speed reaching signal (SAG R).
Setting: "1" = 1 ms
Setting range: 0 to 32767

#6225

Specifies the feedrate for the rapid traverse section in circle cutting (G12, G 13) .
Setting : " 1" = 1 mm/min or 0.1 inch/rein
Setting range : 0 to 30000

#6226

#6227

Specifies the maximum feedrate for F1-digit designation.
Setting : " 1" = 1 mm/min or 0.1 inch/rein
Setting range : 0 to 30000
Note : The maximum feedrate for F1 -F4 commands is set in #6226 and that for F5-F9 commands in #6227. Any feedrates increased on manual pulse generators are bunched into these settings.

#6228

Specifies the maximum feedrate for the linear axes (x, Y, z, u, v, w).
Setting : " 1" = 1 mm/min or 0.1 inch/rein
Setting range : 0 to 30000

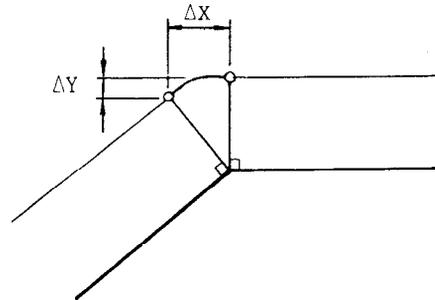
#6229

Specifies the maximum feedrate for the rotary axes (A, B, C).
Setting : " 1" = 1000 deg/min
Setting range : 0 to 30000
Note: Any feedrate greater than those set in #6228 and #6229 are bunched into those settings.

#6230

When a circular path is drawn in tool radius compensation outside a corner approaching 180°, the movement follows on a very small circular arc. In this, arc movement is considered to affect the workpiece surface machining, this parameter is used to set the critical arc value.

Setting : "l" = 0.001 mm/min or 0.001 inch/rein
Setting range : 0 to 32767



The corner arc setting is ignored when:
 $\Delta X \leq \#6230$
 $\Delta Y \leq \#6230$
Standard setting = 5

#6231

Specifies the F_0 speed for rapid traverse override.
Setting : "1" = 1 mm/min or 0.1 inch/rein
Setting range : 0 to 30000

#6232

Specifies the feedrate in the skip function (G31).
Setting : " 1" = 1 mm/min or 0.1 inch/rein
Setting range : 0 to 30000

Note : This setting is effective when parameter #6019D₄ = 1.

#6233

to

#6264

#6233 to #6264

Specifies the feedrate for the respective positions on the jog feedrate select switch.

Setting : "I" = 1 mm/min or 0.1 inch/rein

Setting range: 0 to 30000

Typical settings

Typical Settings mm/min

Switch position	Parameter		Continuous manual feedrate	
	N umber	Setting	#6250 = 0	#6265 = 10
0	#6233	0	0	0
1	#6234	1	1	0.1
2	#6235	2	2	0.2
3	#6236	4	4	0.4
4	#6237	6	6	0.6
5	#6238	8	8	0.8
6	#6239	10	10	1.0
7	#6240	12	12	1.2
8	#6241	15	15	1.5
9	#6242	20	20	2.0
10	#6243	25	25	25
11	#6244	30	30	30
12	#6245	40	40	40
13	#6246	50	50	50
14	#6247	60	60	60
15	#6248	80	80	80
16	#6249	100	100	100
17	#6250	120	120	120
18	#6251	150	150	150
19	#6252	200	200	200
20	#6253	250	250	250
21	#6254	300	300	300
22	#6255	400	400	400
23	#6256	500	500	500
24	#6257	600	600	600
25	#6258	800	800	800
26	#6259	1000	1000	1000
27	#6260	1200	1200	1200
28	#6261	1500	1500	1500
29	#6262	2000	2000	2000
30	#6263	2500	2500	2500
31	#6264	3000	3000	3000

#6265

The manual feedrates set in parameters #6233 to #6264 can be reduced to a tenth of the original setting. This applies to the settings on all switch positions lower than the value specified in this parameter #6265.

Setting : 0 to 32 (switch position)

#6266

#6267

#6268

#6269

#6266 to #6269:

Specifies the maximum spindle speed, respectively, for gears 1, 2, 3 and 4 (specifiable only in S5-digit designation).

Setting : "I" = 1 rein-l or 10 rein-l
(Set by parameter #6022 D7)

Setting range : 0 to 32767

#6270

Specifies the speed command output value to the spindle motor when a gear shift (GR O) input is entered.

Specifies No. of bits occupied in 32767 or 4095 directly.

Setting value: $\frac{\text{Gear shift spindle motor speed}}{\text{Spindle motor max speed}}$
(Command = 10 V)

x 4095 --- 12-bit output

$\frac{\text{Gear shift spindle motor speed}}{\text{Spindle motor max speed}}$
(Command = 10 V)

x 32767 --- Analog output

Setting range: 0 to 32767

#6271

#6272

#6273

PARAMETER NUMBERS AND THEIR CONTENTS
(Cent'd)

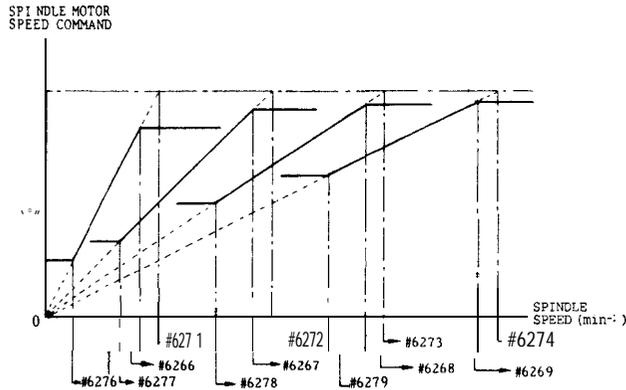
#6274

#6271 to #6274:

Specifies the maximum speed of the spindle D/A (10 V/4095), respectively, for gears 1, 2, 3 and 4 each selected by an input signal (specifiable in S5-digit designation). Set the spindle speed applicable when the speed command voltage is 10 V.

Setting : "1" = 1 rein-l or 10 rein-l
(Set by parameter #6022 D7)

Setting range : 1 to 32767



#6275

Specifies the spindle motor speed in effect when a spindle orientation (SOR) input is entered (specifiable in S5-digit designation).

Setting : "1" = 1 rein-l

Setting range : 1 to 32767

#6276

#6277

#6278

#6279

#6276 to #6279:

Specifies the minimum speed of the spindle, respectively, for gears 1, 2, 3 and 4 each selected by an input signal (specifiable in S5-digit designation).

Setting : "1" = 1 rein-l or 10 min⁻¹
(Set by parameter #6022 D7)

Setting range : 0 to 32767

#6280 X-axis

#6281 Y-axis

#6282 Z-axis

#6283 4th axis

#6280 to #6283 :

Specifies the rapid traverse rate, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 30000

Note : Be sure to turn the power on and off after changing the parameter.

(Reference)

The following calculations are made between rapid traverse rate and acceleration and deceleration for re-computing the rapid traverse rate and second-stage time constant switching rate. The example of calculations shown below is made for the parameters related to X-axis, but the same can be applied also to Y-axis, Z-axis or 4th-axis.

Rapid traverse rate :

$$V_a = \#6280 \rightarrow \text{Rate } V_a \text{ recreated by the calculations.}$$

Acceleration/deceleration 1st-stage time constant :

$$t_a = \#6286$$

2nd-stage time constant switching rate :

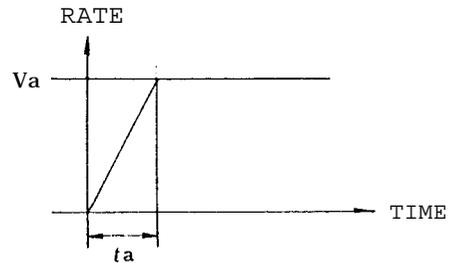
$$V_b = \#6292 \rightarrow \text{Rate } V_b \text{ recreated by the calculations.}$$

Acceleration/deceleration 2nd-step time constant : #6298

(a) Where 2nd-step acceleration and deceleration are not used :

$$\text{Condition : } V_b = 0 \\ t_b = 0$$

$$t'a = \left(\frac{V_a}{7.5} * \frac{100}{MP} \right) / \frac{t_a}{8} \quad V'a = t'a * \frac{t_a}{8} * 7.5$$

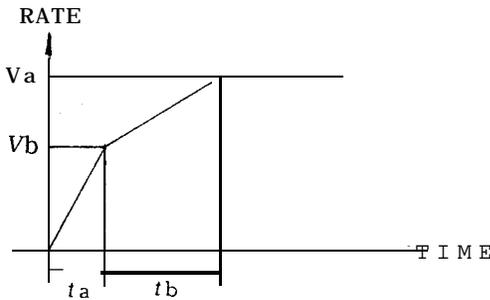


(b) Where 2nd-step acceleration and deceleration are used :

$$t'a = \left(\frac{V'b}{7.5} * \frac{100}{MP} \right) / \frac{t'a}{8}$$

$$t'b = V'b + \left(\frac{V'a - V'b}{7.5} * \frac{100}{MP} \right) / \frac{t'b}{8}$$

$$V'b = t'a * \frac{t'a}{8} \quad V'a = V'b + t'b * \frac{t'b}{8}$$

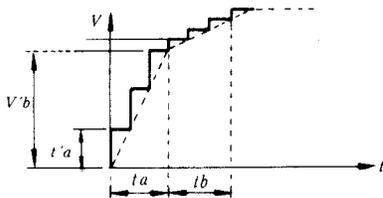


(c) Supplement for setting when 2nd-step acceleration/deceleration used

t'a : 1st-step acceleration/deceleration constant to set a multiple of 8 ms

t'b : 2nd-step acceleration/deceleration constant to set a multiple of 8 ms

t'a : 1st-step acceleration/deceleration constant. The result of calculation can be rounded. Therefore, Va and t'a must be set in such a way that the value to be rounded will be as small as possible.



(Example of X-axis)

$$t'a = (\#6269) \times \left(\frac{8}{7.5} \right) \times \frac{1}{(\#6286)} \dots$$

..... Numbers below decimal point omitted.

$$V'b = t'a \times \frac{t'a}{8}$$

$$t'b = (\#6280 - \#6292) \times \left(\frac{8}{7.5} \right) \times \frac{1}{(\#6298)} \dots$$

..... Numbers below decimal point omitted.

$$V'a = V'b + t'b \times \frac{t'b}{8}, Mp = 100$$

Actual rapid traverse : $V'a \times 7.5$ (mm/min)
 Actual changing speed : $V'b \times 7.5$ (mm/min)
 Actual acceleration/deceleration time is as set by parameter.

Notes :

1. By omitting the resultant calculation, rapid traverse setting may differ from actual speed. (The part below the decimal point is omitted for t'a or t'b value. In results, an error is generated in V'a or V'b).

2. MP indicates #6450, #6451, #6452 and #6453.

Input Status			Rapid Traverse				
ROV1	ROV2	ROV4	X-axis	Y-axis	Z-axis	α -axis	
1	1	0	#6280 Set speed	#6281 Set speed	#6282 Set speed	#6283 Set speed	100%
0	1	0	(#6280 Set speed) $\times \frac{1}{2}$	(#6281 Set speed) $\times \frac{1}{2}$	(#6282 Set speed) $\times \frac{1}{2}$	(#6283 Set speed) $\times \frac{1}{2}$	50%
1	0	0	(#6280 Set speed) $\times \frac{1}{4}$	(#6281 Set speed) $\times \frac{1}{4}$	(#6282 Set speed) $\times \frac{1}{4}$	(#6283 Set speed) $\times \frac{1}{4}$	25%
0	0	0	#6231 Set speed				F0
0	0	1	#6266 Set speed				F1
			#6267 Set speed				F2

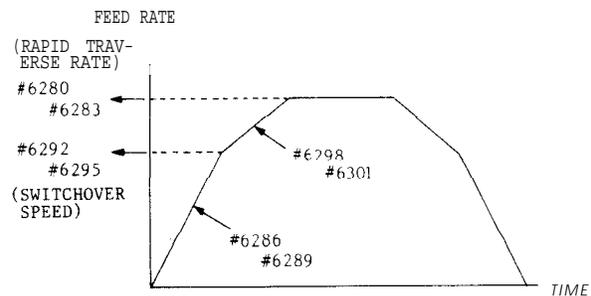
#6286	X-axis
#6287	Y-axis
#6288	Z-axis
#6289	4th axis

#6286 to #6289 :

Specifies the first-stage time constant in linear acceleration/deceleration, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 ms

Setting range : 8 to 32767



PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

Notes :

1. Setting should be made by multiples (X8).
2. Be sure to turn the power on and off after changing the parameter

#6291	Spindle
-------	---------

Specifies the linear acceleration speed constant

Setting : "1" = 1 ms

Setting range : 8 to 32767

#6292	X-axis
-------	--------

#6293	Y-axis
-------	--------

#6294	Z-axis
-------	--------

#6295	4th axis
-------	----------

#6292 to #6295 :

Specifies the second-stage time constant **switchover** speed in linear acceleration/deceleration, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 24000

Note : Be sure to turn the power on and off after changing the parameter.

#6298	X-axis
-------	--------

#6299	Y-axis
-------	--------

#6300	Z-axis
-------	--------

#6301	4th axis
-------	----------

#6298 to #6301 :

Specifies the second-stage time constant in linear acceleration/deceleration, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 ms

Setting range : 8 to 32767

Note : Setting should be made by multiples (X8).

#6304	X-axis
-------	--------

#6305	Y-axis
-------	--------

#6306	Z-axis
-------	--------

#6307	4th axis
-------	----------

#6304 to #6307 :

Specifies the traverse distance for reference point return, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : "1" = 0.001 mm or 0.0001 inch/rein

Setting range : 0 to 32767

#6310	X-axis
-------	--------

#6311	Y-axis
-------	--------

#6312	Z-axis
-------	--------

#6313	4th axis
-------	----------

#6310 to #6313 :

Specifies the approach speed 1 for reference point return, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 30000

#6316	X-axis
-------	--------

#6317	Y-axis
-------	--------

#6318	Z-axis
-------	--------

#6319	4th axis
-------	----------

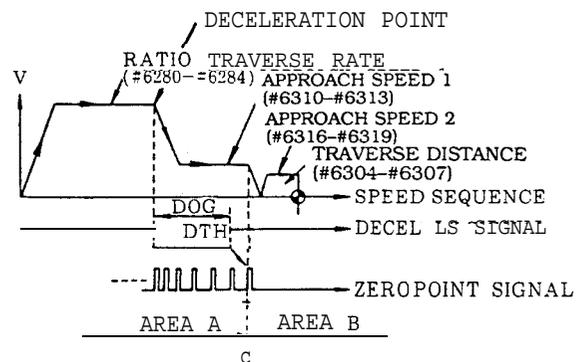
#6316 to #6319 :

Specifies the approach speed 2 for reference point return, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 30000

Note : The parameters related with reference point return operations are as follows.



c

. Reference point return direction:

#6010 D0 - D3

· Reference point return enabled/disabled:

#6016 D0 - D3

#6322	X-axis
#6323	Y-axis
#6324	Z-axis
#6325	4th axis

#6322 to #6325 :

Specifies the number of the end point for pitch error compensation, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : 0 to 511

#6328	X-axis
#6329	Y-axis
#6330	Z-axis
#6331	4th axis

#6328 to #6331 :

Specifies the number of the start point for pitch error compensation, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : 0 to 511

#6334	X-axis
#6335	Y-axis
#6336	Z-axis
#6337	4th axis

#6334 to #6337 :

Specifies the reference point for pitch error compensation, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : 0 to 511

#6340	
-------	--

Specifies the external deceleration speed for rapid traverse.

Setting : "1" = 1 mm/min or 0.1 inch/rein
(common to all axes)

Setting range : 0 to 30000

#6341	
-------	--

Specifies the external deceleration speed for cutting feed.

Setting : "1" = 1 mm/min (common to all axes)

Setting range : 0 to 30000

#6342	X-axis
-------	--------

#6343	Y-axis
-------	--------

#6344	Z-axis
-------	--------

#6345	4th axis
-------	----------

#6342 to #6345 :

Specifies the offset in external workpiece coordinate system shift, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : "1" = 0.001 mm

Setting range : 0 to ±32767

Note : Usually, these parameters are automatically set from the machine tool side through the external data input function.

#6348	
-------	--

Specifies the maximum speed for handle feed on the rotary axes (A, B, C) .

Setting : "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 24000

#6350	X-axis
-------	--------

#6351	Y-axis
-------	--------

#6352	Z-axis
-------	--------

#6353	4th axis
-------	----------

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

#6350 to #6353 :

Specifies the rapid traverse acceleration/deceleration constants X-, Y-, Z- and 4th-axes handle feed, respectively.

Setting : "1" = 1ms

Setting range : 8 to 32767

#6355	
-------	--

#6356	
-------	--

For tool pot indication

#6355: Sets tool pot indication start No.

#6356: Sets tool pot indication end No.

#6357	X-axis
-------	--------

#6358	Y-axis
-------	--------

#6359	Z-axis
-------	--------

#6357 to #6359:

Specifies the time between ESP and SVOFF for X-, Y- and Z-axis, respectively.

Setting : "1" = 8 ms

Setting range : 0 to 32767

#6362	
-------	--

#6363	
-------	--

#6364	
-------	--

#6362: Tool pot SP
#6363: Tool pot MG
#6364: Tool pot WT

#6383	
-------	--

By the input of x100, the scale factor of handle becomes effective.

Setting : "1" = × 1

#6384	
-------	--

to to

#6391	
-------	--

Title display code setting after turning the power on :
1st line

#6392	
-------	--

to to

#6399	
-------	--

Title display code setting after turning the power on :
2nd line

#6400	X-axis
-------	--------

#6401	Y-axis
-------	--------

#6402	Z-axis
-------	--------

#6403	4th axis
-------	----------

#6405	Spindle
-------	---------

Backlash compensation values shown in the order of X-, Y-, Z-, 4th-axes and spindle.

Setting : "1" = 1 pulse

Setting range: 0 to ±8191

Note: Be sure to turn the power on and off after changing the parameter.

#6406	X-axis
-------	--------

#6407	Y-axis
-------	--------

#6408	Z-axis
-------	--------

#6409	4th axis
-------	----------

#6411	Spindle
-------	---------

Command unit position loop gains are set in the order of X-, Y-, Z-, 4th-axes and spindle.

Setting : "1" = 0.01 s

Setting range : 0 to 32767

#6412	X-axis
-------	--------

#6413	Y-axis
-------	--------

#6414	Z-axis
-------	--------

#6415	4th axis
-------	----------

Acceleration and deceleration time constants for control unit during normal cutting are set in the order of X-, Y-, Z- and 4th-axes.

Setting : "l" = 1 ms

Setting range : 0 to 32767

#6418	X-axis
#6419	Y-axis
#6420	Z-axis
#6421	4th axis

Acceleration and deceleration bias for control unit during normal cutting is set in the order of X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 32767

#6424	X-axis
#6425	Y-axis
#6426	Z-axis
#6427	4th axis

Acceleration and deceleration time constants for control unit during screw/tap cutting is set in the order of X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 ms

Setting range : 0 to 32767

#6430	X-axis
#6431	Y-axis
#6432	Z-axis
#6433	4th axis

Acceleration and deceleration bias for control unit thread/tap cutting is set.

Setting : "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 32767

#6436	X-axis
#6437	Y-axis
#6438	Z-axis
#6439	4th axis

Overshoot during G60 in the order of X-, Y-, Z- and 4th-axes.

Setting : "1" = 1 pulse

Setting range : 0 to 32767

#6444	X-axis
#6445	Y-axis
#6446	Z-axis
#6447	4th axis

Setting : "l" = 0.001 mm/rev or 0.0001 inch/rev

Setting range : 1 to 65535

Notes :

1. Be sure to turn the power on and off after changing the parameter.
2. For input of 32768 to 65535, indication is shown by minus.

Example

Input	Indication
32768	→ -32768
32769	→ -32767
65535	→ -1

#6449	
-------	--

Feedback pulse number for 1 rotation of spindle PG.

Setting: "1" = 1 pulse/rev

Setting range: 1 to 32768

#6450	X-axis
#6451	Y-axis
#6452	Z-axis
#6453	4th axis

Minimum unit of travel in the order of X-, Y-, Z- and 4th-axes.

Setting : "1" = 0.00001 mm or 0.000001 inch

Setting range: 100 is set.

Note : Be sure to turn the power on and off after changing the parameter.

#6456	X-axis
-------	--------

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

#6457 Y-axis

#6458 Z-axis

#6459 4th axis

Servo input in the order of X-, Y-, Z- and 4th-axes.

Setting: "1" = 1 r/rein/volt

Setting range : 250 is set.

#6462

Spindle indexing speed rate.

Setting: "1" = 500 pps

Setting range: 0 to 32767

#6463

Spindle indexing creep rate.

Setting: "1" = 500 pps

Setting range: 0 to 32767

#6464

Spindle indexing & spindle stop confirmation timer.

Setting: "1" = 8 ms

Setting range: 0 to 255

#6465

Spindle indexing and origin position.

Setting: "1" = 1P

Setting range: 0 to 4095

#6466 F1

#6467 F2

Rapid traverse override of F1 and F2.

Setting: "1" = 1 mm/min or 0.1 inch/rein

Setting range : 0 to 30000

#6468

Synchronization offset gain upon solid tap stationary cutting feed

Setting: "1" = 1/256 times

Setting range: -32768 to 32767

#6469

Synchronization offset gain upon solid tap acceleration/deceleration feed

Setting: "1" = 0.0001 times

Setting range : 0 to 32767

#6470

Solid tap R point return feedrate magnification

Setting: "1" = 0.1 times

Setting range: 0 to 32767

#6471

Solid tap spindle revolutions upper limit

Setting: "1" = 1 min⁻¹

Setting range: 0 to 32767

#6472

Spindle revolutions against the 10 V command of the gear used on the solid tap

Setting: "1" = 1 rein-l

Setting range: 0 to 32767

#6473

Spindle stop acknowledgment revolutions in the G93 spindle position loop

Setting: "1" = 1 min⁻¹

Setting range: 0 to 30

#6476

In-position width area upon solid tap R point lead-in

Setting: "1" = 1 spindle PG pulse

Setting range: 0 to 32767

#6480 X-axis

#6481 Y-axis

#6482 Z-axis

#6483 4th axis

Projected C-phase shift distance

Setting: "1" = 1 pulse

Setting range: -32768 to 32768

#6486

Spindle position loop gain (for spindle indexing)

Setting: "1" = 0.01 1/s

Setting range: 0 to 32767

#6488

Form compensation linear accel/decel maximum acceleration

Setting: "1" = 1/64 m/s²

Setting range: 1 to 32768

#6489

#6490

CAL ERROR (DIVISION) area setting

#6494 I X-axis

#6495 Y-axis

#6496 Z-axis

#6497 4th axis

Feed forward gain

#6580

#6581

#6582

#6583

#6584

#6585

#6586

#6587

#6588

#6589

#6590

#6591

#6592

#6593

#6594

#6595

#6596

#6597

#6598

#6599

Sets the M codes used to call up the macro program (maximum of 20 M codes).

#6600 X-axis

#6601 Y-axis

#6602 Z-axis

#6603 4th axis

#6600 to #6603 :

Specifies the plus direction boundary value for stored stroke limit 1, respectively, on the X-, Y-, Z- and 4th-axes.

Setting: "1" = 1 pulse

Setting range: 0 to ±99999999

PARAMETER NUMBERS AND THEIR CONTENTS
(Cont'd)

#6606	X-axis
#6607	Y-axis
#6608	Z-axis
#6609	4th axis

#6606 to #6609 :

Specifies the minus direction boundary value for stored stroke limit 1, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6612	X-axis
#6613	Y-axis
#6614	Z-axis
#6615	4th axis

#6612 to #6615 :

Specifies the distance between the first and the second reference point, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6618	X-axis
#6619	Y-axis
#6620	Z-axis
#6621	4th axis

#6618 to #6621 :

Specifies the distance between the first and the third reference point, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6624	X-axis
-------	--------

#6625	Y-axis
#6626	Z-axis
#6627	4th axis

#6624 to #6627 :

Specifies the distance between the first and the fourth reference point, respectively, on the X-, Y-, Z- and 4th-axes.

Setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6630	X-axis
#6631	Y-axis
#6632	Z-axis
#6633	4th axis

#6630 to #6633 :

Specifies the value for automatic coordinate system setting at the time of inch input, respectively, on the X-, Y-, Z-, and 4th-axes. A desired value should be set in inches for the distance between the first reference point and the reference point of the coordinate system to be established.

Setting : " 1" = 0.0001 in.

Setting range : 0 to *99999999

#6636	X-axis
#6637	Y-axis
#6638	Z-axis
#6639	4th axis

#6636 to #6639 :

Specifies the value for automatic coordinate system setting at the time of metric input, respectively, on the X-, Y-, Z-, and 4th-axes. A desired value should be set in millimeters for the distance between the first reference point and the reference point of the coordinate system to be established.

Setting : "1" = 0.001 mm

Setting range : 0 to *99999999

Note : Each setting effective only for an axis with parameter #6015 at " 1".

#6642	X-axis
#6643	Y-axis
#6644	Z-axis
#6645	4th axis

#6642 to #6645 :

Specifies the compensation interval in pitch error compensation, respectively, on the X-, Y-, Z and 4th-axes.

Setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6650	X-axis
#6651	Y-axis
#6652	Z-axis

#6650 to #6652 (optional) :

Specifies stored stroke limit 3 for X-, Y-, and Z-axis, respectively.

Plus (+) boundary setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6653	X-axis
#6654	Y-axis
#6655	Z-axis

#6653 to #6655 (optional) :

Specifies stored stroke limit 3 for X-, Y-, and Z-axis, respectively.

Minus (-) boundary setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6656	X-axis
#6657	Y-axis
#6658	Z-axis

#6656 to #6658 (optional) :

Specifies stored stroke limit 4 for X-, Y-, and Z-axis, respectively.

Plus (+) boundary setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6659	X-axis
#6660	Y-axis
#6661	Z-axis

#6659 to #6661 (optional) :

Specifies stored stroke limit 4 for X-, Y-, and Z-axis, respectively.

Minus (-) boundary setting : " 1" = 1 pulse

Setting range : 0 to ±99999999

#6662	X-axis
#6663	Y-axis
#6664	Z-axis

#6662 to #6664 (optional) :

Specifies stored stroke limit 5 for X-, Y-, and Z-axis, respectively.

Plus (+) boundary setting : " 1" = 1 pulse

Setting range : 0 to *99999999

#6665	X-axis
#6666	Y-axis
#6667	Z-axis

#6665 to #6667 (optional) :

Specifies stored stroke limit 5 for X-, Y-, and Z-axis, respectively.

Minus (-) boundary setting : "1" = 1 pulse

Setting range : 0 to *99999999

#6674	
-------	--

Sets the multiplication factor of characters of the display at power on.

#6675	
-------	--

Sets the CRT starting position of the title display at power on.

#6676	
-------	--

Handle input pulse lag pulse clamp amount

Setting: "1" = 1 pulse (input pulse)

PARAMETER NUMBERS AND THEIR CONTENTS
(Cent'd)

#6677	
-------	--

Sets no. of remaining movement pulses to be clamped of handle.

" 1" = 1 pulse

#6680	X-axis
-------	--------

#6681	Y-axis
-------	--------

#6682	Z-axis
-------	--------

#6683	4th axis
-------	----------

External work coordinate shift offset amount:
6 digits"

Setting range : " 1" = 0 to ±799999

#6690	X-axis
-------	--------

#6691	Y-axis
-------	--------

#6692	Z-axis
-------	--------

#6693	4th axis
-------	----------

Form compensation servo follow-up coefficient

Setting range : 0 to 65535

#8000	number 0
-------	----------

to to

#8511	number 511
-------	------------

#8000 to #8511

Specifies the respective values of lead screw error compensation.

Setting : 0 to ±15 (output increment)

Note : Parameters #6322 to #6339 determine specific combinations of settings and axes.

APPENDIX-3 STORED LEADSCREW ERROR COMPENSATION

This function automatically compensates for lead-screw error on each axis according to the compensation data set by parameter and is effective after completion of reference point return. The compensation data are made on the distances between the reference point on each axis and specified points.

Compensation axes : X-, Y-, Z- and 4th-axes.
(including rotary axis)

No. of correction points: 512 Max.

Compensation base point: Reference point

Compensation interval: 10000 pulses or more

However, if the following conditions are satisfied, compensation interval can be less than 10000 pulses.

$$\left\{ \begin{array}{l} -\frac{FX}{60} \times 10^6 \\ \times 10^{-3} \times N + 100 \text{ pulses or more} \end{array} \right.$$

F : Maximum feeding speed
(quicker between rapid traverse or cutting feeding)

N : Compensation magnification
(by parameter setting)

(Example) 15 m/min, compensation magnification : 2

$$\frac{15 \times 10^6}{60} \times 10^{-3} \times 2 + 100 = 4100 \text{ pulses}$$

Note : With 6000 pulse interval at 24 m/min for 3-multiplication, on the contrary to the above example, position is shifted by improper pulse counting.

Data setting system: Absolute /incremental
(Set by Parameter #6039D2)

Compensation value:

Minimum compensation unit: 1 pulse (least output increment)

Compensation multiplication: X3 max.

One-time-compensation value: 15 pulses max.
X (Compensation multiplication)

Note 1:

Regardless of absolute/incremental setting, the difference between neighboring compensation values should be (15 pulses x compensation multiplication) and below.

Note 2:

Maximum set value in case of absolute setting is ± 127 pulses. Compensation multiplication is taken on this value.

Note 3:

No. of correction points on each axis can be arbitrary as far as the total compensation points are within 512.

Note 4:

Where the 4th and 5th axis is a rotary axis , operation is possible within ± 200 revolution maximum.

Note 5:

For the axis requiring no leadscrew error compensation, set the parameter for compensation multiplication factor at "0. "

Note 6:

Setting at base point is to be made in such a manner that the pitch error compensation value will not occur. In the example shown below, the incremental set point of #8002 should be zero. Ales, set the absolute set point in such a manner that the difference between #8001 and #8002 becomes "0. "

APPENDIX-3 STORED LEADSCREW ERROR COMPENSATION (Cent'd)

Table A3. 1

	Axis	Parameter #	Functions
Compensation interval	X to α	#6642 to #6645	10000 OR MORE "1" = 1 pulse
Absolute /incremental setting switchable		#6039 D2	"0" = Incremental setting "1" = Absolute setting
Compensation reference no.	X to α	#6334 to #6337	Value of parameter # of compensation on each point minus 8000 will be written
Compensation max point	X to α	#6322 to #6325	
Compensation min point	X to α	#6328 to #6331	
Compensation value on each point	X to α	#8000 to #8511	-15 to +15 (Incremental setting) "1" = 1 pulse
Compensation multiplication	X to α	#6168 to #6171	0 to 3 "1" = 1X

The figure below shows the example of writing the data for X axis.

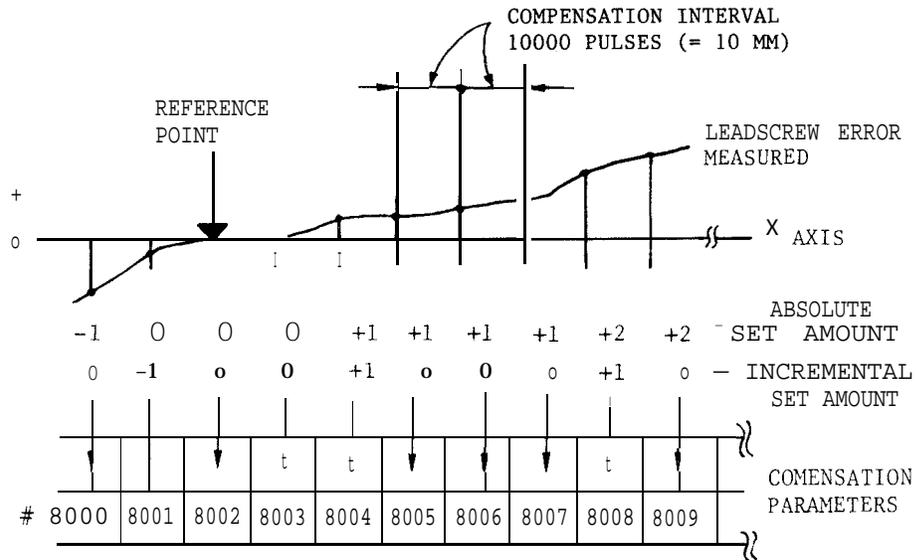
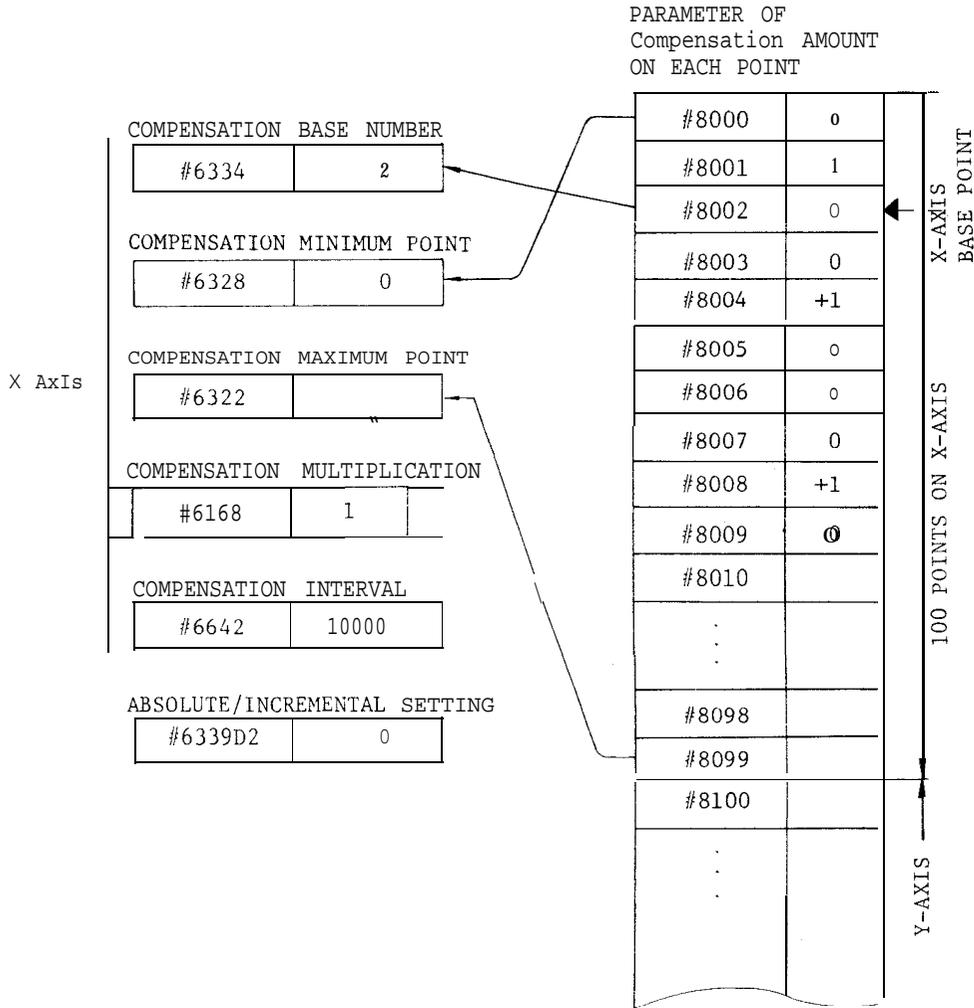


Fig. A3.1

In the above figure,
 Compensation interval: 10000 pulses
 Absolute/incremental: Incremental
 Compensation multiplication : x 1
 Compensation point on X axis: 100 points
 Each parameter is set as follows :



Note : Write the data so that each axis reference point compensated value will be "0".

Fig. A3.2

APPENDIX-3 STORED LEADSCREW ERROR COMPENSATION (Cent'd)

To use the 4th-axis as the rotary axis, follow the rules shown below in addition to the rules for setting X-, Y-, and Z-axes.

(1) Compensation Interval

The compensation interval should be more than 10000 pulses and the quotient obtained by dividing 360000 by the compensation interval become a positive integer.

(2) Compensation Amount at Reference Point

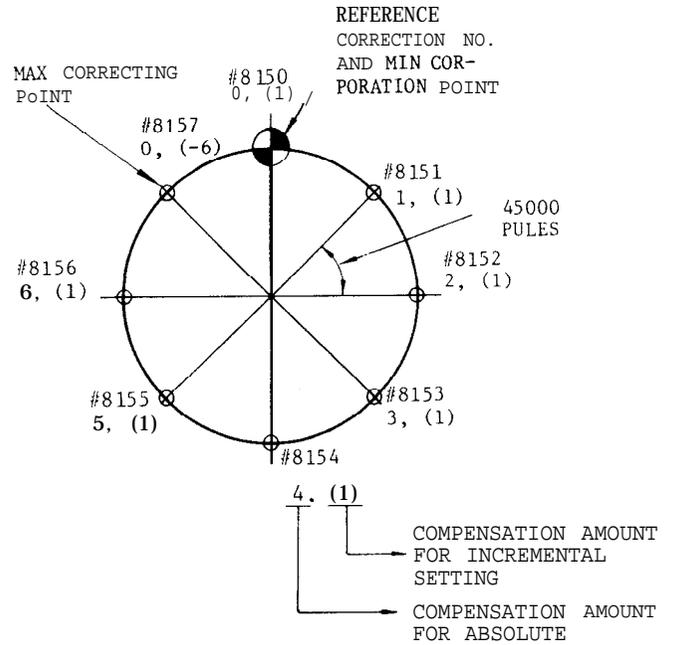
The compensation amount to be set to the reference point should be as follows :

- a. Absolute setting . . . "0"
- b. Incremental setting . . . "0"

In the case of incremental setting, set such a value for the compensation amount at compensation maximum point that the sum of the compensation amount of each point becomes "0."

(3) Sample Writing

If the compensation interval is 45000 pulses (one rotation divided by 8) as shown below, set the parameters as follows.



Compensation parameter at each point

		Parameter	Absolute setting	Incremental setting
Compensation reference No.		#8150	0	0
Compensation min point		#8151	1	1
Compensation max point		#8152	2	1
Compensation multiplication factor		#8153	3	1
		#8154	4	1
		#8155	5	1
		#8156	6	1
		#8157	0	-6
Compensation point				
		#6337	150	
		#6331	150	
		#6325	157	
		#6071	1	
		#6645	45000	

4th axis of rotary axis

APPENDIX-4 LIST OF STANDARD **INPUT/OUTPUT** SIGNALS

Table A4. 1 shows standard input/output signals. For custom-built signals depending on the system, refer to the list of I/O signals provided on the system.

DISPLAY

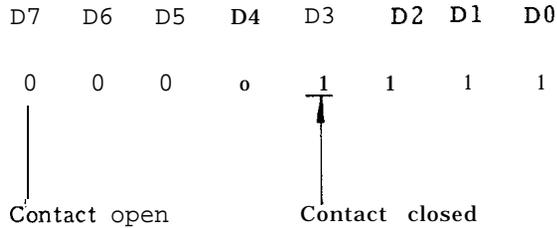


Table A4. 1 List of Standard Input Signals

	D ₇	D ₆	D ₅	D ₄	D ₃	D ₂	D ₁	D ₀
#1300	EDT	MEM	MDI	T	s	H	J	RT
	EDIT	MEMRY	MANUAL	TAPE	STEP	HANDLE	MANUAL	RAP ID
			DATA	INPUT			FEED	TRAVERSE
#1301	OVC	ROV2	ROV1	OV16	ov8	OV4	OV2	OV1
	OVERRIDE CANCEL RAPID TRAVERSE RATE OVERRIDE			FEEDRATE OVERRIDE				
#1302	-α	+α	-z	+Z	-Y	+Y	-x	+x
	JOB"PB							
#1303	SPC	SPB	SPA	JV16	JV8	JV4	JV2	JV1
	SPINDLE SPEED OVERRIDE			MANUAL FEEDRATE OVERRIDE				
#1304	DRS	MP4	MP2	MP1	Hα	HZ	HY	HX
	DISPLAY RESET	HANDLE PULSE MULTIPLY			HANDLE AXIS			
#1305	APL	MLK	OPT	DRN	BDT	DLK	S BK	
	M-FUNCTION LOCK	MACHINE LOCK	OPTIONAL STOP	DRY RUN	BLOCK DELETE	DISPLAY LOCK	SINGLE BLOCK	

Table A4. 1 List of Standard Input Signals (Cont'd)

	'7	'6	'5	D ₄	'3	'2	'1	'0
#1 306	SRN	F1	RET	TLM1	ZRN	EDTLK	SP	ST
	PROGRAM RESTART	F1 - DIGIT	RETRACT	TOOL LENGTH	ZERO RETURN	EDIT LOCK	FEED HOLD	CYCLE START
#1 307	PINT	ZNG	ABs		MI α	MIZ	MIY	MIX
	PROGRAM INTER-	Z-AXIS LOCK	MANUAL ABSOLUTE		MIRROR IMAGE			
#1 308	9BDT	8BDT	7 BDT	6BDT	5BDT	4BDT	3BDT	2BDT
	SPECIAL BLOCK DELETE							
#1 309	ZRN2		4NG	EWS	PST		WSH	
	MANUAL 2ND REFERENCE POINT RETURN		4TH AXIS NEGLECT	EXTERNAL WRITING SIGNAL	TACTILE SENSOR SIGNAL			
#1 310	DEVSLTI							
	TAPE OPERATION DEVICE DESIGNATION							
#1311								
#1 312	PLYBK	TCNT	SENSON	TLCTN	TLSKP	TLRST	ESC1	ESCO
	PLAYBACK	CUTTING TIME COUNT DISABLE SIGNAL	TACTILE SWITCH ENABLE SIGNAL	TOOL LIFE CONTROL			EXT STROKE CHECK SELECTION	
							0 0 Invalid	
							0 1 3rd	
							1 0 4th	
							1 1 5th	
#1 313	RWDH	FSCH	FSMEM	FSCM	SLPC			
	HIGH-SPEED REWIND & AUTO START	FEEDRATE, SPINDLE SPEED	FEEDRATE, SPINDLE SPEED	FEEDRATE, SPINDLE SPEED	SPINDLE LOOP COMMAND			
		MEMORY CHANGE	STORAGE	EDITING MODE	INPUT			
#1314	SPE	SPD	ROV4		ECLM			
	SPINDLE OVERRIDE		RAPID TRAVERSE RATE OVERRIDE		EXT PROGRAM CLEAR			

	'7	'6	'5	D ₄	'3	'2	'1	'0
#1315	MANINT	SSM	PPR		NTCRQ	TCFIN		
	MACHINING INTERRUPTION POINT RETURN SIGNAL	SENSOR STOP MODE	MACHINING INTERRUPTION POINT RETURN MODE		NEW TOOL CHANGE REQUEST	END OF TOOL CHANGE		
#1316	EFIN	FIN	RWD	EOP	ERS	EXTC	STLK	MRD
	COMMAND CYCLE	MST COMPLE- TION	EXTERNAL REWIND	END PRO- GRAM	EXTERNAL RESET	EXTERNAL TIME COUNT	CYCLE START	FUNCTION PREP COMPLETED
#1317	S-INV	S-FIN	SSTP	SAGR	SOR	GRB	GRA	GRO
	SPINDLE REVERSE	S FIN	SPINDLE COMMAND 0	SPINDLE COINCI- DENCE	SPINDLE INDEXING	SPINDLE GEAR SELECTION	GEAR SHIFT	
#1318	ERR2	ERR1	ERR0	SENS(G)	SENS(T)	EXOUT	EXVER	EXIN
	DEC TO STOP	IMMEDI- ATE STOP	SINGLE BLOCK STOP	GAP ELIMINATION	TOOL BREAK	EXTERNAL OUTPUT	EXTERNAL COLLATION	EXTERNAL INPUT
#1319	*-L α	*+L α	*-LZ	*+LZ	*-LY	*+LY	*-LX	*+LX
	OVERTRAVEL							
#1320	HOF5				*IT α	*ITZ	*ITY	*ITX
	AUTO HANDLE	MODE OFFSET			AXIS INTERLOCK			
#1321	*-ED α	*+ED α	*-EDZ	*+EDZ	*-EDY	*+EDY	*-EDX	*+EDX
	EXTERNAL DECELERATION							
#1322	SONPB				*SVOF α	*SVOFZ	*SVOFY	*SVOFX
	SERVO POWER ON				SERVO OFF			
#1323	UI 7	UI 6	UI 5	UI 4	UI 3	UI 2	UI 1	UI 0
	USER 'MACRO							

Table A4. 1 List of Standard Input Signals (Cent'd)

	'7	'6	'5	'4	'3	'2	'1	'0
#1 324	UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8
	USER MACRO							
#1 325	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
	EXTERNAL DATA INPUT							
#1 326	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
	EXTERNAL DATA INPUT							
#1 327	EDCL	EDAS2	EDAS1	EDAS0	EDSD	EDSC	EDSB	EDSA
	EXTERNAL DATA INPUT CONTROL SIGNAL							
#1 328	EDAS3				*DEC α	*DECZ	*DECY	*DECX
	EXTERNAL COORDINATE SYSTEM SHIFT ABS/INC				REFERENCE POINT DECELERATION LS			
#1 329		TL64	TL32	TL16	TL8	TL4	TL2	TL1
	TOOL LIFE CONTROL TOOL GROUP NO. INPUT							
#1330								
#1331	SD18	SDI7	SDI6	SD15	SDI4	SD13	SD12	SDI1
	S 5-DIGIT COMMAND EXTERNAL INPUT							
#1332	SDI16	SDI15	SDI14	SDI13	SDI12	SDI11	SDI10	SDI9
	S 5-DIGIT COMMAND EXTERNAL INPUT							

	'7	'6	'5	'4	'3	'2	'1	'0
#1335	FSCLS	WASKAN1						

CLEAR FS END OF
 EDITING MANUAL
 DATA SKIP
 SIGNAL

#1338	WN128	WN64	WN32	WN16	WN8	WN4	WN2	WN1
-------	-------	------	------	------	-----	-----	-----	-----

EXTERNAL WORK NO. SEARCH INPUT

#1341	SID 8	SID 7	SID 6	SID 5	SID 4	SID 3	SID 2	SID 1
-------	-------	-------	-------	-------	-------	-------	-------	-------

SPINDLE
 INDEXING
 POSITION
 DESIGNATION

#1342					SID12	SID11	SID10	SID9
-------	--	--	--	--	-------	-------	-------	------

SPINDLE
 INDEXING
 POSITION
 DESIGNATION

#1343					SPCUT	SIDXI	SIDXINC	SIDX
-------	--	--	--	--	-------	-------	---------	------

STOP RESTART SPINDLE SPINDLE
 SPINDLE SPINDLE INDEXING INDEXING
 INDEXING. INDEXING POSITION REQUEST
 INCREMENTAL
 DESIGNATION

Table A4.2 List of Standard Output Signals

	'7	'6	'5	D ₄	D ₃	'2	'1	'0
#1 200	M30	M0 2	M0 1	MOO	DEN	OP	SPL	STL
					POSITION- ING COM- PLETED	FEED- ING	TEMPO- RARY STOP	CYCLE START
#1201	2ZPα	2ZPZ	2ZPY	2ZPX	1ZPα	1ZPZ	1ZPY	1ZPX
		SECOND REFERENCE POINT COMPLETION				FIRST REFERENCE POINT COMPLETION		
#1 202	4ZPα	4ZPZ	4ZPY	4ZPX	3ZPα	3ZPZ	3ZPY	3ZPX
		FOURTH REFERENCE POINT COMPLETION				THIRD REFERENCE POINT COMPLETION		
#1203								
#1204								
#1205			TLCHA	TLCHB	TCF	TCHGF		
		TOOL CHANGE SIGNAL	NEW TOOL SELECTION SIGNAL	TOOL CHANGE READING SIGNAL	END OF GROUP 1 TOOL CHANGE SIGNAL			
#1206							RPDO	SINVA ₁
							RAPID TRAVERSE OUTPUT	SPINDLE REVERSE OUTPUT
#1207					S IDXA	SIDXO	SLPS	G93M
					END OF SPINDLE INDEXING OUTPUT	DURING SPINDLE INDEXING OUTPUT	SPINDLE LOOP STATUS OUTPUT	G93 MODE OUTPUT
#1208	WSFTER	WSFTK						
		END OF WORK COORDINATE SYSTEM SCREEN SETTING						

	'7	'6	'5	'4	'3	'2	'1	'0
#1209	MANINTK					FSCLRK	FSCE	FSMD
	END OF MACHINING INTERRUPTION POINT RETURN SIGNAL					END OF FS DATA CLEAR	END OF FS MEMORY CHANGE	UNDER FS EDITING MODE
#1210							SEGOUTS	SEGDNC
						DURING DNC HIGH-SPEED CUTTING PULSE DISTRIBUTION	DURING DNC HIGH-SPEED CUTTING COMMUNICATION	
#1211					MIMGM α	MIMGMZ	MIMGMY	MIMGMX
	DURING MIRROR IMAGE MODE							
#1212								
#1213								
#1214								
#1215	G84S							
	G74/G84 EXECUTION							
#1216	T8/T28	T7/T24	T6/T22	T5/T21	T4/T18	T3/T14	T2JT12	T1/T11
	T-FUNCTION BINARY/BCD OUTPUT							
#1217	T16/T48	T15/T44	T14/T42	T13/T41	T12/T38	T11/T34	T10/T32	T9/T31
	T-FUNCTION BINARY/BCD OUTPUT							

Table A4.2 List of Standard Output Signals (Cont'd)

	D ₇	'6	'5	'4	'3	'2	'1	'0
#1218	TAP	M04S	TLMO	G80S	EREND	ESEND	RST	AL
	TAPPING	SPINDLE	TOOL LENGTH MEASURE- MENT	CHANNED CYCLE	EXTERNAL DATA INPUT COMPLET- ED	EXTERNAL DATA INPUT COMPLET- ED	RESET	ALARM
#1219	SRV	SSP	FMF	EF	BF	TF	SF	MF
	SPINDLE REVERSE	SPINDLE STOP	MF	EXTERNAL OPERATION	B-FUNC- TION	T-FUNC- TION	S-FUNC- TION	M-FUNC- TION
	FOR CANNED CYCLE							
#1222	Ma	M7	M6	M5	M4	M3	M2	M1
	M-FUNCTION BINARY OUTPUT							
#1223	Os	EDTS	IER	4NGC	AUTO	w	RDY	RWD
	ORIENTA- TION	EDITING ERROR	INPUT ERROR	4TH AXIS NEBLECT	AUTO- MATIC	MANUAL	PREPARA- TION COMPLETED	REWIND
#1224	SDA8/ SB8	SDA7/ SB7	SDA6/ SB6	SDA5/ SB5	SDA4/ SB4	SDA3/ SB3	SDA2/ SB2	SDA1/ SB1
	S 5-DIGIT ANALOG OUTPUT/ s 4-DIGIT 12-BIT NON-CONTACT OUTPUT							
#1225	SDA16	SDA15	SDA14	SDA13	SDA12/ SB12	SDA11/ SB11	SDA10/ SB10	SDA9/ SB9
	S 5-DIGIT ANALOG OUTPUT/ s 4-DIGIT 12-BIT NON-CONTACT OUTPUT							
#1232	B8/B28	B7/B24	B6/B22	B5/B21	B4/B18	B3/B14	B2/B12	B1/B11
	B-FUNCTION BINARY/BCD OUTPUT							
#1233	B16/B48	B15/B44	B14/B42	B13/B41	B12/B38	B11/B34	B10/B32	B9/B31
	B-FUNCTION BINARY/BCD OUTPUT							
#1234	S28	s24	s22	S21	s18	S14	S12/GRH	S11/GRL
	S-FUNCTION BCD OUTPUT						HIGH- SPEED GEAR	LOW- SPEED GEAR

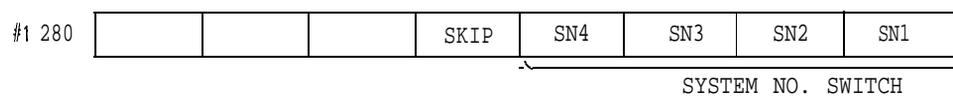
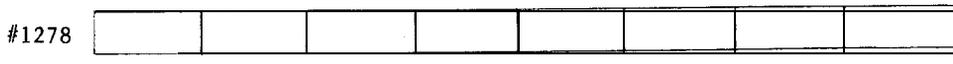
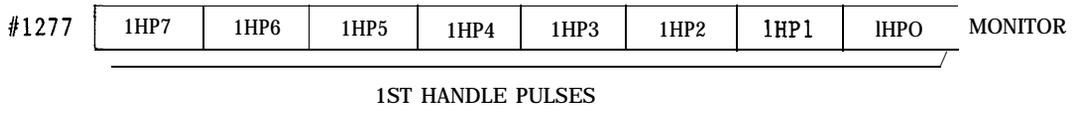
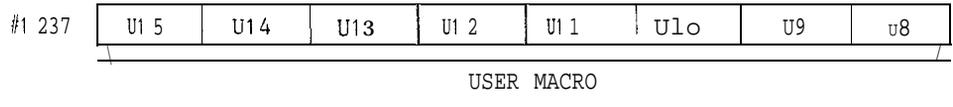
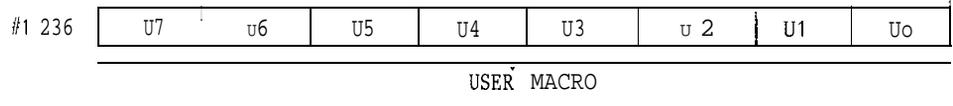
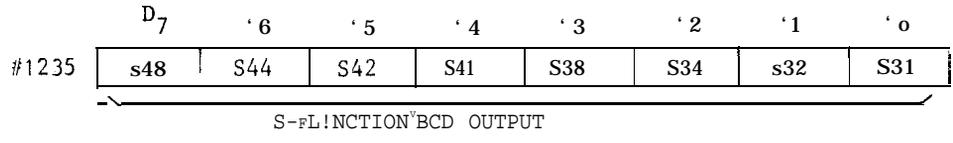


Table A4.2 List of Standard Output Signals (Cent'd)

	'7	'6	'5	'4	'3	'2	'1	'0
#1281		OFFPB		ON-PB	OLD	SVALM	ESP	OHT
		POWER OFF PUSHBUTTON		POWER ON PUSHBUTTON	OVERLOAD	SERVO ALARM	EMER- GENCY STOP	OVERHEAT
#1282	XSD	BALM		1	EXAXIS	PAGEO		EXCMOS
	SD20 0: NOT PROVIDED 1: PROVIDED	BATTERY ALARM		CONSTANT "1"	OPTION AXIS 0: NO SR51 BOARD 1: SR51 BOARD		CONTACT SIGNAL MONITOR FOR MAINTENANCE	
#1283					SNS4	SNS3	SNS2	SNS1
					SYSTEM NO. SWITCH			
#1284	SVON	NRD						
	SERVO NC POWER READY ON							
#1285	0	0	0	0	0	0	0	1 1
	CONSTANTS "1"							
#1286	0	0	0	0	0	0	0	0
	CONSTANTS "0"							
#1287						Pcs	PBS	PAS
						PG MONITOR FOR SPINDLE		
#1288	TGONX	PCX	PBX	PAX	*ALX	*OLX	FUX	SRDX
	PG MONITOR FOR X-AXIS				SERVO UNIT MONITOR FOR X-AXIS			
#1289	TGONY	PCY	PBY	PAY	*ALY	*OLY	FUY	SRDY
	PG MONITOR FOR Y-AXIS				SERVO UNIT MONITOR FOR Y-AXIS			

	D ₇	'6	'5	D ₄	'3	'2	D ₁	'0
#1290	TGONZ	PcZ	PBZ	PAZ	*ALZ	*OLZ	FUZ	SRDZ
	PG MONITOR FOR Z-AXIS				SERVO UNIT MONITOR FOR Z-AXIS			

#1291	TGON4	PC4	PB4	PA4	*AL4	*OL4	FU4	SRD4
	PG MONITOR FOR 4TH AXIS				SERVO UNIT MONITOR FOR 4TH AXIS			

#1292								
-------	--	--	--	--	--	--	--	--

#1293				ZNGC	ABS C	EDTLKC		
	Z-AXIS MANUAL EDIT				NEG- ABSOLUTE LOCK			
	LECT							

} SETTING MONITOR

#1294	AFLC	MLKC	OPTC	DRNC	BTDC	DLKC	STLKC	SBKC
	AUX FUNC- TION LOCK	MACHINE LOCK	OPTION- NAL STOP	DRY RUN	OPTIONAL BLOCK SKIP	DISPLAY LOCK	START LOCK	SINGLE BLOCK

#1295			PLBKC		MI _α C	MIZC	MIYC	MIXC
					MIRROR IMAGE AXIS			

#1297	ALM28	ALM24	ALM22	ALM21	ALM18	ALM14	ALM12	ALM11
	ALARM CODE MONITOR (LOWER 2-DIGIT BCD CODE)							

#1298					ALM38	ALM34	ALM32	ALM31
					ALARM CODE MONITOR (UPPER 1-DIGIT BCD CODE)			

APPENDIX-5 LIST OF ALARM **CODES**

Table A5. 1

Alarm No.	Spindle Operation	Type of Alarm
000 to 099	Stop at block end	Tape format error warning
100 to 199	Stop at block end	Macro, operation, external input/ output error, sequence errors (1) (2) (3)
200 to 299	Decelerate to stop	Overtravel, reference point return, positioning, machine ready
300 to 399	Decelerate to stop	servo, emergent y stop , overload FG, RPG

Table A5.2 List of Alarm Codes

Code	Causes	Code	Causes
000	POWER OFF TURN OFF POWER ONCE AT REFERENCE POINT RETURN ERROR	010	TH ERROR TAPE HORIZONTAL PARITY ERROR
001	ZR UNREADY (X) REFERENCE POINT RETURN NOT COMPLETED X	011	TV ERROR TAPE VERTICAL PARITY ERROR
002	ZR UNREADY (Y) REFERENCE POINT RETURN NOT COMPLETED Y	012	OVERFLOW (128CH) BUFFER CAPACITY OVERFLOW IN A BLOCK (128 CHARACTERS)
003	ZR UNREADY (Z) REFERENCE POINT RETURN NOT COMPLETED Z	013	PROG ERROR (NO ADDRESS) ADDRESS PLUS NO DATA AND NEXT ADDRESS COMMANDED, OR NO ADDRESS PLUS DATA
004	ZR UNREADY (4) REFERENCE POINT RETURN NOT COMPLETED 4	014	PROG ERROR (" ", "O") SIGN " ", "O" NOT CORRECTLY USED
005		015	PROG ERROR (UNUSABLE CH) UNUSABLE CHARACTER PROGRAMMED IN INSIGNIFICANT DATA AREA
006		016	PROG ERROR (UNUSABLE AXIS) INPUTS A, B, C, U, V, W DIFINED AS ADDITIONAL AXIS OR B-FUNCTION
007		017	PROG ERROR (8 DIGITS) INPUT DATA OVERFLOW
008		018	
009		019	

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
020	PROG ERROR (G) UNUSABLE G CODE OR G CODE NOT INCLUDED IN OPTIONS PROGRAMMED	030	PROG ERROR (F) NO F-COMMAND IN FEED COMMAND
021	PROG ERROR (G) G CODE IN 1, 4, *GROUP PROGRAMMED SIMULTANEOUSLY IN A BLOCK	031	PROG ERROR (R = O) CIRCLE WITH RADIUS O COMMANDED IN CIRCULAR ARC COMMAND
022	PROG ERROR (G02/03, G43/44) G43, G44 COMMANDED IN CIRCULAR INTERPOLATION MODE (G02, G03)	032	PROG ERROR (G02/03) COMMANDS ON THREE AXES IN CIRCULAR ARC COMMAND WITHOUT HELICAL INTERPOLATION OPTION
023	PROG ERROR (G) UNUSABLE G CODE COMMANDED IN CANNED CYCLE	033	PROG ERROR (G02/03) COMMAND ON MORE THAN FOUR AXES IN CIRCULAR ARC PLANE WHOSE ARC CANNOT BE SELECTED FROM THE COMMAND
024	PROG ERROR (G, G41/42) UNUSABLE CODE COMMANDED DURING COMPENSATION MODE	034	PROG ERROR (G02/03) CIRCULAR ARC R DESIGNATION ERROR
025	PROG ERROR (G70/71/72) G70 TO G72 COMMANDED EXCEPT IN CANNED CYCLES	035	PROG ERROR (D, H) TOO LARGE NO. OF H OR D CODE FOR TOOL RADIUS COMPENSATION AND TOOL LENGTH COMPENSATION
026	PROG ERROR (G41/42) RISE ERROR AT TOOL RADIUS COMPENSATION C (COMMAND WHICH CANNOT BE ACCOMMODATED CORRECTLY COMPENSATION G MODE) (SEE NOTES).	036	PROG ERROR (G10-P) TOO LARGE P (NUMBER DESIGNATION) WHEN OFFSET IS PROGRAM-INPUT
027	PROG ERROR (G41/42) ERROR AT TOOL RADIUS COMPENSATION C (ERROR IN CIRCULAR INTERPOLATION MODE)	037	PROG ERROR (G10-R) TOO LARGE R WHEN WORK COORDINATE SYSTEM IS PROGRAM-INPUT
028	PROG ERROR (G77R) REFERENCE POINT RETURN COMMAND AT G77	038	PROG ERROR (G10, Q2) TOO LARGE P WHEN WORK COORDINATE SYSTEM IS PROGRAM-INPUT
029	PROG ERROR (G106) THE GROUP 01 G CODE COMMANDED IN THE CANNED CYCLE IS OTHER THAN G01.	039	PROG ERROR (G106 I, J, X, Y) THE I OR J COMMAND CANNOT BE FOUND. BOTH I AND J ARE COMMANDED. AXIS OTHER THAN THE X OR Y AXES IS COMMANDED.
Notes :			
1. No move command in three blocks in series at G41 (G42) command.			
2. M00 commanded when rise.			
3. Rise at circular interpolation block.			

Code	Causes	Code	Causes
040	PROG ERROR (M98, G65/66) P NOT PROGRAMMED IN M98, G65, G66, G25. . P DIFFERENT FROM Q IN NO. IN G25 MODE, • G25 AND M98/M99 PROGRAMMED SIMULTANEOUSLY.	049	PROG ERROR (G41/42) REVERSE OR ALMOST REVERSE COMMANDED IN M97 MODE.
		050	SCALING ERROR UNUSABLE G CODE (G92, G28 TO G30, G36 TO G38, G70 TO G72) IN SCALING MODE.
041	PROG ERROR (NO PROG) PROGRAM NO. (SEQUENCE NO.) NOT FOUND WHEN PROGRAM IS CALLED BY M91, M98, M99, G65, G66, G25, G, M, T.	051	SCALING ERROR ERROR IN G51 AND G50 BLOCK FORMAT . SCALING FACTOR ZERO.
042	PROG ERROR (M98, NEST) SUBPROGRAM (G25) OR MACRO CALL FIVE-NESTED.	052
043	PROG ERROR (G52 Q2) LOCAL COORDINATE SYSTEM SPECIFIED WITH WORK COORDINATE NOT SET.	053
044	PROG ERROR (G12/13) IN CIRCLE CUTTING, PROGRAMMED RADIUS R IS SMALLER THAN COMPENSATION D .	054
045	CAL ERROR (G41/42) CALCULATION IMPOSSIBLE DURING TOOL RADIUS COMPENSATION MODE C .	055	PROG ERROR (M, S, T, B) M, S, T, B COMMANDS IN THE BLOCK IN WHICH M, S, T, B CODE CANNOT BE COMMANDED.
046	PROG ERROR (G41/42) DURING TOOL RADIUS COMPENSATION C MODE, CIRCULAR ARC OUTSIDE OF COMPENSATION PLANE PROGRAMMED .	056	PROG ERROR (AXIS) AXIS COMMAND IN G04, G20, AND G21 BLOCKS.
047	PROG ERROR (G41/42) COMPENSATION PLANE CHANGED DURING COMPENSATION C MODE.	057	PROG ERROR (S COM) S COMMAND EXCEEDS #6274. DISREGARDED WHEN #6274 = 0
048	PROG ERROR (G41/42) INTERSECTION POINT NOT OBTAINED BY INTERSECTION COMPUTATION.	058	MIRROR IMAGE (G28) G28/G30 COMMANDED DURING MIRROR IMAGE.

Table A5.2 List of Alarm Codes (Con'd)

Code	Causes	Code	Causes
059	ZR UNREADY REFERENCE POINT RETURN NOT COMPLETED ON THE AXIS WHICH HAS G29 COMMAND OR G30 COMMAND*	069	
060		070	PROG ERROR (M02/30/99) MEMORY OPERATION FINISH COMMAND NOT GIVEN .
061		071	
062		072	
063		073	
064		074	
065		075	RS232C ERROR (BAUD RATE) RS232C INTERFACE DISAGREEMENT OF NO. OF BITS AND NO. OF BAUD RATES.
066	RESET UNREADY (AFTER EDITING) CYCLE-STARTED WITHOUT RESETTING AFTER EDITING .	076	RS232C ERROR (SIGNAL LEVEL) DATA TRANSMISSION FAILURE THROUGH RS232C INTERFACE.
067		077	RS232C ERROR (OVERRUN) 10 CHARACTERS MORE HAVE BEEN READ IN AFTER STOP CODE HAS BEEN TRANSMITTED THROUGH RS232C INTERFACE.
068	EDITING INHIBIT AREA EDITING ATTEMPTED IN PROHIBITED AREA.		

Code	Causes	Code	Causes
078		088	PROG ERROR (G36/37/38) TOUCH SWITCH CALCULATION ERROR AT G36 TO G38 COMMANDS.
079		089	
080	PROG ERROR (G10, G22/23) G10, G22, AND G23 COMMANDED WITH AXIS DATA.	090	PROG ERROR (G93) A CODE THAT CANNOT BE COMMANDED DURING THE BLOCK OF G93 OR G93 MODE HAS BEEN COMMANDED.
081			
082		091	TAP ERROR SLPC (SPINDLE LOOP COMMAND) HAS NOT BEEN TURNED ON DURING EXE- CUTION OF SOLID TAPPING.
083		092	
084	MIRROR IMAGE (G36/37/38) MIRROR IMAGE IS ON WITH G36 TO G38.	093	
085	PROG ERROR (G36/37) COMMAND OF I (J) ON MORE THAN ONE AXIS AT G36 (G37).	094	
086	PROG ERROR (G38) COMMAND OTHER THAN K AT G38	095	
087	PROG ERROR (G31, G36/37/38) TOUCH SWITCH NOT ON WHEN MOTION REACHES AT END POINT BY G31, G36 TO G38 COMMANDS.	096	

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
097		107	MACRO ERROR (FORMAT) ERROR IN THE FORMAT AND EQUATION.
098		108	MACRO ERROR (UNDEFIN #NO) UNDEFINED VARIABLE NO. DESIGNATED.
099		109	MACRO ERROR (#NO NOT LEFT) COMMANDED PROHIBITED VARIABLE AS SUBSTITUTION IN LEFT-HAND SIDE OF THE EQUATION.
100	CAL ERROR (FIXED POINT) MAGNITUDE OF FIXED POINT DATA EXCEEDING UPPER LIMIT	110	MACRO ERROR ([1 ^S LIMIT) MULTIPLE LAYERS OF PARENTHESES EXCEEDING THE UPPER LIMIT (5) .
101	CAL ERROR (FLOATING) EXPONENT OF FLOATING POINT DATA EXCEEDING ALLOWABLE RANGE	111	MACRO ERROR (MOVE G66-M99) MOVE COMMAND IN M99 FINISHING COMMAND OF MACRO CALLED BY G66.
102	CAL ERROR (DIVISION) CALCULATION DIVISOR 'ZERO OR OVERFLOW ERROR' .	112	MACRO ERROR (5) LEVEL FOR CALLING MACRO EXCEEDING FOUR LEVELS.
103	CAL ERROR (SQUARE ROOT) ROOT VALUE IS A NEGATIVE (-).	113	MACRO ERROR (6)
104	PROG ERROR (DOUBLE ADD) SAME ADDRESS REPEATED IN A BLOCK.	114	MACRO ERROR (DO-FORMAT) NO. OF DOs AND ENds NOT THE SAME.
105	MACRO ERROR (CONSTANT) CONSTANTS USABLE IN USER MACRO EXCEEDING THE LIMIT.	115	MACRO ERROR ([]UNMATCH) NO. OF LEFT BRACKETS AND RIGHT BRACKETS NOT THE SAME.
106	MACRO ERROR TOO MANYG CODES COMMANDED FOR CANCELLING G67.	116	MACRO ERROR (DO-END NO.) CONDITION $1 \leq m \leq 3$ NOT ESTABLISHED IN DOm,

Code	Causes	Code	Causes
117	MACRO ERROR (10)	127	LIFE CTRL ERROR (T5 & T9999) ENTERED COMMAND FOR T-5 DIGITS WITHOUT REGISTERING THE GROUP OR THERE IS AN ERROR IN THE T9999L AAA COMMAND. T9999L AAA COMMANDED ALTHOUGH THIS SETTING SHOULD BE COUNTED BY M20/M30.
118	MACRO ERROR (GOTO N) CONDITION $0 \leq n \leq 9999$ NOT ESTABLISHED OR NO SEQUENCE NO. IN GOTO n.		
119	MACRO ERROR (12)	128	LIFE CTRL ERROR (T5 & H999) ENTERED COMMAND FOR T5 DIGITS ALTHOUGHT ALL OF THE CONTENTS OF THE GROUP WERE IN SKP STATUS, OR ENTERED H(D)999 COMMAND WITHOUT REGISTERING TOOL NUMBERS H-NO AND D-NO.
120	PRTN ERROR (NOT FOUND) SEQUENCE NO. NOT FOUND IN PART PROGRAM.		
121	PRTN ERROR (G92) G92 COMMANDED THROUGH MDI OPERATION DURING PROGRAM RESTART .	129	PROG ERROR (G54 TO G59 J) SPECIFIED AN "EXCESSIVELY LARGE J IN THE WORK COORDINATE SYSTEM OR THE MODE WAS OTHER THAN G00/G01.
122	PRTN ERROR (G54-G59) G54 TO G59 COMMANDED THROUGH MDI OPERATION DURING PROGRAM RESTART.		
123	PRTN ERROR (ORG) COORDINATE SYSTEM CHANGED BY DEPRESSING THE CRT OPERATION CONTROL PANEL ORG KEY DURING PROGRAM RESTART .	130	
124	PRTN ERROR (MDI MOVE) AXIS OPERATED BY MDI AFTER PROGRAM RESTART .	131	
125	PROG ERROR (G122/123/124) ERROR EXISTS IN SPECIFICATION OF P, I, T, L, H, and D WITH COMMANDS G122/123.	132	
126	PROG ERROR (G122 DATA OVR) TOOL LIFE CONTROL DATA COMMANDS EXCEEDED CAPACITY.	133	

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
134	NO PROG (EXT) 0 NO. NOT FOUND WHEN EXECUTING WORK NO. SEARCH A COMMAND.	144	
135		145	
136		146	
137		147	
138		148	AS AFTER READ SUBPROG/MACRO AN ADDRESS SEARCH OPERATION WAS MADE DURING THE EXECUTION OF SUBPROGRAMS.
139		149	
140		150	ROTATE ERROR (G CODE) G CODE THAT CANNOT BE USED IN G68 MODE HAS BEEN COMMANDED. G68 HAS BEEN COMMANDED DURING TOOL RADIUS COMPENSATION C.
141			
142		151	ROTATE ERROR (FORMAT) THERE IS AN ERROR IN THE FORMAT OF G68 OR G69 COMMAND BLOCK.
143		152	ROTATE ERROR COORDINATE ROTATION COMMAND (G54) IS COMMANDED DURING COMPENSATION C, IN THE COORDINATE ROTATION OF THE EASY SETTER SPECIFICATION. OR PLANE OTHER THAN THE X, Y PLANES IS COMMANDED.

Code	Causes	Code	Causes
153		163	
154		164	
155		165	
156		166	
157		167	
158		168	
159		169	
160		170	MEM ERROR (OFS) TOOL OFFSET TOTAL CHECK ERROR
161		171	
162		172	MEM ERROR (SET) SETTING AREA TOTAL CHECK ERROR.

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
173	MEM ERROR (PRM) PARAMETER AREA TOTAL CHECK ERROR .	183	
174	MEM ERROR (KEEP) TOTAL CHECK ERROR OF MACRO VARIABLES.	184	
175	MEM ERROR (MACR) TOTAL CHECK ERROR OF KEEP MEMORY.	185	
176		186	
177		187	
178		188	
179	OVER TEMP PANEL INSIDE TEMPERATURE TOO HIGH .	189	
180	SEQ ERROR 1 SEQUENCER ERROR (1)	190	
181	SEQ ERROR 3 SEQUENCER ERROR (3) IMMEDIATE STOP	191	
182	SEQ ERROR 2 SEQUENCER ERROR (2) DECELATION STOP	192	

Code	Causes	Code	Causes
193		203	OT (Z) OVERTRAVEL Z
194		204	OT (4) OVERTRAVEL 4
195		205	-
196		206	
197		207	
198		208	
199		209	
200		210	
201	OT (X) OVERTRAVEL X	211	S-OT1 (X) STORED STROKE LIMIT FIRST AREA X .
202	OT (Y) OVERTRAVEL Y	212	S-OT1 (Y) STORED STROKE LIMIT FIRST AREA Y.

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
213	S-OT1 (Z) STORED STROKE LIMIT FIRST AREA Z .	223	S-OT2 (Z) STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) Z .
214	S-OT1 (4) STORED STROKE LIMIT FIRST AREA 4.	224	S-OT2 (4) STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) 4.
215		225	
216		226	
217		227	
218		228	
219		229	
220	S-OT2 (INSIDE) STORED STROKE LIMIT SECOND AREA (INSIDE INHIBIT) .	230	TOOL BROKEN "AT BROKEN TOOL DETECTION BY G32, G33, Z-AXIS MOVES TO THE SET POSITION BUT SENS (T) INPUT NOT TURNED ON. (SPECIAL SPECIFICATIONS)
221	S-OT2 (X) STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) X.		
222	S-OT2 (Y) STORED STROKE LIMIT SECOND AREA (OUTSIDE INHIBIT) Y.	231	ZR ERROR-AREA (X) REFERENCE POINT RETURN AREA ERROR X. THEORETICAL C PHASE TYPE LATCH UNFINISHED.

Code	Causes	Code	Causes
232	ZR ERROR-AREA (Y) REFERENCE POINT RETURN AREA ERROR Y.	242	ZR ERROR-POS (Y) REFERENCE POINT RETURN AREA ERROR Y,
233	ZR ERROR-AREA (Z) REFERENCE POINT RETURN AREA ERROR Z .	243	ZR ERROR-POS (Z) REFERENCE POINT RETURN AREA ERROR Z .
234	ZER ERROR-AREA (4) REFERENCE POINT RETURN AREA ERROR 4.	244	ZR ERROR-POS (4) REFERENCE POINT RETURN AREA ERROR 4.
235		245	
236		246	
237		247	
238		248	
239		249	
240		250	S-OT3-5 (INSIDE) STORED STROKE LIMIT 3RD 4TH, 5TH AREA INSIDE PROHIBITED .
241	ZR ERROR-POS (X) REFERENCE POINT RETURN AREA ERROR X .	251	S-OT3-5 (x) STORED STROKE LIMIT 3RD , 4TH, 5TH AREA OUTSIDE PROHIBITED X .

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
252	S-0T3-5 (Y) STORED STROKE LIMIT 3RD, 4TH, 5TH AREA OUTSIDE PROHIBITED Y .	262	ZR ERROR PRM (Y) RETRACTION BY REFERENCE POINT RETURN (ER PULSE AT C PHASE LATCH > #6305)
253	S-0T3-5 (Z) STORED STROKE LIMIT 3RD , 4TH, 5TH AREA OUTSIDE PROHIBITED Z.	263	ZR ERROR PRM (Z) RETRACTION BY REFERENCE POINT RETURN (ER PULSE AT C PHASE LATCH > #6306)
254		264	ZR ERROR PRM (4) RETRACTION BY REFERENCE POINT RETURN (ER PULSE AT C PHASE LATCH > #6307)
255	ZR DECLS ERROR (X) LS ERROR (X) DECELERATION (DECLS OF X-AXIS STARTS CHAT- TERING ON THE DOG)	265	
256	ZR DECLS ERROR (Y) LS ERROR (Y) DECELERATION (DECLS OF Y AXIS STARTS CHAT- TERING ON THE DOG)	266	
257	ZR DECLS ERROR (Z) LS ERROR (Z) DECELERATION (DECLS OF Z AXIS STARTS CHAT- TERING ON THE DOG)	267	
258	ZR DECLS ERROR (4) LS ERROR (4) DECELERATION (DECLS OF 4 AXIS STARTS CHAT- TERING ON THE DOG)	268	
259		269	
260		270	
261	ZR ERROR PRM (X) RETRACTION BY REFERENCE POINT RETURN (ER PULSE AT C PHASE LATCH > #6304)	271	P-SET ERROR (X) P SET ERROR X IF THIS ERROR OCCURS '-----S IN BLOCK WHERE GOO IS CHANGING TO GO 01, MACHINE STOPS AFTER COMPLETION OF GOO BLOCK).

Code	Causes	Code	Causes
272	P-SET ERROR (Y) P SET ERROR Y (IF THIS ERROR OCCURS IN BLOCK WERE GOO IS CHANGING TO G01, MACHINE STOPS AFTER COMPLETION OF GOO BLOCK).	282	ZR POSITION ERROR (Y) REFERENCE POINT RETURN COMPLETED WITH REFERENCE POINT LS OFF.
273	P-SET ERROR (Z) P SET ERROR Z (IF THIS ERROR OCCURS IN BLOCK WHERE GOO IS CHANGING TO GO I , MACHINE STOPS AFTER COMPLETION OF GOO BLOCK).	283	ZR POSITION ERROR (Z) REFERENCE POINT RETURN COMPLETED WITH REFERENCE POINT LS OFF.
274	P-SET ERROR (4) P SET ERROR 4 (IF THIS ERROR OCCURS IN BLOCK WHERE GOO IS CHANGING TO G01 , MACHINE STOPS AFTER COMPLETION OF GOO BLOCK).	284	ZR POSITION ERROR (4) REFERENCE POINT RETURN COMPLETED WITH REFERENCE POINT LS OFF.
275		285	
276	P-SET ERROR (S) PROGRAM SET ERROR S. SPINDLE P-SET ERROR AT SOLID TAPPING.	286	
277		287	
278		288	
279		289	
280	MACHINE UNREADY MACHINE NOT READY	290	
281	ZR POSITION ERROR (X) REFERENCE POINT RETURN COMPLETED WITH REFERENCE POINT LS OFF.	291	

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
292		302	
293		303	
294		304	
295		305	
296		306	
297		307	
298		308	
299		309	
300		310	SERVO OFF SERVO POWER NOT SUPPLIED
301		311	X-AXIS SERVO UNREADY X-AXIS SERVO NOT READY

Code	Causes	Code	Causes
312	Y-AXIS SERVO UNREADY Y-AXIS SERVO NOT READY	322	CMOS VACANT AREA ERROR THE UNUSED AREA OF THE CMOS IS USED. (THE CMOS MAY BE DESTROYED.) (SET "1" IN #6066 D2 TO RESET THE ERROR.)
313	Z-AXIS SERVO UNREADY Z-AXIS SERVO NOT READY	323	
314	4TH AXIS SERVO UNREADY 4TH AXIS SERVO NOT READY	324	
315		325	SERVO CPU ERROR SERVO CPU IS IN ABNORMAL STATE. (SET "0" IN #6104 D6 TO RESET THE ERROR).
316		326	SKIP MACRO (G, M, T) CALLING OF G, M, T MACRO OF THE MAIN PROGRAM WAS SKIPPED (CHECK MODE) .
317			
318		327	NO PROG GEN (RES→NEXT→PRG) MEMORY LENGTH , FOR WHICH MEMORY GENERATION WAS NOT PERFORMED , HAS BEEN CHANGED BUT NO GENERATION HAS BEEN MADE. (RES→NEXT→PRG)
319			
320	NC UNREADY NC UNREADY P SET NOT READY	328	UNSUITABLE MM51 MEMORY LENGTH DOES NOT MATCH THE MEMORY BOARD.
321		329	PC CPU ERROR PC CPU IS IN ABNORMAL STATE. (SET "0" IN #6014 D7 TO RESET THE ERROR).

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
330	EMERGENCY STOP EMERGENCY STOP	340	
331	FUSE (X) FUSE BLOWN X	341	SERVO ERROR (X) SERVO ERROR X
332	FUSE (Y) FUSE BLOWN Y	342	SERVO ERROR (Y) SERVO ERROR Y
333	FUSE (Z) FUSE BLOWN Z	343	SERVO ERROR (Z) SERVO ERROR Z
334	FUSE (4) FUSE BLOWN 4	344	SERVO ERROR (4) SERVO ERROR 4
335		345	
336		346	SERVO ERROR (S) SERVO ERROR (SPINDLE)
337		347	
338		348	
339		349	

Code	Causes	Code	Causes
350		360	
351	OL (X) OVERLOAD (1) X	361	PG ERROR (X) PG ERROR X
352	OL (Y) OVERLOAD (1) Y	362	PG ERROR (Y) PG ERROR Y
353	OL (Z) OVERLOAD (1) Z	363	PG ERROR (Z) PG ERROR Z
354	OL (4) OVERLOAD (1) 4	364	PG ERROR (4) PG ERROR 4
355		365	
356	OL (S) OVERLOAD (1) (S)	366	PG ERROR (S) PG ERROR SPINDLE
357	OL (OTHERS) OVERLOAD (2) (CONTACTOR UNIT) CEILING RESISTANCE, OTHERS	367	
358		368	
359		369	

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
370		380	
371		381	AXIS ERROR (X) AXIS CPU ERROR
372		382	AXIS ERROR (Y) AXIS CPU ERROR
373	SR-51 UNSUITABLE OPTIONAL AXIS SETTING DOES NOT MATCH THE TYPE OF SR-51.	383	AXIS ERROR (Z) AX-IS CPU ERROR
374	YENET 1/0 ERROR COMMUNICATION ERROR BETWEEN 1/0 BOARD OR SP50 BOARD AND PC50	384	AXIS ERROR (4) AXIS CPU ERROR
375	YENET 1/0 BOARD ERROR . 1/0 BOARD OR SP BOARD TYPE IS WRONG. . 1/0 BOARD OR SP BOARD CHANNEL SETTING IS INCORRECT.	385	
376		386	AXIS ERROR (S) AXIS CPU ERROR
377		387	
378		388	
379		389	

Code	Causes	Code	Causes
390	SERVO ALARM (SERVOPACK) (SR20) SERVOPACK ALARM SPINDLE PG DISCONNECTION OF 4/5 AXIS SR FAILURE	800	
391	SERVO ALARM (X) X-AXIS SERVOPACK ALARM X	801	
392	SERVO ALARM (Y) Y-AXIS SERVOPACK ALARM Y	802	
393	SERVO ALARM (Z) Z-AXIS SERVOPACK ALARM Z	803	
394	SERVO ALARM (4) 4TH AXIS SERVOPACK ALARM 4	804	
395		805	
396		806	
397		810	
398		811	
399		812	

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
813		823	
814		824	
815		825	
816		826	
817		827	
818		828	
819		829	
820	ROM ERROR · ROM CHECK ERROR · RAM CHECK ERROR · REQUIRES HARDWARE CHECK.	830	
821		840	
822		841	

Code	Causes	Code	Causes
842		852	
843		853	
844			
846		856	
847		857	
848		858	
849		859	
850		860	
851		861	

Table A5.2 List of Alarm Codes (Cent'd)

Code	Causes	Code	Causes
862		872	
863		873	
864		874	
865		875	
866		876	
867		877	
868		878	
869		879	
870		880	
871		881	

Code	Causes	Code	Causes
882		892	
883		893	
884		894	
885		895	
886		896	
887		897	
888		898	
889		899	
890		900	
891		901	

Table A5.2 List of Alarm Codes (Cent'd)

Code I	Causes	Code	Causes
902		912	
903		913	
904		914	
905		915	
906		916	
907		917	
908		918	
909		919	
910		920	
911		921	

Code	Causes	Code	Causes
922		932	
923		933	
924		934	
925		935	
926		936	
927		937	
928		938	
929		939	
930			
931			

APPENDIX-6 LIST OF ADDRESS CHARACTERS

Table A6. 1 List of Address Characters

Address Characters	Meanings	B: Basic O: Optional
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	B, O
E	User macro character	O
F	Feedrate	B
G	Preparatory function	B, o
H	Tool length offset number	B
I	X-coordinate of arc center Radius for circle cutting	B o
J	Y-coordinate of arc center Cutting depth for circle cutting	B, O
K	Z-coordinate arc center	B
L	Number of repetitions	B, O
M	Miscellaneous functions	B
N	Sequence number	B
o	Program number	B
P	Dwell time, Program No. and sequence No. designation in subprogram	B o
Q	Depth of cut, shift of canned cycles	O
R	Point R for canned cycles Radius designation of a circular arc	o, B
s	Spindle-speed 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 AL.2 Function Characters

EIA Code	ISO Code	Meanings	Remarks
Blank	N ul	Error in significant data area in EIA Disregarded in ISO	
BS	BS	Disregarded	
Tab	HT	Disregarded	
CR	LF/NL	End of Block (EOB)	
/	CR	Disregarded	
SP	SP	Space	
ER	%	Rewind stop	
UC	/	Upper shift	
LC	/	Lower shift	
2-4-5 bits	(Control out (Comment start)	EIA : Special code
2-4-7 bits)	Control in (Comment end)	
+	+	Disregarded, User macro operator	
		Minus sign, User macro operator	
0 to 9	0 to 9	Numerals	
a to z	A to Z	Address characters, User macro operator	
I	I	Optional block skip, User macro operator	
Del	DEL	Disregarded (Including All Mark)	
	.	Decimal point	
Parameter setting	#	Sharp (Variable)	
*	*	Astrisk (Multiplication operator)	
=	=	Equal mark	
[[Left bracket	
]]	Right bracket	
0		User macro operator	
\$	\$	User macro operator	
@	@	User macro operator	
?	?	User macro operator	

Notes:

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

Table A6.3 Tape Code

EIA CODE									CHARACTERS	ISO CODE								
8	7	6	5	4	3	2	1			8	7	6	5	4	3	2	1	
			0						0			0	0					
				1				0	1			0	0				0	
					2				2			0	0				0	
						3			3			0	0				0	
							4		4			0	0				0	
								5	5			0	0				0	
									6			0	0				0	
									7			0	0				0	
									8			0	0				0	
									9			0	0				0	
									a	A							0	
									b	B							0	
									c	C							0	
									d	D							0	
									e	E							0	
									f	F							0	
									g	G							0	
									h	H							0	
									i	I							0	
									j	J							0	
									k	K							0	
									l	L							0	
									m	M							0	
									n	N							0	
									o	O							0	
									p	P							0	
									q	Q							0	
									r	R							0	
									s	S							0	
									t	T							0	
									u	U							0	
									v	V							0	
									w	W							0	
									x	X							0	
									y	Y							0	
									z	Z							0	
									Blank	NUL							0	
									9S								0	
									Tab	HT							0	
									CR	LF/NL							0	
									--	CR							0	
									SP								0	
									ER	%							0	
									UC	--							0	
									LC	--							0	
									--	(0	
									--)							0	
									+								0	
									--								0	
									o	:							0	
									/								0	
									Del	DEL							0	
									All Mark								0	
									See Note 2.	#							0	
									*								0	
									=								0	
									[00	
]								0	
									\$								0	
									@								0	
									?								0	
									.								0	

Notes:

1. For characters from # to ?, EIA codes have not been agreed upon. In the present system, for the time being, the above provisional codes are used.
2. EIA code of character # can be designated by the parameter #6017.

Table A6.4 Tape Format

No.	Address		Metric output		Inch output		B: Basic O: Optional
			Metric input	Inch input	Metric input	Inch input	
1	Program No.		04		04		B
2	Sequence No.		N4		N4		B
3	G function		G3		G3		B
4	Coordinate Word	Linear axis	a + 53	a + 44	a + 53	a + 44	B
		Rotary axis	b + 53	b + 53	b + 53	b + 53	O
5	Feed /rein		F50	F31	F50	F41	B
6	Feed/rein 1/10		F51	F32	F51	F42	B
7	S- function		S2		S2		B
			S5		S5		O
8	T- function		T2		T2		B
			T4		T4		O
9	M-function		M3		M3		B
10	Tool Offset No.		H2 or D2		H2 or D2		B
11	B- function		B3		B3		O
12	Dwell		P53		P53		B
13	Program No. designation		P4		P4		B
14	Sequence No. designation		P4		P4		B
15	No. of repetitions		L8		L8		B

Table A6.5 Range of Program Commands

No.	Address		Metric Output		Inch Output	
			Metric Input	Inch Input	Metric Input	Inch Input
1	Program Number	O	1 to 9999	1 to 9999	1 to 9999	1 to 9999
2	Sequence Number	N	1 to 9999	1 to 9999	1 to 9999	1 to 9999
3	G- function	G	0 to 199	0 to 199	0 to 199	0 to 199
4	Coordinate address					
	Linear axis		±99999.999 mm	±3937.0078 in.	*99999.999 mm	*9999.9999 in.
	Rotary axis		*99999.999 deg	±99999.999 deg	*99999.999 deg	*99999.999 deg
	Max cumulative value		±99999.999 mm	*9999.99999 in.	*99999.999 mm	*9999.99999 in.
5	Feed per minute	F	1 to 30000 mm/min	0.1 to 1181.1 in. /rein	1 to 76200 mm/min	0.1 to 3000.0 in. /rein
6	Feed per minute 1/10		0.1 to 30000.0 mm/min	0.01 to 1181.10 in. /rein	0.1 to 76200.0 mm/min	0.01 to 3000.00 in. / min
7	S-function	S2	0 to 99	0 to 99	0 to 99	0 to 99
		S5	0 to 99999	0 to 99999	0 to 99999	0 to 99999
8	T-function	T2	0 to 99	0 to 99	0 to 99	0 to 99
		T4	0 to 9999	0 to 9999	0 to 9999	0 to 9999
9	M-function	M	0 to 199	0 to 199	0 to 199	0 to 199
10	Tool offset No.	H	0 to 99 (299)	0 to 99 (299)	0 to 99 (299)	0 to 99 (299)
		D	0 to 99 (299)	0 to 99 (299)	0 to 99 (299)	0 to 99 (299)
11	B-function	B	0 to 999	0 to 999	0 to 999	0 to 999
12	Dwell	P	0 to 99999.999 sec	0 to 99999.999 sec	0 to 99999.999 sec	0 to 99999.999 sec
13	Program No. designation	P	1 to 9999	1 to 9999	1 to 9999	1 to 9999
14	Sequence No. designation	P	1 to 9999	1 to 9999	1 to 9999	1 to 9999
15	No. of repetitions	L	99999999	99999999	99999999	99999999

Table A6.6 Range of Data Setting

Address		Metric Output		Inch Output	
		Metric Input	Inch Input	Metric Input	Inch Input
Least input increment		0.001 or 0.01 mm	0.0001 or 0.001 in.	0.001 or 0.01 mm	0.0001 or 0.001 in.
Max stroke (Distance from reference point)		± 99999.999 mm		± 3937.0074 in.	± 9999.9999 in.
Tool offset amount & Tool radius value		0 to ± 999.999 mm	0 to ± 99.9999 in.	0 to ± 999, 999 mm	0 to ± 99.9999 in.
Min. feed amount at STEP /HANDLE		0.001 mm	0.0001 in.	0.001 mm	0.0001 in.
Unit of area setting for stored stroke limit	Program designation	0.001 mm	0.0001 in.	0.001 mm	0.0001 in.
	Parameter & setting	0.001 mm		0.0001 in.	
Rapid traverse rate		(7.5 to) 1 to 30000 mm/min		(0.75 to) 0.1 to 30000.0 in./min	
Manual jog speed					
Seed at Fo					
Max. feedrate					
Dry run speed		1 to 30000 mm/min		0.1 to 3000.0 in. /rein	
Stored leadscrew compensation, stored stroke limit, and setting area for 2nd to 4th reference points		0 to *99999.999 mm.		0 to ± 3937.0078 in.	0 to ± 9999.9999 in.
		(O : Reference point position)			
Backlash compensation		± 8191 pulses		± 8191 pulses	
Stored leadscrew Compensation	Incremental	0 to 15 pulses		0 to 15 pulses	
	Absolute	0 to 127 pulses		0 to 127 pulses	

Note: 1 pulse = least output increment

Table A6. 7 List of G Codes

G Code	Group	Function	B: Basic O: Optional
G00		Positioning	B
G01		Linear interpolation	B
G02	01	Circular interpolation CW, Helical interpolation CW	B, O
G03		Circular interpolation CCW, Helical interpolation CCW	B, O
G04		Dwell	B
G06		Positioning in error detect off mode	B
G09	*	Exact stop	E
G10		Tool offset value and work coordinate, Shift-value modification	B, O
G12		Circle cutting CW	o
G13		Circle cutting CCW	o
G17		XY plane designation	B
G18	02	ZX plane designation	B
G19		YZ plane designation	B
G20	06	Inch input designation	o
G21		Metric input designation	o
G22	04	Stored stroke limit ON	o
G23		Stored stroke limit OFF	o
G25	*	Program copy	o
G27		Reference point check	1 O
G28		Automatic return to reference point	1 O
G29	*	Return from reference point	o
G30		Return to 2nd, 3rd, 4th reference point	o
G31		Skip function	O
G40		Tool radius compensation cancel	O
G41	07	Tool radius compensation, left	O
G42		Tool radius compensation, right	O
G43		Tool length compensation, plus direction	B
G44	08	Tool length compensation, minus direction	B
G49		Tool length compensation, cancel	B
G45		Tool position offset, extension	B
G46		Tool position offset, retraction	B
G47	*	Tool position offset, double extension	B
G48		Tool position offset, double retraction	B
G50		Scaling OFF	O
G51	15	Scaling ON	o
G52	12	Return to base coordinate system	o
G53	*	Temporary shift to machine coordinate system	o
G54		Shift to work coordinate system 1	O
G55		Shift to work coordinate system 2	O
G56	12	Shift to work coordinate system 3	O
G57		Shift to work coordinate system 4	O
G58		Shift to work coordinate system 5	O
G59		Shift to work coordinate system 6	O

G Code	Group	Function	B: Basic O: Optional
G60	01	Unidirectional approach	o
G61		Exact stop mode	B
G64	13	Exact stop mode cancel	B
G65	*	Non-modal call of user macro	o
G66		Modal call of user macro	o
G67	14	Modal call of user macro cancel	o
G68		Rotation of coordinate ON	o
G69	18	Rotation of coordinate OFF	o
G70		Bolt hole circle	o
G71	*	Arc	o
G72		Line-at-angle	o
G73		Canned cycle 10	O
G74		Canned cycle 11	O
G76		Canned cycle 12	O
G77		Canned cycle 13	O
G80	09	Canned cycle cancel	o
G81		Canned cycle 1, Output for external motion	o
G82		Canned cycle 2	O
G83		Canned cycle 3	O
G84		Canned cycle 4	O
G85		Canned cycle 5	O
G86	09	Canned cycle 6	O
G87		Canned cycle 7	O
G*8		Canned cycle 8	O
G89		Canned cycle 9	O
G90	03	Absolute command designation	B
G91		Incremental command designation	B
G92	*	Programming of absolute zero point	o
G93		Solid tap mode ON	O
G94	05	Solid tap mode OFF	1 O
G98	10	Return to initial point for canned cycles	o
G99		Return to point R for canned cycles	O
G106	*	Automatic corner override	o
G122	17	Tool register start	o
G123		Tool register end	o
G124	*	Tool register cancel	o
G198	*	Form compensation mode ON	o
G199	*	Form compensation mode OFF	o

Notes :

1. The G codes in the * group are non-modal, and are effective only for the block in which they are commanded. They cannot be programmed twice or more in a block. They must be programmed only once in a block of its own.
2. The codes marked with ∇ is automatically selected at power on or reset.
3. The codes marked with ∇ is automatically selected upon power on.

MEMO

YASNAC J50M INSTRUCTIONS

TOKYO OFFICE OhtemachiBldg, 1-6-1 Ohtemachi, Chiyoda-ku, Tokyo, 100 Japan
Phone (03) 3284-9111 Telex YASKAWA J33530 Fax (03) 3284-9034
YASKAWA ELECTRIC AMERICA, INC.
Chicago-Corporate Headquarters 2942 MacArthur Blvd Northbrook, IL 60062-2028, U S A
Phone (708) 291-2340 Fax (708) 498-2430
Chicago-Technical Center 3160 MacArthur Blvd Northbrook, IL 60062-1917, U.S.A
Phone (708) 291-0411 Fax (708) 291-1018
MOTOMAN INC.
805 Liberty Lane West Carrollton, OH 45449, U S A
Phone (513) 847-6200 Fax (513) 847-6277
YASKAWA ELÉTRICO DO BRASIL COMÉRCIO LTDA.
Rua Conde Do Pinhal 8-5, Andar Sala 51 CEP 01501-São Paulo-SP, Brash
Phone (011) 35-1911 Fax (011) 37-7375
YASKAWA ELECTRIC EUROPE GmbH
Am Kronberger Hang 2, 65824 Schwalbach, Germany
Phone (49) 6196-569-300 Fax (49) 6196-888-301
Motoman Robotics AB
Box 130 S-38500. Torsås, Sweden
Phone 0486-10575 Fax 0486-11410
Motoman Robotec GmbH
Kammerfeldstraße 1, 85391 Allershausen, Germany
Phone 08166-900 Fax 08166-9039
YASKAWA ELECTRIC UK LTD.
3 Orum Mains Park Orchardton Woods Cumbernauld, Scotland, G68 9LD U K.
Phone (1236) 735000 Fax (1 236) 458182
YASKAWA ELECTRIC KOREA CORPORATION
Paik Nam Bldg. **901 188-3**, 1-Ga Euijro, Joong-Gu Seoul, Korea
Phone (02)776-7844 Fax (02)753-2639
YASKAWA ELECTRIC (SINGAPORE) PTE. LTD.
Head Office : CPF Bldg, 79 Robinson Road # 13-05, Singapore 0106, SINGAPORE
Phone 221-7530 Telex (87) 24890 YASKAWA RS Fax 224-5854
Service Center : **221 Henderson Road, # 07-20 Henderson Building Singapore 0315, SINGAPORE**
Phone 276-7407 Fax 276-7406
YATEC ENGINEERING CORPORATION
Shen Hsiang Tang Sung Chiang Building 10F 146 Sung Chiang Road, Taipei, Taiwan
Phone (02) 563-0010 Fax (02) 567-4677
SHANGHAI OFFICE Room No. 8B Wan Zhong Building 1303 Yan An Road (West), Shanghai 200050, CHINA
Phone (86) 212-1015 Fax (86) 212-1015
TAIPEI OFFICE Shen Hsiang Tang Sung Chiang Building 10F 146 Sung Chiang Road, Taipei, Taiwan
Phone (02) 563-0010 Fax (02) 567-4677



YASKAWA ELECTRIC CORPORATION

TOE-C843-I 2.30

© Printed in Japan April 1995 95-4 0.3TA
9-CW-114, 94-C84-123, 94-C84-126