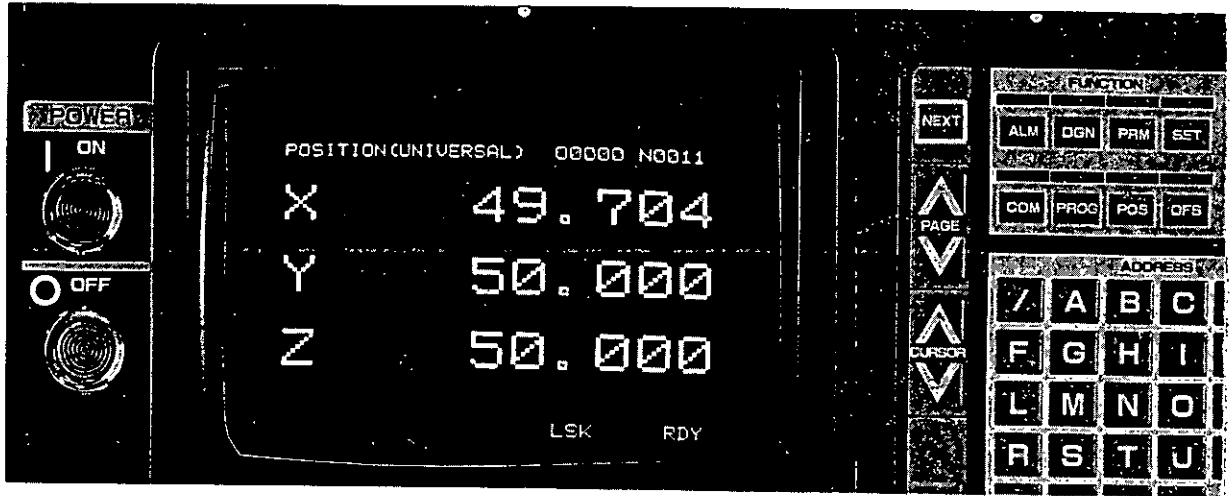


YASNAC MX3

CNC SYSTEM FOR MACHINING CENTERS



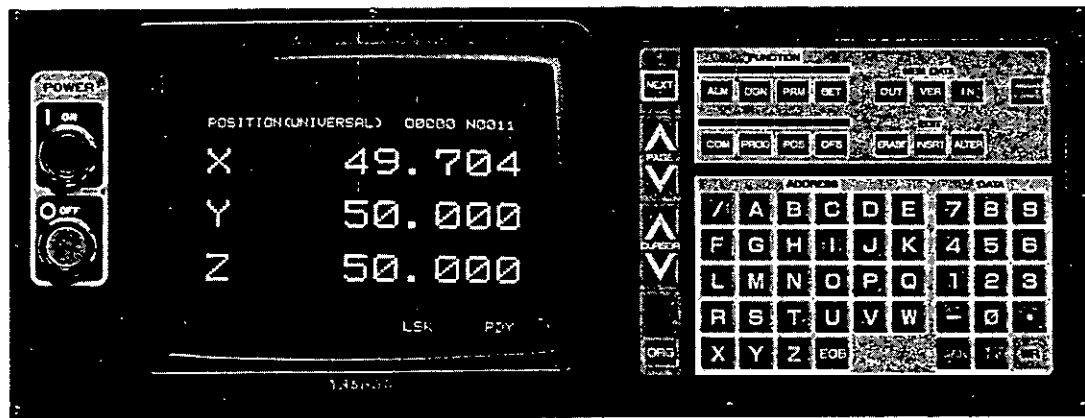
Before initial operation, read these instructions thoroughly, and retain for future reference.



YASKAWA

This manual is primarily intended with 9" CRT character display (basic) to give operators instructions for YASNAC MX3 programming, operation and maintenance. For operation of 14" CRT character display (ACGC, optional), refer to the instruction manual (TOE-C843-8.31) separately provided.

This manual applies to the basic and optional features of YASNAC MX3. The optional features are marked with a dagger. For the specifications of your YASNAC MX3, refer to the machine tool builder's manual.




586-175

YASNAC MX3 Operator's Station
with 9" CRT Character Display

CONTENTS

	Page
1. PREFACE	1
2. PROGRAMMING	1
2.1 INPUT FORMAT	1
2.2 PROGRAM NUMBER AND SEQUENCE NUMBER	6
2.3 COORDINATE WORD	6
2.4 TRAVERSE AND FEED FUNCTIONS	10
2.5 SPINDLE-SPEED FUNCTION (S-FUNCTION)	13
2.6 TOOL FUNCTION (T-FUNCTION)	14
2.7 TOOL COMPENSATION	15
2.8 MISCELLANEOUS FUNCTIONS (M-FUNCTION)	16
2.9 PREPARATORY FUNCTION (G-FUNCTION)	22
2.10 USER MESSAGE DISPLAY †	99
2.11 USER MACRO (G65 AND G66)	100
2.12 SOLID TAP FUNCTION †	128
2.13 AUTOMATIC CORNER OVERRIDE	134
2.14 HIGH-SPEED CONTOURING FUNCTION*	136
3. PART PROGRAM TAPE CODING	139
3.1 TAPE CODE	139
3.2 PROGRAMMING	139
3.3 PART PROGRAM TAPE PUNCHING	142
3.4 PART PROGRAM TAPE HANDLING	142
4. NC OPERATOR'S STATION WITH 9" CRT CHARACTER DISPLAY	143
4.1 PUSHBUTTONS, LAMPS AND KEYS	143
4.2 POWER ON/OFF OPERATION	147
4.3 DISPLAY AND WRITING OPERATION	148
4.4 TAPE INPUT/OUTPUT OPERATIONS OF NC DATA	161
4.5 LOADING PART PROGRAMS INTO MEMORY	164
4.6 EDIT	167
4.7 SUPPLEMENT TO DATA INPUT/OUTPUT INTERFACE	169
4.8 TAPE VERIFYING	172
5. MACHINE CONTROL STATION	175
5.1 SWITCHING UNITS ON THE CONTROL STATION	175
5.2 OPERATION PROCEDURE	185
6. OPERATION PROCEDURE	203
6.1 INSPECTION BEFORE TURNING ON POWER	203
6.2 TURNING ON POWER	203
6.3 MANUAL OPERATION	203
6.4 PREPARATION FOR STORED LEADSCREW ERROR COMPENSATION AND STORED STROKE LIMIT †	204
6.5 PREPARATION FOR AUTOMATIC OPERATION	204
6.6 OPERATION IN TAPE AND MEMORY MODE	205
6.7 MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION	205
6.8 AUTOMATIC OPERATION IN MDI MODE	205
6.9 MDI OPERATION INTERRUPTING AUTOMATIC OPERATION	206
6.10 PREPARATION FOR TURNING OFF POWER	206
6.11 TURNING OFF POWER	206
7. MAINTENANCE	208
7.1 ROUTINE INSPECTION SCHEDULE	208
7.2 BATTERY REPLACEMENT	210
7.3 POWER SUPPLY	211
7.4 THERMAL OVERLOAD RELAY OF SERVO UNIT	211
7.5 MOLDED-CASE CIRCUIT BREAKERS (MCCB)	213
7.6 TROUBLE CAUSES AND REMEDIES	213
APPENDIX-1 LIST OF SETTING NUMBERS	215
APPENDIX-2 LIST OF PARAMETER NUMBERS	227
APPENDIX-3 STORED LEADSCREW ERROR COMPENSATION	258
APPENDIX-4 LIST OF STANDARD INPUT/OUTPUT SIGNALS	262
APPENDIX-5 LIST OF ALARM CODES	271
APPENDIX-6 LIST OF ADDRESS CHARACTERS	299

INDEX

Subject	Chapter	Par.	Page
A ABSOLUTE/INCREMENTAL PROGRAMMING (G90, G91)	2	2.9.26	92
ADDING PART PROGRAM (INSRT KEY)	4	4.6.4	168
ADDRESS AND FUNCTION CHARACTERS	2	2.1.2	3
ADDRESS KEYS	4	4.1.4	145
ADDRESS SEARCH	4	4.3.10	160
ALARM CODE DISPLAY	4	4.3.12	161
ALARM CODES AND REMEDIES	7	7.6.2	213
ALARM MESSAGE DISPLAY BY #8000 COMMAND	2	2.10.1	99
ALARM NUMBER OF USER MACROS	2	2.11.10	123
ARGUMENT DESIGNATION	2	2.11.2	102
AUTO MODE HANDLE OFFSET	2	2.9.31	98
AUTOMATIC ACCELERATION AND DECELERATION	2	2.4.5	13
AOUTOMATIC COORDINATE SYSTEM SETTING †	5	5.2.2	187
AOUTOMATIC CORNER OVERRIDE	2	2.13	134
AOUTOMATIC OPERATION IN MDI MODE	6	6.8	205
AOUTOMATIC RETURN TO REFERENCE POINT (G28) †	2	2.9.14	214
AOUTOMATIC TOOL LENGTH MEASUREMENT †	5	5.2.3	188
AXIS INTERLOCK INPUT	5	5.1.29	183
B BATTERY	7	7.1.3	210
BATTERY REPLACEMENT	7	7.2	210
BEFORE MAINTENANCE CALL	7	7.6.5	214
BUFFER REGISTER	2	2.1.5	5
C CABLE CONNECTOR SPECIFICATIONS	4	4.7.4	171
CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99) †	2	2.9.25	74
CAUTIONS IN PROGRAMMING	3	3.2.3	141
CIRCLE CUTTING (G12, G13) †	2	2.9.9	30
CIRCUIT PROTECTOR TRIP (ALARM NO. 331 TO 335)	7	7.4.1	211
CIRCULAR INTERPOLATION (G02, G03)	2	2.9.4	26
CIRCULAR PATH MODE ON/OFF ON TOOL RADIUS COMPENSATION C (M97, M96) †	2	2.8.6	20
CIRCULAR PROJECTION COMPENSATION	2	2.14.3	138
COMMAND DATA DISPLAY	4	4.3.2	149
CONDITIONS FOR ADDING SOLID TAP FUNCTIONS	2	2.12.1	128
CONSIDERATIONS AND REMARKS FOR USER MACROS	2	2.11.9	120
CONSTANT DISPLAY	4	4.3.1	148
CONTROL COMMANDS	2	2.1.6	117
CONTROL PANEL	7	7.1.1	209
COORDINATE WORD	2	2.3	6
COORDINATE WORD	2	2.3.1	7
CRT CHARACTER DISPLAY	4	4.1.2	144
CURRENT VALUE NUMERICAL SETTING FUNCTION	5	5.2.9	200
CURSOR KEYS	4	4.1.8	146
CYCLE START PUSHBUTTON AND LAMP	5	5.1.2	176
D DATA KEYS	4	4.1.5	145
DECIMAL POINT PROGRAMMING	2	2.1.3	5
DELETING PART PROGRAM BLOCK	4	4.6.2	167
DESCRIPTION ON PROGRAMMING	2	2.12.2	128
DISPLAY AND WRITE OF LOCAL VARIABLES AND COMMON VARIABLES	2	2.11.8	120
DISPLAY AND WRITING OPERATION	4	4.3	148
DISPLAY IN THE  (SETTING) FUNCTION	4	4.3.6	155
DISPLAY LOCK/MACHINE LOCK SWITCH	5	5.1.21	180
DISPLAY OF REGISTERED PROGRAM NUMBER	4	4.5.4	166
DISPLAYING AND RESETTING CURRENT POSITION	4	4.3.4	152
DISPLAYING AND WRITING PARAMETERS	4	4.3.7	158
DISPLAYING AND WRITING TOOL OFFSET AND WORK OFFSET	4	4.3.5	154
DISPLAYING ON/OFF INPUT/OUTPUT SIGNALS	4	4.3.13	161
DRY RUN SWITCH	5	5.1.2	180
DWELL (G04)	2	2.9.6	29

INDEX (Cont'd)

Subject	Chapter	Par.	Page
E EDIT	4	4.6	167
EDIT KEYS	4	4.1.10	146
EDIT LOCK SWITCH †	5	5.1.28	183
EIA/ISO AUTO-SELECT	3	3.1.2	139
EMERGENCY STOP PUSHBUTTON	5	5.1.4	176
EXACT STOP (G09, G61, G64)	2	2.9.7	29
EXERCISES OF USER MACRO	2	2.11.11	123
EXTERNAL DECELERATION INPUT SIGNALS †	5	5.1.30	183
F F 1-DIGIT PROGRAMMING †	2	2.4.4	12
FEED HOLD PUSHBUTTON AND LAMP	5	5.1.3	176
FEED STOP FUNCTION BY SENSOR SIGNAL	5	5.2.10	200
FEEDRATE (F FUNCTION)	2	2.4.2	11
FEEDRATE 1/10	2	2.4.3	12
FEEDRATE OVERRIDE CANCEL SWITCH	5	5.1.13	179
FEEDRATE OVERRIDE SWITCH	5	5.1.12A	178
FEEDRATE, SPINDLE SPEED EDITING FUNCTION	5	5.2.7	197
5TH AXIS CONTROL †	2	2.3.6	9
5TH AXIS NEGLECT INPUT	5	5.1.22C	181
FORM COMPENSATION FUNCTION	2	2.14.2	136
4TH AXIS CONTROL †	2	2.3.5	8
4TH AXIS NEGLECT INPUT	5	5.1.22B	181
FUNCTION KEYS	4	4.1.3	144
G GENERAL PART PROGRAM FORM	3	3.2.2	141
H H- AND D-FUNCTION (H, D CODES)	2	2.7.3	15
HANDLE AXIS SELECT SWITCH †	5	5.1.6	176
HANDLE DIAL † (MANUAL PULSE GENERATOR)	5	5.1.5	176
HANDLE DIALS FOR SIMULTANEOUS CONTROL OF UP TO THREE AXES †	5	5.1.8	177
HELICAL INTERPOLATION (G02, G03) †	2	2.9.5	28
HIGH-SPEED CONTOURING FUNCTION*	2	2.14	136
HOLE PATTERN CYCLES (G70, G71, G72) †	2	2.9.24	72
I IMPORTANT ALARM CODES	7	7.6.4	214
INCH/METRIC DESIGNATION BY G CODE (G20, G21) †	2	2.9.11	33
INPUT FORMAT	2	2.1	1
INPUT FORMAT	2	2.1.1	1
INPUT/OUTPUT SIGNALS	7	7.6.3	213
INPUTTING SETTING DATA AND PARAMETER DATA	4	4.4.2	162
INPUTTING TOOL OFFSETS FROM TAPE	4	4.4.1	161
INSPECTION BEFORE TURNING ON POWER	6	6.1	203
INTERNAL TOGGLE SWITCHES	4	4.3.8	159
J JOG FEEDRATE OVERRIDE SWITCH †	5	5.1.12B	179
JOG FEEDRATE SWITCH	5	5.1.10	178
JOG PUSHBUTTONS	5	5.1.9	177
K KEEPING NC TAPE	3	3.4.2	142
L LABEL SKIP FUNCTION	2	2.1.4	5
LEAST INPUT INCREMENT AND LEAST OUTPUT INCREMENT	2	2.3.7	9
LINEAR INTERPOLATION (G01)	2	2.9.3	25
LIST OF ADDRESS CHARACTERS	APPENDIX-6		299
LIST OF ALARM CODES	APPENDIX-5		271
LIST OF G CODES AND GROUPS	2	2.9.1	22
LIST OF PARAMETER NUMBERS	APPENDIX-2		227
LIST OF SETTING NUMBERS	APPENDIX-1		215
LIST OF STANDARD INPUT/OUTPUT SIGNALS	APPENDIX-4		262
LOADING PART PROGRAMS BY MDI	4	4.5.3	166
LOADING PART PROGRAM TAPE INTO MEMORY	4	4.5.1	164
LOADING PART PROGRAMS INTO MEMORY	4	4.5	164

INDEX (Cont'd)

Subject	Chapter	Par.	Page
M M CODES FOR INTERNAL PROCESSING (M90 TO M199)	2	2.8.2	17
M CODES FOR STOP (M00, M01, M02, M30)	2	2.8.1	16
M-FUNCTION LOCK SWITCH (AUXILIARY FUNCTION LOCK)	5	5.1.23	182
MACHINE CONTROL STATION	5		175
MACHINING INTERRUPTION POINT RETURN FUNCTION*	5	5.2.8	199
MAINTENANCE	7		208
MAKING ADDITION TO A PART PROGRAM	4	4.5.2	165
MANUAL ABSOLUTE SWITCH	5	5.1.24	182
MANUAL CENTERING FUNCTION	5	5.2.11	200
MANUAL OPERATION	6	6.3	203
MANUAL OPERATION INTERRUPTING AUTOMATIC OPERATION	6	6.7	205
MANUAL PULSE MULTIPLY SELECT SWITCH †	5	5.1.7	176
MANUAL REFERENCE POINT RETURN SWITCH †	5	5.1.15	179
MANUAL RETURN TO REFERENCE POINT	5	5.2.1	185
MANUAL SKIP FUNCTION	5	5.2.6	196
MAXIMUM PROGRAMMABLE DIMENSIONS	2	2.3.8	10
MDI OPERATION INTERRUPTING AUTOMATIC OPERATION	6	6.9	206
MEM DATA (MEMORY DATA) KEYS	4	4.1.11	146
MESSAGE DISPLAY BY CONTROL-OUT AND CONTROL-IN	2	2.10.2	99
MIRROR IMAGE AXIS SELECTOR SWITCH	5	5.1.25	182
MIRROR IMAGE ON/OFF (M95, M94)	2	2.8.5	17
MISCELLANEOUS FUNCTION (M-FUNCTION)	2	2.8	16
MODE SELECT SWITCH	5	5.1.1	175
MODIFYING PART PROGRAM BLOCK	4	4.6.3	168
MOLDED-CASE CIRCUIT BREAKERS (MCCB)	7	7.5	213
MULTI-ACTIVE REGISTERS ON/OFF (M93, M92) †	2	2.8.4	17
MULTI-ACTIVE REGISTERS †	2	2.1.6	5
N NC OPERATOR'S STATION WITH 9" CRT CHARACTER DISPLAY	4		143
NC TAPE CHECK	3	3.3.3	142
NC TAPE PUNCH	3	3.3.2	142
NEXT KEY	4	4.1.6	145
O ON-LINE DIAGNOSTICS	7	7.6.1	213
OPERATION COMMANDS	2	2.11.5	115
OPERATION IN TAPE AND MEMORY MODE	6	6.6	205
OPERATION PROCEDURE	5	5.2	185
OPERATION PROCEDURE	6		203
OPERATION TIME DISPLAY	4	4.3.9	159
OPERATIONS USING DATA INPUT/OUTPUT INTERFACE	4	4.7.5	172
OPTIONAL BLOCK SKIP (/1 - /9) †	2	2.2.3	6
OPTIONAL BLOCK SKIP SWITCH	5	5.1.19	180
OPTIONAL STOP SWITCH	5	5.1.18	180
ORG (ORIGIN) KEY	4	4.1.9	146
OTHER M CODES	2	2.8.8	22
OUTLINE OF TOOL COMPENSATION	2	2.7.1	15
OUTPUTTING PART PROGRAM TO PAPER TAPE	4	4.4.5	163
OUTPUTTING SETTING DATA AND PARAMETER DATA TO PAPER TAPE	4	4.4.4	162
OUTPUTTING TOOL OFFSETS TO PAPER TAPE	4	4.4.3	162
OVERLOAD (ALARM NO. 351 TO 355)	7	7.4.2	212
OVERVIEW	2	2.14.1	136
OVERVIEW OF USER MACRO BODY	2	2.11.3	104
P PAGE KEYS	4	4.1.7	146
PAPER TAPE SELECT	3	3.3.1	142
PART PROGRAM DISPLAY	4	4.6.1	167
PART PROGRAM TAPE CODING	3		139
PART PROGRAM TAPE HANDLING	3	3.4	142

INDEX (Cont'd)

Subject	Chapter	Par.	Page
P PART PROGRAM TAPE PUNCHING	3	3.3	142
"+5 V, ±12 V" LEDS (RED) LIT	7	7.3.2	211
"+24 V" LED (RED) LIT	7	7.3.3	211
PLANE DESIGNATION (G17, G18, G19)	2	2.9.10	33
PLAYBACK FUNCTION †	5	5.2.5	193
POSITIONING (G00, G06)	2	2.9.2	25
POWER ON/OFF OPERATION	4	4.2	147
POWER ON/OFF PUSHBUTTONS	4	4.1.1	144
POWER SUPPLY	7	7.3	211
PREFACE	1		1
PREPARATION FOR AUTOMATIC OPERATION	6	6.5	204
PREPARATION FOR STORED LEADSCREW ERROR COMPENSATION AND STORED STROKE LIMIT †	6	6.4	204
PREPARATION FOR TURNING OFF POWER	6	6.10	206
PREPARATORY FUNCTION (G-FUNCTION)	2	2.9	22
PROCESS SHEET	3	3.2.1	139
PROGRAM COPY † (G25)	2	2.9.12B	35
PROGRAM INTERRUPTION ON/OFF (M91, M90) †	2	2.8.3	17
PROGRAM NUMBER	2	2.2.1	6
PROGRAM NUMBER AND SEQUENCE NUMBER	2	2.2	6
PROGRAM RESTART †	5	5.2.4	190
PROGRAMMING	2		1
PROGRAMMING	3	3.2	139
PROGRAMMING OF ABSOLUTE ZERO POINT (G92)	2	2.9.27	92
PUSHBUTTONS, LAMPS AND KEYS	4	4.1	143
R RAPID TRAVERSE RATE	2	2.4.1	10
RAPID TRAVERSE RATE OVERRIDE SWITCH	5	5.1.11	178
REFERENCE POINT CHECK (G27) †	2	2.9.13	37
REFERENCE POINT LAMPS †	5	5.1.16	179
REGISTRATION OF USER MACROS	2	2.11.7	119
REMOTE POWER ON/OFF PUSHBUTTONS	4	4.2.3	148
RESET KEY	4	4.1.12	146
RETURN FROM REFERENCE ZERO (G29) †	2	2.9.15	39
ROTATION OF COORDINATES (G68, G69)	2	2.9.29	97
ROUTINE INSPECTION SCHEDULE	7	7.1	208
S S 2-DIGIT PROGRAMMING	2	2.5.1	13
S 5-DIGIT PROGRAMMING	2	2.5.2	14
SCALING FUNCTION (G50, G51) †	2	2.9.21	61
2ND MISCELLANEOUS FUNCTION (B-FUNCTION) †	2	2.8.9	22
2ND, 3RD AND 4TH REFERENCE POINT RETURN (G30) †	2	2.9.16	40
SEQUENCE NO. COLLATION STOP FUNCTION	4	4.3.11	160
SEQUENCE NUMBER	2	2.2.2	6
SERVO ALARM (ALARM NO. 391 TO 395)	7	7.4.3	212
SERVOMOTOR AND DC MOTOR FOR SPINDLE	7	7.1.2	209
SETTING AND PARAMETER TAPE VERIFYING	4	4.8.1	172
SETTING OF BAUD RATE AND OTHERS OF SERIAL INTERFACE	4	4.7.3	170
SETTIGN OF DATA INPUT/OUTPUT INTERFACE TO BE USED	4	4.7.2	170
SETTING OF LOCAL COORDINATE SYSTEM (G52) †	2	2.9.30	98
SIMULTANEOUSLY CONTROLLABLE AXES OF FIVE-AXIS CONTROL †	2	2.3.4	8
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	179
SKIP FUNCTION (G31) †	2	2.9.17	40
SOLID TAP FUNCTION †	2	2.12	128
SOLID TAP AND ITS RELATION WITH SPINDLE CONTROL I/O	2	2.12.4	130

INDEX (Cont'd)

Subject	Chapter	Par.	Page
S SOLID TAP RELATED FUNCTION	2	2.12.3	130
SOLID TAP RELATED PARAMETER	2	2.12.5	131
"SOURCE" LED [GREEN] UNIT	7	7.3.1	211
SPINDLE INDEXING FUNCTION	5	5.2.12	200
SPINDLE SPEED OVERRIDE SWITCH †	5	5.1.14	179
SPINDLE-SPEED FUNCTION (S-FUNCTION)	2	2.5	13
SPlicing NC TAPE	3	3.4.1	142
START LOCK INPUT (OR SWITCH) †	5	5.1.27	183
STORED LEADSCREW ERROR COMPENSATION	APPENDIX-3		258
STORED STROKE LIMIT (G22, G23) †	2	2.9.12A	34
SUBROUTINE PROGRAM (M98, M99)	2	2.8.7	21
SUMMARY OF EDITING OPERATION	4	4.8.4	174
SUPPLEMENT TO DATA INPUT/OUTPUT INTERFACE	4	4.7	169
SWITCHING UNITS ON THE CONTROL STATION	5	5.1	175
T T 2-DIGIT PROGRAMMING	2	2.6.1	14
T 4-DIGIT † PROGRAMMING	2	2.6.2	14
TAPE CODE	3	3.1	139
TAPE CODE	3	3.1.1	139
TAPE INPUT/OUTPUT OPERATIONS OF NC DATA	4	4.4	161
TAPE VERIFYING	4	4.8	172
THERMAL OVERLOADE RELAY OF SERVO UNIT	7	7.4	211
THIRD TO FIFTH STORED STROKE †	5	5.1.31	184
TOOL COMPENSATION	2	2.7	15
TOOL FUNCTION (T-FUNCTION)	2	2.6	14
TOOL LENGTH COMPENSATION (G43, G44, G49) †	2	2.9.19	54
TOOL LENGTH MEASUREMENT PUSHBUTTON AND LAMP †	5	5.1.26	183
TOOL LIFE CONTROL (G122, G123)	2	2.9.28	93
TOOL OFFSET MEMORY	2	2.7.2	15
TOOL OFFSET VALUE DESIGNATION (G10)	2	2.9.8	30
TOOL OFFSET VALUE TAPE VERIFYING	4	4.8.2	172
TOOL POSITION OFFSET (G45 TO G48)	2	2.9.20	56
TOOL RADIUS COMPENSATION C (G40, G41, G42) †	2	2.9.18	41
TRAVERSE AND FEED FUNCTIONS	2	2.4	10
TROUBLE CAUSES AND REMEDIES	7	7.6	213
TURNING OFF POWER	4	4.2.2	147
TURNING OFF POWER	6	6.11	206
TURNING ON POWER	4	4.2.1	147
TURNING ON POWER	6	6.2	203
TYPES AND FUNCTIONS OF INTERFACE	4	4.7.1	169
U UNIDIRECTIONAL APPROACH (G60) †	2	2.9.23	71
USER MACRO (G65 AND G66)	2	2.11	100
USER MACRO CALL COMMANDS	2	2.11.1	100
USER MESSAGE DISPLAY †	2	2.10	99
V VARIABLES	2	2.11.4	104
VERIFYING PART PROGRAM TAPE	4	4.8.3	172
W WORK COORDINATE SYSTEM SETTING A (G52 TO G59) †	2	2.9.22A	63
WORK COORDINATE SYSTEM SETTING B (G52 TO G59) †	2	2.9.22B	65
WORK COORDINATE SYSTEM SETTING C (G52 TO G59) †	2	2.9.22C	68
WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI	4	4.3.3	151
Z Z-AXIS FEED NEGLECT SWITCH	5	5.1.22A	181



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

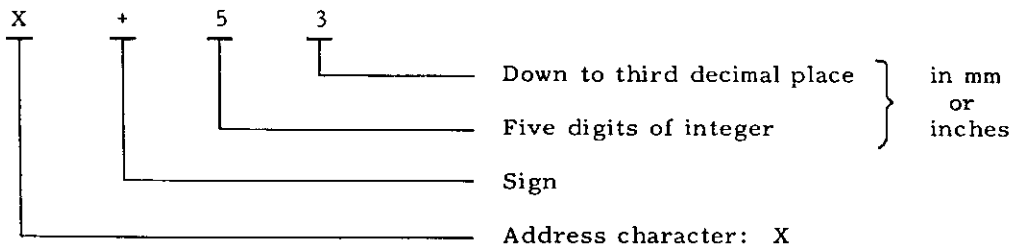
2.1 INPUT FORMAT

2.1.1 INPUT FORMAT

A variable block format conforming to JIS#B 6313 is used for YASNAC MX3.

Table 2.1 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 (;).

• Metric input format

O4 N4 G3 a+43 F5 S2 T2 M3 D(H)2 B3;

• Inch input format

O4 N4 G3 a+34 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.

Japanese Industrial Standard

2.1.1 INPUT FORMAT (CONT'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.		O4		O4		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/min		F50	F31	F50	F41	B
6	Feed/min 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

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 (CONT'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/min
	(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
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 ▽.

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.

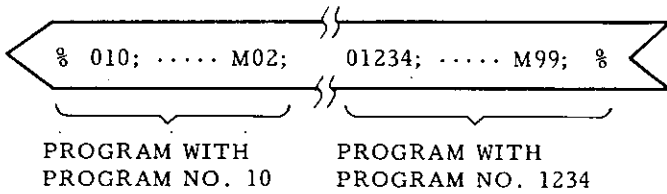
2.2 PROGRAM NUMBER AND SEQUENCE NUMBER

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. (#6021D0)

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 /n 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 register. 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 and 5th axes†	A, B, C or U, V, W	These coordinate words are treated as commands in the directions of the 4th and 5th axes. 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	Simultaneous control of 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 24.

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, $\alpha^{(2)(3)}$ XY, YZ, ZX, $X\alpha^{(1)}$, $Y\alpha^{(1)}$, or $Z\alpha^{(1)}$
Circle 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	One axis, 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 24.
- (3) For circular interpolation axis α , any one of linear axes U, V, and W should be designated.

2.3.4 SIMULTANEOUSLY CONTROLLABLE AXES OF FIVE-AXIS CONTROL†

Table 2.7 shows simultaneously controllable axes.

Table 2.7 Simultaneously Controllable Axes of Five-axis Control

	Simultaneously Controllable Axes
Positioning G00	X, Y, Z, α and β axes
Linear interpolation G01	X, Y, Z, α and β axes
Circular interpolation G02, G03	Two axes, ⁽¹⁾ XY, YZ, ZX, X α , Y α , or Z α
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.
Manual control	One axis, X, Y, Z, α ⁽²⁾ or β .

(1) Circular interpolation is executed only when the 4th axis is linear. Circular interpolation will not function for the 5th axis, if it is linear axis.

(2) Manual pulse generator is of 1-axis or 3-axis.

2.3.5 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.

2.3.5.1 ROTARY AXIS (A, B OR C AXIS)

The rotary axis is defined as follows.

Table 2.8 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")

2.3.5.2 LINEAR AXIS (U, V OR W AXIS)

The linear axes are defined as follows.

Table 2.9 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.

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.

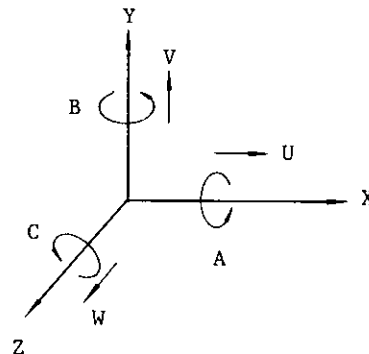


Fig. 2.1 4th Axis in Right-hand Coordinate System

2.3.6 5TH AXIS CONTROL[†]

An additional 5th axis can be incorporated. In this manual, the 5th axis is referred to as β -axis, and represents any of the 6 axes, A, B, C, U, V and W.

2.3.6.1 ROTARY AXIS (A, B OR C AXIS)

The rotary axis is defined as follows.

Table 2.10 Rotary Axes for 5th Axis Control

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" on page 24.)

2.3.6.2 LINEAR AXIS (U, V OR W AXIS)

The linear axes are defined as follows.

Table 2.11 Linear Axes for 5th Axis Control

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.

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.

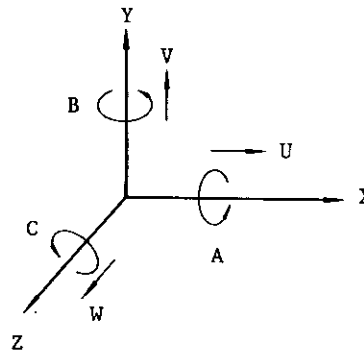


Fig. 2.2 5th Axis in Right-hand Coordinate System

2.3.7 LEAST INPUT INCREMENT AND LEAST OUTPUT INCREMENT

2.3.7.1 LEAST INPUT INCREMENT

The minimum input units that can be commanded by punched tape or MDI are shown in Table 2.12.

Table 2.12 Least Input Increment (#6006D5 = "0.")

	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 $\times 10$ (#6006D5 = "1.")

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

Metric input and inch input can be selected by setting #6001D0.

2.3.7.1 LEAST INPUT INCREMENT (CONT'D)

Notes:

- Selection of metric system or inch system is made by setting (#6001D0).
- Selection of x 1 or x 10 is made by parameter setting (#6006D5).

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 in- to 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.

2.3.7.2 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 2.13 Least Output Increment

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

2.3.8 MAXIMUM PROGRAMMABLE DIMENSIONS

Maximum programmable dimensions of move command are shown below.

Table 2.14 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 2.15 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 24,000 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/min) are programmed.

The programmable range of feedrates is as follows.

Table 2.16 Programmable Range of Feedrate

		Format	Feedrate (Feed/min) range
Metric output	Metric input	F40	F1. -F24000. mm/min
	Inch input	F31	F0.1-F944.8 in./min
Inch output	Metric input	F50	F1. -F60960. mm/min
	Inch input	F31	F0.1-2400.0 in./min

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 G91 (incremental)

G01 X40. Y30. F500 ;

With this command,

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

(mm/min)

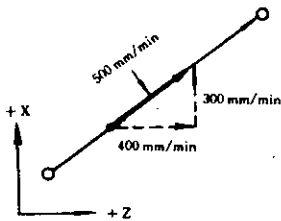


Fig. 2.2

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

With this command,

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

(mm/min)

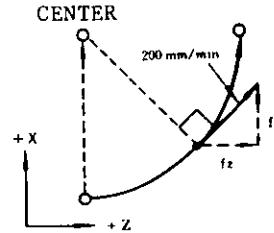


Fig. 2.4

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 G01 X... Y... Z... F400 ;

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

(mm/min)

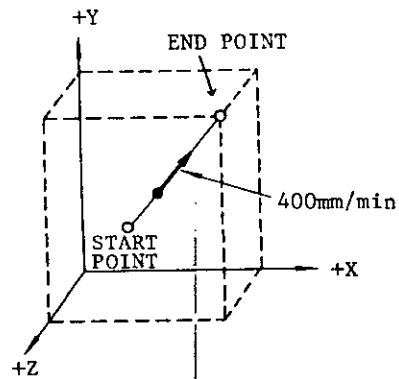


Fig. 2.5

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

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

2.4.2 FEEDRATE (F FUNCTION)(CONT'D)

NOTES:

- If F0 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 2.17 Programmable Range of 1/10 Feedrate

		Format	Feedrate (Feed/min) range
Metric output	Metric input	F41	F1.0-F24000.0 mm/min
	Inch input	F32	F0.01-F944.88 in./min
Inch output	Metric input	F51	F1.0-F60960.0 mm/min
	Inch input	F32	F0.01-F2400.00 in./min

- When parameter #6020 D0 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 2.18(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 2.18(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 2.18(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
F8	#6568
F9	#6569

Setting "1" = 0.1 mm/min

Table 2.18(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 2.18(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/min by the usual F function is not allowed. Specifying 10 mm/min 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 F1-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)

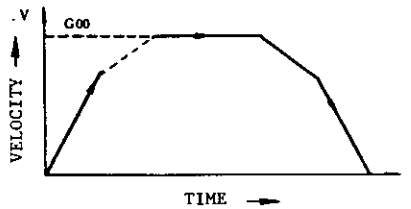


Fig. 2.6

Rapid traverse rate and acceleration/deceleration constant of rapid traverse rate can be set by parameter. (#6280 to #6302)

2.4.5.2 ACCELERATION/DECELERATION OF FEEDRATE

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

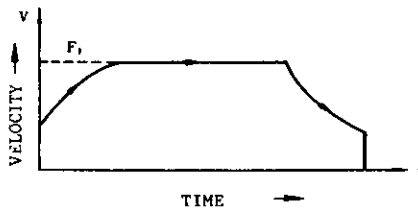


Fig. 2.7 Exponential acceleration deceleration

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 a block, whether the S command is executed together with the move command or after the completion of tool move depends on the machine tool builder. Refer to the machine tool builder's manual.

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.

EXAMPLE

```

G00 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 to the machine when S and two-digit command is issued.

2.5.2 S 5-DIGIT PROGRAMMING

With five digits written after an address character S (S□□□□□), spindle speeds in rpm are directly commanded.

The programmed speeds become effective upon the inputting of an S-command-completion-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 (S0 and other S commands near 0) 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 ;

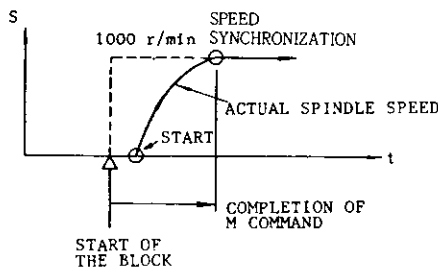
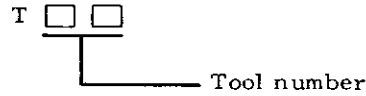


Fig. 2.8

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,

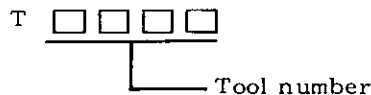
- the two commands are executed simultaneously, or
 - the T command is executed upon completion of the execution of the move command,
- depending on the design of the machine.

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 and 5th⁺) 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	99
Optional	299

The setting range of offset values is as follows.

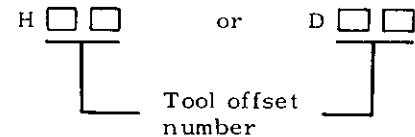
	Linear axis	Rotary axis [†]
Metric input	0 - ± 999.999 mm	0 - ± 999.999 deg
Inch input	0 - ± 99.9999 inch	0 - ± 999.999 deg

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 DATA" on page 137.

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 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.

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

However, for programming ease, it is recommended to divide the numbers into H code part and D code part.

H codes: H01 to H30

D codes: D31 to D99

Table 2.19 H or D Code and Offset Number

Offset method	G code	H or D code	Offset value memory																								
Tool length offset	G43	H 0 1	<table border="1"> <thead> <tr> <th>No.</th> <th>Offset value</th> </tr> </thead> <tbody> <tr><td>01</td><td></td></tr> <tr><td>02</td><td></td></tr> <tr><td>03</td><td></td></tr> <tr><td>04</td><td></td></tr> <tr><td>.</td><td></td></tr> <tr><td>.</td><td></td></tr> <tr><td>.</td><td></td></tr> <tr><td>96</td><td></td></tr> <tr><td>97</td><td></td></tr> <tr><td>98</td><td></td></tr> <tr><td>99</td><td></td></tr> </tbody> </table>	No.	Offset value	01		02		03		04		.		.		.		96		97		98		99	
	No.			Offset value																							
	01																										
02																											
03																											
04																											
.																											
.																											
.																											
96																											
97																											
98																											
99																											
G44																											
G49																											
Tool position offset	G45	D 9 9																									
	G46																										
	G47																										
	G48																										
Tool dia. compensation C (Intersection computing system)	G40																										
	G41																										
	G42																										

2.8 MISCELLANEOUS FUNCTIONS (M-FUNCTION)

The miscellaneous function is specified with the address M and maximum three digits. The function of each M code (M00 to M89) is determined by the machine, except for several M codes. Refer to the machine tool builder's manual for the function of M codes except for the following M codes concerned with the control.

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

• M00 (Program Stop)

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

• M01 (Optional Stop)

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

• M02 (End-of-Program)

M02 is used at the end of program. When given in automatic operation mode, this code stops

the automatic operation after the commands in the block containing M02 have been completed. Although the control is reset in most cases, the details are determined by the machine. Refer to the machine tool builder's manual.

M30 (End-of-Tape)

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

Notes :

- When M00, M01, M02 or M30 is given, it prevents the control from reading ahead the next block of information. The single decoded signal is fed in addition to the 2-digit BCD output for M codes.
- Whether M00, M01, M02 or M30 executes spindle stop, coolant off or some other executions, refer to the machine tool builder's manual.

2.8.2 M CODES FOR INTERNAL PROCESSING (M90 TO M199)

M90 through M199 are used only for internal processing. Even when they are programmed, no external output signal (BIN and decoded output) is sent.

- M90†: 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)†

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.

Note: Program interrupt function during DNC operation is ineffective.

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

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 5 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_{D0} to _{D3} (or mirror image axis designation switch.) For this procedure, refer to 5.1.25, "MIRROR IMAGE AXIS SELECTOR SWITCH" on page 157.

2.8.5 MIRROR IMAGE-ON/OFF (M95, M94) (CONT'D)

- 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.

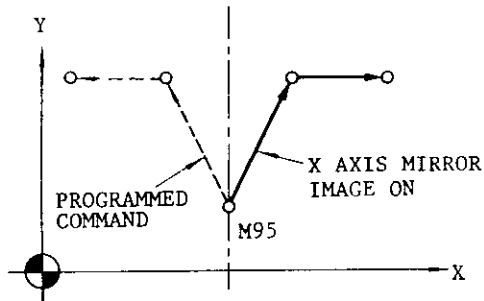


Fig. 2.9

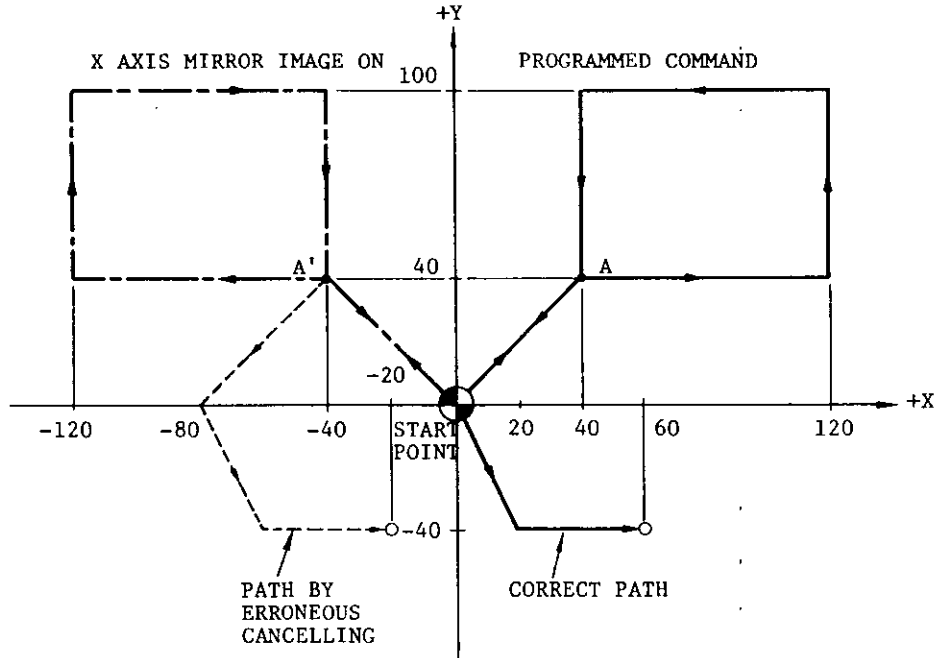
With both the absolute and increment move commands, the same mirror image effect will be obtained. The block including M95 command constitutes the mirror point.

- 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 G29 is used to change tools or for ending machining processes, make sure to cancel the mirror image effect by means of M94. If mirror image effect is not cancelled when G28 or G29 is given, an error "058" is shown.
- 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 cancelling mirror image will be shifted by the difference between both positions.
- When the operation is reset, it will be in M94 mode.

EXAMPLE



```

N01 G92 X0 Y0 ;
N02 M95 ; ..... Mirror image on.
N03 G90 G01 X40. Y40. F300 ;
N04 X120. ;
N05 Y100. ;
N06 X40. ;
N07 Y40. ;
N08 X0 Y0 ;
N09 M94 ; ..... Correct mirror image off.
N10 X20. Y-40. ;
N11 X60. ;

```

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

Fig. 2.10

2.8.5 MIRROR IMAGE-ON/OFF (M95, M94)(CONT'D)

- 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.")

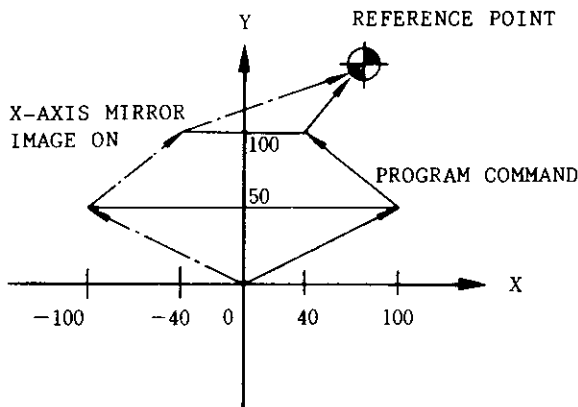
#6005, D1 0: Commands mirror image at the G28 intermediate point
1: Does not command mirror image at the G28 intermediate point

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 (#6116) of the set/reset M codes.

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

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

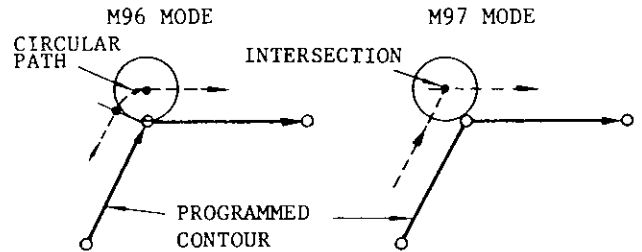


Fig. 2.11

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

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

G01 X... Y... ;
(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)

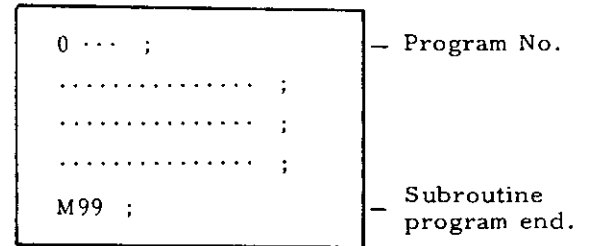
M98 P... L... ;

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.

- Format of subroutine program (M99)

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



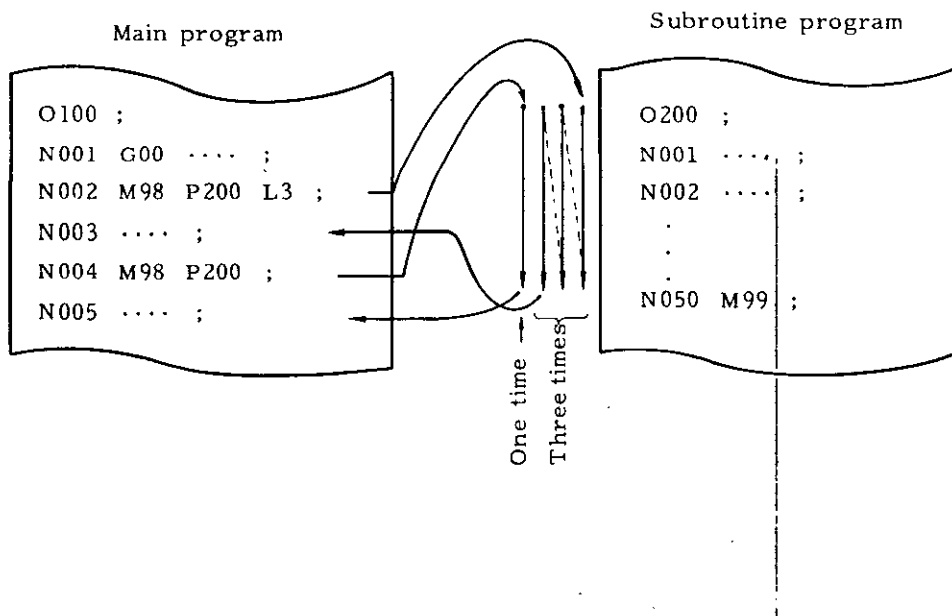
- Automatic return command from subroutine program

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.

EXAMPLE

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



2.8.7 SUBROUTINE PROGRAM (M98, M99) (CONT'D)

- Special use of M99

M99 P... ;

With this command, the main program does not return to the block following the M98 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 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 2.20 Typical Examples of M Codes for Machine

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

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

2.8.9 2ND MISCELLANEOUS FUNCTION (B-FUNCTION)[†]

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

Three digits following the address B give index table positions.

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

- the B command is executed simultaneously with the move command, or
 - B command is executed after the execution of the move command,
- depending on the design of the machine tool. For this refer to the specifications of the machine tool builder.

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

Notes :

- B function standard interface is in 3-digit BCD output.
- With MDI operation on NC panel, "B" is used to specify address for B codes. Therefore, when the control has B-function, the 4th and 5th 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 a G code belonging to the 01 group is commanded during a canned cycle (G73, G74, G76, G77 and G81 through G89), the canned cycle is canceled and these codes in the group 09 becomes G80.
- 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 performed.

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

- G codes in the 01 group may selectively be changed to G00 or kept unchanged after re-setting. (#6005D6)
- During the execution of G92, Display may selectively be made. (#6005D5)

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

2.9.1 LIST OF G CODES AND GROUPS (CONT'D)

Table 2.21 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	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	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		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	15	Scaling OFF	O
G51		Scaling ON	O
G52	12	Return to base coordinate system	O
G53	*	Temporary shift to machine coor- dinate 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	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	B
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	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 ∇ are automatically selected at power on or reset.
3. The code marked ∇ is automatically selected upon power ON.

2.9.2 POSITIONING (G00, G06)

G00 X... Y... Z... ($\alpha\ddagger\dots\beta$) ;
(where α and β = A, B, C, U, V, or W)

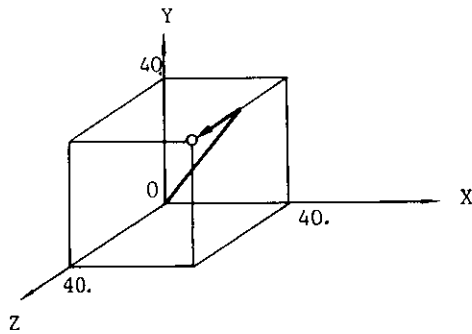
With this command, the tool is sent to the specified position in rapid traverse motions along the 3 axes (5 axes!) simultaneously. If any of the coordinate positions is not specified, the machine does not move along that coordinate axis.

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

Motions in the respective axis directions are independent each other, and therefore, the resultant tool path is not necessarily straight. When programming tool positioning commands, take care to avoid the possibility of tool and workpiece interference.

EXAMPLE

G01 X40. Y40. Z40. ;



Rapid traverse rate

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

Fig. 2.12

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

Error detect OFF positioning (G06)

G06 X... Y... Z... ($\alpha\ddagger\dots\beta\dots$) ;

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.
- 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... ($\alpha\ddagger\dots\beta\dots$) F... ;

where α and β = A, B, C, U, V, or W

With this command, the tool is moved simultaneously in the three (five+) 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.

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

(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 2.9.31, "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 ;

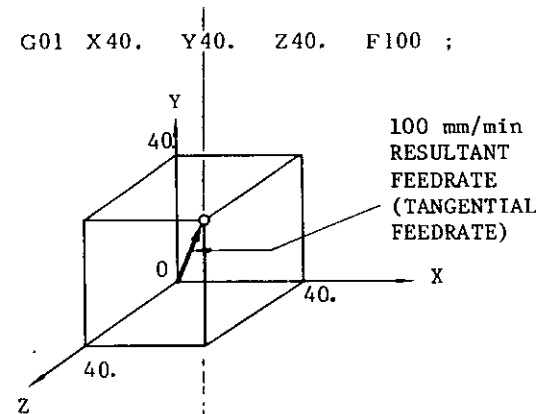


Fig. 2.13

Where the optional 4th or 5th 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.

2.9.3 LINEAR INTERPOLATION (G01) (CONT'D)

Table 2.22 Minimum F Command Unit

F-function			In minimum F command unit	
			Feedrate of basic three axes	Feedrate of rotary axes
Metric output	Metric input	F40	1 mm/min	1 deg/min
	Inch input	F31	0.1 in./min	2.54 deg/min
Inch output	Metric input	F50	1 mm/min	0.3937 deg/min
	Inch input	F31	0.1 in./min	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 paths on the XY,

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

XY plane G17 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\}$ X... Y... $\left\{ \begin{matrix} R... \\ I... J... \end{matrix} \right\}$ F... ;

ZX plane G18 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\}$ Z... X... $\left\{ \begin{matrix} R... \\ K... I... \end{matrix} \right\}$ F... ;

YZ plane G19 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\}$ Y... Z... $\left\{ \begin{matrix} R... \\ J... K... \end{matrix} \right\}$ F... ;

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

G02: Clockwise

G03: Counter-clockwise

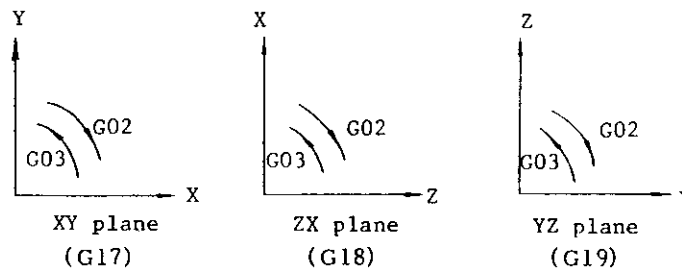


Fig. 2.14

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

- G17: XY plane or X4 plane†
- G18: ZX plane or Z4 plane†
- G19: YZ plane or Y4 plane†

In addition to the plane of circular interpolation, these G codes specify planes for tool radius com-

pensation (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.

EXAMPLE

```
G17 G90 G03 X15. Y40. I-30. J-10. F150 ;
```

(a) Absolute command with (G90)

```
G17 G91 G03 X-40. Y20. I-30. J-10. F150;
```

(b) Incremental command

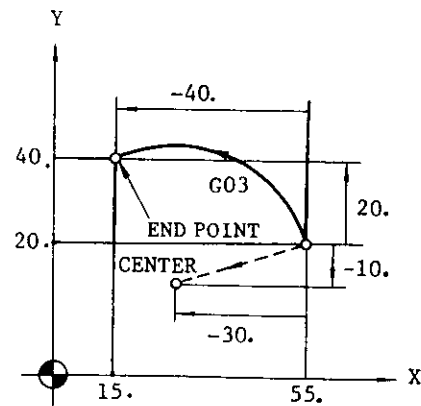
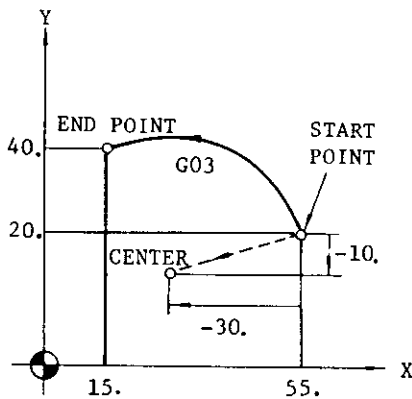


Fig. 2.15

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... ;
```

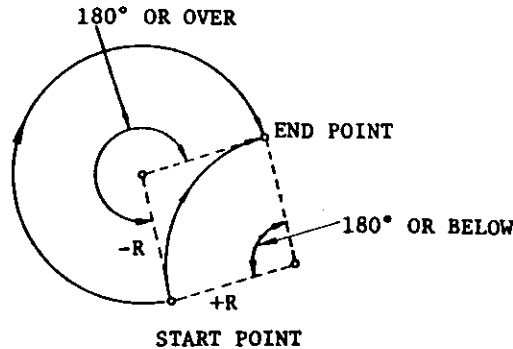


Fig. 2.16

2.9.4 CIRCULAR INTERPOLATION (G02, G03) (CONT'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 $X\alpha$, $Z\alpha$, and $Y\alpha$ planes in addition to the XY, YX, and ZY planes (where $\alpha = U, V, \text{ or } W$)

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

Note:

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

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 X0 Y0 ;
 G02 X0 Y0 I10.J0 F100 ;
 ... complete circle

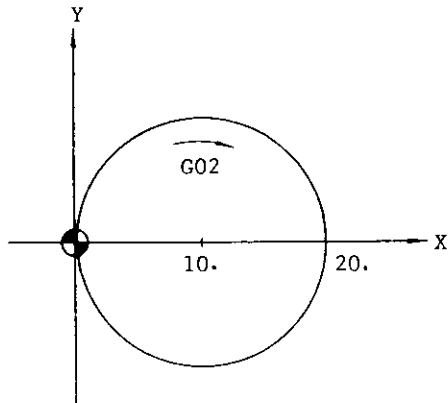


Fig. 2.17

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 0 are commanded as end point. (See the figure below.)

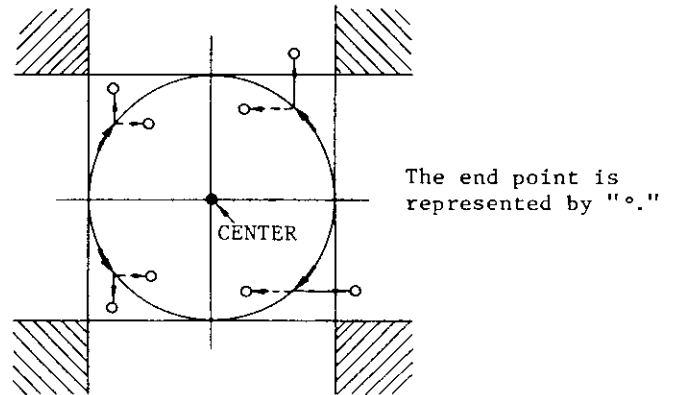


Fig. 2.18

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.

2.9.5 HELICAL INTERPOLATION (G02, G03)†

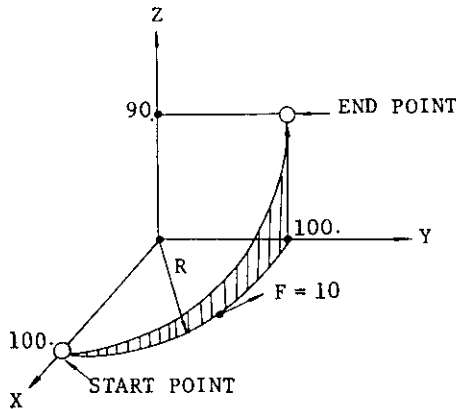
A circular interpolation on a certain plane, and a linear interpolation along an axis not included in that plane can be executed in synchronization, and this combined interpolation is called helical interpolation.

Command format

- (a) For XY plane G17 {G02
G03} X... Y... {R...
I... J...} Z(α)... F... ;
- (b) For ZX plane G18 {G02
G03} Z... X... {R...
K... I...} Y(α)... F... ;
- (c) For YZ plane G19 {G02
G03} Y... Z... {R...
J... K...} X(α)... F... ;
- (d) For Xα plane G17 {G02
G03} X... α... {R...
I... J...} Z... F... ;
- (e) For Zα plane G18 {G02
G03} Z... α... {R...
K... I...} Y... F... ;
- (f) For Yα plane G19 {G02
G03} Y... α... {R...
J... K...} X... F... ;

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).

EXAMPLE



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

Fig. 2.19

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. Therefore, the speed (F') in the direction of linear interpolation is as follows.

$$F' = F \times \frac{\text{(Length covered by linear interpolation)}}{\text{(Length of circular path)}}$$

- 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.

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

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.

2.9.7 EXACT STOP (G09, G61, G64) (CONT'D)

• 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 cancelling the effect of G61.

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 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 Pn X... Y... Z... (α...β...);

(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... (α...β...);. (Jn specifies J1 to J6. 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... ;

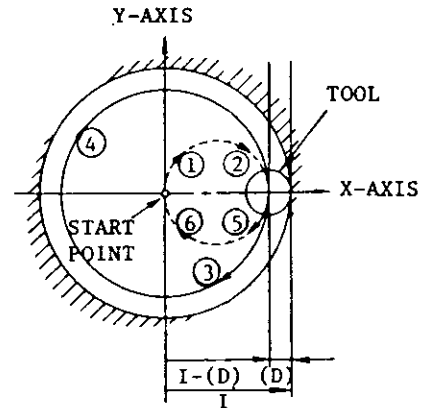
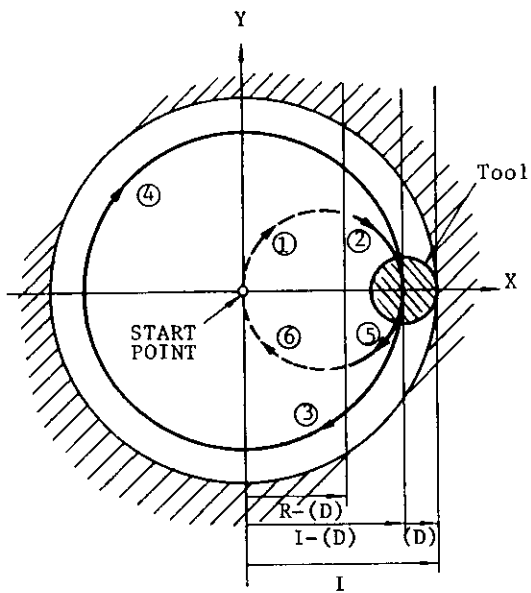


Fig. 2.20

• 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.



Tool path

G12: ① → ② → ③ → ④ → ⑤ → ⑥

G13: ⑥ → ⑤ → ④ → ③ → ② → ①

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

G12: Clockwise (CW)

G13: Counterclockwise (CCW)

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

R: Rapid traverse section
(incremental value with sign)

D: Tool radius compensation No.

F: Cutting feed rate

Fig. 2.21

- 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.

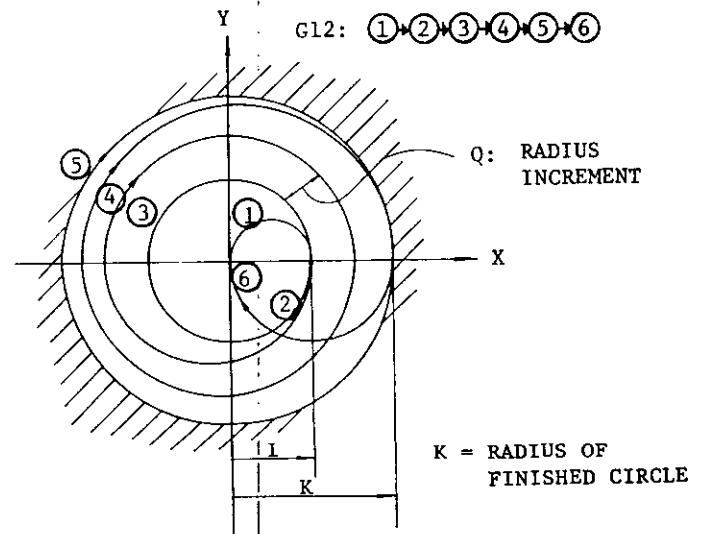
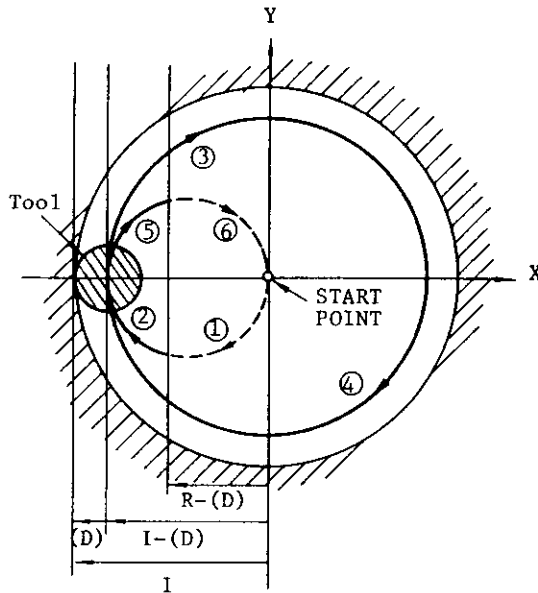


Fig. 2.22

2.9.9 CIRCLE CUTTING (G12, G13)[†] (CONT'D)



Tool path

G12: ① → ② → ③ → ④ → ⑤ → ⑥

G13: ⑥ → ⑤ → ④ → ③ → ② → ①

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

G12: Clockwise (CW)

G13: Counterclockwise (CCW)

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

R: Rapid traverse section
(incremental value with sign)

D: Tool radius compensation value

Fig. 2.23

- 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, J, K and D codes, motions in the - direction of X-axis (symmetrical with respect to Y-axis) can be commanded. In Fig. 2.24, 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.
- 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.

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

If R-d and I-d have different signs, this is also regarded as an error.

- When programming G12(G13), always specify a tool radius compensation number D. If this is not specified, the tool moves without radius compensation.

EXAMPLE

```
G00 Z-40. ;
G12 I50. R40. D10 F300 ; D10 = 10.0mm
G00 Z40. ;
```

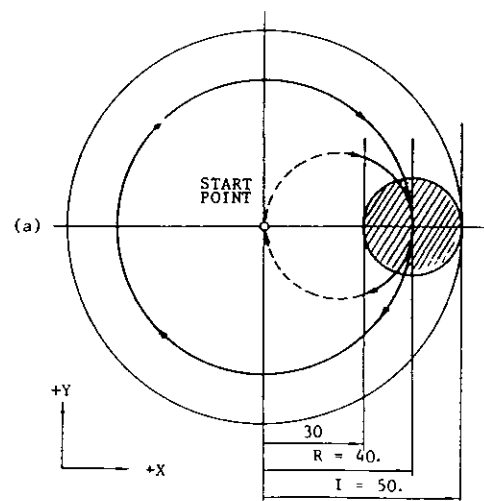


Fig. 2.24

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

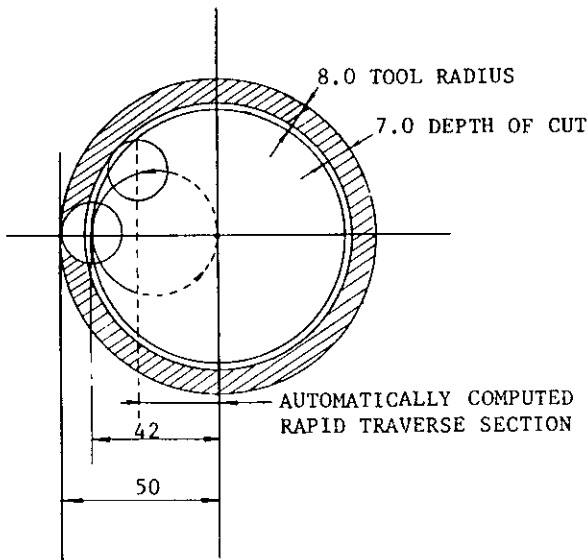


Fig. 2.25

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 fourth axis of rotary axis.

The XY plane (G17) 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

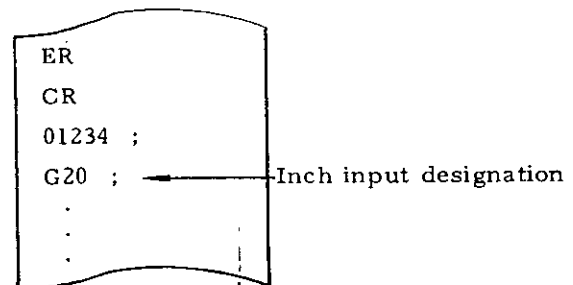
These G codes are programmed at the leading end of a block of its own. If one of these G codes are commanded, the units of all the following motions are changed afterwards.

- 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 D0.

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)

2.9.11 INCH/METRIC DESIGNATION BY G CODE (G20, G21)[†] (CONT'D)

- Take the following procedure after the command of G20/G21 selection.
 - A. Program absolute zero point 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

2.9.12A 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 A_1 and lower limit point B_1 are specified by parameters.

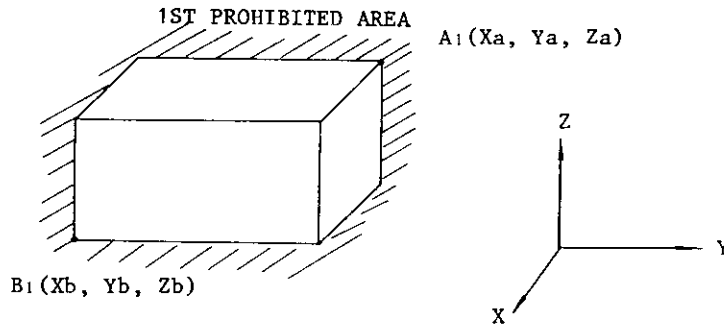


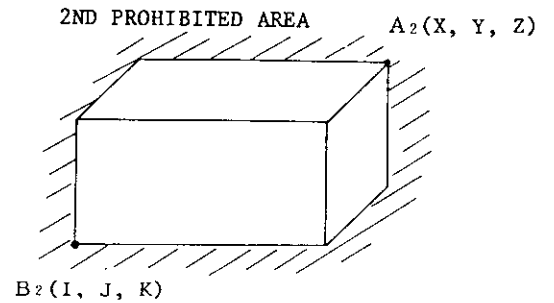
Fig. 2.26

- 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.

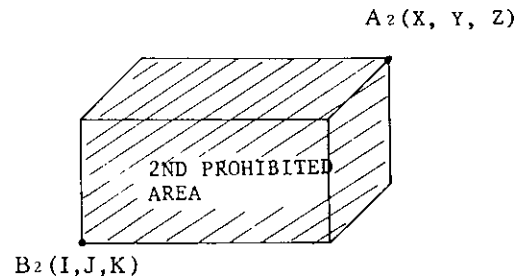
```
G22 X... Y... Z... I... J... K... ;
```

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. 2.27

Table 2.23 Setting of Stored Stroke Limit for Each Axis

		X	Y	Z	Division
1st prohibited area	Point A1	#6600	#6601	#6602	Parameter
	Point B1	#6606	#6607	#6608	
2nd prohibited area	Point A2	#6510	#6511	#6512	Setting
	Point B2	#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.

2.9.12B PROGRAM COPY† (G25)

(1) The command G25P p1 p2 Q q1 q2 L; , executes L times any program from sequence p2 of program p1 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)

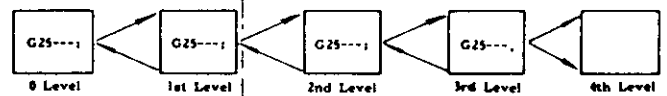
q2; Last sequence

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

L: Number of repetitions (One time if omitted)

(2) 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.



2.9.12B PROGRAM COPY† (G25)(CONT'D)

(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(p1) (p2) Q (q1) (q2) L ...;, the following errors will occur.
 Error "041" when p1 cannot be searched
 Error "041" when p2 cannot be searched
 Error "040" when p1 and q1 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 OL 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.

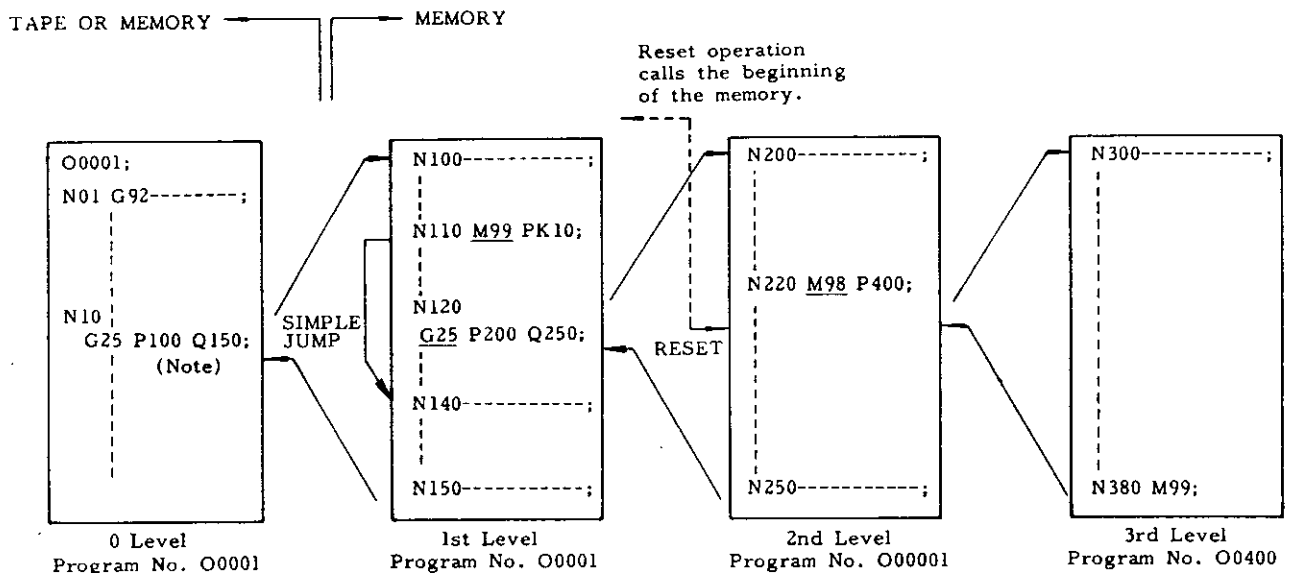
(g) The leading zeros in both commands P and Q can be omitted. However, for five digits or more, the lower four digits will be considered the sequence number.

(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.

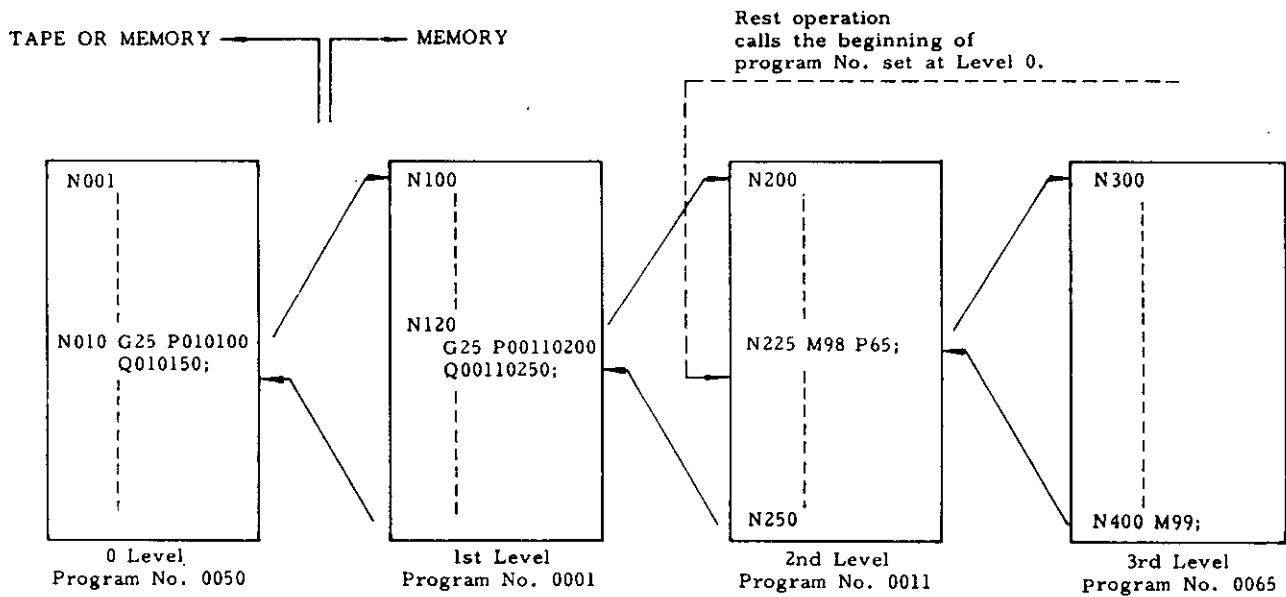
(j) The 0 (q1) at the end of the Q command can be omitted. When omitted, it will automatically be considered to be the same as the leading 0 of the P command.

(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.



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

Example: N10G25 P010100 Q1550; Program number



Note:

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.

2.9.13 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.

G27 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.

2.9.13 REFERENCE POINT CHECK (G27)[†] (CONT'D)

Reference point as meant here is a fixed 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 6.1.15. "MANUAL REFERENCE POINT RETURN SWITCH" on page 170. 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.

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

G28 X... Y... Z... (α^{\dagger} ... β ...);

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 (5 axes[†]). However, the tool will not move in the direction of the axis for which a coordinate instruction is omitted.

EXAMPLE

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

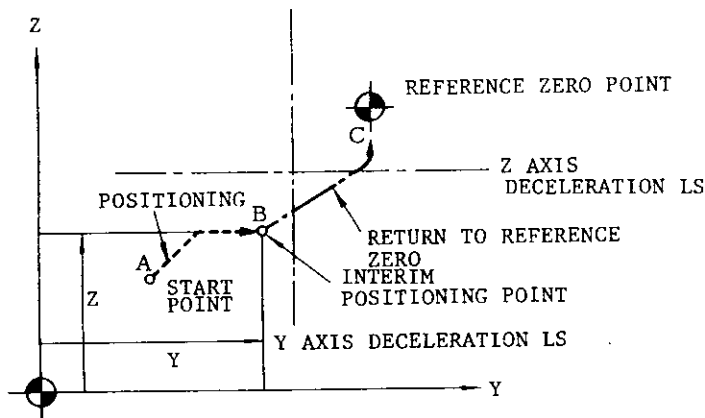


Fig. 2.28

"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. Make it a point 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 6.2.1 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 axes, 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.

2.9.15 RETURN FROM REFERENCE ZERO (G29)†

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

```
G28 Y... Z... ;      Point A → B → C
      .             (Reference zero point)
      .             Point B
      .
      .
G29 Y... Z... ;      Point C → B → D
      .             Point D
```

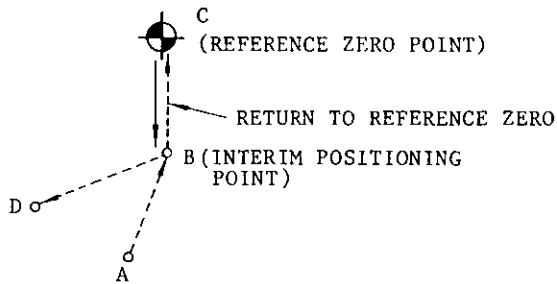


Fig. 2.29

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

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

If G28 is programmed a number of times, the final coordinates of point B which 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.)
```

```
N24 G29 X-40. Y-50. ;
      Equal to the two blocks = G00 X30. Y20. ;
      G00 X-40. Y-50. ;
```

EXAMPLE 2

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

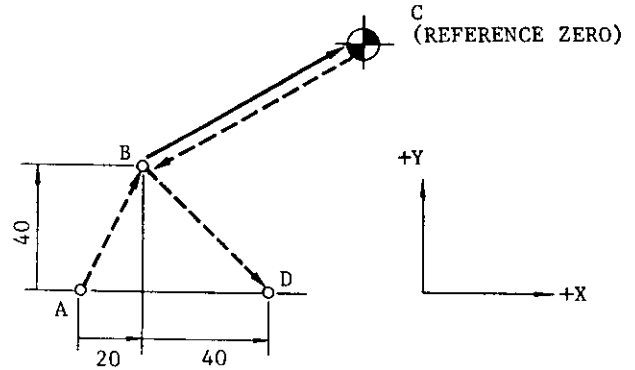


Fig. 2.30

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'.

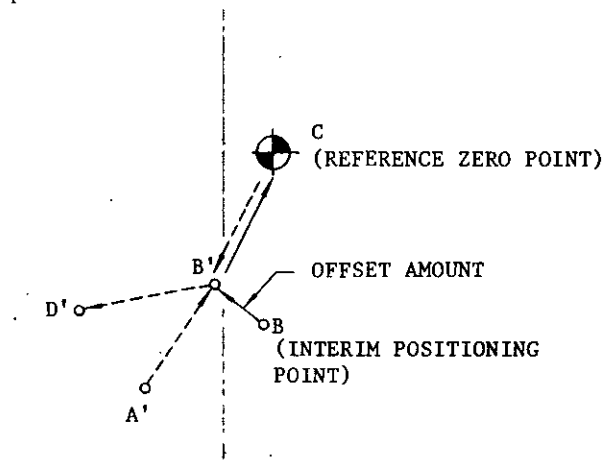


Fig. 2.31

2.9.15 RETURN FROM REFERENCE ZERO (G29)[†] (CONT'D)

- 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 absolute zero (G92, ORG key)
 - Machine lock
 - Manual operation at Manual Absolute Off
 - (2) G28 and G29 are commanded in the blocks following the block containing M94 which cancels mirror image at the different point from the starting point of mirror image.
 - (3) G28 and G29 are commanded after manual operation at Manual Absolute Off.

2.9.16 2ND, 3RD AND 4TH REFERENCE POINT RETURN (G30)[†]

G30 Pn X... Y... Z... (α^{\dagger} ... β^{\dots}) ;
(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 in the command, the tool remains motionless in the direction of that axis.

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

EXAMPLE

G30 P3 X30. X50. ; ... The tool returns to the 3rd reference point moving in the X and Y directions.

Notes :

- Three items except the last one in NOTES of 2.9.14 AUTOMATIC RETURN TO REFERENCE POINT (G28)[†] on page 37, 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 G30. However the interim positioning point is renewed on the axis designated by G30.

2.9.17 SKIP FUNCTION (G31)[†]

G31 X... Y... Z... (α^{\dagger} ... β^{\dots}) 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 0.5 m sec. G31 is non-modal.

EXAMPLE

```
N100 G90 G31 X100. Y50. ;
N200 G01 X80. Y15. ;
```

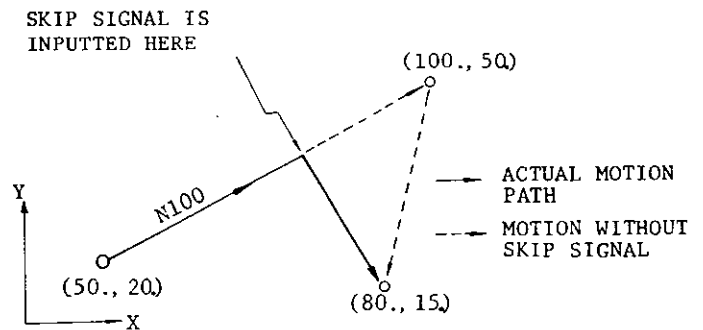


Fig. 2.32

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 G31 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
 #6556 ... storing 5th 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 (#6004D0), the program moves on to the next block automatically.

2.9.18 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 2.24 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 D00 is commanded, tool radius will be regarded as "0."

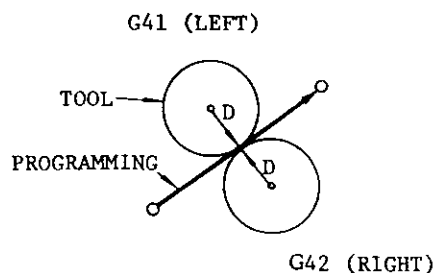


Fig. 2.33

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

2. Designation of compensating plane

The plane in which tool radius compensation is made is designated by G17, G18, G19. They are G codes of 02 group. The XY plane (G17) is in effect at the time power is turned on.

Table 2.25 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.

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

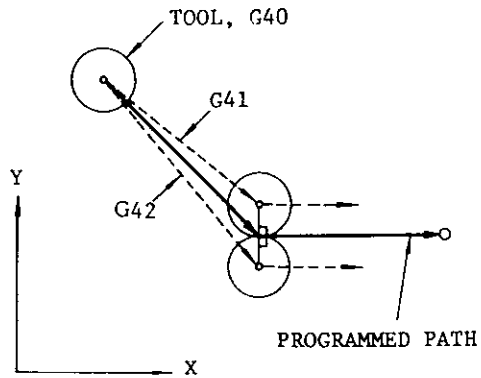
3. Method of entry into compensation mode

When G41(G42) is programmed, the tool moves to an offset position with the distance equal to the radius. The offset position is on the normal line at the start point of the block immediately after G41(G42). If no coordinate instruction is programmed in the block of G41(G42), movement is made by the offset value only. Because G41(G42) accompanies a movement, it is necessary to program G00 or G01 for a G code in group A. An input error (alarm code "026) occurs if a G code other than G00, G01 is programmed.

2.9.18 TOOL RADIUS COMPENSATION C
(G40, G41, G42)+ (CONT'D)

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... ;

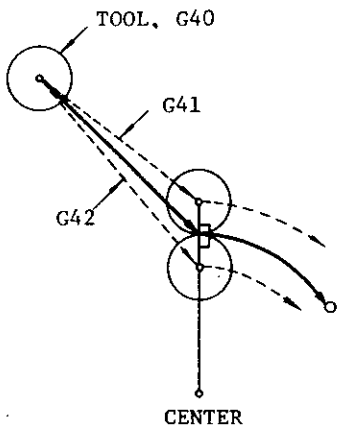
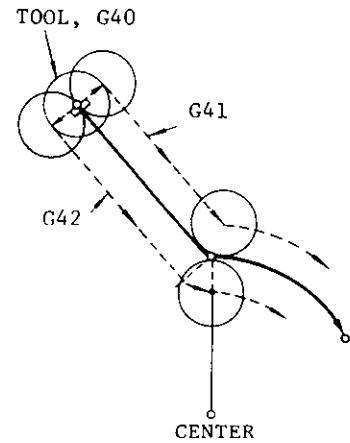


Fig. 2.34

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... ;

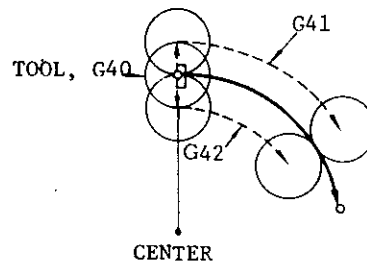


Fig. 2.35

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

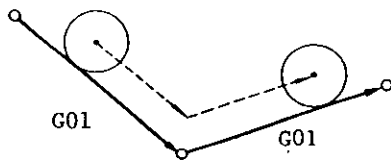


Fig. 2.36

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

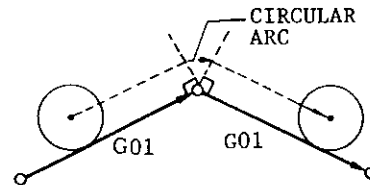


Fig. 2.37

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. Refer to 2.8.6, "CIRCULAR PATH MODE ON/OFF ON TOOL RADIUS COMPENSATION C (M97, M96)" on page 20 for details.

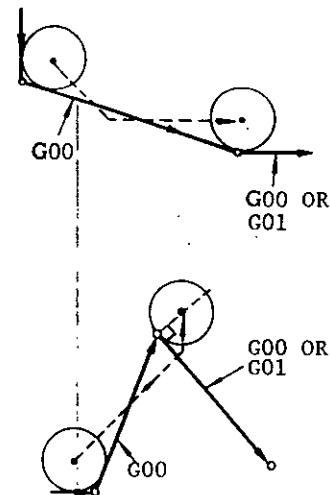
M96 ... Tool radius compensation circular path ON

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. 2.38

2.9.18 TOOL RADIUS COMPENSATION C
(G40, G41, G42)+ (CONT'D)

D. Shape requiring care

Do not program a wedge shape having an acute angle.

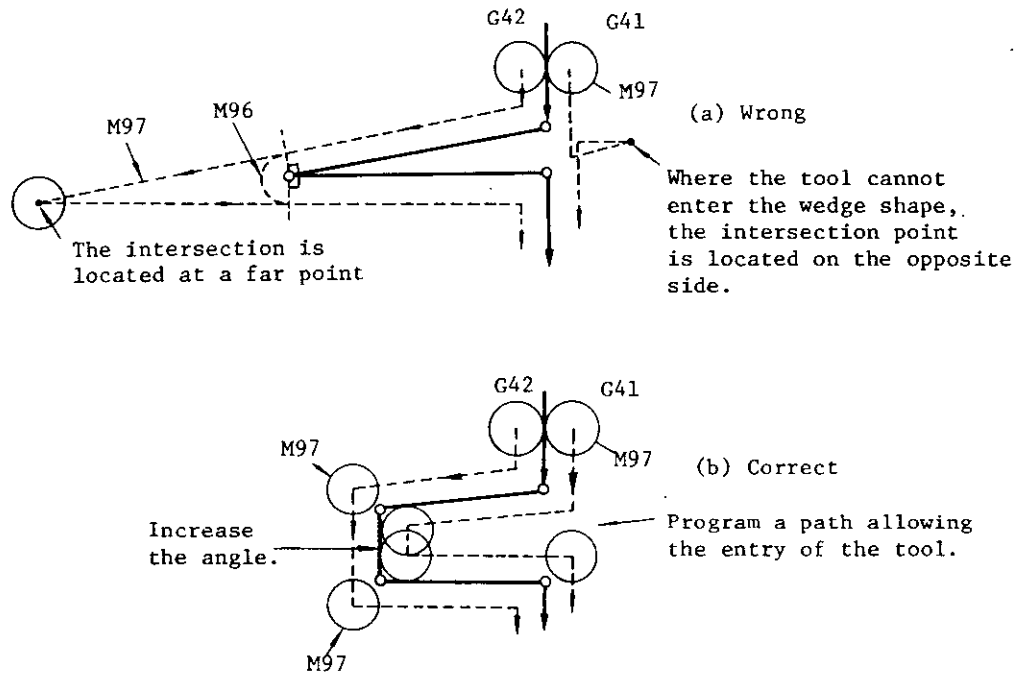


Fig. 2.39

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 movement instruction 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... ;
N011 I... J... ;
N012 Z... ;
.
.
N019 Z... ;
N020 X... Y... ;
.
.
N029 X... Y... ;
N030 G40 X... Y... ;
```

XY plane

Dummy block

Z axis
(3 blocks
or above)

XY plane

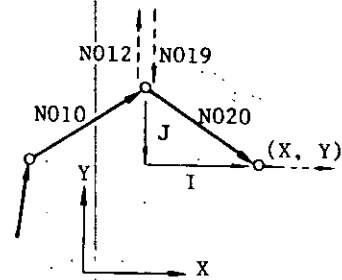


Fig. 2.40

2.9.18 TOOL RADIUS COMPENSATION C
(G40, G41, G42)+ (CONT'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 axes respectively. Suitably use them in accordance with the plane designation.

EXAMPLE

N050 G01 X... Y... ;

N051 G01 I(b) J(-a) ; — Dummy block

N052 Z... ;
N053 Z... ; } Z axis
N059 Z... ;

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

N061 G01 X... Y... ;

- 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.

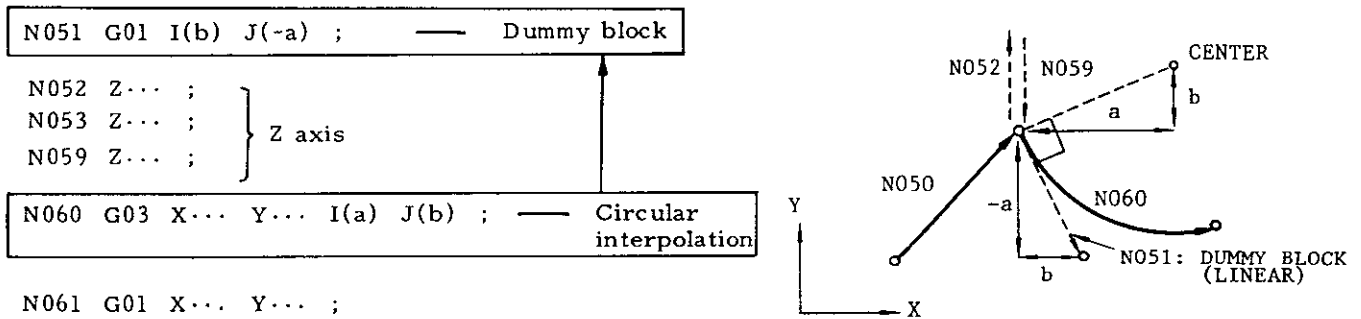


Fig. 2.41

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. Exercise care with the sign of the dummy block data depending on the shape of the circle. The tool stops at point A by the dummy block in preparation for the next circular command.

6. Switching between G41 and G42 in compensation mode

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

EXAMPLE

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

·
·
·

N20 G01 X... Y... F... ;

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

N22 X... ;

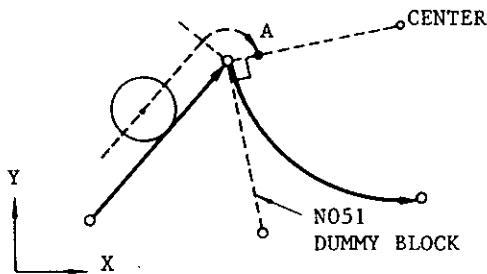
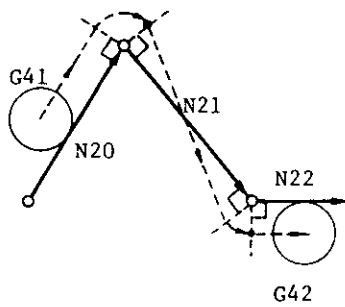
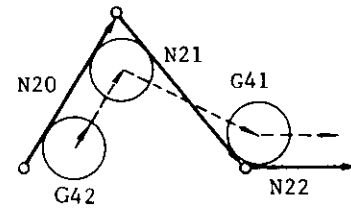


Fig. 2.42



(a) G41 → G42
(M96 mode)



(b) G42 → G41

Fig. 2.43

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 the compensation mode is effective in the block next to the commanded block.

8. Method of cancellation of compensation

G40 is the command for cancelling 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.

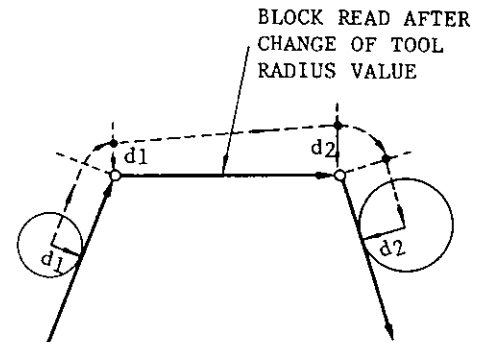


Fig. 2.44

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.

2.9.18 TOOL RADIUS COMPENSATION C
(G40, G41, G42)+ (CONT'D)

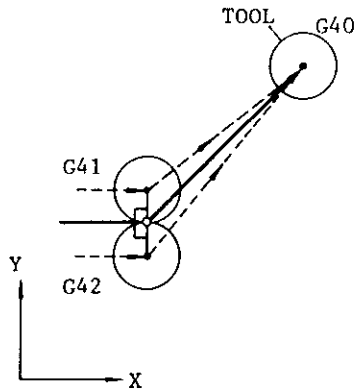
EXAMPLE A

(a) G41 (G42)

```

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

```



(b) G41 (G42)

```

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

```

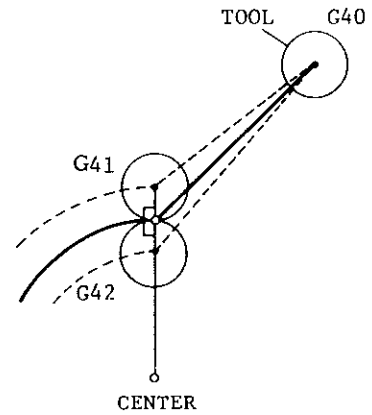


Fig. 2.45

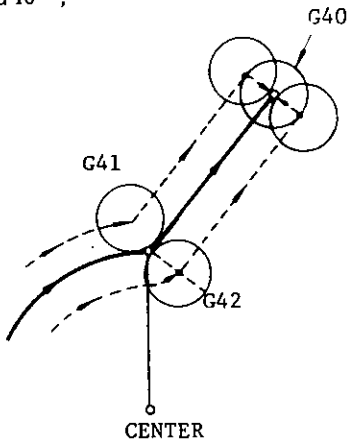
EXAMPLE B

(c) G41 (G42)

```

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

```



(d) G41 (G42)

```

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

```

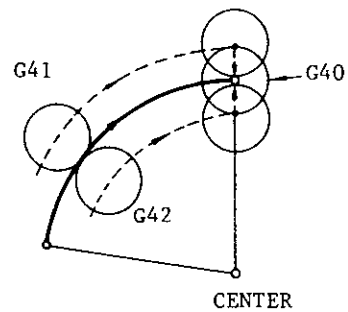


Fig. 2.46

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.

9. Cautions and remarks in tool radius compensation C

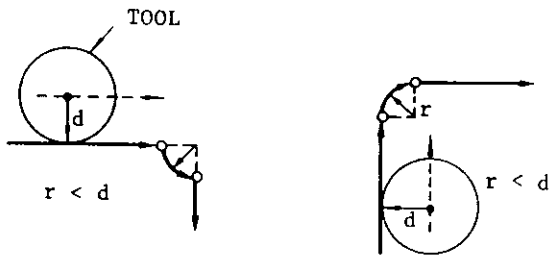
A. Maximum programmable dimensions (Refer to Table 2.14) is not changed even in tool radius compensation C.

B. Programmed shapes that produce input errors

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

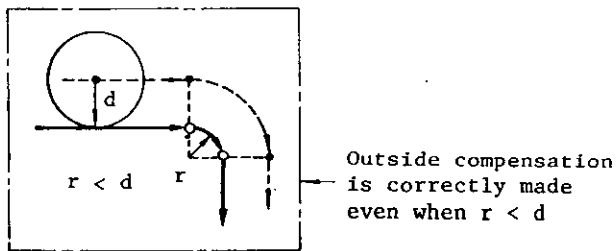
(1) When programming an inside arc with tool compensation, if

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



(a) Inside compensation error

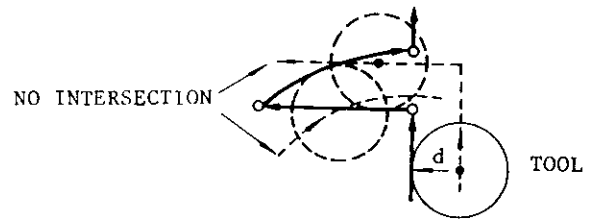
(b) Inside compensation error



Outside compensation is correctly made even when $r < d$

Fig. 2.47

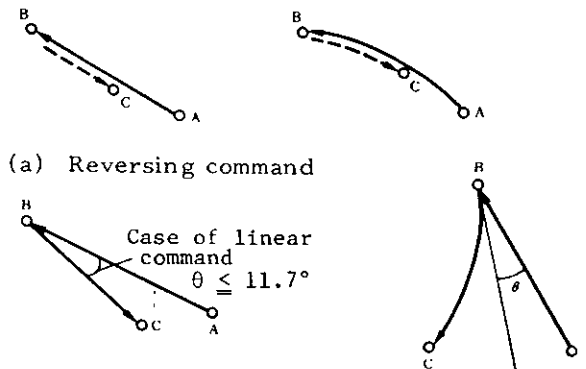
(2) When no intersection point exists on the locus of the offset tool center.



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

Fig. 2.48

(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

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

Fig. 2.49

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

2.9.18 TOOL RADIUS COMPENSATION C
(G40, G41, G42)+ (CONT'D)

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

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

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

D. Tool radius compensation C is applied to the movement path offset by tool 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.

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 compensating functions in themselves. Input error "024" occurs when they are programmed in the compensation mode.

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

I. Subprogram (M98, M99) can be programmed in the compensation mode.

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

COMPLETING POSITION OF PULSE DISTRIBUTION OUT OF THE COMPENSATION PLANE.

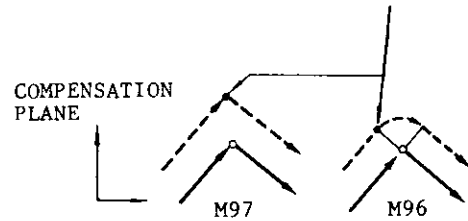


Fig. 2.50

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 G01 X... Y... ;
N101 I... J... ;
N102 X... ;
```

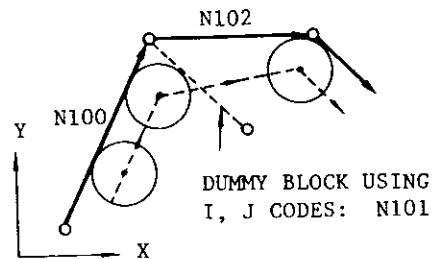


Fig. 2.51

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

```

(G41)
.
.
.
N200 G01 X... Y... ;

N201 I... J... ;
N202 M00 ;
N203 X... Y... ;

```

← Command movement data N203 using I, J

N. Up to 99 radius values can be stored in the 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 G97 mode, it is better than overcut with the M96 mode.

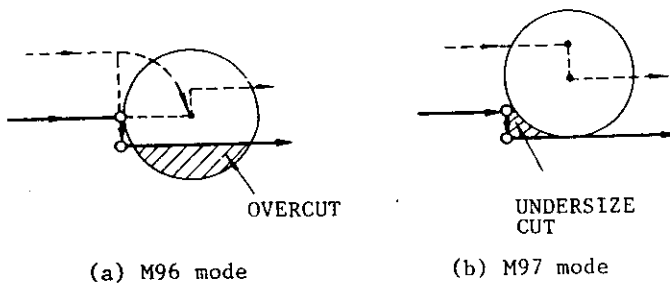


Fig. 2.52

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).

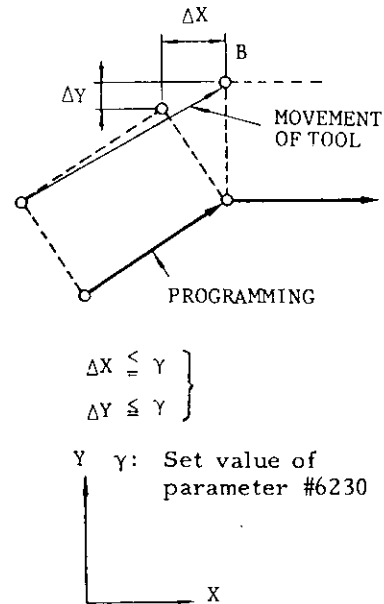


Fig. 2.53

10. Intervention of MDI 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 the compensation mode of G41 or G42 with procedures identical to those of MDI operation, after turning on the SINGLE BLOCK switch to suspend the block, and then, selecting the RAPID or JOG mode.

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

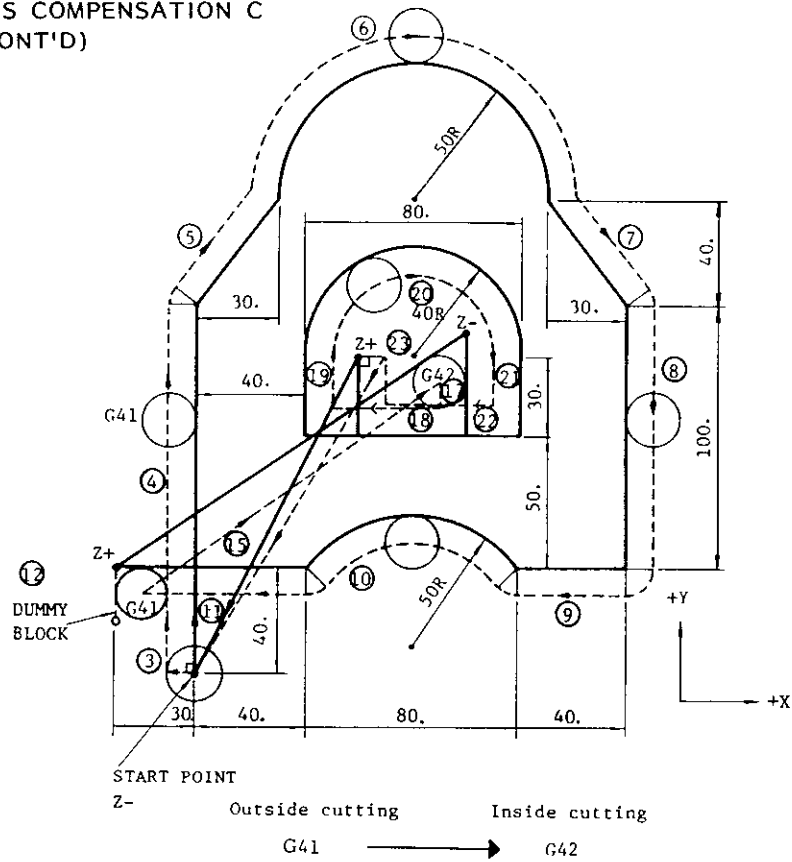
When the CYCLE START button is pushed in the RAPID or JOG mode after programming, the instructions are immediately executed and signals such as BCD output are sent out. Automatic operation can be resumed when CYCLE START is made after returning to the original automatic operation mode.

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

- M00, M01, M02, M30, M90 to M199

2.9.18 TOOL RADIUS COMPENSATION C
(G40, G41, G42)+ (CONT'D)

EXAMPLE A

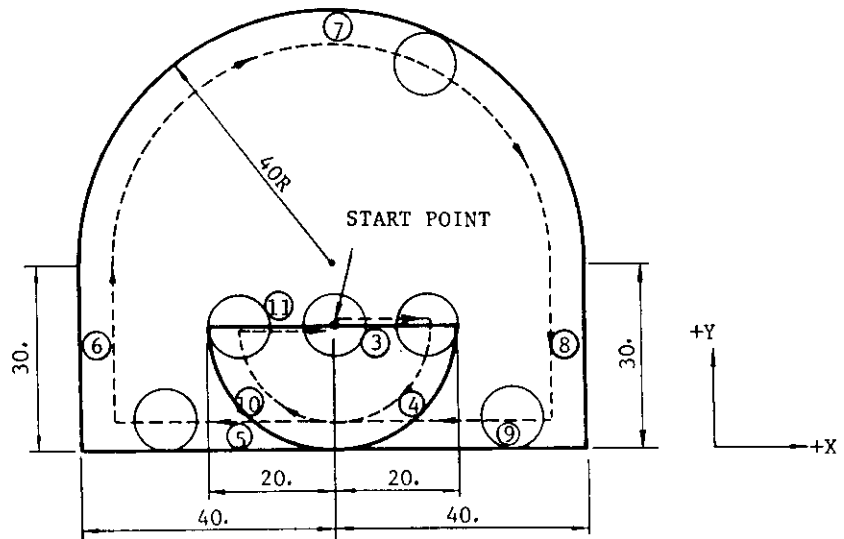


- | | | | |
|-----------------------------|--|----------------------------------|---|
| ① G91 G01 Z-25. ;
F150 ; | Incremental; Z axis lowering | ⑫ J-20. ; | Dummy block (for modifying offset position) |
| ② G17 (G01) F300 ; | XY plane designation, feed command | ⑬ Z25. ; | Z axis elevation |
| ③ G41 D21 ; | Tool radius compensation start command with tool offset No. 21 | ⑭ M01 ; | 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 |
| | | ㉔ Z25. ; | Z axis elevation |
| | | ㉕ G40 (G01) X-60. Y-120. F2000 ; | Compensation cancel command |

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

Fig. 2.54

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. I40. ;
- 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. 2.55

**2.9.19 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... ; (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

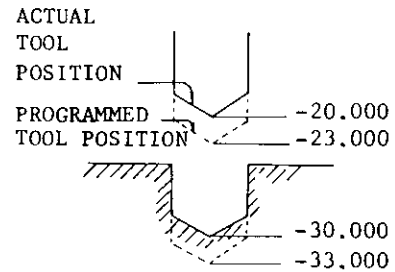
H10 ... Offset value: -3.0
 H11 ... Offset value: 4.0

CRT display including offset value
 (Z direction only)

```

N101 G92 Z0 ;           0.000
N102 G90 G00 X1.0 Y2.0 ; 0.000
N103 G43 Z-20. H10 ;    -23.000
N104 G01 Z-30. F1000 ;  -33.000
N105 G00 Z0 H00 ;      0.000
.
.
.

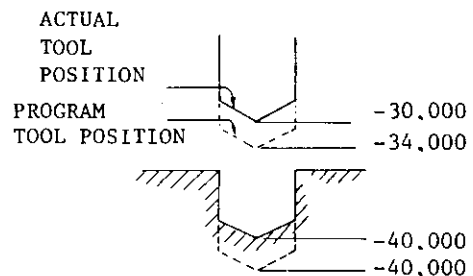
```



```

N201 G00 X-2.0 Y-2.0 ;
N202 G44 Z-30. H11 ;    -34.000
N203 G01 Z-40. F1000 ;  -44.000
N204 G00 Z0 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 this, refer to 4.3.2, "DISPLAY OF COMMAND DATA." G43, G44, and G49 commands must be given in G00 or G01 mode. When G43 is commanded in G02 or G03 mode, no alarm is triggered but correct motion may not be made.

2.9.20 TOOL POSITION OFFSET (G45 TO G48)

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

1. G codes of tool position offset

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.

3. Make program command by incremental designation (G91) for the sake of making the above operation clear. When the command is given by absolute designation (G90), extension and reduction are made along the direction of movement to the movement value from the end point of the preceding block, to the command target point. That is, extension and reduction are made to the incremental movement amount. The programming may become complicated.
4. When programming G45 to G48, designate the tool offset number by a D code simultaneously with axis designation. Because D codes are modal, they may be omitted if the same D code is used. Store the tool radius value in the tool offset value memory.

EXAMPLE

```

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

```

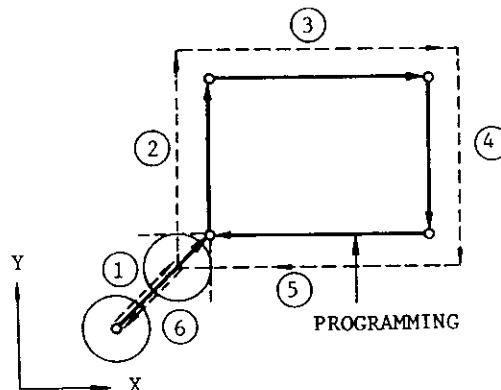


Fig. 2.55

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 G00 G47 X60. D10 ; D10 = 20.
```

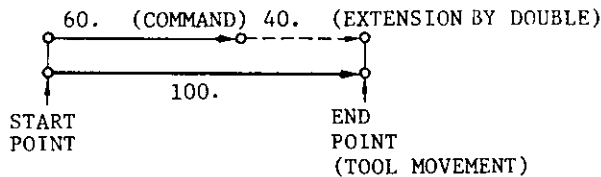


Fig. 2.56

- 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 an instruction same as that described above:

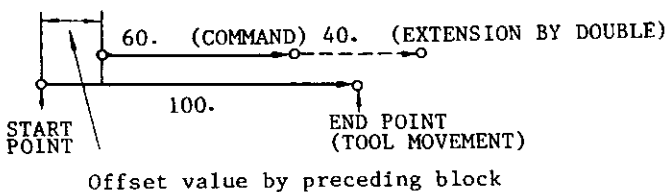


Fig. 2.57

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.
```

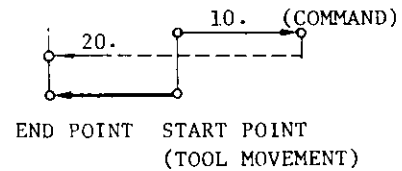


Fig. 2.58

7. The above applies to X and Y axes, 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. I50. D10 ;
D10 = 20.
```

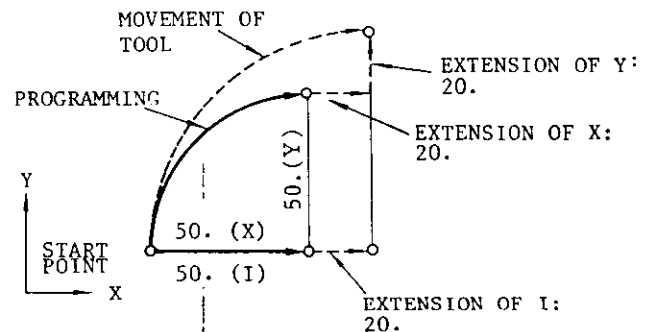
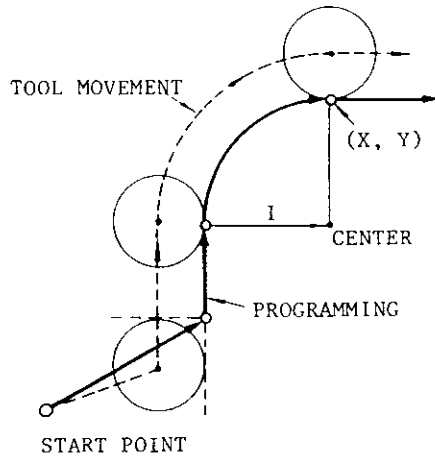


Fig. 2.59

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... ;
```

2.9.20 TOOL POSITION OFFSET (G45 TO G48)
(CONT'D)



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

Fig. 2.60

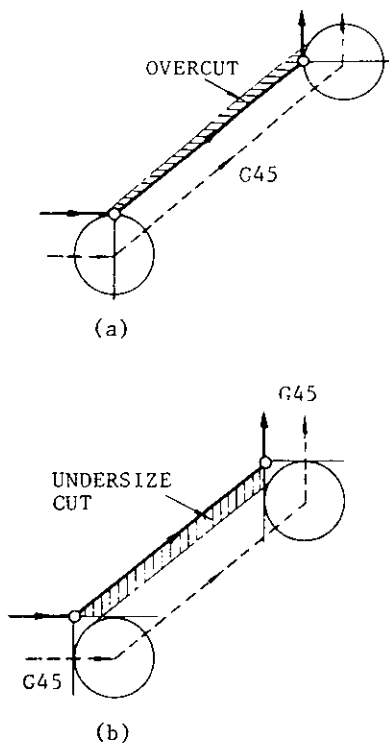


Fig. 2.61

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 instruction is given with other G codes.

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

```
G91 G01 G45 X0 Y0 D10 F... ;
```

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

```
G91 G00 G46 X0 D11 ;
```

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

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

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

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

Setting parameter

```
#6073 D2
```

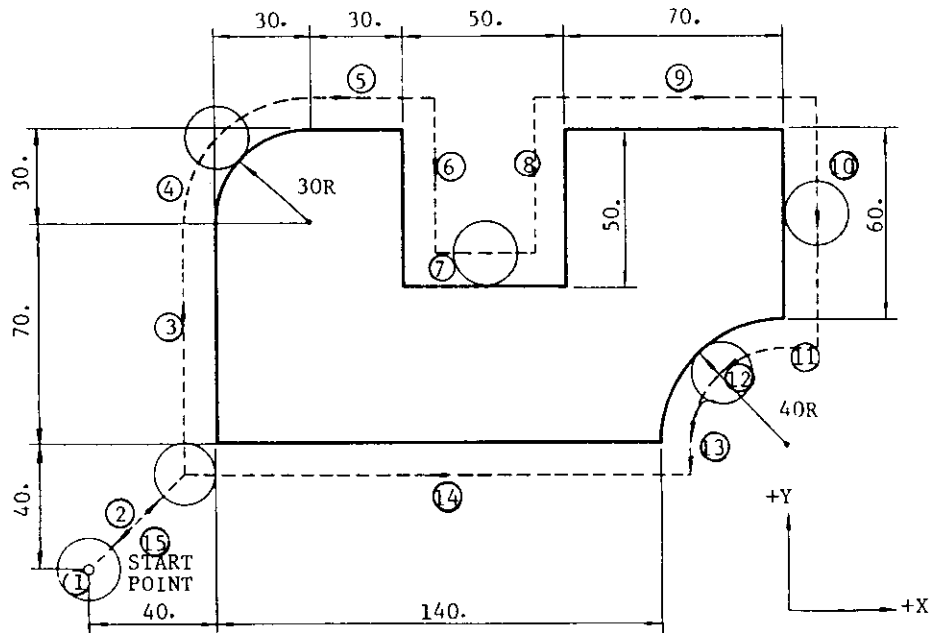
```
= 1: D code only is effective
```

```
= 0: D and H codes are effective
```

Notes :

- When G45 to G48 are programmed as the simultaneous movement instruction along two axes, extension or reduction is made in the two axes. Overcut or undersize cut will occur if this is applied to cutting. Keep this in mind.
- Even when the offset value is changed by MDI, the offset instruction previously programmed will not be affected. It becomes operable when G45 to G48 are programmed thereafter.
- This tool position offset 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 absolute zero point 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

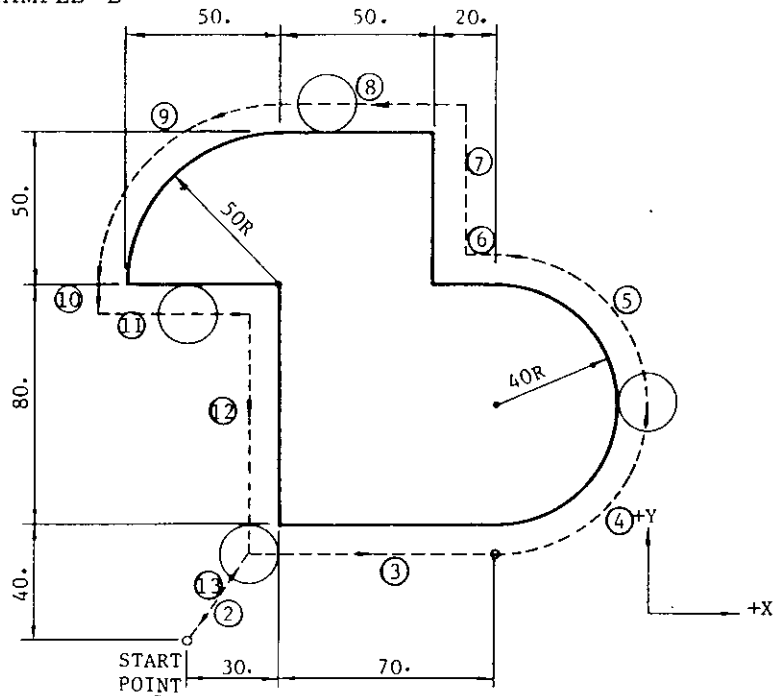


- ① G91 G01 Z-25. F150 ;
- ② G46 X40. Y40. D10 F300 ;
- ③ G45 Y70. ;
- ④ G45 G02 X30. Y30. I30. ;
- ⑤ G45 G01 X30. ;
- ⑥ Y-50. ;
- ⑦ G48 X50. ;
- ⑧ Y50. ;
- ⑨ G47 X70. ;
- ⑩ G47 Y-60. ;
- ⑪ G46 X0 ;
- ⑫ G46 G03 X-40. Y-40. J-40. ;
- ⑬ G46 G01 Y0 ;
- ⑭ G47 X-140. ;
- ⑮ G46 X-40. Y-40. ;
- ⑯ Z25. ;

Fig. 2.62

2.9.20 TOOL POSITION OFFSET (G45 TO G48)
(CONT'D)

EXAMPLE B



- ①
- ① G91 G01 F300 ;
- ② G46 X30. 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

2.9.21 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	0.000001-99.999999
------------------------------	--------------------

When no scale ratio is specified in the program, the ratio set by parameter #6500 becomes effective as the scale ratio.

Command 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 J0 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 zero point and scaling center.

Example

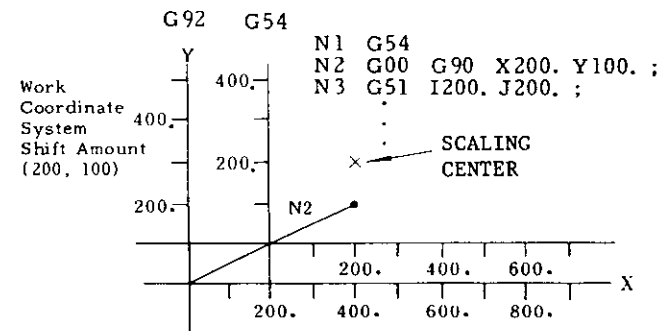
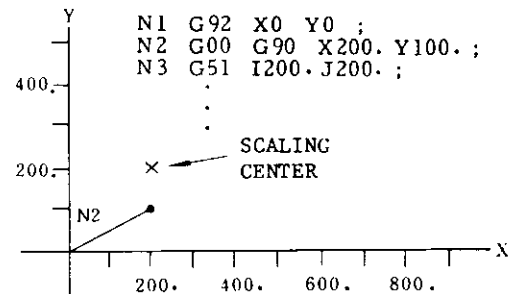


Fig. 2.64

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.

**2.9.21 SCALING FUNCTION (G50, G51)†
(CONT'D)**

- Block commands G51 I... 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.
- Display of command and position will show the values of command and position after scaling is finished.
- The following Gcodes 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 2.26 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 G51 during scaling mode is ignored.

2.9.22A 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 point.

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

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 once 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.

Table 2.27 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.

- 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.

- 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.

EXAMPLE

```

N1 G90 X100. Y200. ;
N2 G54 ;
N3 X100. Y300. ;
N4 X300. Y200. ;
N5 G52 ;
N6 X0 Y0 ;
    
```

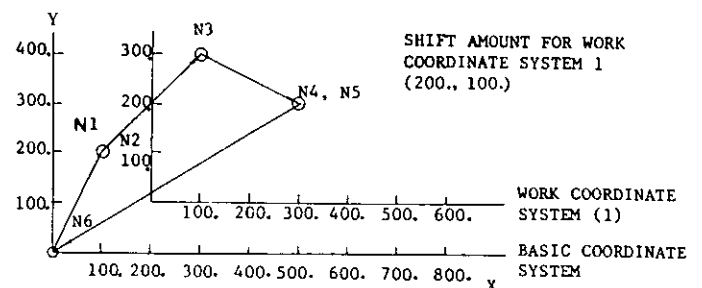


Fig. 2.65

2.9.22A WORK COORDINATE SYSTEM SETTING A
(G52 TO G59)+ (CONT'D)

EXAMPLE (Reference program)

```

N1 G92 X200. Y100. ;
N3 G54 G90 X100. Y200. ;
N4 G53 X300. Y100. ;
N5 X300. Y0 ;
N6 G52 ;
N7 X0 Y0 ;
    
```

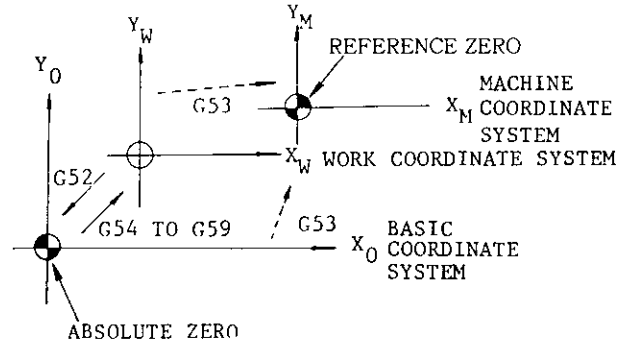


Fig. 2.66

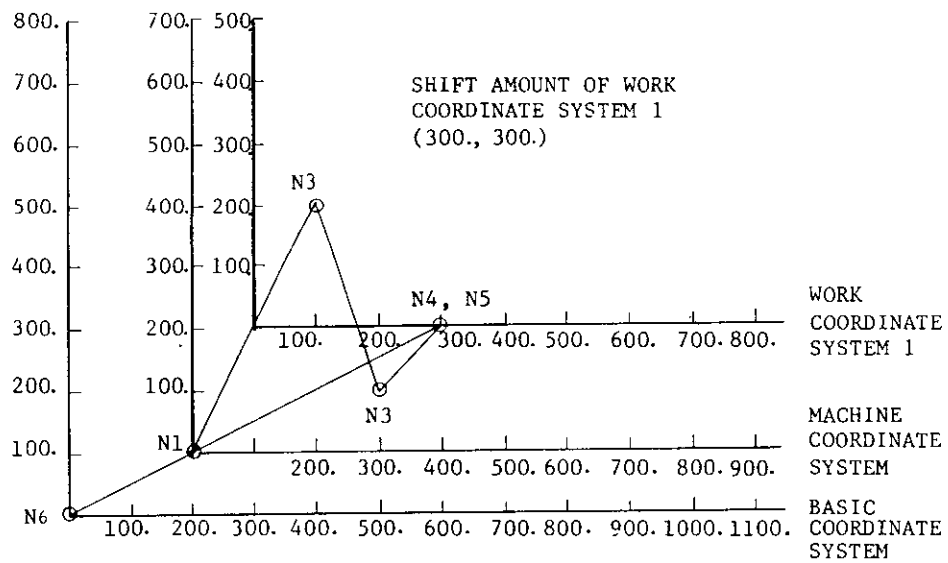


Fig. 2.67

Notes :

- The shift amounts for work coordinate systems can be specified by programs with G10 commands, in addition to the MDI 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 once 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 G90 mode. If they are given in the G91 mode, the command values are regarded as G90 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 Z1100.
```

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.

2.9.22B 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 five axes X,Y,Z, α , β can be set since G54 to G59 and G54J1 to G59J1 are the same. In commands J2 to J5, only three axes X,Y,Z can be set but the 4th and 5th axes cannot be set.

(2) Setting numbers for specifying work coordinate system shift

Table 2.28 Work Coordinate System Setting B

	X	Y	Z	α	β
G54(J1)	#6516	#6517	#6518	#6519	#6520
J2	#6700	#6701	#6702		
J3	#6718	#6719	#6720		
J4	#6736	#6737	#6738		
J5	#6754	#6755	#6756		
G55(J1)	#6522	#6523	#6524	#6525	#6526
J2	#6703	#6704	#6705		
J3	#6721	#6722	#6723		
J4	#6739	#6740	#6741		
J5	#6757	#6758	#6759		
G56(J1)	#6528	#6529	#6530	#6531	#6532
J2	#6706	#6707	#6708		
J3	#6724	#6725	#6726		
J4	#6742	#6743	#6744		
J5	#6760	#6761	#6762		
G57(J1)	#6534	#6535	#6536	#6537	#6538
J2	#6709	#6710	#6711		
J3	#6727	#6728	#6729		
J4	#6745	#6746	#6747		
J5	#6763	#6764	#6765		
G58(J1)	#6540	#6541	#6542	#6543	#6544
J2	#6712	#6713	#6714		
J3	#6730	#6731	#6732		
J4	#6748	#6749	#6750		
J5	#6766	#6767	#6768		
G59(J1)	#6546	#6547	#6548	#6549	#6550
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 or 5th axis of G54(J1) to G59(J1) is provided.

2.9.22B WORK COORDINATE SYSTEM SETTING B (G52 TO G59)† (CONT'D)

Setting Comment Display

SETTING		O1234	N1234
#6520	-12345678	(G54	β)
#6521	0		
#6522	0	(G55	X)
#6523	0	(G55	Y)
#6524	0	(G55	Z)
#6525	0	(G55	α)
#6526	0	(G55	β)
#6527	0		
#6528	0	(G56	X)
#6529	12345678	(G56	Y)
		LSK	RDY

Selection of 4th or 5th 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. G54 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 "129" will result.

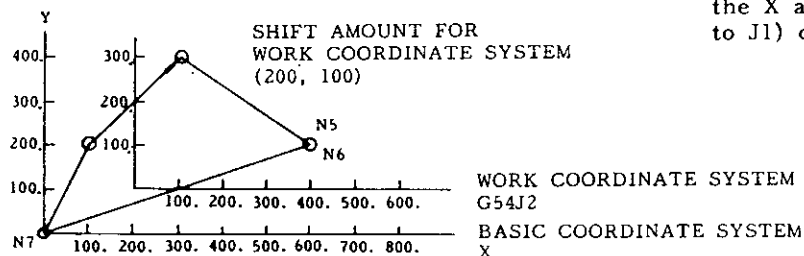
(d) During the J2 to J5 commands, the amount of shift on the 4th and 5th axes will be 0 and the basic coordinate system will become effective again.

(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.



N1 G92 X0 Y0;
 N2 G90 G00 X100. Y200.;
 N3 G54J2;
 N4 X200. Y300;
 N5 X400. Y100;
 N6 G52;
 N7 X0 Y0;

Work Coordinate System 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 G53 command. G53 is a non-modal G code.

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

(8) Changing work coordinate systems with G10

(a) Similar to Specification A, G10 command corrects the work coordinate system from the program.

G10 Q2 Pm Jn X...Y...Z...(α ... β ...);

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

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 J0 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 result if the 4th and 5th axes are 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 axes of #2600's, the Z axes of #2700's, the α axes of #2800's and the B axes of #2900's have the same correspondence as the X axis. Command G54 to G59 (equivalent to J1) only in the case of the α and β axes.

Table 2.29(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 2.29(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 2.29(c)

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	

Table 2.29(d)

System Variables	Work Coordinate System Shift Amount	Axis
#2800	External Work Coordinate system correction amount	α
#2801	G54 (J1)	
⋮	⋮	
#2806	G59 (J1)	
#2900	External Work Coordinate system correction amount	β
#2901	G54 (J1)	
⋮	⋮	
#2906	G59 (J1)	

2.9.22B WORK COORDINATE SYSTEM SETTING B
(G52 TO G59)+ (CONT'D)

Table 2.29 (e)

(d) Program examples

(i) #116 = #2501;

Command G54(J1) assigns the amount of work coordinate system shift of the X axis as the common variable #116.

(ii) #2511 = #4

Command G54 J2 cancels the amount of work coordinate system shift of the 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 means of 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(J2) 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(J2) to G59 (J5).

(11) Notes and remarks

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

(b) Use the G54J2 to J5; commands in the G00 or G01 mode. Alarm "129" will result if used in any other mode.

2.9.22C 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 and 5th axes.

(2) The item on the rotary angle is added to the work coordinate system setting. The 4th and 5th axes can be set to all the work coordinate systems at the same time. See Table 2.29 (e) for the setting numbers.

	X	Y	Z	α	β	R
G54(J1)	#6516	#6517	#6518	#6519	#6520	#6521
J2	#6700	#6701	#6702	#6703	#6704	#6705
J3	#6736	#6737	#6738	#6739	#6740	#6741
J4	#6772	#6773	#6774	#6775	#6776	#6777
J5	#6808	#6809	#6810	#6811	#6812	#6813
G55(J1)	#6522	#6523	#6524	#6525	#6526	#6527
J2	#6706	#6707	#6708	#6709	#6710	#6711
J3	#6742	#6743	#6744	#6745	#6746	#6747
J4	#6778	#6779	#6780	#6781	#6782	#6783
J5	#6814	#6815	#6816	#6817	#6818	#6819
G56(J1)	#6528	#6529	#6530	#6531	#6532	#6533
J2	#6712	#6713	#6714	#6715	#6716	#6717
J3	#6748	#6749	#6750	#6751	#6752	#6753
J4	#6784	#6785	#6786	#6787	#6788	#6789
J5	#6820	#6821	#6822	#6823	#6824	#6825
G57(J1)	#6534	#6535	#6536	#6537	#6538	#6539
J2	#6718	#6719	#6720	#6721	#6722	#6723
J3	#6754	#6755	#6756	#6757	#6758	#6759
J4	#6790	#6791	#6792	#6793	#6794	#6795
J5	#6826	#6827	#6828	#6829	#6830	#6831
G58(J1)	#6540	#6541	#6542	#6543	#6544	#6545
J2	#6724	#6725	#6726	#6727	#6728	#6729
J3	#6760	#6761	#6762	#6763	#6764	#6765
J4	#6796	#6797	#6798	#6799	#6800	#6801
J5	#6832	#6833	#6834	#6835	#6836	#6837
G59(J1)	#6546	#6547	#6548	#6549	#6550	#6551
J2	#6730	#6731	#6732	#6733	#6734	#6735
J3	#6766	#6767	#6768	#6769	#6770	#6771
J4	#6802	#6803	#6804	#6805	#6806	#6807
J5	#6838	#6839	#6840	#6841	#6842	#6843

(3) Setting command display

SETTING	O1234	N1234
#6520	-12345678	(G 54 β)
#6521	3000	(G 54 R)
#6522	0	(G 55 X)
#6523	0	(G 55 Y)
#6524	0	(G 55 Z)
#6525	0	(G 55 α)
#6526	0	(G 55 β)
#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 and 5th axes specification.

Fig. 2.67A

(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 is commanded, the shift amount of the 4th and 5th axes 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 X0 Y0 Z0;
 • Nothing changes under G92.

G54;
 • (G68 X0 Y0 R (value of #6521))
 • Shifts the work coordinate system by G54, and rotates the coordinate system for R,
 • centering the work coordinate system (0, 0).

G55;
 • (G69)
 • (G68 X0 Y0 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) G00 X... Y... Z...
 (α ... β ...);

(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... α... β... 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 J0 is specified, it is regarded as J1.

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

(d) The 4th and 5th axes commands are 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 2.29(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	

2.9.22C WORK COORDINATE SYSTEM SETTING C
(G52 TO G59)+ (CONT'D)

Table 2.29(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 2.29(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	

Table 2.29(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 2.29(j)

System Variables	Work Coordinate System Shift Amount	Axis
#2900	External Work coordinate system correction amount	β
#2901	G54 (J1)	
⋮	⋮	
#2906	G59 (J1)	
#2911	G54 J2	
⋮	⋮	
#2916	G59 J2	
#2921	G54 J3	
⋮	⋮	
#2926	G59 J3	
#2931	G54 J4	
⋮	⋮	
#2936	G59 J4	
#2941	G54 J5	
⋮	⋮	
#2946	G59 J5	

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

Table 2.29(k)

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	

(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 G00 to G01 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 0 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.

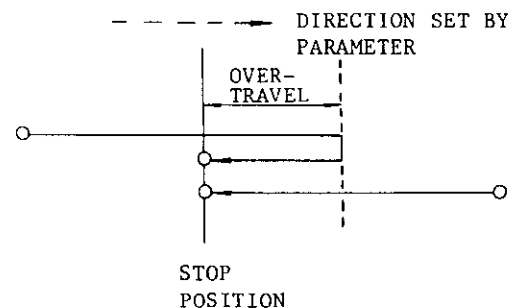
(f) The coordinate rotation plane of this specification is fixed to the G17 plane.

2.9.23 UNIDIRECTIONAL APPROACH (G60)+

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

· G60 X... Y... Z... (α†...β) ;

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 - #6440) once, and then returns to the specified position to stop.



Note: Unidirectional approach is effective in the canned cycle mode. Shift by G76 includes unidirectional approach. Cancel this function before executing G76 for safety purposes.

Fig. 2.68

2.9.24 HOLE PATTERN CYCLES (G70, G71, G72)+ Arc cycle (G71)

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)

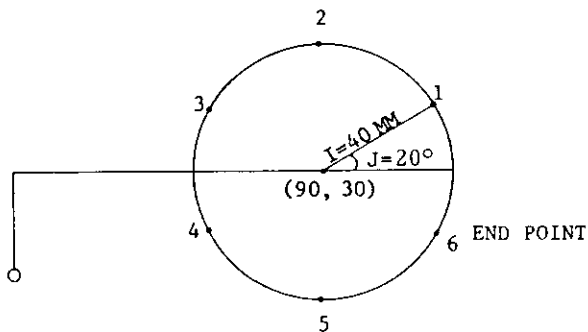
G70 X... Y... I... J... L... ;

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 I, starting at a point located on a line forming J degree with the X axis. In the command,

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

- I is the radius of the bolt hole circle, programmed in a positive number and programmed with an accuracy of the least input increment.
- J is the angular position of the first hole, programmed in degrees with an accuracy of 0.001 degree. CCW direction is regarded positive.
- L is the 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. I40. J20. L6 ;
G80 G00 X0 Y0 ;
```

Fig. 2.69

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

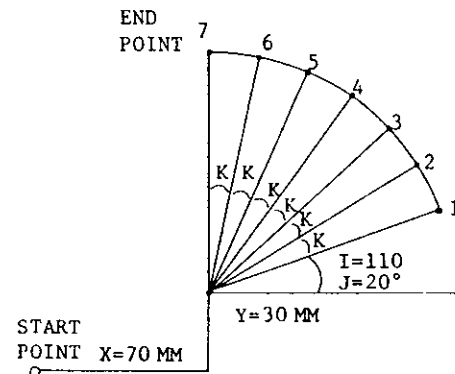
I: Radius of the arc programmed with an accuracy of the least programmable increment, and in positive numbers

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. I110. J20. K15.2 L7 ;
G80 G00 X0 Y0 ;
```

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 the X axis, with a uniform interval of I, starting at X, Y.

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

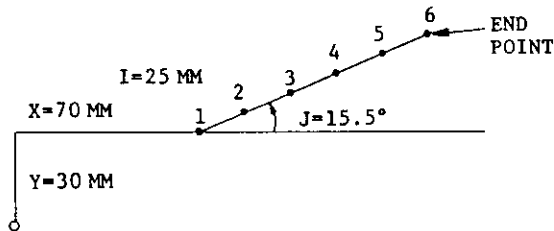
X, Y: Coordinates of the starting point, either in G90 or G91 mode

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. I25. J15.5 L6 ;
G80 G00 X0 Y0 ;
```

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 may 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.

- 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. Make it a point 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.

2.9.25 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99)+

Canned cycles (G73, G74, G76, G77, G80 to G89, G98, G99) 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 cancelling them.

Table 2.30

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:

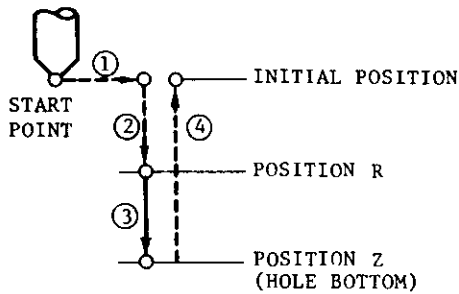
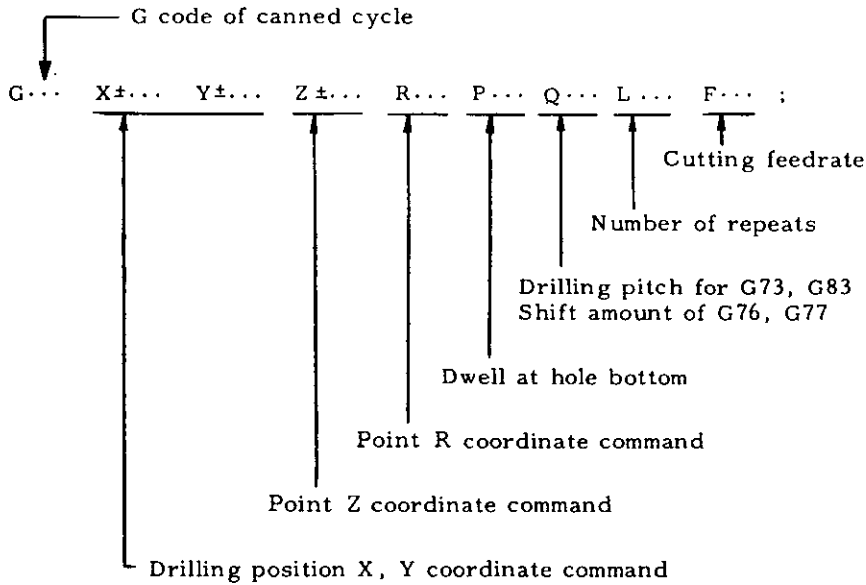


Fig. 2.72

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	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).

2.9.25 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99)+ (CONT'D)

Table 2.31 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>G73 X... Y... Z... R... Q... L... F... ;</p> <p>δ: Setting data</p>
<p>G73 (Variable)</p> <p>High speed deep hole drilling</p>	<p>G73 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 (#6211)</p>	<p>G73 X... Y... Z... R... I... J... K... L... F... ;</p> <p>δ: Setting data</p>
<p>G74</p> <p>Reverse tapping</p>	<p>G74 X... Y... Z... R... P... L... F... ;</p> <p>SPINDLE REVERSE</p> <p>SPINDLE FORWARD AFTER DWELL</p>	<p>G74 X... Y... Z... R... P... L... F... ;</p> <p>SPINDLE REVERSE</p> <p>SPINDLE FORWARD AFTER DWELL</p>

Table 2.31 Canned Cycle (continued)

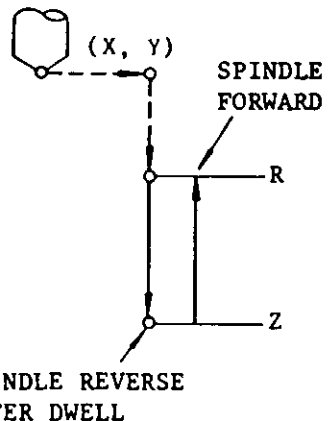
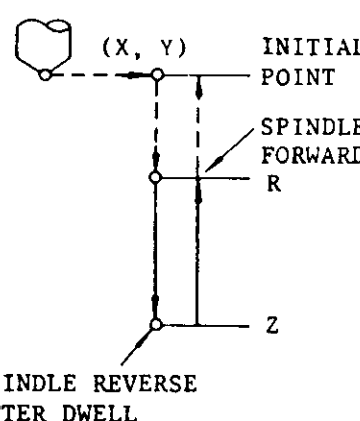
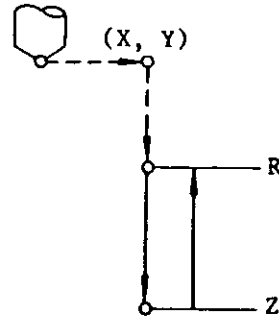
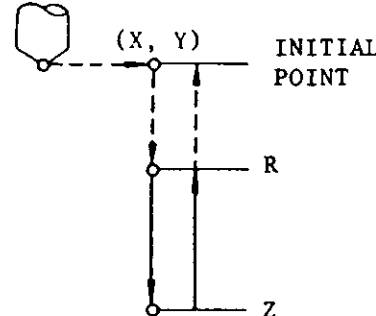
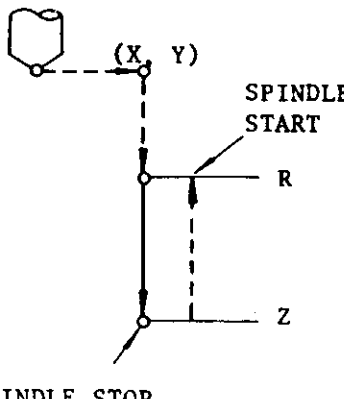
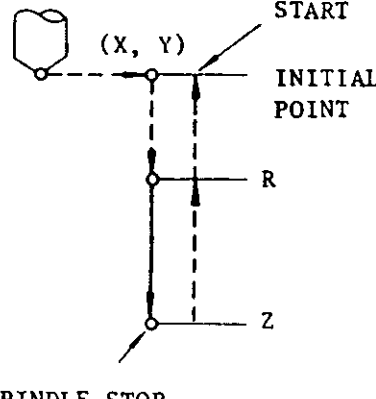
	With G99 (return to R)	With G98 (return to initial point)
G76 Boring	<p>G76 X... Y... Z... R... Q... L... F... ;</p>	
G77 Back boring	<p>G77 X... Y... Z... R... Q... L... F... ;</p> <p>NOT USED</p>	
G80 Cancel	G80 ;	
G81 Drilling	<p>G81 X... Y... Z... R... L... F... ;</p>	

2.9.25 CANNED CYCLES (73, G74, G76, G77, G80 TO G89, G98, G99)+ (CONT'D)

Table 2.31 Canned Cycle (continued)

	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>INITIAL POINT</p> <p>δ: 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>INITIAL POINT</p> <p>δ: Setting data</p>

Table 2.31 Canned Cycle (continued)

	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>SPINDLE REVERSE AFTER DWELL</p>	 <p>INITIAL POINT</p> <p>SPINDLE FORWARD</p> <p>SPINDLE REVERSE AFTER DWELL</p>
G85	G85 X... Y... Z... R... L... F... ;	
Boring		 <p>INITIAL POINT</p>
G86	G86 X... Y... Z... R... L... F... ;	
Boring	 <p>SPINDLE START</p> <p>SPINDLE STOP</p>	 <p>SPINDLE START</p> <p>INITIAL POINT</p> <p>SPINDLE STOP</p>

2.9.25 CANNED CYCLES (G73, G74, G76, G77, G80, TO G89, G98, G99)+ (CONT'D)

Table 2.31 Canned Cycle (continued)

	With G99 (return to R)	With G98 (return to initial point)
G87	G87 X... Y... Z... R... Q... 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

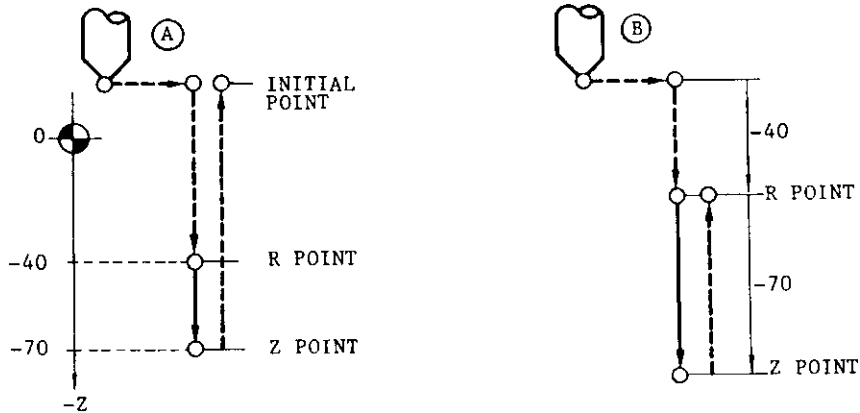
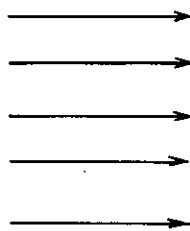


Fig. 2.73

- C. N1 G92 X0 Y0 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 + N3 indicated in the above case. As for the non-programmed addresses, the positions programmed in the earlier blocks are maintained.

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

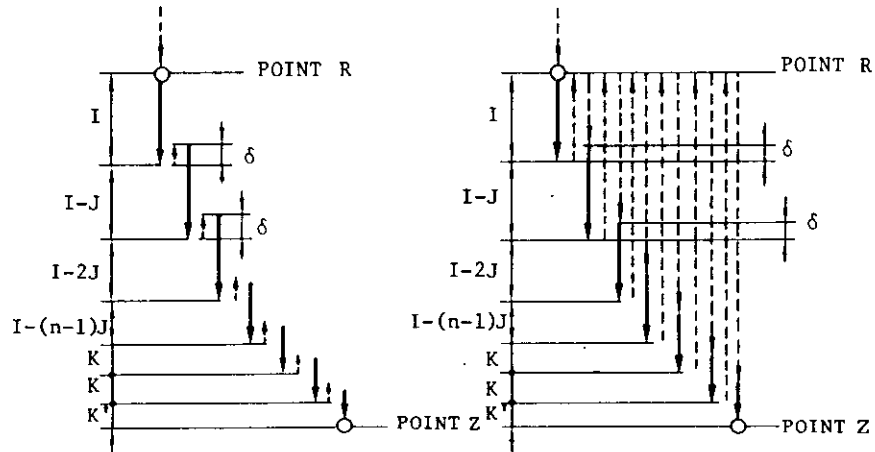
2.9.25 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89, G98, G99)+ (CONT'D)

- 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 instructions 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.

Drilling pitch

1st plunge ...	10 mm	←	I10.
2nd plunge ...	9 mm		
3rd plunge ...	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		←	Z-55.

EXAMPLE

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

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 point and Z point by programming R and Z before entering the canned cycle mode. R point and Z point are cleared when canned cycles are cancelled.
- When executing canned cycles with the address data changed, the block requires any of the following address 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" is triggered 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 made when a G code of 01 group is programmed during canned cycles.

- An program error "024" is triggered when canned cycles are programmed in the tool radius compensation C mode (G41, G42).
- 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 No. 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, α, 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 X0 Y0 Z0
G91 X... Y... R-5.0

	Point R	Point Z
Z-10.0F ;	-5.0	-10.0
X... R-7.0 ;	-7.0	-12.0
X... Z-3.0 ;	-7.0	-10.0
R-4.0Z-11.0 ;	-4.0	-15.0

2.9.25 CANNED CYCLES (G73, G74, G76, G77,
G80 TO G89, G98, G99)+ (CONT'D)

EXAMPLE

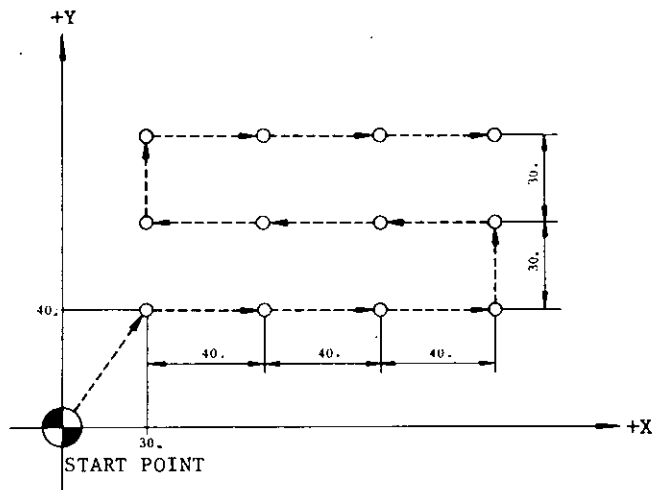


Fig. 2.75

```

.
.
.
N10 G92 X0 Y0 Z0 ;
N11 G90 G98 ; . . . Return to initial point, Absolute
N12 G81 X30. Y40. R-20. Z-30. F200 ; . . . Drilling cycle
N13 M98 P100 ; . . . Jump to subprogram
N14 G00 X0 Y0 ;
N15 T05 ; . . . Tapper selection
N16 M06 ; . . . Tool change
N17 G84 X30. Y40. R-20. Z-30. F2000 ; . . . Tapping cycle
N18 M98 P400 ; . . . Jump to subprogram (Note)
N19 G00 X0 Y0 ;
.
.
.
O400
N100 G91 X40. L3 ;
N101 Y30. ;
N102 X-40. L3 ;
N103 Y30. ;
N104 X40. L3
N105 G90 G80 ;
N106 M99 ;

```

} Subprogram for drilling position pattern.

9.25A 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 MX3 canned cycles.

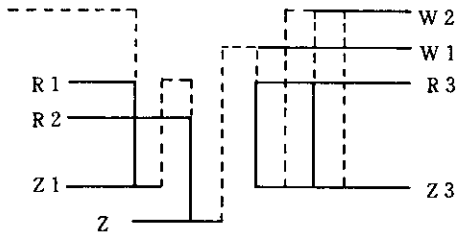
- Initial point alteration
- Improved command procedure for the high-speed deep hole drilling cycle
- 2-step feed
- Reciprocal feed
- Improved command procedure for boring and back boring
- 2-step drilling cycle
- Improved command procedure for the tapping cycle

2.9.25A.1 VARIOUS FUNCTIONS

(1) Initial point alteration

It is no longer necessary to cancel the canned cycle once before changing the initial point in the canned cycle. Address W is used.

Example



```
G99G81 X...Y...R 1...Z 1...F... ;
G98 X...Y...R 2...Z 2...W 1... ;
X...Y...R 3...Z 3...W 2... ;
X...Y... ;
```

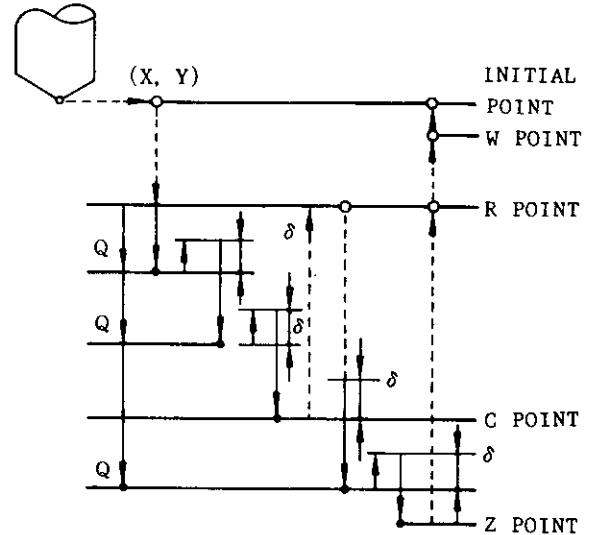
Notes:

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.

2.9.25A.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.

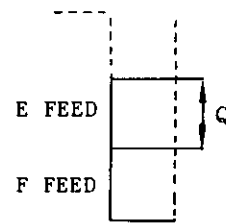
(2) The drill can be moved up to the R point during high-speed drilling. This allows the chips to be removed during the cycle.



• : Dwell
○ : Single-block stop

2.9.25A.3 2-STEP FEED

Delaying the cutting speed near the drilling start point can improve the precision without performing center drilling.

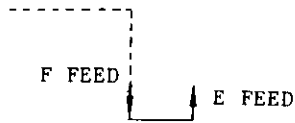


Address E: Speed of the 1st step
Address F: Distance of the 1st step speed (commanded without sign)

2.9.25A.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

2.9.25A.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.

```
G76 X...Y...Z...R...I...J...P...;
      |
      | Dwell time
      |
      | Shift direction designation
```

2.9.25A.6 2-STEP DRILLING CYCLE

Two-step drilling cycle is provided to correspond with drilling cycles of hollow workpieces, 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).

2.9.25A.7 IMPROVED COMMAND PROCEDURE FOR THE TAPPING CYCLE

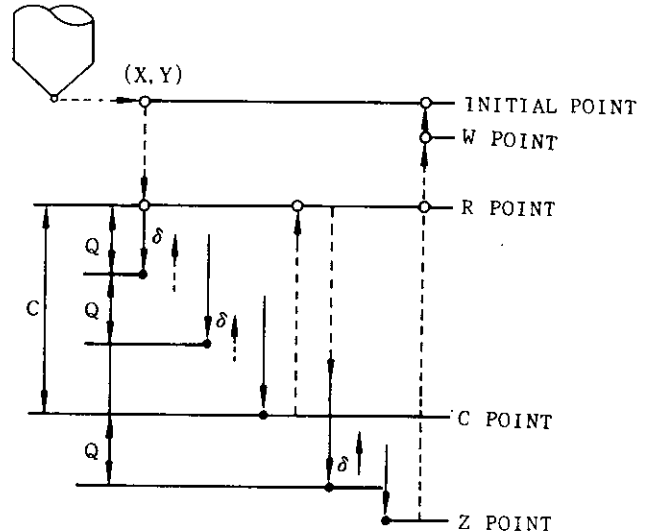
In the tapping cycle, dwell can be commanded upon R point return and initial point return, in addition to the hole-bottom dwell. This can absorb the elongation of the taper.

G73 High-speed deep hole drilling

```
G73 X... Y... Z... R... Q... L... F... C... P...
      W... ; (Fixed pitch)
```

```
G73 X... Y... Z... R... I... J... K... L... F...
      C... P... W... ; (Variable pitch)
```

Example: Fixed pitch



• : Dwell
◦ : Single-block stop

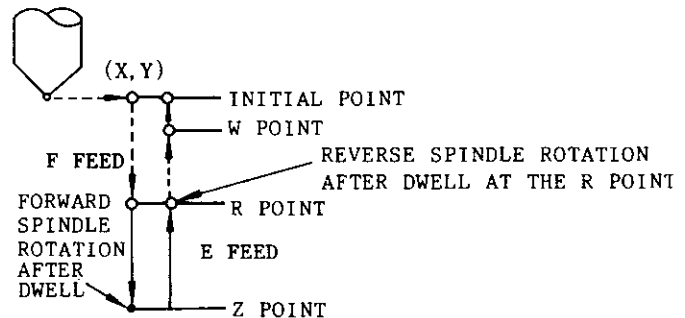
C : Temporary R point return depth (increment)
P : Hole-bottom dwell
W : New initial point (absolute/incremental)
: Setting data (#6211)

Notes:

- 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 ms). PO also follows #6212.
- The default of the hole-bottom dwell can also be set by #6212.
- C and Q are incremental amounts without sign.

G74 Reverse tapping

```
G74 X... Y... Z... R... P... L... F... E... Q...
      W... ;
```



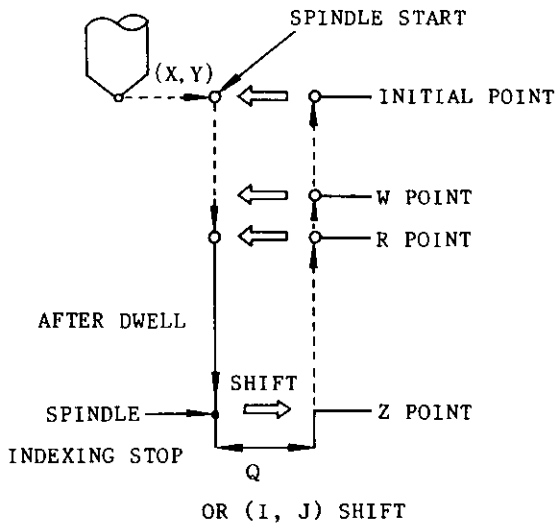
• : Dwell
◦ : Single-block stop

E : Return feedrate (Z point → R point)
Q : 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)



◦ : Dwell
○ : Single-block stop

I : X-axis shift amount (incremental)
J : Y-axis shift amount (incremental)
P : Dwell
W : New initial point

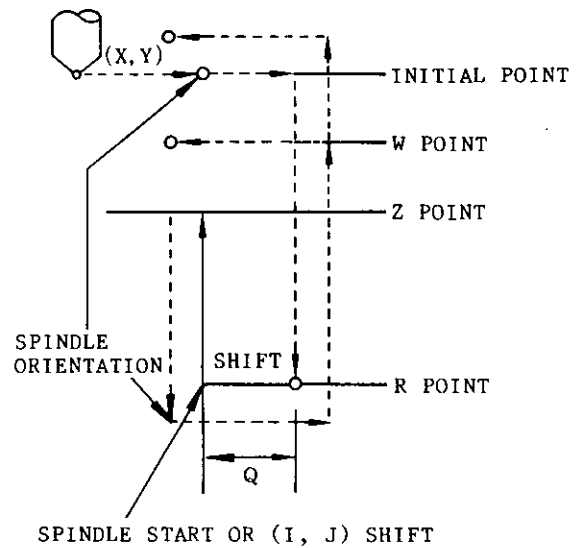
Notes:

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 D0 is ON.
4. Address Q is modal, so a Q command before the I or J, will disregard the I or J. 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 Dog 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)



◦ : Single-block stop

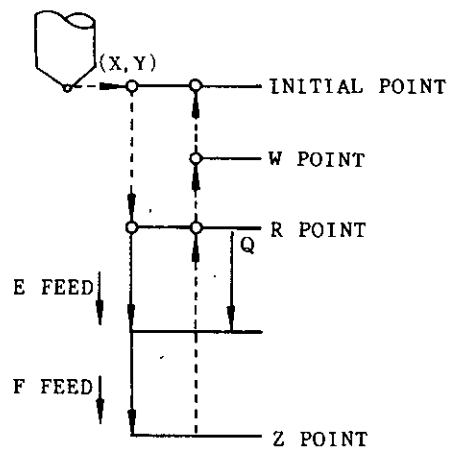
I : X-axis shift amount (incremental)
J : Y-axis shift amount (incremental)
P : Dwell
W : New initial point

Notes:

1. Addresses I and J are disregarded when address Q and address I or J are commanded at the same time.
2. In the dwell after back boring, the program command has priority over the #6210 setting.
3. The shift speed is set in #6223.

G81 Dwell

G81 X... Y... Z... R... L... F... E... Q...
W... ;



◦ : Single-block stop

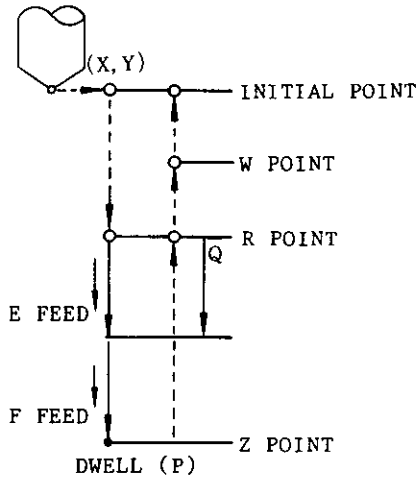
E : 2-step feedrate (1st step: E, 2nd step: F)
Q : Speed switching point (incremental)
W : New initial point (Absolute/incremental)

Note: Return to the R point or initial point after reaching the Z point is made by rapid feed.

2.9.25A.7 IMPROVED COMMAND PROCEDURE FOR THE TAPPING CYCLE (Cont'd)

G82 Spot facing

G82 X... Y... Z... R... P... L... F... E... Q...
W... ;



- : Dwell
- : Single-block stop

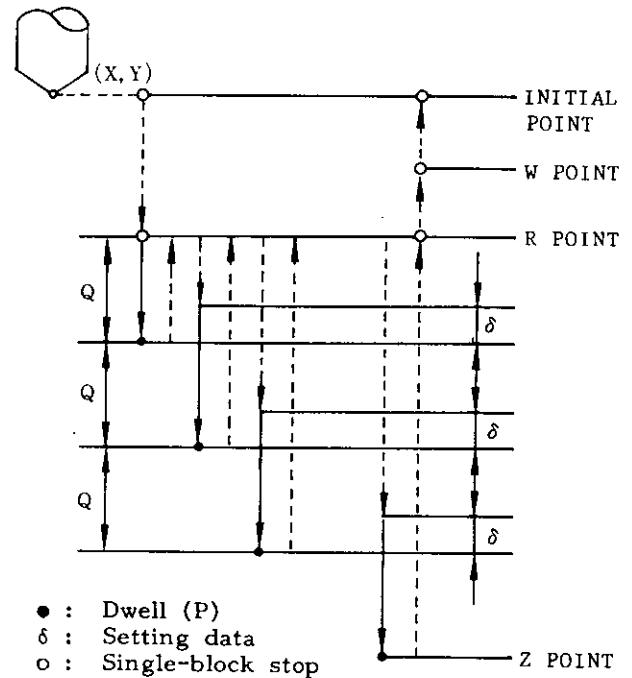
E : 2-step feedrate (1st step: E, 2nd step: F)
Q : Speed switching point (incremental)
W : New initial point (Absolute/incremental)

Note: 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)



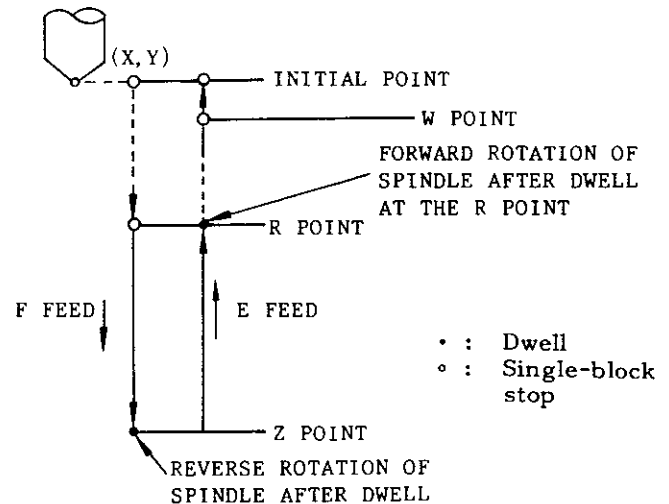
- : Dwell (P)
- δ : Setting data
- : Single-block stop

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

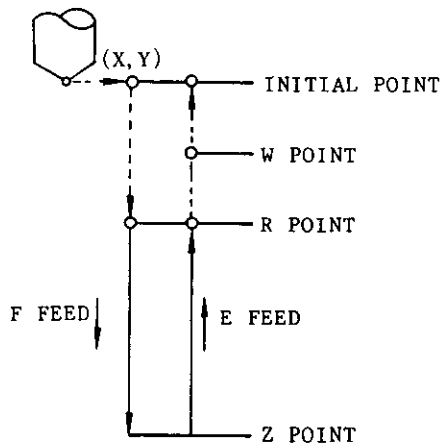
G84 X... Y... Z... R... P... L... F... E... Q...
W... ;



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... ;

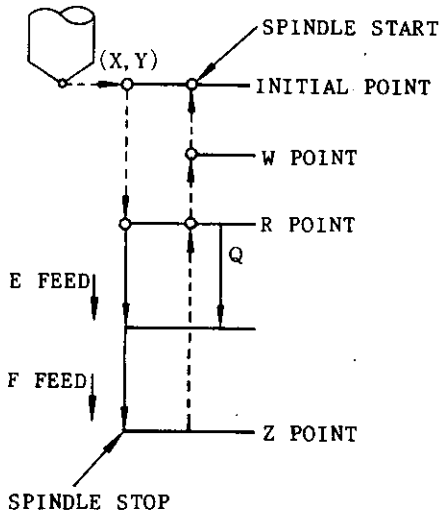


○ : Single-block stop

E : Return cutting feedrate
W : New initial point (Absolute/incremental)

G86 Boring

G86 X... Y... Z... R... L... F... E... Q...
W... ;



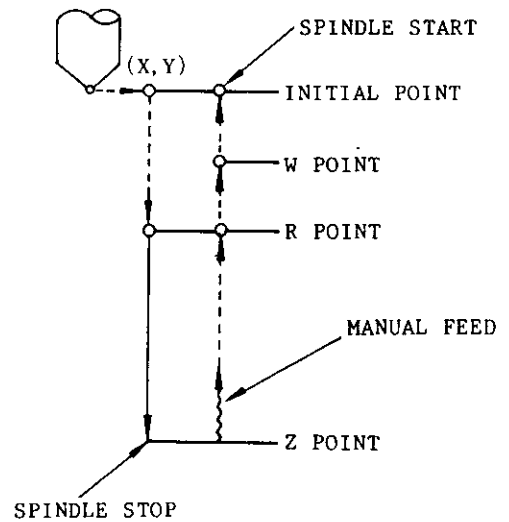
○ : Single-block stop

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... ;

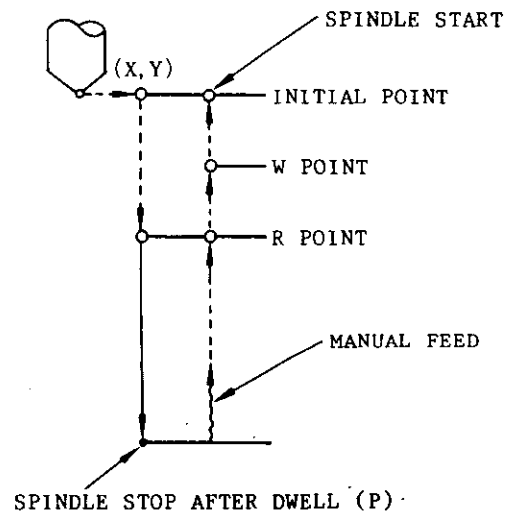


○ : Single-block stop

W : New initial point (absolute/incremental)

G88 Boring

G88 X... Y... Z... R... P... L... F... W... ;



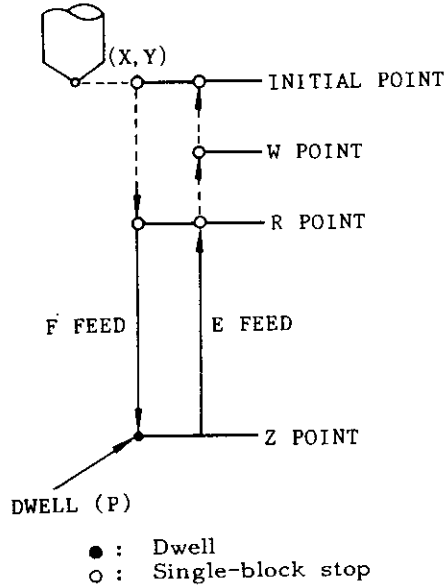
● : Dwell
○ : Single-block stop

W : New initial point (absolute/incremental)

2.9.25A.7 IMPROVED COMMAND PROCEDURE FOR THE TAPPING CYCLE (Cont'd)

G89 Boring

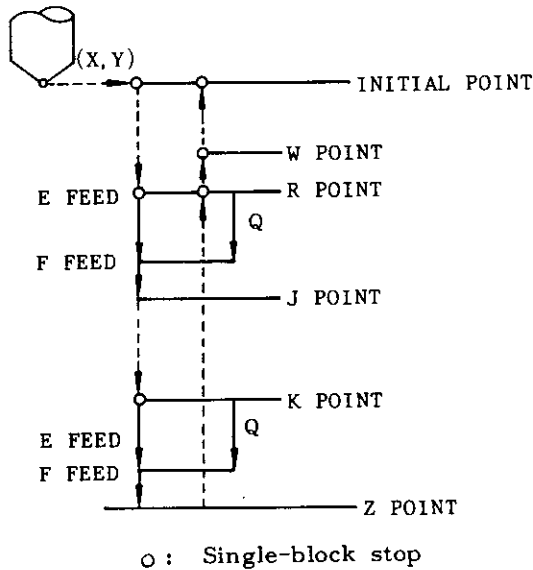
G89 X... Y... Z... R... P... L... F... E...
W... ;



E : Return cutting feedrate
W : New initial point

G181 2-step drilling

G181 X... Y... Z... R... L... F... J... K...
E... Q... W... ;

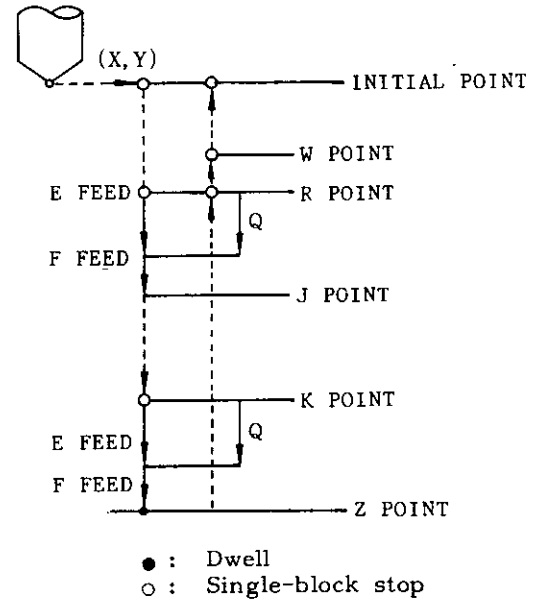


J : 2-step hole rapid feed start point (absolute/incremental)
K : 2-step hole cutting feed start point (absolute/incremental)
E : 2-step feedrate (1st step: E, 2nd step: F)
Q : Speed switching point (incremental)
W : New initial point (absolute/incremental)

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... ;

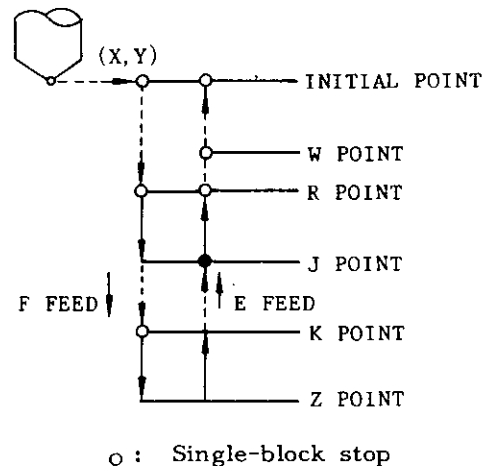


J : 2-step hole rapid feed start point (absolute/incremental)
K : 2-step hole cutting feed start point (absolute/incremental)
E : 2-step feedrate (1st step: E, 2nd step: F)
Q : Speed switching point (incremental)
W : New initial point (absolute/incremental)

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... J... K...
E... W... ;

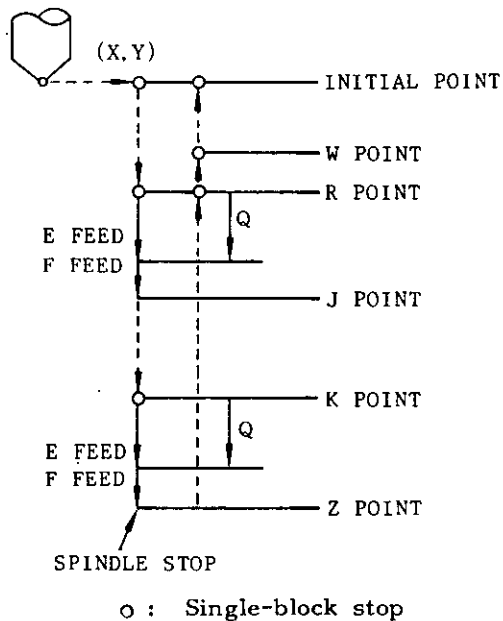


- J : 2-step hole rapid feed start point (return: cutting feed start point) (absolute/incremental)
- K : 2-step hole cutting feed start point (return: rapid feed start point) (absolute/incremental)
- E : Return feedrate
- W : New initial point (absolute/incremental)

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... ;

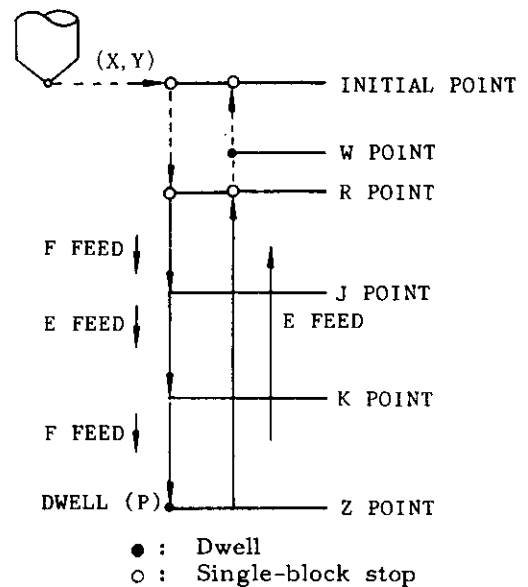


- E : 2-step feedrate (1st step: E, 2nd step: F)
- Q : Speed switching point (incremental)
- J : 2-step hole rapid feed start point (absolute/incremental)
- K : 2-step hole cutting feed start point (absolute/incremental)
- W : New initial point (absolute/incremental)

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... ;

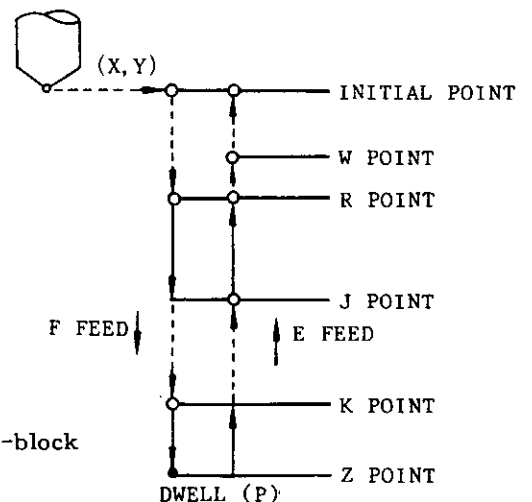


- P : Z point dwell
- J : 2-step hole rapid feed start point (return: cutting feed start point) (absolute/incremental)
- K : 2-step hole cutting feed start point (return: rapid feed start point) (absolute/incremental)
- E : Feedrate between J and K, and return cutting feedrate
- W : New initial point (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... ;



- 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.

2.9.26 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, ($\alpha^\dagger \beta$) are regarded as absolute values.

```
G90 G00 X... Y... Z... ;
          ... Absolute designation
```

- G91 . . . Incremental designation

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

```
G91 G01 X... Y... Z... ;
          ... Incremental designation
```

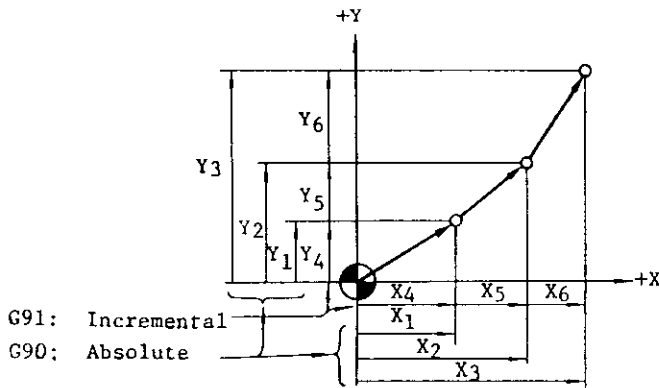


Fig. 2.76

- 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"	G90
"1"	G91

2.9.27 PROGRAMMING OF ABSOLUTE ZERO POINT (G92)

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

- G92 X... Y... Z... ($\alpha^\dagger \dots \beta \dots$) ;

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

EXAMPLE

```
G92 X500. Y300. Z400. ;
```

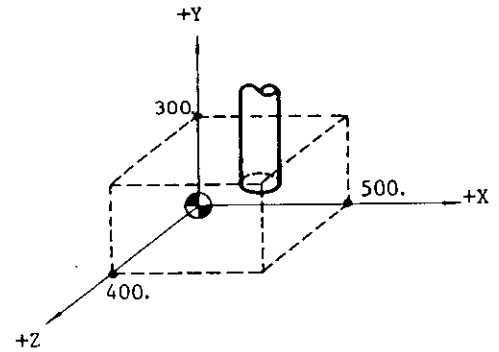


Fig. 2.77

- 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 G92 before executing the automatic operation.
- The programmed absolute zero point is not affected by reset operation. Perform any of the following operations for resetting the absolute zero point.
 1. Use ORG key (see 4.1.9).
 2. Write G92 X0 Y0 Z0 α^\dagger 0 β 0 ; in MDI mode, and then execute.
 3. Turn the power off and on again.

2.9.28 TOOL LIFE CONTROL (G122, G123)

2.9.28.1 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 controlled...
 .. 256 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 only 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, however, 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 O1234 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									

2.9.28.2 TOOL LIFE CONTROL DATA DISPLAY

With the function set to **[OFS]**, tool life control data as shown below will be possible by pressing **[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 you desire to index.

(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">

- Press the **[OFS]** key and page to the tool life control display.
- Press the **[PAGE]** key and page to the tool life control data display.
- Key-in the numerical value of the object group.
- Press the **[CURSOR]** key. Data display of the object group will then appear.

GROUP NO.		LIFE CLASSIFICATION			
TOOL LIFE CONTROL		O1234 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
T0024	100	101	500	153	SKP
T9001	212	214	1000	25	
T0005	023	022	9000	00	
T1278	*	*	500	0	
T0054	099	100	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		

2.9.28.3 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

- Press the **[OFS]** key and page to the tool life control display.
- Use the **[PAGE]** key or group search and search the group to be rewritten.
- 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.)
- Key-in the numerical value.
- Press the **[WR]** key. The data at the cursor position will then be rewritten.

(b) Write of 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.

2.9.28.3 TOOL LIFE CONTROL DATA SETTING
(CONT'D)

(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.

(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 **O** and **WR** and life classification will change to MINUTES. Key-in **1** and **WR** and life classification will change to COUNTS. Key-in **2** and **WR** and 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 2.32 Data Erasure Method

Erasure Type	Page	Cursor Position	Key-in Data
Erasure of all groups	All pages	Possible at any place.	G - 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 "0."

(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 2.33 Tool Life Control Data Format

Tape Format	Meaning
G122;	Tool registration start.
P□□□IΔ;	Commands group No. after P. Commands life classification after I.
TΔΔΔH○○○D□□□ L×××;	Specifies tool No. after T. Specifies tool length compensation No. after H. Specifies tool radius compensation No. after D. Specifies tool life(1-9999) after L.
TΔΔΔH○○○D□□□ L×××;	
⋮	
P□□□IΔ;	
TΔΔΔH○○○D□□□ L×××;	Sets new group data.
G123	Tool registration end.

(c) G code (G124) 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 G124;. If command G122 is entered without executing G124, the following will result.

(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 cancelling 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 G122 and G123 so these commands can be omitted when setting to the same tool life.

(v) The following life classifications can be set by specifying I.

- 10.... Controlling by time (1 to 9999 minutes)
- 11.... Controlling by counts (1 to 9999 times)
- 12.... Controlling by distance (1 to 9999 inches
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 commandable range. Any other command values will result in an error.
 $0 \leq T \leq 9998$, $0 \leq H$, $D \leq 299$, $0 \leq L \leq 9999$, $1 \leq P \leq 128$, $0 \leq I \leq 2$

(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 G122; ... G123; section. (Alarm will result if such a command is entered.)

(ix) Do not switch over the mode to execute a manual MDI command while executing G122;. (Alarm will result 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 be in the "minutes" state.

2.9.28.4 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 2.34 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 T10128 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.

2.9.28.4 EXECUTION OF TOOL LIFE CONTROL (CONT'D)

(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.

(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.

2.9.28.5 PARAMETERS AND SETTINGS FOR TOOL LIFE CONTROL

(1) Settings

#6004 (D6)

Clears life data registered prior to the G122; command

0: Does not clear 1: Clears

#6204

Specifies group number when resetting for tool change. However, this is only effective when #6020 D5 is "0." Setting 1 to 128

(2) Parameters

#6020 D7

Specifies group number for tool change skipping (TL SKP)

0: Currently specified group

1: External signal (TL1 to TL64)

#6020 D6

Selection of a T command group during an M06 command.

0: Latest T command

1: Immediately Prior T command

#6020 D5

Group number command for tool change reset (TLRST)

0: Setting #6204 1: External input (TL1 to TL64)

#6020 D4

T/M selection when tool life control is by count

0: Counts with T9999L $\Delta\Delta\Delta$

1: Counts with M02/M30

2.9.28.6 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 :

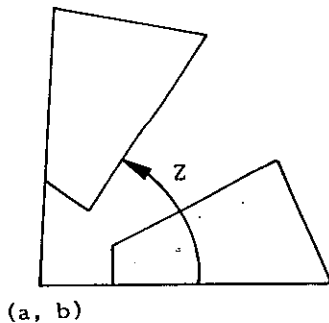
- (1) Do not rewrite the life control data when executing life control.
- (2) Even if M06 is executed in manual MDI, life control will not be executed. Do not execute commands T10001 through T10128 in manual MDI.
- (3) If the part program memory capacity is 150 m or 320 m, use 13 m as the tool life control data. This will then reduce the part program memory capacity by that amount.
- (4) If OVR and SKP occur at the same time, T-NO will display SKP on a priority basis.

2.9.29 ROTATION OF COORDINATES (G68, G69)

(1) $\left. \begin{matrix} G17 \\ G18 \\ G19 \end{matrix} \right\} G68 \quad a \dots b \dots R \dots;$

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.

G17...XY plane or X4 plane } (But the 4th axis is limited to straight line axis.)
 G18...ZX plane or Z4 plane }
 G19...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.....b.....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 activates 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.

2.9.30 SETTING OF LOCAL COORDINATE SYSTEM (G52)†

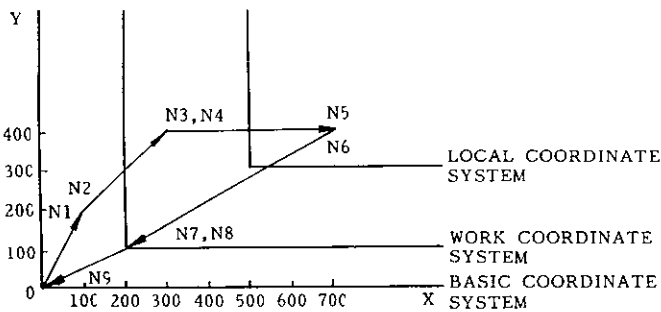
- (1) G52 Q2 X.....Y.....Z.....α.....β.....;
 (α and β respectively shows the 4th and 5th axes.)

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.

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 X0 Y0 Z0 (α0 β0);

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" is triggered if the command is made if the work coordinate system has not been set.

(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 cancelling the work coordinate system (G54 to G59) if the above option has not been added.

2.9.31 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.

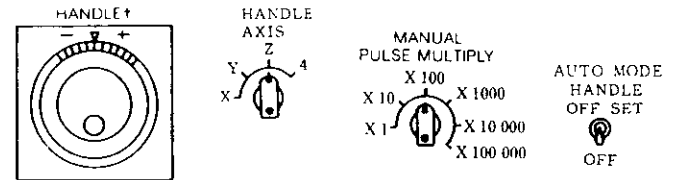
- (1) Operating procedures are as follows:

(a) Turn on the auto mode handle offset switch.
 (b) Select an axis to be moved by means of the handle 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		Effective/Invalid
	No.	Setting	
X-axis	#6010 (Change) (HOFSX)	1	Effective
		0	Invalid
Y-axis	#6010 (HOFSX)	1	Effective
		0	Invalid
Z-axis	#6010 (HOFSZ)	1	Effective
		0	Invalid
4th-axis	#6011 (HOFS4)	1	Effective
		0	Invalid
5th-axis	#6011 (HOFS5)	1	Effective
		0	Invalid

(e) The auto mode handle offset switch is #13207 (HOFS).

(f) For the function stated above, manual absolute function is invalid.

(g) For unmoved axis, handle offset is in valid.

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.

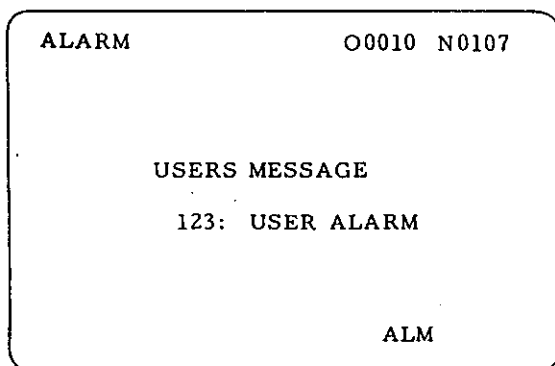


Fig. 2.78 Sample Display of Alarm Message by #8000 Command

Note: If a command with a wrong alarm message is specified, a format error will be caused.

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 [C2] key and [SHIFT] key, and character "(" may be entered.

b. Depress [C3] 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.

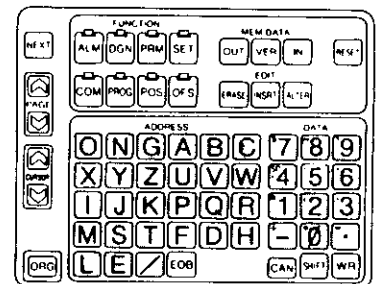
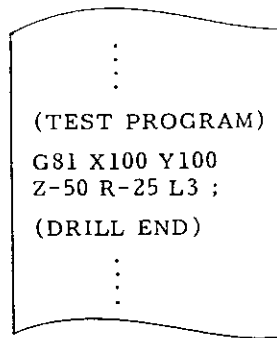


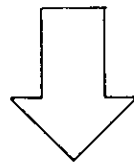
Fig. 2.79

Characters Enclosed in Control-Out and Control-In (Thick-Lettered Keys) for 9" CRT

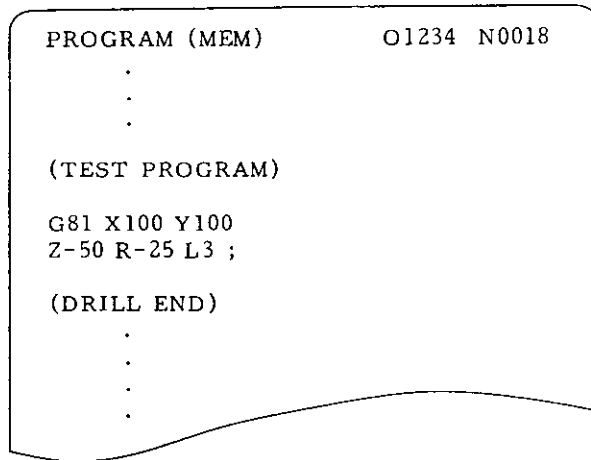
2.10.2 MESSAGE DISPLAY BY CONTROL-OUT AND CONTROL-IN (CONT'D)



Program Example



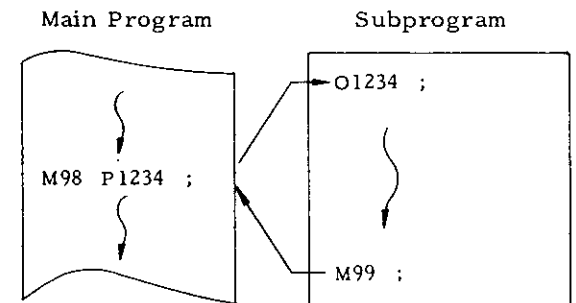
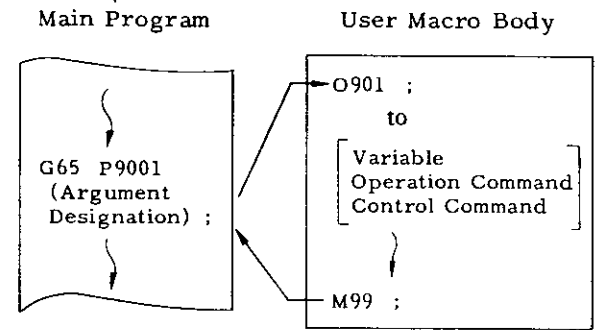
Message Display by Control-out and Control-in



2.11 USER MACRO (G65 AND G66)

Special programs written by the machine builder or user by the use of a group of instructions 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."

2.11.1 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 1.3 ARGUMENT DESIGNATION.

(2) Modal Call (G66 and 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 maker 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 O9010.

#6121 ... Sets G code which calls the macro of program number O9011.

to

#6129 ... Sets G code which calls the macro of program number O9019.

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 M00, M01, M02, M30, M90 through M99 may be set for the parameter.

#6130 ... Sets M code which calls the macro of program number O9001.

#6131 ... Sets M code which calls the macro of program number O9002.

#6132 ... Sets M code which calls the macro of program number O9003.

#6133 ... Sets M code which calls the macro of program number O9004.

For macro call by arbitrary M code macro, argument designation is available.
#6073 D4 1 : Argument designation applicable
0 : Argument designation not applicable

	M code
#6580	Program No. O9005 macro calling M code
#6581	Program No. O9006 macro calling M code
#6582	Program No. O9007 macro calling M code
#6583	Program No. O9008 macro calling M code
#6584	Program No. O9009 macro calling M code
#6585	Program No. O9020 macro calling M code
#6586	Program No. O9021 macro calling M code
#6587	Program No. O9022 macro calling M code
#6588	Program No. O9023 macro calling M code
#6589	Program No. O9024 macro calling M code
#6590	Program No. O9025 macro calling M code
#6591	Program No. O9026 macro calling M code
#6592	Program No. O9027 macro calling M code
#6593	Program No. O9028 macro calling M code
#6594	Program No. O9029 macro calling M code
#6595	Program No. O9030 macro calling M code
#6596	Program No. O9031 macro calling M code
#6597	Program No. O9032 macro calling M code
#6598	Program No. O9033 macro calling M code
#6599	Program No. O9034 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.

NOTE THAT THE MACRO CALL BY M CODE DOES NOT PERMIT ARGUMENT DESIGNATION.

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.

2.11.1 USER MACRO CALL COMMANDS (CONT'D)

(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 O9000 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

0 ... T code designation is handled as a T code.

1 ... T code designation is handled as a macro call command to call the macro of program number O9000.

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.

(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	
O61 ;	O62 ;
DO1 ;	DO1 ;
DO2 ;	M98P63 ;
M98P62 ; - 1	GOTO20 ;
GOTO10 ;	END1 ;
END2 ;	N20M99 ;
END1 ;	
N10M30 ;	

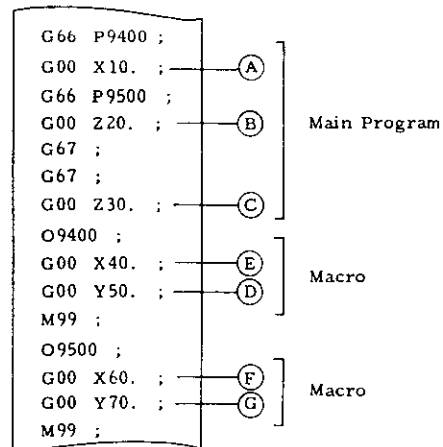
M98 or /G65 can be specified at the above "-1."

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)

2.11.2 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 2.35 Argument Designation I

Address of Argument Designation I	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 may be omitted.

Sample Program

```

#1 #2 #24 #26
↑ ↑ ↑ ↑
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 may 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 2.36.

Table 2.36 Argument Designation II

Address of Argument Designation II	Variables in User Macro Body
A	#1
B	#2
C	#3
I1	#4
J1	#5
K1	#6
I2	#7
J2	#8
K2	#9
I3	#10
J3	#13
K3	#12
I4	#13
J4	#14
K4	#15
I5	#16
J5	#17
K5	#18
I6	#19
J6	#20
K6	#21
I7	#22
J7	#23
K7	#24
I8	#25
J8	#26
K8	#27
I9	#28
J9	#29
K9	#30
I10	#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 may be omitted.

2.11.2 ARGUMENT DESIGNATION (CONT'D)

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 on the next page.

Table 2.37 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	0
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 #6020D0 = 1, D2 = 1 for addresses E and F, and parameter #6006D5 = 1 for the other addresses.

(4) Considerations in Argument Designation

A. Argument designation types I and II may be used concurrently. If the same variable has been duplicated, the last one is validated.

B. For both types I and II, addresses I, J, and K should be designated in this order. The other addresses may be designated in any order.

C. In the argument designation part, negative sign and decimal point may 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.

2.11.3 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) Operation Commands

- A. Arithmetical operations (+, -, *, /, ...)
- B. Functional operations (SIN, COS, ROUND, ...)

(3) Control Commands

- A. Branch command (IF <qualification>
GO TO n)
- B. Repeat command (WHILE <qualification>
DO m)

Using these commands, a program which requires complicated operations and conditional judgements may be written in the general format. Hence, 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.

2.11.4 VARIABLES

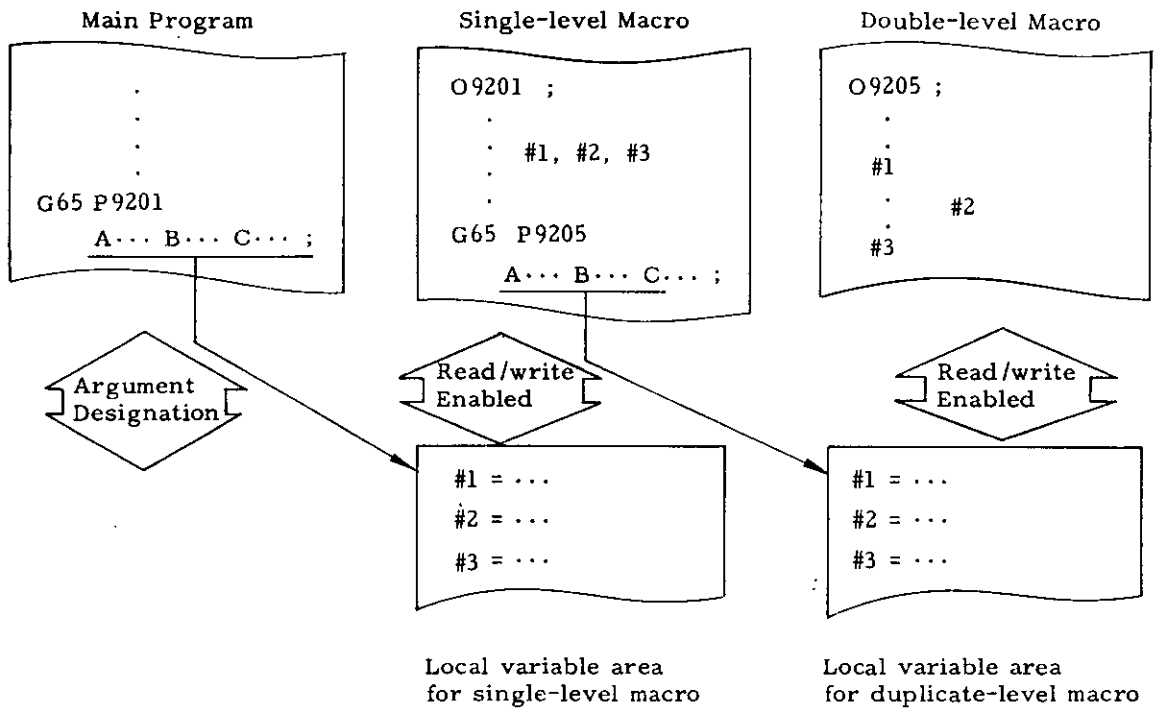
Instead of directly assigning a value to an address in a user macro body, the address may be designated by a variable. When this variable is called during execution, the corresponding value is searched from the variable area to provide the address value.

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.



Hence, 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

```

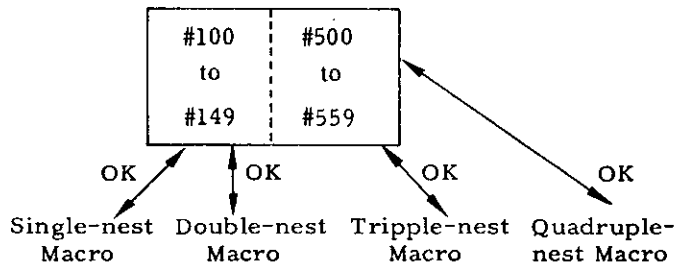
O1000;           O8000;
M98P8000;       # 1 = 1;
T # 1;          # 2 = 2;
                :
T # 2;          :
                :
                :
                M99;
M30;

```

(2) Common Variables (#100 through #149, #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 Variable Area

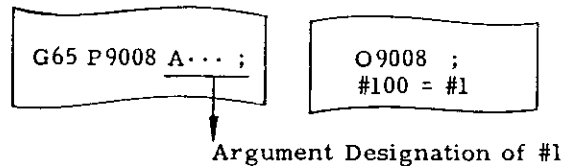


Common variables are divided into the following two types depending on clear conditions:

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 D1 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:



2.11.4 VARIABLES (CONT'D)

(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 shift 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.

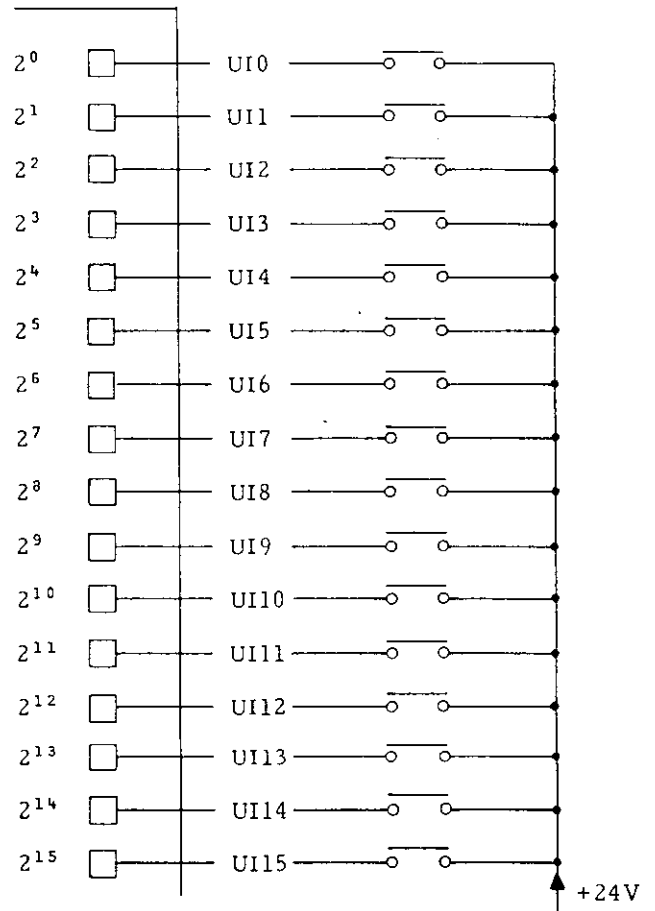
A. Interface Input Signals (#1000 Through #1015, #1032)†

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
UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0
2^7	2^6	2^5	2^4	2^3	2^2	2^1	2^0
#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008
UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8
2^{15}	2^{14}	2^{13}	2^{12}	2^{11}	2^{10}	2^9	2^8

Variable Value	Input Signal
1	Contact Closed
0	Contact Open

YASNAC



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] GO TO 100 ;
```

Bit 2^{15} (UI15) is read and, if it is "0," a branch is made to sequence number N100.

```
#130 = #1032 AND 255
```

Bits 2^0 through 2^7 (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.

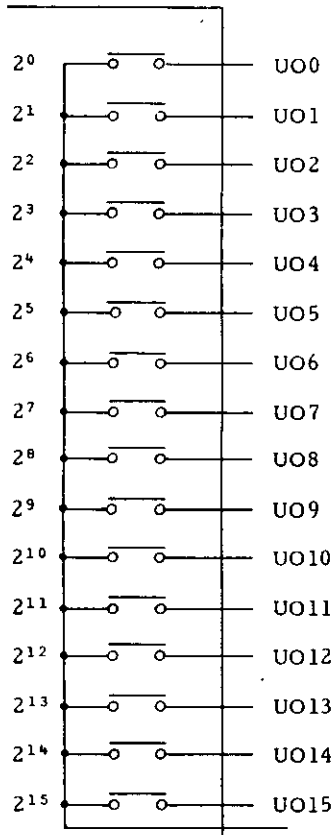
B. Interface Output Signals (#1100 Through #1115, #1132)[†]

a. When one of system variables #1100 through #1115 is specified to the left-hand of an operational expression, an on or 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	#1106	#1105	#1104	#1103	#1102	#1101	#1100
U07	U06	U05	U04	U03	U02	U01	U00
2 ⁷	2 ⁶	2 ⁵	2 ⁴	2 ³	2 ²	2 ¹	2 ⁰
#1115	#1114	#1113	#1112	#1111	#1110	#1109	#1108
U15	U14	U13	U12	U11	U10	U09	U08
2 ¹⁵	2 ¹⁴	2 ¹³	2 ¹²	2 ¹¹	2 ¹⁰	2 ⁹	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 #1132 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. Hence, 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 #1100 through #1115, the values are handled as follows:

"Blank" is assumed to be "0."
Values other than "blank" and 0 are assumed to be "1."

Sample Program

#1107 = #10 ; (#10 = 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 output signal of bits 2⁴ through 2⁷ (U04 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.

2.11.4 VARIABLES (CONT'D)

c. The relationships between the tool offset numbers and the system variables are as shown below:

Table 2.38 System Variables and Tool Offset No.

System Variable	Tool Offset No.
#2001	01
#2002	02
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
.	.
#2098	98
#2099	99
#2100	100
.	.
.	.
.	.
.	.
#2299	299

Table 2.39 System Variables and Work Coordinate System Shift Amount

System Variable	Work Coordinate System Shift Amount	Spindle
#2500	External work coordinate system correction amount	X
#2501	G54 (J1)	
.	.	
#2506	G59 (J1)	Y
#2600	External work coordinate system correction amount	
#2601	G54 (J1)	
.	.	Z
#2606	G59 (J1)	
#2700	External work coordinate system correction amount	
#2701	G54 (J1)	α
.	.	
#2706	G59 (J1)	
#2800	External work coordinate system correction amount	β
#2801	G54 (J1)	
.	.	
#2806	G59 (J1)	β
#2900	External work coordinate system correction amount	
#2901	G54(J1)	
.	.	β
#2906	G59(J1)	

Note: When "work coordinate system setting B" feature is provided, System variables listed in the table on page 102 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	
# 2611 ⋮	G54 J2 ⋮	Y
# 2616	G59 J2	
# 2711 ⋮	G54 J2 ⋮	Z
# 2716	G59 J2	
# 2521 ⋮	G54 J3 ⋮	X
# 2526	G59 J3	
# 2621 ⋮	G54 J3 ⋮	Y
# 2626	G59 J3	
# 2721 ⋮	G54 J3 ⋮	Z
# 2726	G59 J3	
# 2531 ⋮	G54 J4 ⋮	X
# 2536	G59 J4	
# 2631 ⋮	G54 J4 ⋮	Y
# 2636	G59 J4	
# 2731 ⋮	G54 J4 ⋮	Z
# 2736	G59 J4	
# 2541 ⋮	G54 J5 ⋮	X
# 2546	G59 J5	
# 2641 ⋮	G54 J5 ⋮	Y
# 2646	G59 J5	
# 2741 ⋮	G54 J5 ⋮	Z
# 2746	G59 J5	

D. Alarm Message Display (#3000)

When a condition to be alarmed occurs in a user macro program, system variable #3000 may be specified to put the machine in the alarm state.

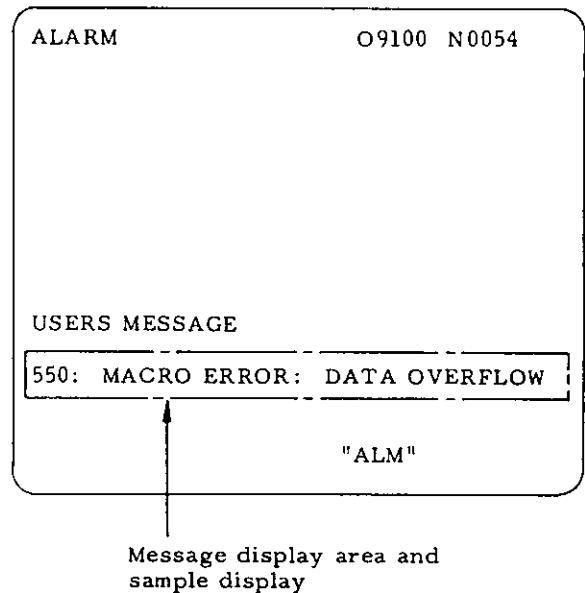
#3000 = n (<alarm message>);

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. Press ALM function key.

The alarm number and message are displayed on the bottom of CRT screen.



2.11.4 VARIABLES (CONT'D)

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 OVER-
          FLOW)
```

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 "0"	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 "0."
```

Restrictions

a. The accuracy of clock 1 is 8 ms. When 4294968000 msec has been reached, an overflow occurs, setting the clock to "0."

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
┌──────────┴──────────┐
G65 P9351 A5. ;          Argument designation
                        of variable #1
                        ... Macro to wait for
                        5 seconds
```

Macro Program

```
O9351 ;
#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 may 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	FIN 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)

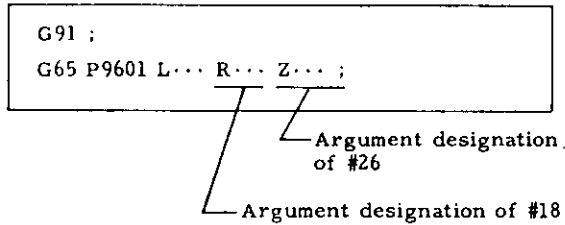
When the value listed in the following table is substituted in system variable #3004, feed hold, feedrate override, and exact stop can be made valid or invalid.

#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

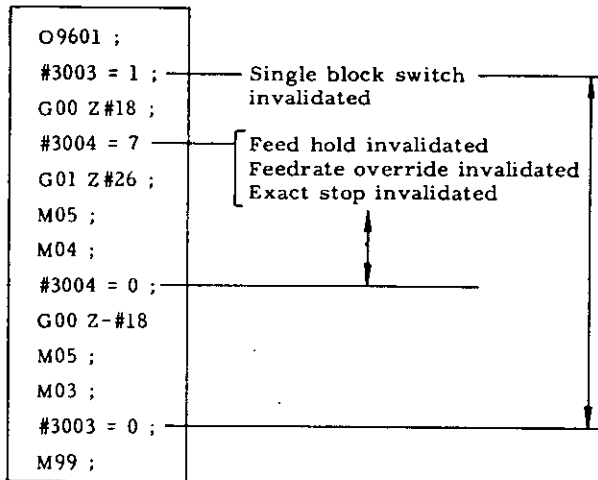
Sample Program

Tapping Cycle (for Incremental Command)

Main Program



Macro Program



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

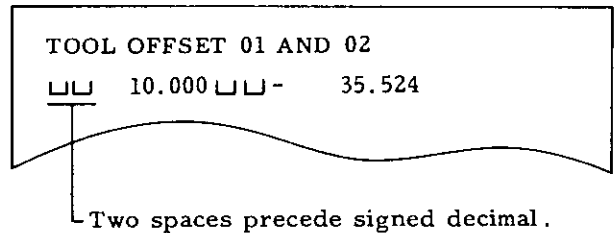
variable at the right-hand is output via RS232C interface as sign and decimal 9-digit data (4 after and 5 before decimal point). The value is rounded out to the fifth digit after the decimal point. When the value is of b digits or more before the decimal point, the asterisk is output.

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 RS232C 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 ; ... = -35.524 mm
#3100 = ( ) ;
```

Printout Data



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
#4001	G code (group 01)
to	to
#4021	G code (group 21)
#4102	B code
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Sequence number
#4115	Program number
#4119	S code
#4120	T code

2.11.4 VARIABLES (CONT'D)

#4001 through #4120 cannot be placed to the left-hand of the operation expression.

Sample Program

Main Program

```
G65 P9602 <Argument Designation>;
```

Macro Program

```
O9602 ;
#1 = #4001 ;
G00 X... Y... ;
G01 Z... F... ;
G03 X... Y... R... ;
G00 Z... ;
G#1 ;
M99 ;
```

G codes (G00 through G03) of 01 group are retained.

G codes of 01 group are restored.

J. Positional Information (#5001 Through #5105)

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)	
#5005	5th block end position (ABSIO)	

System Variable	Positional Information	Read During Move
#5021	X-axis current position (ABSMT)	Enabled
#5022	Y-axis current position (ABSMT)	
#5023	Z-axis current position (ABSMT)	
#5024	4th current position (ABSMT)	
#5025	5th 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)	
#5045	5th current position (ABSOT)	
#5061	X-axis skip signal position (ABSKP)	Enabled*
#5062	Y-axis skip signal position (ABSKP)	
#5063	Z-axis skip signal position (ABSKP)	
#5064	4th skip signal position (ABSKP)	
#5065	5th skip signal position (ABSKP)	
#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	
#5105	5th servo position deflection amount	

* Reading of #5021 to #5025, #5041 to #5045, and #5101 to #5105, when commanded during movement, will be performed after completion of the movement of the preceding block. Therefore radius compensation is not performed.

Abbreviation	ABSIO	ABSMT	ABSOT	ABSKP
Meaning	End position of block immediately before.	Command current position (same as POSITION MACHINE display).	Command current position (same as POSITION UNIVERSAL display).	Position at which skip signal did not go on in G31 block.
Coordinate system	Work coordinate System	Machine coordinate system	Work coordinate system	Work coordinate system
Tool Position, Tool length, Tool radius compensation Amount	Not included	-	Included	Included

Notes:

1. When the skip signal is not turned on in G31 block, the skip signal position is at the end of G31 block.

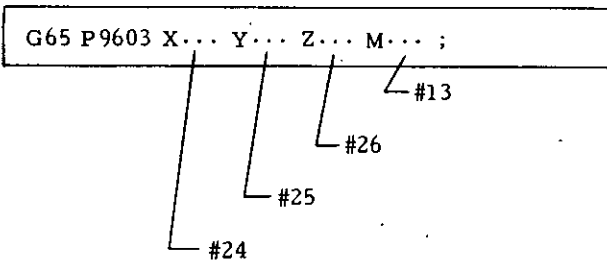
2. The "input unit x 10" feature is valid up to the macro call block (the argument designation part by G65 or G66) but is invalid in the user macro body.

3. System variables #5001 through #5105 may not be placed to the left-hand of operational expression.

Sample Program

The tool is positioned to the specified location (X, Y, Z) on machine coordinate system, performs the specified M feature, and returns to the start point.

Main Program



Macro Program

```
O9603 ;
#1 = #5001 ;
#2 = #5002 ;
#3 = #5003 ;
G91 ;
G00 X [#24-#5021] Y [#25-#5022];
G00 Z [#26-#5023];
M#13 ;
G00 Z#3 ;
G00 X#1 Y#2 ;
M99 ;
```

List of Variables

Variable No.	Meaning
#1 to #33	Local variables.
#100 to #149	Common variables (reset to "blank" at power-off).
#500 to #559	Common variables (retained at power-off).
#1000 to #1015	Interface input signals (each signal for each bit).
#1032	Interface input signal ($\sum_{i=0}^{15} \# [1000 + i * 2i]$).
#1100 to #1115	Interface output signals (each signal for each bit).
#1132	Interface output signal ($\sum_{i=0}^{15} \# [1100 + i * 2i]$).
#2001 to #2299	Tool offset amount.
#2500 to #2906	Work coordinate system shift amount, external work coordinate system correction amount.
#3000	Alarm message display.
#3001	Clock 1 (in units of 1 ms).
#3002	Clock 2 (in units of 1 s).
#3003	Single block stop, auxiliary function complete wait control.
#3004	Feed-hold, feedrate-override, and exact-stop control.
#3100	RS232C data output (print out feature).
#4001 to #4120	Current value of modal information command.

2.11.4 VARIABLES (CONT'D)

Variable No.	Meaning
#5001 to #5005	End position of immediately preceding block (for each axis).
#5021 to #5025	Current position of machine coordinate system (for each axis).
#5041 to #5045	Current position of POS. UNIVERSAL (for each axis).
#5061 to #5065	Position at which G31 skip signal is turned on (for each axis).
#5083	Valid tool length compensation amount.
#5101 to #5105	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 directly:

#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; G01X100.Y20.F1; G01X100.Y20.F01; G01X100.Y20.F#1; (F5.0 specification)	F1-digit command Direct command Direct command Direct command
#1 = 0.001 to 0.499 #1 = 1.0 to 9.0 Integers. (2.0, 3.0, 4.0...)	No lower digit error Lower 1-digit command
#1 = 0.50 to 9.499 (Digits below decimal point are rounded off)	Lower 1-digit command
#1 = 9.500 or more (F5.1 specification)	Direct command
#1 = 0.001 to 0.499 #1 = 0.1 to 0.9 (0.2, 0.3, 0.4...)	No lower digit error Lower 1-digit command
#1 = 0.050 to 0.949 (Digits of 0.01 or below are rounded off)	Lower 1-digit command
#1 = 0.950 or more	Direct command

Notes:

(1) Address /, O and N may not refer to variables.

Sample /#8, N#100 ... Error.

(2) A variable number may 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.

Designation and function of <blank> is classified in the following two versions A and B. The control is set for either version. Versions A and B cannot be changed.

Item	Meaning of (Blank)															
1 Concept of "#0"	<ul style="list-style-type: none"> • #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.	<ul style="list-style-type: none"> • 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	<ul style="list-style-type: none"> • Where #2 is <blank> command G00 x #2; is equivalent to G00 (Address is Ignored.) F figure 1, F figure 2; commands Numeral 1 and numeral 2 include indirect commands. <table style="margin-left: 20px;"> <tr> <td>#1</td> <td>#2</td> <td></td> </tr> <tr> <td>Blank</td> <td>Full</td> <td>→ Blank</td> </tr> <tr> <td>Blank</td> <td>Blank</td> <td>→ Numeral 2</td> </tr> <tr> <td>Full</td> <td>Blank</td> <td>→ Numeral 1</td> </tr> <tr> <td>Full</td> <td>Full</td> <td>→ Numeral 2...Note</td> </tr> </table>	#1	#2		Blank	Full	→ Blank	Blank	Blank	→ Numeral 2	Full	Blank	→ Numeral 1	Full	Full	→ Numeral 2...Note
#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.	<ul style="list-style-type: none"> • Where #2 is <blank> #3 is 0 <ul style="list-style-type: none"> ① Condition "IF #3 NE #2" is not established. ② Condition "IF #3 EQ #2" is established. 															
5 Others	<table style="margin-left: 20px;"> <tr> <td>#3=#[#0+#0]...#3 is blank.</td> <td rowspan="5" style="font-size: 2em; vertical-align: middle;">}</td> <td rowspan="5" style="vertical-align: middle;">In these commands, #3=0,</td> </tr> <tr> <td>#3=#2 * #0;</td> </tr> <tr> <td>#3=#0 + #0;</td> </tr> <tr> <td>#3=#0/#0;</td> </tr> <tr> <td>#3=5*#0;</td> </tr> </table> <ul style="list-style-type: none"> #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. <p style="margin-left: 20px;">Blanks in conditions except EQ and NE are treated as 0.</p>	#3=#[#0+#0]...#3 is blank.	}	In these commands, #3=0,	#3=#2 * #0;	#3=#0 + #0;	#3=#0/#0;	#3=5*#0;								
#3=#[#0+#0]...#3 is blank.	}	In these commands, #3=0,														
#3=#2 * #0;																
#3=#0 + #0;																
#3=#0/#0;																
#3=5*#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

2.11.5 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.

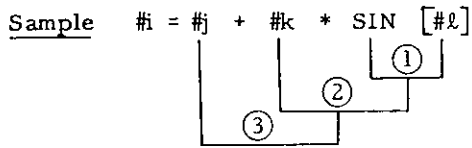
(4) Functions

#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.

2.11.5 OPERATION COMMANDS (CONT'D)

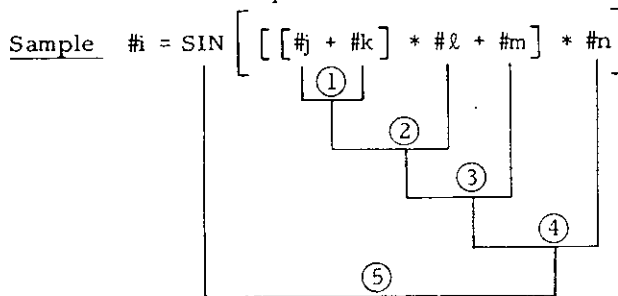
(5) Combinations of Operations

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 may 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

$G00 X [\text{ROUND} [\#10]] ;$
means

$G00 X12.376 ;$
because 8 of 12.3758 is rounded.

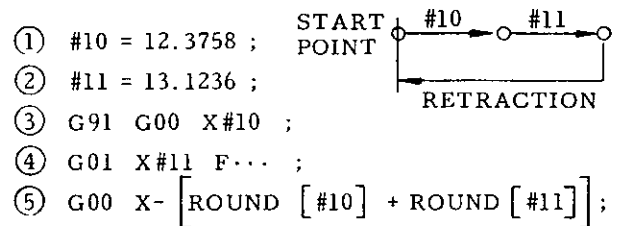
This command is also equivalent to

$G00 X\#10 ;$

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:



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 $G00 X- [\#10 + \#11] ;$

then, the movement is made by the following amount:

$$\begin{aligned} X- [\#10 + \#11] &= X - [12.3758 + 13.1236] \\ &= X - [25.4994] \\ &= X - [25.499] \end{aligned}$$

On the other hand, block movement of

③+④ is

$$\begin{aligned} X\#10 + X\#11 &= X12.376 + X13.124 \\ &= X25.500 \end{aligned}$$

Hence, 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.

2.11.6 CONTROL COMMANDS

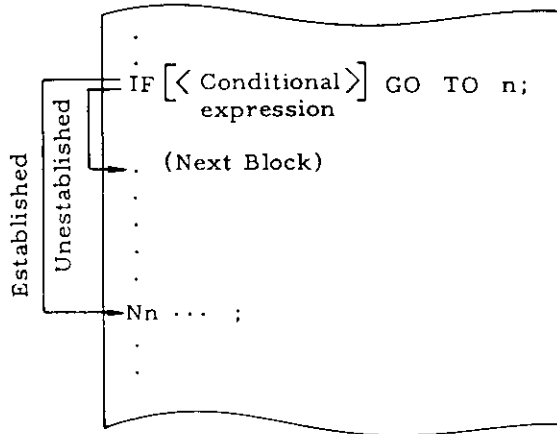
The commands which control the flow of micro-program are of the following two types:

- A. Branch Command ... IF [\langle conditional expression \rangle] GO TO n ;
- B. Repeat Command ... WHILE [\langle conditional expression \rangle] DO m ;

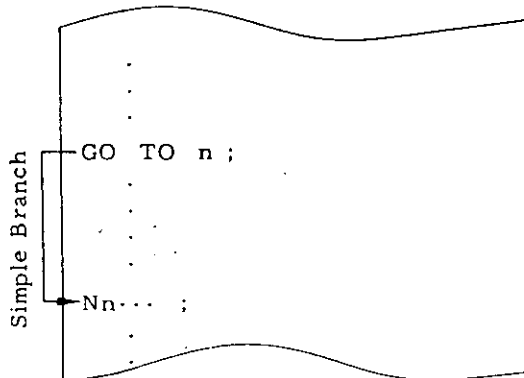
(1) Branch Command

A. IF [\langle conditional expression \rangle] GO TO n ;

If \langle conditional expression \rangle 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 may be changed. If the condition is not satisfied, the program proceeds to the next block.



IF [\langle conditional expression \rangle] may be omitted to provide a simple branch command as shown below:



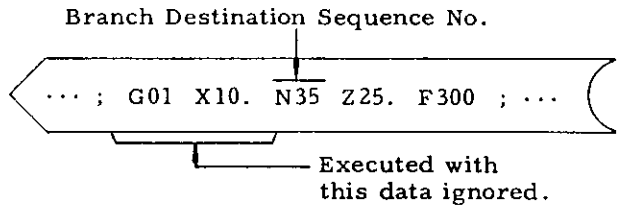
Conditional expressions are EQ, NE, GT, LT, GE, and LE. They are represented as follows:

Conditional Expression	Meaning
# EQ #	(# = #)
# NE #	(# \neq #)
# GT #	(# > #)
# LT #	(# < #)
# GE #	(# \geq #)
# LE #	(# \leq #)

A constant and \langle expression \rangle may be used to # and #j. A variable and \langle expression \rangle may be used for n.

Notes:

1. The sequence number of the destination of the branch by a branch command must be located at the head of that block. Otherwise, the data prior to the sequence number is ignored as shown below:



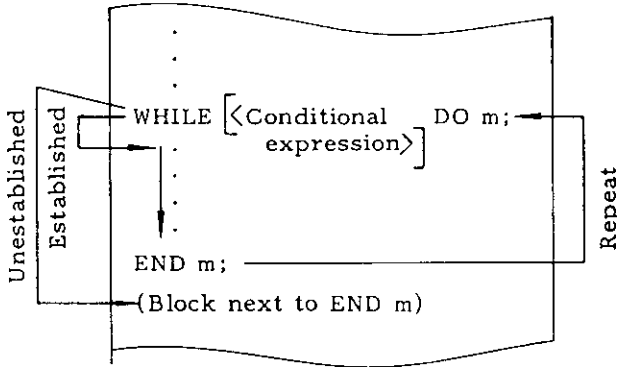
2. The reverse branch on the program takes longer execution time than the forward branch.

2.11.6 CONTROL COMMANDS (CONT'D)

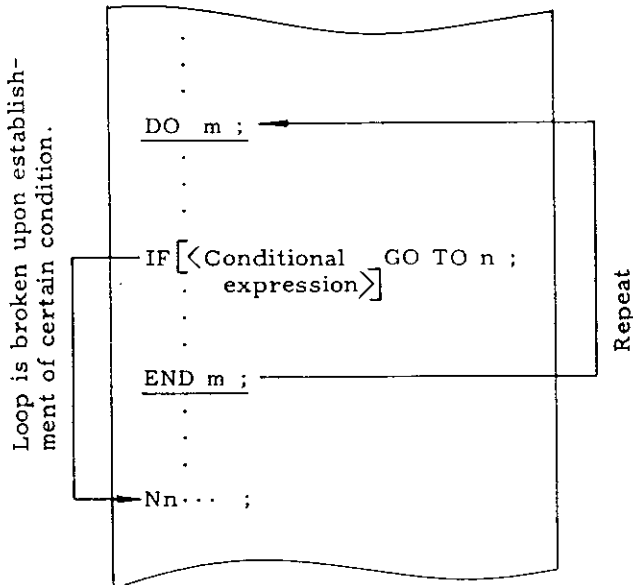
(2) Repeat Command

A. WHILE [\langle conditional expression \rangle] DO m;
 (m = 1, 2 and 3)
 .
 .
 .
 END m ;

While \langle conditional expression \rangle is satisfied, the blocks between DO m and END m are repeated. When it is unsatisfied, the processing branches to the block next to END m.

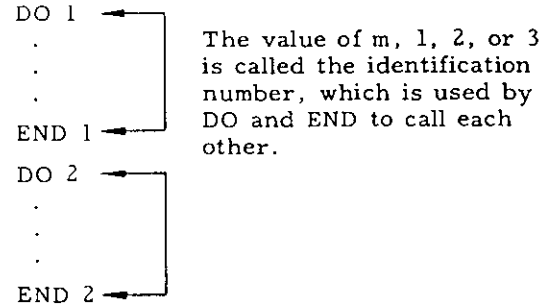


When the specification is made omitting WHILE [\langle conditional expression \rangle], the blocks between DO m and END m are repeated infinitely. Generally, this is used in the format shown below.

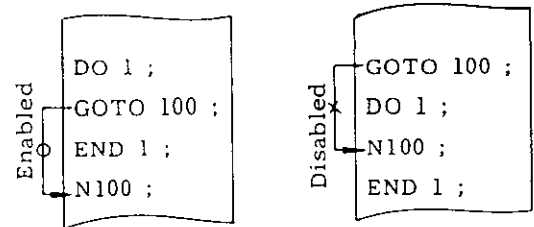


Notes:

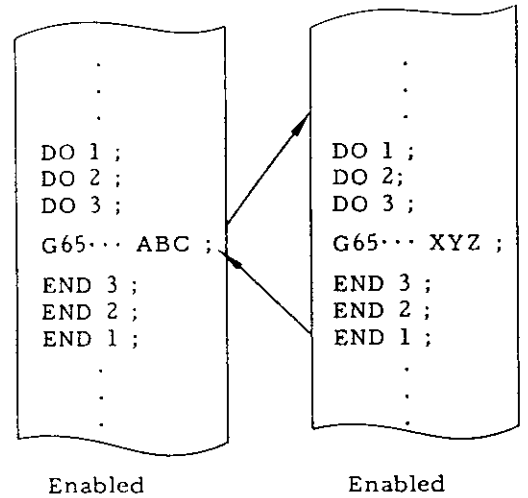
- DO m should be specified before END m.
- m of DO m and END m should have the same value. However, only 1, 2, or 3 may be specified in m.



- The same identification number may be used repeatedly except where repeat ranges overlap.
- GO TO n enables to get out of DO loop but it does not enable to get into DO loop as shown below:



- Triple DO-loop nesting is permitted for each micro program.



6. The codings shown below cause an error:

```
(i) DO 1 ;
      .
      .
      DO 1 ;
      .
      .
      END 1 ;
```

The return destination of END 1 is unknown.

```
(ii) DO 1 ;
      .
      .
      END 1 ;
      .
      .
      END 1 ;
```

One END 1 is missing.

```
(iii) DO 1 ;
      .
      .
      DO 2 ;
      .
      .
      END 1 ;
      .
      .
      END 2 ;
```

Overlap of DO ranges is not permitted. Rewrite as shown right.

```
DO 1 ;
      .
      .
      DO 2 ;
      .
      .
      END 2 ;
      .
      .
      END 1 ;
```

```
(iv) DO 1 ;
      .
      .
      DO 2 ;
      .
      .
      DO 3 ;
      .
      .
      DO 1 ;
      .
      .
      END 1 ;
      .
      .
      END 3 ;
      .
      .
      END 2 ;
      .
      .
      END 1 ;
```

Quadruple nesting. Max. nesting permitted is triple.

```
(v) DO 1 ;
      .
      .
      .
      N7000... ;
      .
      .
      .
      END 1 ;
      .
      .
      IF... GO TO 7000 ;
```

DO loop may not be entered from outside.

2.11.7 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 may 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 may 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.

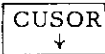

2.11.8 DISPLAY AND WRITE OF LOCAL VARIABLES AND COMMON VARIABLES

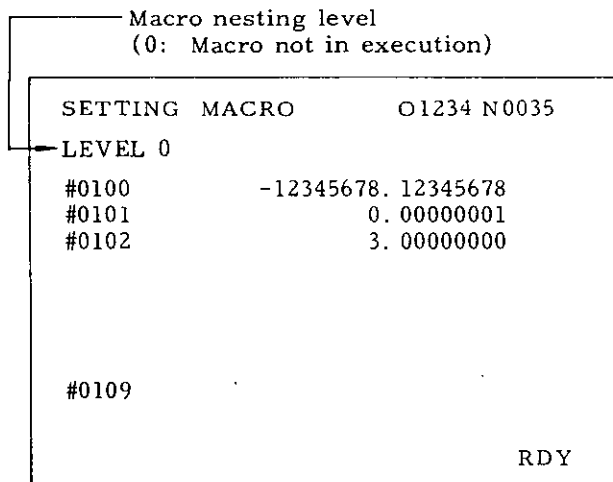
Local variables (#1 through #33) and common variables (#100 through #149, #500 through #559) can be displayed and written by the following operations:

(1) Display Operations



A. Display of Variables

a. Press SET function key. Mode select position may be provided anywhere.

b. Key-in the variable number and press  key or  key. 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. Press  key or  key, and the display may be scrolled up or down.

B. Remarks

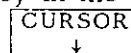
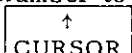
a. Common variables may always be displayed for review.

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."

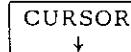
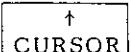


(2) Write Operations

A. Write of Values to Variables

a. Press SET function key. Mode select position may be provided anywhere.

b. Key-in the variable number to be written. Press  key or  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. Press  key or  key or  key or  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 may 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.

2.11.9 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 ... $\pm 10^{-308}$

C. Constant Values Usable in <Expression>

\pm (8 digits above decimal point). (7 digits below decimal points).

Sample Maximum value ± 99999999.9999999

Minimum value ± 0.0000001

D. Operational Accuracy

Decimal 15 digits significant.

E. Macro Call Maximum Nesting Level

Quadruple (four-fold).

F. Maximum Nesting Level of Repeat Command

Triple (three-fold) for each macro.

G. Repeat Command (DO) Identifier m

m = 1, 2, and 3.

H. Maximum Nesting Level of Brackets

Quintuple (five-fold).

(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 G65 and G66 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 MDI 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 may 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 #6004D1 = 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.2.11.4, F on page 103) and setting #6004 D1 mentioned above operate as shown below:

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.

2.11.9 CONSIDERATIONS AND REMARKS FOR USER MACROS (CONT'D)

(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				o	o				o	o			o			
*	(o	o	o	o				o	o	o			
*)	o	o	o	o	o	o	o	o	o	o	o	o	o	o	o	o
+	Add	o	o	o	o	o					o	o	o	o	o	o	o
-	Subtract	o			o						o	o	o	o	o	o	o
:	For comment	o			o	o	o			o	o	o	o	o	o	o	o
/	Divide		o	o	o	o	o	o		o	o	o	o	o	o	o	o
#	Variable	Parameter designation									o		o		o	o	
*	*	o			o	o	o			o	o	o	o	o	o	o	o
*	=	o			o	o	o			o	o	o	o	o	o	o	o
*	[o	o	o	o	o	o	o	o	o	o	o	o	o	o	o	o
*]	o	o	o	o	o	o	o	o	o	o	o	o	o	o	o	o
\$	For comment	o			o	o	o				o	o	o	o	o	o	o
@					o	o	o	o	o					o			
?		o			o	o	o	o	o		o	o	o	o	o	o	o
.	Decimal point	o	o	o	o	o	o	o	o	o	o	o	o	o	o	o	o

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 ... [
 - #6111 ...]
 - #6112 ... *
 - #6113 ... =
 - #6114 ... (
 - #6115 ...)
- } Hole pattern setting parameter

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	3	2	1
o			o	o	o		

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 ... (,), *, =.

2.11.10 ALARM NUMBER OF USER MACROS

Shown below are the user-macro-associated alarms and their causes.

Code	Cause
105	MACRO ERROR (CONSTANT) The number of constants is in excess of the specified range.
106	MACRO ERROR There are too many G67 cancel codes.
107	MACRO ERROR (FORMAT) A format other than expression has an error.
108	MACRO ERROR (UNDEFIN #NO) The value not defined as a variable number is designated.
109	MACRO ERROR (#NO NOT LEFT) The variable of assignment statement is the one that is disabled for assignment.
110	MACRO ERROR ([] 5 LIMIT) The bracket nesting level is in excess of the upper limit (5).
111	MACRO ERROR (MOVE G66 - M99) A move command is specified in the macro end command M99 called by G66.
112	MACRO ERROR (5) The macro call nesting level is in excess of the upper limit (4).
113	---

Code	Cause
114	MACRO ERROR (DO FORMAT) DO and END are not paired.
115	MACRO ERROR ([] UNMATCH) The format of <expression> has an error.
116	MACRO ERROR (DO - END NO.) DO m is not in the range of $1 \leq m \leq 3$.
117	---
118	MACRO ERROR (GO TO N) GO TO n is not in the range of $0 \leq n \leq 9999$.

2.11.11 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.

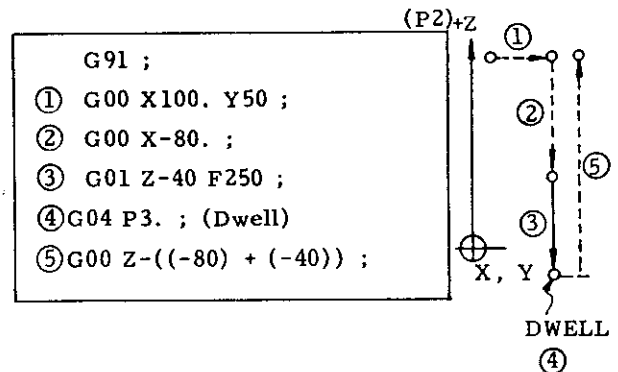
S (Student): Where shall we start?

T: An example of usual G82 command takes the following format:

(P1)

```
G91 ; (... Incremental Designation)
G82 X100. Y50. R-80. Z-40. P3.0 F250 ;
```

This command is divided into the following and executed within the NC unit:



2.11.11 EXERCISES OF USER MACRO (CONT'D)

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?

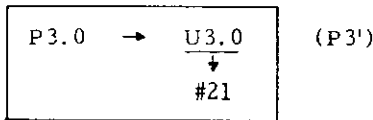
T: Type 1 for small number of variables. This type allows the use of X, Y and Z and therefore makes the argument designation easier to understand.

S: OK. When type 1 is used, we have the following variables.

X100.	Y50.	R-80.	Z-40.	P3.0	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.

```
(P4)
G91 ;
① G00 X#24 Y#25 ;
② G00 Z#18 ;
③ G01 Z#26 F#9 ;
④ G04 P#21 ;
⑤ G00 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 G00 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:

```
(P6)
G91 ;
G65 P9082 X100. Y50. R-80. Z-40.
U3.0 F250. ;
```

The user macro body is as follows:

```
(P7)
O9082 ;
G00 X#24 Y#25 ;
G00 Z#18 ;
G01 Z#26 F#9 ;
G04 P#21 ;
G00 Z- [ ROUND [#18] + ROUND [#26] ] ;
M99 ;
```

Program number "O9082" of the user macro is arbitrary.

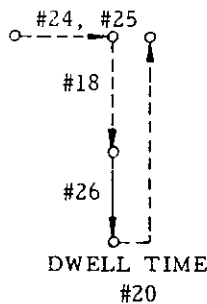
T: That looks OK.

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)

```
O9000 ;
#100 = #18 ;
#101 = #26 ;
#102 = #21 ;
#103 = #9 ;
M99 ;
```

(P9)

```
O9082 ;
G00 X#24 Y#25 ;
G00 Z#100 ;
G01 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. U3.0 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 "10." 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 G00 through G03.

S: It looks difficult ...

What would you say to the following program?

T: Well done!

(P11)

Macro Call Program

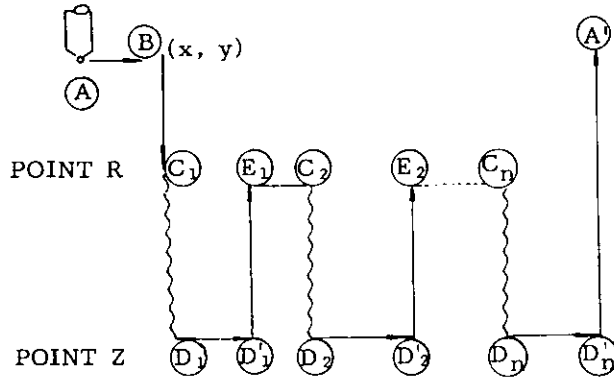
```
G91 G99;
G65 P9000 R-80. Z-40. U3.0 F250;
G65 P9082 X100. Y50.;
G65 P9082 X...Y...;
.
.
.
G98;
G65 P9082 X...Y...;
```

User Macro Body

(P12)

```
O9000;
#100 = #18;
#101 = #26;
#102 = #21;
#103 = #9;
#104 = 0
-----
O9082;
#104 = #104 + 1
#1 = #4001; ... GO to G3
#2 = #4010; ... G98/G99
G00 X#24 Y#25;
IF[#104 NE1] GO TO 1;
IF[#2 EQ99] GO TO 2;
G00 Z#100;
N1 G01 Z#101 F#103;
G04 P#102;
IF[#2 EQ 98] GO TO 2;
G00 7-[#101]
GO TO 3;
N2 G00 Z-[ROUND [#100] + ROUND [#101]];
N3 G#1; ... Restore G Code
M99;
```

2.11.11 EXERCISES OF USER MACRO (CONT'D)



```
O9000 (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([G CODE READING SET]) --> PosC[POSITIONING TO B, C, ... Cn]
      PosC --> Dec104{#104 ≠ 1}
      Dec104 -- Yes --> PosC
      Dec104 -- No --> PosZ[POSITIONING TO POINT C]
      PosZ --> Cut[CUTTING, DWELL AT POINT C → D]
      Cut --> G82{G98/G99}
      G82 -- G98 --> PosA[POSITIONING TO POINT A']
      G82 -- G99 --> PosE[POSITIONING TO POINT E]
      PosA --> RestoreG[RESTORE OF G CODE]
      PosE --> RestoreG
      RestoreG --> Jump[M99]
  
```

G CODE READING SET
 #104 = #104 + 1
 #1 = #4001 G0 - G3
 #2 = #4010 G98/G99

POSITIONING TO B, C, ... C_n
 G00 X #24 Y #25

POSITIONING TO POINT C
 G00 Z #100

CUTTING, DWELL AT POINT C → D
 G01 Z #101 F #103
 G04 P #102

POSITIONING TO POINT A'
 G00 Z --- [ROUND (#100) + ROUND (#101)]

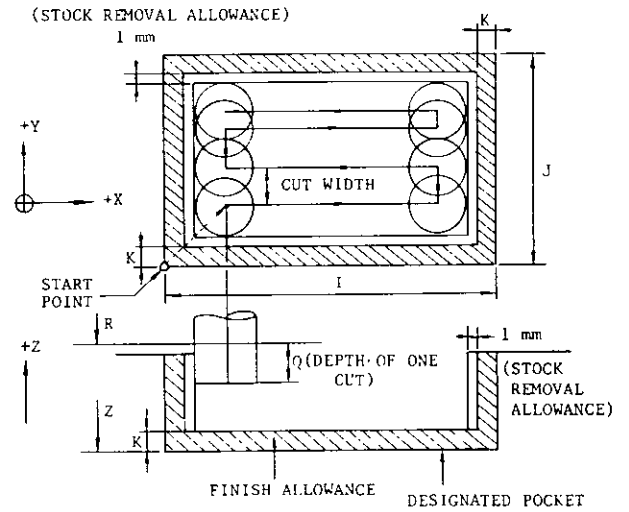
POSITIONING TO POINT E
 G00 Z --- (#101)

RESTORE OF G CODE
 G #1

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:

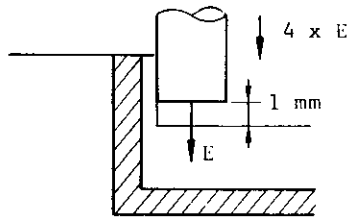


B. Macro Call Command

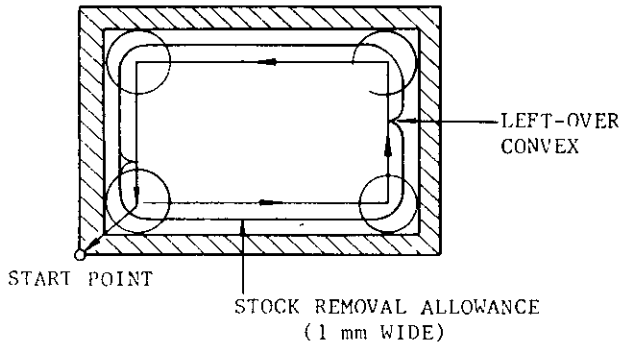
```
G65 P9061 X... Y... Z... R... I... J... K...
      T... Q... D... F... E... ;
```

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-axis 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

```
O9061 ;
#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
G00 X#27 Y#28 ;
G00 Z#18 ;
#32 = #18 ; ... #32: Cut bottom in execution
DO 1 ;
#32 = #32 - #17 ;
IF [#32 GT#29] GO TO 1 ;
#32 = #29 ;
```

```
N1 G01 Z#32 F#8 ;
G01 X#30 F#9 ;
#33 = 1 ;
WHILE [#33 LE#14] DO 2 ;
IF [#33 EQ#14] GO TO 2 ;
G01 Y [#28 + #33 * #13] F#9 ;
GO TO 3 ;
N2 G01 Y [#25 + #5 - #11] ;
N3 IF [#33 AND 1 EQ 0] GOTO 4 ;
G01 X#27 ;
GO TO 5 ;
N4 G01 X#30 ;
N5 #33 = #33 + 1 ;
END 2 ;
G00 Z#18 ; ;
IF [#32 LE#29] GO TO 6 ;
G00 X#27 Y#28 ;
G01 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
G00 X#27 Y#28 ;
G01 Z#32 F#8 ;
G01 X#30 F#9 ;
      Y#31 ;
      X#27 ;
      Y#28 ;
G00 Z#18 ;
G00 X#24 Y#25 ; ... Return to start point
G#15 ; ... Restore of G90/G91
M99 ;
```

α cut
loop

Stock
removal
cycle

2.12 SOLID TAP FUNCTION †

Solid tap functions can be optionally added to YASNAC MX3.

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 MX3.

- (1) S5-digit analog output is provided as an option for YASNAC MX3.
- (2) Yaskawa VS-626MTIII 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 application/resetting.

2.12.2.2 SOLID TAP CYCLE COMMAND

The solid tap can be made by specifying as below after G93 specification.

G84/74 X__Y__Z__R__ (P__) F__ (S__) (L__);

Tap Reverse tap Tapping position Z point R point Dwell Pitch Spindle speed Number repeats

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) 1=0.0001 mm/rev.
Range 0.0001 to 200.0000 mm/rev.
Input (in.) 1=0.000001 inch/rev.
Range 0.000001 to 7.874015 inch/rev

Programmable Range of S

1=1 rev.
Range 1 to 4500 rev.

NOTE: F×S cannot specify a program exceeding 24000.0 mm/min., 944.8 inch/min.

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.

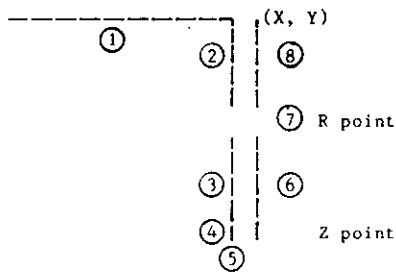
The solid tap mode is also selected by another way, namely, by spindle indexing and positioning to the home position, after spindle stop by parameter (#6055 D1). However, the spindle can be indexed only at the time when feedback pulse for one spindle revolution is 4096 pulses (#6061 * #6449=4096).

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. R1. 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 G00 or G01 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

G00, 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 cancelling 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 point 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 0. 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.

Accordingly, to actuate G93 with Dry Run ON for program check, keep Dry Run SW ON from the first and do not reset it before completion.

2.12.2.5 PRECAUTIONS (Cont'd)

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.

Accordingly, for check of 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.

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-axis 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 × 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.

NOTE

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

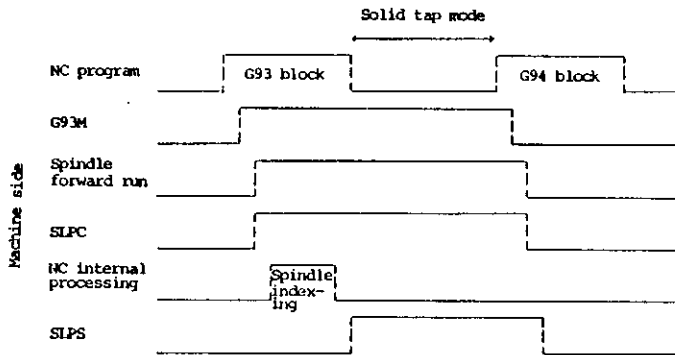
The signal is output at the time when NC unit forms position control loop to the spindle for solid tap. G93 block ends at the time this signal is output.

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 I/O

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.

Moreover, 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 phase A and phase 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 0: 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 0: Spindle not indexing
1: Spindle indexing

(3) G00 Error Detect in Solid Tap

- #6065 D0 0: Error Detect ON
1: Error Detect OFF

*(4) Display of Simultaneous Error Peak in Solid Tap

- #6065 D1 0: 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 0: 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.

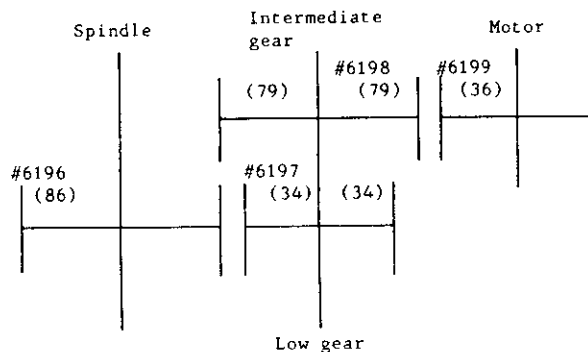
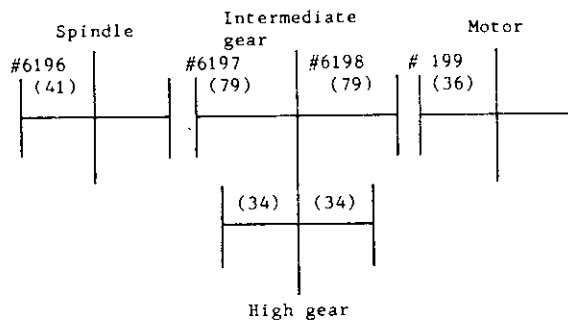
2.12.5 SOLID TAP-RELATED PARAMETER
(CONT'D)

* (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 example).
#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).



* (7) Spindle Feedback Pulse Multiplication
#6061

D3	D2	D1	D0	Multiplication
0	0	0	1	X1
0	0	1	0	X2
0	1	0	0	X4
1	0	0	0	X8

Standard setting: X4

* (8) Number of Feedback Pulses per Revolution of Spindle PG

- #6449 Range 1 to 32767
Unit: Pulse/rev.
Standard setting: 1024

NOTE

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 rpm
Standard setting: 6000

(10) Maximum Spindle Speed at Solid Tap

- #6471 Range 1 to 32767
Unit 1 = 1 rpm
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 = 0.01S-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 MX3 (DE8410207) 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 MX3 (DE8410207) for adjustment.

(17) Spindle/Z-axis Simultaneous Compensation Parameter (K2)

#6469 Range 0 to 32767
Standard: 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 MX3 (DE8410207) 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 rpm
Standard setting: 30

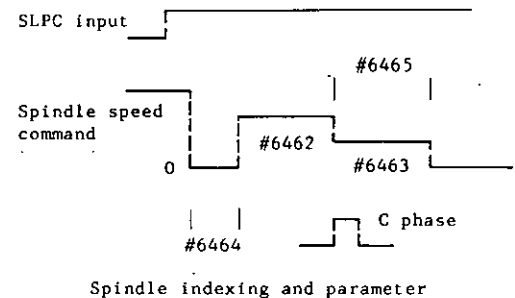
It sets the revolving speed to confirm spindle stop at the time of forming position loop of spindle with G93. Position loop of spindle is formed after G93 is specified and the spindle speed decreases to a value lower than this parameter. Set the value lower than the parameter to confirm zero speed at 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 (22) cover the parameter about spindle indexing at the time of specifying G93.



Note:

In case of setting original position 0, it stops on C phase after one turn 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.

G106 G01 X... Y... I... (or J...)

F...;

G106 : Non-modal G code

I : Cutting margin (X-axis direction)

J : Cutting margin (Y-axis direction)

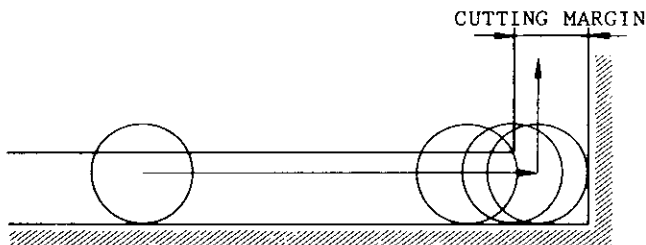


Fig. 2.80

(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

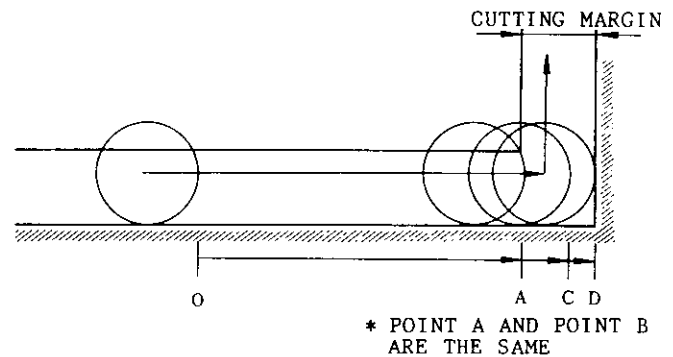


Fig. 2.81

(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

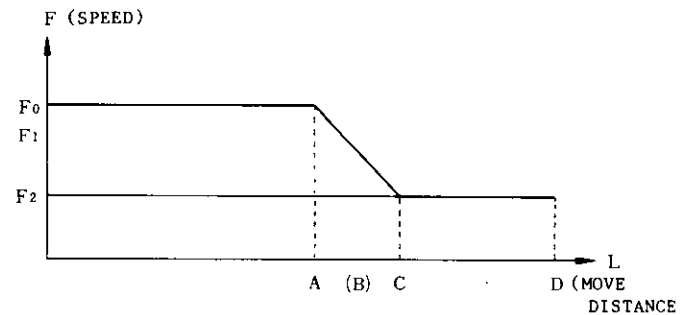


Fig. 2.82

$F_0 = (\text{Commanded feedrate}) * (\text{Override } \%)$

$F_1 = F_0$

$F_2 = F_0 * (\#6217) \dots (\text{given as } n\% \text{ of } F_0)$

Note: The operator no longer needs 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

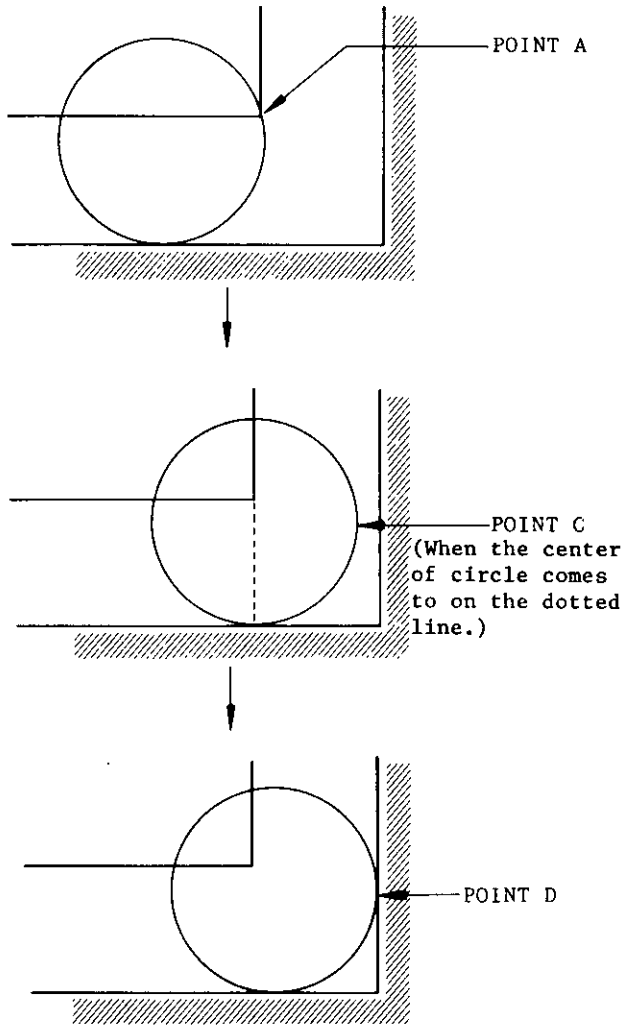


Fig. 2.83

(b) When the tool radius is larger than or equal to the cutting margin

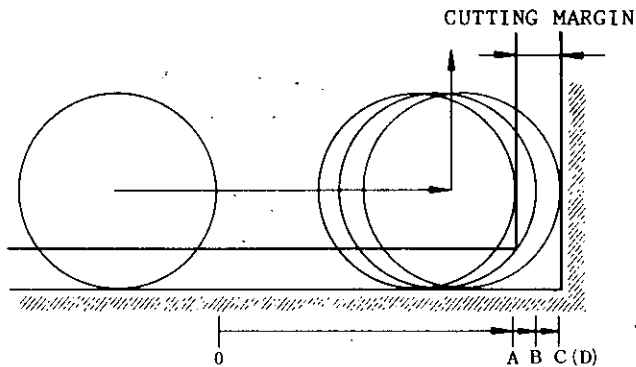


Fig. 2.84

- (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

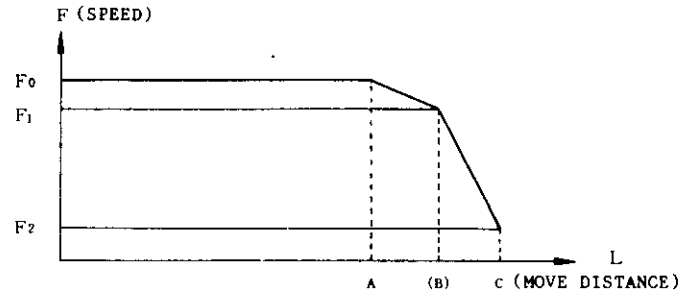


Fig. 2.85

$$F_0 = (\text{Commanded feedrate}) * (\text{Override } \%)$$

$$F_1 = F_0 * (\#6216) \dots (\text{given as } n\% \text{ of } F_0)$$

$$F_2 = F_0 * (\#6217) \dots (\text{given as } n\% \text{ of } F_0)$$

(iv) Speed change positions

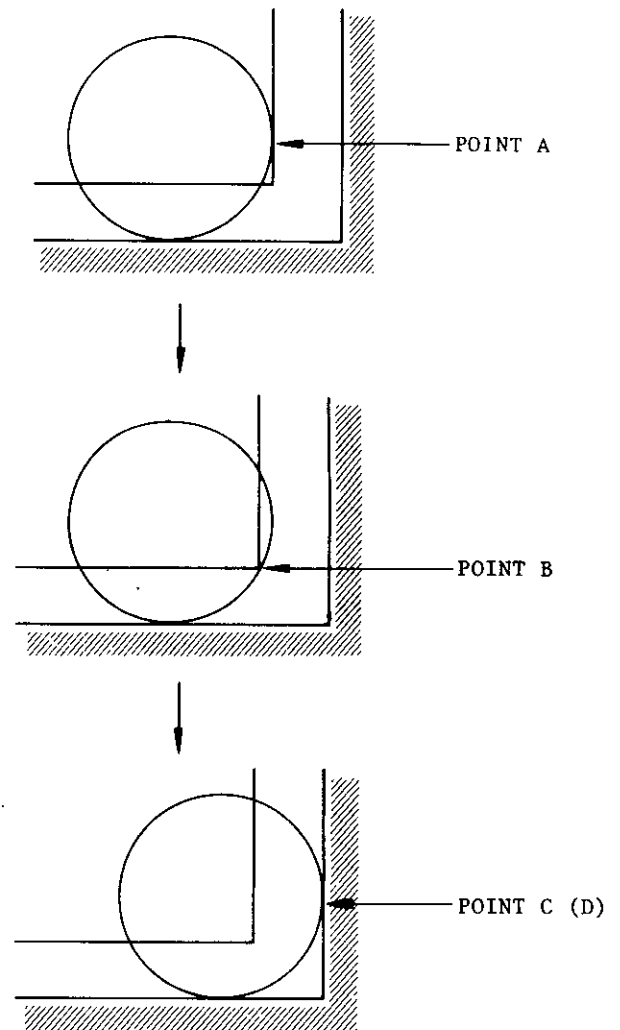


Fig. 2.86

2.13 AUTOMATIC CORNER OVERRIDE (CONT'D)

(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 $F0 \leq F1 \leq F2$, the speed is clamped at the value immediately before (F1 against F2). For example, if F1 is set at 80% and F2 at 90% by mistake, calculation is made with F2 as 80%. There will thus be no automatic corner override between A and B, and the speed is clamped at F1.

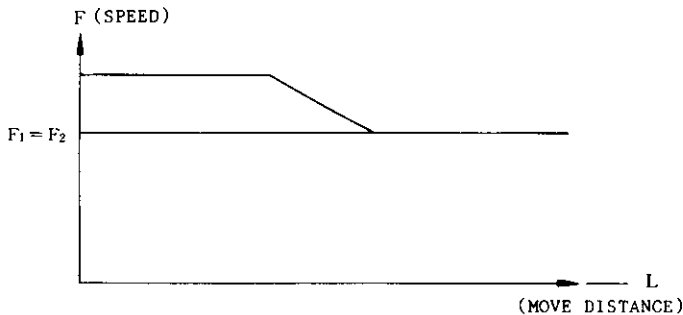


Fig. 2.87

(e) Set #6216, #6217 between 1 and 100. (Do not set "0.")

(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 G106 block.

(h) No override is performed by the G106 command, if the tool radius is 1/2 or less the cutting margin.

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/decel, or by the influence of the servo follow-up delay.

So the linear interpolation with linear acceleration (G198) and linear interpolation with linear deceleration (G199) are developed and added to the preparatory functions. This limits the acceleration in case the speed variation is too radical since it may shock the machine.

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 G199 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. The 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)

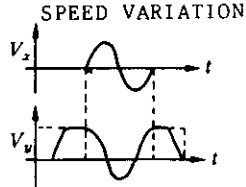
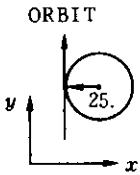
Command G199 under the G01 mode in the form compensation mode, to perform linear deceleration and position at the commanded point. The form compensation mode turns off after positioning. The move command must be large enough for deceleration.

(b) Program example

(i) How to command linear accel/decel in a block with discontinuous command speed

EXAMPLE (CIRCULAR CUT WITH FORM COMP.) ;

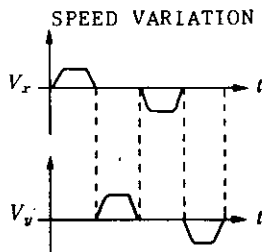
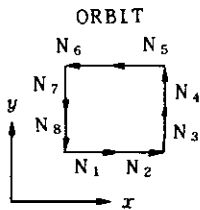
```
G01 G91 F10000;
N1 Y25. G198;
N2 G02 I25;
N3 G01 Y25. G199;
M30;
```



(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 (SQUARE CUT WITH FORM COMP.) ;

```
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 G199.
- (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
 $100/0.1 = 100 [(m/min)/s]$
 $100/(60/64) = 106.7$

Set 107 as the maximum acceleration parameter.

(4) How to set the form compensation parameter

Set the servo follow-up coefficient Kx, Ky ... K5 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 (ms)
 Servo follow-up coefficient: kx

$$kx = \left(\frac{1}{1 - \exp\left(-\frac{8}{Te}\right)} - 1 \right) * 8192$$

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)
Servo follow-up coefficient kx (X-axis)	-#6580 (0 to 65535)
Servo follow-up coefficient ky (Y-axis)	-#6581 (0 to 65535)
Servo follow-up coefficient kz (Z-axis)	-#6582 (0 to 65535)
Servo follow-up coefficient k4 (4th axis)	-#6583 (0 to 65535)
Servo follow-up coefficient k5 (5th axis)	-#6584 (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

Y axis: #6057 D7 D6

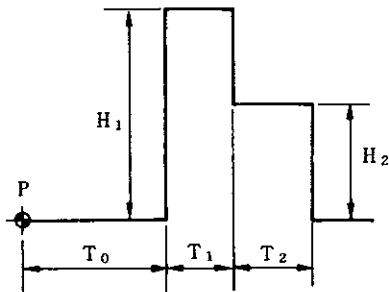
Z axis: #6058 D7 D6

0 0: Circular projection compensation off

1 0: Circular projection compensation on

Always turn off the NC power one time 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

T0 : Wait time from the start point to the first offset

T1 : Width of the 1st offset

T2 : Width of the 2nd offset

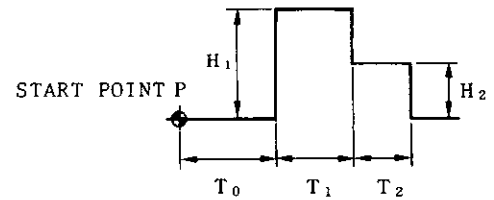
(Symbol)	(Param No.)	(Description)	(Setting range)	(Unit)
H ₁ X	#6077	(X-axis 1st height)	0 to 255	1 = 500 [pps]
H ₁ Y	#6078	(Y-axis 1st height)	0 to 255	1 = 500 [pps]
H ₁ Z	#6079	(Z-axis 1st height)	0 to 255	1 = 500 [pps]
H ₂ X	#6080	(X-axis 2nd height)	0 to 255	1 = 500 [pps]
H ₂ Y	#6081	(Y-axis 2nd height)	0 to 255	1 = 500 [pps]
H ₂ Z	#6082	(Z-axis 2nd height)	0 to 255	1 = 500 [pps]
T ₀ X	#6083	(X-axis comp. start wait time)	0 to 255	1 = 2 [ms]
T ₀ Y	#6084	(Y-axis comp. start wait time)	0 to 255	1 = 2 [ms]
T ₀ Z	#6085	(Z-axis comp. start wait time)	0 to 255	1 = 2 [ms]
T ₁ X	#6086	(X-axis 1st width)	0 to 255	1 = 2 [ms]
T ₁ Y	#6087	(Y-axis 1st width)	0 to 255	1 = 2 [ms]
T ₁ Z	#6088	(Z-axis 1st width)	0 to 255	1 = 2 [ms]
T ₂ X	#6089	(X-axis 2nd width)	0 to 255	1 = 2 [ms]
T ₂ Y	#6090	(Y-axis 2nd width)	0 to 255	1 = 2 [ms]
T ₂ Z	#6091	(Z-axis 2nd width)	0 to 255	1 = 2 [ms]

Notes:

- Any change made in parameters #6056 to #6057 becomes effective only after turning the power OFF and then ON again. D6 of #6056 to #6058 must always be set 0.
- Any change made in parameters #6077 to #6091 becomes effective by NC RE.SET.

(3) Adjustment

The following are standard setting and adjustment procedures.



EXAMPLE

G92 X0 Y0 Z0

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 3.1.1 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 will be read automatically in that code.

For punching tapes, the code must be selected by the setting of #6000D7.

When "0" is set with #6000D7 ... EIA code

When "1" is set with #6000D7 ... ISO code

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.

The diagram shows a process sheet layout with two main sections: LABEL and PROGRAM NO. Below these are fields for EOR and EOB. A grid of characters is shown below, with columns labeled N, G, X, Y, Z, I, J, K, and F. The grid contains characters G, X, Y, Z, I, J, K, R, Q, L, P, Q, α, α, T, H/D, S, M.

LABEL				PROGRAM NO.				
[] EOR				EOB []				
N	G	X	Y	Z	I	J	K	F
					R	Q	L	
		P	Q	α	α	T	H/D	S
								M

Fig. 3.1 Example of Process Sheet

3.2.1 PROCESS SHEET (CONT'D)

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											
								1											
								2											
								3											
								4											
								5											
								6											
								7											
								8											
								9											
								a	A										
								b	B										
								c	C										
								d	D										
								e	E										
								f	F										
								g	G										
								h	H										
								i	I										
								j	J										
								k	K										
								l	L										
								m	M										
								n	N										
								o	O										
								p	P										
								q	Q										
								r	R										
								s	S										
								t	T										
								u	U										
								v	V										
								w	W										
								x	X										
								y	Y										
								z	Z										
								Blank	NUL										
								BS											
								Tab	HT										
								CR	LF/NL										
								-	CR										
								SP											
								ER	%										
								UC	-										
								LC	-										
								-	(
								-)										
								+											
								-											
								o	:										
								/											
								Del	DEL										
								All	Mark										
								See Note 2.	#										
								*											
								=											
								[
]											
								\$											
								@											
								?											
								.											

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.

3.2.2 GENERAL PART PROGRAM FORM

Part programs are generally written on tapes in the following formats.

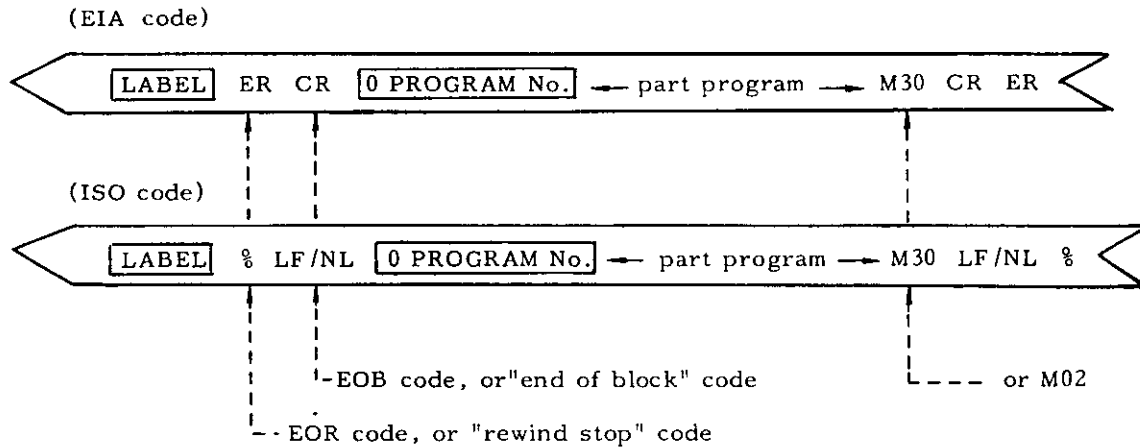


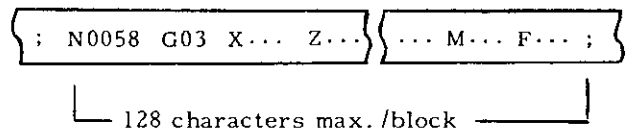
Fig. 3.2

To facilitate classification and handling of tapes, any identifying labels may 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 may be written here, and also out-of-parity codes may 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.

- 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.



(b) Maximum number of effective characters that can be written in a block

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 resets 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.

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.

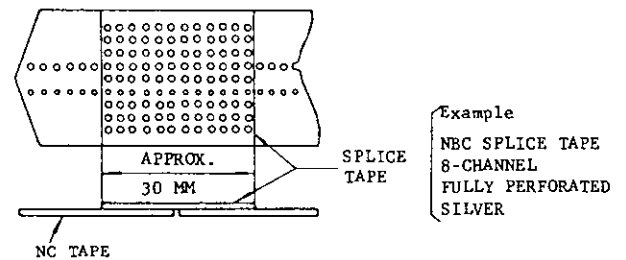


Fig. 3.4 Splicing Part Program Tape

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 STATION 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.

For operation of NC operator's panel with 14" CRT display, see the instruction manual for NC Operator's Station with 14" CRT Character Display (TOE-C843-8.31)

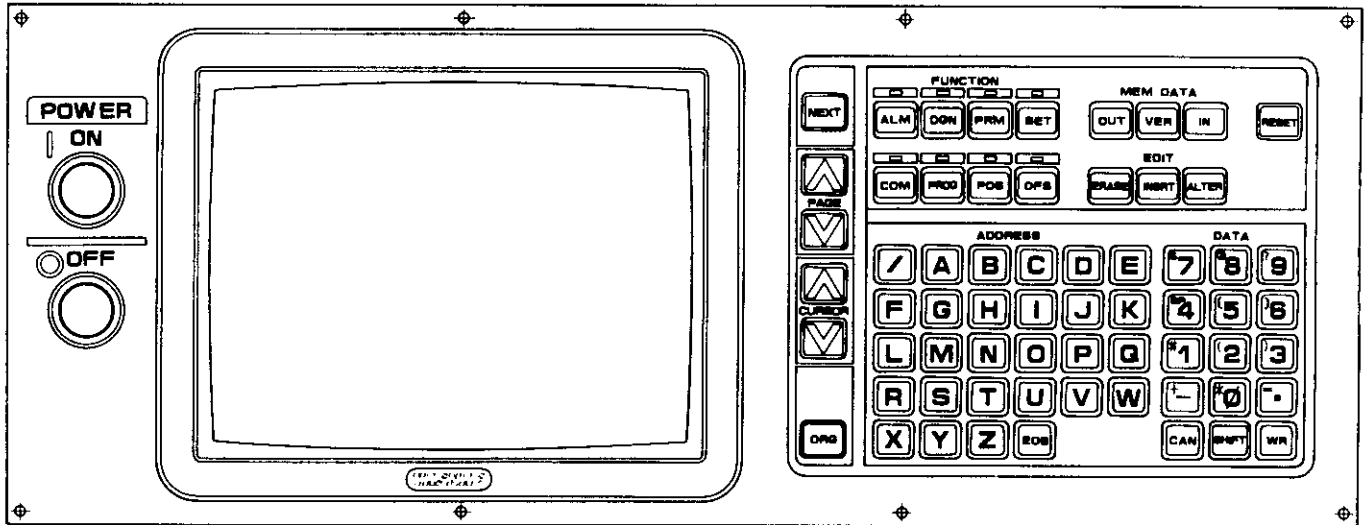


Fig. 4.1 Standard NC Operator's Station with 9" CRT Character Display
(Keyboard on Right Side of CRT)

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.

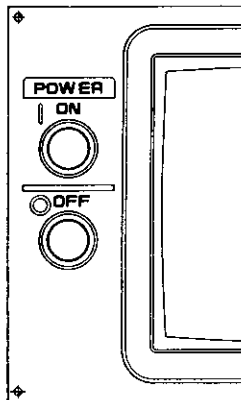


Fig. 4.2

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:

Numerals - [0] through [9], [-], [.]

Alphabetic characters - [A] through [Z]

Special code - [;] (EOB), [/] (slash), etc.

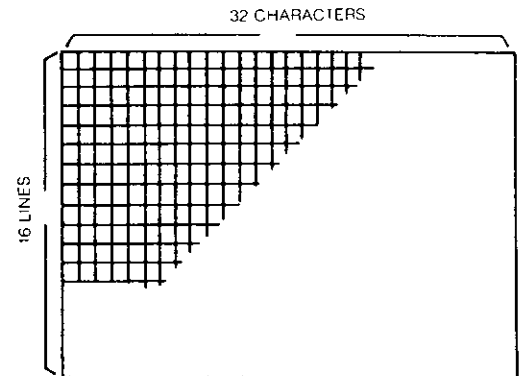


Fig. 4.3 Braun Tube

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.

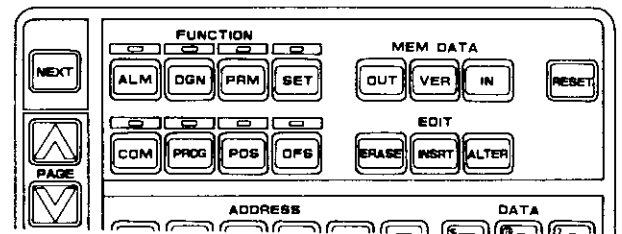


Fig. 4.4

- [ALM] (Alarm) key:

Select this key for display of alarm and status codes. The function becomes effective when the power is turned on or an alarm occurs.

- **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.

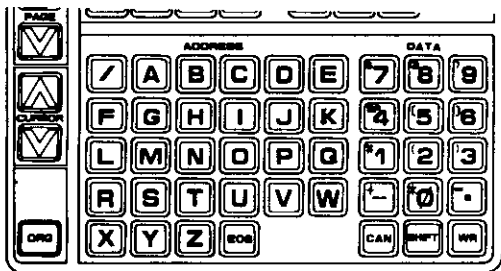


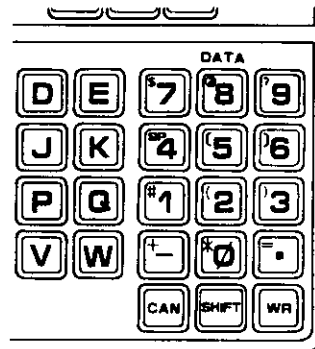
Fig. 4.5

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 0 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.



Note:

- ***0** to ***9** 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.

Fig. 4.6

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.

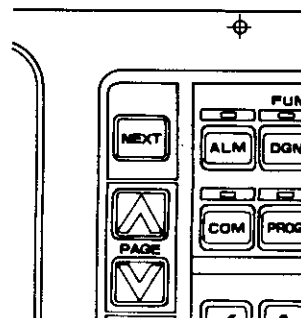




Fig. 4.7

4.1.7 PAGE KEYS

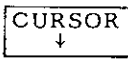
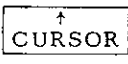
The PAGE key is used to display the next page or the previous page when CRT display is regarded as page.

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 forward.
- Depressing  key moves the cursor backward.
- Keeping the cursor control key depressed makes the cursor move automatically forward or backward.

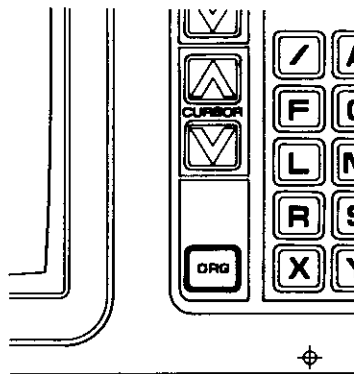


Fig. 4.8

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.

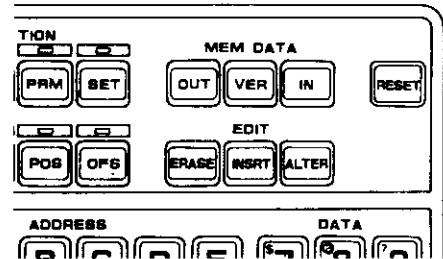


Fig. 4.9

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.

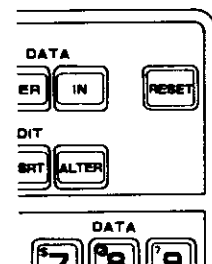


Fig. 4.10

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 of A group

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.

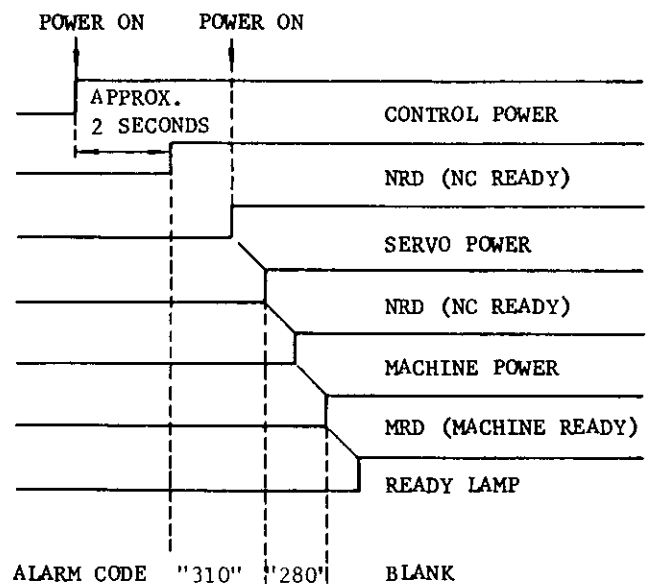


Fig. 4.11 Sequence of Turning on Operation

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.

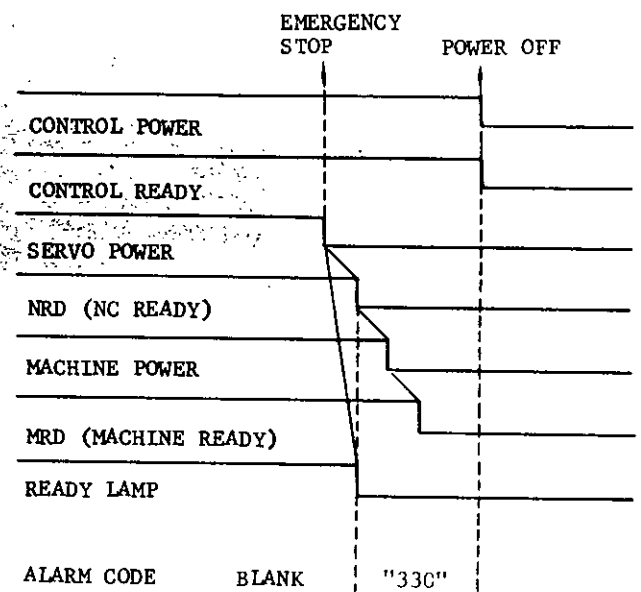


Fig. 4.12 Sequence of Turning off Operation

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.

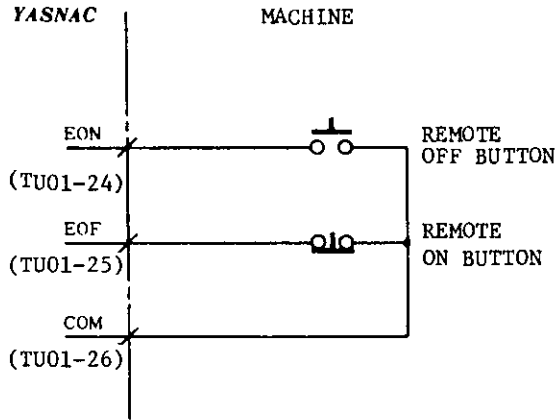


Fig. 4.13 Connections of Remote ON/OFF Pushbuttons

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 | |

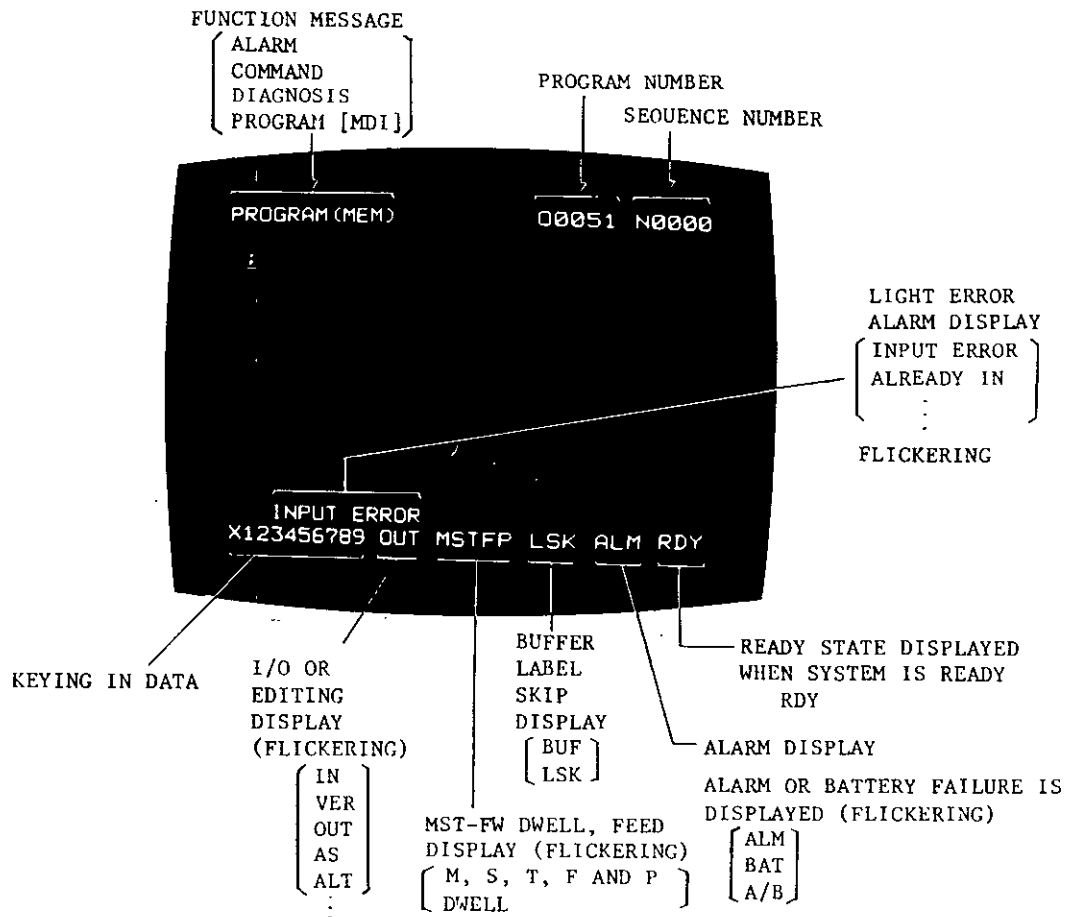


Fig. 4.14

- 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 display 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.
 "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.
 "MEMORY 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 digits 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

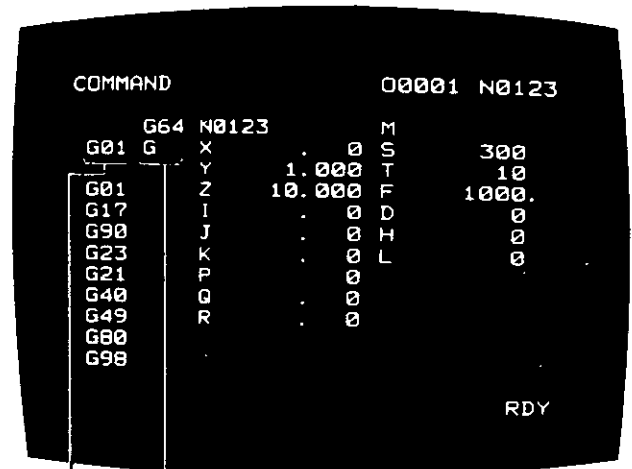
↑
PAGE

 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.

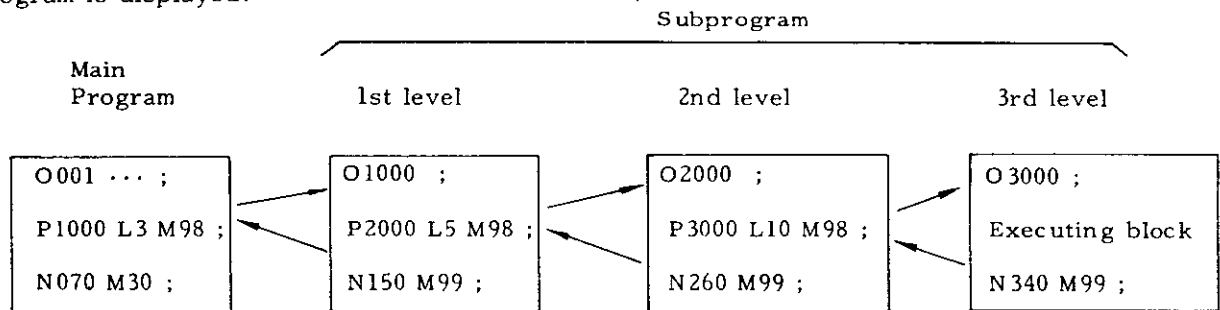
1. The data shows the contents of the active register during an automatic operation or a feed hold.
2. While the control is stopped at a block end, the contents of the buffer register are displayed. If the buffer register blank (BUF is not displayed), the contents of the just executed block are displayed.
3. In the MDI operation, the current block data are displayed after cycle is indicated.



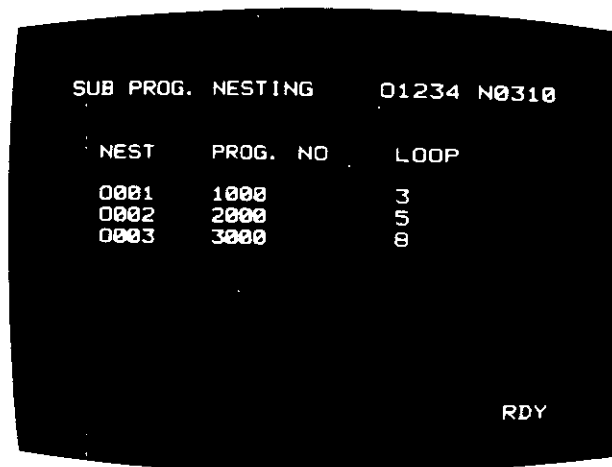
Display of non-modal G code
 Display of G code under execution

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.

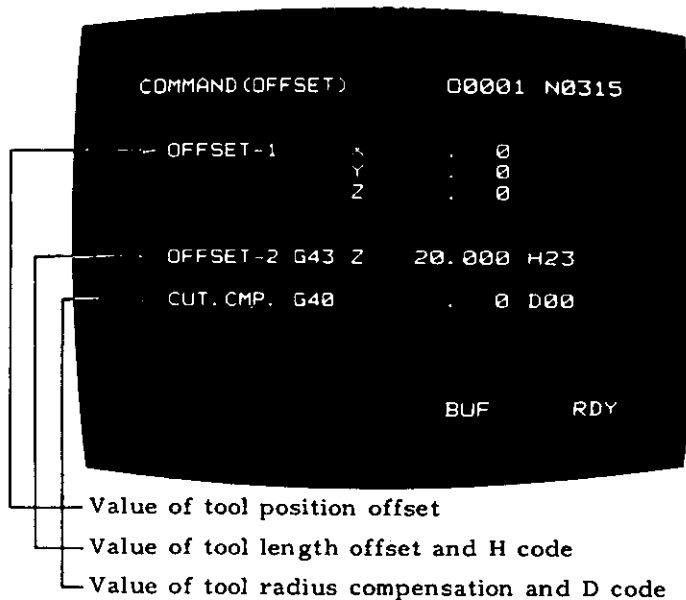


Remaining number of repetitions
 Sequence number of subprogram start
 Subprogram level (up to 4)

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, it is possible to write data into blocks by MDI and perform operation. (MEM mode permits displaying only.) The following operations are possible when function PROG. is selected.

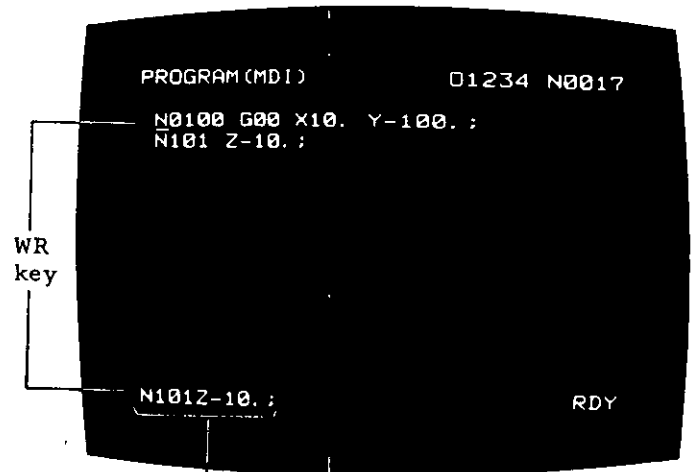
1. In MDI mode

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 may 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** (down arrow), **CURSOR** (up arrow), ERASE, INSRT, and ALTER keys permit editing multi-block data written in. Address (word) pointed to by the cursor will be edited.

The **CURSOR** (down arrow) and **CURSOR** (up arrow) 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, it is possible to edit only the current page. Note: Editing is possible in MDI mode and EDIT mode.

4.3.3 WRITING IN BLOCKS AND DISPLAYING CONTENTS BY MDI (CONT'D)

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.

2. In EDIT mode

See 4.6 EDITING OPERATION OF PART PROGRAMS.

Programs written by MDI can be executed repeatedly by M99.

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 may be displayed at a time. When execution of the ninth has been completed, the next page appears with the tenth line of the last screen appearing at the top.

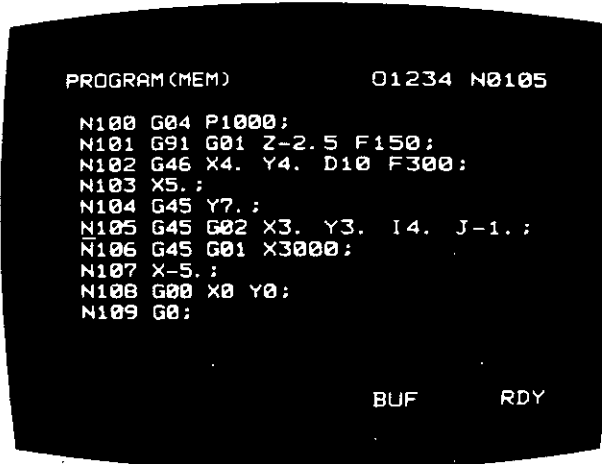


Fig. 4.18

4.3.4 DISPLAYING AND RESETTING CURRENT POSITION

It is possible to observe the current position in any mode. Operate as follows.

1. Depress the POS key. Any of the following screens will appear.
 - A. Current position display-universal (POSITION [UNIVERSAL])
 - B. Current position display-external (POSITION [EXTERNAL])
 - C. Current position display-increment (POSITION [INCREMENT])
 - D. Current position display-all (POSITION)
 - E. Servo positioning error display (POSITION [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 ineffective 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 in reference to 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." This is possible 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." This is possible in any modes and even during operation.
- These displaying and resetting operations are the same as with the case of POSITION (UNIVERSAL) #6005D5 = 0 (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.

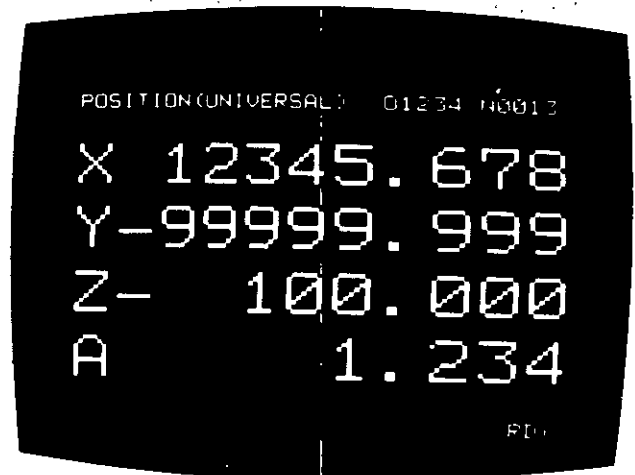


Fig. 4.20 Current Position Display (External)-Example (with 4-axis control)

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)

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)

4.3.4.4 CURRENT POSITION DISPLAY (ALL):
POSITION (Cont'd)

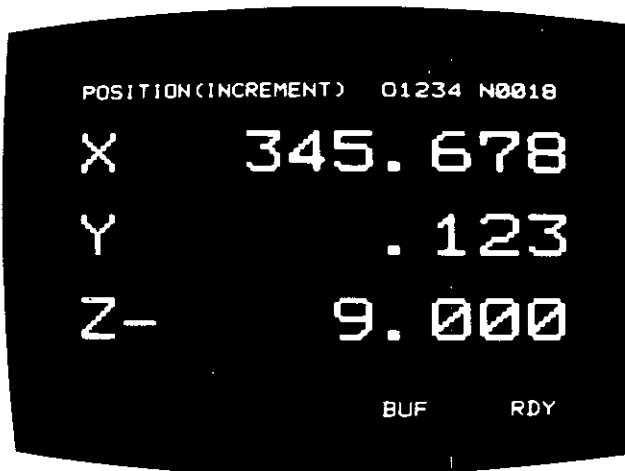


Fig. 4.21 Current Position Display
(Increment) - Example

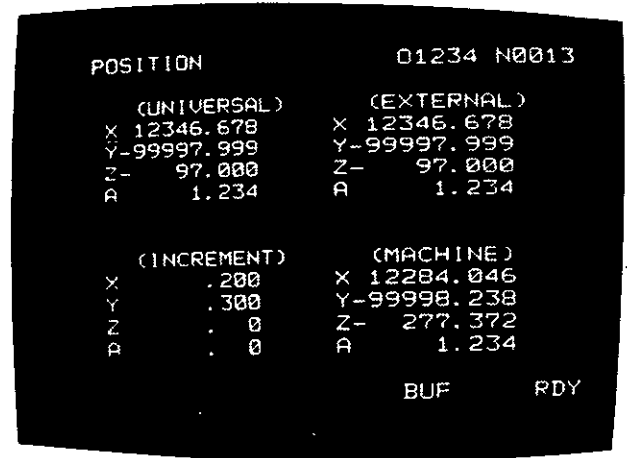


Fig. 4.22 Current Position Display
(All) - Example

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 may be displayed and re-written 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

CURSOR
↓

 or

↑
CURSOR

 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.

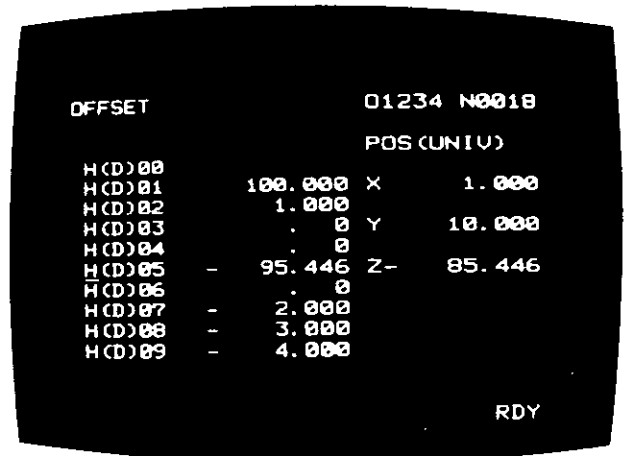


Fig. 4.23 (a)



3. Depress the

CURSOR
↓

 or

↑
CURSOR

 key to move to a smaller or larger tool offset number. If you move the cursor 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 may 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 may 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.
- It is possible to rewrite tool offset data 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) Selection 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.

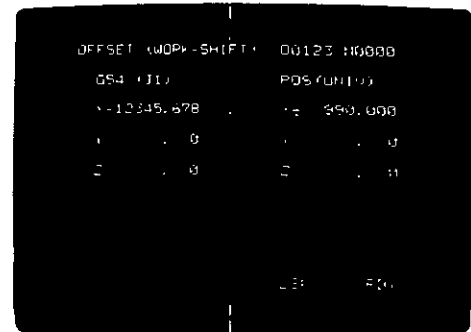


Fig. 4.23 (b) Example of Work Offset Display Screen

4.3.6 DISPLAY IN THE  (SETTING) FUNCTION

4.3.6.1 DISPLAYING AND WRITING SETTING DATA

In this system, varying 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.

4.3.6.1 DISPLAYING AND WRITING SETTING DATA (Cont'd)

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.

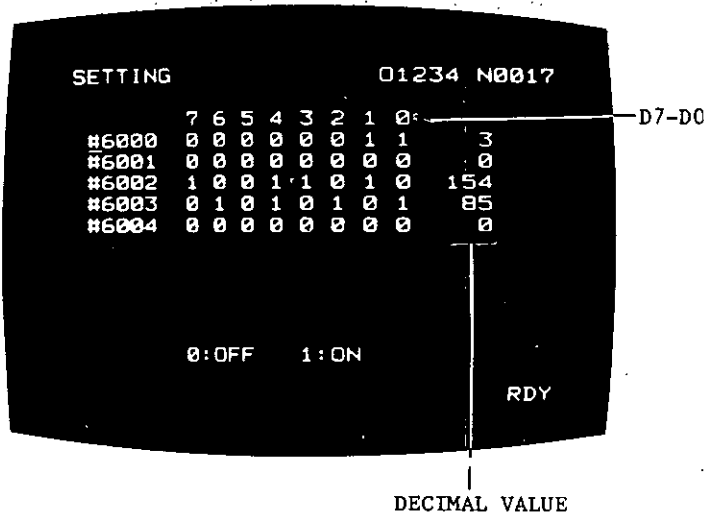


Fig. 4.24 Setting (Decimal model)-Example

B. Decimal mode

Setting numbers of #6200-#6219 and #6500-#6599 are associated with setting data of decimal mode.

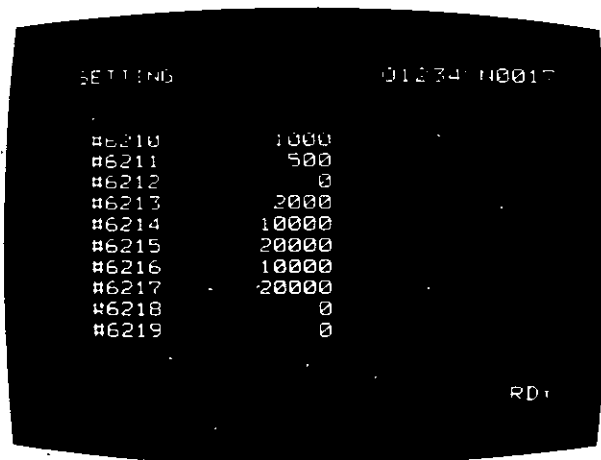


Fig. 4.25 (a) Setting (Decimal mode)-Example

2. Displaying setting data

- Enter a setting number then depress the **CURSOR** 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

- Designate a desired setting number.
- Depress the **INSRT** key. The cursor moves to the bit data from a setting number. Designate the data of D7.
- Depress the **CURSOR** key. Each time the key is depressed, the cursor moves one bit toward D0. Locate the cursor at a desired bit position.
- Depress the **WR** key. The designated bit data reverses (0 to 1 or 1 to 0). If you depress the **WR** key again, the bit data will reverse again. Normally, "1" designates ON state and "0" off state.
- 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	0	0	0	0	0	0	0	0
2 5 5 WR	1	1	1	1	1	1	1	255

- Repeat steps (2) through (5) to write desired data. If you keep the **CURSOR** or **CURSOR** key depressed, the cursor will move column by column in the screen automatically.
- 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. In decimal mode

- Designate a desired setting number.
- 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  function key.

(c) Depress the  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  key. The entered numeral is now written.

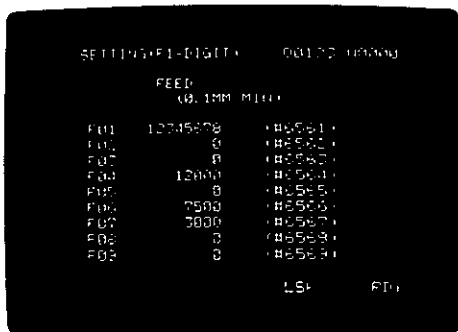


Fig. 4.25 (b) Example of F1-Digit Data Input Display Screen

< 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

To write parameters. At this position, Cycle start is prevented.

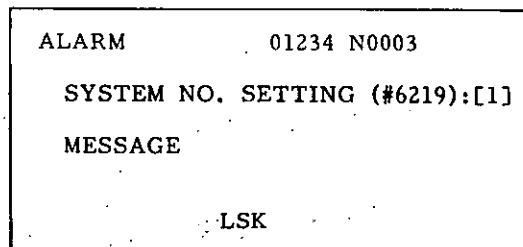
Notes :

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 "2" 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."



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, varying 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 may be displayed at any time even during automatic operation.

1. Kinds of parameters

Parameters are displayed either in decimal mode in binary mode.

PARAMETER	7	6	5	4	3	2	1	0	
#6010	0	0	0	0	0	0	1	1	3
#6011	0	0	0	0	0	0	0	0	0
#6012	0	0	0	0	1	1	1	0	14
#6013	0	0	0	0	0	0	0	0	0
#6014	0	0	0	0	0	1	1	0	6
#6015	0	0	1	0	0	1	1	1	39
#6016	0	0	1	0	0	1	0	0	36
#6017	0	0	0	0	0	1	0	0	4
#6018	0	0	1	0	0	0	0	0	32
#6019	0	0	0	0	0	1	0	0	4

0:OFF 1:ON

RDY

Fig. 4.26 Parameters (in binary mode)-Example

Parameter numbers #6005-#6045 are assigned to binary mode. Those of #6050 and up are assigned to decimal mode.

PARAMETER	Value
#6600	1000000
#6601	2000000
#6602	5000000
#6603	0
#6604	0
#6605	0
#6606	1000000
#6607	1000000
#6608	5000000
#6609	0

RDY

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, "DISPLAYING AND WRITING SETTING DATA" on page 136.

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, "DISPLAYING AND WRITING SETTING DATA" on page 136.

B. Parameters cannot be rewritten unless the system No. switch is set at "1."

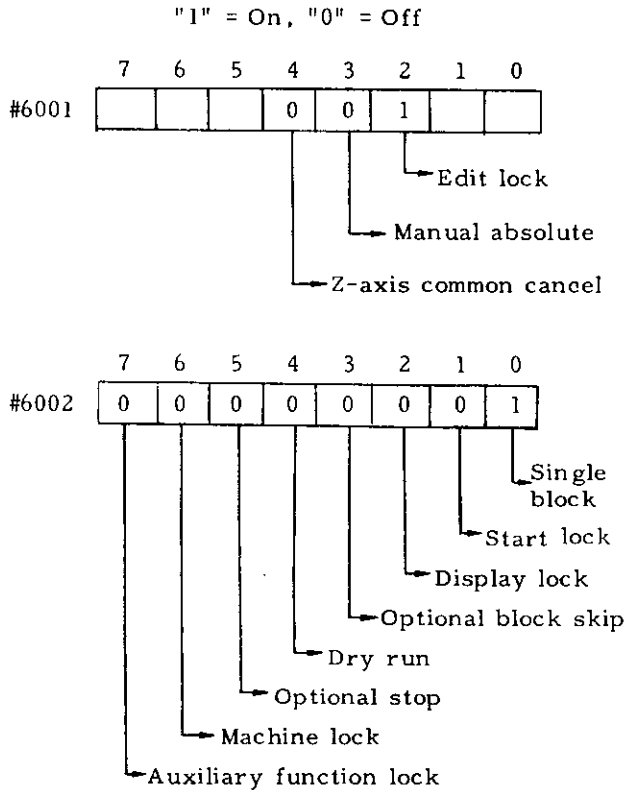
After rewriting parameters, be sure to reset the system No. switch at "0."

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 (2-stage accel/decel switch speed)
 - #6292 to #6296 (2-stage accel/decel switch speed)
 - #6298 to #6302 (Linear accel/decel time)
 - #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

4.3.8 INTERNAL TOGGLE SWITCHES

The following switches may be easily turned on and off on the NC operator's station 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 station is ORed with that of the NC operator's panel will determine the machine condition.

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 station work.

4.3.9 OPERATION TIME DISPLAY

The system counts the duration of automatic operation and it may 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.

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 may 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.

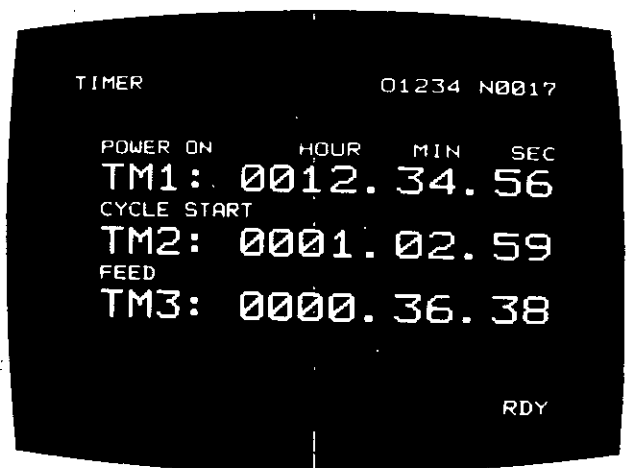


Fig. 4.28

4.3.10 ADDRESS SEARCH

Search continues until a 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 not more than 10 characters headed by address) to be searched.
- E. Depress the

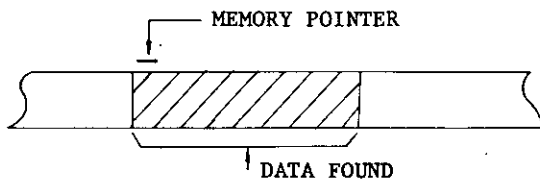
CURSOR ↓

 key. Search starts. "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 you depress a key (CAN normally) of the control station.

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/ N005/ N0005. 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

CURSOR ↓

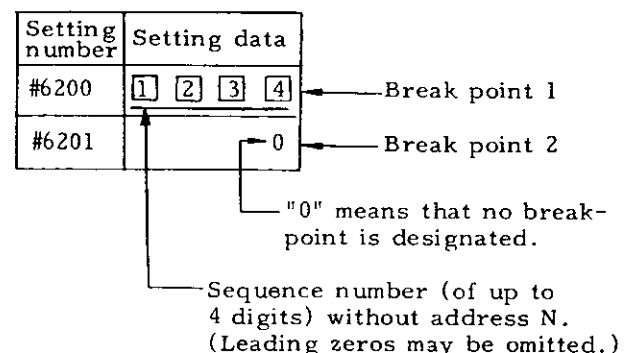
 key.

The designated program number will be searched. The result of search is as described in 2. In MEM mode, you may depress the CYCLE START button immediately after completion of search to start automatic operation from the beginning of the program.

4.3.11 SEQUENCE NO. COLLATION STOP FUNCT

It is possible to suspend operation at the end of a block by designating a sequence number in set function. Location is 6200 and 6201.

1. If the current sequence number is found to be equal to a sequence number designated as setting data during automatic operation, operation will stop after execution of the block like in single block operation.
2. The designated sequence number is called a breakpoint and up to two breakpoints may be designated.
3. Setting numbers are as follows for designating breakpoints.



4. "BREAKPOINT!" appears blinking when operation has stopped at a breakpoint. To restart, depress the CYCLE START button.

Note: If the breakpoint function is not used, set the contents of #6200 and #6201 to "0."

4.3.12 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 may 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, it is not necessary operate the PAGE key.

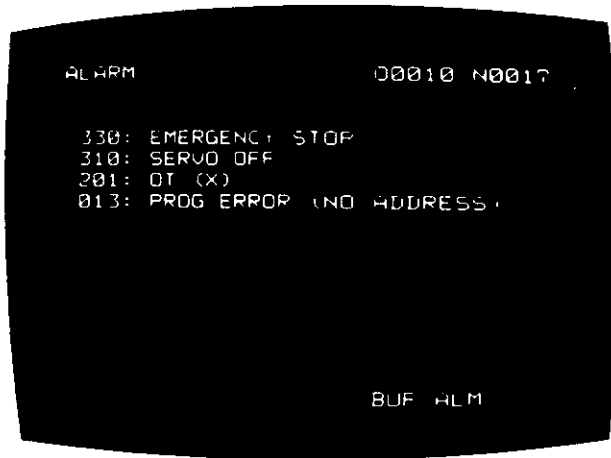


Fig. 4.29 Alarm Codes and Message Displayed-Example

To reset the alarm status and screen, remove the cause of alarm then depress the RESET key.

For the detail of alarm codes, see Appendix 5, "LIST OF ALARM CODES" on page 240.

4.3.13 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. This is possible at any time even during automatic operation.

For more detail of this operation, see 8.6.3, "DIAGNOSTICS OF INPUT/OUTPUT SIGNALS" on page 198.

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 may be read from and written onto tape. A tape reader will 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 Pars. 4.7.2 and 4.7.3 for how to set the type of input/output device (setting #6003) and baud rate (parameter #6026).

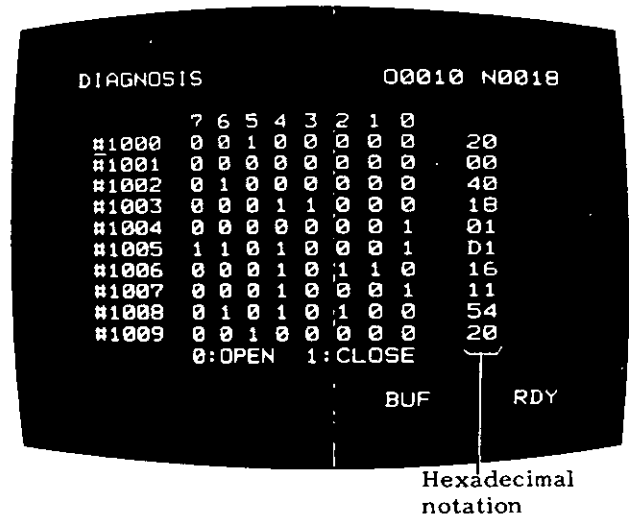
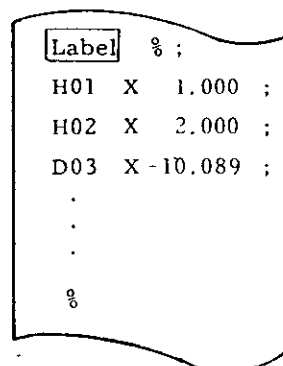


Fig. 4.30 State of I/O Signals Displayed-Example

4.4.1 INPUTTING TOOL OFFSETS FROM TAPE

Though tool offsets are normally input by MDI operation, they may also be entered by means of paper tape.

1. The tape format of tool offsets is as follows.



Note:
Either H or D may 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.

4.4.1 INPUTTING TOOL OFFSETS FROM TAPE (Cont'd)

E. Depress the IN key.

The tape reader starts to read the tape. "IN" blinks on the CRT while the data are read.

F. The tape reader stops when it has read "%" (or ER). "IN" disappears from the CRT. Now the tool offset data have been read into memory.

4.4.2 INPUTTING SETTING DATA AND PARAMETER DATA

Though setting data and parameter data are normally input by MDI operation, they may also be entered by means of paper tape. Setting data and parameter data may be input from a single tape.

1. The tape format is as follows.

Label	%	;
N6000	X128	;
N6001	X 5	;
N6002	X 0	;
.	.	.
.	.	.
.	.	.
		%

Note:
 "%" is used in the ISO code and "ER" in the EIA code.

2. The input operation is as follows.

- A. Select EDIT mode.
- B. Depress the RESET key.
- C. Depress the PRM key.
- D. Set the setting/parameter data tape onto the tape reader.
- E. Depress IN key.

The tape reader starts to read the tape. "IN" blinks on the CRT while the data are read.

F. The tape reader stops when it has read "%" (or "ER"). "IN" disappears from the CRT. Now the setting/parameter data have been read into memory.

Turn on power again because the control is in the HOLD state (key inoperative) at completion of input.

4.4.3 OUTPUTTING TOOL OFFSETS TO PAPER TAPE

The tool offset data set in the system may be output to paper tape.

1. The output operation is as follows.

- A. Select EDIT mode.
- B. Depress the RESET key.
- C. Depress the OFS key.
- D. Check that the punch is ready for operation.

E. Depress the OUT key.

The paper tape punch punches the tool offset data onto paper tape and stops automatically when all contents of the tool offset memory have been output.

F. To suspend the operation, depress the RESET key.

At this time, the output operation cannot be resumed. Restart from the beginning.

2. The tape format is the same as that described in 4.4.1, "INPUTTING TOOL OFFSETS FROM TAPE" on page 143.

4.4.4 OUTPUTTING SETTING DATA AND PARAMETER DATA TO PAPER TAPE

1. The output operation is as follows.

- A. Select EDIT mode.
- B. Depress the RESET key.
- C. Depress the PRM key.
- D. Check that the punch is ready for operation.

E. Depress the OUT key.

The paper tape punch punches the setting parameter data onto paper tape continuously.

F. To suspend the operation, depress the RESET key.

At this time, the output operation cannot be resumed. Restart from the beginning.

2. The tape format is the same as that described in 4.4.2, "INPUTTING SETTING DATA AND PARAMETER DATA."

4.4.5 OUTPUTTING PART PROGRAM TO PAPER TAPE

- (1) The part program of the designated program number is punched out by the following operations:
 - a. Connect the external equipment such as the tape puncher to the NC via the data input/output interface.
 - b. Make the external equipment relay for operation.
 - c. Power on the NC.
 - d. Select the EDIT mode.
 - e. Depress the PROG function key.
 - f. Check to see if the external equipment is ready.
 - g. Depress the RESET key.
 - h. Depress "O" key and key in program number.
 - 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 flashing.

- j. To discontinue the punch out operation, depress the RESET key. However, the discontinued operation cannot be resumed. Go back to f. and repeat the operations all over 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 may be output to the external equipment by the above operations except that 0 - 9 9 9 9 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 #6231D₃ = 1.

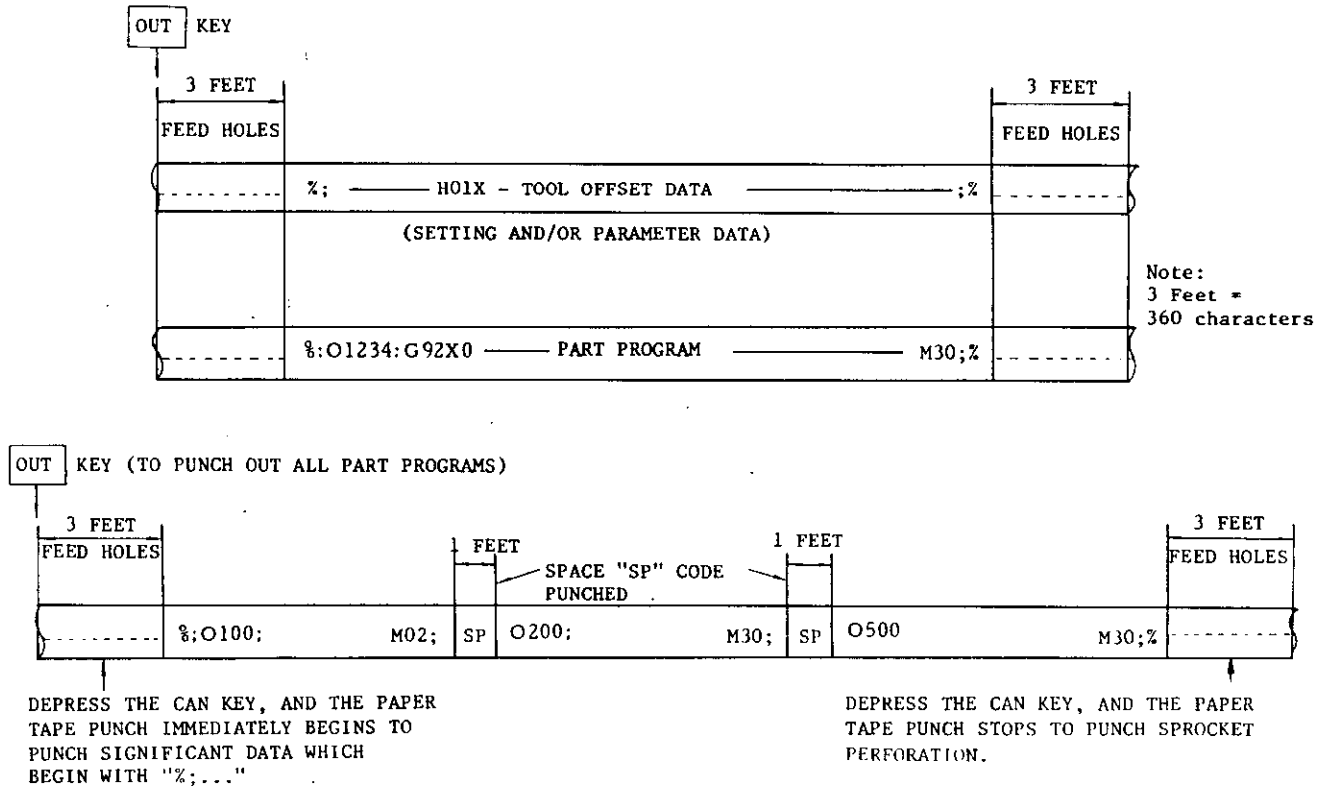
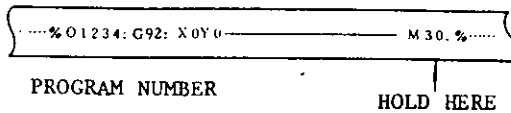


Fig. 4.31 Data and Program Formats on Paper Tape

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 device.

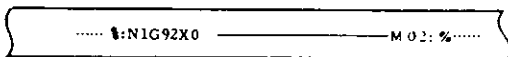


- d. Depress the RESET key.
- e. Depress the IN key.

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 a duplicate of the program number as in 1. Operation ends with error if the designated program number is not found on the tape.

When the tape reader has read "M02 ;," "M03 ;," or "M99 ;," it stops and "IN" 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 device.



The tape stops at this location when loading is completed.

- d. Depress the RESET key.
- e. Depress the address O key then enter the program number.
- f. Depress the IN key.

The system starts to read 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 may 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 "00000" 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 "0," "-", "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 "0," "-", "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 X0 Z0 ;

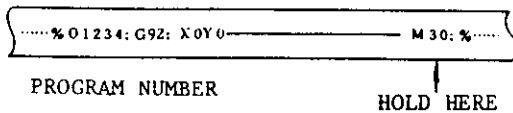
When this is in the first block, position the cursor to N and key in as follows (in EDIT and PROG modes):

Oxxxx ; N1 ALTER

- (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. Key in "O" and PROGRAM NUMBER.
- f. Depress the IN key.

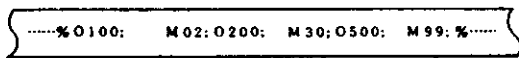
The program number entered from the key is registered in preference to the program number punched on the tape. At this time, the program number on the tape is written to the part program memory simply as a label. M02 ;, M30 ; or M99 ; is read and the storing operation is completed.

Note :

- 1. 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 device.



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 depression of the IN key. The tape reader stops at the position of "%."

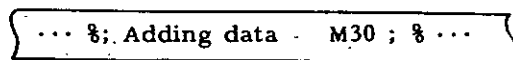
4.5.2 MAKING ADDITION TO A PART PROGRAM

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 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.

Note: You cannot add data to a program from the middle of it. If necessary, delete the last part of the program by editing operation and perform this adding.

4.5.3 LOADING PART PROGRAMS BY MDI

Part programs may 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 then enter the part program number and depress the WR key.
The designated program number will be registered. If this number already exists, "ALREADY IN" blinks and, in this case, it is required to delete the registered program number.
- 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.

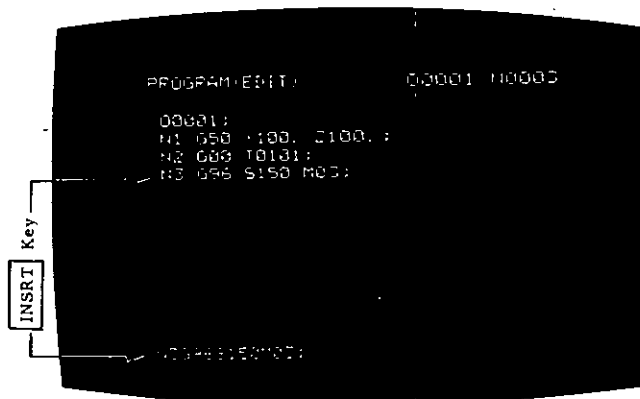


Fig. 4.32

- 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

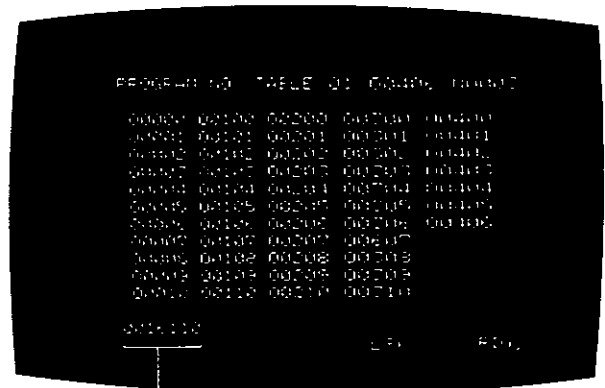
- (2) All program numbers already registered are displayed. By depressing

PAGE

 or

↑
PAGE
↓

 key, the page shown below may be obtained.



REMAINING NUMBER OF CHARACTERS IN PART PROGRAM MEMORY

The remaining number of characters in part program memory is displayed in the lower left corner of the screen.

Fig. 4.33

Note: This screen displays only the registered program numbers. A program number is registered by depressing the PROG function key in EDIT mode.

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

4.6.1 PART PROGRAM DISPLAY

Stored program contents can be displayed, and checked by the operator.

Part Program Call

1. Select the EDIT mode.
2. Depress the RESET and PROG keys.
3. Input the program number with ADDRESS O and 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.

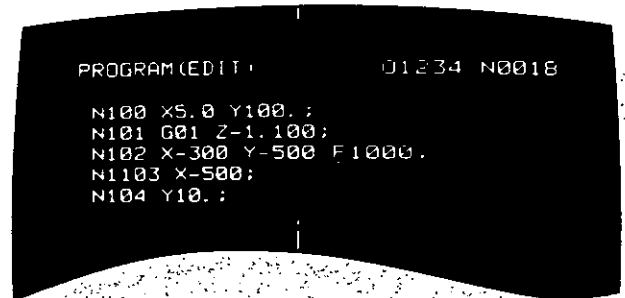
4.6.2 DELETING PART PROGRAM BLOCK

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

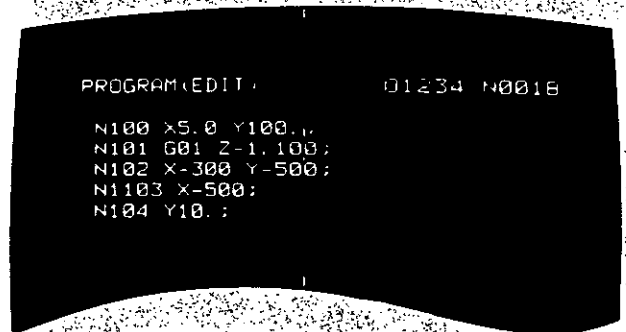


Fig. 4.35

4.6.2 DELETING PART PROGRAM BLOCK (CONT'D)

Deletion of Program No.

Enter the program No. with address O and depress the ERASE key. The specified 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. "0" 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

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.

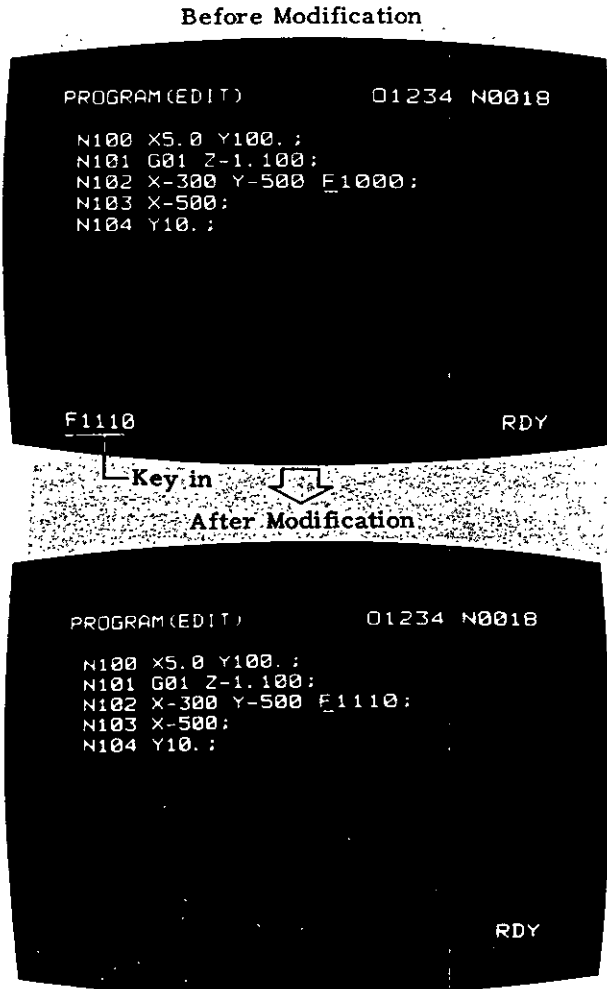


Fig. 4.36

Words less than 32 characters can replace one word specified.

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 #6005 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.

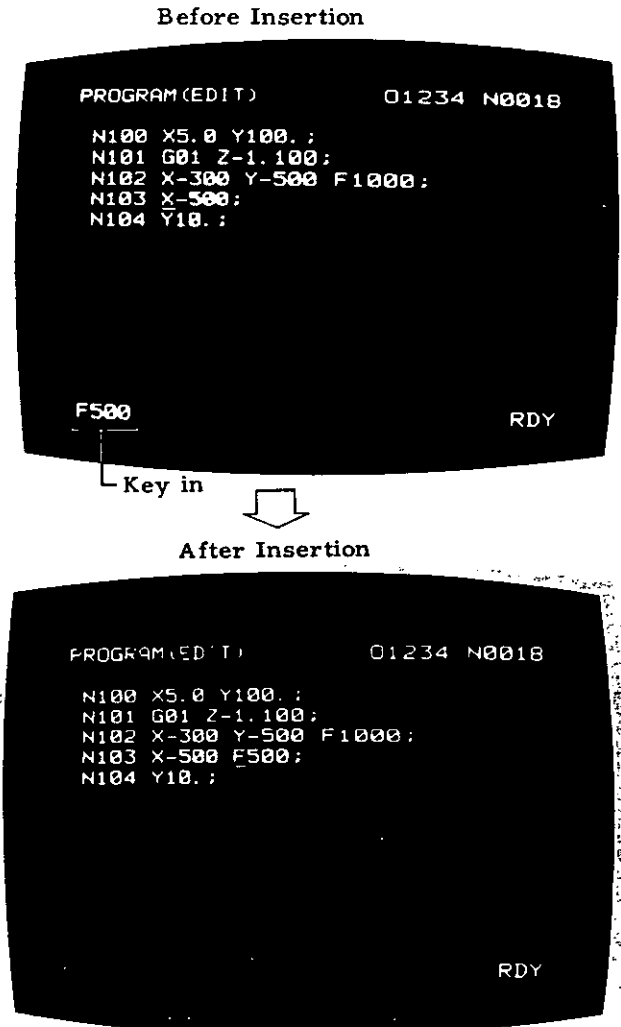


Fig. 4.37

After insertion, CURSOR indicates the last word keyed in 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 **[2]** key and **[SHIFT]** key, and character "(" may be entered.
- b. Depress **[3]** 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.

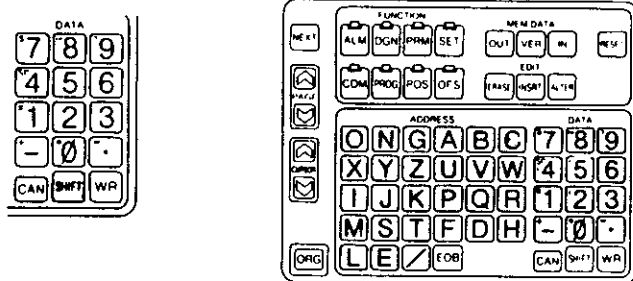


Fig. 4.38

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	②
Name of interface	RS232C Interface No. 1	RS232C Interface No. 2
Type of interface	Serial-Voltage Interface	
Data transmission speed	Parameter setting	
	Input: #6026 Output: #6028	Input: #6027 Output: #6029
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. 	

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.

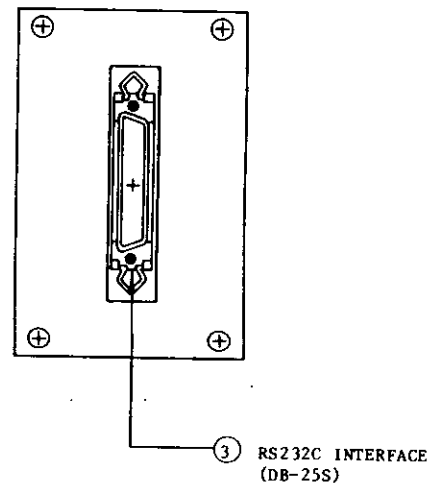


Fig. 4.39 Data I/O Interface Receptacles in Control Cabinet

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

INVCE1 (#6003,D1)	IDVCE0 (#6003,D0)	Data Input Interface to be used
0	1	RS232C Interface No.1
1	0	RS232C Interface No.2

Note: PTR interface is for the standard tape reader. Usually, this interface is set.

(2) Setting of Data Output Interface to Be Used

ODVCE1 (#6003,D5)	ODVCE0 (#6003,D4)	Name of Interface
0	1	RS232C Interface No.1
1	0	RS232C Interface No.2

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.

(1) Current Loop or RS232C Interface

As shown below, the data is set for input and output combined or separately.

#6028 D6

0 ... Data is set for input and output combined.

1 ... Data is set for input and output separately.

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
	9600	1	0	1	0

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

d. Setting of Baud Rate

Input	#6027D3	#6027D2	#6027D1	#6027D0	
Output	#6029D3	#6029D2	#6029D1	#6029D0	
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
	9600	1	0	1	0

e. Setting Stop Bit

Input	#6027D4	= 1: Stop bit as 2 bits
Output	#6029D4	= 0: Stop bit as 1 bit

f. Setting Control Code Sending

Input	#6027D5	= 1: Does not send control code
Output	#6029D5	= 0: Sends control code

Notes:

1. Set the baud rate and stop bit length according to the specifications of the input/output equipment to be used.
2. 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 RS232C 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			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			IO BUSY
	Not used	25			ER (OR IO ALARM)

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 RS232C 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			
		1			ER (OR IO ALARM)
	Not used	25			

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" on page 137.
- (2) The storing of part programs into memory. For details, see 4.5, "LOADING PART PROGRAMS INTO MEMORY" on page 140.
- (3) Tape-verification of part programs, tool offset amount, setting data, and parameter data. For details, see 4.8, "TAPE VERIFYING" on page 147.

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 4.9, "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, "INPUT 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 keyed-in 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 keyed-in program number. Hence, when verifying a tape containing program numbers, no program number should be keyed in.

(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	EDIT	PRM	Parameter number + <input type="text" value="CURSOR"/> Data + <input type="text" value="WR"/>	
	Storing from tape (Note 4) (Note 6)		1			RESET + IN	
	Punch out (Note 3)					<input type="text" value="RESET"/> + <input type="text" value="OUT"/>	
	Matching with tape (Note 4)					<input type="text" value="RESET"/> + <input type="text" value="VER"/>	
Setting	Storing from NC operator's panel keyboard			EDIT	SET	Setting number + <input type="text" value="CURSOR"/> Data + <input type="text" value="WR"/>	
	Storing from tape		1			<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"/>	
Offset	Storing from NC operator's panel keyboard			EDIT	OFS	Offset number + <input type="text" value="CURSOR"/> 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 NC operator's panel keyboard	OFF		EDIT	PROG	<input type="text" value="O"/> + Program number + <input type="text" value="WR"/> Repeat of edit operation "addition of address data"	
	Storing from tape	One part program	Tape with number O			OFF	<input type="text" value="RESET"/> + <input type="text" value="IN"/>
			Tape without number O			OFF	<input type="text" value="RESET"/> + <input type="text" value="O"/> + 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="O"/> + -9999 + <input type="text" value="OUT"/>
	Matching with tape	One part program	Tape with number O				<input type="text" value="RESET"/> + <input type="text" value="VER"/>
			Tape without number O (Note 1)				<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 2)				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 2)				OFF	<input type="text" value="CURSOR"/> (Set to address data just before addition) + Address data + <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 5)				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
Clear	Designated part program		OFF	EDIT	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="O"/> + -9999 + <input type="text" value="ERASE"/>	

Notes:

1. 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."
2. Within the limit of 32 characters, addition of multiple address data and the change to one address data are permitted.
3. Setting is punched out at the same time.
4. If the tape contains setting information, it is also stored and matched at the same time.
5. 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.
6. When data has been stored from a parameter tape, turn the power on and off.

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, JOG, and HANDLE modes are called manual operation mode, and TAPE, MDI, and MEM, automatic-operation mode in this manual.

RAPID: Allows the tool to traverse rapidly or return to reference zero by manual operation.

JOG: Allows the tool to feed continuously by manual operation. Feedrate is set by JOG FEEDRATE switch.

STEP: Allows the tool to feed manually by step each time JOG pushbutton is depressed.

HANDLE†: Allows the tool to feed by operating the manual pulse generator†.

TAPE: Automatically controls the system from NC tape.

MDI: Allows the operator to insert up to 10 blocks of data through the DATA keyboard and control the system automatically with the data.

MEM: Automatically controls the system with the stored part program.

EDT: Stores the part program into memory and edit the part program.

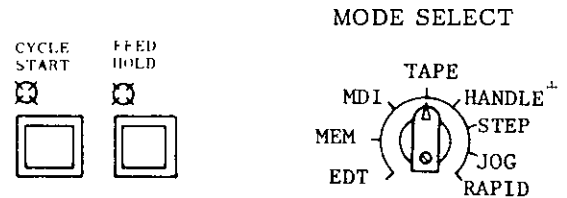


Fig. 5.2

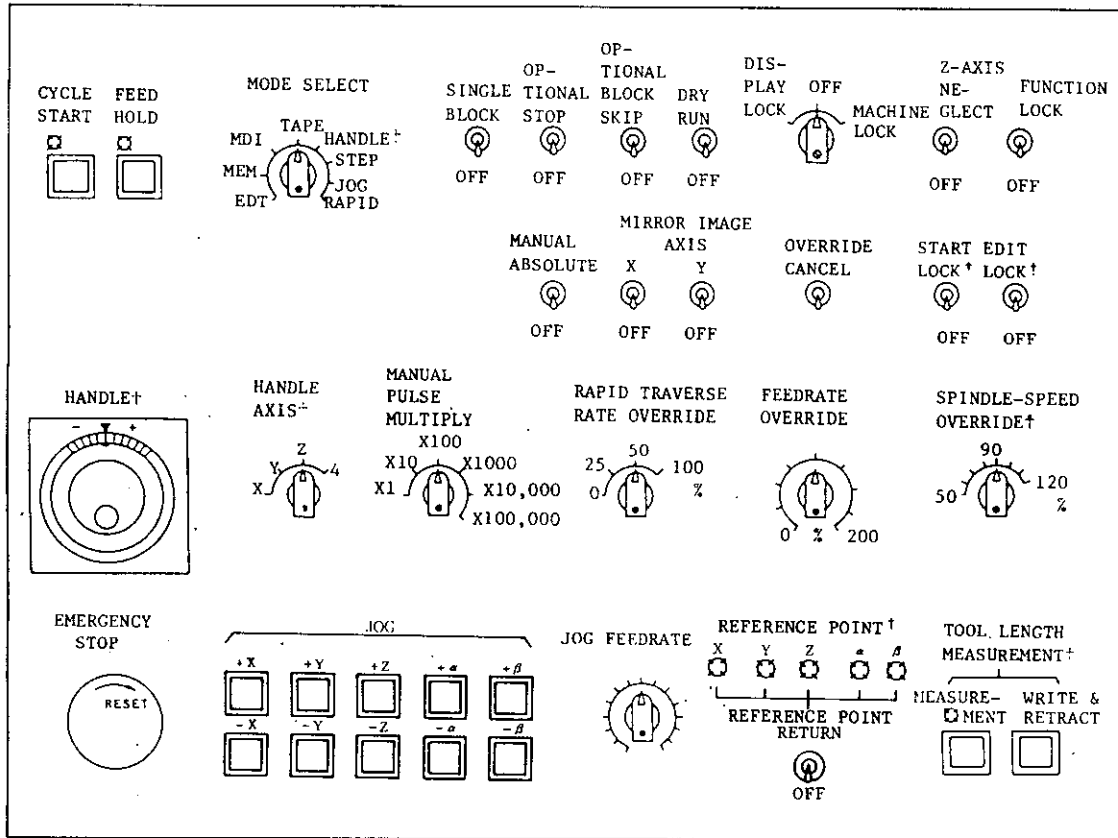


Fig. 5.1 Machine Control Station

5.1.2 CYCLE START PUSHBUTTON AND LAMP

Depress this pushbutton to start the system in the automatic operation mode (TAPE, MDI 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. Depressing the FEED HOLD pushbutton during dwell by G04 works on completion of the current block.

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.

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" and "310" are 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 deleted from page.
3. Turn on the servo power again by depressing POWER ON pushbutton. NC ALARM LAMP is extinguished and READY lamp lights up.

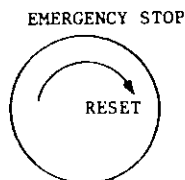


Fig. 5.3

5.1.5 HANDLE DIAL[†] (MANUAL PULSE GENERATOR)

The dial is used as a manual pulse generator to feed the tool manually with the MODE SELECT switch set to the HANDLE. HANDLE operation is effective for an axis. Procedure of HANDLE operation is as follows:

1. Set the MODE SELECT switch to the HANDLE.
2. Select the axis to be operated with HANDLE AXIS select switch.
3. Set the move amount per graduation of the dial by setting MANUAL PULSE MULTIPLY switch.
4. Rotate the dial to move the selected axis.

Turning it clockwise causes the axis to move in the plus direction. The axis moves in the minus direction by turning it counterclockwise.

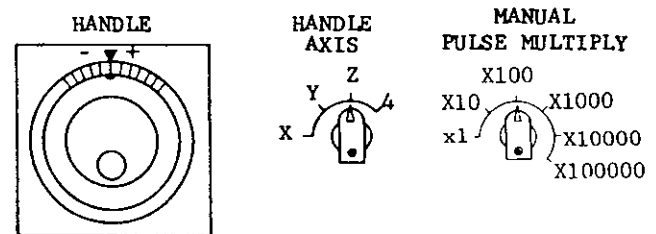


Fig. 5.4

5.1.6 HANDLE AXIS SELECT SWITCH[†]

This switch is used to select an 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 6.1. 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.

Note: Handle coasting can be prevented by #6677 (handle stock pulse scrap capacity). With this parameter installed, when handle is turned quickly there will be a gap between quality of handle turning and that of movement.

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 HANDLE DIALS FOR SIMULTANEOUS CONTROL OF UP TO THREE AXES[†]

When a manual pulse generator is connected for each axis, the tool can be manually moved along selected three of the five axes (X,Y,Z, α and β) simultaneously.

1. The tool move distance per graduation of the HANDLE dial for the manual pulse generator is determined by the MANUAL PULSE MULTIPLY switch (Table 5.1). This switch is effective on all the three axes.
2. Set the mode select switch to HANDLE, and turn the HANDLE dials for the desired axes in the positive or negative direction.

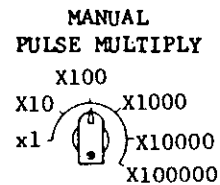
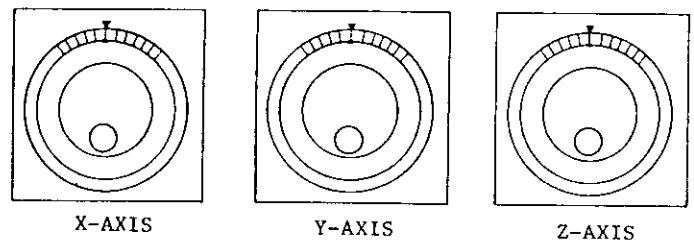
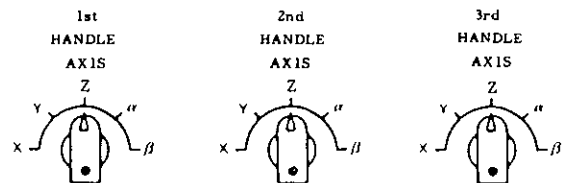


Fig. 5.5

5.1.9 JOG PUSHBUTTONS

This pushbutton is used to feed the tool manually.

- With any of pushbuttons +X, -X, +Y, -Y, +Z, or -Z (+ α , - α , + β , - β)[†] 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.

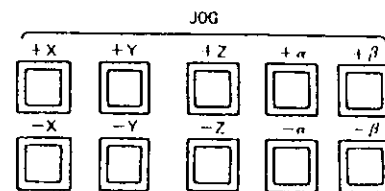


Fig. 5.6

5.1.10 JOG FEEDRATE SWITCH

The JOG FEEDRATE switch is used to select the jog feedrate in the JOG mode. Up to 32 steps of feedrate can be specified. Jog feedrate depends on the machine tool. For definite values, refer to the machine tool builder's manual. See Table 6.7. The JOG feedrate can be preset by parameters #6233 to 6264.

JOG FEEDRATE RAPID TRAVERSE
RATE OVERRIDE

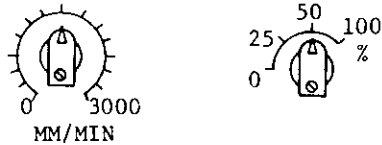


Fig. 5.7

5.1.11 RAPID TRAVERSE RATE OVERRIDE SWITCH

This switch is used to adjust the traverse rate by F0, 25, 50 and 100%. 100% Rate is the rapid traverse rate set by parameter #6280 to 6283. The switch is effective both in automatic operation including G00 command and in manual operation (RAPID mode). F0 is set by parameter #6231.

5.1.11.1 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%, F0, F1, and F2."

(2) The F1 and F2 feedrates are the feedrates given by the parameter (#6466, #6467).

5.1.12A 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 5.3 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 and fifth axes[†], if provided, is shown by deg/min.

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	—	—

OVERRIDE
CANCEL

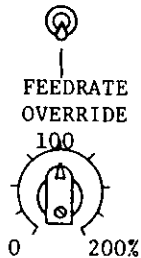


Fig. 5.8

5.1.12B 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.



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.

SPINDLE-SPEED
OVERRIDE†

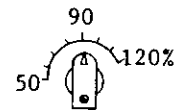


Fig. 5.9

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 6.2.1. "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.

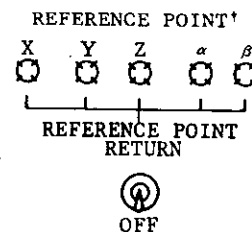


Fig. 5.10

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.

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, MDI or MEM mode causes the tool to move at the speed selected by the JOG FEEDRATE switch, ignoring all programmed F-functions. F commands can be displayed as they are programmed. This switch may be used to check the program.

Rapid traverse (G00) rate for dry run operation can be set by setting parameter # "6006D2."

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.
- Rapid traverse rate override is kept effective during dry run operation.
- During tapping, the set speed when tapping starts will be kept. It cannot be changed by JOG operation during dry run operation.

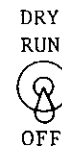


Fig. 5.11

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 FEEDHOLD pushbutton.

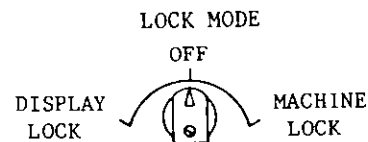


Fig. 5.12

"OFF"

Usual operation is made at "OFF" position in both manual and automatic operation. The machine and the position display operate according to the command by automatic operation or manual operation.

"DISPLAY LOCK"

This position is used to exclude the axis movement value from the position display. Current position display is not updated, though the machine moves.

"MACHINE LOCK"

Setting the switch at MACHINE LOCK inhibits axis movement including Zero Return. The position display is updated. M-, S-, and T-functions are executed. This position is selected to preset the display or to check the tape data.

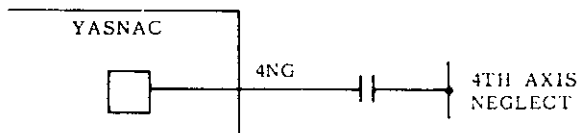
5.1.22A Z-AXIS FEED NEGLECT SWITCH

The switch is used for dry run operation or drawing-check operation on the X-Y plane. Turning on the switch causes the Z-axis in MACHINE LOCK condition. The Z-axis movement is inhibited, though the position display is updated.

Operate the switch when the machine is stopped. That is, the switch does not function except when the machine is stopped at the block end by SINGLE BLOCK switch or temporarily stopped by FEED HOLD pushbutton.

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).



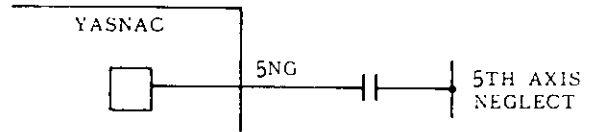
(2) The ON/OFF switching of parameter #6023 D6 is equivalent to the ON/OFF switching of the 4th axis neglect input. The result of these two is determined by their logical OR.

4NG Input	#6023 D6	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 (Opened)

Note: When 4NC input or #6023 D6 is switched over, be sure to switch on the power supply again. Otherwise, the switching is ineffective.

5.1.22C 5TH AXIS NEGLECT INPUT

(1) When the POWER ON switch is pushed with a 5th axis neglect input on (contact is closed), the system is equivalent to one without the 5th axis control. If motion along the 5th axis is commanded in this state, the machine remains motionless along the 5th axis, but the position display indicates the suppressed motion. (Machine lock state).



(2) The ON/OFF switching of parameter #6024 D6 is equivalent to the ON/OFF switching of the 5th axis neglect input. The result of those two is determined by their logical OR.

5NG Input	#6024 D6	Logical OR (5GNC for 5th Axis Neglect)
ON	ON (1)	ON (Closed)
ON	OFF (0)	ON (Closed)
OFF	ON (1)	ON (Closed)
OFF	OFF(0)	OFF (Opened)

Note: When 5NC input or #6024 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.



Fig. 5.13

- The following M codes are executed even if the switch is set on.
 - M00, M01, M02, M30
Both its decoded signals and its BCD codes are sent out to the machine.
 - M90 to M99
BCD code is not sent out.
- Turning on the M-FUNCTION LOCK switch during automatic operation becomes effective on the block after the next block of the current block.

5.1.24 MANUAL ABSOLUTE SWITCH

- When MANUAL ABSOLUTE SWITCH is on.

When automatic operation is restarted after interrupted by manual operation, the tool performs the rest of the command in the interrupted block from the end point of manual operation. The tool moves in parallel with the path specified by the program.

When the command of the next block is G00 or G01, the tool moves automatically to the target coordinate specified by the program. Then the operation is performed according to block of data.

When the command of the next block is G02 or G03 (circular interpolation), the interpolation is performed in parallel with programmed command. The tool automatically returns to the target coordinate when G00 or G01 is commanded after the circular interpolation.

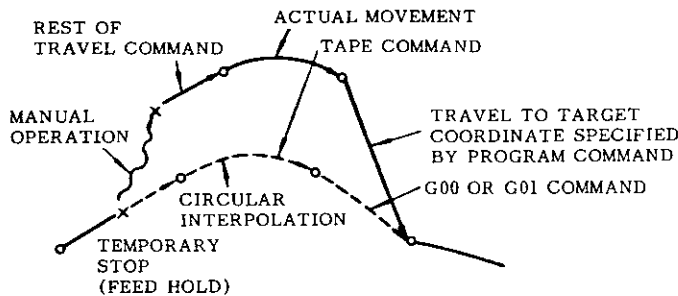


Fig. 5.14

- 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.

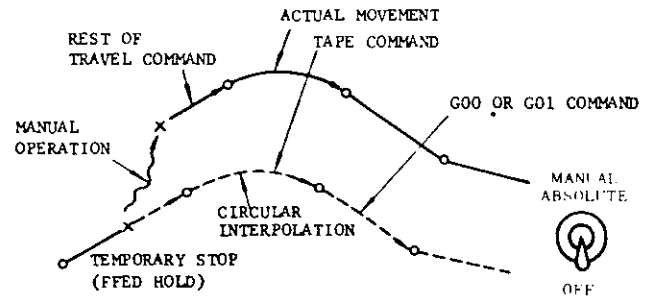


Fig. 5.15 Tool Movement with MANUAL ABSOLUTE Switch Off

The parallel shift is reset by executing Reference Zero Return manually, automatically by G28, or operating the RESET key. The command value is forced to change to the current position. Thus the shift value is reset.

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.

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 # "6000D0-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 M95 (Mirror Image ON) mode, never operate the MIRROR IMAGE AXIS switch.

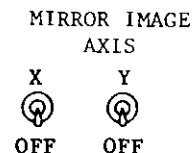


Fig. 5.16

5.1.26 TOOL LENGTH MEASUREMENT PUSH-BUTTON AND LAMP†

Use the WRITE button to automatically store the amount of Z-axis move manually made between "home-position" and "base-position" directly in the tool offset memory. For operating procedure, refer to 5.2.3 Automatic Tool Length Measurement†.

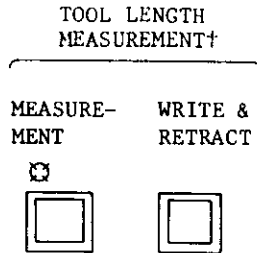


Fig. 5.17

5.1.27 START LOCK INPUT (OR SWITCH)†

When the START LOCK is on, CYCLE START pushbutton does not function. Use the START LOCK input to prevent operating the machine in abnormal condition during automatic operation. The input may be used as on/off switch on control station for machine.

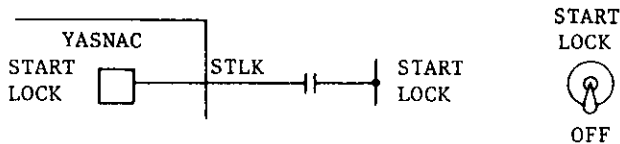


Fig. 5.18

5.1.28 EDIT LOCK SWITCH†

Turning on the EDIT LOCK switch prevents the function of ERS, INS, ALT, and EOB keys, and storing from NC tape. 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.

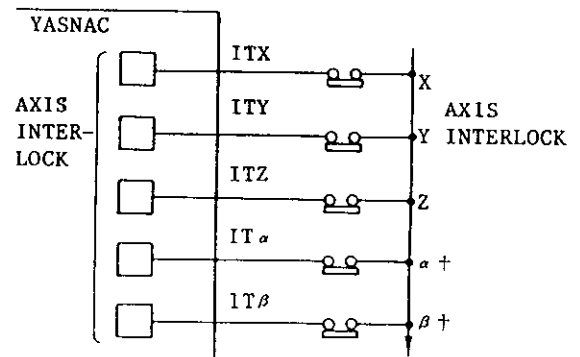


Fig. 5.19

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 (G94)

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.

5.1.30 EXTERNAL DECELERATION INPUT SIGNALS+ (Cont'd)

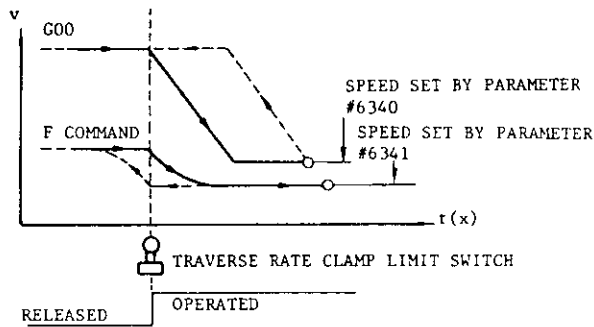


Fig. 5.20

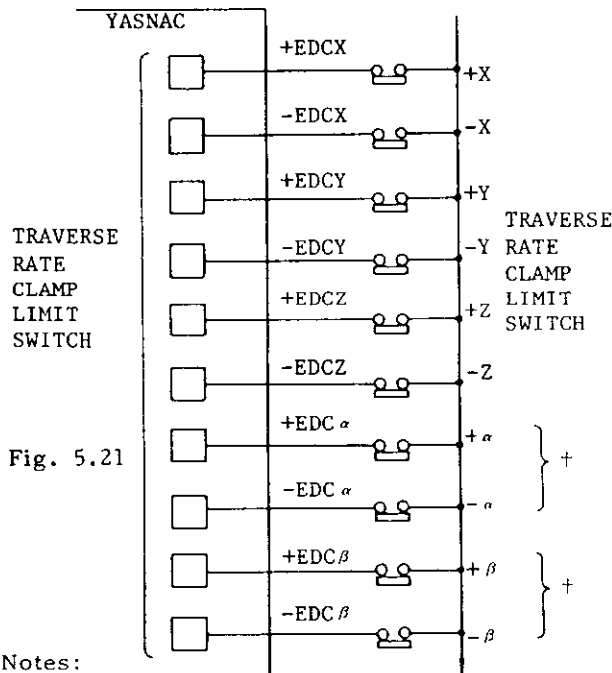


Fig. 5.21

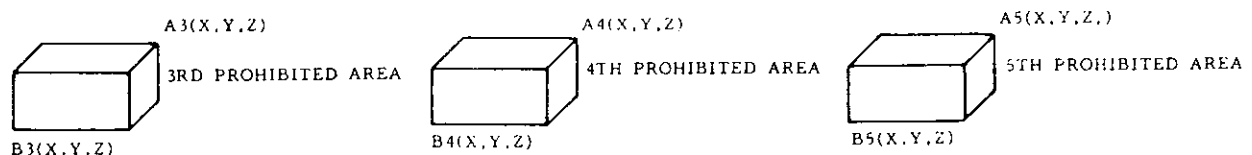
- Notes:
- The external deceleration function is ineffective on feedrate specified in mm per revolution of the spindle (mm/rev).
 - It is also ineffective on the HANDLE feed.

5.1.31 THIRD TO FIFTH STORED STROKE+

By adding the stored stroke limit function (G22, G23), it is possible to set the third, 4th and fifth prohibited areas simultaneously. However, selecting the ON/OFF state of this function must be by means of 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.

		X	Y	Z	
3rd	Point A3	#6650	#6651	#6652	Parameter
	Point B3	#6653	#6654	#6655	
4th	Point A4	#6656	#6657	#6658	
	Point B4	#6659	#6660	#6661	
5th	Point A5	#6662	#6663	#6664	
	Point B5	#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.

#1312		Meaning
D1	D0	
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."

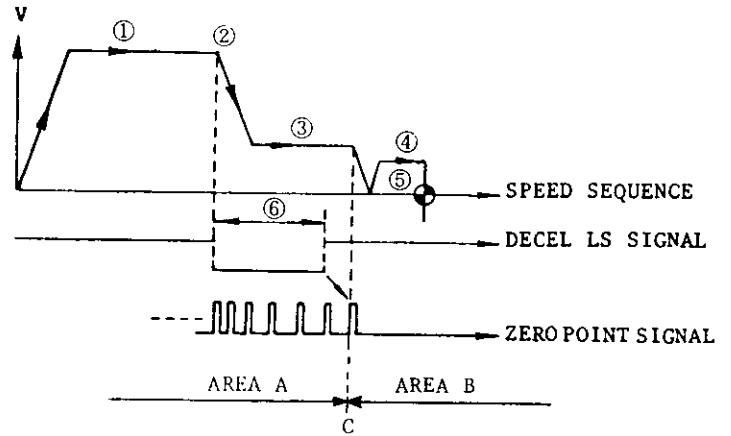
Note: Same as the notes for section 2.9.12 "STORED STROKE LIMIT" on page 34.

5.2 OPERATION PROCEDURE

5.2.1 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.



- | | |
|--|--------------------------------------|
| ① RATIO TRAVERSE RATE
(#6280-#6284) | ④ APPROACH SPEED 2
(#6316-#6320) |
| ② DECELERATION POINT | ⑤ TRAVERSE DISTANCE
(#6304-#6308) |
| ③ APPROACH SPEED 1
(#6310-#6314) | ⑥ DOG WIDTH |

Fig. 5.22

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.

5.2.1 MANUAL RETURN TO REFERENCE POINT (Cont'd)

- 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.

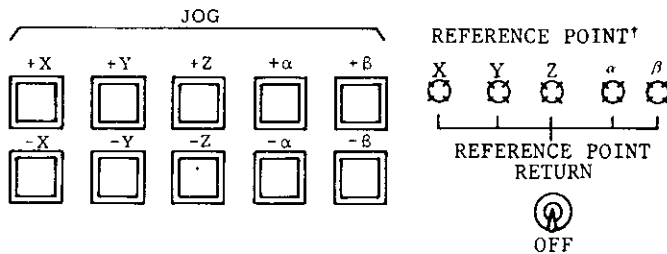


Fig. 5.23

5.2.1.1 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.

5.2.1.2 EASY REFERENCE POINT RETURN

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.

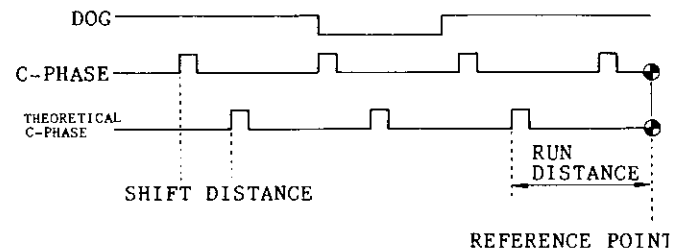


Fig. 5.24

(2) Related parameters

- #6010 D0 to D4: Reference point return direction of each axis
1: -/0: +
- #6016 D0 to D4: Reference point return of each axis
1: Enabled/0: Disabled
- #6304 to #6308: Run distance from the theoretical C-phase
Setting range: -32768 to 32767
- #6310 to #6314: Reference point return approach speed 1
Setting range: 0 to 24000
- #6316 to #6320: Reference point return approach speed 2
Setting range: 0 to 24000
- #6444 to #6448: Move distance per motor rotation
Setting range: 1 to 65535
- #6480 to #6484: 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 #6308) 0, theoretical C-phase shift distance (#6480 to #6484) 1000.

(ii) Perform reference point return.

(iii) If it stopped before the machine zero point, add the distance from that position to the machine zero point to the theoretical C-phase shift distance. If it stopped beyond the machine zero point, subtract the distance from that position to the machine zero point from the theoretical C-phase shift distance.

(iv) Fine adjustment for the true machine zero point can be made by the theoretical C-phase shift distance or by the run distance.

(b) When the machine zero point and the deceleration limit switch are at 1/2 the distance of the ball screw pitch.

(i) Adjust the machine zero point assuming that the above machine zero point (iii) is at 1/2 the ball screw pitch. Fine adjustment for the true machine zero 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 321 to 235 occur.

(b) If the setting of the theoretical C-phase shift distance (#6480 to #6484) and the run distance (#6304 to #6308) are not appropriate, pull-back occurs, to cause alarms 261 to 265. 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.

(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
#6640	5th coordinate

2. Parameters for inch input system

Parameter	Meaning
#6630	X coordinate
#6631	Y coordinate
#6632	Z coordinate
#6633	4th coordinate
#6634	5th coordinate


3. Axis can be selected by parameter #6015 for both metric and inch systems.

5.2.3 AUTOMATIC 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.

- a. 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.
 - b. Increase the correction number by 1, in preparation for the next writing.
 - c. Return the tool to the Z-axis home position.
1. Measuring method (parameter #6039D4 = "0")
- a. Mount a tool on the spindle, and move it to a Z-axis position which is to be set as the home position. Any position may be set as the home position, but for facilitating tool changing process, the tool changing position may be set up as the home position.

- b. Select the manual operation mode (RAPID, JOG, HANDLE or STEP) using MODE SELECT switch.


- c. Push the function key .

The offset number specified previously and related data are displayed.

- d. The page covering tool offset values specified the tool offset number keyed in will be shown. The specified number is shown by



- e. Push 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.

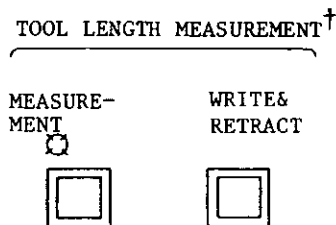


Fig. 5.24

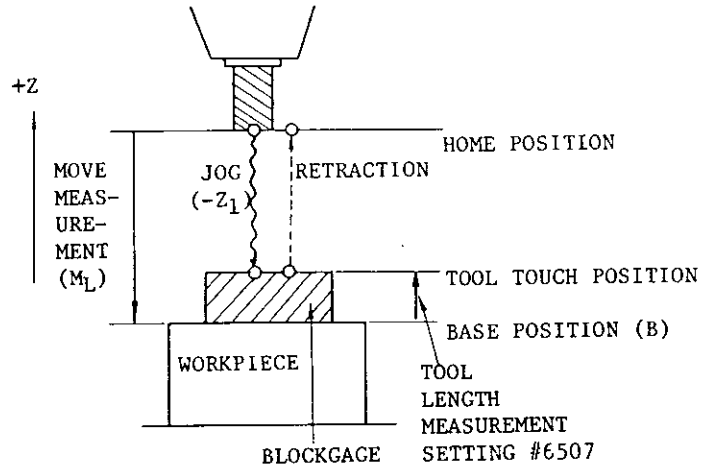



Fig. 5.25

Home position is the point where measurement starts. Measurement is made with the point temporarily determined as coordinate 0.

Note: To display this position as coordinate 0, 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 controlling Z-axis manually. Tool touch position is the position specified arbitrarily based on the base position (machine workpiece surface). The difference between tool touch position and base position is set in advance in setting #6507 ("1" = input unit). The difference is regarded as thickness of block gage. Write it in plus value.

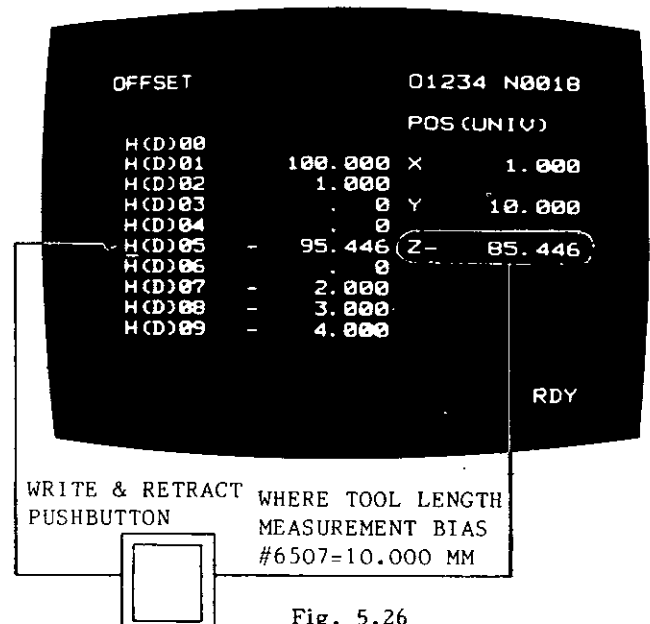


Fig. 5.26

The figure above shows the value in the offset number 05.

g. Depress the WRITE & RETRACT button
 ... The NC equipment executes the next operation

i. The distance from home position to base position, 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 home position 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 MDI 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. Push 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 home position is different from the tool change position, the new tool may be brought to the reference surface directly without first returning to the home position. Once the home position has been set up by the use of the MEASUREMENT key, repositioning to the home position is not necessary.

When the WRITE & RETRACT key is depressed with H100 designated, offset amount will not be changed. Execute TLM operation by moving the CURSOR AFTER H01.

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.

$$R_L = -(A - M_L)$$

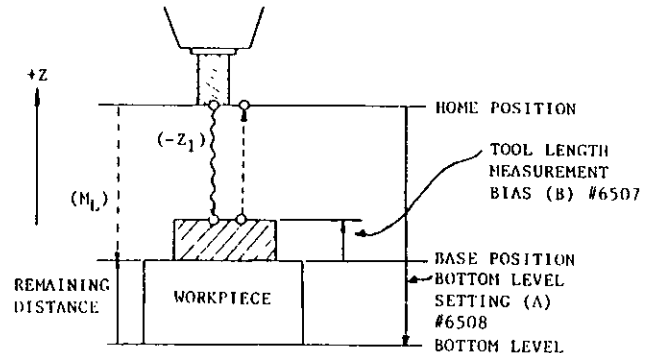


Fig. 5.27



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 [UNIVERSAL].

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,

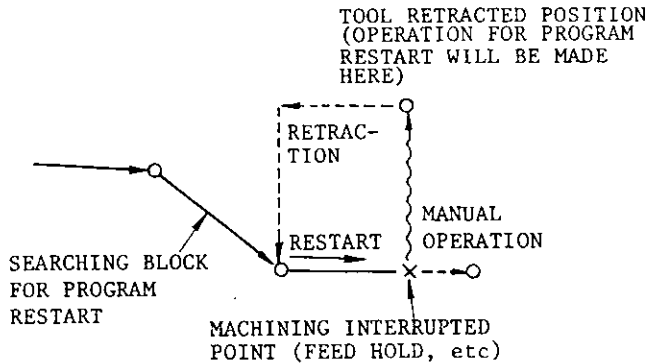
... This results in the automatic writing of the same tool offset measured as written in step 1.

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.

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.



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.

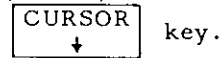
Operation Procedure

When the tool is broken during machining operation, the automatic operation may be restarted in the following procedure:

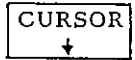
- (1) Press FEED HOLD button to stop the operation and make the tool escape.
- (2) Replace the broken tool and perform the associated chores. Change the tool offset, if required.
- (3) Turn on PROGRAM RESTART switch on the machine control station.
- (4) Press PROG function key to display the part program 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:

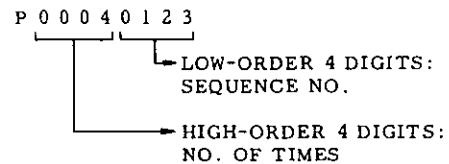
Set memory mode, key-in the program number (Oxxxx) to be restarted, and press



- (6) Key-in P, 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 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.

- (7) When the search is completed, the CRT screen automatically displays the program restart information.

PROGRAM RETURN	O0001 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-	258.774
Y	483.123 T 0015 0025
Z-	182.347 S 0500
A	73.442 B 0000
PRN RDY	

Fig. 5.28 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 MDI 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, 5th 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 D0

= 1: Prohibit execution of the M code macro

= 0: Allow execution of the M code macro #6073 D1

#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 pressed.

Operational 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.

(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

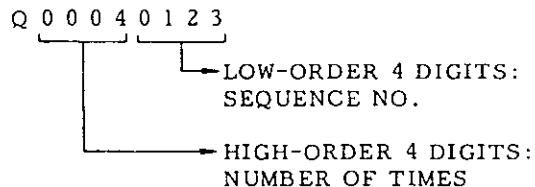
(Oxxxx) to be restarted, and depress

CURSOR
↓

key.

5.2.4.2 PROGRAM RESTART OF TYPE Q
(Cont'd)

(6) Key-in Q, sequence number, and CURSOR
↓
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 MDI 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, 5th 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, 5th 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 MDI 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.

d. A manual intervention was performed during the axis shift to the return position.

(8) In type P, any of the following operations performed in the time between the discontinuation of machining and the search operation for program restart will cause an error:

a. The setting of coordinate system was specified.

ERROR ... 121 PRN ERROR (G92)

b. The setting of work coordinate system was specified.

ERROR ... 122 PRTN ERROR (G54 - G59)

c. The coordinate system was modified by operation ORG key.

ERROR ... 123 PRTN ERROR (ORG)

In type P, the correct program restart is made possible only for the blocks that follow the one for which the coordinate-system setting was performed last before the discontinuation of machining.

(9) If a specified block has not been found, it will cause an error.

+ Error 120 PRTN ERROR (NOT FOUND)

(10) After turning power on, operation for program restart without any returning to reference point will cause an error.

+ Error — ZR UNREADY

(11) After search operation for program restart is completed, if shaft is transferred by MDI before moving shaft, it will cause an error.

+ Error 124 PRTN ERROR (MDIMOVE)

(12) In program restart operation during modifying tool dia, MDI cannot instruct M.S.T or B. In this case, use manual MDI operation (interruption into actual buffer during modifying codes).

(13) "PRN" blinks at the last line of CRT screen between the times when program restart search is instructed and when last axis-Z is completed to return.

(14) In DNC operation or tape operation program can not be restarted.

5.2.5 PLAYBACK FUNCTION+

The current axis position during movement can be stored as command value in the part program. This function is permitted in the PLAYBACK mode when PLAYBACK switch is turned on in manual operation mode.

To turn on PLAYBACK mode, activate the PLAYBACK switch on the machine control station or set #6000 D5 at "1." Procedure for storing program in the PLAYBACK mode is as follows.

(1) Set the control in the manual operation mode (RAPID, JOG, STEP, or HANDLE).

(2) Turn on the PLAYBACK switch.

(3) Depress the PROG key of function keys. The CRT screen shows the contents of part program number O plus 4 digits and the current value at the bottom.

(4) Register the new program number after address O. To register O1234, for example, key-in 0, 1, 2, 3, 4 and depress the WR key. The numbers are displayed as they are keyed-in. See Fig.

```

PROG(PLAYBACK)      O0001 N0010
O0001;
N0001 G92 X140 Y100 Z30;
N0002 G00 S1200 T0202 M03;
N0003 G41;
N0004 X0 Z500;
N0005 G01 Z0 F20;
N0006 X2000;
N0007 Z-2000;
N0008 Y2000;
N0009 X3000 Y4000;
  POS X1234, 567 Y2345, 678
  (UNIV) Z-2345, 789
                                LSK   RDY
  
```

Fig. 5.29

```

PROG(PLAYBACK)      O1234 N0010
O1234;
                                LSK   RDY
  
```

EDIT BUFFER REGISTER

Fig. 5.30

(5) Move the axis to the position which will be written as command.

(6) Depress the address key for the axis whose position is written.

(7) Depress the WR key.

The displayed current value is the command value of the selected address. Key-in the numerical data before depressing the WR key, and the total amount of keyed-in value plus current value will be the new command value.

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.)

5.2.5 PLAYBACK FUNCTION+ (Cont'd)

(9) Depress the EOB key.

(10) Depress the WSRT key.

A block of data is stored in the part program.

(11) Repeat steps (5) through (10) until the desired data is 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.

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. 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.

5. 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

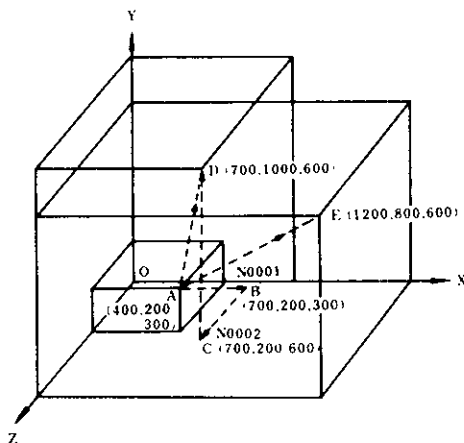


Fig. 5.31

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 G00 when machine was moved from point A to point B. Sequence No. for this data block is N0001.

(1) Key-in N, 0, 0, 0, 1.

Do not depress the WR key. If depressed, "INPUT ERROR" will appear on the screen.

(2) Key-in G, 0, 0.

(3) Depress WR key after X.

(4) Depress the EOB key.

(5) Depress the INSRT key.

Then, block of N0001 G00 X0.700; will be stored in the part program.

B. Procedure for storing current position data using G01, F02 when machine was moved from point B to point C. Sequence No. for this data block is N0002.

(1) Key-in N, 0, 0, 0, 2.

(2) Key-in G, 0, 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 G00 when machine was moved from point C to point D. Sequence No. for this data block is N0003.

(1) Key-in N, 0, 0, 0, 3.

(2) Key-in G, 0, 0.

(3) Depress the WR key after Y.

(4) Depress the EOB key.

(5) Depress the INSRT key.

Then, block of N0003 G00 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 N0004.

(1) Key-in N, 0, 0, 0, 4.

(2) Key-in G, 0, 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.

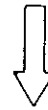
Then block of N0004 G01 X0.700 Y1.000 Z0.600 F100; will be stored in the part program.

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 N0005.

PROG (PLAYBACK)		O0010N0000	
O0010;			
G91 G28 X0 Y0 Z0;			
G92 X9 Y0 Z0;			
G90 G00 X0.4 Y0.2 Z0.3;			
N0001 G00 X0.700;			
N0002 G01 Z0.600 F20;			
N0003 G00 Y1.000;			
N0004 G01 X0.700 Y1.000 Z0.600 F100;			
POS	X	.700	Y1.000
(UNIV)	Z	.600	
N0005 G01X0.5			RDY

Fig. 5.32



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)		O0010N0000	
O0010;			
G91 G28 X0 Y0 Z0;			
G92 X0 Y0 Z0;			
G90 G00 X0.4 Y0.2 Z0.3;			
N0001 G00 X0.700;			
N0002 G01 Z0.600 F20;			
N0003 G00 Y1.000;			
N0004 G01 X0.700 Y1.000 Z0.600 F100;			
POS	X	700	Y1.000
(UNIV)	Z	600	
N0005 G01 X1.200			RDY

Fig. 5.33

- (1) Key-in N, 0, 0, 0, 5.
- (2) Key-in G, 0, 1.
- (3) Depress the WR key after keying-in X0.5.
- (4) Depress the WR key after keying-in Y-0.2
- (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 N0005 G01 X1.200 Y0.800 Z0.600 F0.600 F100; will be stored in the part program.

5.2.5.1 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

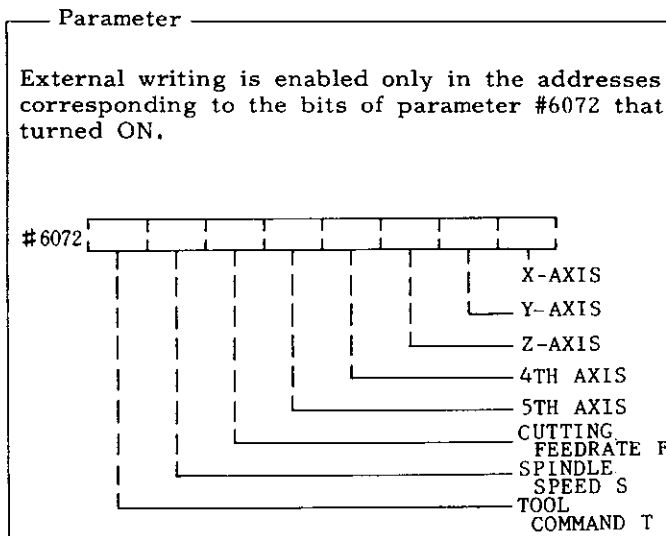
The operation procedure is as usual. Refer to the Operator's manual (Par. 5.2.5.1, Playback function").

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, overridden values of the current cutting feedrate command F or spindle speed S become the command value.

(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 canceled by the cancel key.

Note:

The following are the command values of F, S, and T.

- F: Current command F × Override (%)
- S: Current command S × 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 (P1) 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.

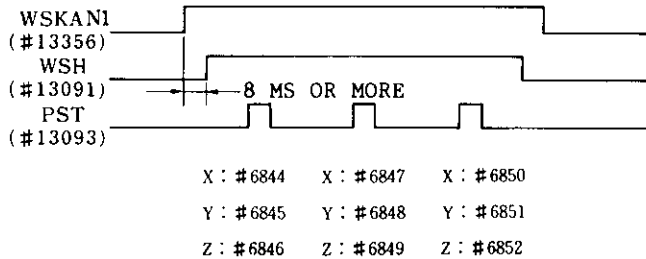
(d) Contact the tactile sensor at the second measurement point (P2).

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

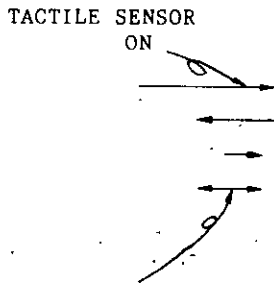
- (a) POS [MACHINE] is used for the manual skip measurement. Do not forget the reference point return before the measurement.
- (b) If writing is done without completing the reference point return, alarm occurs when the PST signal turns on. (ZR UNREADY (X)).
- (c) The WSKAN1, WSH signals selected before selecting the manual mode are invalid.

It enters the manual skip mode ( LED flashes)

only when the WSKAN1, WSH signals are selected after selecting the manual mode. The PST signal is also effective only when activated (turned on from off) under the writing mode.

(d) Always touch (SKIP signal ON) the datum plane with the same axis.

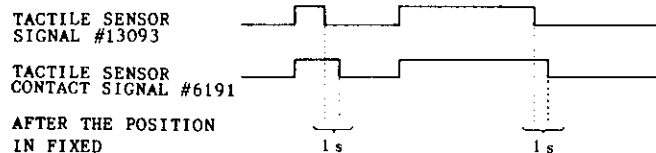
(e) When the tactile sensor turns on, it must be pulled back for a certain amount (parameter #6578), or it cannot be moved in the same direction again.



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.



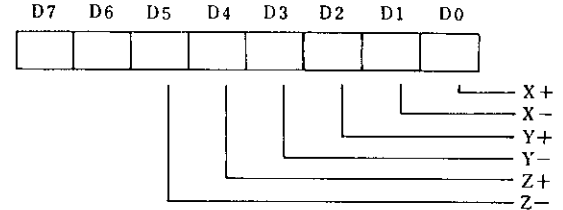
When the tactile sensor signal turns on, the tactile sensor contact signal turns on after the position is fixed, and when the tactile sensor signal turns off, the tactile sensor contact signal turns off after 1 second.

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) I/O

(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.

5.2.7 FEEDRATE, SPINDLE SPEED EDITING FUNCTION (Cont'd)

- (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.
 - (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.

(EXAMPLE)

(Before FS editing)

```

0 1 2 3 ;
G28 X10. Y10. Z10. ;
G92 X100. Y100. Z100. ;
G00 G90 F100. ;
M03 S3000 ;
G01 X... ;
    Y... ; → FSMEM Closes when the override reaches 80%
    Z... ;   and S override 110%.
    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... ;
G00 X... Y... Z... ;
M30 ;

```

(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.

(After FS editing)

```

0 1 2 3 ;
G28 X10. Y10. Z10. ;
G92 X100. Y100. Z100. ;
G00 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... ;
G00 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

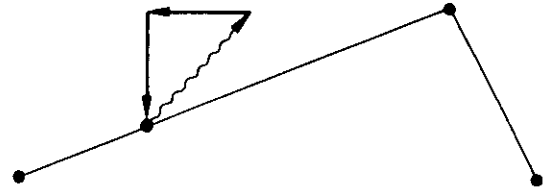
- (a) When the FSMEM is closed two or more times in a single block, the later feedrate becomes effective. However, the FSMEM count is not incremented.
- (b) When the FSMEM is closed in the subprogram, the FS data are inserted in the subprogram itself. This needs care, as the subprogram may also be called 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 times or more, or just
	128 times
Warning #6181	DO 128 or 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).

(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 "0".

- (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.

X 1 2 3 . 4 5 6

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 36578. 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.

(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 store 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 MTIII 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 feed back pulse per spindle rotation must be 4096.
- (e) The solid tap function option must be effective

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, always 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

#6199 Spindle side gears
#6198 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 × #6449) per spindle rotation become 4096 pulses.

(5) Spindle revolutions against command voltage 10 V in the spindle indexing execution gear

#6472 Setting unit: 1 = 1 r/min
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 command

#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 command

#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 = r/min
Setting range: 0 to 30
Standard setting value: 30

(13) Spindle revolutions upper limit for solid tap

#6471 Setting unit: 1 = 1 r/min
Setting range: 1 to 32767
Standard setting value: 4500

5.2.12.3 I/O SIGNALS

The I/O 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 D0
#1342 D3 to D0

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.

5.2.12.3 I/O SIGNALS (Cont'd)

(2) Spindle indexing input

#1343 D0 1: Spindle indexing request ON
0: 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 D1 1: Spindle indexing incremental request ON
0: 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
0: Spindle indexing restart request OFF

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.

(5) Spindle indexing execution output

#1207 D2 1: Spindle indexing execution
0: Not spindle indexing execution

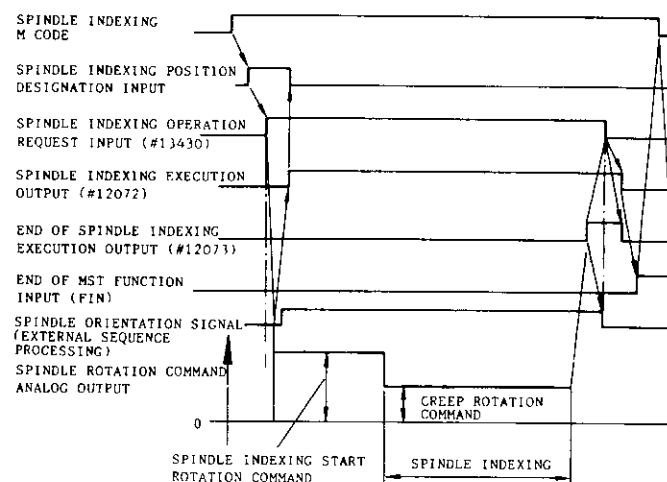
(6) End of spindle indexing execution output

#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 "X10000" is regarded as "X100."
4. Rotate HANDLE dial.
Turning the dial clockwise:
The machine moves in the positive direction.
Turning the dial counterclockwise:
The machine moves in the negative direction.

6.4 PREPARATION FOR STORED LEADSCREW ERROR COMPENSATION AND STORED STROKE LIMIT †

Return to Reference Point

With an NC equipped with the stored leadscrew 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 6.2.1)
- b. Execute G91 G28 X0 Y0 Z0 ; 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 - 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.

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 6.2.1 MANUAL RETURN TO REFERENCE POINT.)
 - The G92 command according to the part program should be executed by MDI.

G92 X... Y... Z... ;

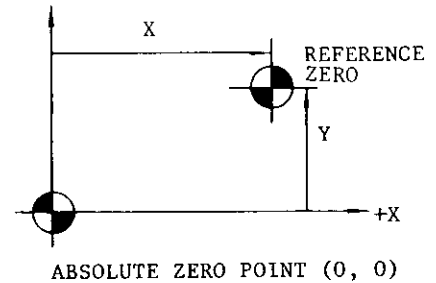


Fig. 6.1

If "G92 X0 Y0 Z0 ;" 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 G00 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 zero, return the machine to the reference point by manual return to reference point to reference zero.

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, MDI or MEM).
6. Depress CYCLE START pushbutton, and the machine will resume the automatic operation.

Notes :

- Where MODE SELECT switch is changed without depressing FEED HOLD pushbutton.
 - a. When the automatic-mode (TAPE, MDI or MEM) is changed to the manual-mode (HANDLE, JOG or RAPID), the machine rapidly slows down and stops.
 - b. When the automatic-modes are changed the machine is stopped at the block end.
 - Where the machine is restarted by depressing CYCLE START button, the tool path shifted due to manual operation will be changed by ON-OFF operation of MANUAL ABSOLUTE switch. Refer to 6.1.24, "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 MDI 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, M00, 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 on page 134.
3. Depress CYCLE START button, and automatic operation can be executed in MDI 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 blocks of data may be read in advance.
2. Display the data on CRT DISPLAY according to 4.3.2 COMMAND DATA DISPLAY, and check it on page 132.
3. Set MODE SELECT switch to MDI operation.
4. Write the data referring to 4.3.3 Writing in Blocks and Displaying Contents by MDI on page 134.
Execute the data by depressing the CYCLE START button.
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 — four blocks are read ahead.
- 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 MDI operation, the command becomes effective and is executed as soon as written. This procedure is used to add a new data to an active buffer. However, M00, M01, M02, M30 and M90 to M99 cannot be written.

- 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.
- Tool radius compensation and canned cycle modes are not 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" on page 142.
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.

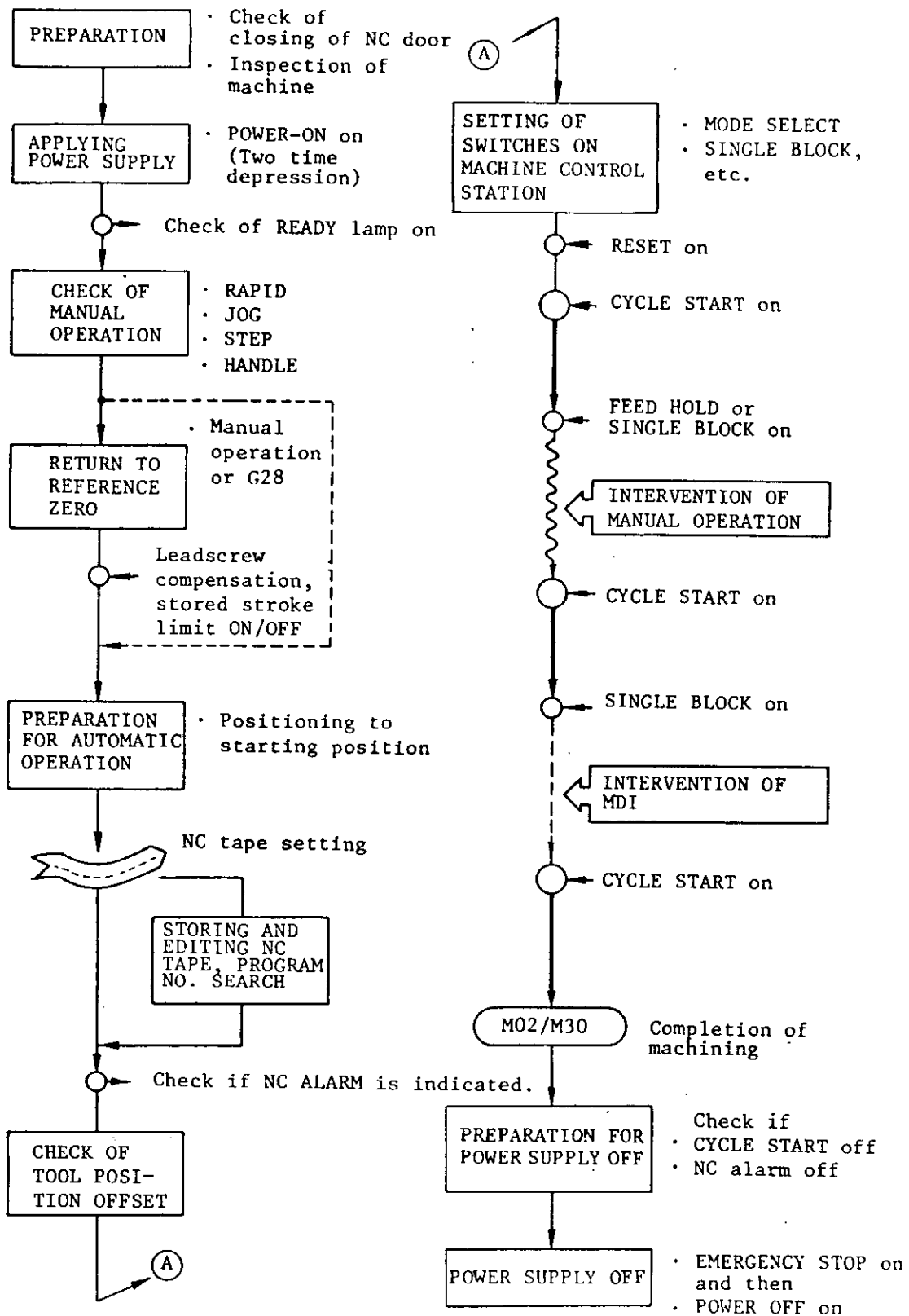


Fig. 6.4 Operating Procedure

7. MAINTENANCE

7.1 ROUTINE INSPECTION SCHEDULE

The following table shows the minimum require-

ments to be observed for maintenance according to time in order to keep the equipment in optimum condition for extended period.

Table 7.1.0 Inspection Schedule

Items		Frequency	With the system-off	With the system-on	Remarks
Tape reader	Cleaning of reading head	Daily	<input type="radio"/>		Including light source part.
	Cleaning of tape tumble box	Weekly	<input type="radio"/>		
	Lubricating of tension arm shaft end	As required	<input type="radio"/>		
Control panel	Tight closing the doors	Daily	<input type="radio"/>		
	Checking for loose fit and gaps of side plates and worn door gaskets	Monthly	<input type="radio"/>		
Servomotor and DC motor for spindle	Vibration and noise	Daily		<input type="radio"/>	Feel by hand, and do the audible inspection.
	Motor contamination and breakage	Daily or as required	<input type="radio"/>	<input type="radio"/>	Inspect visually.
	Clearance of ventilation openings		<input type="radio"/>	<input type="radio"/>	Inspect mainly spindle DC motor.
	Burned spots, cracks, wear, and pressure of brushes	Every three months	<input type="radio"/>		Check the length of brushes.
	Roughened commutator surface		<input type="radio"/>		Check dark bar, threading and grooving of commutator.
	Dirt in interior of motor		<input type="radio"/>		Clean with compressed air.
Battery		Daily	<input type="radio"/>	<input type="radio"/>	See if alarm for BATTERY is displayed on CRT screen.

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.

Note: When trouble occurs in feeding or winding tape with 8 inch diameter reels, open the front door and brush away dust around the photo-coupler by using a blower brush.

7.1.1 CONTROL PANEL

The control panel is a dustproof, sheet-steel enclosure with gasketed doors.

- A. Front and rear doors of the control should be shut tightly, even if the control is not operating.
- B. When inspecting the control with the door open, upon completion of inspection, lock the door by turning two door locks with the key attached to the control panel. Turning direction of door locks is as follows.

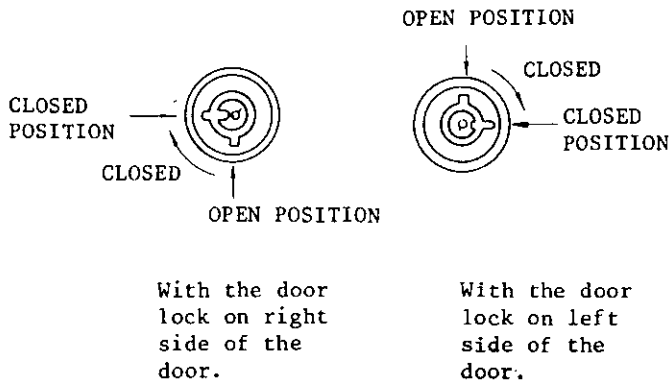


Fig. 7.1

Note: If the optional door interlocking switch is provided, opening the door shuts off the main power supply and stops all operations.

- C. Check gaskets on the rims of front and rear doors.
- D. See if the inside of enclosure is dusty. Clean it, if necessary.
- E. Check for any opening in the door base with the doors shut tightly.

7.1.2 SERVOMOTOR AND DC MOTOR FOR SPINDLE

- 1. Vibration and noise.

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.

Check the motor exterior visually. If dirt or damage should be observed, inspect the motor by removing the machine cover. Refer to the machine tool builder's manual.

- 3. Clearance of ventilation window blockage

Check the ventilation window of DC spindle motor. If it is clogged with dust or dirt, inspect DC spindle motor removing the machine cover. Refer to the machine tool builder's manual.

Inspection of commutators and brushes is essential for maintaining the excellent performance of the control. Inspection work to be executed is described in the following three items.

Quarterly Inspection of Commutators and Brushes

The carbon dust from brushes, accumulated around the commutator, inside the motor, may cause motor troubles such as the layer short of armature and the flashover of commutator. In the worst case, it may lead to fatal damage. To avoid this, be sure to have an inspection on the commutators and brushes at least every three months.

Be sure to turn off the power supply to the NC before starting to check the brushes and motor interior.

For this, turning the circuit breaker on the power supply unit (DCP UNIT) for the servo control unit (CPCR-MR-K) off is not sufficient. To prevent electric shocks and shorting, be sure to shut off the supply of power to the NC.

- 4. Carbon brushes

- A. Under normal operating conditions, brush wears by 2 to 4 mm per 1000 operating hours. If wear is excessive, check to see if oil has contaminated commutator surface, or if abnormal overcurrent flow through motor circuit.
- B. When brush length becomes shorter than those shown below, replace the brush with a new one.

Minertia motor junior series: 6 mm or below

DC motor for spindle: 17 mm or below

7.1.2 SERVOMOTOR AND DC MOTOR FOR SPINDLE (Cont'd)

- C. If either of brush, or pigtail is broken, brush assembly must be replaced as a whole unit.

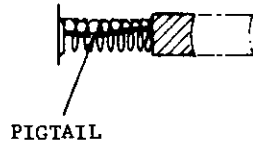


Fig. 7.2

Note: When replacing the brush assembly, consult the company.

5. Commutator surface

- A. Visually check surface roughness of the commutator through inspection window.
- After 100 to 200 operating hours, the commutator should take on a polished light brown or chocolate color. The motor has developed an ideal commutator film and needs no attention other than to be kept clean.

- B. See if a blackened bar, threading (or grooving) is on the commutator. If any of the above is observed, investigate the cause of trouble.

Threading or grooving on the commutator surface may be due to too small motor load. Blackened bar is a result of carbon dust in commutator slots, or accidentally produced sparkings. If the carbon dust is a cause of blackened bar, wipe the commutator with a clean dry cloth to smooth the surface. If sparking occurs, contact the maintenance representative.

6. Motor inside (dirty)

- A. Visually check the motor interior through inspection window.
- The dried carbon dust will not affect motor running, but it is recommended that the inner parts such as commutator, brush-holders and brushes be cleaned with a dry compressed air (air pressure: 2-4 kg/cm²).
- B. Where oily carbon dust exists inside the motor due to poor oil seal or defective enclosure, contact Yaskawa.

7. Servomotor with oil seal

As the life expectancy of oil seal and brush is 5000 hours (about five years), the inspection and maintenance by the company should be done every 5 years. If possible, yearly inspection taking less than 8 hours is recommended.

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.

The control with a bubble memory board (optional) does not require a battery.

7.2 BATTERY REPLACEMENT

The battery is used as power source for memory in order to prevent programming data stored in memory, such as parameter, tool offset and part program from erasing.

When the battery is 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.

Replacing Procedure

1. Depress POWER OFF pushbutton on the operator's station.
2. Open the front door of the control. The battery of the memory(printed circuit) board can be seen on the CPU module which is mounted on rear of the front door.
3. 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.
4. Check to see if 1LED on memory board is illuminated. Fig. 8.2.1 shows the arrangement of LED and the battery. If illuminated, replace the battery.

(5) Verify if the LED/1 on the main board (MB20) lights up or not. If the LED/1 lights up, the replacement of battery "BR-2/3A" is necessary. Fig. 7.2.1 shows the arrangement of LEDs and batteries. The above is the procedure for checking defective batteries.

"BR-2/3A" YASKAWA type: JZNC-GBA01

(6) While the power supply is on, disconnect the old battery from connector and remove it from the battery holder.

(7) Insert new battery into the battery holder and then plug it in to the connector.

NOTE: The connector may be inserted even upside down.

(8) Turn the power off.

(9) Where there is a door interlock switch, depress the moving part. Then, the switch will return to its original position. When the door is "open", the power cannot be turned on.

(10) Close the front door positively.

(11) Turn the power on.

(12) Confirm that the battery alarm display on the CRT screen of NC operation panel has 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.

7.3 POWER SUPPLY

Various kinds of protective functions are provided for compound power supply (CPS-10N). 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, ±12 V" LEDs (RED) LIT

(1) The red LED lights up when an overcurrent due to the short-circuit of output from +5 V or ±12 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.

7.4 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) 335(5)
Overload	351(X) 352(Y) 353(Z) 354(4) 355(5)
Servo alarm	390 for all axes

7.4.1 CIRCUIT PROTECTOR TRIP (ALARM NO. 331 TO 335)

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

335: FUSE(5) ... for 5th axis

When a circuit protector is tripped, and the alarm No. 331 to 335 is displayed, do not attempt to take measures, but the user should immediately notify our service department.

7.4.2 OVERLOAD (ALARM NO. 351 to 355)

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 abnormally 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
- 355: OL(5) ... 5th 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 our service department.

7.4.3 SERVO ALARM (ALARM NO. 391 TO 395)

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 7.2 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.

8-Segment LEDs (red)

Display	Indication (When Lit)	Relation With Output Signals
.	Base breaking state	· Base of power circuit in Servopack is interrupted. · Self-hold is made until reset.
-	(No current through motor)	
1.	Overcurrent detected	
2.	Circuit protector trip detected	
3.	Abnormal regeneration detected	
4.	Overvoltage detected	
5.	Overspeed detected	
6.	Undervoltage detected	
7.	Overload detected	
A.	Heat sink overheat detected	
b.	A/D error detected	
F.	Open phase detected	
C.	Runaway prevention activation detected	
	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 MCCBs which can be turned on and off externally.

Generally, when these MCCBs 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 DIAGNOSTICS

On-line 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 5, "LIST OF ALARM CODES" on the last part of this manual.

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 MX3. Custom-built signals are provided for optional machine interface equipped with some type of YASNAC controls.

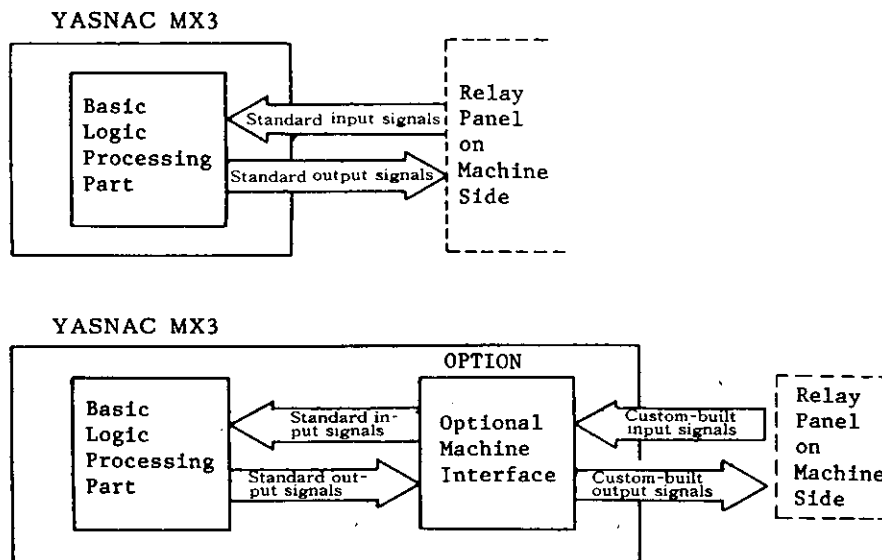


Fig. 7.3

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

349: PC CPU ERROR....This occurs in the event of a PC CPU error.

371: FG-SV SYNC ERROR....This occurs when there is no synchronization between SERVO CPU and FG.

372: FG NOT READY....This occurs when READY from FG CPU has not been received.

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.

385: AXIS ERROR (5)....This occurs when READY from 5th axis has not been received.

(2) Alarms that occur when the hardware related to optional items is changed:

347: UNFINISHED PROM GEN....This occurs if no initialization has been made after changing the memory length of part program.

384: 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.

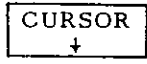
1. Depress the DGN key.

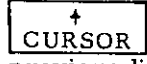


A page containing the diagnostic number specified previously occurs on the CRT screen. The input/output signals are shown in "1," "0" and hexadecimal digit.

"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 next diagnostic number (line). When down to the last lower line, the next page is displayed on 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.

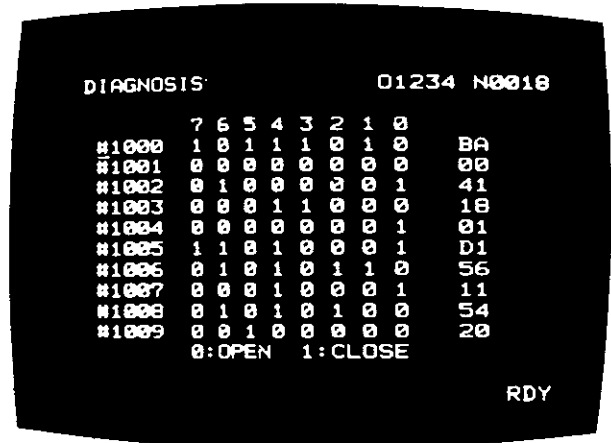


Fig. 7.4 Example of Input/Output Signal Display

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 correct 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 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 1.1

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

Appendix table 1-2 lists setting numbers and their functions.

SETTING NUMBERS AND THEIR CONTENTS

#6000	D7		D5	D4	D3	D2	D1	D0
-------	----	--	----	----	----	----	----	----

- D7 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
- D4, D3, D2, D1, D0
Selects whether to make the mirror image of the 5 axis, 4 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 cancelling switch
1: ON
0: OFF
- D3 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: Changes input units to inch inputs
0: Changes input units to MM 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	D7	D6	D5	D4	D3		D1	D0
-------	----	----	----	----	----	--	----	----

- D7 Tape operation device designation
0: PTR
1: Communication module
- D6 Communication module device designation
0: RS-232C
1: RS-422
- D5, D4: Selects the output device of the data input interface
- D3 When outputting EOB in ISO code;
0: Output with LF.
1: Output with CRLF.
- D1, D0: Selects the input device of the data input interface

SETTING NUMBERS AND THEIR CONTENTS
(Cont'd)

Setting Code	Input Device	Output Device	Parameter for Setting Baud Rate
0 0 0 1	RS232C No.1	RS232C	#6026 (#6029) D3 to D0
1 0 1 1	RS232C No.2	RS232C No.2	#6027 (#6028) D3 to D0

#6004	D7	D6	D5	D4	D3		D1	D0
-------	----	----	----	----	----	--	----	----

- D7 The group G02 G code upon reset is;
0: G17.
1: The G code commanded immediately before is retained.
- D6 1: Cancels life data when executing command G122
0: Does not cancel life data when executing command G122
- D5 The group G03 G code upon reset;
0: Follows PM6005 D0.
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.
- 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.
0: Causes alarm "087"

#6200 Break point -1

#6201 Break point -2

Sets break point sequence.
Setting range 1 through 9999. Will not stop at 0.

#6204

Specifies group number during tool change and reset.
Setting range 1 through 128.

#6207

Sets program number.
Specifies program number of tape when inputting tape without a program number.
Setting range 1 through 9999.

Note: If 0 is set, input error may cause except when the number on the upper right of CRT is "0."

#6210

Sets dwell time when executing G76 and G77.
Setting "1" = 1 ms
Setting range: 0 - 32767

#6211

Sets amount of S when executing G73,
Setting "1" = 0.001 mm
"1" = 0.0001 inch
Setting range: 0 - 32767

#6212

Dwell time setting for G83
Setting: "1" = 1 ms
Setting range: 0 to 32767

#6213

Sets shift amount "δ" when executing G83.
Setting: "1" = 0.001 mm
or "1" = 0.0001 inch
Setting range: 0 - 32767

#6219

System No. switch setting
Setting range: 0, 1, 4

#6500

#6501

Sets scaling multiple. Multiple = #6500/
#6501
Scale multiple = 1 when setting is "0."

#6506

Sets angle during execution of commands G76 and G77.
Setting: "1" = 0.001 deg

#6507

Sets tool length measuring bias.
Setting: "1" = Least input increment

#6508

Sets tool length measuring bottom level.
Setting: "1" = Least input increment

#6510	X-axis
#6511	Y-axis
#6512	Z-axis

Sets stored stroke limit of the X, Y and Z axes and the boundary value in the positive direction of the 2nd prohibited area.

Setting range: 0 - ±99999999
Setting: "1" = 1 pulse

#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 range: 0 - ±99999999
Setting: "1" = 1 pulse

#6516	X-axis
#6517	Y-axis
#6518	Z-axis
#6519	4th axis
#6520	5th axis

Work coordinate system setting G54 (J1)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6521	
-------	--

Rotary angle setting G54 (J1)
Setting: "1" = 0.001 deg

#6522	X-axis
#6523	Y-axis
#6524	Z-axis
#6525	4th axis
#6526	5th axis

Work coordinate system setting G55 (J1)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6527	
-------	--

Rotary angle setting G55 (J1)
Setting: "1" = 0.001 deg

#6528	X-axis
#6529	Y-axis
#6530	Z-axis
#6531	4th axis
#6532	5th axis

Work coordinate system setting G56 (J1)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6533	
-------	--

Rotary angle setting G56 (J1)
Setting: "1" = 0.001 deg

#6534	X-axis
#6535	Y-axis
#6536	Z-axis
#6537	4th axis
#6538	5th axis

Work coordinate system setting G57 (J1)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6539	
-------	--

Rotary angle setting G57 (J1)
Setting: "1" = 0.001 deg

#6540	X-axis
#6541	Y-axis
#6542	Z-axis
#6543	4th axis
#6544	5th axis

Work coordinate system setting G58 (J1)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6545	
-------	--

Rotary angle setting G58 (J1)
Setting: "1" = 0.001 deg

SETTING NUMBERS AND THEIR CONTENTS
(Cont'd)

#6546	X-axis
#6547	Y-axis
#6548	Z-axis
#6549	4th axis
#6550	5th axis

Work coordinate system setting G59 (J1)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6551	
-------	--

Rotary angle setting G59 (J1)
Setting: "1" = 0.001 deg

#6552	X axis
#6553	Y axis
#6554	Z axis
#6555	4th axis
#6556	5th axis

Coordinate value at skip signal ON
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6561	F1
#6562	F2
#6563	F3
#6564	F4
#6565	F5
#6566	F6
#6567	F7
#6568	F8
#6569	F9

F1-digit speed setting
Setting range: 0 - 240000 "1" = 0.1 mm/min
Setting: "1" = Least input increment

#6570	
-------	--

Tool setter measurement longitudinal direction
reference value
Setting: "1" = input unit
Setting range: ±99999999

#6571	
-------	--

#6572	
-------	--

Tool setter measurement diameter direction
reference value
Setting: "1" = input unit
Setting range: ±99999999

#6573	X-axis
-------	--------

Workpiece setter measuring probe radius
Setting: "1" = input unit
Setting range: +99999999

#6574	X-axis
-------	--------

Workpiece setter measuring probe radius
Setting: "1" = input unit
Setting range: -99999999

#6575	Y-axis
-------	--------

Workpiece setter measuring probe radius
Setting: "1" = input unit
Setting range: +99999999

#6576	Y-axis
-------	--------

Workpiece setter measuring probe radius
Setting: "1" = input unit
Setting range: -99999999

#6577	Z-axis
-------	--------

Workpiece setter measuring probe radius
Setting: "1" = input unit
Setting range: -99999999

#6578	
-------	--

Workpiece setter measurable return amount
Setting: "1" = input unit
Setting range: 99999999

#6579	
-------	--

Workpiece setter measuring point pull away
amount

#6580	
#6581	
#6582	
#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 SPECIFICATION B

#6700	X-axis
#6701	Y-axis
#6702	Z-axis

Work coordinate system setting G54 (J2)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6703	X-axis
#6704	Y-axis
#6705	Z-axis

Work coordinate system setting G55 (J2)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6706	X-axis
#6707	Y-axis
#6708	Z-axis

Work coordinate system setting G56 (J2)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6709	X-axis
#6710	Y-axis
#6711	Z-axis

Work coordinate system setting G57 (J2)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6712	X-axis
#6713	Y-axis
#6714	Z-axis

Work coordinate system setting G58 (J2)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

**WORK COORDINATE SYSTEM SETTING
SPECIFICATION B (Cont'd)**

#6715 X-axis

#6716 Y-axis

#6717 Z-axis

Work coordinate system setting G59 (J2)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6718 X-axis

#6719 Y-axis

#6720 Z-axis

Work coordinate system setting G54 (J3)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6721 X-axis

#6722 Y-axis

#6723 Z-axis

Work coordinate system setting G55 (J3)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6724 X-axis

#6725 Y-axis

#6726 Z-axis

Work coordinate system setting G56 (J3)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6727 X-axis

#6728 Y-axis

#6729 Z-axis

Work coordinate system setting G57 (J3)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6730 X-axis

#6731 Y-axis

#6732 Z-axis

Work coordinate system setting G58 (J3)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6733 X-axis

#6734 Y-axis

#6735 Z-axis

Work coordinate system setting G59 (J3)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6736 X-axis

#6737 Y-axis

#6738 Z-axis

Work coordinate system setting G54 (J4)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6739 X-axis

#6740 Y-axis

#6741 Z-axis

Work coordinate system setting G55 (J4)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6742 X-axis

#6743 Y-axis

#6744 Z-axis

Work coordinate system setting G56 (J4)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6745 X-axis

#6746 Y-axis

#6747 Z-axis

Work coordinate system setting G57 (J4)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6748 X-axis

#6749 Y-axis

#6750 Z-axis

Work coordinate system setting G58 (J4)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6751 X-axis

#6752 Y-axis

#6753 Z-axis

Work coordinate system setting G59 (J4)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6754 X-axis

#6755 Y-axis

#6756 Z-axis

Work coordinate system setting G54 (J5)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6757 X-axis

#6758 Y-axis

#6759 Z-axis

Work coordinate system setting G55 (J5)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6760 X-axis

#6761 Y-axis

#6762 Z-axis

Work coordinate system setting G56 (J5)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6763 X-axis

#6764 Y-axis

#6765 Z-axis

Work coordinate system setting G57 (J5)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6766 X-axis

#6767 Y-axis

#6768 Z-axis

Work coordinate system setting G58 (J5)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

#6769 X-axis

#6770 Y-axis

#6771 Z-axis

Work coordinate system setting G59 (J5)
Setting range: 0 - ±99999999
Setting: "1" = Least input increment

Note: #6772 to #6899 is unused area. Do not use this area.

WORK COORDINATE SYSTEM SETTING
SPECIFICATION C

#6700	X-axis
#6701	Y-axis
#6702	Z-axis
#6703	4th axis
#6704	5th axis

Work coordinate system setting G54 (J2)
Setting: "1" = Least input increment
Setting range: ±99999999

#6705	
Rotary angle setting G54 (J2) Setting: "1" = 0.001 deg	
#6706	X-axis
#6707	Y-axis
#6708	Z-axis
#6709	4th axis
#6710	5th axis

Work coordinate system setting G55 (J2)
Setting: "1" = Least input increment
Setting range: ±99999999

#6711	
Rotary angle setting G55 (J2) Setting: "1" = 0.001 deg	
#6712	X-axis
#6713	Y-axis
#6714	Z-axis
#6715	4th axis
#6716	5th axis

Work coordinate system setting G56 (J2)
Setting: "1" = Least input increment
Setting range: ±99999999

#6717	
Rotary angle setting G56 (J2) Setting: "1" = 0.001 deg	
#6718	X-axis
#6719	Y-axis
#6720	Z-axis
#6721	4th axis
#6722	5th axis

Work coordinate system setting G57 (J2)
Setting: "1" = Least input increment
Setting range: ±99999999

#6723	
Rotary angle setting G57 (J2) Setting: "1" = 0.001 deg	
#6724	X-axis
#6725	Y-axis
#6726	Z-axis
#6727	4th axis
#6728	5th axis

Work coordinate system setting G58 (J2)
Setting: "1" = Least input increment
Setting range: ±99999999

#6729	
Rotary angle setting G58 (J2) Setting: "1" = 0.001 deg	
#6730	X-axis
#6731	Y-axis
#6732	Z-axis
#6733	4th axis
#6734	5th axis

Work coordinate system setting G59 (J2)
Setting: "1" = Least input increment
Setting range: ±99999999

#6735

Rotary angle setting G59 (J2)
Setting: "1" = 0.001 deg

#6736 X-axis

#6737 Y-axis

#6738 Z-axis

#6739 4th axis

#6740 5th axis

Work coordinate system setting G54 (J3)
Setting: "1" = Least input increment
Setting range: ±99999999

#6741

Rotary angle setting G54 (J3)
Setting: "1" = 0.001 deg

#6742 X-axis

#6743 Y-axis

#6744 Z-axis

#6745 4th axis

#6746 5th axis

Work coordinate system setting G55 (J3)
Setting: "1" = Least input increment
Setting range: ±99999999

#6747

Rotary angle setting G55 (J3)
Setting: "1" = 0.001 deg

#6748 X-axis

#6749 Y-axis

#6750 Z-axis

#6751 4th axis

#6752 5th axis

Work coordinate system setting G56 (J3)
Setting: "1" = Least input increment
Setting range: ±99999999

#6753

Rotary angle setting G56 (J3)
Setting: "1" = 0.001 deg

#6754 X-axis

#6755 Y-axis

#6756 Z-axis

#6757 4th axis

#6758 5th axis

Work coordinate system setting G57 (J3)
Setting: "1" = Least input increment
Setting range: ±99999999

#6759

Rotary angle setting G57 (J3)
Setting: "1" = 0.001 deg

#6760 X-axis

#6761 Y-axis

#6762 Z-axis

#6763 4th axis

#6764 5th axis

Work coordinate system setting G58 (J3)
Setting: "1" = Least input increment
Setting range: ±99999999

#6765

Rotary angle setting G58 (J3)
Setting: "1" = 0.001 deg

#6766 X-axis

#6767 Y-axis

#6768 Z-axis

#6769 4th axis

#6770 5th axis

Work coordinate system setting G59 (J3)
Setting: "1" = Least input increment
Setting range: ±99999999

**WORK COORDINATE SYSTEM SETTING
SPECIFICATION C (Cont'd)**

#6771

Rotary angle setting G59 (J3)
Setting: "1" = 0.001 deg

#6772 X-axis

#6773 Y-axis

#6774 Z-axis

#6775 4th axis

#6776 5th axis

Work coordinate system setting G54 (J4)
Setting: "1" = Least input increment
Setting range: ±99999999

#6777

Rotary angle setting G54 (J4)
Setting: "1" = 0.001 deg

#6778 X-axis

#6779 Y-axis

#6780 Z-axis

#6781 4th axis

#6782 5th axis

Work coordinate system setting G55 (J4)
Setting: "1" = Least input increment
Setting range: ±99999999

#6783

Rotary angle setting G55 (J4)
Setting: "1" = 0.001 deg

#6784 X-axis

#6785 Y-axis

#6786 Z-axis

#6787 4th axis

#6788 5th axis

Work coordinate system setting G56 (J4)
Setting: "1" = Least input increment
Setting range: ±99999999

#6789

Rotary angle setting G56 (J4)
Setting: "1" = 0.001 deg

#6790 X-axis

#6791 Y-axis

#6792 Z-axis

#6793 4th axis

#6794 5th axis

Work coordinate system setting G57 (J4)
Setting: "1" = Least input increment
Setting range: ±99999999

#6795

Rotary angle setting G57 (J4)
Setting: "1" = 0.001 deg

#6796 X-axis

#6797 Y-axis

#6798 Z-axis

#6799 4th axis

#6800 5th axis

Work coordinate system setting G58 (J4)
Setting: "1" = Least input increment
Setting range: ±99999999

#6801

Rotary angle setting G58 (J4)
Setting: "1" = 0.001 deg

#6802	X-axis
#6803	Y-axis
#6804	Z-axis
#6805	4th axis
#6806	5th axis

Work coordinate system setting G59 (J4)
 Setting: "1" = Least input increment
 Setting range: ±99999999

#6807	
#6808	X-axis
#6809	Y-axis
#6810	Z-axis
#6811	4th axis
#6812	5th axis

Rotary angle setting G59 (J4)
 Setting: "1" = 0.001 deg

Work coordinate system setting G54 (J5)
 Setting: "1" = Least input increment
 Setting range: ±99999999

#6813	
#6814	X-axis
#6815	Y-axis
#6816	Z-axis
#6817	4th axis
#6818	5th axis

Rotary angle setting G54 (J5)
 Setting: "1" = 0.001 deg

Work coordinate system setting G55 (J5)
 Setting: "1" = Least input increment
 Setting range: ±99999999

#6819	
-------	--

Rotary angle setting G55 (J5)
 Setting: "1" = 0.001 deg

#6820	X-axis
#6821	Y-axis
#6822	Z-axis
#6823	4th axis
#6824	5th axis

Work coordinate system setting G56 (J5)
 Setting: "1" = Least input increment
 Setting range: ±99999999

#6825	
#6826	X-axis
#6827	Y-axis
#6828	Z-axis
#6829	4th axis
#6830	5th axis

Rotary angle setting G56 (J5)
 Setting: "1" = 0.001 deg

Work coordinate system setting G57 (J5)
 Setting: "1" = Least input increment
 Setting range: ±99999999

#6831	
#6832	X-axis
#6833	Y-axis
#6834	Z-axis
#6835	4th axis
#6836	5th axis

Rotary angle setting G57 (J5)
 Setting: "1" = 0.001 deg

Work coordinate system setting G58 (J5)
 Setting: "1" = Least input increment
 Setting range: ±99999999

#6837	
-------	--

Rotary angle setting G58 (J5)
 Setting: "1" = 0.001 deg

WORK COORDINATE SYSTEM SETTING
SPECIFICATION C (Cont'd)

#6838	X-axis
#6839	Y-axis
#6840	Z-axis
#6841	4th axis
#6842	5th axis

Work coordinate system setting G59 (J5)
Setting: "1" = Least input increment
Setting range: ±99999999

#6843	
-------	--

Rotary angle setting G59 (J5)
Setting: "1" = 0.001 deg

#6844	X-axis P1
#6845	Y-axis P1
#6846	Z-axis P1

Easy setter measurement point monitor
Tool setter measurement point monitor

#6847	X-axis P2
#6848	Y-axis P2
#6849	Z-axis P2
#6850	X-axis P3
#6851	Y-axis P3
#6852	Z-axis P3

Easy setter measurement point monitor

APPENDIX - 2 LIST OF PARAMETER NUMBERS

Parameter numbers are classified in the following five groups:

Table 2.1

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		D6	D5	D4	D3			D0
-------	--	----	----	----	----	--	--	----

- 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 is capable of coordinate system zero setup 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 is incapable of coordinate system zero setup, while capable of zero clear in all modes.

D₄, D₃: Status of G codes at power on.

D ₄	D ₃	Initial status
1	0	Sets the G code in the 08 group to G44 on power application.
0	1	Sets the G code in the 08 group to G43 on power application.
0	0	Sets the G code in the 08 group to G49 on power application.

- D₀** 1: Sets the G code in the 05 group to G91 on power application.
 0: Sets the G code in the 05 group to G90 on power application.

#6006	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

D₇, D₆ Signs of S5-digit analog (SDA) output

D ₇	D ₆	Sign	
1	1	Minus	Minus (1)
1	0	Plus	Plus (1)
0	1	Minus	Minus (2)
0	0	Plus	Plus (2)
		SINV signal off	SINV signal on

Note: The functions of (1) and (2) are the same.

- D5** 1: Sets the least input increment ×10
 0: Does not set the least input increment to ×10
- 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.

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

- 0: Provides no check on the spindle speed match signal (SAGR).
- 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 power application.
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 D0 to D4.

- D₀ 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	D0
-------	----	----	----	----	----	----	----	----

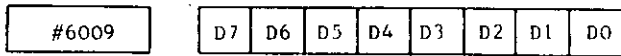
- D₇ 1: Disables start interlock after edit.
0: Enables start interlock after edit.
- D₆ 1: Employs the newly entered tool compensation value in place of the old value.
0: Adds the newly entered tool compensation value to the soterd value to establish another offset.
- D₅ 1: Enables error detect ON mode 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.
- D₀ 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.

#6008	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

- D₇ 1: Rewinds at M30 command in Tape operation with YASNAC standard tape reader (No RS-232C)
0: Does not rewind at M30 command in Tape operation with YASNAC standard tape reader (No.2 RS-232C)
- D₆ 1: Rewinds at M30 command in Tape operation with Yasnac standard tape reader.
0: Does not rewind at M30 command in tape operation with Yasnac standard tape reader.
- D₅ 1: 0 is changeable with **ALT** operation.
0: 0 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 G91 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.
- D₀ 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.



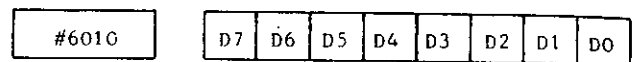
- D7 1: Up to 256 characters can be set in 1 tape read block.
 0: Up to 128 characters can be set in 1 tape read block.
- D6 1: Feed is provided when output from the NC.
 0: Feed is not provided when output from the NC.
- D5 1: Space code exists between programs at all program outputs (0 - 9999 OUT).
 0: Space code does not exist between programs at all program outputs (0 - 9999 OUT).
- D4 1: RST is output when both RWD and EOP are input in M02/M30, or when EOP alone is input.
 0: RST is not output when both RWD and EOP are input in M02/M30, or when EOP alone is input.

Note:

- D3 1: G68 is effective by the coordinate rotation command.
 0: G54 is effective by the coordinate rotation command.
- 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.
- D1 } Select which handle is used for the lower
 D0 } 1-digit speed correction function, where
 there are three manual pulse generators.

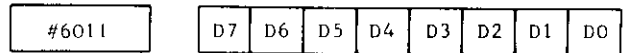
D1	D0	Selected handle
0	0	None
0	1	1st handle
1	0	2nd handle
1	1	3rd handle

Note: Effective when the optional parameter 36048 D4 is 1.



- D7, D6, D5
 Specify whether or not the automatic pulse is effective on the Z-, Y-, and X-axis, respectively.
 1: Effective
 0: Ineffective
 (Automatic handle Offset)
- D4, D3, D2, D1, D0
 Specify the direction of reference point return on the 4th-, 5th-, 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 D4 after turning on power.



- D7 1: Convert the ":" into "O" upon tape in/out.
 0: Do not convert the ":" into "O" upon tape in/out.
- D6, D5
 Specify whether or not the automatic pulse is effective on the 5th, and 4th axes, respectively.
 1: Effective
 0: Ineffective
- D4, D3, D2, D1, D0
 Command reference-point interlock for 5th, 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."

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**



D4, D3, D2, D1, D0

Specify whether or not the plus-direction external deceleration signal is effective on the 4th-, 5th-, Z-, Y- and X-axes, respectively.

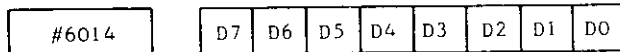
- 1: Makes the plus-direction external deceleration signal effective.
- 0: Makes the plus-direction external deceleration signal ineffective.



D4, D3, D2, D1, D0

Specify whether or not the minus-direction external deceleration signal is effective on the 4th-, 5th-, Z-, Y- and X-axes, respectively.

- 1: Makes the minus direction external deceleration signal effective.
- 0: Makes the minus direction external deceleration signal ineffective.



D7 1: PC CPU error is triggered.
0: PC CPU error is not triggered.

D6 1: SV CPU error is triggered.
0: SV CPU error is not triggered.

D5 1: ACGC2 SYNC ERROR or AMGC SYNC ERROR occurred.

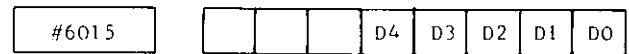
0: ACGC2 SYNC ERROR or AMGC SYNC ERROR did not occur.

D4, D3, D2, D1, D0

Specify the direction of the G60 unidirectional approach upper limit on the 4th-, 5th-, Z-, Y- and X-axes, respectively.

- 1: Minus direction
- 0: Plus direction

Note: The approach upper limit is set with #6062 to #6065.



D4, D3, D2, D1, D0

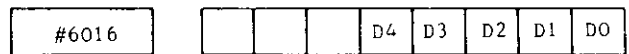
Specify whether or not the automatic coordinate system setting is effective on the 4th-, 5th-, Z-, Y- and X-axes, respectively

- 1: Effective
- 0: Ineffective

Note: The automatic coordinate system is established with the following parameters:

Metric system; #6636 - #6639

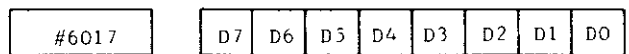
Inch system; #6630 - #6633



D4, D3, D2, D1, D0

Specify whether or not reference point return is effective on the 4th-, 5th-, Z-, Y- and X-axes, respectively.

- 1: Makes reference point return effective.
- 0: Makes reference point return ineffective.



D7 - D0

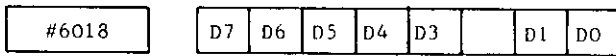
Specify whether or not a hole is to be made on channels 8-1, respectively, in a code corresponding to symbol "#" (used with user macro) in the EIA code.

- 1: Hole
- 0: No hole

Example:
D7 - D0 = 01001001

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.

Note: The specification of D7 - D0 = 0000000 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 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 P SET at G00 and G01 to G00 commands.

0: Waits for P SET at G00 and G01 to G00 commands.

D1 1: Outputs the FMF signal twice in a canned cycle.

0: Outputs the FMF signal once in a canned cycle.

D0 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 "8" during tape operation.

1: Effective

0: Ineffective

D6 1: When parameter outputs, only #7000 - #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

Specify the tool shift direction in the canned cycles of G76 and G77 (effective when #6019 = 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).

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6020	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

- 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 prior 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 $\Delta\Delta\Delta$ 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 F23 (mm/rev) for the feed per minute in the metric system.
- D1 1: Sets F32 (mm/min.) for the feed per minute in the metric system.
- 0: Sets F31 (mm/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.

#6021	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

- D7 1: Makes editing interlock O9000 through O9999 effective.
- 0: Makes editing interlock O9000 through O9999 ineffective.

- D6 1: Erases and stores the previous O 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 O0 with O through 9999 OUT.
- 0: Does not output O0 with 0 through 9999 OUT.
- D2 1: Displays the 0 number when the power supply is turned on and off.
- 0: Displays 0 when 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.
- 0: Does not consider M02, M30 and M99 as the program end when machining data is stored into memory.

#6022	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

- D7 1: Sets input unit of parameter setting to 10 RPM when specifying the S5 digits.
- 0: Sets input unit of parameter setting to 1 RPM 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 O9000 through O9000 Interlock effective.
0: Makes editing display O9000 through O9999 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 possible even without parity.
0: Disables read of ISO tape without parity.

#6023	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

- 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 (SR20-**).

D4-D0

Set the address for pan-out and CRT display on the additional axis.

Address	D4	D3	D2	D1	D0
A	0	0	0	0	1
B	0	0	0	1	0
C	0	0	0	1	1
U	1	0	1	0	1
V	1	0	1	1	0
W	1	0	1	1	1

#6024	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

- D7 When 5th axis is rotary axis, stored stroke limit is

- 1: Effective
- 0: Ineffective

- D6 Ignorance of additional axis (5NG)

- 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 (SR20-**).

D4-D0: Set address of pan-out and CRT display on the additional axis.

Address	D4	D3	D2	D1	D0
A	0	0	0	0	1
B	0	0	0	1	0
C	0	0	0	1	1
U	1	0	1	0	1
V	1	0	1	1	0
W	1	0	1	1	1

Note: These parameters cannot be written when parameter #6030-D7 is "0."

#6025								
-------	--	--	--	--	--	--	--	--

Parameter #6025 cannot be written at any time.

#6026		D5	D4	D3	D2	D1	D0
-------	--	----	----	----	----	----	----

#6027		D5	D4	D3	D2	D1	D0
-------	--	----	----	----	----	----	----

#6026, #6027:

- D5 1: Does not allow the control code (DC1 - DC4) to be used on the I/O device.
0: Allows the control code (DC1 - DC4) to be used on the I/O device.

- D4 1: Employs 2 stop bits on the I/O device.
0: Employs 1 stop bit on the I/O device.

D3 - D0

Baud rate setting

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

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(SIO-1) and #6027 on input device 2(SIO-2).

#6028			D5	D4	D3	D2	D1	D0
-------	--	--	----	----	----	----	----	----

#6029			D5	D4	D3	D2	D1	D0
-------	--	--	----	----	----	----	----	----

D5 1: Does not use control codes (DC1-DC4).
0: Uses control codes (DC1-DC4)

D4 1: Sets stop bits as two bits in the output device.
0: Sets stop bits as one bit

D3 to D0 Baud rate setting

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: 1. #6028 is for setting output device 1 (SIO-1).
2. #6029 is for setting output device 2 (SIO-2).

#6050	D7	D6	D5					D0
#6051	D7	D6	D5					D0
#6052	D7	D6	D5					D0
#6053	D7	D6	D5					D0
#6054	D7	D6	D5					D0
#6055	D7	D6	D5				D1	D0

Shown in the order of X-axis Y-axis, Z-axis, 4th-axis, 5th-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.

0: Servo control is performed.

D1 1: Perform spindle indexing when entering the solid tap mode by G93.

0: Do not perform spindle indexing when entering the solid tap mode by G93.

D0 1: Operation confirmation of each unit is not performed during system operation.

0: Operation confirmation of each unit is performed during system operation.

Note: Normal setting is "0."

Note: When these parameters are changed, be sure to turn the power off and on.

#6056					D3	D2	D1	D0
-------	--	--	--	--	----	----	----	----

X-axis, Y-axis, Z-axis, 4th axis, 5th axis, spindle in this order.

D3 to D0

Set the magnification of the feedback pulse input the NC unit.

	D3	D2	D1	D0
×1	0	0	0	1
×2	0	0	1	0
×4	0	1	0	0
×8	1	0	0	0

Note: Always turn off the power once when change is made to the setting.

D7, D6

X-axis Circular Projection Compensation

D7	D6	Compensation
0	0	None
1	0	Provided

#6057					D3	D2	D1	D0
-------	--	--	--	--	----	----	----	----

X-axis, Y-axis, Z-axis, 4th axis, 5th axis, spindle, in this order.

D3 to D0

Set the magnification of the feedback pulse input to the NC unit.

	D3	D2	D1	D0
×1	0	0	0	1
×2	0	0	1	0
×4	0	1	0	0
×8	1	0	0	0

Note: Always turn off the power once when change is made to the setting.

D7, D6

Y-axis Circular Projection Compensation

D7	D6	Compensation
0	0	None
1	0	Provided

#6058					D3	D2	D1	D0
-------	--	--	--	--	----	----	----	----

X-axis, Y-axis, Z-axis, 4th axis, 5th axis, spindle, in this order.

D3 to D0

Set the magnification of the feedback pulse input to the NC unit.

	D3	D2	D1	D0
×1	0	0	0	1
×2	0	0	1	0
×4	0	1	0	0
×8	1	0	0	0

Note: Always turn off the power once when change is made to the setting.

D7, D6

Z-axis Circular Projection Compensation

D7	D6	Compensation
0	0	None
1	0	Provided

#6059					D3	D2	D1	D0
-------	--	--	--	--	----	----	----	----

#6060					D3	D2	D1	D0
-------	--	--	--	--	----	----	----	----

#6061					D3	D2	D1	D0
-------	--	--	--	--	----	----	----	----

Shown in the order of X-axis, Y-axis, Z-axis, 4th-axis, 5th-axis and spindle.

D3 to D0: They set the pulse scale factor of feedback pulses to be input into NC unit.

	D3	D2	D1	D0
×1	0	0	0	0
×2	0	0	1	0
×4	0	1	0	0
×8	1	0	0	0

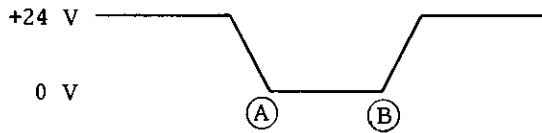
Note: Be sure to turn the power on and off after changing the parameter.

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6062			D5	D4				
-------	--	--	----	----	--	--	--	--

D4: This sets the signal state of processing start of "skip" input for skip function.

D5: This sets the signal state of processing start of "PINT" input for pint 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.

Note: Be sure to turn the power on and off after changing the parameter.

Note: Be sure to set "0" except in D4 and D5.

#6063					D3		D1	
-------	--	--	--	--	----	--	----	--

D1: This determines the ENABLE/DISABLE of control circuit of "SKIP" input for skip function.

D3: This determines the ENABLE/DISABLE of control circuit of "PINT" input for pint function.

Note: Be sure to set "1" when using the skip function and program interruption function.

Note: Be sure to turn the power on and off after changing the parameter to set "0" except in D1 and D3.

#6064				D4	D3	D2	D1	D0
-------	--	--	--	----	----	----	----	----

D4, D3, D2, D1, D0

Shown in the order of 5th-axis, 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.

#6065	D7					D2	D1	D0
-------	----	--	--	--	--	----	----	----

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: Always turn off the power once after setting this parameter.

D2 1: The T number of the executed offset register in the tool nose coordinate system and the coordinate system offset memory are compared and checked.

0: The T number of the executed offset register in the tool nose coordinate system and the coordinate system offset memory are not compared or checked.

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 pulse display section, and the minus peak value is displayed on the Z-axis error pulse display section.)

D0 1: Do not wait for PSET during the tapping cycle by the solid tap function.

0: Wait for PSET during the tapping cycle by the solid tap function.

#6066	D7	D6	D5	D4	D3	D2		D0
-------	----	----	----	----	----	----	--	----

D7 1: JOG speed of 4 and 5 axis is 1/10 of 3 axis.

0: JOG speed of 4 and 5 axis is not 1/10 of 3 axis.

D6 1: At power ON, M97.

0: At power ON, N96.

D5 1: H/D changing 149/149 sets.

0: H/D changing 99/199 sets.

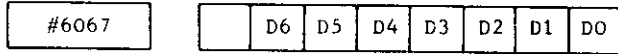
D4 1: With interference check function

0: Without interference check function

D3 1: Cross point calculation automatic selector is performed

0: Cross point calculation automatic selector is not performed

- D2 1: CMOS free area check function is invalid
 0: CMOS free area check function is valid
- D0 1: Workpiece setter, tool setter, tactile sensor input signal selection
 1: Tactile sensor signal hardware interruption
 0: Tactile sensor signal I/O



Specify the format of the communication module RS-232C.

D6, D5 Parity Check Designation

D6	D5	Check status
0	0	No parity
0	1	Even parity
1	0	Odd parity

D4 Character bit setting

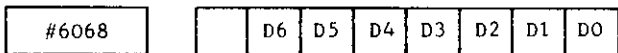
- 1: 8 bits
 0: 7 bits

D3 Stop bit setting

- 1: 2 bits
 0: 1 bit

D2, D1, D0 Baud Rate Setting

Baud rate	D2	D1	D0
1200	0	0	0
2400	0	0	1
4800	0	1	0
9600	0	1	1
19200	1	0	0



Specify the format of the communication module RS-422.

D6, D5 Parity Check Designation

D6	D5	Check status
0	0	No parity
0	1	Even parity
1	0	Odd parity

D4 Character bit setting

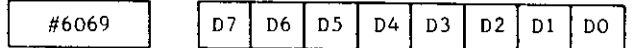
- 1: 8 bits
 0: 7 bits

D3 Stop bit setting

- 1: 2 bits
 0: 2 bits

D2, D1, D0 Baud Rate Setting

Baud rate	D2	D1	D0
1200	0	0	0
2400	0	0	1
4800	0	1	0
9600	0	1	1
19200	1	0	0



Specify the D7, D6, D5 communication module protocol.

	D7	D6	D5
Protocol 1	0	0	0
Protocol 2	0	0	1
Protocol 3	0	1	0
Protocol 4	0	1	1

D4 Control character transmission designation

- 0: No transmission
 1: Transmission

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

D3, D2 DNC Operation Start
Buffer Storage Amount

D3	D2	Buffer storage
0	1	128 bytes
1	0	256 bytes
1	1	512 bytes
0	0	2048 bytes

- D1 1: Triggers alarm also on the NC side when alarm occurs in the DNC.
0: Does not trigger alarm on the NC side when alarm occurs in the DNC.
- D2 1: Device designation judgment input designation: setting (#60037)
0: Device designation judgment input designation: I/O (#13107)

#6070	DNCWAT	TIMEOUT				D2	D1	D0
-------	--------	---------	--	--	--	----	----	----

- D7 DAT binary data after the WAT command: 150 bytes (15-segment) buffering function designation
1: Performs buffering
0: Does not perform buffering

Note: Measurement to the MAKINO TMS operation machine.

- D6 TIMEOUT #6069 When D1 = 1
1: TIMEOUT error alone is not output.
0: Normal

D2, D1, D0

DNC high-speed cutting multi-segment designation

D2	D1	D0	Segment
0	0	0	8-ms mono-segment
0	0	1	4-ms multi-segment
0	1	0	2-ms multi-segment
0	1	1	1-ms multi-segment (Currently cannot be used)
1	0	0	16-ms multi-segment (Currently cannot be used)

#6072	D7	D6	D5	D4	D3	D2	D1	D0
-------	----	----	----	----	----	----	----	----

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
D4	5th axis
D5	Cutting feedrate F
D6	Spindle speed S
D7	Tool command T

#6073				D4	D3	D2	D1	D0
-------	--	--	--	----	----	----	----	----

- 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: Triggers double-address error
- D2 Tool position offset (G45 to G48) number command
1: H code, D code are effective
0: D code alone is effective
- D1 The TR
1: Prohibits
0: Performs the M code macro call upon restarting the program
- D0 The TR
1: Prohibits
0: Performs the M code macro call upon restarting the program

#6077 to #6091 are circular projection compensation (soft hammer) parameters, used as byte parameters

#6077	1st X-axis speed command
#6078	1st Y-axis speed command
#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 offset start waiting time
#6084	Y-axis offset start waiting time
#6085	Z-axis offset start waiting time
#6086	H1X effective time
#6087	H1Y effective time
#6088	H1Z effective time
#6089	H2X effective time
#6090	H2Y effective time
#6091	H2Z effective time

Setting: "1" = 2 ms

Setting range: 0 to 255

#6092	D7								
-------	----	--	--	--	--	--	--	--	--

- D7 1: Displays the macro call T code upon restarting the program
- 0: Does not display the macro call T code upon restarting the program

#6093								D1	D0
-------	--	--	--	--	--	--	--	----	----

D1, D0

Internal Buffer Transmission Method
(IF → NC) Setting Parameter

D1	D0	Transmission byte
0	0	128 × 8
0	1	256 × 4
1	0	512 × 2
1	1	1024 × 1

#6107	
-------	--

Specifies the number of manual pulse generators.

Setting: "1" = 1

Setting range: 0 - 5

Note: When change is made to this parameter, always turn off the power once.

#6110	([)
#6111	(])
#6112	(*)
#6113	(=)
#6114	(()
#6115	())

#6110 to #6115:

Specify punches of the codes corresponding to the symbols of EIA codes used in the user macro body.

1: Punched

0: Not punched

Setting range: 0 - 255

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6116	
#6117	
#6118	
#6119	

Maximum 4 types of M codes to stop advance reading.

Setting range: 0 - 255

#6120	G-1
#6121	G-2
#6122	G-3
#6123	G-4
#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 - 255

#6130	M-1
#6131	M-2
#6132	M-3

#6133	M-4
-------	-----

#6130 to #6133:

Specify up to 4 M codes for calling user macros.

Setting range: 0 - 255

Note: M00, M01, M02, M30 and M90 - 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.

#6136	1st byte
-------	----------

#6137	2nd byte
-------	----------

#6138	3rd byte
-------	----------

DNC high-speed cutting data
Start character

#6139	
-------	--

DNC high-speed cutting data axis designation
Setting range: 0 to 2

#6141	
-------	--

#6142	
-------	--

#6143	
-------	--

#6144	
-------	--

#6145	
-------	--

#6146	
-------	--

#6147	
-------	--

#6148	
-------	--

#6149	
-------	--

#6141 to #6149:

Specify 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.

Setting range: 0 - 255 /pulse

#6150	
#6151	
#6152	
#6153	
#6154	

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, 4-axis and 5-axis.

Setting: "1" = 1%

Setting range: 1 - 200 (stored setting = 120)

#6155	
-------	--

Set the magnification against the deviation pulse at the maximum speed of the spindle in the spindle alarm zone.

Setting: "1" = 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	
#6160	

Sets Error Defect ON area for X-, Y-, Z-, 4th and 5th-axis, respectively.

Setting: "1" = 1 pulse

Setting range: 0 - 255

Note: When this parameter setting is changed, turn on the power.

#6161	
-------	--

Spindle error detect ON area (for solid tap)

Setting: "1" = 1 pulse

Setting range: 0 to 255

Note: Always turn off the power once when change is made in parameters #6150 to #6161.

#6162	
#6163	
#6164	
#6165	
#6166	
#6167	

M codes to stop advance reading setting expanded area.

#6168	X-axis
#6169	Y-axis
#6170	Z-axis
#6171	4th axis
#6172	5th axis

Set pitch error compensation setting pulse multiplication factor for X-, Y-, Z-, 4th-, and 5th-axis, respectively.

Setting: "1" = 1 multiplication

Setting range: 0 - 3

#6174	
-------	--

Spindle error detect ON area (for spindle indexing)

Setting: "1" = 1 pulse

Setting range: 0 to 255

#6180	
-------	--

FSMEM (FS storage) count

Setting range: 0 to 128

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6181

FSMEM (FS storage) warning

Setting range: 0 to 128

#6196

Intermediate gear

#6197

Spindle gear

#6198

Spindle and spindle motor gear ratio (motor side)

#6199

Spindle and spindle motor gear ratio (spindle side)

#6210

G76/G77 dwell timer

Setting: "1" = 1 ms

Setting range: 0 - 32767

#6211

G73 8 amount

Setting: "1" = 0.001 mm/min or 0.0001 inch/min

Setting range: 0 - 32767

#6212

G73/83 default dwell

Setting: "1" = 1 ms

Setting range: 0 to 32767

#6213

G83 8 amount

Setting: "1" = 0.001 mm/min or 0.0001 inch/min

Setting range: 0 - 32767

Specify 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.

#6217

Automatic corner override F2 (command feed n%)

Setting: "1" = 1%

#6218

Automatic corner override F3 (command feed n%)

Setting: "1" = 1%

#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.

Setting: "1" = 1 ms

Setting range: 0 - 32767

#6221

Specifies the interval from gear output (GRH, GRL) unit SF transmission when an S5-digit designation is added.

Setting: "1" = 1 ms

Setting range: 0 - 32767

#6222

Specify 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/min.

Setting range: 0 - 24000

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 - 24000

Note: This specification is effective when #6019D0 = 1.
If #6019D0 = 0, rapid traverse is effective regardless of this parameter specification.

#6224

Specifies the delay time for checking the spindle speed reaching signal (SAGR).

Setting: "1" = 1 ms

Setting range: 0 - 32767

#6225

Specifies the feedrate for the rapid traverse section in circle cutting (G12, G13).

Setting: "1" = 1 mm/min. or 0.1 inch/min.

Setting range: 0 - 24000

#6226

#6227

#6266, #6277

Specify the maximum feedrate for F1-digit designation.

Setting: "1" = 1 mm/min.

Setting range: 0 - 24000

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 rotary axes (A, B, C).

Setting: "1" = 1 mm/min. or 0.1 inch/min.

Setting range: 0 - 24000

#6229

Specifies the maximum feedrate for the rotary axes (A, B, C).

Setting: "1" = 1 mm/min.

Setting range: 0 - 24000

Note: any feedrate greater than those set in #6228 and #6229 are bunched into those settings.

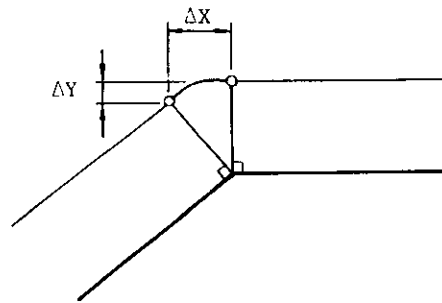
#6230

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: "1" = 0.001 mm (metric system)

"1" = 0.001 in. (inch system)

Setting range: 0 - 32767



The corner arc setting is ignored when:

$$\Delta X \leq \#6230$$

$$\Delta Y \leq \#6230$$

Standard setting = 5

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6231

Specifies the F_0 speed for rapid traverse override.

Setting: "1" 1 mm/min. or 0.1 inch/min.

Setting range: 0 - 24000

#6232

Specifies the feedrate in the skip function (G31).

Setting: "1" = 1 mm/min. or 0.1 inch/mm.

Setting range: 0 - 24000

Note: This setting is effective when parameter #6019D₄ = 1.

#6233

to

to

#6264

#6233 to #6264

Specify the feedrate for the respective positions on the jog feedrate select switch.

Setting: "1" = 1 mm/min. or 0.1 inch/min.

Setting range: 0 - 24000

Typical settings

Table 3.3.6 Typical Settings mm/min

Switch position	Parameter		Continuous manual feedrate	
	Number	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 each 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 - 32 (switch position)

#6266

#6267

#6268

#6269

#6266 to #6269:

Specify the maximum spindle speed, respectively, for gears 1, 2, 3 and 4 (specifiable only in S5-digit designation).

Setting: "1" = 1 r/min or "1" = 10 r/min

Setting range: 0 - 32767

#6270

Specifies the speed command output value to the spindle motor when a gear shift (GR 0) input is entered (specifiable only in S5-digit designation).

Specifies No. of bits occupied in 32767 (16 bits) or 4095 (12 bits) directly.

Setting value: $\frac{\text{Gear shift spindle motor speed}}{\text{Spindle motor max speed}}$
(Command = 10 V)

× 4095 --- 12-bit output

$\frac{\text{Gear shift spindle motor speed}}{\text{Spindle motor max speed}}$
(Command = 10 V)

× 32767 --- Analog output

Setting range: 0 - 32767

#6271

#6272

#6273

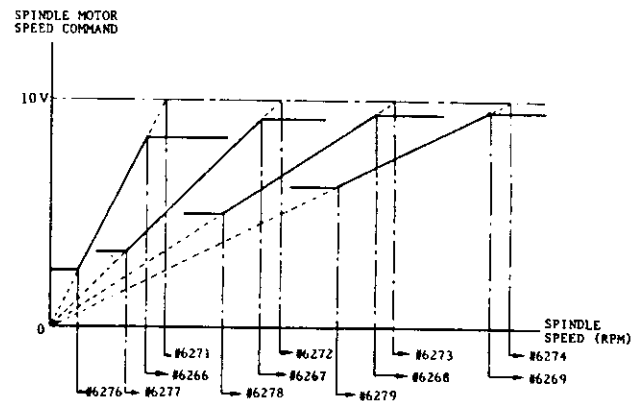
#6274

#6271 to #6274:

Specify 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 r/min or "1" = 10 r/min
(For parameter #6022 D7)

Setting range: 1 - 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 r/min
(For parameter #6022 D7)

Setting range: 1 - 32767

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6276	
#6277	
#6278	
#6279	

#6276 to #6279:

Specify the maximum 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 rpm or "1" = 10 rpm

Setting range: 0 - 32767

#6280	X-axis
#6281	Y-axis
#6282	Z-axis
#6283	4th axis
#6284	5th axis

#6280 to #6284:

Specify the rapid traverse rate, respectively, on the X-, Y-, Z-, 4th- and 5th-axes.

Setting: "1" = 1 mm/min, or "1" = 0.1 inch/min.

Setting range: 0 - 24000

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, 4th-axis or 5th-axis.

Rapid traverse rate: $V_a = \#6280 \rightarrow$ Rate $v'a$ recreated by the calculations.

Acceleration & deceleration 1st-stage time constant:

$$t_a = \#6286$$

"2nd-stage time constant switching rate:

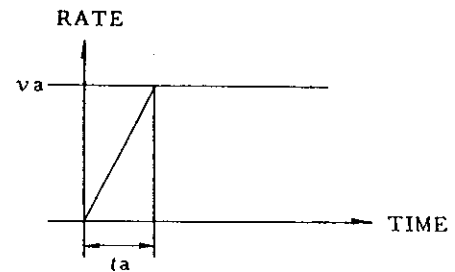
$v_b = \#6292 \rightarrow$ Rate $v'a$ recreated by the calculations.

Acceleration & deceleration 2nd-stage time constant: #6298

(a) Where 2nd-stage acceleration & deceleration are not used:

Condition: $v_b = 0$
 $t_b = 0$

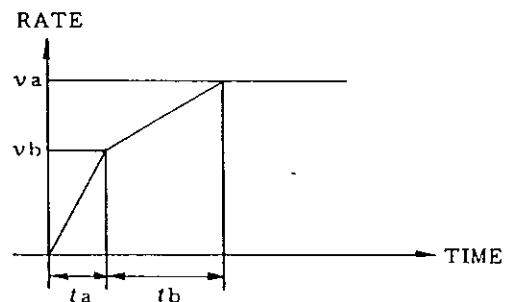
$$v'a = \left(\frac{v_a}{7.5} * \frac{100}{MP} \right) / \frac{t_a}{8}$$



(b) Where 2nd-stage acceleration & deceleration are used:

$$v'b = \left(\frac{v_a}{7.5} * \frac{100}{MP} \right) / \frac{t_a}{8}$$

$$v'a = v'b + \left(\frac{v_a - v_b}{7.5} / \frac{100}{MP} \right) * \frac{t_b}{8}$$



Note 1: Setting of 24,000 mm/min. may sometime becomes 23,999 mm/min. in rapid traverse rate.

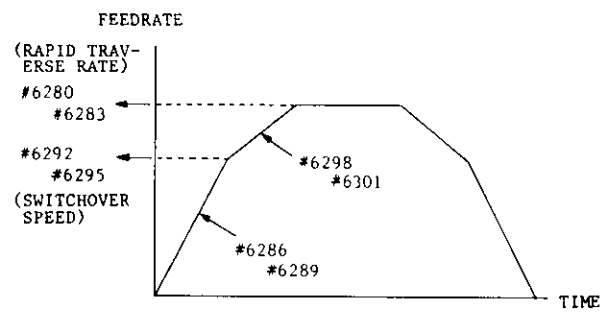
Note 2: MP indicates #6450 and #6451.

#6286	X-axis
#6287	Y-axis
#6288	Z-axis
#6289	4th axis
#6290	5th axis

#6286 to #6290:

Specify the first-stage time constant in linear acceleration/deceleration, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 1 ms
 Setting range: 8 - 32767



#6291	Spindle
-------	---------

Sets linear lower limit speed constant
 Setting: "1" = 1 ms
 Setting range: 1 - 255

#6292	X-axis
#6293	Y-axis
#6294	Z-axis
#6295	4th axis

#6296	5th axis
-------	----------

#6292 to #6296:

Specify the second-stage time constant switchover speed in linear acceleration/deceleration, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 1 mm/min. or "1" = 0.1 inch/min.
 Setting range: 0 - 24000

Note: When this parameter is changed, turn off and on power.

#6298	X-axis
#6299	Y-axis
#6300	Z-axis
#6301	4th axis
#6302	5th axis

#6298 to #6302:

Specify the second-stage time constant in linear acceleration/deceleration, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 1 ms
 Setting range: 8 - 32767

Note: Setting should be made by multiples (x8, x16, x24...)

#6304	X-axis
#6305	Y-axis
#6306	Z-axis
#6307	4th axis
#6308	5th axis

#6304 to #6308:

Specify the traverse distance for reference point return, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 0.001 mm or "1" = 0.0001 inch/min.

Setting range: 0 - 32767

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6310	X-axis
#6311	Y-axis
#6312	Z-axis
#6313	4th axis
#6314	5th axis

#6310 to #6314:

Specify the approach speed 1 for reference point return, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 1 mm/min. or "1" = 0.1 inch/min.

Setting range: 0 - 24000

#6316	X-axis
#6317	Y-axis
#6318	Z-axis
#6319	4th axis
#6320	5th axis

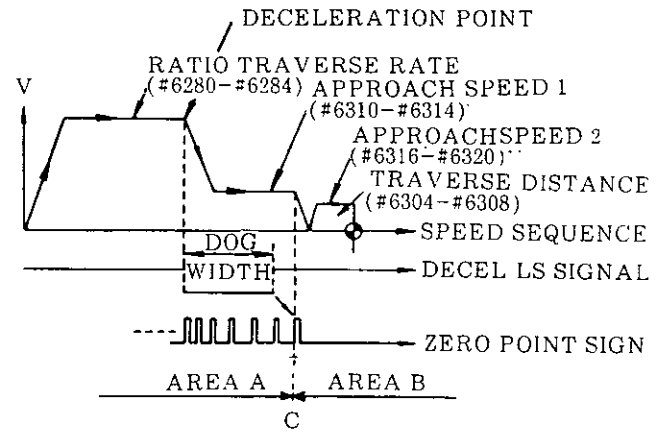
#6316 to #6320:

Specify the approach speed 2 for reference point return, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 1 mm/min. or "1" = 0.1 inch/min.

Setting range: 0 - 24000

Note: The parameters associated with reference point return operations are as follows.



Reference point return direction:
#6010 D₀ - D₃

Reference point return enabled/disabled:
#6016 D₆ - D₃

#6322	X-axis
#6323	Y-axis
#6324	Z-axis
#6325	4th axis
#6326	5th axis

#6322 to #6326:

Specify the number of the start point for pitch error compensation, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: 0 - 511

#6328	X-axis
#6329	Y-axis
#6330	Z-axis
#6331	4th axis
#6332	5th axis

#6328 to #6332:

Specify the number of the start point for pitch error compensation, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: 0 - 511

#6334	X-axis
#6335	Y-axis
#6336	Z-axis
#6337	4th axis
#6338	5th axis

#6334 to #6338:

Specify the reference point for pitch error compensation, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: 0 - 511

#6340	
-------	--

Specifies the external deceleration speed for rapid traverse.

Setting: "1" = 1 mm/min. or "1" = 0.1 inch/min.
(common to all axes)

Setting range: 0 - 24000

#6341	
-------	--

Specifies the external deceleration speed for cutting feed.

Setting: "1" = 1 mm/min. (common to all axes)

Setting range: 0 - 24000

#6342	X-axis
#6343	Y-axis
#6344	Z-axis
#6345	4th axis
#6346	5th axis

#6342 to #6346:

Specify the offset in external workpiece coordinate system shift, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: 1 = 0.001 mm

Setting range: 0 - ±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 "1" = 0.1 inch/min.

Setting range: 0 - 24000

#6350	X-axis
#6351	Y-axis
#6352	Z-axis
#6353	4th axis
#6354	5th axis

#6350 to #6354:

Specify the rapid traverse accel/decel constants X-, Y-, Z-, and 4th-, and 5th-axis handle feed, respectively.

Setting: "1" = 1 msec

Setting range: 8 - 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:

Specify the time between ESP and SVOF for X-, Y-, and Z-axis, respectively.

Setting: "1" = 8 msec

Setting range: 0 - 32767

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#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.

#6384

to to

#6391

Title display code setting during power turn-on:
1st line

#6392

to to

#6399

Title display code setting during power turn-on:
2nd line

#6400

#6401

#6402

#6403

#6404

#6405

Backlash compensation values shown in the order of X-axis, Y-axis, Z-axis, 4th-axis, 5th-axis 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

#6407

#6408

#6409

#6410

#6411

Command unit position loop gains are set in the order of X-axis, Y-axis, Z-axis, 4th-axis, 5th-axis and spindle.

Setting: "1" = 0.01 sec⁻¹

Setting range: 0 - 32767

#6412

#6413

#6414

#6415

#6416

Acceleration and deceleration time constants for control unit during ordinary cutting are set in the order of X-axis, Y-axis, Z-axis, 4th-axis, 5th-axis.

Setting: "1" = 1 ms

Setting range: 0 - 32767

#6418

#6419

#6420

#6421

#6422

Acceleration and deceleration bias for control unit during ordinary cutting is set in the order of X-axis, Y-axis, Z-axis, 4th-axis and 5th-axis.

Setting: "1" = 1 mm/min. or "1" = 0.1 inch/min.

Setting range: 0 - 32767

#6424	X-axis
#6425	Y-axis
#6426	Z-axis
#6427	4th axis
#6428	5th axis

Setting of acceleration and deceleration time constants for control unit during screw/tap cutting in the order of X-axis, Y-axis, Z-axis, 4th-axis and 5th-axis.

Setting: "1" = 1 ms

Setting range: 0 - 32767

#6430	X-axis
#6431	Y-axis
#6432	Z-axis
#6433	4th axis
#6434	5th axis

Setting of acceleration and deceleration bias for control unit thread/tap cutting.

Setting: "1" = 1 mm/min. or "1" = 0.1 inch/min.

Setting range: 0 - 32767

#6436	X-axis
#6437	Y-axis
#6438	Z-axis
#6439	4th axis
#6440	5th axis

Overshoot during G60 in the order of X-axis, Y-axis, Z-axis, 4th-axis and 5th-axis.

Setting: "1" = 1 pulse

Setting range: 0 - 32767

#6444	X-axis
#6445	Y-axis
#6446	Z-axis

#6447	4th axis
#6448	5th axis

Setting: "1" = 0.001 mm/rev or "1" = 0.001 inches/rev

Setting range: 1 - 65535

Note 1: Be sure to turn the power on and off after changing the parameter.

Note 2: For input of 32768-65535, 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 - 32768

#6450	X-axis
#6451	Y-axis
#6452	Z-axis
#6453	4th axis
#6454	5th axis

Minimum unit of travel in the order of X-axis, Y-axis, Z-axis, 4th-axis, and 5th-axis.

Setting: "1" = 0.00001 mm or "1" = 0.000001 inches

Setting range: 100 is set.

Note: Be sure to turn the power on and off after changing the parameter.

#6456	X-axis
#6457	Y-axis
#6458	Z-axis
#6459	4th axis
#6460	5th axis

Servo input in the order of X-axis, Y-axis, Z-axis, 4th-axis, and 5th-axis.

Setting: "1" = 1 rpm/volt

Setting range: 250 is set.

PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)

#6462

Spindle indexing speed rate.
Setting: "1" = 500 pps
Setting range: 0 - 32767

#6463

Spindle indexing creep rate.
Setting: "1" = 500 pps
Setting range: 0 - 32767

#6464

Spindle indexing & spindle stop confirmation timer.
Setting: "1" = 8 ms
Setting range: 0 - 255

#6465

Spindle indexing and origin position.
Setting: "1" = 1P
Setting range: 0 - 4095

#6466 F1

#6467 F2

Rapid traverse override of F1 and F2 in the order shown.
Setting: "1" = 1 mm/min. or "1" = 0.1 inch/min.
Setting range: 0 to 24000

#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 accel/decel feed
Setting: "1" = 1/256 times 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 r/min
Setting range: 0 to 32767

#6472

Spindle revolutions against the 10 V command of the gear used on the solid tap
Setting: "1" = 1 r/min
Setting range: 0 to 32767

#6473

Spindle stop acknowledgment revolutions in the G9 spindle position loop
Setting: "1" = 1 r/min
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
#6484	5th axis
#6485	6th axis

#6495	Y-axis
#6496	Z-axis
#6497	4th axis
#6498	5th axis
#6499	6th 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

#6487	
-------	--

#6488	
-------	--

Form compensation linear accel/decel maximum acceleration

Setting: "1" = 1/64m/S²

Setting range: 1 to 32768

#6489	
-------	--

#6490	
-------	--

CAL ERROR (DIVISION) area setting

#6491	
-------	--

#6492	
-------	--

#6493	
-------	--

#6494	X-axis
-------	--------

Feed forward gain

#6580	
-------	--

#6581	
-------	--

#6582	
-------	--

#6583	
-------	--

#6584	
-------	--

#6585	
-------	--

#6586	
-------	--

#6587	
-------	--

#6588	
-------	--

#6589	
-------	--

#6590	
-------	--

#6591	
-------	--

#6592	
-------	--

#6593	
-------	--

#6594	
-------	--

#6595	
-------	--

#6596	
-------	--

#6597	
-------	--

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6598	
#6599	

Set 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
#6604	5th axis

#6600 to #6604:

Specify the plus direction boundary value for stored stroke limit 1, respectively, on the X-, Y-, Z-, 4th- and 5th-axes.

Setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6606	X-axis
#6607	Y-axis
#6608	Z-axis
#6609	4th axis
#6610	5th axis

#6606 to #6610:

Specify the plus direction boundary value for stored stroke limit 1, respectively, on the X-, Y-, Z-, 4th- and 5th-axes.

Setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6612	X-axis
#6613	Y-axis
#6614	Z-axis
#6615	4th axis
#6616	5th axis

#6612 to #6616:

Specify the distance between the first and the second reference point, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6618	X-axis
#6619	Y-axis
#6620	Z-axis
#6621	4th axis
#6622	5th axis

#6618 to #6622:

Specify the distance between the first and the third reference point, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6624	X-axis
#6625	Y-axis
#6626	Z-axis
#6627	4th axis
#6628	5th axis

#6624 to #6628:

Specify the distance between the first and the fourth reference point, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes.

Setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6630	X-axis
#6631	Y-axis
#6632	Z-axis
#6633	4th axis
#6634	5th axis

#6630 to #6634

Specify the value for automatic coordinate system setting at the time of inch input, respectively, on the X-, Y-, Z-, and 4th, and 5th-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 - ±99999999

#6636	X-axis
#6637	Y-axis
#6638	Z-axis
#6639	4th axis
#6640	5th axis

#6636 to #6640:

Specify the value for automatic coordinate system setting at the time of metric input, respectively, on the X-, Y-, Z-, 4th-, and 5th-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 - ±99999999

Note: Each setting is effective only for an axis with parameter #6015 at "1."

#6642	X-axis
#6643	Y-axis
#6644	Z-axis
#6645	4th axis
#6646	5th axis

#6642 to #6646:

Specify the compensation interval in pitch error compensation, respectively, on the X-, Y-, Z-, 4th-, and 5th-axes

Setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6650	X-axis
#6651	Y-axis
#6652	Z-axis

#6650 to #6652:

Specify stored stroke limit 3 for X-, Y-, and Z-axis, respectively.

Plus (+) boundary setting: "1" = 1 pulse

Setting range: 0 - ±99999999

**PARAMETER NUMBERS AND THEIR CONTENTS
(CONT'D)**

#6653 X-axis

#6654 Y-axis

#6655 Z-axis

#6653 to #6655:

Specify stored stroke limit 3 for X-, Y, and Z-axis, respectively.

Minus (-) boundary setting: "1" pulse

Setting range: 0 - ±99999999

#6656 X-axis

#6657 Y-axis

#6658 Z-axis

#6656 to #6658:

Specify stored stroke limit 4 for X-, Y-, and Z-axis, respectively.

Plus (+) boundary setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6659 X-axis

#6660 Y-axis

#6661 Z-axis

#6659 to #6661 (optional):

Specify stored stroke limit 4 for X-, Y-, and Z-axis, respectively.

Minus (-) boundary setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6662 X-axis

#6663 Y-axis

#6664 Z-axis

#6662 to #6664 (optional):

Specify stored stroke limit 5 for X-, Y-, and Z-axis, respectively.

Plus (+) boundary setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6665 X-axis

#6666 Y-axis

#6667 Z-axis

#6665 to #6667 (optional):

Specify stored stroke limit 5 for X-, Y-, and Z-axis, respectively.

Minus (-) boundary setting: "1" = 1 pulse

Setting range: 0 - ±99999999

#6674

Sets the multiplication factor of characters of the display at power on.

#6675

Sets the position of the display at power on.

#6676

Handle input pulse clamp amount

Setting: "1" = 1 pulse (input pulse)

#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

#6684 5th axis

External work coordinate shift offset amount:
6 digits

Setting range: "1" = 0 - ±799999

#6685	
#6686	
#6687	
#6688	
#6689	
#6690	
#6691	
#6692	
#6693	
#6694	
#6695	
#6696	
#6697	
#6698	
#6699	

#8000	number 0
to	to
#8511	number 511

#8000 to #8511

Specify the respective values of pitch error compensation.

Setting: 0 - ±15 (output increment)

Note: Parameters #6322 to #6339 determine specific combinations of settings and axes.

Always turn off the power once when change is made to parameters #6680 to #6684 and #6686 to #6690.

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, 4th-, and 5th-axes.
(including rotary axis)

No. of correction points: 512 Max.

Compensation base point: Reference point

Compensation interval: 10000 Pulses or more

Data setting system: Absolute/incremental
(Set by Parameter #6039D2)

Compensation value:

Minimum compensation unit: 1 pulse (least output increment)

Compensation multiplication: X13 max.

One-time-compensation value: 15 pulses max.
(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. Also, set the absolute set point in such a manner that the difference between #8001 and #8002 becomes "0."

Table 3.1

	Axis	Parameter #	Functions
Compensation interval	X to β	#6642 to #6646	6000 OR MORE "1" = 1 pulse
Absolute/incremental setting switchable		#6039D2	"0" = Incremental setting "1" = Absolute setting
Compensation reference no.	X to β	#6334 to #6338	Value of parameter # of compensation on each point minus 8000 will be written
Compensation max point	X to β	#6322 to #6326	
Compensation min point	X to β	#6328 to #6332	
Compensation value on each point	X to β	#8000 to #8511	-15 to +15 (Incremental setting) "1" = 1 pulse
Compensation multiplication	X to β	#6168 to #6172	0 to 3 "1" = 1X

The figure below shows the example of writing the data for X axis.

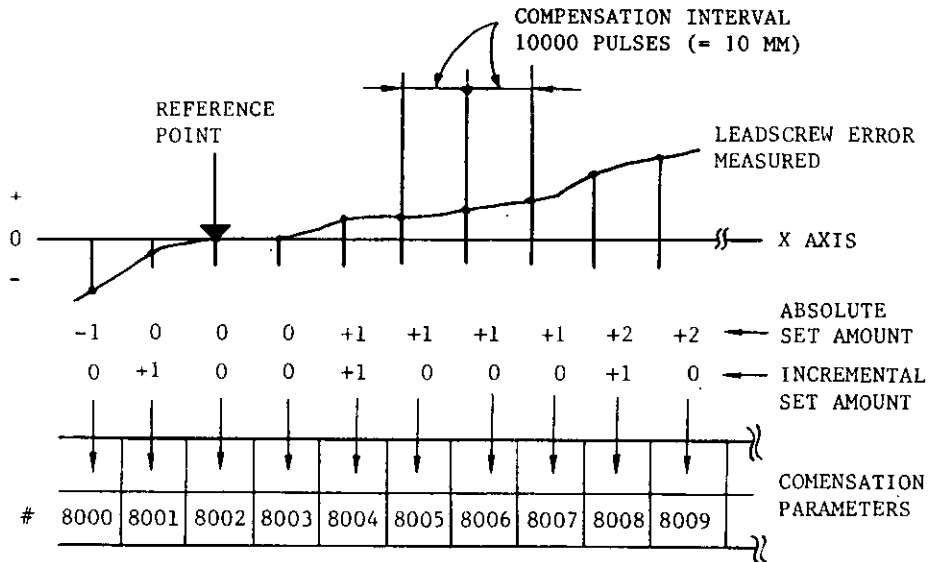


Fig. A3.1

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:

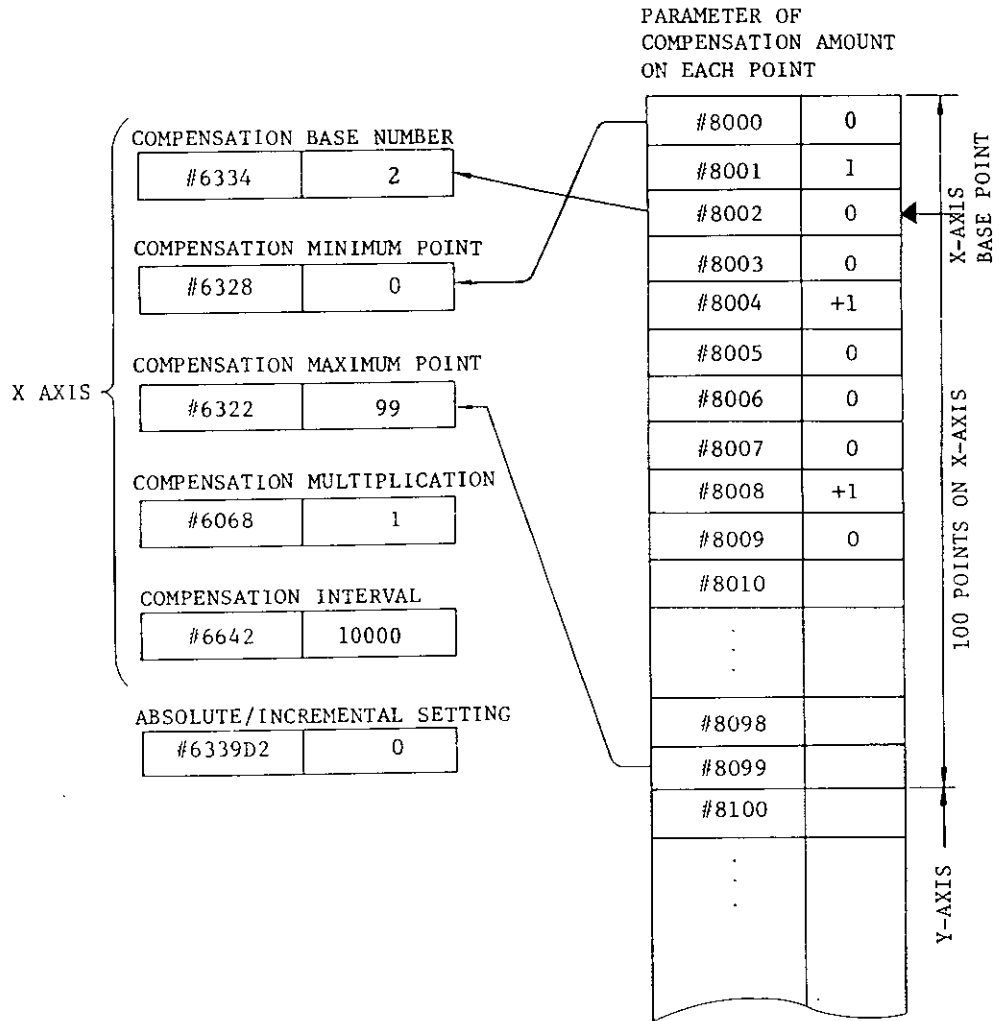


Fig. A3.2

To use the 4th and 5th axes 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

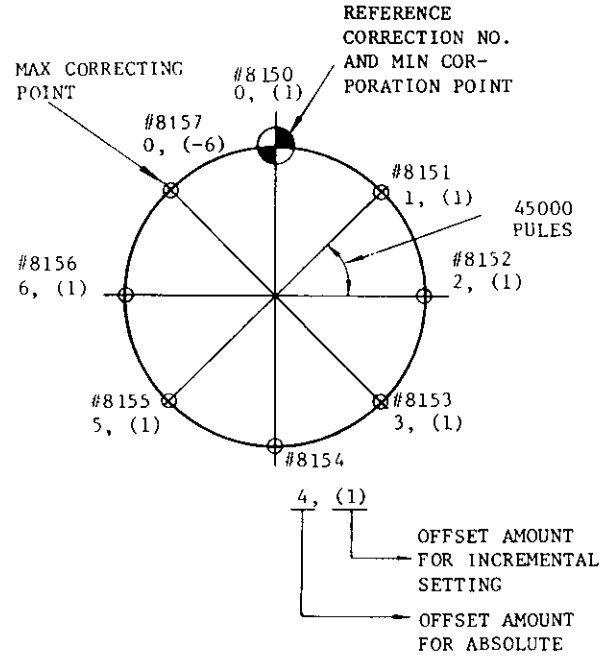
The compensation amount to be set to the reference point should be as follows:

- 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.



		Offset parameter at each point			
		Parameter	Absolute setting	Incremental setting	
4th axis of rotary axis	Offset reference No.	#6337	150		
	Offset min point	#6331	150		
	Offset max point	#6325	157		
	Offset multiplication factor	#6071	1		
	Offset point	#6645	45000		
			#8150	0	0
			#8151	1	1
			#8152	2	1
		#8153	3	1	
		#8154	4	1	
		#8155	5	1	
		#8156	6	1	
		#8157	0	-6	
		⋮	⋮	⋮	

APPENDIX - 4 LIST OF STANDARD INPUT/OUTPUT SIGNALS

Table 4.1 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

D7	D6	D5	D4	D3	D2	D1	D0
0	0	0	0	1	1	1	1
↑ Contact open				↑ Contact closed			

Table 4.1 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	RAPID
			DATA INPUT				FEED	TRAVERSE
#1301	OVC	ROV2	ROV1	OV16	OV8	OV4	OV2	OV1
	OVERRIDE	RAPID TRAVERSE			FEEDRATE OVERRIDE			
	CANCEL	RATE OVERRIDE						
#1302	-α	+α	-Z	+Z	-Y	+Y	-X	+X
	JOB PB							
#1303	SPC	SPB	SPA	JV16	JV8	JV4	JV2	JV1
	SPINDLE SPEED			MANUAL FEEDRATE OVERRIDE				
	OVERRIDE							
#1304	DRS	MP4	MP2	MP1	Hα	HZ	HY	HX
	DISPLAY	HANDLE PULSE MULTIPLY			HANDLE AXIS			
	RESET							
#1305	AFL	MLK	OPT	DRN	BDT	DLK		SBK
	M-	MACHINE	OPTIONAL	DRY	BLOCK	DISPLAY		SINGLE
	FUNCTION	LOCK	STOP	RUN	DELETE	LOCK		BLOCK
	LOCK							
#1306	SRN	F1	RET	TLM1	ZRN	EDTLK	SP	ST
	PROGRAM	F1-	RETRACT	TOOL	ZERO	EDIT	FEED	CYCLE
	RESTART	DIGIT	LENGTH	LENGTH	RETURN	LOCK	HOLD	START
#1307	PINT	ANG	ABS	M18	M1α	M1Z	M1Y	M1X
	PROGRAM	Z-AXIS	MANUAL	MIRROR IMAGE				
	INTER-	LOCK	ABSOLUTE					
#1308	9BDT	8BDT	7BDT	6BDT	5BDT	4BDT	3BDT	2BD
	SPECIAL BLOCK DELETE							
#1309	ZRN2	5NG	4NG	EWS	PST		WSH	
	MANUAL	5TH	4TH	EXTERNAL	TACTILE			
	2ND	AXIS	AXIS	WRITING	SENSOR			
	REFERENCE	NEGLECT	NEGLECT	SIGNAL	SIGNAL			
	POINT							
	RETURN							
#1310	DEVSLTI			2H8	2Hα	2HZ	2HY	2HX
	TAPE	SECOND HANDLE AXIS SELECT						
	OPERATION							
	DEVICE							
	DESIGNATION							

Table 4.1 List of Standard Input Signals (Cont'd)

	D ₇	D ₆	D ₅	D ₄	D ₃	D ₂	D ₁	D ₀
#1311				3H β	3H α	3HZ	3HY	3HX
	THIRD HANDLE AXIS SELECT							
#1312	PLYBK	TCNT	SENSON	TLCTN	TLSKP	TLRST	EXC1	ESCO
	PLAYBACK CUTTING TIME COUNT SWITCH DISABLE SIGNAL		TACTILE TOOL LIFE CONTROL ENABLE SIGNAL		TOOL LIFE CONTROL		EXT STROKE CHECK SELECTION	
#1313	RWDH	FSCH	FSMEM	FSCM	SLPC		*- L β	*+ L β
	HIGH-SPEED FEEDRATE, FEEDRATE, FEEDRATE, SPINDLE REWIND & SPINDLE SPINDLE SPINDLE LOOP AUTO START SPEED SPEED SPEED COMMAND MEMORY STORAGE EDITING INPUT CHANGE MODE						OVERTRAVEL	
#1314	SPE	SPD	ROV4		ECLM		*-ED β	*+ED β
	SPINDLE OVERRIDE		RAPID TRAVERSE RATE OVERRIDE		EXT PROGRAM CLEAR		EXTERNAL DECELERATION	
#1315	MANINT			H β	NTCRQ	TCFIN	- β	+ β
	MACHINING INTERRUPTION POINT RETURN SIGNAL		HANDLE AXIS SELECT CHANGE REQUEST		NEW TOOL END OF TOOL CHANGE REQUEST		MANUAL FEED	
#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	GST
	SPINDLE REVERSE	S FIN		SPINDLE COINCI- DENCE	SPINDLE INDEXING	GEAR SELECTION		GEAR SHIFT
#1318	ERR2	ERR1	ERR0	SENS(G)	SENS(T)	EXOUT	EXVER	EXIN
	DEC TO STOP	IMMEDI- ATE STOP	SINGLE GAP BLOCK STOP	TOOL ELIMINATION BREAK		EXTERNAL OUTPUT	EXTERNAL COLLATION	EXTERNAL INPUT
#1319	*-L α	*+L α	*-LZ	*+LZ	*-LY	*+LY	*-LX	*+LX
	OVERTRAVEL							
#1320	HOF5			*IT β	*IT α	*ITZ	*ITY	*ITX
	AUTO MODE HANDLE 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			

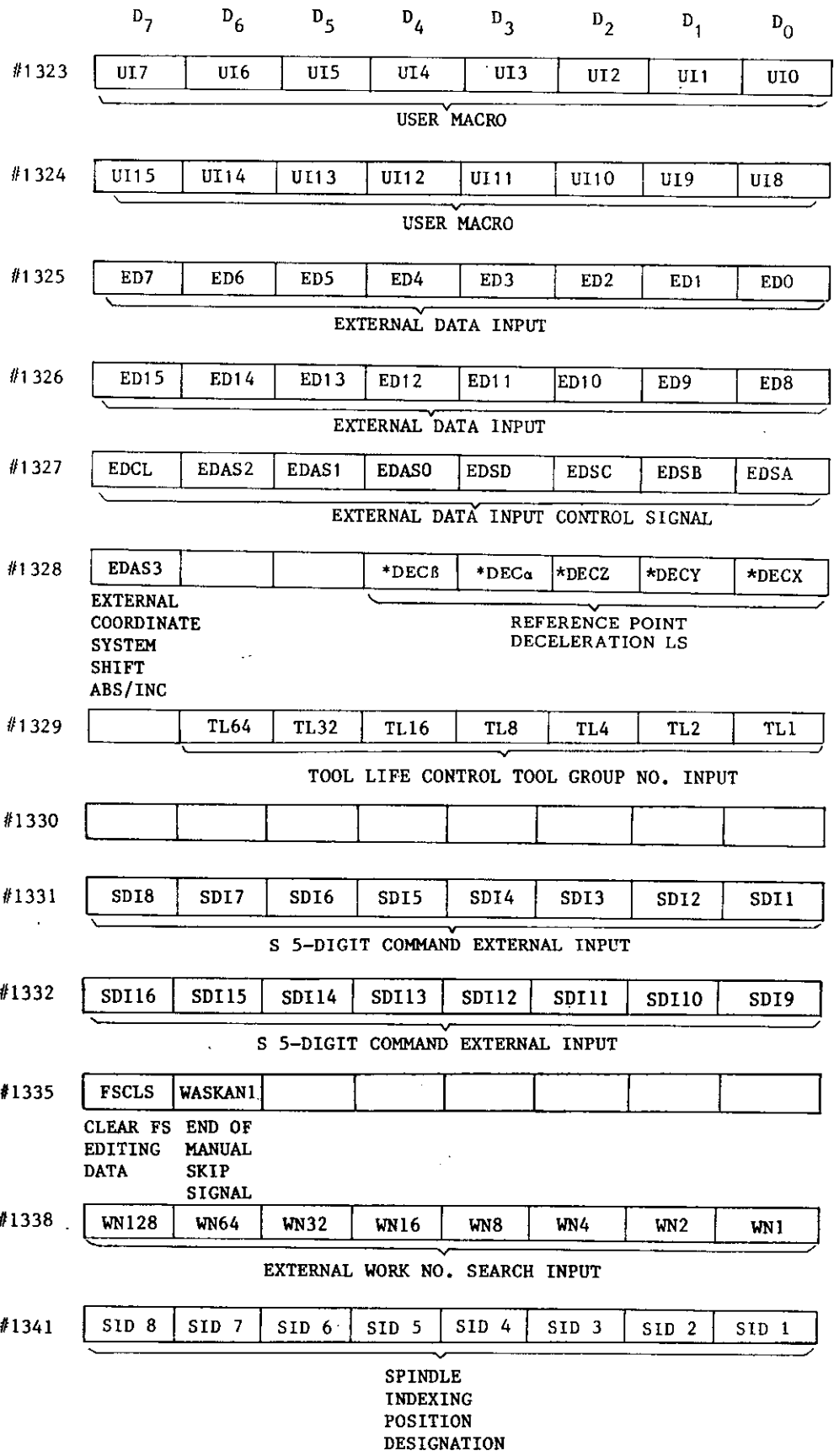


Table 4.1 List of Standard Input Signals (Cont'd)

	D ₇	D ₆	D ₅	D ₄	D ₃	D ₂	D ₁	D ₀
#1342					SID12	SID11	SID10	SID9
					SPINDLE INDEXING POSITION DESIGNATION			
#1343					SPCUT	SIDX1	SIDXINC	SIDX
					STOP SPINDLE INDEXING	RESTART SPINDLE INDEXING	SPINDLE INDEXING POSITION INCREMENTAL DESIGNATION	SPINDLE INDEXING REQUEST

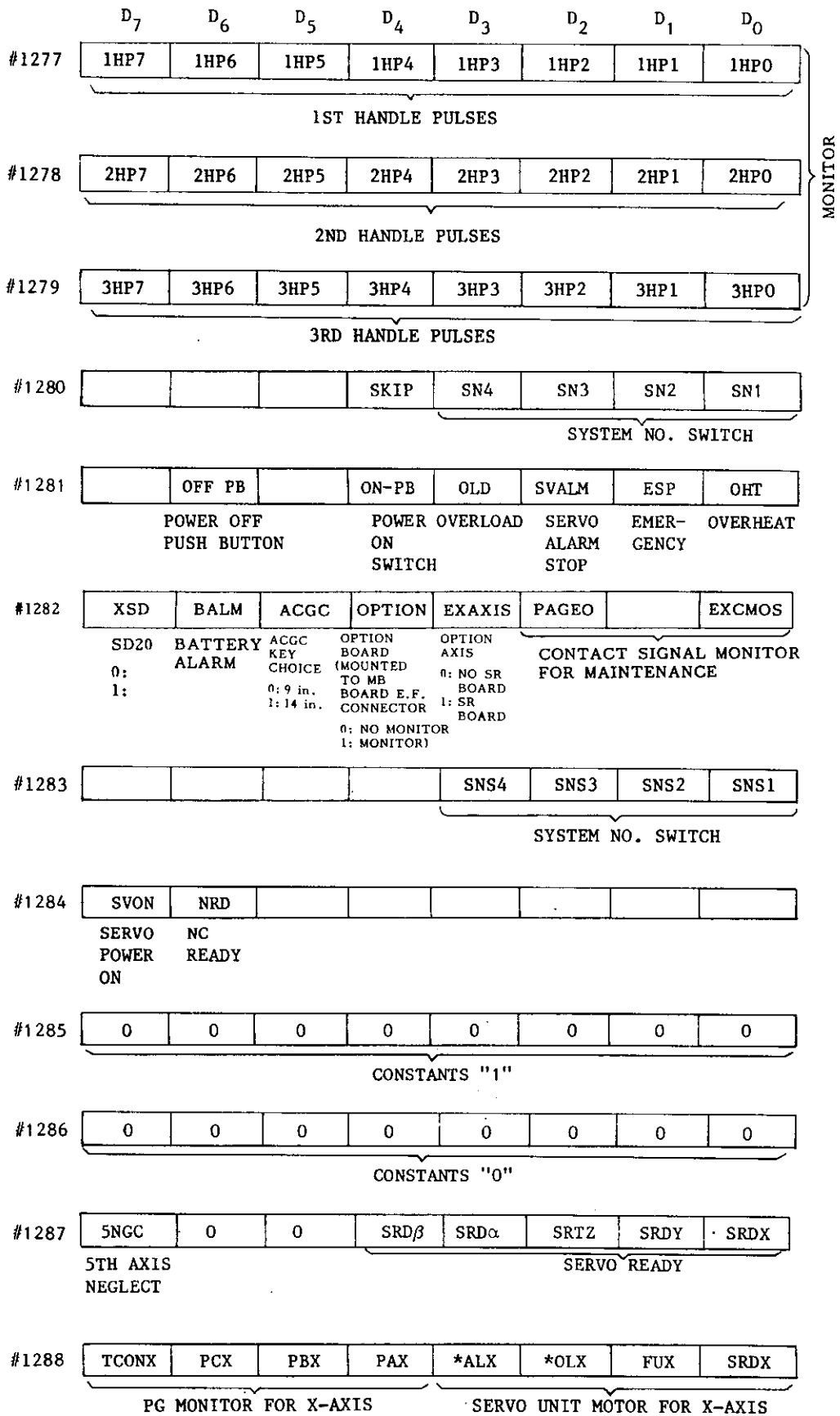
Table 4.2 List of Standard Output Signals

	D ₇	D ₆	D ₅	D ₄	D ₃	D ₂	D ₁	D ₀
#1200	M30	M02	M01	M00	DEN	OP	SPL	STL
					POSITION- ING COM- PLETED	FEED- ING	TEMPO- RARY STOP	CYCLE START
#1201	2ZP α	2ZPZ	2ZPY	2ZPX	1ZP α	1ZPZ	1ZPY	1ZPX
	SECOND REFERENCE POINT LAMP				FIRST REFERENCE POINT LAMP			
#1202	4ZP α	4ZPZ	4ZPY	4ZPX	3ZP α	3ZPZ	3ZPY	3ZPX
	FOURTH REFERENCE POINT LAMP				THIRD REFERENCE POINT LAMP			
#1203					4ZP β	3ZP β	2ZP β	1ZP β
					FIFTH REFERENCE POINT LAMP			
#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
							RAPID TRAVERSE OUTPUT	SPINDLE REVERSE OUTPUT
#1207					SIDXA	SIDXO	SLPS	G93M
					END OF SPINDLE INDEXING OUTPUT	DURING SPINDLE INDEXING OUTPUT	SPINDLE LOOP STATUS OUTPUT	G93 MODE OUTPUT

	D ₇	D ₆	D ₅	D ₄	D ₃	D ₂	D ₁	D ₀
#1208	WSFTER	WSFTK						
	END OF WORK COORDINATE SYSTEM SCREEN SETTING							
#1209	MANINTK					FSCLRK	FSCE	FSMD
	END OF MACHINING INTERRUPTION POINT RETURN SIGNAL					END OF FS DATA CLEAR	END OF FS MEMORY CHANGE	FS UNDER FS EDITING MODE
#1210							SEGOUTS	SEGDNC
							DURING DNC HIGH-SPEED CUTTING PULSE DISTRIBUTION	DURING DNC HIGH-SPEED CUTTING COMMUNICATION
#1211				MIMGMB	MIMGMa	MIMGMZ	MIMGMY	MIMGMX
	DURING MIRROR IMAGE MODE							
#1212								
#1213								
#1214								
#1215	G84S							
	G84 EXECUTION							
#1216	T8/T28	T7/T24	T6/T22	T5/T21	T4/T18	T3/T14	T2/T12	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							
#1218	TAP	MO4S	TLMO	G8OS	EREND	ESEND	RST	AL
	TAPPING	SPINDLE LENGTH MEASURE- MENT	TOOL CYCLE	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- FION
	FOR CANNED CYCLE							

Table 4.2 List of Standard Output Signals (Cont'd)

	D ₇	D ₆	D ₅	D ₄	D ₃	D ₂	D ₁	D ₀
#1222	M8	M7	M6	M5	M4	M3	M2	M1
	M-FUNCTION BINARY/BCD OUTPUT							
#1223	OS	EDTS	IER	4NGC	AUTO	MAN	RDY	RWD
	ORIENTA- TION	EDITING	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
#1235	S48	S44	S42	S41	S38	S34	S32	S31
	S FUNCTION BCD OUTPUT							
#1236	U7	U6	U5	U4	U3	U2	U1	U0
	USER MACRO							
#1237	U15	U14	U13	U12	U11	U10	U9	U8
	USER MACRO							
#1238								
#1239								



MONITOR

Table 4.2 List of Standard Output Signals (Cont'd)

	D ₇	D ₆	D ₅	D ₄	D ₃	D ₂	D ₁	D ₀	
#1289	TGONY	PCY	PBY	PAY	*ALY	*OLY	FUY	SRDY	
	PG MONITOR FOR Y-AXIS				SERVO UNIT FOR Y-AXIS				
#1290	TGONZ	PCZ	PBZ	PAZ	*ALZ	*OLZ	FUZ	SRDZ	
	PG MONITOR FOR Z-AXIS				SERVO UNIT MOTOR FOR Z-AXIS				
#1291	TGON4	PC4	PB4	PA4	*AL4	*OL4	FU4	SRD4	
	PG MONITOR FOR 4TH AXIS				SERVO UNIT MOTOR FOR 4TH AXIS				
#1292	TGON5	PC5	PB5	PA5	*AL5	*OL5	FU5	SRD5	
	PG MONITOR FOR 5TH AXIS				SERVO UNIT MOTOR FOR 5TH AXIS				
#1293				ZNCC	ABSC	EDTLKC			SETTING MONITOR
				Z-AXIS MANUAL EDIT NEG- ABSOLUTE LOCK LECT					
#1294	AFLC	MLKC	OPTC	DRNC	BTDC	DLKC	STLKC	SBKC	
	AUX FUNC- MACHINE TION LOCK LOCK		OPTION- NAL STOP	DRY RUN	OPTIONAL BLOCK SKIP	DISPLAY LOCK	START LOCK	SINGLE BLOCK	
#1295			PLBKC	M1BC	M1oC	M1ZC	M1YC	M1XC	
	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 5.1

Alarm No.	Spindle Operation	Type of Alarm
000 to 099	Stop at block end	Tape format error alarm
100 to 199	Stop at block end	Macro, operation, external input/ output error, sequence error (1) (2) (3)
200 to 299	Decelerate to stop	Overtravel, reference point return, positioning, machine ready
300 to 399	Decelerate to stop	Servo, emergency stop, overload FG, RPG

Table 5.2 List of Alarm Codes

Code	Causes	Code	Causes
000		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.	014	PROG ERROR ("-", "0")
			SIGN "-", "0" NOT CORRECTLY USED
005	ZR UNREADY (5) REFERENCE POINT RETURN NOT COMPLETED.	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	
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
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 = 0) CIRCLE WITH RADIUS 0 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 ACCOMODATED 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		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.			

Table 5.2 List of Alarm Codes (Cont'd)

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	
048	PROG ERROR (G41/42) INTERSECTION POINT NOT OBTAINED BY INTERSECTION COMPUTATION.	058	MIRROR IMAGE (G28) G28 COMMANDED DURING MIRROR IMAGE.
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
059	ZR UNREADY	069	
	REFERENCE POINT RETURN NOT COMPLETED ON THE AXIS WHICH HAS G29 COMMAND OR G30 COMMAND.		
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)	076	RS232C ERROR (SIGNAL LEVEL)
	CYCLE-STARTED WITHOUT RESETTING AFTER EDITING.		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			
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.			

Table 5.2 List of Alarm Codes (Cont'd)

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	
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
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 ([] ⁵ 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 ([] UNMACH) NO. OF LEFT BRACKETS AND RIGHT BRACKETS NOT THE SAME.
106	MACRO ERROR TOO MANY G CODES COMMANDED FOR CANCELLING G67.	116	MACRO ERROR (DO-END NO.) CONDITION $1 \leq m \leq 3$ NOT ESTABLISHED IN DOm.
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.			

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
117	MACRO ERROR (10)	127	LIFE CTRL ERROR (T5 & T9999)
118	MACRO ERROR (GOTO N) CONDITION $0 \leq n \leq 9999$ NOT ESTABLISHED OR NO SEQUENCE NO. IN GO TO n.		ENTERED COMMAND FOR T5 DIGITS WITHOUT REGISTERING THE GROUP OR THERE IS AN ERROR IN THE T9999L $\Delta\Delta\Delta$ COMMAND.
119	MACRO ERROR (12)	128	LIFE CTRL ERROR (T5 & H9999)
120	PRTN ERROR (NOT FOUND) SEQUENCE NO. NOT FOUND IN PART PROGRAM.		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.
121	PRTN ERROR (G92) G92 COMMANDED THROUGH MDI OPERATION DURING PROGRAM RESTART.	129	PROG ERROR (G54 TO G59 J)
122	PRTN ERROR (G54-G59) G54 TO G59 COMMANDED THROUGH MDI OPERATION DURING PROGRAM RESTART.		SPECIFIED AN EXCESSIVELY LARGE J IN THE WORK COORDINATE SYSTEM OR THE MODE WAS OTHER THAN G00/G01.
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	
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
134		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.
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.			

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
153		163	DNC MODULE 71051/8530 ERROR 8251 ERROR
154		164	DNC MODULE CHECK SUM ERROR CHECK SUM ERROR
155		165	DNC MODULE COMMAND ERROR COMMAND ERROR
156		166	
157		167	DNC MODULE ROM CHECK ERROR COMMUNICATION MODULE CHECK ERROR
158		168	DNC HISPEED MODE ERROR HIGH SPEED CUTTING MODE ERROR
159		169	DNC I/F SYNC ERROR DNC CPU ERROR
160	DNC MODULE TIME OUT DNC TIME OUT ERROR.	170	MEM ERROR (OFS) TOOL OFFSET TOTAL CHECK ERROR
161	DNC MODULE DR-LINE ERROR DR LINE ERROR	171	
162	DNC MODULE PACKET LENGTH ERROR PACKET LENGTH ERROR	172	MEM ERROR (SET) SETTING AREA TOTAL CHECK ERROR.
<p>Notes :</p> <ol style="list-style-type: none"> 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
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	HDLC SYSTEM SOFT ERROR SYSTEM ERROR WHEN ACGC CONNECTED.
181	SEQ ERROR 3 SEQUENCER ERROR (3) IMMEDIATE STOP	191	HDLC SOFT COMMAND ERROR SYSTEM ERROR WHEN ACGC CONNECTED.
182	SEQ ERROR 2 SEQUENCER ERROR (2) DECELATION STOP	192	HDLC 8530 SEND ERROR SYSTEM ERROR WHEN ACGC CONNECTED.
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.			

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
193	HDLC 8530 RECEIVE ERROR SYSTEM ERROR WHEN ACGC CONNECTED.	203	OT (Z) OVERTRAVEL Z
194	HDLC NAK ERROR (10 TIMES) SYSTEM ERROR WHEN ACGC CONNECTED.	204	OT (4) OVERTRAVEL 4
195	HDLC CMOS FILE ERROR SYSTEM ERROR WHEN ACGC CONNECTED.	205	OT (5) OVERTRAVEL 5
196	HDLC I/O DATA ERROR SYSTEM ERROR WHEN ACGC CONNECTED.	206	
197	HDLC NC REQUEST ERROR SYSTEM ERROR WHEN ACGC CONNECTED.	207	
198	HDLC OPERATION ERROR SYSTEM ERROR WHEN ACGC CONNECTED.	208	
199	AMGC COMMUNICATION SYSTEM ERROR AMGC COMMUNICATION SYSTEM ERROR	209	
200	ACGC2 COMMUNICATION SYSTEM ERROR ACGC120 COMMUNICATION SYSTEM ERROR	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.
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
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	S-OT1 (5) STORED STROKE LIMIT 1ST AREA 5.	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. VIRTUOL C PHASE TYPE LATCH UNFINISHED.
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.			

Table 5.2 List of Alarm Codes (Cont'd)

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	ZR ERROR-AREA (5) ----- REFERENCE POINT RETURN ERROR 5.	245	ZR ERROR-POS (5) ----- REFERENCE POINT RETURN POSITION ERROR 5.
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.
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
252	S-OT3-5 (Y) STORED STROKE LIMIT 3RD, 4TH, 5TH AREA OUTSIDE PROHIBITED Y.	262	ZR ERROR PRM (Y) RETRACTION BY HOME POSITION (ER PULSE AT C PHASE LATCH > #6305)
253	S-OT3-5 (Z) STORED STROKE LIMIT 3RD, 4TH, 5TH AREA OUTSIDE PROHIBITED Z.	263	ZR ERROR PRM (Z) RETRACTION BY HOME POSITION (ER PULSE AT C PHASE LATCH > #6306)
254		264	ZR ERROR PRM (4) RETRACTION BY HOME POSITION (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	ZR ERROR PRM (5) RETRACTION BY HOME POSITION (ER PULSE AT C PHASE LATCH > #6308)
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	ZR DECLS ERROR (5) LS ERROR (5) DECELERATION (DECLS OF 5 AXIS STARTS CHAT- TERING ON THE DOG)	269	
260		270	
261	ZR ERROR PRM (X) RETRACTION BY HOME POSITION (ER PULSE AT C PHASE LATCH > #6304)	271	P-SET ERROR (X) P SET ERROR X
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.			

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
272	P-SET ERROR (Y) P SET ERROR Y	282	
273	P-SET ERROR (Z) P SET ERROR Z	283	
274	P-SET ERROR (4) P SET ERROR 4	284	
275	P-SET ERROR (5) PROGRAM SET ERROR 5	285	
276		286	
277		287	
278		288	
279		289	
280	MACH UNREADY MACH RDY OFF	290	
281		291	
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
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
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.			

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
312	Y-AXIS SERVO UNREADY	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	323	
314	4TH AXIS SERVO UNREADY	324	
315	5TH AXIS SERVO UNREADY	325	SERVO CPU ERROR SERVO CPU IS IN ABNORMAL STATE.
316		326	SKIP MACRO (G, M, T) CALLING OF G, M, T MACRO OF THE MAIN PROGRAM WAS SKIPPED (CHECK MODE).
317			
318		327	UNFINISHED PROG GEN MEMORY LENGTH, FOR WHICH MEMORY GENERATION WAS NOT PERFORMED, HAS BEEN CHANGED BUT NO GENERATION HAS BEEN MADE.
319			
320	NC UNREADY NC UNREADY P SET UNREADY	328	UNSUITABLE MM21 MEMORY LENGTH SET BY PARAMETER DOES NOT MATCH THE MEMORY BOARD.
321		329	PC CPU ERROR PC CPU IS IN ABNORMAL STATE.
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
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	FUSE (5) FUSE BLOWN 5	345	SERVO ERROR (5) SERVO ERROR 5
336		346	SERVO ERROR (S) SERVO ERROR (SPINDLE)
337		347	
338		348	
339		349	
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.			

Table 5.2 List of Alarm Codes (Cont'd)

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	OL (5) OVERLOAD (1) 5	365	PG ERROR (5) PG ERROR 5
356		366	
357	OL (OTHERS) OVERLOAD (2)	367	
358		368	
359		369	
<p>Notes:</p> <ol style="list-style-type: none"> 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
370		380	
371	SV-FG SYNC ERROR SERVO CPU IS NOT SYNCHRONIZED WITH FG CPU.	381	AXIS ERROR (X) AXIS CPU ERROR
372	FG NOT READY READY FROM FG CPU HAS NOT BEEN RECEIVED.	382	AXIS ERROR (Y) AXIS CPU ERROR
373	SR-20 UNSUITABLE OPTIONAL AXIS SETTING DOES NOT MATCH THE TYPE OF SR-20.	383	AXIS ERROR (Z) AXIS CPU ERROR
374		384	AXIS ERROR (4) AXIS CPU ERROR
375		385	AXIS ERROR (5) AXIS CPU ERROR
376		386	
377		387	
378		388	
379		389	

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.

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
390	SERVO ALARM (SERVOPACK) (SR20)	800	
	SERVOPACK ALARM SPINDLE PG DISCONNECTION OF 4/5 AXIS SR FAILURE		
391	SERVO ALARM (X)	801	
	X-AXIS SERVOPACK ALARM X		
392	SERVO ALARM (Y)	802	
	Y-AXIS SERVOPACK ALARM Y		
393	SERVO ALARM (Z)	803	
	Z-AXIS SERVOPACK ALARM Z		
394	SERVO ALARM (4)	804	
	4 AXIS SERVOPACK ALARM 4		
395	SERVO ALARM (5)	805	
	5 AXIS SERVOPACK ALARM 5		
396		806	
397		810	RAM ERROR
398		811	
399		812	
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
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. FOR DETAILS, REFER TO THE YASNAC MX3 MAINTENANCE MANUAL.	830	CPU ERROR
821		840	CPU ERROR
822		841	
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.			

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
842		852	
843		853	
844		854	
845		855	
846		856	
847		857	
848		858	
849		859	
850		860	
851		861	
<p>Notes:</p> <ol style="list-style-type: none"> 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
862		872	
863		873	
864		874	
865		875	
866		876	
867		877	
868		878	
869		879	
870		880	
871		881	

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.

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
882		892	
883		893	
884		894	
885		895	
886		896	
887		897	
888		898	
889		899	
890		900	
891		901	
<p>Notes :</p> <ol style="list-style-type: none"> 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
902		912	
903		913	
904		914	
905		915	
906		916	
907		917	
908		918	
909		919	
910	TAPE MEM ERROR	920	TAPE ERROR
911		921	
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.			

Table 5.2 List of Alarm Codes (Cont'd)

Code	Causes	Code	Causes
922		932	
923		933	
924		934	
925		935	
926		936	
927		937	
928		938	
929		939	
930			
931			
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.			

APPENDIX – 6 LIST OF ADDRESS CHARACTERS

Table 6.1 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 6.2 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, User macro operator	
Del	DEL	Disregarded (Including All Mark)	
.	.	Decimal point	
Parameter setting	#	Sharp (Variable)	EIA: Special code
*	*	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 6.3 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											
								1											
								2											
								3											
								4											
								5											
								6											
								7											
								8											
								9											
								a	A										
								b	B										
								c	C										
								d	D										
								e	E										
								f	F										
								g	G										
								h	H										
								i	I										
								j	J										
								k	K										
								l	L										
								m	M										
								n	N										
								o	O										
								p	P										
								q	Q										
								r	R										
								s	S										
								t	T										
								u	U										
								v	V										
								w	W										
								x	X										
								y	Y										
								z	Z										
								Blank	NUL										
								BS											
								Tab	HT										
								CR	LF/NL										
								-	CR										
								SP											
								ER	%										
								UC	-										
								LC	-										
								-	(
								-)										
								+											
								-											
								o	:										
								/											
								Del	DEL										
								All Mark											
								See Note 2.	#										
								*											
								=											
								[
]											
								\$											
								@											
								?											
								.											

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 6.4 Tape Format

No.	Address		Metric output		Inch output		B: Basic O: Optional
			Metric input	Inch input	Metric input	Inch input	
1	Program No.		O4		O4		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/min		F50	F31	F50	F41	B
6	Feed/min 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 6.5 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 24000 mm/min	0.1 to 944.8 in./min	1 to 60960 mm/min	0.1 to 2400.0 in./min
6	Feed per minute 1/10		0.1 to 24000.0 mm/min	0.01 to 944.88 in./min	0.1 to 60960.0 mm/min	0.01 to 2400.00 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 6.6 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 24,000 mm/min		(0.75 to) 0.1 to 2400.0 in./min	
Manual jog speed					
Seed at Fo					
Max. feedrate		1 to 24,000 mm/min		0.1 to 2400.0 in./min	
Dry run speed					
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.
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 6.7 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	B
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	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
G36		Automatic centering	O
G37	*	Automatic centering	O
G38		Z-axis reference surface offset	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	15	Scaling OFF	O
G51		Scaling ON	O
G52	12	Return to base coordinate system	O
G53	*	Temporary shift to machine coord- inate system	O
G54		Shift to work coordinate system 1	O
G55	12	Shift to work coordinate system 2	O
G56		Shift to work coordinate system 3	O

G Code	Group	Function	B: Basic O: Optional
G57		Shift to work coordinate system 4	O
G58	12	Shift to work coordinate system 5	O
G59		Shift to work coordinate system 6	O
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	Rotation of coordinate ON	O
G69		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
G88		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	B
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.

YASNAC MX3

CNC SYSTEM FOR MACHINING CENTERS

TOKYO OFFICE Ohtemachi Bldg, 1-6-1 Ohtemachi, Chiyoda-ku, Tokyo, 100 Japan
Phone (03) 3284-9111, -9145 Telex YASKAWA J33530 Fax (03) 3284-9034

SEOUL OFFICE Seoul Center Bldg, 91-1, So Kong Dong, Chung-ku, Seoul, Korea
Phone (02) 776-7844 Fax (02) 753-2639

TAIPEI OFFICE Union Commercial Bldg, 14F, 137, Nanking East Road, Sec 2, Taipei, Taiwan
Phone (02) 507-7065, -7732 Fax (02) 506-3837

YASKAWA ELECTRIC AMERICA, INC. : SUBSIDIARY
Chicago-Technical Center
3160 MacArthur Blvd. Northbrook, Illinois 60062-1917, U.S.A
Phone (708) 291-0411 Fax (708) 291-1028

Los Angeles Office
7341 Lincoln Way, Garden Grove, California 92641, U.S.A.
Phone (714) 894-5911 Telex (230) 678396 YASKAWAUS TSTN Fax (714) 894-3258

New Jersey Office 30 Two Bridges Road, Fairfield, New Jersey 07006, U.S.A.
Phone (201) 575-5940 Fax (201) 575-5947

YASKAWA ELECTRIC EUROPE GmbH : SUBSIDIARY
Am Kronberger Hang 2, 65824 Schwalbach, Germany
Phone (49) 6196-569-300 Fax (49) 6196-888 301

YASKAWA ELECTRIC (SINGAPORE) PTE. LTD.
CPF Bldg, 79 Robinson Road No. 13-05, Singapore 0106
Phone 2217530 Telex (87) 24890 YASKAWA RS Fax (65) 224-5854



YASKAWA ELECTRIC CORPORATION

YASKAWA

TOE-C843-9.30E

© Printed in Japan August 1994 86-9
586-175, 582-126