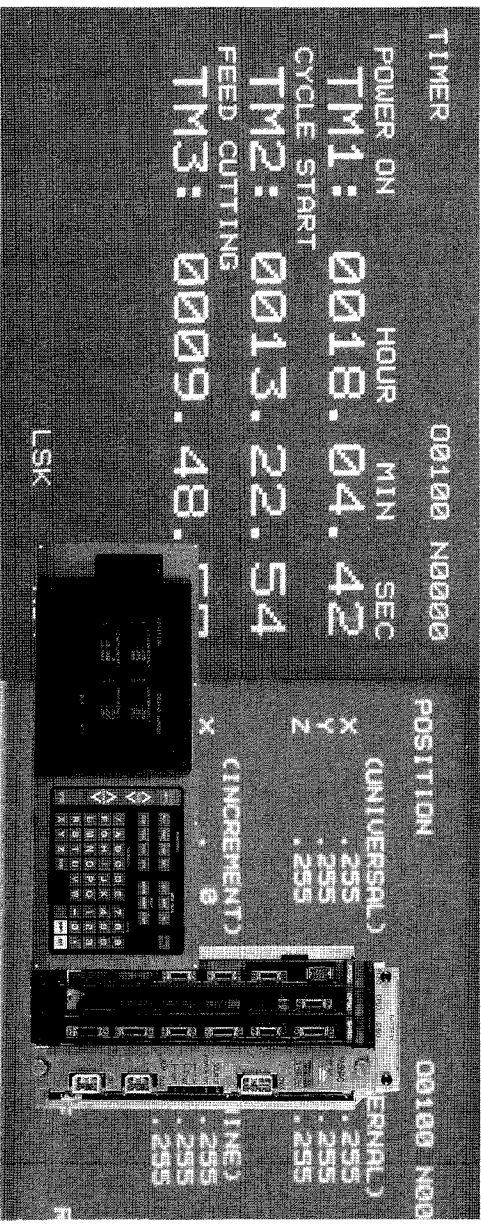


# YASNAC J50M DESCRIPTIVE INFORMATION

CNC SYSTEM FOR MACHINING CENTERS



YASKAWA

SIE-C843-12.30

## 1 INTRODUCTION

The YASNAC J50M is a high-performance CNC for the simultaneous control of 2 or 3 axes of a driven machine, with emphasis placed on high-speed machining, and programming capability.

### FEATURES

1. Ultra-high-speed Performance

"High-speed, computing system" is achieved by installing a 32-bit microprocessor in the YASNAC J50M.

2. Significant Downsizing (Miniaturized)

YASNAC J50M is significantly downsized because it has surface mounted devices and customized gate arrays.

This manual explains both basic and optional features of YASNAC J50M as well as the servo system.

You can determine your own hardware requirements after carefully reading this manual.

<p>This manual is subject to change without notification due to product improvements, model changes, etc.</p>
---

## CONTENTS

<b>1 INTRODUCTION</b>	<b>1</b>	2.35 SUBROUTINE PROGRAM (M98, M99)	5
<b>2 BASIC FEATURES</b>	<b>1</b>	2.36 PARAMETER SETTING	6
2.1 CONTROLLED AXES	1	2.37 SETTING FUNCTION	6
2.2 SIMULTANEOUS CONTROLLABLE AXES	1	2.38 INTERNAL DATA TAPE INPUT	6
2.3 LEAST INPUT INCREMENT (MINIMUM INPUT UNIT)	1	2.39 OPERATION TIME DISPLAY	6
2.4 LEAST OUTPUT INCREMENT (MINIMUM OUTPUT UNIT)	1	2.40 ADDRESS SEARCH	6
2.5 MAX. PROGRAMMABLE DIMENSIONS	1	2.41 PROGRAM NUMBER	6
2.6 NC TAPE	1	2.42 LABEL SKIP	6
2.7 TAPE CODE	1	2.43 CONTROL IN/OUT	6
2.8 EIA/ISO AUTO-RECOGNITION	1	2.44 TV CHECK	6
2.9 TAPE FORMAT	1	2.45 SEQUENCE NUMBER BREAK POINT	6
2.10 DECIMAL POINT INPUT	1	2.46 SINGLE BLOCK	6
2.11 BUFFER REGISTER	2	2.47 OPTIONAL STOP	7
2.12 RAPID TRAVERSE RATE	2	2.48 OPTIONAL BLOCK SKIP	7
2.13 FEEDRATE RANGE	2	2.49 DRY RUN	7
2.14 AUTOMATIC ACCELERATION / DECELERATION	2	2.50 MACHINE LOCK	7
2.15 FEED FUNCTION (F-FUNCTION)	2	2.51 DISPLAY LOCK	7
2.16 FEEDRATE OVERRIDE AND FEEDRATE OVERRIDE CANCEL	2	2.52 Z-AXIS COMMAND NEGLECT	7
2.17 PREPARATORY FUNCTIONS (G-FUNCTION)	2	2.53 AUXILIARY FUNCTION LOCK	7
2.18 ABSOLUTE/INCREMENTAL PROGRAMMING (G90/G91)	2	2.54 MANUAL ABSOLUTE ON/OFF	7
2.19 PROGRAMMING OF ABSOLUTE ZERO POINT (G92)	2	2.55 MIRROR IMAGE	7
2.20 POSITIONING (G00, G06)	3	2.56 INTERNAL TOGGLE SWITCHES	7
2.21 LINEAR INTERPOLATION (G01)	3	2.57 ORIGIN KEY	7
2.22 CIRCULAR INTERPOLATION (G02, G03)	3	2.58 INTERLOCK	7
2.23 DWELL (G04)	3	2.59 START LOCK AND EDIT LOCK	7
2.24 EXACT STOP CHECK (G09, G61, G64)	3	2.60 AUTOMATIC COORDINATE SYSTEM SETTING	7
2.25 MISCELLANEOUS FUNCTION (M-FUNCTION)	4	2.61 FEED HOLD	8
2.26 SPINDLE-SPEED FUNCTION (S-FUNCTION)	4	2.62 EMERGENCY STOP	8
2.27 TOOL FUNCTION (T-FUNCTION)	4	2.63 OVERTRAVEL	8
2.28 TOOL LENGTH COMPENSATION (G43, G44, G49)	4	2.64 REMOTE RESET	8
2.29 TOOL POSITION OFFSET (G45 THROUGH G48)	4	2.65 REMOTE POWER ON/OFF	8
2.30 OFFSET MEMORY	5	2.66 MACHINE READY INPUT SIGNAL	8
2.31 TOOL OFFSET VALUE	5	2.67 NC READY OUTPUT SIGNAL	8
2.32 BACKLASH COMPENSATION	5	2.68 NC ALARM OUTPUT SIGNAL	8
2.33 MANUAL FEED	5	2.69 NC RESET OUTPUT SIGNAL	8
2.34 PROGRAM STORAGE AND EDITING	5	2.70 RS-232C INTERFACE	8
		2.71 ON-LINE DIAGNOSTICS	8
		2.72 POSITION DETECTOR INTERFACE	8
		2.73 INPUT/OUTPUT CONNECTORS	9
		2.74 POWER INPUT A	9
		2.75 AMBIENT CONDITIONS	9
		2.76 PAINT COLOR AND DIMENSIONS	9

## CONTENTS (Cont'd)

### 3 BASIC OPTIONS 9

- 3.1 AC SERVO CONTROL UNITS 9
- 3.2 AC SERVOMOTORS 9

### 4 OPTIONS 10

- 4.1 NC OPERATOR'S STATION 10
- 4.2 TAPE READER 10
- 4.3 TAPE READER WITH REELS 10
- 4.4 F1-DIGIT COMMAND 10
  
- 4.5 S5-DIGIT PROGRAMMING WITH 12-BIT OUTPUT 10
- 4.6 T4-DIGIT PROGRAMMING 10
- 4.7 ADDITIONAL OFFSET MEMORY 10
- 4.8 ADDITIONAL PART PROGRAM STORAGE 10
- 4.9 ADDITIONAL PROGRAM NUMBER REGISTRATION 10
  
- 4.10 4TH AXIS CONTROL 10
- 4.11 MANUAL PULSE GENERATOR FOR ONE AXIS AT A TIME 11
- 4.12 REFERENCE POINT RETURN (G27, G28, G29) 11
- 4.13 2ND, 3RD, AND 4TH REFERENCE POINT RETURN 11
- 4.14 EXTERNAL DECELERATION 11
  
- 4.15 TOOL LENGTH MEASUREMENT 11
- 4.16 OPTIONAL BLOCK SKIP B 12
- 4.17 2ND AUXILIARY FUNCTION (B-FUNCTION) 12
- 4.18 JOG FEEDRATE OVERRIDE 12
- 4.19 PROGRAM COPY 12
  
- 4.20 HELICAL INTERPOLATION (G02, G03) 12
- 4.21 CIRCLE CUTTING B (G12, G13) 12
- 4.22 INCH/METRIC DESIGNATION BY G CODE 13
- 4.23 UNIDIRECTIONAL APPROACH (G60) 13
- 4.24 WORK COORDINATE SYSTEM SETTING A (G52 TO G59) 14
  
- 4.25 WORK COORDINATE SYSTEM SETTING B (G54J TO G59J) 14
- 4.26 TOOL RADIUS COMPENSATION C (G40 TO G42) 14
- 4.27 OUTPUT FOR EXTERNAL MOTION (G80, G81) 15
- 4.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89) 15
- 4.29 HOLE PATTERN CYCLES (G70, G71, G72) 22
  
- 4.30 SCALING FUNCTION 22
- 4.31 MACRO PROGRAM (G65, G66, G67) 22
- 4.32 EXTERNAL DATA INPUT 23
- 4.33 SKIP FUNCTION (G31) 23
- 4.34 STORED STROKE LIMIT (G22, G23) 23
  
- 4.35 STORED LEADSCREW ERROR COMPENSATION 23
- 4.36 USER MESSAGE DISPLAY 24
- 4.37 PROGRAM RESTART 24
- 4.38 PROGRAM INTERRUPTION (M90, M91) 24
- 4.39 PLAYBACK FUNCTION 24
  
- 4.40 EXTERNAL INPUT, COLLATION, AND OUTPUT 24
- 4.41 TOOL LIFE CONTROL (G122, G123) 24
- 4.42 COORDINATE ROTATION 24
- 4.43 LOCAL COORDINATE SYSTEM SETTING 25
- 4.44 AUTOMATIC OPERATION MODE HANDLE OFFSET 25

### 5 BUILT-IN TYPE PROGRAMMABLE CONTROLLER (PC) 25

- APPENDIX 1 LIST OF DATA 26
- APPENDIX 2 DIMENSIONS in mm (inch) 31

# INDEX

Subject	Chapter	Section	Page
A ABSOLUTE/INCREMENTAL PROGRAMMING (G90/G91)	2	2.18	2
AC SERVO CONTROL UNITS	3	3.1	9
AC SERVOMOTORS	3	3.2	9
ADDITIONAL OFFSET MEMORY	4	4.7	10
ADDITIONAL PART PROGRAM STORAGE	4	4.8	10
ADDITIONAL PROGRAM NUMBER REGISTRATION	4	4.9	10
ADDRESS SEARCH	2	2.40	6
AMBIENT CONDITIONS	2	2.75	9
AUTOMATIC ACCELERATION/DECELERATION	2	2.14	2
AUTOMATIC COORDINATE SYSTEM SETTING	2	2.60	7
AUTOMATIC OPERATION MODE HANDLE OFFSET	4	4.44	25
AUXILIARY FUNCTION LOCK	2	2.53	7
B BACKLASH COMPENSATION	2	2.32	5
BASIC FEATURES	2		1
BASIC OPTIONS	3		9
BUFFER REGISTER	2	2.11	2
BUILT-IN TYPE PROGRAMMABLE CONTROLLER (PC)	5		25
C CANNED CYCLES (G73, G74, G76, G77, G80 TO G89)	4	4.28	15
CIRCLE CUTTING B (G12, G13)	4	4.21	12
CIRCULAR INTERPOLATION (G02, G03)	2	2.22	3
CONTROL IN/OUT	2	2.43	6
CONTROLLED AXES	2	2.1	1
COORDINATE ROTATION	4	4.42	24
D DECIMAL POINT INPUT	2	2.10	1
DIMENSIONS in mm (inch)	APPENDIX 2		31
DISPLAY LOCK	2	2.51	7
DRY RUN	2	2.49	7
DWELL (G04)	2	2.23	3
E EIA/ISO AUTO-RECOGNITION	2	2.8	1
EMERGENCY STOP	2	2.62	8
EXACT STOP CHECK (G09, G61, G64)	2	2.24	3
EXTERNAL DATA INPUT	4	4.32	23
EXTERNAL DECELERATION	4	4.14	11
EXTERNAL INPUT, COLLATION, AND OUTPUT	4	4.40	24
F F1-DIGIT COMMAND	4	4.4	10
FEED FUNCTION (F-FUNCTION)	2	2.15	2
FEED HOLD	2	2.61	8
FEEDRATE OVERRIDE AND FEEDRATE OVERRIDE CANCEL	2	2.16	2
FEEDRATE RANGE	2	2.13	2
4TH AXIS CONTROL	4	4.10	10
H HELICAL INTERPOLATION (G02, G03)	4	4.20	12
HOLE PATTERN CYCLES (G70, G71, G72)	4	4.29	22
I INCH/METRIC DESIGNATION BY G CODE	4	4.22	13
INPUT/OUTPUT CONNECTORS	2	2.73	9
INTERLOCK	2	2.58	7
INTERNAL DATA TAPE INPUT	2	2.38	6
INTERNAL TOGGLE SWITCHES	2	2.56	7
INTRODUCTION	1		i
J JOG FEEDRATE OVERRIDE	4	4.18	12

## INDEX (Cont'd)

Subject	Chapter	Section	Page
L LABEL SKIP	2	2.42	6
LEAST INPUT INCREMENT (MINIMUM INPUT UNIT)	2	2.3	1
LEAST OUTPUT INCREMENT (MINIMUM OUTPUT UNIT)	2	2.4	1
LINEAR INTERPOLATION (G01)	2	2.21	3
LIST OF DATA	APPENDIX 1		26
LOCAL COORDINATE SYSTEM SETTING	4	4.43	25
M MACHINE LOCK	2	2.50	7
MACHINE READY INPUT SIGNAL	2	2.66	8
MACRO PROGRAM (G65, G66, G67)	4	4.31	22
MANUAL ABSOLUTE ON/OFF	2	2.54	7
MANUAL FEED	2	2.33	5
MANUAL PULSE GENERATOR FOR ONE AXIS AT A TIME	4	4.11	11
MAX. PROGRAMMABLE DIMENSIONS	2	2.5	1
MIRROR IMAGE	2	2.55	7
MISCELLANEOUS FUNCTION (M-FUNCTION)	2	2.25	4
N NC ALARM OUTPUT SIGNAL	2	2.68	8
NC OPERATOR'S STATION	4	4.1	10
NC READY OUTPUT SIGNAL	2	2.67	8
NC RESET OUTPUT SIGNAL	2	2.69	8
NC TAPE	2	2.6	1
O OFFSET MEMORY	2	2.30	5
ON-LINE DIAGNOSTICS	2	2.71	8
OPERATION TIME DISPLAY	2	2.39	6
OPTIONAL BLOCK SKIP	2	2.48	7
OPTIONAL BLOCK SKIP B	4	4.16	12
OPTIONAL STOP	2	2.47	7
OPTIONS	4		10
ORIGIN KEY	2	2.57	7
OUTPUT FOR EXTERNAL MOTION (G80, G81)	4	4.27	15
OVERTRAVEL	2	2.63	8
P PAINT COLOR AND DIMENSIONS	2	2.76	9
PARAMETER SETTING	2	2.36	6
PLAYBACK FUNCTION	4	4.39	24
POSITION DETECTOR INTERFACE	2	2.72	8
POSITIONING (G00, G06)	2	2.20	3
POWER INPUT A	2	2.74	9
PREPARATORY FUNCTIONS (G-FUNCTION)	2	2.17	2
PROGRAM COPY	4	4.19	12
PROGRAM INTERRUPTION (M90, M91)	4	4.38	24
PROGRAM NUMBER	2	2.41	6
PROGRAM RESTART	4	4.37	24
PROGRAMMING OF ABSOLUTE ZERO POINT (G92)	2	2.19	2
PROGRAM STORAGE AND EDITING	2	2.34	5
R RAPID TRAVERSE RATE	2	2.12	2
REFERENCE POINT RETURN (G27, G28, G29)	4	4.12	11
REMOTE POWER ON/OFF	2	2.65	8
REMOTE RESET	2	2.64	8
RS-232C INTERFACE	2	2.70	8
S S5-DIGIT PROGRAMMING WITH 12-BIT OUTPUT	4	4.5	10
SCALING FUNCTION	4	4.30	22
2ND AUXILIARY FUNCTION (B-FUNCTION)	4	4.17	12
2ND, 3RD, AND 4TH REFERENCE POINT RETURN	4	4.13	11
SEQUENCE NUMBER BREAK POINT	2	2.45	6

## INDEX (Cont'd)

Subject	Chapter	Section	Page
S			
SETTING FUNCTION	2	2.37	6
SIMULTANEOUS CONTROLLABLE AXES	2	2.2	1
SINGLE BLOCK	2	2.46	6
SKIP FUNCTION (G31)	4	4.33	23
SPINDLE-SPEED FUNCTION (S-FUNCTION)	2	2.26	4
START LOCK AND EDIT LOCK	2	2.59	7
STORED LEADSCREW ERROR COMPENSATION	4	4.35	23
STORED STROKE LIMIT (G22, G23)	4	4.34	23
SUBROUTINE PROGRAM (M98, M99)	2	2.35	5
T			
T4-DIGIT PROGRAMMING	4	4.6	10
TAPE CODE	2	2.7	1
TAPE FORMAT	2	2.9	1
TAPE READER	4	4.2	10
TAPE READER WITH REELS	4	4.3	10
TOOL FUNCTION (T-FUNCTION)	2	2.27	4
TOOL LENGTH COMPENSATION (G43, G44, G49)	2	2.28	4
TOOL LENGTH MEASUREMENT	4	4.15	11
TOOL LIFE CONTROL (G122, G123)	4	4.41	24
TOOL OFFSET VALUE	2	2.31	5
TOOL POSITION OFFSET (G45 THROUGH G48)	2	2.29	4
TOOL RADIUS COMPENSATION C (G40 TO G42)	4	4.26	14
TV CHECK	2	2.44	6
U			
UNDIRECTIONAL APPROACH (G60)	4	4.23	13
USER MESSAGE DISPLAY	4	4.36	24
W			
WORK COORDINATE SYSTEM SETTING A (G52 TO G59)	4	4.24	14
WORK COORDINATE SYSTEM SETTING B (G54J TO G59J)	4	4.25	14
Z			
Z-AXIS COMMAND NEGLECT	2	2.52	7

## 2 BASIC FEATURES

### 2.1 CONTROLLED AXES

3 axes (X, Y and Z)

### 2.2 SIMULTANEOUS CONTROLLABLE AXES

3 axes (positioning and linear interpolation)  
2 axes (circular interpolation)

All axes (manual operation except for the manual pulse generator)

### 2.3 LEAST INPUT INCREMENT (MINIMUM INPUT UNIT)

The least input increment is the minimum programmable length expressed in millimeters, inches or in degrees.

	Linear Axis	Rotary Axis <sup>†</sup>
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 parameter.

	Input Increment×10	
	Linear Axis	Rotary Axis <sup>†</sup>
Metric Input	0.01 mm	0.01 deg.
Inch Input	0.001 in.	0.01 deg.

† Optional

Metric input and inch input can be selected by setting numbers.

### 2.4 LEAST OUTPUT INCREMENT (MINIMUM OUTPUT UNIT)

The least output increment is the minimum unit of movement through which the machines can move, expressed in millimeters or inches.

	Linear Axis	Rotary Axis <sup>†</sup>
Metric Output	0.001 mm	0.001 deg.
Inch Output	0.0001 in.	0.001 deg.

† Optional

### 2.5 MAX. PROGRAMMABLE DIMENSIONS

		Linear Axis	Rotary Axis <sup>†</sup>
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.

† Optional

### 2.6 NC TAPE

8-channel black paper tape, EIA RS-277, ISO 1154, JIS C6246

### 2.7 TAPE CODE

EIA RS-244-A and ISO 840

Refer to Tables 1.1 and 1.2 in Appendix 1.

### 2.8 EIA/ISO AUTO-RECOGNITION

When the first EOB code is read in Label Skip mode, the code in use is automatically sensed.

### 2.9 TAPE FORMAT

Variable block format conforming to JIS B6313. The format differs with metric/inch input or output. For details of the formats, refer to Tables 1.3 and 1.4 in Appendix 1.

### 2.10 DECIMAL POINT INPUT

Numerical values containing a decimal point can be input. Addresses with which decimal points can be used are as follows:

- Coordinates: X, Y, Z, I, J, K, Q, R ...
- Feedrate: F
- Dwell time: P

Normally, when numbers without a decimal point are input, the control treats "1" as 0.001 mm, 0.0001 inch, or 0.001 deg. However, the control can be set by parameters to treat "1" as 1 mm, 1 in. or 1 deg.



## 2.11 BUFFER REGISTER

During normal operation, one block of data is read in advance and compensation is computed 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.

## 2.12 RAPID TRAVERSE RATE

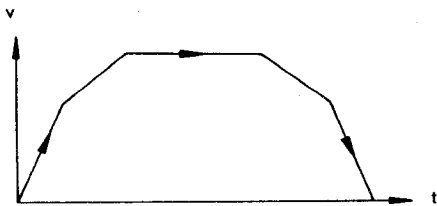
Up to 30,000 mm/min, or 1181.10 in./min, as set by parameters, is programmed independently for each axis.

## 2.13 FEEDRATE RANGE

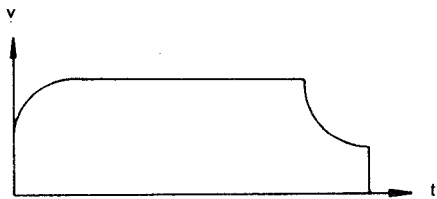
Feedrate is programmable between 1 and 30,000 mm/min, or between 0.1 and 2400 in./min. The upper limit can be set by parameters according to the machine.

## 2.14 AUTOMATIC ACCELERATION / DECELERATION

(1) In positioning and manual feeding, motion can be automatically accelerated and decelerated linearly. Two-stage linear acceleration/deceleration can also be set as shown below, independently for each axis.



(2) Feed acceleration is exponential, and is applied commonly to all the axes.



The time constants for the above curves are set by parameters.

## 2.15 FEED FUNCTION (F-FUNCTION)

Tool feedrates are selected within the following ranges by F codes.

		Format	Feedrate (Feed/Minute) Range
Metric Output	Metric Input	F40	F1. - F30000 mm/min
	Inch Input	F31	F0.1 - F1181.10 in./min
Inch Output	Metric Input	F50	F1. - F76200 mm/min
	Inch Input	F31	F0.1 - F3000 in./min

Note: Minimum input values can be reduced to 1/10 by parameters.

## 2.16 FEEDRATE OVERRIDE AND FEEDRATE OVERRIDE CANCEL

(1) Rapid traverse rate override

Rapid traverse rates can be reduced to F0, 25%, 50% or 100% of the original traverse rate. F0 is set by parameters.

(2) Feedrate override

The feedrates programmed by F codes can be modified between 0% to 200% in 10% increments.

(3) Feedrate override cancel

When this switch is turned on, any feedrate override effect is cancelled, and the tool moves at the originally programmed feedrates.

## 2.17 PREPARATORY FUNCTIONS (G-FUNCTION)

G codes consisting of address G plus up to 3 digits, specify work for the respective blocks. For details of the G codes, refer to Table 1.5 in Appendix 1.

(1) Ordinary G codes include non-modal G-codes marked with\*, and modal G-codes belonging to groups 01 through 15. The G-codes which belong to division B are basic G-codes.

(2) G100 through G199 are expansion G-codes. They are used to call G-codes for user macro option, etc.

## 2.18 ABSOLUTE/INCREMENTAL PROGRAMMING (G90/G91)

With the following G-codes, the tool movement can selectively be programmed either in absolute values or in increments:

G90: absolute designation

G91: incremental designation

## 2.19 PROGRAMMING OF ABSOLUTE ZERO POINT (G92)

With a command "G92 X... Y... Z...;" an absolute coordinate system is established with the current tool position having the specified coordinate values.

## 2.20 POSITIONING (G00, G06)

(1) G00 X... Y... Z... ;

With this command, the tool moves at the rapid traverse rate to the specified coordinate position, moving independently in each coordinate direction. The motion after positioning will be in the ERROR DETECT ON mode. G00 is a 01 group modal G code. The ERROR DETECT OFF mode can be entered by parameters.

(2) G06 X... Y... Z... ;

With this command, after executing a positioning similar to G00, the program advances to the next block in the ERROR DETECT OFF mode. G06 is non-modal, and is effective only in the programmed block.

Note: In the ERROR DETECT ON mode, the command of the next block is executed only after the servo-lag pulses in the current block are reduced to a permissible number. The ERROR DETECT OFF mode is where the command of the next block is executed immediately after the distribution of the pulses in the current block, regardless of the servo-lag pulses. In this mode, the corners of the workpieces are slightly rounded.

## 2.21 LINEAR INTERPOLATION (G01)

G01 X... Y... Z... F... ;

With this command, the tool moves along the specified straight line at a feedrate specified by the F code.

## 2.22 CIRCULAR INTERPOLATION (G02, G03)

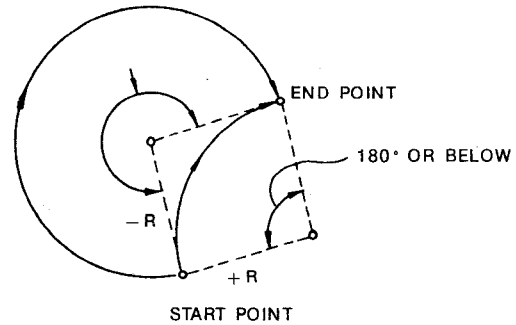
(1) G02 (G03) X... Y... I... J... F... ;

These commands move the tool along the specified circular path at feedrate specified by the F code. X and Y specify the end point of the circular motion, and I and J specify the center of the circular path in XY plane. With the proper selection of address, similar circular interpolation is programmed also in the XY and ZX planes. G02 is for clockwise motion, and G03 is for counterclockwise motion.

(2) G02 (G03) X... Y... R... F... ;

Circular interpolation is also possible by designating the radius R with the above command. When  $R > 0$ , a circular path with a center angle smaller than  $180^\circ$  is programmed, and when  $R < 0$ , the center angle of the circular path is larger than  $180^\circ$ .

180° OR OVER



G02 X... Y... R±... F... ;

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

This command moves the tool around a designated complete circle n times. When L is not programmed, the tool moves only once around the circle.

(4) G codes for plane designation (G17 to G19)

The plane for programming circular interpolation is specified by the following G codes:

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

Note :

1. Circular interpolation is possible over two or more quadrants.
2. Circular interpolation is also possible with respect to the optional 4th linear axis.

## 2.23 DWELL (G04)

G04 P... ;

With this command, the tool remains motionless for the duration of time specified by the P code. The minimum and the maximum programmable dwell times are 0.001 and 99999.999 seconds, respectively.

## 2.24 EXACT STOP CHECK (G09, G61, G64)

This function is effective only in the blocks of feedrate which is controlled by interpolation.

(1) Exact stop (G09)

A block containing G09 is executed in the ERROR DETECT ON mode. When the workpiece is required to be machined with a sharp corner, this code is programmed. G09 is non-modal, and is effective only in the programmed block.

(2) Exact stop check mode (G61)

When G61 is programmed, all the subsequent blocks are executed in the ERROR DETECT ON mode until G64 is programmed.

## 2.24 EXACT STOP CHECK (G09, G61, G64) (Cont'd)

(3) Exact stop check mode cancel (G64)

This code is for cancelling the G61 command.

Note:

1. When the power supply is turned on, the status corresponding to G64, that is, the ERROR DETECT OFF mode, is on.
2. Rapid traverse motion is controlled by G00 and G06, and not influenced by these exact stop G codes.

## 2.25 MISCELLANEOUS FUNCTION (M-FUNCTION)

Miscellaneous functions are programmed with address M and up to these digits. The M codes are grouped in the following three categories:

(1) M codes for internal processing, decode signal outputting, and 3-digit BCD outputting.

M00: Program stop  
M01: Optional stop  
M02: Program end (reset)  
M30: Tape end (reset and rewind)

(2) M codes only for internal processing

M90<sup>†</sup>: Program interrupt off  
M91<sup>†</sup>: Program interrupt on  
M92<sup>†</sup>: Multi-active register off  
M93<sup>†</sup>: Multi-active register on  
M94: Mirror image off  
M95: Mirror image on  
M96<sup>†</sup>: Tool radius compensation C; circular path mode  
M97<sup>†</sup>: Tool radius compensation C; intersection calculation mode  
M98: Subroutine program call  
M99: Subroutine program end  
M100 to 199:  
† indicates options.

(3) M codes exclusively for outputting 3-digit BCD signals are those other than the above.

## 2.26 SPINDLE-SPEED FUNCTION (S-FUNCTION)

The following output mode can be selected.

S 5-digit programming, analog output (Basic option). Outputs analog voltage of  $\pm 10$  V max as D/A converter.

The control outputs spindle gear ratio change commands (4 max) when it receives the RPM value specified program. It then outputs analog voltage corresponding to the changed gear ratio. Speed ranges corresponding to the changed gear ratio. Speed ranges for individual gear ratio are set by parameter.

In either output mode, spindle speed override can be accomplished. This function permits overrides by steps of 10% within a range of 50 to 120% to the spindle output command. (Input points: 3)

Instead of this function, S5-digit programming with 12-bit output is selected. It outputs 12-bit binary signal without a sign (4095 maximum).

## 2.27 TOOL FUNCTION (T-FUNCTION)

Tool numbers are specified by two digits following the address T. Commands to the machine are sent in 2-digit BCD.

Note: T4-digit programming with T4-digit BCD output is available as an option.

## 2.28 TOOL LENGTH COMPENSATION (G43, G44, G49)

This is a tool position offset function only effective in the Z-axis direction. With G43 (G44) Z... H... ; or G43 (G44) H... ; the tool is offset by the value stored in the tool offset memory specified by the H code in plus (+) or minus (-) direction, with respect to the point of the Z-axis movement.

G Code	Meaning
G43	Tool length compensation in plus (+) direction
G44	Tool length compensation in minus (-) direction
G49	Tool length compensation command cancel

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

## 2.29 TOOL POSITION OFFSET (G45 THROUGH G48)

These tool position offsets are used mainly for compensating for the radius differences when machining simple rectangular workpieces.

G01 G45 (G46) X... D... F... ;

With this command, the feed length of the tool in the specified axis is extended or retracted by the length stored in the specified tool offset memory.

G Code	Meaning
G45	Extension
G46	Retraction
G47	Double extension
G48	Double retraction

These G codes are non-modal, and are effective only in the block in which they are programmed. When circular interpolation is included in the same block in which a tool position offset is programmed, the radius and the end point are extended also. In this case, proper compensation for tool radius is possible only for machining 1/4, 3/4 and 4/4 circles.

### 2.30 OFFSET MEMORY

The two digits following the address H or D are called tool offset numbers, and these numbers are assigned to the 99 tool offset values stored in the tool offset memory. Any desired tool offset value can be designated with the tool length compensation command (specified by the H code) or the tool position offset command (specified by the D code among the stored values).

Note: The 99 tool offset values can also be used with the tool radius compensation C function (option). Tool offset memories can be expanded up to 299.

### 2.31 TOOL OFFSET VALUE

The range of tool offset value that can be written in the tool offset memory is as follows:

Metric Input	0 to ±999.999 mm
Inch Input	0 to ±99.9999 inches

### 2.32 BACKLASH COMPENSATION

This function is for compensating for the backlash in the driving system of the machines. Backlashes between 0 and ±8191 P can be compensated independently in each axes (p representing the minimum output unit). The desired compensation values are preset by parameters.

### 2.33 MANUAL FEED

Manual feed is possible in the following three modes, simultaneously in all three axes.

#### (1) Manual rapid traverse (RAPID)

The tool moves at the rapid traverse rate, independently in all three axes.

#### (2) Manual JOG feed (JOG)

After setting the JOG FEEDRATE switch at the desired speed (32 available), the tool will move at that feedrate while any of the JOG buttons is depressed.

#### (3) Step feed (STEP)

Each time the desired JOG button is pushed, the tool moves through the distance specified by the MANUAL PULSE MULTIPLY switch. The distance are in the following multiples of pulses: × 1, × 10, × 100, × 1000, × 10,000, × 100,000.

### 2.34 PROGRAM STORAGE AND EDITING

Part program can be loaded into memory for tapeless operation and for editing.

(1) Memory capacity is equivalent to 40 meters of tape. (Note 1)

(2) Part program, added with a program number of 4-digit numerals, can be stored in memory (from paper tape or MDI). In the basic mode, up to 99 program numbers can be stored in memory. (Note 2)

(3) The stored part program can be edited by ERASE, INSERT, and ALTER keys. Editing is done in one to several words at a time.

(4) The OUT, VER, and IN keys are used to output the stored part programs to external equipment (option), to collate them with punched cards, and store them from tape readers. (Note 3)

(5) Address search function permits the specified program number to be searched for the purpose of an automatic operation (MEM mode).

Note:

1. Optionally, the part program storage may be extended to 320 meters.
2. Optionally, the number of stored programs may be extended to 999.
3. To output the part program to an external equipment, the optional "data input/output interface" is required.

### 2.35 SUBROUTINE PROGRAM (M98, M99)

Subroutine programs with program numbers can be retrieved and executed as many times as desired.

#### (1) Retrieving subroutine programs (M98)

M98 P... L... ;

With this command, the subroutine program with the number designated by P is retrieved and executed L times. When no L-digit is defined, the subroutine program is executed only once. The retrieved subroutine program may also retrieve further subroutine programs up to four nestings.

## 2.35 SUBROUTINE PROGRAM (M98, M99) (Cont'd)

### (2) Subroutine program end (M99)

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

O .....	;	...	Program No.
.....	;		
.....	;		
.....	;		
.....	;		
M99	;	...	Subroutine program

### (3) Special use of M99

M99 P... ;

With this command, the control does not advance to the subsequent block after executing the subroutine program, but returns to the block with the sequence number specified by P.

## 2.36 PARAMETER SETTING

Parameters for machine constants such as backlash compensation values and rapid traverse rate can be written.

## 2.37 SETTING FUNCTION

Any of the functions can be selectively switched on and off.

## 2.38 INTERNAL DATA TAPE INPUT

Normally, tool offset values, parameter data, and setting data are input from MDI. With this function, these data can be entered into the respective memories via tape reader.

With ordinary part programs, any desired tool offset values can be changed into desired tool offset values can be changed into new values with the command "G10 P... R... , (P = tool offset number, R = tool offset value).

## 2.39 OPERATION TIME DISPLAY

With this function, the cumulative times of the following operations can be displayed:

- (1) Total time after switching the power supply on
- (2) Total time of automatic operation
- (3) Total automatic cutting (interpolation motion) time

## 2.40 ADDRESS SEARCH

All address data, including program numbers in the part program storage can be searched with an MDI command.

## 2.41 PROGRAM NUMBER

Up to 4 digits can be written as program numbers immediately after the address 0. However, the maximum number of program numbers that can be registered is 99. A part program starts with a program number, and ends with M02, M30 or M99.

## 2.42 LABEL SKIP

The Label Skip function becomes effective and "LABEL SKIP" is displayed when:

- (1) the power supply is turned on,
- (2) control is reset.

When the Label Skip function is effective, all the tape information before the first EOB code is ignored. When LABEL SKIP lamp is on in the MEM or EDIT mode, it indicates that there is a pointer at the beginning of the part program.

## 2.43 CONTROL IN/OUT

Data between a control out" (" and control in")" is ignored as insignificant.

## 2.44 TV CHECK

This function checks whether the number of characters including EOB is odd or even. If the number is odd, the block is regarded as an input error, and the operation is interrupted automatically. This function is turned on and off with parameters.

Note: The TV check does not count the characters between control out and control in.

## 2.45 SEQUENCE NUMBER BREAK POINT

During automatic operation, a single-block-stop can be applied after the execution of a block by specifying the sequence number of the desired block. The specified sequence number is called a break point, and up to 2 break points can be set with the setting function.

## 2.46 SINGLE BLOCK

While the SINGLE BLOCK switch (at the machine side) is turned on, automatic operation with tape or the memory are performed block by block.

## 2.47 OPTIONAL STOP

While the OPTIONAL STOP switch (at the machine side) is on, operation stops automatically after executing a block including M01.

## 2.48 OPTIONAL BLOCK SKIP

While the OPTIONAL BLOCK SKIP switch (at the machine side) is on, a command block starting with "/" is neglected.

## 2.49 DRY RUN

With this function, the feedrates for automatic operation are converted into manual operation feedrates for convenience during dry-run. While the DRY RUN switch (at the machine side) is on, the following feedrates are available.

	DRY RUN On
Feed	Feedrate for Manual Continuous Feed
Rapid Traverse	Rapid Traverse or Manual Continuous Feed (Parameter Selection)

## 2.50 MACHINE LOCK

This function allows NC commands to be executed, with M, S, and T functions functioning normally, and the current positions to be continuously updated and displayed, with the machine standing still.

## 2.51 DISPLAY LOCK

With this function, the POS-EXTERNAL display is locked while the machine is operated under automatic or manual control.

## 2.52 Z-AXIS COMMAND NEGLECT

While Z-AXIS NEGLECT switch is on, the machine motion only in the Z-axis is locked.

## 2.53 AUXILIARY FUNCTION LOCK

While this switch is on, no BCD code is output for M, S, and T (and B+) functions.

## 2.54 MANUAL ABSOLUTE ON/OFF

While this switch is on, manual movement distances are added to the absolute register, and the coordinate system remains unchanged. While the switch is off, manual movement distances are not added, and the coordinate system is shifted in parallel with the movement.

## 2.55 MIRROR IMAGE

Mirror image axis for symmetrical machining can be specified with switches (for X, Y or the 4th<sup>+</sup> axis). Mirror image control is applied to the tool path between M95 and M94 on the program with respect to the specified axis.

## 2.56 INTERNAL TOGGLE SWITCHES

With this function, the toggle switches for the function described in paragraph 2.47 SINGLE BLOCK through 2.56 MIRROR IMAGE can be eliminated at the machine control station, and the necessary setting can be performed from the NC operator's station.

## 2.57 ORIGIN KEY

The ORG key designates the current tool position designated at point "0." Each axis is controlled independently.

## 2.58 INTERLOCK

Tool movement for control axes can be locked during motion with this function. When interlock is applied, the tool stops after deceleration, and when interlock is cleared, the tool resumes the motion.

When interlock is applied during an interpolation covering two or more axes, the interpolation motion is stopped.

## 2.59 START LOCK AND EDIT LOCK

The following functions can be turned on and off by switches.

### (1) START LOCK

When this function is on, the CYCLE START key is made ineffective.

### (2) EDIT LOCK

When the EDIT LOCK is on, editing and storing of part programs are inhibited.

## 2.60 AUTOMATIC COORDINATE SYSTEM SETTING

With this function, a new coordinate system having coordinate values set by parameters in advance is automatically set up at the reference point after executing manual return to reference point. The coordinate system established by this function is equivalent to the one set by G92.

## 2.61 FEED HOLD

The feedhold function interrupts tool feed temporarily during automatic operations. Feedhold operation is disregarded during threading.

## 2.62 EMERGENCY STOP

This function makes all the commands ineffective. The servo power supply is turned off, and all moving members of the machine are stopped by dynamic brake.

## 2.63 OVERTRAVEL

With this function, the tool motion is stopped upon receipt of a stroke-end signal from the machine. When the machine is stopped by this function, the machine member must be moved backward by manual feed.

## 2.64 REMOTE RESET

With this function, the NC is reset with an external signal. When the NC is reset, all the commands become ineffective, and tool motion is stopped immediately.

## 2.65 REMOTE POWER ON/OFF

In addition to the POWER ON/OFF keys on the NC Operator's station, the control is provided with input terminals for receiving remote power on/off signals.

## 2.66 MACHINE READY INPUT SIGNAL

When the machine is ready for operation, this signal is transmitted from machine to control. When this signal is received with the control in the "NC ready" condition, operation may be started immediately.

When this signal drops off during operation, "machine error" status occurs, and all the functions become ineffective.

## 2.67 NC READY OUTPUT SIGNAL

When the NC is correctly energized, and is ready for control function, an "NC ready" signal is output to the machine.

## 2.68 NC ALARM OUTPUT SIGNAL

When one or more alarm states are present, an NC alarm signal is output to the machine. When the cause is eliminated and the reset procedure is followed, this signal is stopped.

## 2.69 NC RESET OUTPUT SIGNAL

While RESET key or REMOTE RESET key is depressed, an NC reset signal is output to the machine.

## 2.70 RS-232C INTERFACE

RS-232C interface is provided in order to connect with tape puncher, external tape reader or other external devices.

Interface Type	Serial Voltage Interface
Communication Speed	110 to 4800 baud
Connector	DB-25S
Maximum Cable Length	15 m
Output from Memory	Possible
Storage in Memory	Possible
Tape Mode Operation	Possible

Note : The following are the data items which are objects for output from memory and storage in memory.

- (a) Part program
- (b) Offset data, machine tool coordinate data, machine tool abrasion data
- (c) Setting and parameter data

## 2.71 ON-LINE DIAGNOSTICS

During operation, the following self-diagnoses are made online:

- (1) 3-digit Alarm Code and Alarm Message Display.
- (2) System Diagnosis
  - a. System memory total check.
  - b. RAM check (when power is input).
  - c. Watchdog timer.
- (3) Input/Output Signal Diagnosis

## 2.72 POSITION DETECTOR INTERFACE

Position detected by feedback signal from the rotary-type pulse generator. The motion per rotation of the pulse generator is varied by the number of pulses from the pulse generator as shown below.

	Pulse Generator	Motion per Rotation of Pulse Generator
Metric Output	5000 p/rev	2.5, 5, 10 mm
	6000 p/rev	2, 3, 4, 6, 8 mm
Inch Output	5000 p/rev	0.25, 0.5 in
	6000 p/rev	0.2, 0.3, 0.4, 0.6, 0.8 in

The motion per rotation of the pulse generator of X-axis becomes one-half of the above values.

Note: Pulse multiplication can be set by servo unit or NC.

### 2.73 INPUT/OUTPUT CONNECTORS

The control is connected with the machine control circuit via "Half pitch connectors."

### 2.74 POWER INPUT A

Standard input  
200/220/230 VAC +10%, -15%, 3-phase, 50/60 Hz,  
±1 Hz

### 2.75 AMBIENT CONDITIONS

(1) Ambient temperature:

for operation: 0°C to 45°C (32°F to 113°F)

for storage: -20°C to +65°C (-4°F to +149°F)

(2) Relative humidity: 10 to 90% R.H.

(3) Vibration: 4.9 m/s<sup>2</sup> max.

Note: When the ambient conditions do not conform to the above requirements, or when organic solvent or other fumes are present in high concentration we offer special measures.

### 2.76 PAINT COLOR AND DIMENSIONS

(1) The following dimensions are available for the selection to suit the machine to be controlled.

· Control unit  
150 (W)×350 (H)×185 (D) mm  
5.90 (W)×13.78 (H)×7.28 (D)

(2) Paint Color

- NC operator's station: Munsell N1.5 (gray)
- Enclosure inner/outer surface: Munsell 4Y7.7/1.2 (light gray)

## 3 BASIC OPTIONS

The optimal servo components are available for selection to meet the requirements of the machine being controlled.

### 3.1 AC SERVO CONTROL UNITS

Transistorized PWM AC servo control units are further miniaturized to be available for use in either or the following systems:

(1) NC board built-in system: Free standing type.

(2) External system: Supplied in unit.  
Connection cable should be 10 m max.

Servo capacity is as follows:

No.	F Series Motor Maximum Continuous Torque kg · cm	SERVOPACK Type
1	30	CACR-SR05SB1 <input type="checkbox"/> F
2	60	CACR-SR10SB1 <input type="checkbox"/> F
3	90	CACR-SR15SB1 <input type="checkbox"/> F
4	120	CACR-SR20SB1 <input type="checkbox"/> F
5	230	CACR-SR30SB1 <input type="checkbox"/> F
6	380	CACR-SR44SB1 <input type="checkbox"/> F

### 3.2 AC SERVOMOTORS

The following AC servo motors that incorporate the feedback unit, consisting of position-detecting pulse generator (PG) and speed-detecting tachometer generator (TG).

No.	F Series Motor Maximum Continuous Torque kg · cm	SERVOMOTOR Type
1	30	USAFED-06F <input type="checkbox"/> *
2	60	USAFED-09F <input type="checkbox"/>
3	90	USAFED-13F <input type="checkbox"/>
4	120	USAFED-20F <input type="checkbox"/>
5	230	USAFED-30F <input type="checkbox"/>
6	380	USAFED-44F <input type="checkbox"/>

\* According to the type of detector, either A or B is entered in .

A : 6000 p/rev

B : 5000 p/rev



## 4 OPTIONS

### 4.1 NC OPERATOR'S STATION

The separate stations are available in three configurations :

- Keyboard on right side of CRT  
See Fig. A2.2.
- Keyboard below CRT  
See Fig. A2.3.

### 4.2 TAPE READER

The tape reader unit specified below can be provided as an independent unit or incorporated in a free-standing cabinet. For connections of the tape reader, RS-232C interface is used.

- Read speed : 200 char./sec
- Reading system : LED-photoelectric

### 4.3 TAPE READER WITH REELS

Free-standing type cabinets can be provided with the following tape reader with reels. Tape reel unit can be provided as an independent unit.

(1) 6-inch reel

Reel diameter: 150 mm (6 inches)  
Tape length: 80 m (262 ft.)

(2) 8-inch reel

Reel diameter: 200 mm (8 inches)  
Tape length: 180 m (590 ft.)

The indicated tape lengths are for tapes with 0.108 mm (4 inches) thickness. The following are common to both reader.

Read speed: 200 char./sec  
Rewind speed: 200 char./sec  
Reading system: LED photoelectric

### 4.4 F1-DIGIT COMMAND

With this function, feedrates can be programmed by one digit following an address F (F1 through F9). The actual feedrates corresponding to F1 through F9 are preset by parameters.

When this function is adopted, 1 through 9 mm/min commands of the ordinary F-function (direct designation of feedrates in mm/min) cannot be used.

### 4.5 S5-DIGIT PROGRAMMING WITH 12-BIT OUTPUT

S 5-digit programming with 12-bit output S 5-digit programming analog output can replace S 5-digit programming 12-bit output specification. The output is made by 12-bit binary (4095 maximum) signal.

### 4.6 T4-DIGIT PROGRAMMING

Instead of the basic T 2-digit programming, tool numbers can be programmed with 4 digits following the address T. The control outputs corresponding 4-digit BCD code.

### 4.7 ADDITIONAL OFFSET MEMORY

Basic offset memory capacity of 99 can be expanded up to 1199.

### 4.8 ADDITIONAL PART PROGRAM STORAGE

Instead of the basic part program storage of tape length 40 m (131 ft.) the following are available.

	Tape Length
1	80 m (262 ft.)
2	160 m (524 ft.)
3	320 m (1049 ft.)

### 4.9 ADDITIONAL PROGRAM NUMBER REGISTRATION

Either of the following number of registrable programs can be selected to replace the basic number of 99.

	Total Number of Registrable Programs
1	199
2	999

Note that if the optional "additional program number registration" is adopted, the storage capacity for the part program is reduced by the following amount.

Additional Number of Registered Programs	Reduced Storage Capacity
199	2 m (6.6 ft.)
999	18 m (59 ft.)

### 4.10 4TH AXIS CONTROL

Any one out of the three rotary axes A, B and C and linear axes, U, V and W, can be controlled as 4th axis in addition to the three basic axes. The following simultaneous controls are possible.

- Simultaneous controllable 4 axes:
  - Positioning (G00, G06, G60)
  - Linear interpolation (G01)
  - Manual feed except by manual pulse generator

- Simultaneous controllable 2 axes:  
Circular interpolation (G02, G03)
- For linear 4th axis, stored stroke limit can be set at 1st prohibit area only.
- Program Restart, User Macro, Stored Lead-screw Error Compensation are effective with the 4th axis.
- Tool Length Compensation, Tool Compensation C, High-speed Cutting are ineffective with the 4th axis.

#### 4.11 MANUAL PULSE GENERATOR FOR ONE AXIS AT A TIME

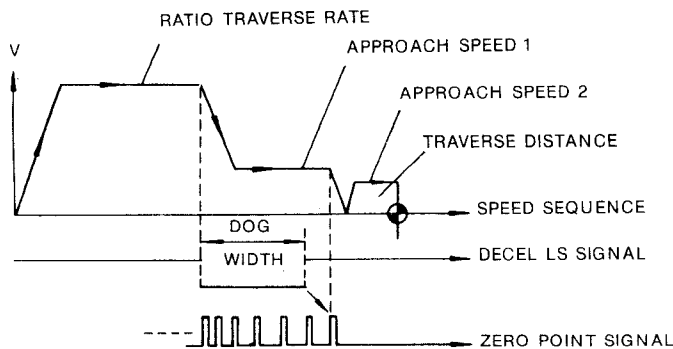
With the manual pulse generator, the machine can be moved in response to the manual turning of the pulse generator by handle. The generator has a dial graduated into 100, and the travel distance per graduation can be set to 1, 10 and 100 pulses. The controlled axes are selected with the select switch, and the tool is controlled only in the selected axial direction.

#### 4.12 REFERENCE POINT RETURN (G27, G28, G29)

All of the following methods for returning the tool to the reference point are possible.

##### (1) Manual return to reference point

After the ZERO RETURN switch is turned on, the tool can be returned to the reference point by manual operation. The approach speed and the traverse distance are set by parameters.



##### (2) Reference point check (G27)

G27 X... Y... Z... ;

With this command, the programmed point is checked for coincidence with the reference point. If the programmed point does not coincide with the reference zero point even in one axial direction, this is regarded as an error.

##### (3) Automatic return to reference point (G28)

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

With this command, the tool first moves to the specified intermediate position, and from there, automatically returns to the reference point.

##### (4) Return from reference point (G29)

G29 X... Y... Z... ;

With this command, the tool starts from the reference point, moves through the intermediate point specified by G28 previously, and finally moves to the position specified by G29.

##### (5) Rapid return to reference point

The manual and automatic reference point return motions can be made in rapid return mode by specifying in the specifications. In this rapid return mode, the tool does not follow the deceleration sequence started by decelerations LS, but moves directly to the reference point as if the tool were moving in the G00 positioning mode. This rapid return mode becomes effective, however, only after the tool has been once returned to the reference point in all axes in the normal mode.

#### 4.13 2ND, 3RD, AND 4TH REFERENCE POINT RETURN

G30 Pn X... Y... Z... ; (Pn = P2, P3, and P4)

With this command, the tool is returned to the 2nd, 3rd, or 4th reference point after positioning in the specified intermediate position.

P2: 2nd reference point

P3: 3rd reference point

P4: 4th reference point

The reference points can be set by parameters.

#### 4.14 EXTERNAL DECELERATION

When an external deceleration limit switch is tripped, the tool speed, both rapid traverse and feed, are reduced to the levels set by parameters. This switch can be installed in both directions on all axes.

#### 4.15 TOOL LENGTH MEASUREMENT

With an actual tool mounted on the machine spindle, the tip of the tool is brought to the base position for the Z-axis by manual operation, and RETRACT button is pushed. Then, the control executes the following operations:

- (1) Automatic storing of the distance from the home position in the Z-axis to the current position in the tool offset memory at the specified tool offset number.

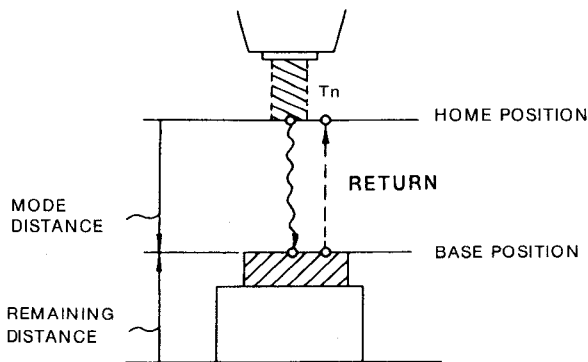
#### 4.15 TOOL LENGTH MEASUREMENT (Cont'd)

(2) Increasing the tool offset number by "1" in preparation for the next write operation.

(3) Returning the tool to the Z-axis home position.

Note:

1. Instead of "moved distance," "remaining distance" can be stored by parameters.
2. Instead of pushing the RETRACT button, the appropriate key on the NC Operator's Station may be depressed for tool length measurement. The tool does not return to the home position.



#### 4.16 OPTIONAL BLOCK SKIP B

In addition to the ordinary block skip (programmed with "/" or "/1"), 8 skip functions may be built into the control. These functions are programmed with codes "/2" through "/9," and when the respective switches are on, these blocks are skipped.

#### 4.17 2ND AUXILIARY FUNCTION (B-FUNCTION)

2nd auxiliary function command can be used with three digits following address B. The control outputs the corresponding 3-digit BCD code to the machine.

Note: When this B function is incorporated in the control, the B-axis for 4th axis or 5th axis control cannot be adopted.

#### 4.18 JOG FEEDRATE OVERRIDE

This function permits overrides in 21 steps of 10% per step within a range of 0 to 200% in relation to the jog feedrate. JOG FEEDRATE OVERRIDE switch serves as the FEEDRATE OVERRIDE switch.

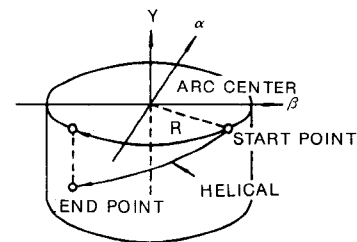
#### 4.19 PROGRAM COPY

```
G25 P ... ... Q ... ... L ... ;
      p1 p2   q1 q2
```

This command executes any program from sequence p2 of program p1 to sequence q2 of program q1, L times. Omission of L executes the program one time. G25 can be commanded in the copied program. Nesting up to 4th level can be applied. During canned cycles, program copy can be commanded.

#### 4.20 HELICAL INTERPOLATION (G02, G03)

Circular interpolation in any desired plane can be synchronously combined with a linear interpolation perpendicular to the plane of circular interpolation. For example, with G02 (G03) X... Y... I... J... Z... F... ;, circular interpolation in the XY plane is combined with linear interpolation in the Z-axis.



Note:

1. Instead of I, J and K, arc center can be specified by radius R.
2. Helical interpolation is also possible with a 4th axis which is a linear axis.
3. Tool radius compensation C is effective only on the circle projected on the plane of circular motion.

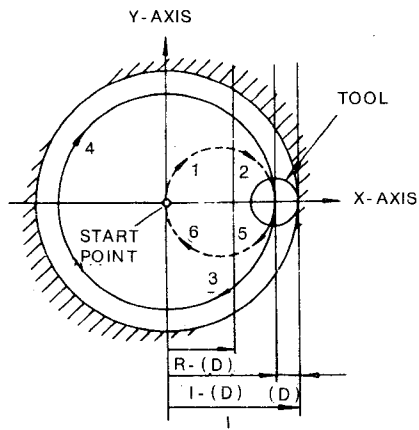
#### 4.21 CIRCLE CUTTING B (G12, G13)

With this function, all the motions involved in cutting a circle can be programmed in one block.

(1) Rapid approach distance designation R

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

With this command, the circle cutting operation shown below is executed. The rapid approach distance is specified by R.



**Tool path**

G12: 1→2→3→4→5→6

G13: 6→5→4→3→2→1

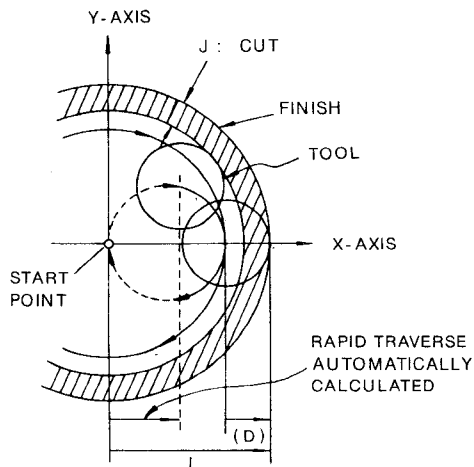
(D) indicates tool offset value specified.  
G12 is for CW and G13 is for CCW.

- I: radius of finished circle (with sign, incremental)
- R: rapid approach distance (with sign, incremental)
- D: tool offset number
- F: feedrate

(2) Automatic calculation of rapid approach distance

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

When the cutting depth is designated incrementally by the address J, instead of three distance R, the rapid approach distance just short of the tool contact point of the stock surface is automatically calculated.



(3) Repetition of circular motion by L

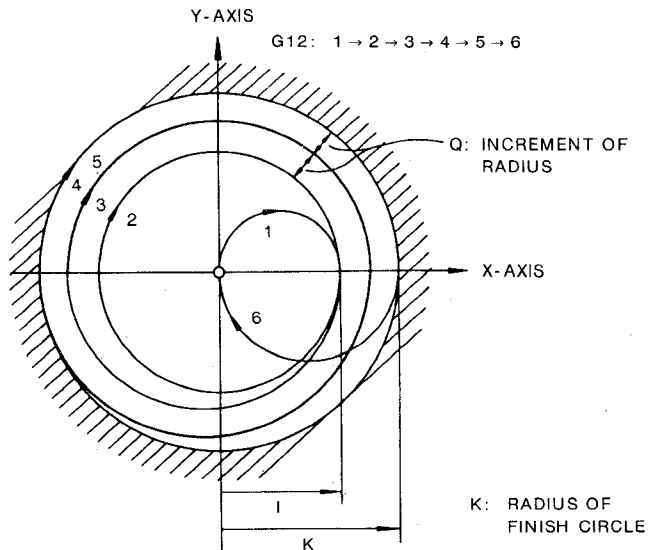
G12 (G13) I.. D... J... L... F... ;

With this command, the circular motion is repeated L times.

(4) Spiral circular cutting by Q and K

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

With this command, the tool moves along a spiral path as shown below to cut a circle. For the sake of simplicity, an imaginary tool with the radius  $D = 0$  is shown.



Note: This function incorporates the tool radius compensation function.

#### 4.22 INCH/METRIC DESIGNATION BY G CODE

With the following G codes, metric and inch input system are selectively designated.

G Code	Input unit
G20	Inch
G21	Metric

These G codes are written at the beginning of a program in a single block.

Note:

1. G20 and G21 commands rewrite inch/metric selecting parameter. Therefore, the condition of G20 or G21 at power switch-on time depends on the parameter.
2. The inch/metric selection by parameter setting is a basic function.

#### 4.23 UNIDIRECTIONAL APPROACH (G60)

G60 X... Y... Z... ;

With this command, the tool is brought to the specified position. However, in the direction specified by parameter only the tool first overruns and returns to the specified position. The overrun distance is set by parameters.

This G60 command is effective to stop the tool with high accuracy.

#### 4.24 WORK COORDINATE SYSTEM SETTING A (G52 TO G59)

With this function, tool motion can be programmed in the following coordinate system, in addition to the basic coordinate system set up by the G92 command.

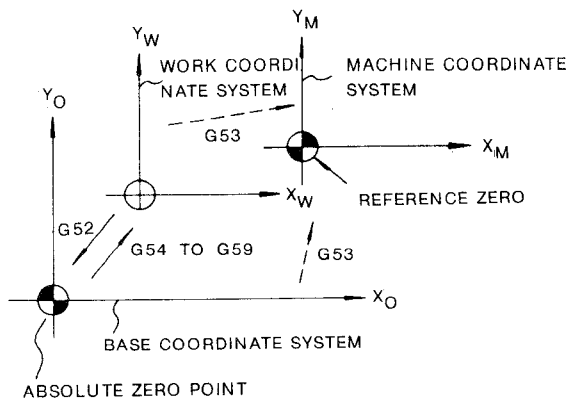
(1) When any of the G codes from G54 through G59 is programmed, a new coordinate system which is shifted from the basic coordinate system by a preset amount for the selected G code is established. This newly established coordinate system is referred to as a work coordinate system, and thereafter, the tool is controlled on the work coordinate system. Up to 6 work coordinate systems can be set up for use.

(2) Return to the basic coordinate system (G52) G52 ;

With this command, the current work coordinate system is cancelled, and the basic coordinate system is re-established.

(3) Temporary shift to machine coordinate system (G53)

During the execution of programs on a work coordinate system or on the basic coordinate system, the tool may temporarily be shifted to the position (X, Y, Z) on the machine coordinate system with a command G53 G00 X... Y... Z... ;. G53 is a non-modal G code. The machine coordinate system is one in which the origin is the reference point.



#### 4.25 WORK COORDINATE SYSTEM SETTING B (G54J TO G59J)

This is the expansion function of work coordinate system setting A. Up to 30 types of work coordinate systems can be set with 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 five axes X, Y, Z,  $\alpha$ ,  $\beta$  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 axis cannot be set. The other functions are the same as work coordinate system setting A.

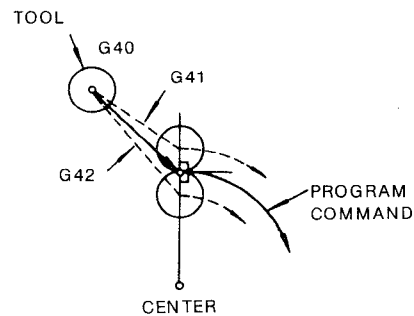
#### 4.26 TOOL RADIUS COMPENSATION C (G40 TO G42)

This function is for automatically offsetting the tool path to the right or to the left of the programmed path by a distance equal to the radius of the tool used.

(1) The meaning of these G codes is as follows.

G Code	Meaning
G40	Cancel tool radius compensation
G41	Tool radius compensation, left
G42	Tool radius compensation, right

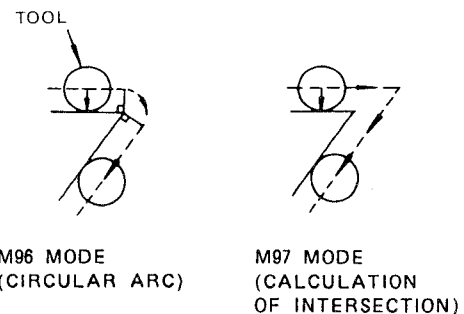
Note: When power is applied or the control is reset, the control is in the state of G code marked with  $\blacktriangledown$ .



(2) D code for tool radius designation

Tool offset numbers are specified by two digits following address D. The actual tool radius values are stored in the tool offset memory corresponding to these tool offset numbers. The maximum tool radius that can be set is as follows:

±999.999 mm  
±99.9999 in.



#### 4.27 OUTPUT FOR EXTERNAL MOTION (G80, G81)

G81 X... Y... L... ;

With this command, the control outputs an external motion signal to the machine. The L digit specifies the number of repeated positioning motions. G81 is modal, and remains effective on all the motion commands until it is cancelled by G80. G81 may selectively be used for this function or for the canned cycle 1 depending on parameter setting.

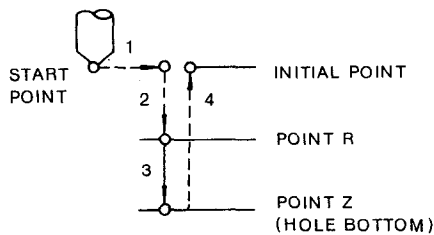
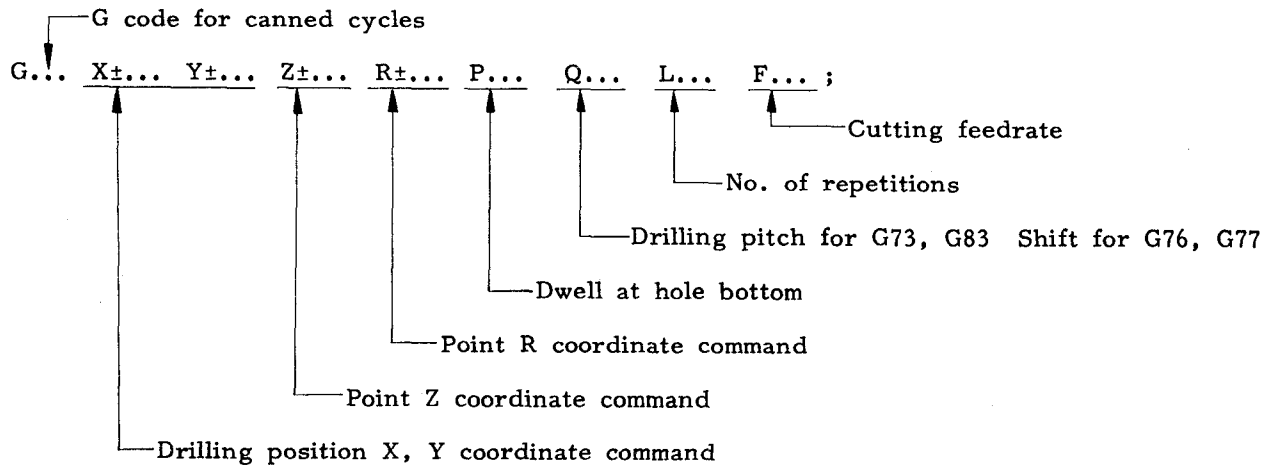
#### 4.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89)

Canned cycles are simplified programs for specific sequential movements covering several blocks by single-block commands. The following 13 canned cycles are available. G80 is for cancelling these canned cycles.

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 orientation→ Shift	Rapid traverse →Shift, spindle start	Boring
G77	Spindle orientation→ Shift→ Rapid traverse →Shift →Feed	Dwell —	Rapid traverse →Spindle orientation 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

## 4.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89) (Cont'd)

### Command Format



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

- ① Positioning the drill (position X, Y)
- ② Rapid traverse to point R
- ③ Drilling to point Z
- ④ Return to point R or to initial point

The number of repetitions is specified by the address L, and when no number is specified, the cycle is executed only once. When L = 0 is programmed, only positioning to (X, Y) is executed.

The shift direction is G76 and G77 can be specified at any angle by parameters. The return position in the Z-axis direction after executing canned cycles can be specified by the following G codes.

G Code	Meaning
G98	Initial point
G99	Point R

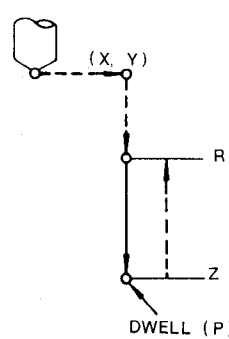
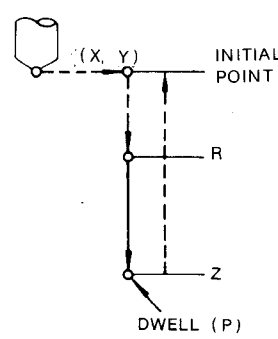
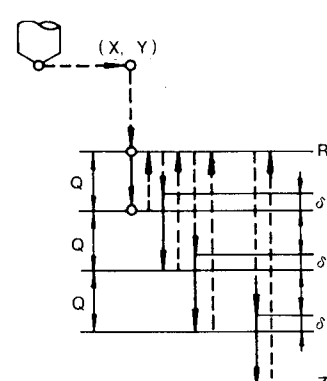
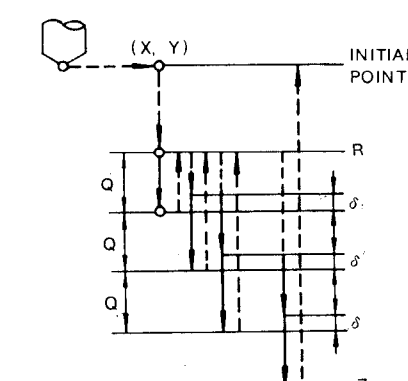
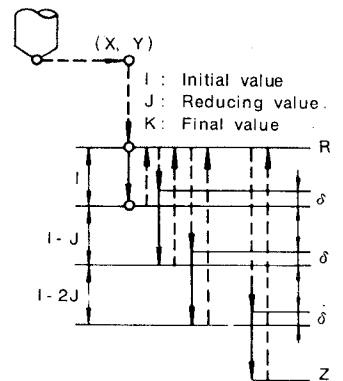
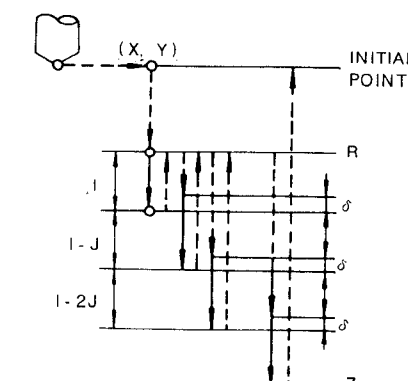
Note: When power is applied or the control is reset, the control is in the state marked with

	With G99 (return to R)	With G98 (return to initial point)
<p>G73 (Fixed pitch) High speed deep hole drilling</p>	<p>G73 X... Y... Z... R... Q... L... F... ;</p> <p><math>\delta</math>: Setting data</p>	<p><math>\delta</math>: Setting data</p>
<p>G73 (Variable) 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><math>\delta</math>: Setting data</p>	<p><math>\delta</math>: Setting data</p>
<p>G74 Reverse tapping</p>	<p>G74 X... Y... Z... R... P... L... F... ;</p> <p>SPINDLE REVERSE</p> <p>R</p> <p>Z</p> <p>SPINDLE FORWARD AFTER DWELL</p>	<p>(X, Y)</p> <p>START POINT</p> <p>SPINDLE REVERSE</p> <p>R</p> <p>Z</p> <p>SPINDLE FORWARD AFTER DWELL</p>

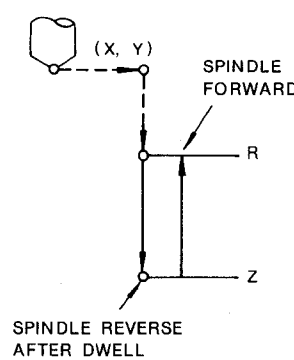
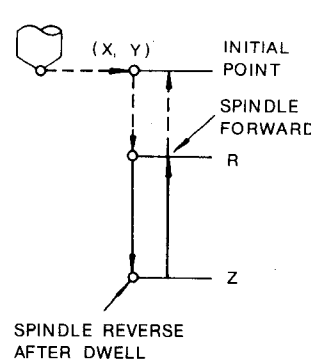
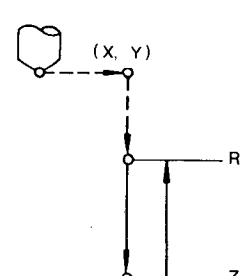
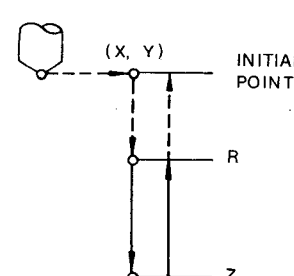
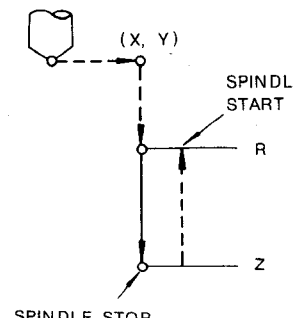
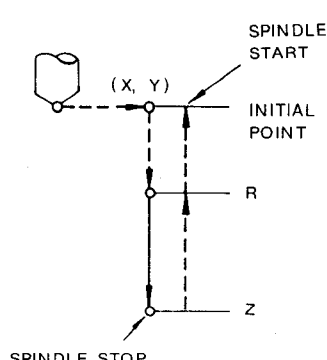


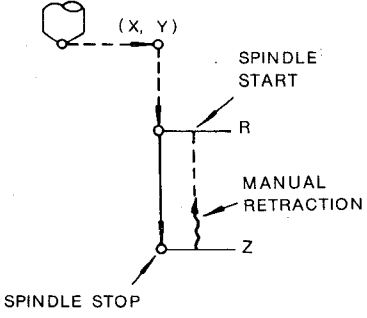
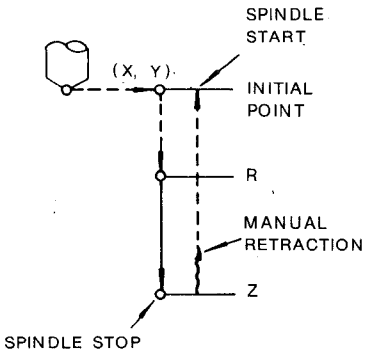
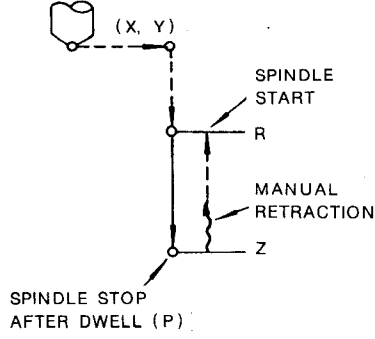
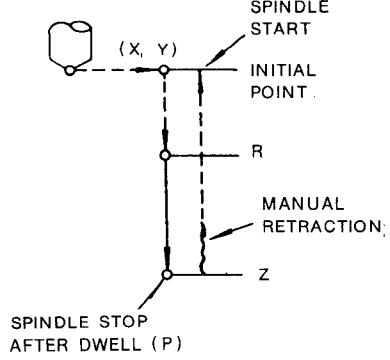
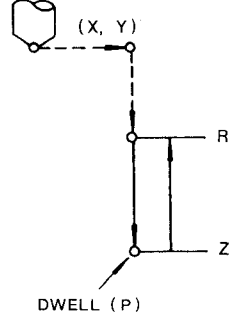
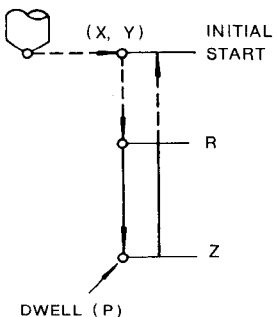
### 4.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89) (Cont'd)

	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>	

	With G99 (return to R)	With G98 (return to initial point)
<p>G82 Tapping</p>	<p>G82 X... Y... Z... R... P... L... F... ;</p> 	
<p>G83 Boring</p>	<p>G83 X... Y... Z... R... L... F... ;</p>  <p><math>\delta</math>: Setting data</p>	 <p><math>\delta</math>: Setting data</p>
<p>G83 Boring</p>	<p>G83 X... Y... Z... R... L... F... ;</p>  <p>I: Initial value J: Reducing value K: Final value</p> <p><math>\delta</math>: Setting data</p>	 <p><math>\delta</math>: Setting data</p>

### 4.28 CANNED CYCLES (G73, G74, G76, G77, G80 TO G89) (Cont'd)

	With G99 (return to R)	With G98 (return to initial point)
<p>G84 Tapping</p>	<p>G84 X... Y... Z... R... P... L... F... ;</p> 	
<p>G85 Boring</p>	<p>G85 X... Y... Z... R... L... F... ;</p> 	
<p>G86 Boring</p>	<p>G86 X... Y... Z... R... L... F... ;</p> 	

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

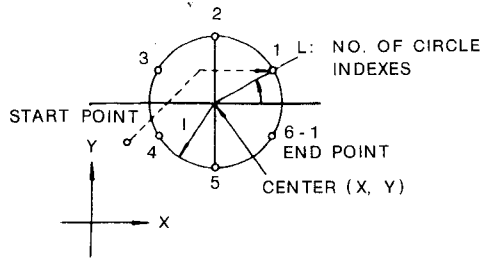
## 4.29 HOLE PATTERN CYCLES (G70, G71, G72)

This function, used in combination with canned cycles, permits simple drilling of holes in specific position patterns. G70, G71 and G72 automatically calculate the hole locations in specific patterns using the auto-programming technique.

(1) Bolt hole circle (G70)

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

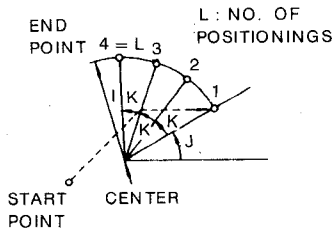
This command specifies the locations equally dividing the circumference into L parts.



(2) Arc (G71)

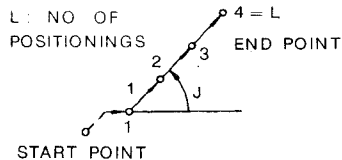
G71 X... Y... I... J... K... L... ;

This command specifies L locations on an arc.



(3) Line-at-angle (G72)

This command specifies L locations on any straight line.



Note: When executing drilling cycles with G70 to G72, the canned cycle with L = 0 (G73, G76, G77, G81 to G89) should in principle be programmed in the preceding block.

## 4.30 SCALING FUNCTION

With this function, geometrical shapes specified by part programs can be enlarged or reduced in any desired ratio.

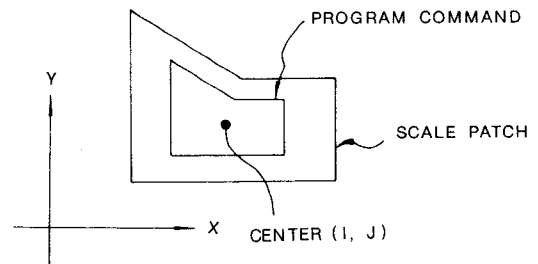
(1) G51 I... J... K... P... ;

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.

(2) G50: Command cancels the scaling mode.

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

Enlarging and reducing range: 0.000001 - 99.999999



## 4.31 MACRO PROGRAM (G65, G66, G67)

The special subprograms prepared by the machine makers or users may be stored in the part program memory, called and executed. Such special programs are called main macro.

(1) Macro Program Simple Call (G65)

G65 P... L... <argument designation>;

The main macro with the program number specified by P is executed L times. The argument designation means that real number is allocated to a variable, and that value is written after the address.

(2) Macro Programm Modal Call (G66, G67)

G66 P... L... <argument designation>;

This command generates the macro call mode, and every time motion commands are executed, the main macro specified by P is executed L times.

G67 ;

This command cancels the macro call mode.

(3) Multiple Call

A called main macro can call another macro, and this process can be nested to 4 levels of macros.

#### (4) Main Macro

The main macro, written in the format of sub-program (starting with 0 macro number and ending with M99), is provided with the following functions for high operation capabilities.

##### a. Normal variable

Many local and common variable can be used.

##### b. System variable

Various internal control data (various current values, offset values, parameters, clocks, etc.) and external input/output data can be directly processed in the macro as system variable.

##### c. Control statement

(i) IF [< conditional expression >] GO TO n ;

The above conditional branch control statement can be used.

(ii) WHILE [< conditional expression >] DO m ;

The above conditional performance control statement can be used.

##### d. Arithmetic operation

(i) +, -, OR, XOR can be used.

(ii) \*, /, AND, SIN, COS, TAN, ... FUP, etc. can be used.

### 4.32 EXTERNAL DATA INPUT

Data can be transmitted to the control to let the machine operate in a specified manner. For this transmission, interfaces for 24-bit input signals (data: 16 bits) and 2-bit output signals (answer signals) must be prepared and connected to the control.

#### (1) External work number search

Work numbers between 1 and 9999 can be searched.

#### (2) External offset correction

External offset values may be added to or replace the offset values stored under the currently specified offset number. 0 to  $\pm 7.999$  mm (BCD data), 0 to .7999 in. (Binary data).

#### (3) External work coordinate system correction

External data can be added to the shift-value memory for G54 through G59 for the specified axis. 0 to  $\pm 7.999$  mm (BCD data), 0 to  $\pm 32767$  in. (Binary data)

### 4.33 SKIP FUNCTION (G31)

G31 X... Y... Z... F... ;

With this command, a special linear interpolation is executed. While the tool is in linear motion under this command, the motion is interrupted immediately upon receipt by the control of a skip signal, and the program advances to the next block.

### 4.34 STORED STROKE LIMIT (G22, G23)

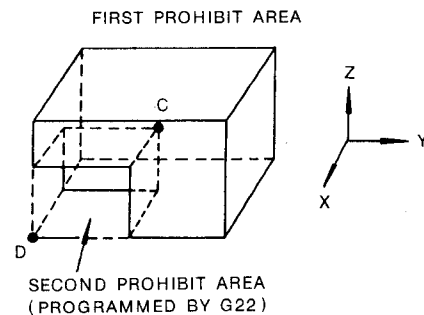
This function is for checking the current tool position during manual and automatic operations for clearance from the prohibited area established by a G22 command in all axes. When the tool enters the prohibited area, the operation is interrupted, and an error message is displayed.

#### (1) 1st prohibited area

The outside of the boundary specified by parameters is the 1st prohibited area, which is usually used as a substitute for overtravel limit switches.

#### (2) 2nd prohibited area

The boundary for the 2nd prohibited area is specified by parameters or by programming G22. The inside or the outside of the boundary may selectively be made as the prohibited area by parameter setting.



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

(Point C) (Point D)

With this command, the tool position check for clearance from the 2nd area is started.

G23 ;

With this command, the check function is cancelled.

### 4.35 STORED LEADSCREW ERROR COMPENSATION

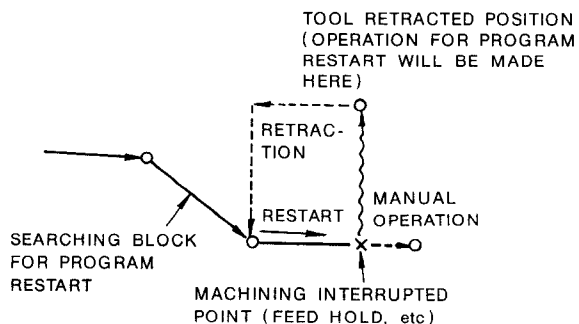
This function is for compensating for the pitch error in the ball screw of the machine. The compensation data is stored by parameter setting in the control in advance. Up to 512 points for all axes can be corrected, and number of correction points for each axis is specified by parameters arbitrarily.

### 4.36 USER MESSAGE DISPLAY

Any message programmed by the user can be displayed on the CRT. In a part program, when #8000 = n [< alarm message >] ; is programmed, 3-digit alarm number "n" and alarm message (up to 32 characters) can be specified.

### 4.37 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.

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 instance. Program restart of type Q is used if the coordinate system is changed by any of the following operations performed after the interrupting automatic operation.

### 4.38 PROGRAM INTERRUPTION (M90, M91)

M91 P... ;

When this command is input once, then during the execution of all subsequent instructions, when a program interruption signal is received, the current program is interrupted (motion is stopped after deceleration) and the program jump to the instruction having the program number specified by P. This function remains effective until M90 command is issued.

M90 ; is the command for cancelling the program interruption function.

M Code	Meaning
M90	Program interruption off
M91	Program interruption on

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

When a fault detection signal during cutting is connected to this function, faults can be handled automatically.

### 4.39 PLAYBACK FUNCTION

Turning on the PLAYBACK switch in the manual operation mode makes the special edit mode.

In this mode, the current axis position during movement can be stored as command value in the part program.

### 4.40 EXTERNAL INPUT, COLLATION, AND OUTPUT

Deletion, input, collation, and output of part program can be commanded to the part program stored in the control by external contact input. To execute this function, RS232C interface is used as the transmission line of part program data. Data input and output interface should be provided.

### 4.41 TOOL LIFE CONTROL (G122, G123)

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.

Maximum number of tools to be controlled .. 256

Maximum number of groups to be registered ... 128

Maximum number of tools per group ... 16

### 4.42 COORDINATE ROTATION

G17 }  
G18 } G68 a ... b ... R...;  
G19 }

With this command, the following move commands will be executed through the rotation to the angle (0.001 deg) specified by R with point (a, b) as a center. The command G69; cancels the coordinate rotation mode.

#### 4.43 LOCAL COORDINATE SYSTEM SETTING

G52 Q2 X ... Y ... Z ... α ... β ...;  
 This command sets the local coordinate system. This is the system shifted by the commanded value from the work coordinated system. After setting, the tool moves in the new coordinate system. To cancel this mode, command G52 Q2 X0 Y0 Z0 α0 β0;

#### 4.44 AUTOMATIC OPERATION MODE HANDLE OFFSET

This function synchronizes the movement by manual handle with the automatic operation during tape operation, MDI operation, and memory mode operation. This can offset the shifted distance due to mounting of the workpiece.

### 5 BUILT-IN TYPE PROGRAMMABLE CONTROLLER (PC)

- (1) Process time (Approx 2.7 μsec/step)
  - High-speed scanning time -- 8 msec
  - Low-speed scanning time -- 8 msec × n
- (2) Number of I/O points (basic/optional)
  - (a) Standard general-purpose I/O modules  
 Type JANCD-FC810 (Max. 3 modules)
    - Input — 112 points/module
    - Output — 96 points/module
  - (b) Mini general-purpose I/O modules  
 Type JANCD-SP50 (contained in 9" CRT panel)

- (3) Main program function
  - (a) Register (internal relay):  
 500 maximum (4000 points)
  - (b) Timer: 90 maximum
  - (c) Keep memory (keep relay):  
 900 maximum (7200 points)
- (4) Message display (optional)

Alarm messages can be displayed on the CRT display by sequence programs. (Macro instruction "SUBP23")

	SP50-1 (points/module)	SP50-2 (points/module)
Input	64	64
Output	32	56

Total number of combined standard and mini I/O modules is 4 maximum.



## APPENDIX 1 LIST OF DATA

Table 1.1 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 1.2 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	(	Control out (Comment start)	EIA: special code
2-4-7	)	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	
/	/	Optional block skip	
Del	DEL	Disregarded (In- cluding All Mark)	
.	.	Decimal point	
Param- eter setting	#	Sharp (Variable designation)	
*	*	Asterisk (Multipli- cation operator)	
=	=	Equal mark	
[	[	Left bracket	
]	]	Right bracket	
\$	\$	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 parameter.

Table 1.3 Tape Format

No.	Items	Metric		Inch		B: Basic O: Optional
1	Program No.	04		04		B
2	Sequence No.	N4		N4		B
3	G Function	G3		G3		B
4	Coordinate word Linear axis Rotary axis	a+	a+	a+	a+	B O
		53	44	53	44	
		b+	b+	b+	b+	
		53	53	53	53	
5	Feed per min	F50	F31	F50	F41	B
6	Feed per 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

## APPENDIX 1 LIST OF DATA (Cont'd)

Table 1.4 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 Rotary axis		±99999.999 mm ±99999.999 deg	±3937.0078 in. ±99999.999 deg	±99999.999 mm ±99999.999 deg	±9999.9999 in. ±99999.999 deg
5	Feed per minute	F	1 to 30000 mm/min	0.1 to 1181.1 in./min	1 to 76200.0 mm/min	0.1 to 3000.0 in./min
6	Feed per minute I/10	F	0.1 to 30000.0 mm/min	0.01 to 1181.10 in./min	0.1 to 76200.0 mm/min	0.01 to 3000.00 in./min
7	S-function	S2	0 to 99	0 to 99	0 to 99	0 to 99
		S5	0 to 99999	0 to 99999	0 to 99999	0 to 99999
8	T-function	T2	0 to 99	0 to 99	0 to 99	0 to 99
		T4	0 to 9999	0 to 9999	0 to 9999	0 to 9999
9	M-function	M	0 to 199	0 to 199	0 to 199	0 to 199
10	Tool offset No.	H	0 to 99	0 to 99	0 to 99	0 to 99
		D	0 to 99	0 to 99	0 to 99	0 to 99
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 1.5 List of G Codes

G code	Group	Function	B: Basic O: Optional
G00	01	Positioning (rapid feeding)	B
G01		Linear interpolation	B
G02		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		02	XY plane designation
G18	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	
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
G33	01	Thread cutting	O
G36	*	Automatic centering	O
G37		Automatic centering	O
G38		Z-axis reference surface offset	O
G40	07	Tool radius compensation cancel	O
G41		Tool radius compensation, left	O
G42		Tool radius compensation, right	O
G43	08	Tool length compensation, plus direction	B
G44		Tool length compensation, minus direction	B
G49		Tool length compensation, cancel	B
G45	*	Tool position offset, extension	B
G46		Tool position offset, retraction	B
G47		Tool position offset, double extension	B
G48		Tool position offset, double retraction	B
G50	15	Scaling OFF	O
G51		Scaling ON	O
G52	12	Return to base coordinate system, Local coordinate system setting	O
G53	*	Temporary shift to machine coordinate system	O
G54	12	Shift to work coordinate system 1	O
G55		Shift to work coordinate system 2	O

## APPENDIX 1 LIST OF DATA (Cont'd)

Table 1.5 List of G Codes (Cont'd)

G code	Group	Function	B: Basic O: Optional	
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	
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		Coordinate rotation mode	O	
G69		Coordinate rotation mode cancel	O	
G70	*	Bolt hole circle	O	
G71		Arc	O	
G72		Line-at-angle	O	
G73	09	Canned cycle 10	O	
G74		Canned cycle 11	O	
G76		Canned cycle 12	O	
G77		Canned cycle 13	O	
G80		Canned cycle cancel	O	
G81		Canned cycle, 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		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 coordinate	B	
G98	10	Return to initial point for canned cycles	O	
G99		Return to point R for canned cycles	O	
G122	17	Tool register start	O	
G123		Tool register end	O	
G124	*	Tool register cancel	O	

**Notes:**

- The codes marked with ◻ are automatically selected at power on or reset.
- G codes of \* group are non-modal.  
They should not be commanded a together with the other G codes in one block.

## APPENDIX 2 DIMENSIONS in mm (inch)

Due to ongoing product modification/improvement, dimensions and specifications are subject to change without notice.

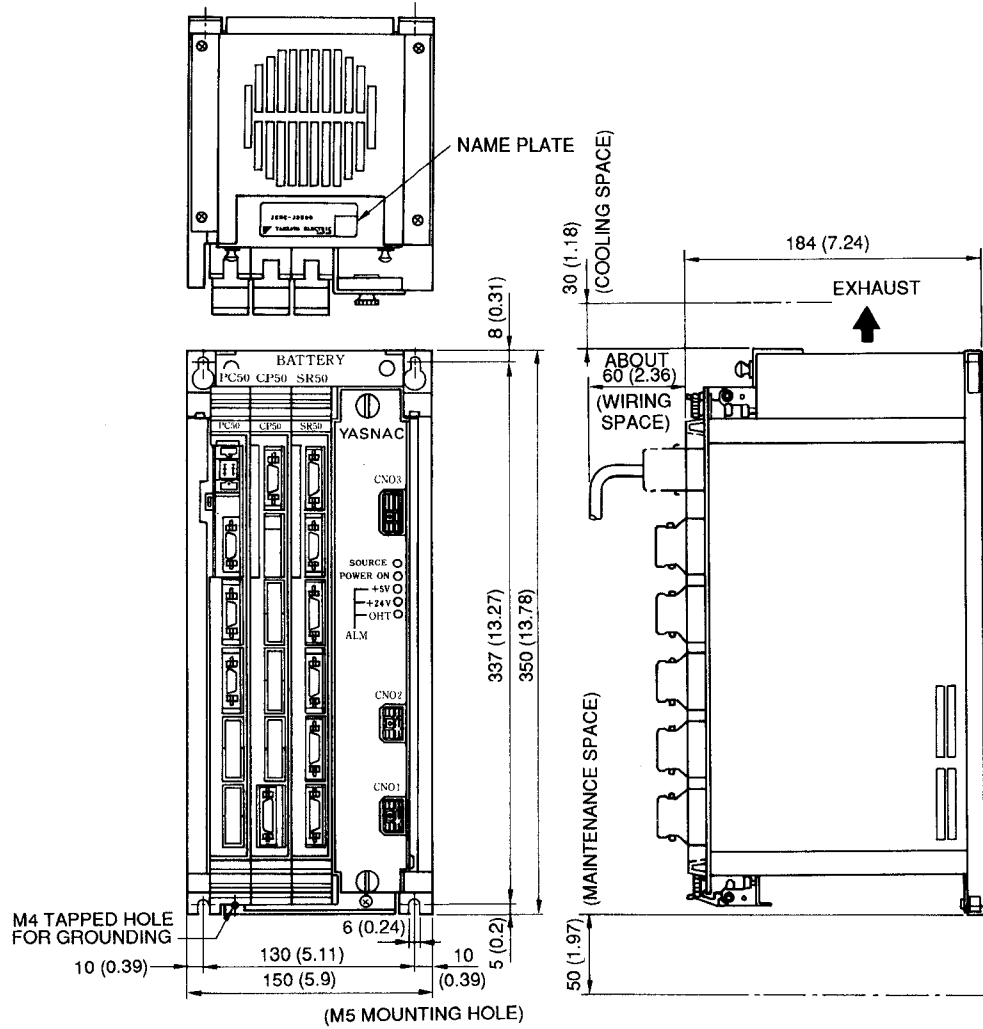


Fig. 2.1 Module Type CPU Rack

## APPENDIX 2 DIMENSIONS in mm (inch) (Cont'd)

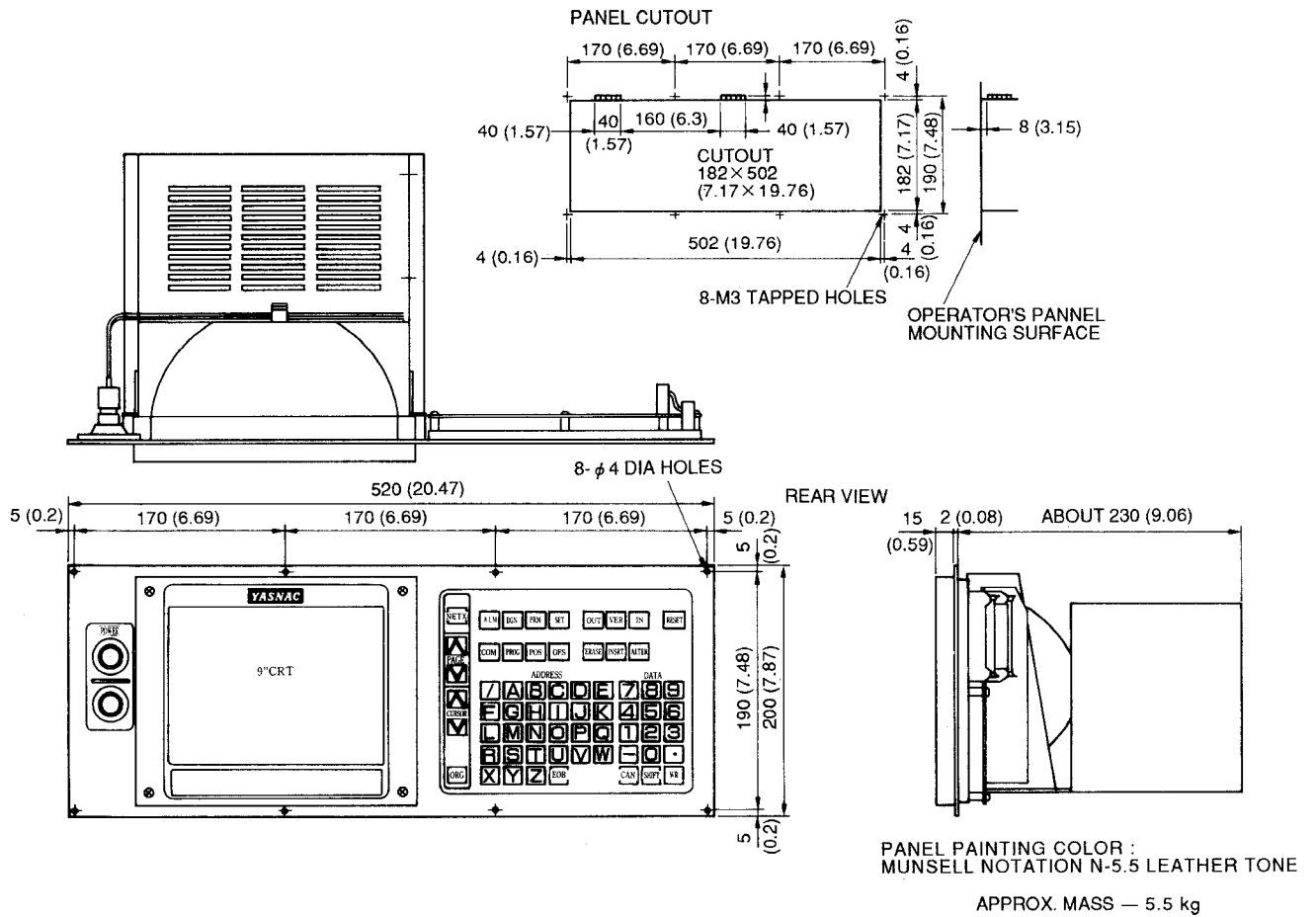


Fig. 2.2 NC Operator's Station with 9" Monochromatic CRT Display  
(Keyboard on right side of CRT) — with Power On/Off Pushbutton

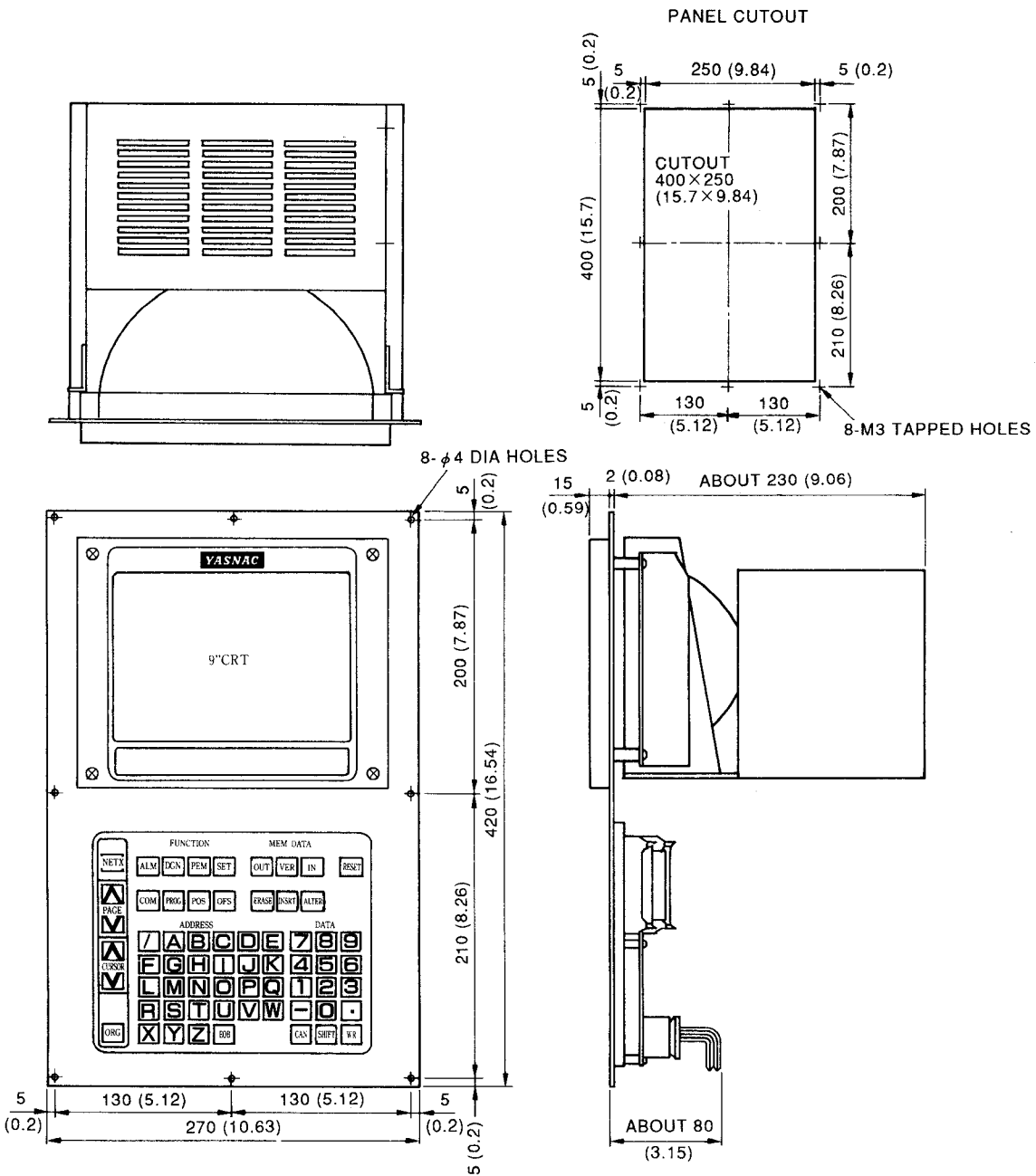


Fig. 2.3 NC Operator's Station with 9" Monochromatic CRT Display (Keyboard below CRT)



## APPENDIX 2 DIMENSIONS in mm (inch) (Cont'd)

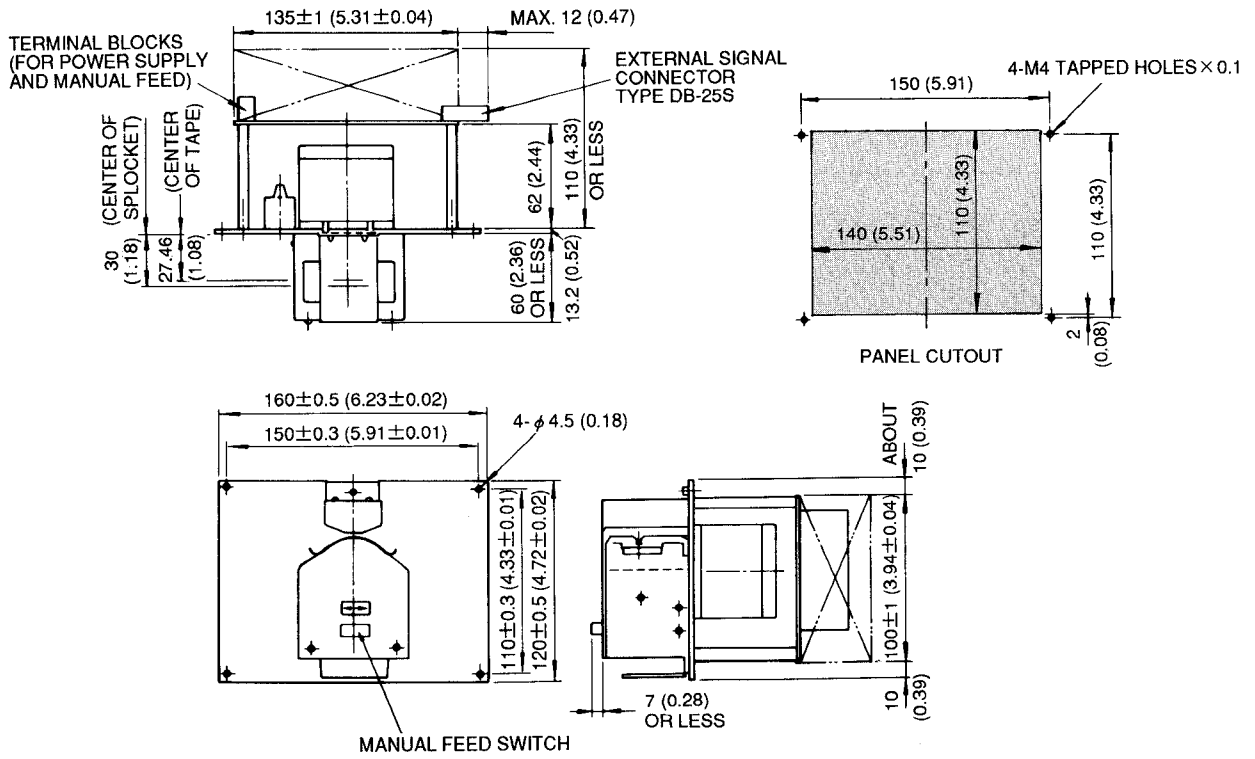


Fig. 2.4 Tape Reader Unit

**MEMO**

# YASNAC J50M

## DESCRIPTIVE INFORMATION

**TOKYO OFFICE** New Pier Takeshiba South Tower, 1-16-1, Kaigan, Minatoku, Tokyo 105 Japan  
Phone 81-3-5402-4511 Fax 81-3-5402-4580

**YASKAWA ELECTRIC AMERICA, INC.**

**Chicago-Corporate Headquarters** 2942 MacArthur Blvd. Northbrook, IL 60062-2028, U.S.A.

Phone 1-847-291-2340 Fax 1-847-498-2430

**Chicago-Technical Center** 3160 MacArthur Blvd. Northbrook, IL 60062-1917, U.S.A.

Phone 1-847-291-0411 Fax 1-847-291-1018

**MOTOMAN INC.**

805 Liberty Lane West Carrollton, OH 45449, U.S.A.

Phone 1-937-847-6200 Fax 1-937-847-6277

**YASKAWA ELÉTRICO DO BRASIL COMÉRCIO LTDA.**

Avenida Brigadeiro Faria Lima 1664-5° CJ 504/511, São Paulo, Brazil

Phone 55-11-815-7723 Fax 55-11-870-3849

**YASKAWA ELECTRIC EUROPE GmbH**

Am Kronberger Hang 2, 65824 Schwalbach, Germany

Phone 49-6196-569-300 Fax 49-6196-888-301

**Motoman Robotics AB**

Box 504 S38525 Torsås, Sweden

Phone 46-486-48800 Fax 46-486-41410

**Motoman Robotec GmbH**

Kammerfeldstraße 1, 85391 Allershausen, Germany

Phone 49-8166-900 Fax 49-8166-9039

**YASKAWA ELECTRIC UK LTD.**

Unit2 Centurion Court Brick Close, Kiln Farm, Milton Keynes MK11 3JA, United Kingdom

Phone 44-1908-565874 Fax 44-1908-565891

**YASKAWA ELECTRIC KOREA CORPORATION**

Paik Nam Bldg. 901 188-3, 1-Ga Euljiro, Joong-Gu Seoul, Korea

Phone 82-2-776-7844 Fax 82-2-753-2639

**YASKAWA ELECTRIC (SINGAPORE) PTE. LTD.**

151 Lorong Chuan, #04-01, New Tech Park Singapore 556741, Singapore

Phone 65-282-3003 Fax 65-289-3003

**YATEC ENGINEERING CORPORATION**

Shen Hsiang Tang Sung Chiang Building 10F 146 Sung Chiang Road, Taipei, Taiwan

Phone 886-2-563-0010 Fax 886-2-567-4677

**BEIJING OFFICE** Room No. 301 Office Building of Beijing International Club, 21

Jianguomenwai Avenue, Beijing 100020, China

Phone 86-10-532-1850 Fax 86-10-532-1851

**SHANGHAI OFFICE** 27 Hui He Road Shanghai 200437 China

Phone 86-21-6553-6060 Fax 86-21-6553-6060

**YASKAWA JASON (HK) COMPANY LIMITED**

Rm. 2909-10, Hong Kong Plaza, 186-191 Connaught Road West, Hong Kong

Phone 852-2803-2385 Fax 852-2547-5773

**TAIPEI OFFICE** Shen Hsiang Tang Sung Chiang Building 10F 146 Sung Chiang Road, Taipei, Taiwan


Phone 886-2-563-0010 Fax 886-2-567-4677



YASKAWA

YASKAWA ELECTRIC CORPORATION

SIE-C843-12.30

© Printed in Japan July 1997 95-2 0.2TA 

*Due to ongoing product modification/improvement, data subject to change without notice.*

94-C84-114, 94-C84-123, 94-C84-126