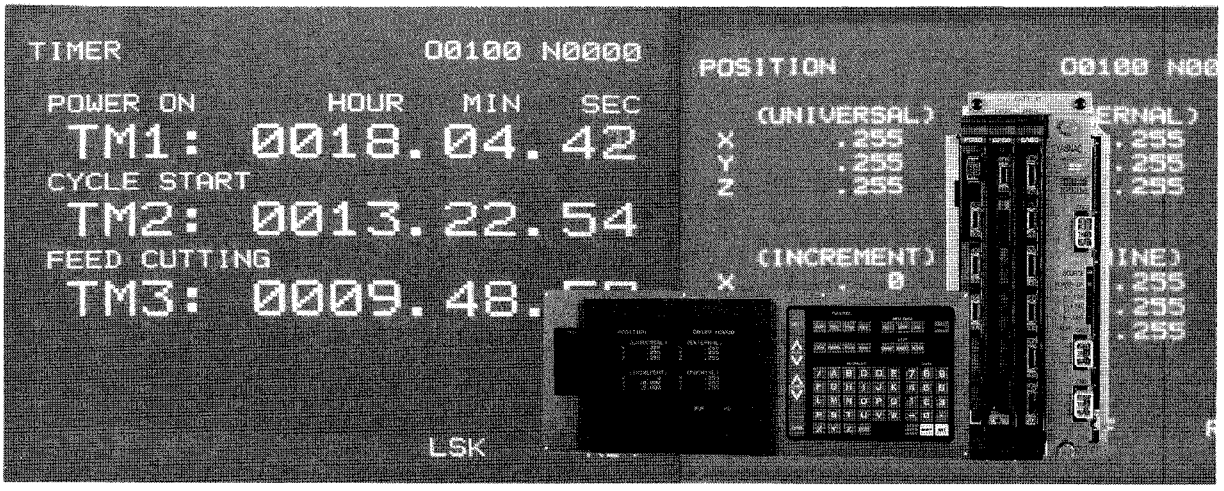
YASNAC J50L
DESCRIPTIVE INFORMATION

CNC SYSTEM FOR TURNING APPLICATIONS



1 INTRODUCTION

The YASNAC J50L is a high-performance CNC for the simultaneous control of 2 or 3 axes of a lathe machining tools, with emphasis placed on high-speed machining, and programming capability.

FEATURES

1. U1tra-high-speed Performance
   "High-speed, computing system" is achieved by installing a 32-bit micro-processor in the YASNAC J50L.
2. Significant Downsizing (miniaturized)
   YASNAC J50L is significantly downsized because it has surface mounted devices and customized gate arrays.

   This manual explains both basic and optional features of YASNAC J50L as well as the servo system.
   You can determine your own hardware requirements after carefully reading this manual.

This manual is subject to change without notification due to product improvements, model changes, etc.

CONTENTS

INDEX

INDEX (Cont'd)

INDEX (Cont'd)

2 BASIC FEATURES

2.1 CONTROLLED AXES

2 Axes (X and Z)

2.2 SIMULTANEOUS CONTROLLABLE AXES

2 Axes for both automatic and manual operations, except with manual pulse generator.

2.3 LEAST INPUT INCREMENT

It is the minimum programmable length that can be input from tapes or MDI.

	x1	x10 (10 times the input increment)
Metric Input	0.001 mm	0.01 mm
Inch Input	0.0001 inch	0.001 inch

X-axis is specified for diameter.

Notes:

1. Inch or metric input can be selected at the setting. Selection of G20/G21 is optional.
2. Selection of x 1 or x 10 is made by setting parameters.
3. X-axis can be specified in radius by parameter setting.

2.4 LEAST OUTPUT INCREMENT

It is the minimum unit of movement through which the machine can move.

	X-axis (Radius Value)	Z-axis
Metric Output	0.0005 mm	0.001 mm
Inch Output	0.00005 inch	0.0001 inch

Note: Inch or metric output is selected by parameter setting.

2.5 MAXIMUM PROGRAMMABLE DIMENSIONS

The incremental and absolute commands can specify the movement in each axis in the following ranges.

Metric Output	Metric input	± 99999.999 mm
	Inch input	± 3937.0078 inch
Inch Output	Metric input	± 99999.999 mm
	Inch input	± 9999.9999 inch

2.6 TAPE CODE

EIA.RS-244-A and ISO 840.

Note: Refer to Tables 1.1 and 1.2 in Appendix 1.

2.7 EIA·ISO AUTO-RECOGNITION

Either EIA or ISO can be read by manually setting applicable parameters. By setting the automatic identification parameters, EOB code is read and the code in use is automatically sensed.

2.8 TAPE FORMAT

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

2.9 DECIMAL POINT PROGRAMMING

Numerical values containing a decimal point can be input from punched tapes and MDI. Addresses with which decimal points can be used are as follows:

Coordinates	Angles	Feedrate	Dwell Time
X, Z, U, W, I, K, R	A, B	F, E	U, P

2.10 BUFFER REGISTER

(1) During the ordinary automatic operation, data is read in one block ahead, processed for, say offset, and stored in the buffer register for the succeeding operation.

(2) In the tool radius compensation (optional) mode, data is read in two blocks ahead (when necessary four blocks ahead), processed for compensation, and stored in the register for the following operation.

(3) Preceding data leading is not done in blocks with the following M codes,

M00, M01, M02, and M30,

as well as in the parameter-specified preceding-read-inhibit M codes (up to 6).

2.11 RAPID TRAVERSE RATE AND FEEDRATE

The rapid traverse feed, manual feed, and rapid feed override F_0 can be set to the upper limit shown below.

Metric Input	30,000 mm/min
Inch Input	1181.1 inches/min

Notes:

1. Depending on the motor and machine systems, the upper limit is further restricted.
2. The upper limit for X-axis speed is half the above.

2.12 FEED FUNCTION (F-, E-FUNCTION)

(1) Feed per revolution (G99 mode)

F (normal feed) and E (accurate feed) commands can specify the tool feed rate per rotation of spindle (mm/rev or inch/rev).

		Format	Range of Feed Per Revolution
Metric Output	Metric Input	F 32	F 0.01 – F 500.00 mm/rev
		E 34	E 0.0001 – E 500.0000 mm/rev
	Inch Input	F 24	F 0.0001 – F 19.6850 inches/rev
		E 26	E 0.000004 – E 19.685000 inches/rev
Inch Output	Metric Input	F 32	F 0.01 – F 1270.00 mm/rev
		E 34	E 0.0003 – E 1270.0000 mm/rev
	Inch Input	F 24	F 0.001 – F 50.0000 inches/rev
		E 26	F 0.000010 – E 50.000000 inches/rev

The feedrates are limited by spindle-speed S as follows:

Metric Output	F(E) x S≦24,000 mm/min
Inch Output	F(E) x S≦2,400 inches/min

(The upper limit of X-component of speed is half of the above.)

(2) Feed per minute (G98) mode

F command specifies the tool feedrate per minute as follows:

		Format	Range of Feed per Minute
Metric Output	Metric Input	F 50	F 1. – F 24000. mm/min
	Inch Input	F 32	F 0.01 – F 944.88 inches/min
Inch Output	Metric Input	F 50	F 1. – F 60960. mm/min
	Inch Input	F 42	F 0.01 – F 24000.00 inches/min

Note:

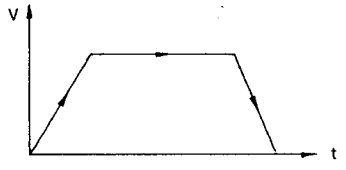
1. Depending on the conditions of the motor or machine system, the upper limit of mm/min and inch/min is further restricted.
2. The upper limit for X-component of speed is the half of the above values.

2.13 AUTOMATIC ACCELERATION/ DECELERATION

The following acceleration/deceleration is done automatically.

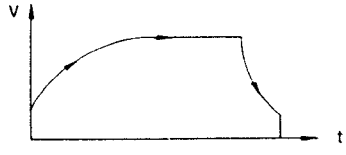
(1) In positioning and manual feed

Linear automatic acceleration/deceleration is done, independently for each axis.



(2) In machining feed

Exponential automatic acceleration/deceleration is common to each axis. In thread cutting and normal feed, this can be set independently for each axis.



2.14 FEEDRATE OVERRIDE AND FEEDRATE OVERRIDE CANCEL

(1) Rapid traverse feed override

Rapid traverse rate can be modified to F_0 and 25, 50, and 100% of the original traverse rate. The F_0 is a constant speed set by parameter.

(2) Feed override

The F-commanded feedrate can be modified in the range of 0 to 200% in 10% increments.

(3) Override cancel

Turning this switch on cancels any override effect, causing the tool to move at the originally specified speed.

2.15 PREPARATORY FUNCTION (G-FUNCTION)

The address G and the following numerals up to 3 digits specify a block and its meaning. For details of G-codes, refer to Table 1.6 in Appendix 1.

(1) Ordinary G-codes

a. G codes in 01 to 11 groups are modal. Once specified, they are effective until other G-codes in the same group are specified.

b. G codes in *-marked groups are non-modal, and effective only in the specified block.

(2) Special G codes

a. Special G code I can be used in the basic mode (by parameter switching).

b. Special code II is optional.

2.16 ABSOLUTE/INCREMENTAL PROGRAMMING (G90/G91)

(1) Absolute programming (X, Z, G90)

Addresses X and G are used to specify an absolute value. In use of special code I or II, X and Z commands in G90 mode specify an absolute value. However, in G90 mode, address U and W remain as incremental commands.

(2) Incremental programming (U, W, G91)

Addresses U and W are used to specify an incremental value. In the use of special code I or II, X and Z commands in G91 mode specify an incremental value.

(3) Combined command

In the same block,

X··· W···; or U··· Z;

can be used as a combined command. In the use of G90 or G91, however, both cannot be specified in the same block.

(4) Addresses I, K, and R for circular interpolation are invariably incremental values.

2.17 PROGRAMMING OF ABSOLUTE ZERO POINT (G50)

(1) G50 X··· Z··· ;

This command establishes the absolute coordinate system (= coordinate system) such that the current tool position becomes the specified coordinate value.

(2) G50 U··· W··· ; (incremental setting)

This command establishes a new coordinate system in which the coordinate system already established with G50 has been shifted by incremental value U. W.

2.18 POSITIONING (G00)

G00 X(U)··· Z(W)··· ;

This command moves the tool to the specified position at rapid traverse rate and independently for each axis. The travel is not necessarily linear.

The G00 is a modal G-code. In G00 positioning, pulse distribution is started after ERROR DETECT ON, and after distribution, when ERROR DETECT ON is again detected, operation goes to the next block.

2.19 POSITIONING IN ERROR DETECT OFF MODE (G06)

G06 X(U)··· Z(W)··· ;

Positioning by this command differs from G00 in the following points:

a. G06, being non-modal, is effective only in the specified block.

b. G06 starts pulse distribution without ERROR DETECT check, and after distribution is completed, immediately goes to the next block.

In G06 positioning, the corners of workpiece are slightly rounded.

Note: ERROR DETECT ON means the state where the servo-lag pulses are reduced to a permissible number. At this time the command pulse position and the actual tool position nearly coincide.

2.20 LINEAR INTERPOLATION (G01)

G01 X(U)··· Z(W)··· F(E)··· ;

This command moves the tool to the specified target position along a straight line at the specified feed rate.

2.21 CIRCULAR INTERPOLATION (G02, G03)

(1) G02(G03)
X(U)··· Z(W)··· I··· K··· F(E)··· ;

This command moves the tool to the specified end position along the specified circular path.

G02: Circular interpolation in clockwise (CW)

G03: Circular interpolation in counterclockwise (CCW)

X(U), Z(W): End position

I, K: X- and Z-components of the center of the circular path with respect to the starting point.

F(E): Feedrate in the tangential direction of the arc.

(2) Circular interpolation is possible across multiple quadrants or along the full circle.

Notes: Optionally, a circular path can be specified by radius R.

2.22 DWELL (G04)

(1) G04 U(P)··· ;

This command causes the tool to remain motionless for the time specified by U or P before the program goes to the next block. The dwell can be increased to a maximum of 8388.607 seconds in increment of 0.001 second. A numerical value containing decimal point can be specified.

(2) G04 U3.5 ; --- A3.5-seconds dwell

2.23 REFERENCE POINT RETURN CHECK (G27)

G27 X(U)··· Z(W)··· ;

This command moves the tool to the specified intermediate position at rapid traverse rate and both axes at the same time, then check if that point is the "reference point." If they do not coincide even in one axis, the automatic operation is stopped as error occurrence. If coincident, the REFERENCE POSITION" lamp lights.
The check is not carried out for unspecified axes.

Note: The reference point is the unique position of machine tools to which the tool is returned by "manual return to reference point" or by "G28 automatic return to reference point."

2.24 AUTOMATIC RETURN TO REFERENCE POINT (G28)

(1) G28 X(U)··· Z(W)··· ;

This command automatically returns the tool to the reference point. The tool is positioned at the specified intermediate position both axes at the same time, then undergoes "Reference Point Return Operation" both axes at the same time.

(2) Reference-point-return operation is as shown below.

a. First time after power is on, the operation is performed in a low-speed mode as shown below.

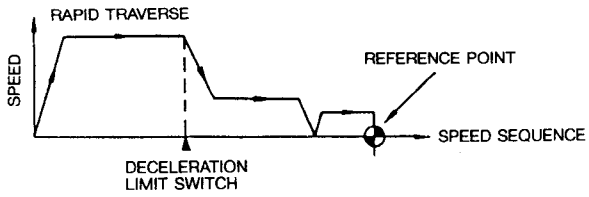


Fig. 1

b. Second time and beyond, the tool returns to the reference point in the same rapid traverse positioning as G00 command.

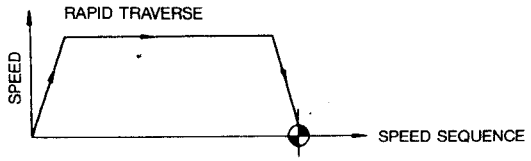


Fig. 2

Note: By parameter setting, it is possible to apply the same low-speed return as Figure 1 at second time and thereafter.

2.25 RETURN FROM REFERENCE POINT (G29)

The tool returned to the reference point by G28 command is moved by the command of

G29 X(U)··· Z(W)··· ;

to the previously specified intermediate position determined by multiple G28 commands, at rapid traverse rate and both axes at the same time; then the tool is moved to the position specified by the G29 command, with both axes at the same time. No movement takes place for unspecified axes.

Note: G29 can be used for a return from the second reference point (option) by G30, like a return in G28 command.

2.26 MULTI-START THREADCUTTING (G32)

(1) G32 X(U)··· Z(W)··· F(E)··· ;

This command allows straight thread, taper thread, and scroll thread to be cut.

Type of Thread	Commands
Straight Thread	G32 Z(W)··· F(E) ··· ;
Taper Thread	G32 X(U) ··· $\begin{matrix} Z(W)\cdots \\ F(E)\cdots \end{matrix}$;
Scroll Thread	G32 X(U) ··· F(E) ··· ;

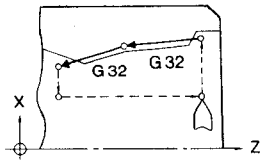
F specifies ordinary thread lead; E precise thread lead. The range of thread lead specification is the same as the command range of feed per revolution (mm/rev or inch/rev) in 2.14 Feed Function.

(2) Continuous thread cutting

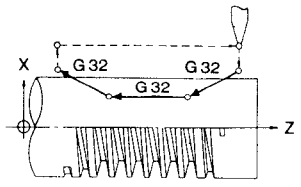
G32 X(U)··· Z(W)··· F(E)··· ;
X(U)··· Z(W)··· ;
X(U)··· Z(W)··· ;

With this type of continuously programmed block command for threading, thread cutting is allowed to continue to the next thread cutting operation, reducing the stop time to "0."

Examples:



(a) Pipe joint



(b) Chamfering

Note:
If thread lead specification is changed midway, then the thread becomes irregular near the boundary of blocks.

2.27 CANNED CYCLES (G90, G92, G94)

(1) Turning cycle A

G90 X(U)··· Z(W)··· I··· F(E)··· ;

This command performs outer diameter straight and taper cutting cycle.

(2) Threading cycle

G92 X(U)··· Z(W)··· I··· F··· ;

This command performs straight and taper cutting cycle. Thread is cut when thread-cut input (CDZ) is ON. The length of thread can be set by parameter in the range of 0 to 25.5L in 0.1L increment, where L is lead. An M-code output is generally used to on/off input signal CDZ.

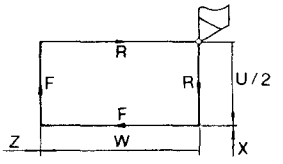
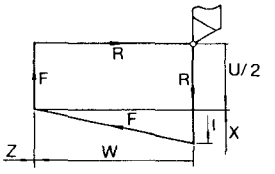
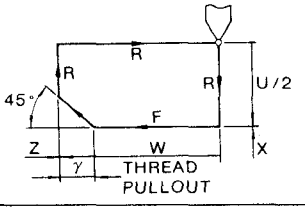
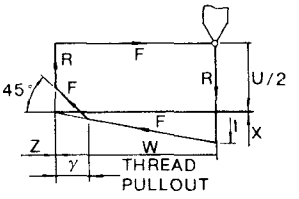
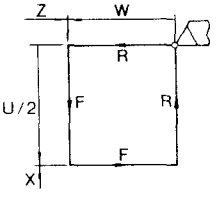
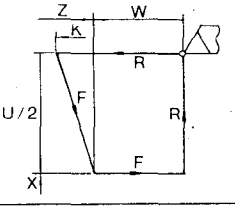
(3) Facing cycle B

G94 X(U)··· Z(W)··· K··· F(E)··· ;

This command performs front surface and front surface taper cutting cycle.

2.27 CANNED CYCLES (G90, G92, G94) (Cont'd)

Canned Cycle Operation

G Code	Straight Cycle	Taper Cycle
G90 Turning Cycle A	G90 X(U)··· Z(W)··· F(E)··· ; 	G90 X(U)··· Z(W)··· I ··· F(E)··· ; 
G92 Threading Cycle	G92 X(U)··· Z(W)··· F(E)··· ; 	G92 X(U)··· Z(W)··· I ··· F(E)··· ; 
G94 Facing Cycle B	G94 X(U)··· Z(W)··· F(E)··· ; 	G94 X(U)··· Z(W)··· K··· F(E)··· ; 

2.28 FEED FUNCTION DESIGNATION (G98, G99)

(1) G98 ;

This command specifies the feed per minute (mm/min, inch/min) mode.

(2) G99 ;

This command specifies the feed per revolution (mm/rev, inch/rev) mode. Parameters are used to select which code is initially to be set at power ON.

2.29 SPECIAL G-CODE I

Instead of the standard G code, the special G code I can be used by parameter setting. For details of the special G code I, refer to Table 1.6 List of G Codes in Appendix 1. Only G codes change; no change in the function occurs.

2.30 MISCELLANEOUS FUNCTION (M-FUNCTION)

The address M and the following numerals up to three digits are used to command the following miscellaneous functions:

(1) The following are used to stop read ahead and for decode output as well as 2-digit BCD output.

M00: Program stop
M01: Optional stop
M02: Program end
M30: Tape end

(2) The following are used for internal processing (mark * indicates option).

* M92: 1-block buffering
* M93: 4-block buffering
* M96: Tool radius compensation, circular path
* M97: Tool radius compensation, intersection computing mode
* M98: Subprogram call
* M99: Subprogram end

(3) The following are for 2-digit BCD output.

M-codes other than above.

(4) A maximum of 6 M-codes to stop read ahead can be specified by parameter setting.

Note: 3-digit BCD output is optionally available.

2.31 SPINDLE-SPEED FUNCTION (S-FUNCTION)

Spindle speed function called S 4-digit programming A is executed.

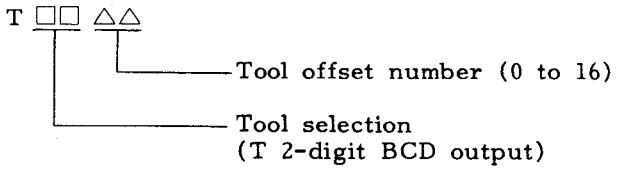
(1) The spindle rpm can be designated by command consisting of address S followed by 4 digits, instead of the basic S 2-digit command. As the output to the machine (spindle), an analog voltage (±10 V max.) is output from the DA converter.

(2) The control makes necessary computation to meet the programmed rpm, outputs the signals (up to 4 signals) to shift the spindle gear ratio, and outputs an analog voltage or 12-bit binary signal suited to the shifted gear ratio. The speed ranges and other required data are set by parameters.

(3) The maximum spindle rpm can be designated by G50S command.

2.32 TOOL FUNCTION (T-FUNCTION)

The address T and the following 4-digit numerals are used to specify tool selection and tool offset number. This is T 4-digit programming.



Selecting this option automatically provides 16 sets of offset memories corresponding to the offset number.

2.33 TOOL POSITION OFFSET

(1) When T function specifies the tool offset number, the content of the tool offset memory corresponding to the specified offset number is algebraically added to the program-specified coordinate value in both X- and Y-axes, and the tool is moved to this corrected position.

(2) In the basic mode, offset memory can be specified up to 16 sets, 1 to 16. Specifying 0 cancels the tool position offset.

(3) Using MDI, initially write the range of tool offsets in the offset memory. Absolute values can be written by the use of X and Z address keys.

Note: Offset memory can be extended to 50 sets by installation of optional T 4-digit command. Refer to "Additional offset memory (optional)."

2.34 BACKLASH COMPENSATION

The lost motion of a machine can be compensated in the range of -8192 to 8192P with each axis indepenently, where P is the least output increment. The value of compensation is initially set in parameter.

2.35 MANUAL FEED

Manual feed is possible in the following three modes, with both axes at the same time.

(1) Manual rapid traverse (RAPID)

The tool moves at the rapid traverse rate, independently in each axis,

(2) Manual jog feed (JOG)

The tool moves at the speed specified by JOG FEEDRATE selection switch, independently in each axis. Any speed curve can be specified in the specifications.

(3) Manual step feed (STEP)

The tool moves step by step in the increment value specified by the MANUAL PULSE MULTIPLY switch.

Multiplication	x1	x10	x100	x1000	x10000	x100000
Metric Input	0.001	0.01	0.1	1.	10.	100. mm/step
Inch Input	0.0001	0.001	0.01	0.1	1.	10. in./step

2.36 MANUAL RETURN TO REFERENCE POINT

After the REFERENCE POINT switch is turned on, the tool can be returned to the reference point by manual operation. Like"2.24 Automatic Return to Reference Point (G28)," the return to the reference point can be made in the low-speed mode by parameter setting.

2.37 BUILT-IN TYPE NC OPERATOR'S STATION

The NC operator's station is provided with 9" monochromatic CRT display (keyboard on right side of CRT). Dimensions are shown in Fig. 2.5.

NC operator's station consisting of membrane keyboard and 9" CRT can permit the efficient writing and displaying of a variety of data.

(1) Display: 32 characters × 16 characters/screen, Monochromatic display

(2) MDI: Display and writing-in any mode of alarm, diagnosis, parameter, setting, command, program, position, or offset.

2.38 PART PROGRAM STORAGE AND EDITING

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

(1) Storage 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, to collate them with punched cards, and store them from tape readers (Note 3.) (option).

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

Notes:
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 "RS232C interface" is required.

2.39 SUBROUTINE PROGRAM (M98, M99)

(1) Subprogram Call (M98)

M98 P··· Q··· L··· ;

With this command, the subprogram starting with sequence number Q is called from the part programs having the program number specified by P and the subprogram is executed L times.

If P is omitted, the subprogram starting with sequence number Q is called from the main program.

If Q is omitted, the starting subprogram having P-specified program number is called.

If L is omitted, the execution is only once.

(2) Subprogram end (M99)

M99 ;

This command is added to the end of the subprogram to end it. After completion of the subprogram, control returns to the block immediately following the main program that has called the subprogram.

M99 P··· ;

If this command is added to the end of the program, control returns to the P-specified sequence number in the main program.

(3) Multiple call

A subprogram can call nested subprograms up to 4 times successively.

2.40 PARAMETER STORAGE

Parameters for machine constants such as backlash compensation values and rapid traverse rate can be set to determine or change the specifications. Set parameters while the SYSTEM switch No. 1 is set at No. 1 and the control in idle condition.

2.41 SETTING FUNCTION

Any of the functions can be selected to on or off. This is possible if the SYSTEM No. switch is set to normal "0."

2.42 INTERNAL DATA TAPE INPUT

Normally, tool offset values, parameter data, and setting data are input from the MDI. These data may be stored, via RS232C interface, in respective memories.

Note: These stored data may be output to an external equipment by installing optional RS232C interface. (Example output: tape punch out, type out)

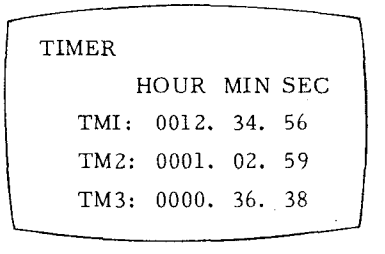
2.43 OPERATION TIME DISPLAY

The cumulative time of the following operations can be displayed:

(1) Total time after switching power supply on: POWER ON

(2) Total time of automatic operation: CYCLE START

(3) Total cutting time (during interpolation moving): FEED



The above time display is stored after power is turned off. The display can be reset to 0 by operation from the panel.

2.44 ADDRESS SEARCH

Through MDI operation, all data can be searched for: on NC tape (TAPE mode) or on part program (MEM, EDT mode). Either single address data or arbitrary data up to 32 characters can be searched. Program numbers and sequence numbers can also be searched for.

Example

```
X15.5 M01;
```

Data consisting of any type and arrangement of characters can be searched.

2.45 PROGRAM NUMBER

A program number can be a maximum of 4-numerical digits following the address 0. It can be written at the head of the program.

2.46 LABEL SKIP

The label skip function is effective and message "LSK" is displayed on the lower part of CRT while:

(1) The power supply is being turned on, and

(2) Control is being reset.

When the label skip function is effective, all the tape information before the first FOB code is ignored. In the MEM or EDIT mode, "LSK" display means that a pointer is at the head of the selected part program.

2.47 CONTROL IN/OUT

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

2.48 SINGLE BLOCK

When the SINGLE BLOCK switch is turned on, automatic operation with tape or memory is performed block by block.

2.49 OPTIONAL BLOCK SKIP

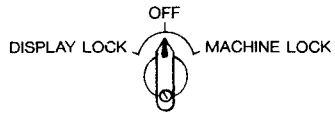
With the OPTIONAL BLOCK SKIP switch on, data is ignored from command "/" of "/1" to EOB on a block.

2.50 DRY RUN

With the DRY RUN switch on, the feedrates for automatic operation (rapid traverse and F(E) specified feedrates) are ignored and the following feedrates are available. This dry run function is used to test programs.

	DRY RUN on	
Feed	Feedrate for manual jog feed	
Rapid Traverse	Rapid traverse or manual jog feedrate	Selectable by setting

2.51 MACHINE LOCK AND DISPLAY LOCK



(1) Off Condition

Normal automatic and manual operations are carried out in off condition.

(2) Machine lock condition

In machine lock condition, NC commands are executed while the machine is standing still. M, S, and T functions operate normally, and the current position is continuously updated and displayed. The same applies to manual operation.

(3) Display lock condition

In display lock condition, the machine moves normally, but no change occurs in the displays of the current value,external and the current value (optional). These displays do not change if feed is done manually.

2.52 AUXILIARY FUNCTION LOCK

While this switch is on, no BCD code is output for M, S, and T functions. However, the operation is normal for decoding M00, M01, M02, and M30, and for internal handling of M codes (M90 to M109).

Note: The miscellaneous function lock does not affect S-4 digit command (option)

2.53 MANUAL ABSOLUTE ON/OFF

(1) While ON

Manual movement distances are added to the absolute register, and the coordinate system remains unchanged.

(2) While OFF

Manual movement distances are not added, and the coordinate system is shifted in parallel to the movement.

2.54 EDIT LOCK

With this switch on, the following editing operations are inhibited.

a. ERASE, INSERT, and ALTER key operations.

b. NC tape storing operation.

2.55 INTERLOCK

When an interlock signal is on during automatic operation, the tool stops after deceleration in both X and Z axes; when the signal is cleared, the tool resumes the motion.

2.56 RAPID PULL OUT OF THREADING

When this signal is input, thread is cut during thread cutting cycle of G92 and G76x. When the signal is off, thread is not cut. In normal usage, any M codes are output if this signal (CDZ) is turned on and off.

2.57 ERROR DETECT

While this signal is turned on, operation goes to the next block after pulses are distributed for feed and ERROR DETECT is on (square corner). If the signal is off, the operation goes to the next block immediately after pulses are distributed. In normal usage, if any M codes are output, then this signal (SMZ) is turned on and off.

Note: This function is effective only for feed. The error detect for positioning is controlled only by G00 and G06.

2.58 DOOR INTERLOCK

When the door is open, the power supply is turned off.

2.59 FEED HOLD

Depressing this pushbutton temporarily interrupts the tool feed during automatic operations. It does not function, however, to stop thread cutting. The operation is resumed by depressing CYCLE START pushbutton.

2.60 EMERGENCY STOP

Depressing the EMERGENCY pushbutton makes all commands ineffective. The servo power supply is turned off, and all moving members are stopped by dynamic brake.

2.61 OVERTRAVEL

This function is to stop the tool motion by receiving 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.

2.62 REMOTE RESET

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

2.63 REMOTE POWER ON/OFF

In addition to the POWER ON/OFF pushbutton on the NC operator's station, the power can be turned on and off by inputting an external contact signal.

2.64 MACHINE READY INPUT SIGNAL

This signal indicates on the control that the machine is ready for operation. The operation is possible when the signal is received in the condition of "Servo Power ON." When the signal is off during operation, all functions are stopped, with "Machine Unready."

2.65 CONTROL POWER ON OUTPUT SIGNAL

An output signal to indicate that the power is input to the control section.

2.66 SERVO POWER ON OUTPUT SIGNAL

An output signal to indicate that the servo power is normally input in the condition of NC power-on.

2.67 TOOL MOVE OUTPUT SIGNAL AND THREADCUTTING OUTPUT SIGNAL

(1) Moving Signal

An output signal to indicate the tool is moving in automatic operation mode.

(2) Threading Signal

A signal to be output specifically during thread cutting.

2.68 NC ALARM OUTPUT SIGNAL

A signal to be output during any one of alarms except for inpur error. The signal is off immediately after the cause is removed and the reset procedure is followed.

2.69 INPUT ERROR OUTPUT SIGNAL

A signal to be made on by an error relating to program input, such as one in part program parity, format, or numerals. The signal is turned off by reset operation. The signal is not output for a "Simple Error" displayed on CRT.

2.70 NC RESET OUTPUT SIGNAL

A signal to be turned on by pushing the reset key on the NC operator's station or by input of external reset signal.

2.71 EXTERNAL ERROR INPUT SIGNAL

(1) External Error Detect Input 0

When this signal is input, the operation stops after the end of the current block, and alarm is displayed.

(2) External error detect input 1

When this signal is input, the current motion is immediately stopped, and alarm is displayed.

2.72 RS-232C INTERFACE PORT

RS-232C interface is provided to connect tape puncher, separate type tape reader unit and other external equipment.

Interface Type	Serial voltage interface
Transmission Speed	110 to 9600 baud
Connector	DB-25S
Max Cable Length	15 m
Output from Memory	Possible
Storage in Memory	Possible
Tape Mode Operation	Possible

Note : Data which are output from memory or stored in memory are as follows :

- Part program
- Offset data, tool coordinate data and tool wear data
- Setting and parameter data

2.73 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

2.74 POSITION DETECTOR INTERFACE

Position and speed is detectec by feedback signal from the rotary-type pulse generator. (Note 2.) The motion per rotation of the pulse generator is varied by the number of pulses from the pulse generator as shown in the table 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 inch
	6000 P / rev	0.2, 0.3, 0.4, 0.6, 0.8 inch

The "motion per rotation of pulse generator" in X-axis is the half of the above values.

Note: The multiplication of pulses can be set from servo unit and the NC.

2.75 INPUT/OUTPUT CONNECTORS

The connection of the machine and the NC control is made via "Half pitch connectors" directly to the CPU rack.

2.76 POWER INPUT

The standard input power is as follows:

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

2.77 AMBIENT CONDITIONS

(1) Ambient Temperature
For operation: 0°C to +45°C
For storage: -20°C to +65°C

(2) Relative Humidity: 10% to 90% RH

(3) Vibration: 4.9 m/s^2 max.

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

2.78 SPINDLE PULSE GENERATOR

This provides the spindle position detector connected to the lathe spindle with 1 : 1 ratio. The number of pulses are:

phase A, phase B	1024 P/rev
phase C (zero point pulse)	1 P/rev

a. 4000 rpm max. (provided with oil seal)
b. 6000 rpm max. (not provided with oil seal)

3 BASIC OPTIONS

3.1 AC SERVO CONTROL UNITS

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

(1) NC board built-in system:
Built in the free-standing type cabinets.

(2) External install system:
Supplied in unbundled type to the free-standing type cabinets. Cable connection to be less than 10 meters (30 feet).

Servo capacity is as follows:

No.	Continuous Max Torque of F Series AC Servomotor	Type CACR-
1	30 kg · cm	SR05BB
2	60 kg · cm	SR10BB
3	90 kg · cm	SR15BB
4	120 kg · cm	SR20BB
5	230 kg · cm	SR30BB
6	380 kg · cm	SR44BB

3.2 AC SERVOMOTORS

The following AC servo motors that incorporate the position-detecting pulse generator (PG) and speed-detecting pulse generator (PG) are available. The rated speed of AC servomotor is 1500 rpm.

No.	Continuous Max Torque of F Series AC Servomotor	Type USAFED-
1	30 kg•cm	05F[]*
2	60 kg•cm	09F[]
3	90 kg•cm	13F[]
4	120 kg•cm	20F[]
5	230 kg•cm	30F[]
6	380 kg•cm	44F[]

* [] indicates depending on detector type(P/rev) as follows:

A: 6000 P/rev
B: 5000 P/rev

4 OPTIONS

4.1 SEPARATE TYPE NC OPERATOR'S STATION

The separate stations are available in two configurations :

- Keyboard on right side of CRT
  (The power on/off push button not provided)

See Fig. 2.2.

- Keyboard below CRT

See Fig. 2.3.

4.2 SEPARATE TYPE TAPE READER

The separate type tape reader can be connected to the NC control through RS-232C interface port 1.

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

4.3 TAPE READER WITH REELS

Attached type 1 can be provided with the following tape reader with reels.

(1) 6-inch reel

Reel diameter: 150 mm (6 inches)
Tape length: 80 m (262 ft.)
Tape thickness: 0.108 mm

The tape reader speed determines tape read and rewind speed.

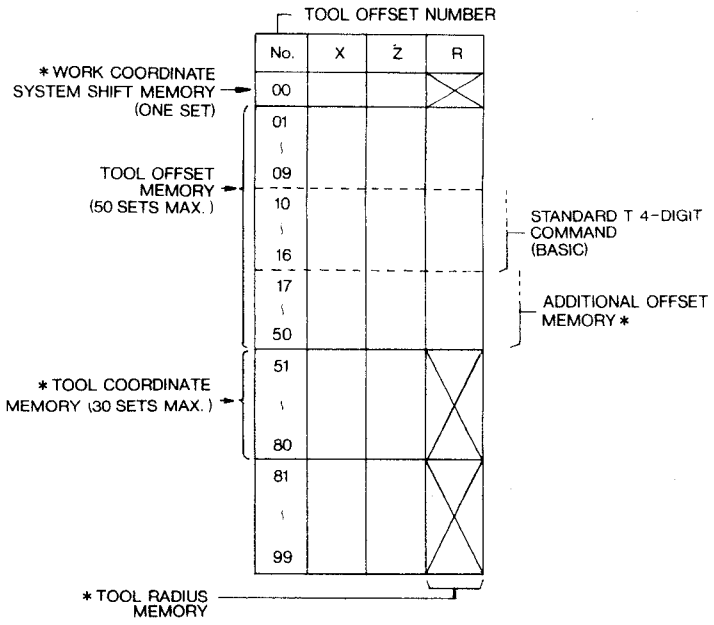
4.4 M3-DIGIT BCD OUTPUT

A digit is added to the basic M 2-digit BCD output to provide a total of M 3-digit BCD output. M codes being output are M00 to M89 and M110 to M999.

4.5 ADDITIONAL TOOL OFFSET MEMORY

To meet T 4-digit command, 50 sets of offset memories are available to replace 16 sets.

The map of offset memory including other options are shown as reference.



4.6 ADDITIONAL PART PROGRAM STORAGE

Any of the following storage capacities can be selected to replace the memory equivalent to the basic tape length of 40 meters.

	Total Storage Capacity
1	80 m (262 ft.)
2	160 m (5254 ft.)
3	320 m (1049 ft.)

4.7 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

4.7 ADDITIONAL PROGRAM NUMBER REGISTRATION (Cont'd)

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.)

4.8 OPERATION TIME DISPLAY B

The CRT displays the cumulative time while the external signal (EXTC) is being input.

	HOUR	MIN	SEC	
TM4:	9999.	59.	59	max.

4.9 STEP-MODE SIMULTANEOUS ONE-AXIS OPERATION

The step-mode of operation is enabled one-axis-at-a-time using the axis selection switch. The axis moves by the amount of turning of the manual pulse generator. The manual pulse generator is graduated in 100 scales per rotation. The motion per rotation can be specified by the manual step multiple selection switch as follows:

Multiplication Switch	x1	x10	x100 or greater
Metric Input	0.001 mm	0.01 mm	0.1 mm
Inch Input	0.0001 inch	0.001 inch	0.01 inch

The motion by the manual pulse generator is possible when "Automatic mode handle offset switch (HOFS input)" is on during the automatic operation mode. This motion, however, is impossible during execution of positioning command (G00, G06). This function is convenient for regnerative thread cutting, etc.

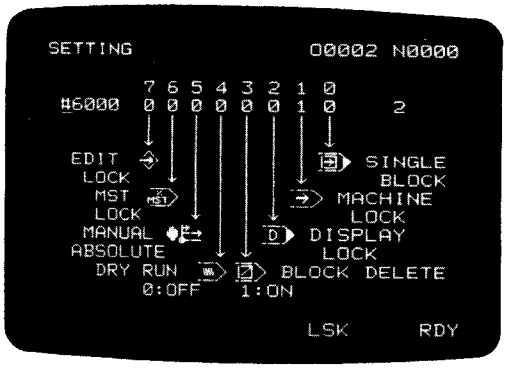
Notes:

1. The automatic operation mode means TAPE, MDI (manual data input), and MEM (memory) operation modes.
2. Installation of the manual pulse generator (one axis at a time) eliminates the function of manual step feed.

4.10 INTERNAL TOGGLE SWITCHES

Input to the following 8 switches (all basic) can be turned on and off by setting operation on the NC control panel (Setting #6000).

0: Single block
1: Machine lock
2: Display lock
3: Optional block skip (Block delete)
4: Dry run
5: Manual absolute
6: Miscellaneous function lock
7: Edit lock

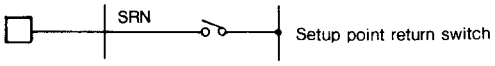


582-217

Example of Internal Toggle Switches CRT Display

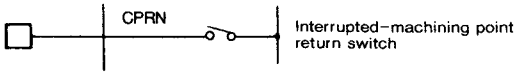
4.11 G50 POINT RETURN

This function manually returns the tool to the position specified by G50 coordinate system setting command. Turn on the Setup Point Return Switch (SRN input) and perform manual jog or manual rapid traverse operation, then the tool automatically stops at the setup position.



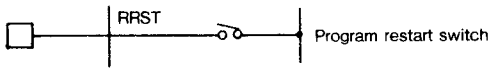
4.12 RESTART AFTER MANUAL INTERRUPTION

This function manually returns the tool to the position where the operation has been changed from the automatic mode to the manual to carry out a manual operation. Turn on the Interrupted-Point Return Switch (CRRN input), and manually feed the tool with jog or rapid traverse in the direction at which the mode has been changed from automatic to manual, then the tool automatically stops at the mode-changed position.



4.13 PROGRAM RESTART

This function resumes the program at any sequence number. Turn on the program resume switch (PRST input), search for the sequence number, and then turn off the switch. Then, use MDI to output required M, S, and T codes, and depress the Cycle Start button to resume the program.



4.14 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.

4.15 BUFFERING FUNCTION (M92, M93)

(1) 4-block Buffering (M93)

The "M93 ;" command specifies a 4-block read-ahead mode until "M92" command is issued. The data read ahead in the following 4 blocks is stored in the buffer for the succeeding operations. If the data read ahead in 4 blocks contains a program whose operation time is longer than the time needed to read and compute the data in the 4 blocks following the above 4, then stop time between blocks is eliminated. Thus, this is effective in avoiding bright streaks on works that might be caused by stop time between blocks.

(2) 1-block Buffering (M92)

The "M92 ;" command cansels the 4-block read-ahead mode and the operation returns to the normal 1-block read-ahead mode.

Note: During tool radius compensation in M93 mode, a maximum of 6-block read-ahead mode might occur, because 2 blocks are allowed as the block without move command.

4.16 INCH/METRIC SETTING

The following G codes specify the input increment either as metric input or inch input.

G Code	Input Increment
G20	Inch input
G21	Metric input

In general, these G codes are specified in a single block at the head of program.

Notes:

1. Inch/metric input can be selected by setting operation (basic).
2. G20/G21 command rewrites the above setting data. Thus, the setting data determines either G20 or G21 at the time of power on.

4.17 RADIUS PROGRAMMING FOR CIRCULAR INTERPOLATION (G22, G23)

G22(G23) X(U)··· Z(W)··· R··· F(E)··· ;

This command specifies the radius of an arc by address R instead of specifying the center of arc by I and K. Note that:

Where R > 0: an arc less than 180°,

R < 0: an arc greater than 180°

Note: The radius-designated arc can be specified by G02 and G03 instead of G22 and G23 respectively.

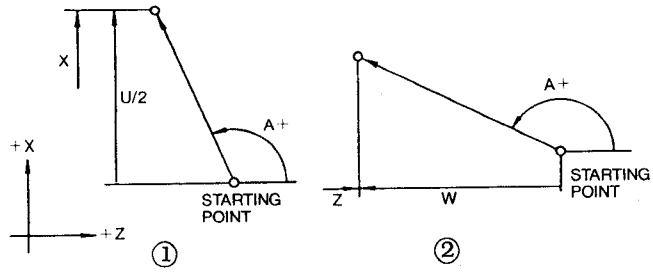
4.18 ANGLE PROGRAMMING FOR LINEAR INTERPOLATION (G01)

G01 X(U)··· A··· F(E)··· ; ①

or

G01 Z(W)··· A··· F(E)··· ; ②

Either command specifies an angle-designated linear interpolation.



Plus (+) sign of angle A indicates a counterclockwise angle from +Z axis.

4.19 TOOL OFFSET VALUE SETTING (G10)

G10 P··· X(U)··· Z(W)··· R··· ;

This command sets or corrects a tool offset on the part program,

where P: Tool offset number

X: } Tool offset value in absolute setting
Z: }

U: } Tool offset value in incremental setting (= additive write)
W: }

* R: Tool radius value in absolute setting

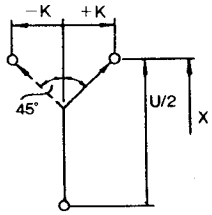
There is no change in compensation values for the omitted addresses. An offset value tape may be prepared using G10 format and stored in the tool offset memory all at once.

4.20 CORNERING (G11, G12)

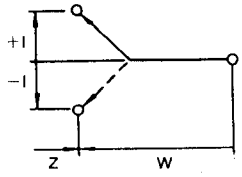
(1) Beveling (G11)

G11 X(U)··· K··· F(E)··· ; ① or
G11 Z(W)··· I··· F(E)··· ; ②

This command performs beveling of K or I at the end of blocks. The operation is limited to a single axis command for X or Z axis.



①



②

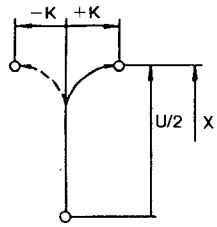
Note: A chamfer can be commanded by G01 code instead of G11 as shown below:

G01 X(U)··· K··· F(E)··· ; or
G01 Z(W)··· I··· F(E)··· ;

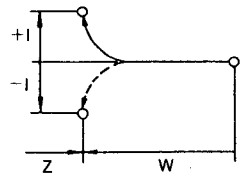
(2) Rounding (G12)

G12 X(U)··· K··· F(E)··· ; 1 or
G12 Z(W)··· I··· F(E)··· ; 2

The command performs a rounding with radius K or I at the end of blocks. The operation is limited to a single axis command for X or Z axis.



①



②

Note: Rounding can be specified with the following G01-code format instead of G12.

G01 X(U)··· R··· F(E)··· ; or
G01 Z(W)··· R··· F(E)··· ;

Address R replaces K and I.

4.21 MULTIPLE CORNERING (BEVELING/ROUNDING) (G111, G112)

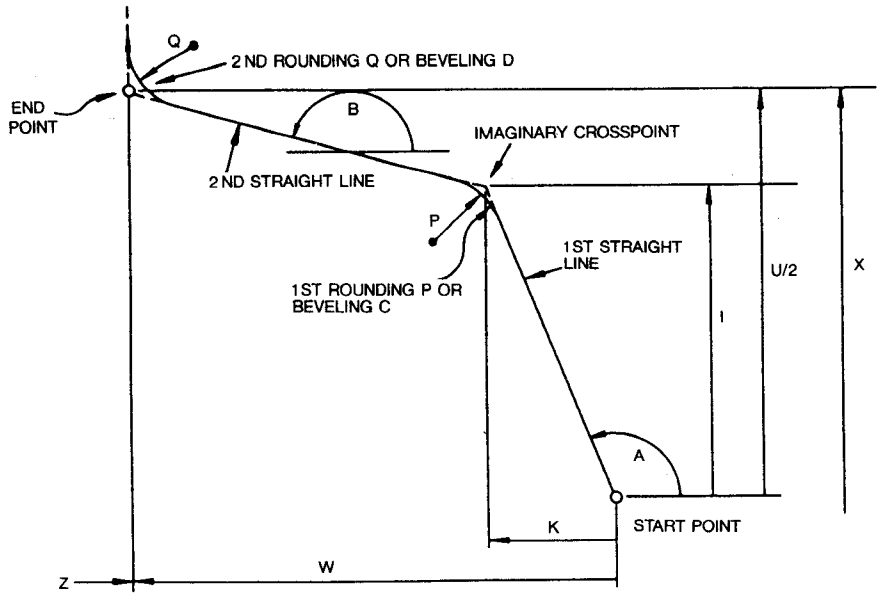
Programming a beveling/rounding operation on a work having a given taper or arc portion requires a complicated computation. The combined beveling/rounding codes provide an autoprogramming function to carry out such a complicated computation automatically within the CNC.

The following two functions are available:

G Code	Group	Function
G111	X*	Taper combined beveling/rounding
G112		Arc combined beveling/rounding

(1) Taper Combined Beveling/Rounding (G111)

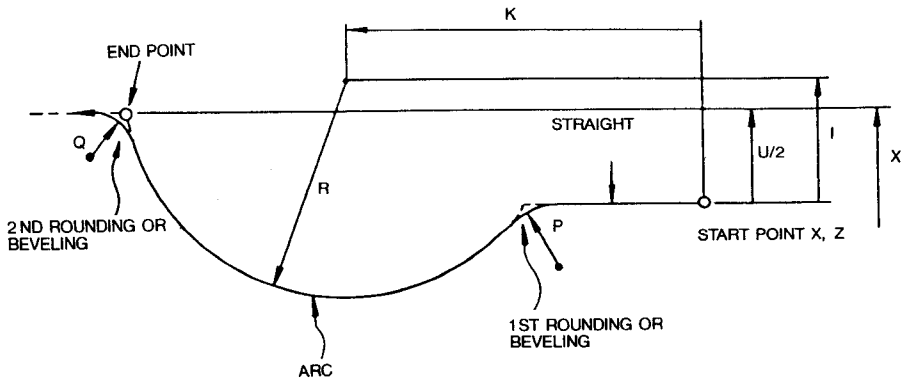
The following taper contour can be specified:



$$G111 \left\{ \begin{array}{l} X(U)\cdots\ Z(W)\cdots\ I\cdots\ K\cdots \\ X(U)\cdots\ A\cdots\ I\cdots\ B\cdots \\ A\cdots\ Z(W)\cdots\ B\cdots\ K\cdots \end{array} \right\} P(C)\cdots Q(D)\cdots\ ;$$

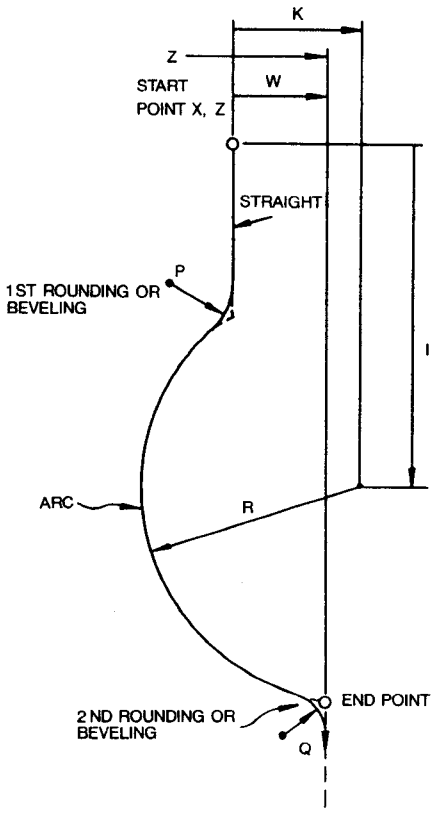
(2) Arc Combined Beveling/Rounding (G112)

The following contour of an arc and straight line can be specified.



i) G112 X(U)··· I··· K··· R··· P(C)··· Q(D)··· ;

4.21 MULTIPLE CORNERING (BEVELING/ROUNDING) (G111, G112) (Cont'd)



ii) G112 Z(W)··· I··· K··· R··· P(C)··· Q(D)··· ;

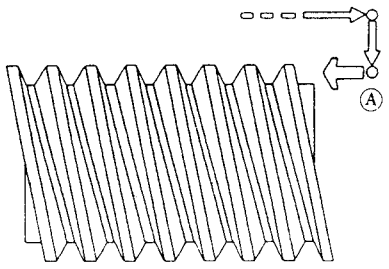
4.22 MULTI-START THREADCUTTING (G32)

This function allows multiple threadcutting to provide multiple grooves in a lead. The function permits grooves to be cut without shifting the start point of thread grooves.

G32 X(U)··· Z(W)··· B··· F(E)··· ;

The above command starts cutting at the spindle position shifted by angle B from the position where the spindle start point synchronous pulses are generated. B is a shift angle of the spindle for multiple thread and specified in the range of 0 to 360°.

Note: Continuous threading cannot include multiple thread.



Two-Start Threads

4.23 VARIABLE LEAD THREADCUTTING (G34)

G34 X(U)··· Z(W)··· K··· F(E)··· ;

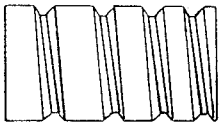
This command allows a variable lead thread to be cut where a lead increment per rotation of thread is specified by address K. The range of K is as follows:

	Range of K
Metric Input	±0.0001 – ±100.0000 mm
Inch Input	±0.000001 – ±1.000000 inch

Other specifications are the same as G32.

Notes:

1. G34 cannot be used for continuous threading and multiple threading.
2. G34 cannot use the function of "temporary interrupt during threading."



Increasing-Lead Threads

4.24 2ND REFERENCE POINT RETURN (G30)

G30 X(U)··· Z(W)··· ;

This command moves the tool to the specified intermediate position, then to the second reference point. Parameters are used to determine the position of the 2nd reference point by its distance from the 1st reference point.

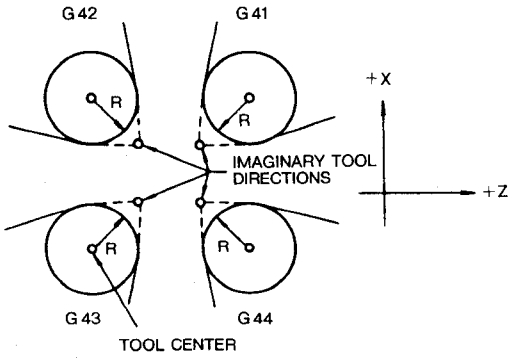
Note: If G29 is specified immediately after G30 command, then the tool motion is the same as G28: return to the 1st reference point.

4.25 TOOL NOSE RADIUS COMPENSATION (G40 TO G44)

This function compensates for cutting errors caused by the roundness of the tool.

(1) G Codes in Compensation Mode (G41 to G44)

Four directions of G41 to G44 are used to specify the imaginary tool directions with respect to the tool centers and to specify to enter the compensation mode.



G Code to Cancel Compensation (G40)

The command "G40 ;" cancels the tool radius compensation.

(3) Specifying compensation direction.

T ± □□△△;

+: The compensation of tool center is to the right of advancement.

-: The compensation of tool center is to the left of advancement.

(4) Offset Memory for Setting Tool Radius

The number of memories to store tool radius depends on the usable tool offset number.

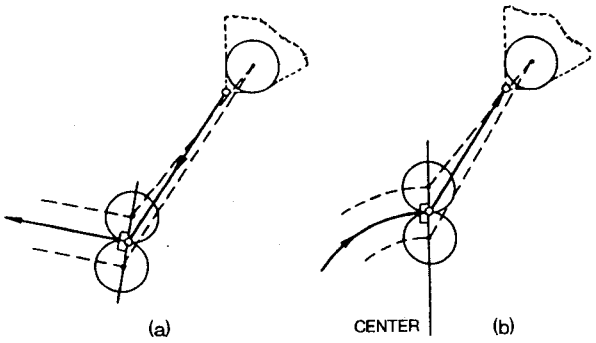
Where T 3-digit specified: 9 (basic)

Where T 4-digit specified: 16 or 50

(5) Motion at Start of Compensation and at Cancel

a. At the start of compensation, the tool center comes on the normal line of the start point of the block immediately following G41, G42, G43, or G44 command.

b. At the cancel of compensation, the tool center comes on the normal line of the end point of the block immediately before G40 command.



(6) Motion during Compensation Mode

a. For inner corner (less than 180° tangent angle)

--- Cross point is computed, and the tool passes through the cross point.

b. For outer corner (more than 180° tangent angle)

M Code	Functions
M96	Tool nose radius compensation-arc round mode
M97	Tool nose radius compensation-crosspoint computation mode.

(7) The compensation is normally done even in continuance of two blocks not having move command, during compensation mode. In the case of three blocks or more, a temporary cancel condition occurs. A dummy block can be used.

4.26 MULTIPLE REPETITIVE CYCLE (G70 TO G76)

By specifying a finish contour with this command, the tool path is automatically computed for rough cutting and rough finishing, and the cutting is carried out. Because of the capability to handle a finish contour having depression, the programming time is drastically reduced.

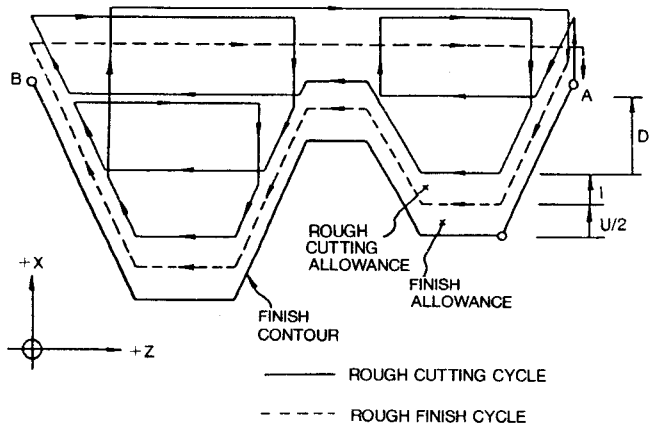
(1) Stock Removal in Turning (G71)

The specified format is shown in the list. By specifying I, K, and tool radius compensation, the tool radius compensation is carried out during the rough finish along the broken line. The R1 command can specify a finish contour having depression with a maximum of three interrupted points. A maximum of 45-block programming can be used to specify a finish contour. The depth of cut D may have cut-override in the range of 10 to 200% in steps of 10%.

4.26 MULTIPLE REPETITIVE CYCLE (G70 TO G76) (Cont'd)

(2) Stock Removal in Facing (G72)

The same as G71 except that the operation is parallel to X-axis. Example of cutting path when R1 is specified in G71.



(3) Pattern Repeating (G73)

This cycle carries out cutting by dividing the operation in D-times and ends it, leaving an allowance for finishing. Tool radius compensation, if specified, is carried out throughout the cycle. Finish contour can be programmed up to 45 blocks.

(4) Finishing Cycle (G70)

After the end of rough cutting specified by G71, G72, or G73, finish cutting is carried out according to the finish contour specified by G70 command. Tool radius compensation, if specified, is done throughout the cycle. The memory search function for finish contour program allows such a sequence as: rough cutting cycle (A) → rough cutting cycle (B) → finish cycle (A) → finish cycle (B).
For this purpose, the internal memory to store the finish contour program is installed separately from the part program memory of 45 blocks.

(5) Peck Drilling in Z-axis (G74)

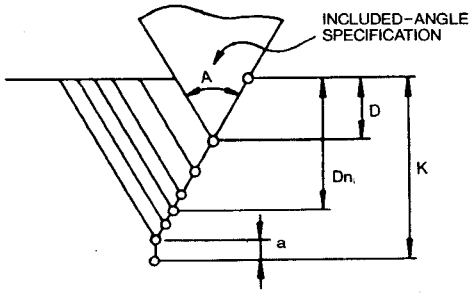
The tool nose radius compensation is not effective. If R1 is specified, the allowance per cutting is made available only up to the start point of cutting.

(6) Grooving in X-axis (G75)

The same as G74 except that the operation is parallel to X-axis.

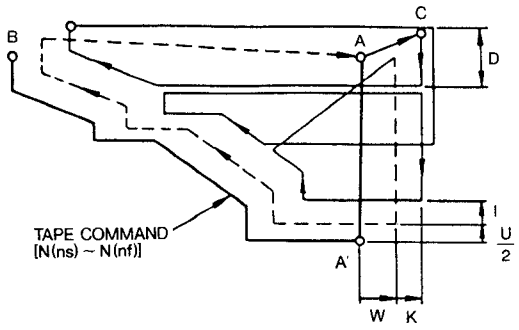
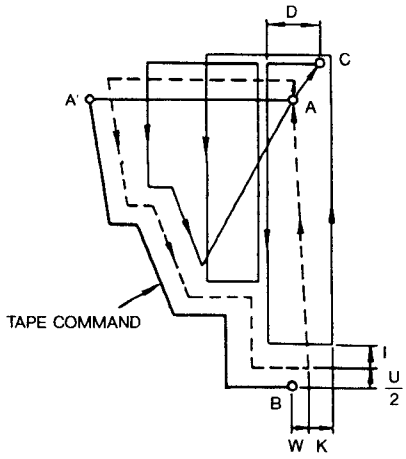
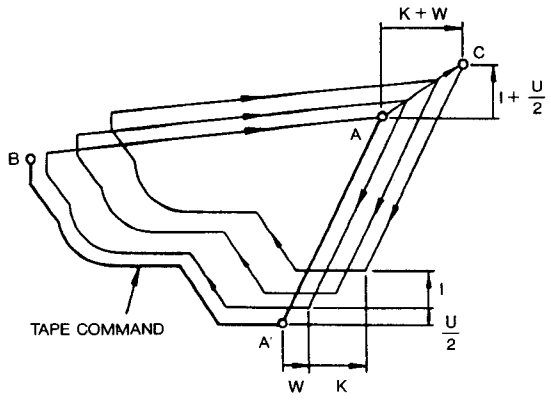
(7) Automatic Threading Cycle (G76)

The cutting near the point B is as follows:

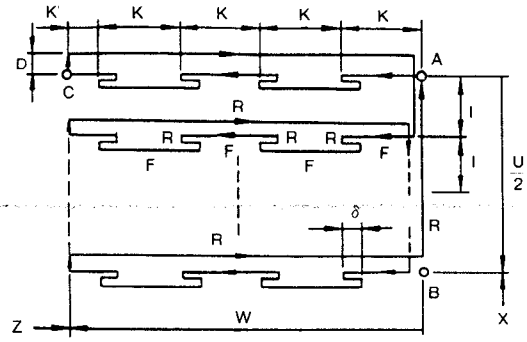
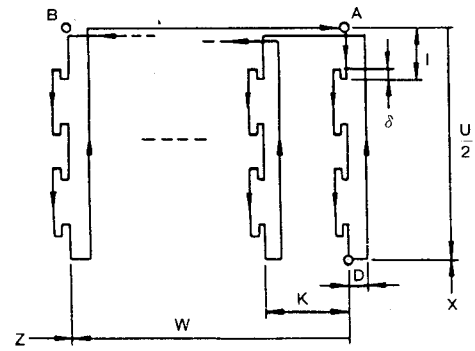
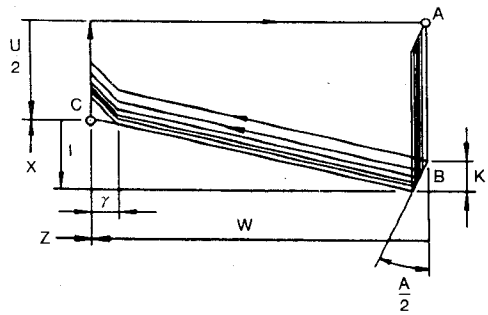


A: 0, 29°, 30°, 55°, 60°, 80°
$D_n = \sqrt{n}\ D$

The tool nose radius compensation is not effective. When the threadcutting input is on, threadcutting is carried out.

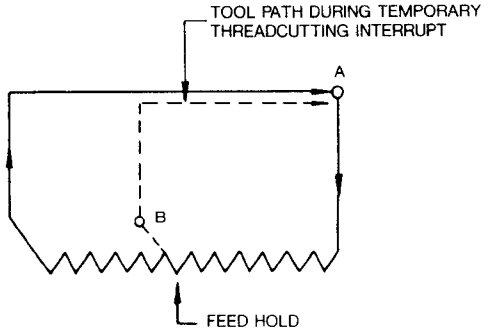
Code	Cutting Cycle	Command
G 71 Stock Removal in Turning		G 71 } (ns) (nf) G 72 } P··· Q··· U··· W··· G 73 } I··· K··· D··· F(E)··· S··· (R1) ; (ns) N····· ····· ; F ····· ; S ····· ; M ····· ; (nf) N ····· ····· ;
G 72 Stock Removal in Facing		Finish contour program A → A' → B P, Q··· Sequence No. to start and end cycle U, W··· Finish allowance in X and Z directions (U: Specified diameter) I, K ··· For G 71, G 72: Rough finish allowance in X and Z directions For G 73: Cutting allowance in X and Z directions D ··· For G 71, G 72: Depth of cut in rough cutting For G 73: Number of cutting operations I ≦ D ≦ 127 (Specify unsigned D) (Specify U, W, I, and K with signs)
G 73 Pattern Repeating		
G 70	N(ns) to N(nf) finish cutting carried out	G 70 P··· Q··· ;

4.26 MULTIPLE REPETITIVE CYCLE (G70 TO G76) (Cont'd)

Code	Cutting Cycle	Command
G74 Peck Drilling in Z-axis	 δ: SETTING	G74 / G75 } X(U)··· Z(W)··· I··· K··· D··· F(E)··· (R1) ; For G74: X(U) ··· X-component of point B Z(W) ··· Z-component of point C I ··· Motion in X direction K ··· Cutting value in Z direction D ··· Clearance at cut bottom For G75, exchange the motion and cutting value for X and Z directions. (Specify I, K, and D without sign; set constant values with parameters.)
G75 Grooving in X-axis	 δ: SETTING	
G76 Automatic Thread Cutting	 	G76 X(U)··· Z(W)··· I··· K··· D··· F(E)··· A··· ; X(U) ··· X-component of point C Z(W) ··· Z-component of point C I ··· Taper distance on X-axis K ··· Thread height D ··· Depth of cut at first time A ··· Included angle (deg) (Specify K and D without sign.) $\frac{1}{6}K \leqq D \leqq K$

4.27 THREADCUTTING INTERRUPTION

When the feed hold button is depressed during G92 or G76 cycle, the tool immediately carries out threadcutting and returns to the start point (A) as shown below. It is also possible, by parameter setting, to stop the tool at the position (B) where the threadcutting is finished. The tool returns to the start point by depressing the cycle start.



4.28 CONSTANT SURFACE SPEED CONTROL (G96, G97)

For this function, install the optional "S 4-digit command."

(1) Constant Peripheral Speed Control (G96)

G96 S··· (M03) ;

This command continuously computes the spindle revolution to have it reach the peripheral speed specified by S, and outputs the corresponding analog voltage or 12-bit binary signal. The computation at this time assumes that the current value of X axis indicates the diameter of the work. The S is specified in the following units:

Metric Input	m/min
Inch Input	feet/min

This control can be selected to meet any of four spindle gear steps.

(2) Constant Peripheral Speed Control Cancel (G97)

G97 S··· (M03) ;

This command cancels the constant peripheral speed control and specifies the spindle rpm with S 4-digit.

4.29 SPINDLE-SPEED OVERRIDE

With installed S 4-digit command (option), the spindle rpm override can be applied by an external input in the range of 50 to 120% and in 10% increments.

4.30 SPECIAL G-CODE II

The optional special G code II can replace the standard G code, by setting parameter. This option cannot permit selection of special G code I. For the detail of special G code II, refer to Table 1.6 List of G codes in Appendix 1. Only the G codes change; the function does not change.

4.31 OPTIONAL BLOCK SKIP B (/2 TO /9)

Install 8 switches on the machine corresponding to commands /2 to /9. If the switch corresponding to /n is ON, then the block specified by /n is skipped. In more detail, the data from /n to ; in that block is skipped. When the command is N50 G00 X10./5 M00 ; and No. 5 switch is ON, "M00" is skipped.

4.32 X-AXIS MIRROR IMAGE

This function is to invert the program-specified X-axis move direction. Either or both of the following functions can be installed.

(1) X-axis Mirror Image by Input Signal

When the X-axis mirror image switch (MIX input) is turned on, the sign of the specified value on the X-axis is inverted and the corresponding operation is carried out.

(2) Mirror Image by Programming

In general, this function is used with a G50 coordinate system setting command to make programming under the assumption that one of the facing tool boxes is in the same quadrant as the other.

G68; (Mirror Image by Programming ON)
This command inverts the sign of the X-axis specified-values of the following programs and carries out the operation.

G69; (Mirror Image by Programming OFF)
This command cancels the facing tool box mirror image.

4.33 AUTOMATIC COORDINATE SYSTEM SETTING

At the time of completing manual return to the reference point, it is possible to automatically set the coordinate system. The values of coordinate system to be set are set as parameters in advance.

4.34 WORK COORDINATE MULTI-SHIFT (G50T)

This function is used with the optional "work measurement value direct input." The combined function is effective in reducing the setting time of tool offsets, and programming of tool exchange in any positions.

(1) Work Coordinate System Setting (G50 T)

G50 T□□△△;

- Tool offset number (0 - 50 max)
- Tool coordinate memory number (51 - 80 max)

With this command, the coordinate system such as shown below can be set in both X and Z axes. The given coordinate system is called the work coordinate system.

[Work coordinate system set value] =

[Display current value] + [Contents of specified tool coordinate memory] + [Contents of specified tool offset number]

Note: Value of the current value display universal.

(2) Tool Coordinate Memory

The tool coordinate memory can be used up to the following number of sets in accordance with the number of sets of usable tool offset memories.

	Tool Offset Memory	Usable Tool Coordinate Memory
1	0-9	51-59
2	0-16	51-66
3	0-50	51-80

By operating the work measurement value direct input, the tool coordinate memory stores the positional data (X, Z) of the tool.

(3) Returning of Displayed Current Value to Origin

G51 ;

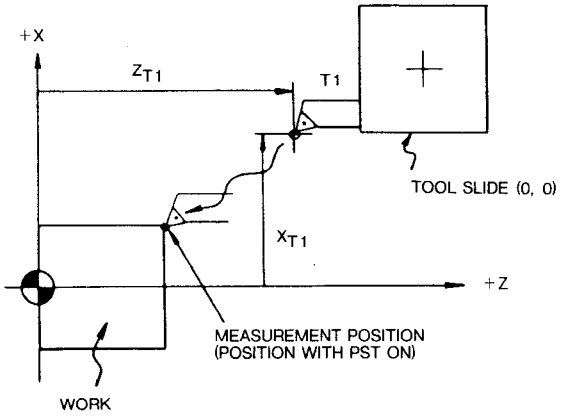
This command cancels the work coordinate system and establishes the coordinate system where the currently displayed value is (0, 0).

4.35 MDI OF MEASURED WORK INPUT

In the manual operation mode, let the tool edge get in contact with the outer diameter (or end face) of the work, and depress the current value memory pushbutton (PST input), then the current value of the tool is temporarily stored in the register. After removing the tool, measure the correct value of work's outer diameter (or the end face coordinate value), and write the value into any of the tool coordinate memory numbers 51 to 80, using the same procedure as offset writing. When the write [WR] is depressed, the following computation result is stored as the data of the selected tool coordinate memory.

[Data of tool coordinate memory] = [Written measurement value] − [Current value temporarily stored in the register]

Thus, the automatically stored value is the distance from the origin (⊕) of the desired work coordinate to the position of the tool edge when the tool slide has positioned where the current value indicates (0, 0).



The above value corresponds to X_{T1}, Z_{T1} in the above example.

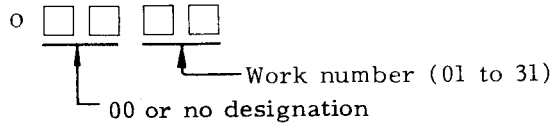
4.36 WORK COORDINATE SYSTEM SHIFT

If the coordinate system at programming does not coincide with the coordinate system set by G50 or automatic coordinate system setting, then the dislocation is corrected by the shift function. Using MDI operation, write the amount to be shifted into the tool compensation number "0" or "00." Then, when the coordinate system is set later*, it is shifted by this amount.

* G50, work coordinate system setting, and automatic coordinate system setting are all effective.

4.37 EXTERNAL WORK NUMBER SEARCH A

By specifying work number (01 to 31) with the "External work number selection switch (5-point input), the corresponding program number can be searched for. The correspondence between the work number and program number is as follows:



4.38 EXTERNAL DATA INPUT

External data can be received for the functions shown below. For this purpose, the required interfaces are:

- For data input: 16 points
- For function input: 8 points
- For answer signal output: 2 points

(1) External Number Search C

Program numbers 1 to 9999 can be externally searched for.

(2) External Tool Compensation C

Tool offset value of 0 to ±7.999 mm (or 0 to ±0.7999 inch) may be incrementally added to or replace the content of the currently specified offset memory.

(3) External Work Coordinate System Shifting

With installed work coordinate shift (option), the value to be shifted in the range of 0 to ±7.999 mm (or 0 to ±0.7999 inch) is received at a time and added to tool offset number "0" or "00."

4.39 SPINDLE INDEXING

This function requires the optional S 4-digit command to be installed. Externally specify the spindle index position by inputting 12-bit binary bits ("4096" = 360°), then input a spindle index signal to start indexing. When it is stopped within the allowance around the specified position, the indexing is complete. The allowance, indexing speed (rapid and approach), and decelerating position are specified by parameters.

4.40 STORED STROKE LIMIT (G36 TO G39)

This function is to ensure safety by preventing the tool from entering the specified prohibited area in both manual and automatic operations.

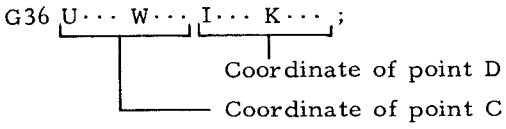
(1) 1st Prohibited Area (Outside Prohibited)

The coordinates of points A and B are set by parameters, not by G code.

(2) 2nd Prohibited Area (Inside Prohibited)

The coordinates of points C and D can be determined at setting.

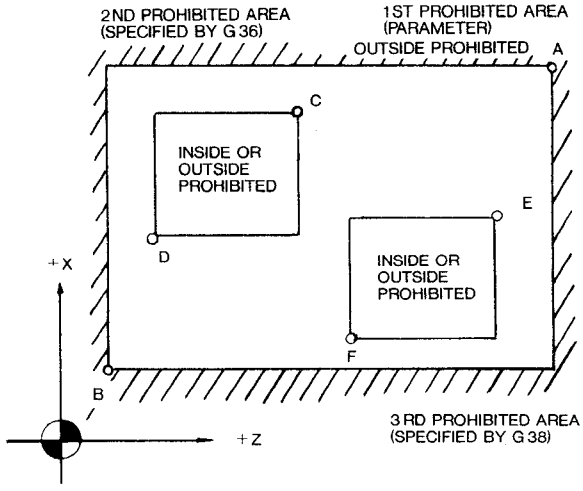
In addition to setting, the command of



enables to set the area and to turn on the area check function. This 2nd area can be specified as either inside or outside prohibited by parameters.

G37 ;

This single block command turns off the area check function.



4.40 STORED STROKE LIMIT (G36 TO G39) (Cont'd)

(3) 3rd Prohibited Area

The coordinates of points E and F can be specified at setting. The 3rd area can be specified as either inside or outside prohibited by parameters.

In addition to setting, the command of

G38 U··· W··· I··· K··· ;

Coordinate of point F

Coordinate of point E

enables to set the area and to turn on the area check function.

G39 ;

This single block command turns off the area check function.

(4) Coordinate and Unit of Setting

Points A through F are specified by the absolute value in the machine coordinate system, or the distance from the reference point. Accordingly, this function is effective only after the return to the reference point is done manually or automatically.

(5) Remaining Distance Display

The distance from the current tool position to the prohibited area can be displayed on the CRT.

4.41 STORED STROKE LIMIT FOR EACH TOOL

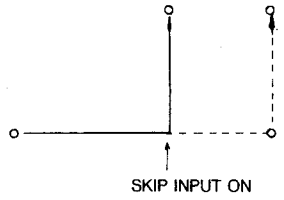
The 3rd area check function corresponding to tool can be made effective by externally specifying tool numbers. Use the special input to specify tool number (4 bits) to specify any of the tools (01 to 15) and turn on "input to select the area (TPS)." Then, the prohibited area corresponding to the selected tool is set as the 3rd prohibited area of stored stroke limit. Initially the prohibited area corresponding to each tool of 01 to 15 is set by parameters.

4.42 SKIP FUNCTION (G31)

(1) G31 X(U)··· Z(W)··· F(E)··· ;

This command performs a special linear interpolation. While the tool is moving along this interpolation, if "skip signal input" is turned on, then the remaining motion is skipped and the tool moves according to the next block command. If the next block command is incremental, the tool moves in incremental mode.

Example: G31 W100. ;
G00 U50. ;



(2) The delay time from a skip signal ON to the start of processing is less than 0.5 milli seconds

(3) In addition to F(E) command, the feedrates can be initially set by parameters. Which is to be used, command or parameters, is specified by other parameters.

(4) The coordinate value where the skip signal is ON is stored as parameter data. The stored value can be used as variables in the user macro.

4.43 TOOL LIFE CONTROL (G122, G123)

With this function, a tool, after the use of specified time or number of cycles for processing, is automatically replaced with the initially specified tool. Instead of the ordinary tool number, the function uses the tool specifications called "Tool group number." Installing this function requires the optional "T 4-digit command" and "Additional offset memory."

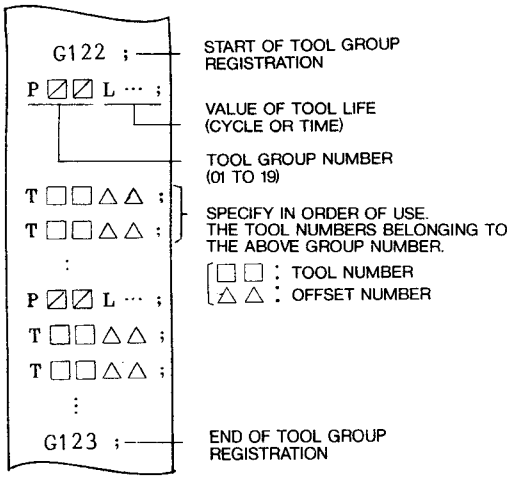
(1) Tool Group Number and Tool Life

Tool Group Number	Tool Life Setting Range
01 to 09	Processing cycle control: 0 to 9999 cycles
10 to 19	Processing time control: 0 to 9999 minutes

"Tool life" is the processing cycles or time served by a tool. When the tool life is reached, successively selected tool group of the same type is represented by "Tool group number." Which is used to control tools, the cycle or time is specified by the tool group number.

(2) Tool Group Registration

The following G codes and formats are used to register the relationship between the tool group number and tool number, from punched cards to the NC control.



Note:
The above data is registered as the setting data of the control. Thus, the registration is possible by setting operation.

(3) Tool Registration into Offset Memory

The tool numbers to be used are written as setting data into the tool offset memories 01 through 50.

T ☐☐ △△

Offset Number	Tool Number	Procedence (n)
01	01	1
02	01	2
03	01	3
04	01	4
05	01	5
06	02	1
07	02	2
.	.	.
.	.	.
.	.	.

One tool number can use a maximum of 5 offset numbers.

(4) Tool Life Control Command

On the part program, the tool life control is specified by the following T 4-digit command.

a. T ☒☒ 9 ☐ ;

- ☐: Offset number with precedence n (n = 1, 2, 3, 4, and 5)
- ☒☒: Tool group number specification (01 to 19)

This command controls the life of tools having the specified tool group number while making positional offset according to the offset number of precedence n.

b. T ☒☒ 90 ;

This command cancels the currently used offset of ☒☒ group.

c. T ☒☒ 99 ; (☒☒ ; 01 to 09)

This command is used only in the controlling of processed number. When this command is read, the NC control adds 1 to the processing number of ☒☒ group.

(5) Interface

This function requires the following interfaces:

- For input of group number specification and others: 7 points
- Tool exchange request output: 1 point

Note:

If the tool number is 30 or less, this function is used with the work coordinate system.

G50T ☒☒ 90 ;

- ☒☒: Group number specification

4.44 MACRO PROGRAMS (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 macro programs.

(1) Macro Program Simple Call (G65)

The macro program 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 Program 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 macro program can call another macro program, and this process can be nested to 4 levels of macros.

4.44 MACRO PROGRAMS (G65, G66, G67) (Cont'd)

(4) Macro Program

The macro program, written in the format of subprogram (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 variables 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>] G0 T0 n ;

The above conditional branch control statement can be used.

(ii) WHILE [<conditional expression>] D0 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.

4.45 STORED LEADSCREW ERROR COMPENSATION

This function compensates for an lead error in the ball screw of the machine. The compensation data is initially set by parameters. This function is effective after the power is on and the tool is returned to the reference point manually or automatically.

The specifications are:

(1) Compensation Axes: X-axis, Z-axis

(2) Number of Compensation: Max. 256 points at Z axis

(3) Base Point of Compensation: Reference point

(4) Compensation Interval: 6000 pulses or more

(5) Data Setting Method: Absolute or incremental (selected by parameters)

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

(a) Standard general-purpose I/O modules
Type JANCD-FC810 (Max. 3)

- Input — 112 points/module
- Output — 96 points/module

(b) Mini general-purpose I/O modules
Type JANCD-SP50

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

* Inside of the 9"CRT panel

(3) 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")

APPENDIX 1 LIST OF DATA

Table 1.1 Address Characters

B : Basic
O : Optional

Address	Meaning	Section
A	Angle designation for G01 and G111, included angle for G76	O
B	Spindle shift angle of multiple thread, Angle designation for G111	O
C	Macro program character, Rapid threading pull-out value tor G111, G112	O
D	Depth of cut and number of cutting cycles for G71 to G76	O
E	Specifications for precise feed and precise lead for cutting	B
F	Specifications for normal feed and normal lead for cutting	B
G	Preparatory function (G-function)	B
H	Macro program character	O
I	X-component of arc center, canned cycle parameter, rapid threading pull-out value (radius value), X-axis component of imaginery cross-point for G111	B, O
J	Macro program character	O
K	Z-component of arc center, canned cycle parameter, rapid threading pull-out value, Z-axis component of imaginery cross-point for G111	B, O
	Incremental value of variable lead thread	O
L	Number of subprogram repetitions	B, O
M	Miscellaneous function (M-function)	B
N	Sequence number	B
O	Program number	B
P	Dwell canned cycle starting sequence number, program number, macro program number	B, O
Q	Subprogram starting sequence number, canned cycle ending sequence number	B, O
R	Radius of arc, rounding value, tool radius value	B, O
S	Spindle function (S-function), maximum spindle revolution	B
T	Tool function (T-function), tool coordinate memory number	B, O
U	X-axis incremental command value, dwell, canned cycle parameter	B, O
V	Macro program character	O
W	Z-axis incremental command value, canned cycle parameter	B, O
X	X-axis coordinate value	B
Y	Macro program character	O
Z	Z-axis coordinate value	B

APPENDIX 1 LIST OF DATA (Cont'd)

Table 1. 2 Function Characters

EIA Code	ISO Code	Function	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, Macro program operator	
−	−	Minus sign, Macro program operator	
0 to 9	0 to 9	Numerals	
a to z	A to Z	Address characters	
/	/	Optional block skip, Macro program operator	
Del	DEL	Disregarded (Including All Mark)	
.	.	Decimal point	
Parameter starting	#	Sharp (Variable designation)	
*	*	Asterisk (Multiplication operator)	
=	=	Equal mark	
[	[	Left bracket	
]	]	Right bracket	
$	$	Macro program operator	
@	@	Macro program operator	
?	?	Macro program operator	

Note:

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 automatically recognized. Tape code for punching is selected by setting function.

Table 1. 3 Tape Format

	Address	Metric Output		Inch Output		B: Basic O: Option
		Metric Input	Inch Input	Metric Input	Inch Input	
1	Program number	O 4		O 4		B
2	Sequence number	N 4		N 4		B
3	G-function	G 3		G 3		B
4	Coordinate address a: X, Z, I, K, U, W, R	a + 53	a + 44	a + 53	a + 44	B
5	Feed per minute	F 50	F 32	F 50	F 42	B
6	Feed per revolution and thread lead	F 32	F 24	F 42	F 24	B
		E 34	E 26	E 44	E 26	B
7	S-function	S 4		S 4		B
8	T-function	T (2 + 2)		T (2 + 2)		B
9	M-function	M 3		M 3		B
10	Dwell	U (P) 43		U (P) 43		B
11	Program number designation	P 4		P 4		B
12	Sequence number designation	Q (P) 4		Q (P) 4		B, O
13	Number of repetitions	L 7		L 7		B
14	Angle designation for straight line	A (B) 33		A (B) 33		O
15	Angle designation for multiple thread	B 3		B 3		O

Note:

1. Inch/Metric output is set by setting parameter # 6007 D 3.
2. Inch/Metric input is set by setting parameter # 6001 D 0.
3. F codes for feed/min or feed/rev can be switched by G 98 and G 99.

APPENDIX 1 LIST OF DATA (Cont'd)

Table 1. 4 List of Program Commands

Address		Metric Output		Inch Output	
		Metric Input	Inch Input	Metric Input	Inch Input
Program number	O	1−9999		1−9999	
Sequence number	N	1−9999		1−9999	
G-function	G	0−199		0−199	
Coordinate address X, Z, I, K, U, W, R		± 99999.999 mm	± 3937.0078 in.	± 99999.999 mm	± 9999.9999 in.
Feed per minute	F	1−24000 mm/min	0.01−944.88 in./min	1−60960 mm/min	0.01−2400.00 in./min
Feed per revolution and thread lead	F	0.01−500.00 mm/rev	0.0001−19.6850 in./rev	0.01−1270.00 mm/rev	0.0001−50.0000 in./rev
	E	0.0001−500.0000 mm/rev	0.000004−19.685000 in./rev	0.0003−1270.0000 mm/rev	0.000010−50.000000 in./rev
S-function	S4	0−9999		0−9999	
T-function	T3	0 − 999		0 − 999	
	T4	0−9999		0−9999	
M-function		0−999		0−999	
Dwell U, P		0.001−99999.999 sec		0.001−99999.999 sec	
Program number direction P		1−9999		1−9999	
Sequence number direction P, Q		1−9999		1−9999	
Number of repetitions L		0−99999999		0−99999999	
Angle of straight line* A, B		0− ±360.000°		0− ±360.000°	
Angle of multiple thread B		0−360°		0−360°	

*For angle designation of included angle for G76, see (7) Automatic Threading Cycle (G76) in 3.29 Multiple Repetitive Cycle (G70 to G76).

Table 1. 5 Data Setting Range

Item		Metric Output (Screw)		Input Output (Screw)	
		Metric Input	Inch Input	Metric Input	Inch Input
Least input increment		0.001 or 0.01 mm	0.0001 or 0.001 inch	0.001 or 0.01 mm	0.0001 or 0.001 inch
Tool offset		0– ±8388.607 mm	0– ±330.2601 inches	0 – ± 9999.999 mm	0– ±838.8607 inches
Tool radius		0– ±99.999 mm	0– ±9.9999 inches	0– ±99.999 mm	0– ±9.9999 inches
Minimum step/handle feed		0.001 mm	0.0001 inch	0.001 mm	0.0001 inch
Stored stroke limit area designation unit	Program designation	0.001 mm	0.0001 inch	0.001 mm	0.0001 inch
	Parameter & setting	0.001 mm		0.0001 inch	
Rapid traverse rate	Upper limit value	24 m/min		2400 inches/min	
Manual jog					
F_0					
2 nd reference point coordinate value		0– ±99999.999 mm		0–9999.9999 inches	
Backlash compensation value		–8192 to 8192 pulses Note (1)		–8192 to 8192 pulses	

Note:
1. 1-pulse = least output increment
2. X-axis designated with diameter (except for pulse display)

APPENDIX 1 LIST OF DATA (Cont'd)

Table 1. 6 List of G Codes

B : Basic
O : Optional

G Code	Special G Code I	Special G Code II	Group	Function		Section
G00	G00	G00	01	Positioning (rapid traverse feed)		B
G01	G01	G01		Linear interpolation, angle programming for linear interpolation		B, O
G02	G02	G02		Circular interpolation CW, (radius R designation)		B, O
G03	G03	G03		Circular interpolation CCW, (radius R designation)		B, O
G04	G04	G04	*	Dwell		B
G06	G06	G06		ERROR DETECT OFF positioning		B
G10	G10	G10		Tool offset value setup		O
G11	G11	G11	01	Beveling	Cornering	O
G12	G12	G12		Rounding		O
G20	G20	G70	05	Inch input specification		O
G21	G21	G71		Metric input specification		O
G22	G22	G22	01	Radius programming for circular interpolation CW		O
G23	G23	G23		Radius programming for circular interpolation CCW		O
G27	G27	G27	*	Reference point return check		B
G28	G28	G28		Automatic return to reference point		B
G29	G29	G29		Return from reference point		B
G30	G30	G30		Return to 2nd reference point		O
G31	G31	G31		Skip function		O
G32	G33	G33	01	Threadcutting, continuous threadcutting, multi-start threadcutting		B, O
G34	G34	G34		Variable lead threadcutting		O
G36	G36	G36	07	Stored stroke limit 2nd area ON		O
G37	G37	G37		Stored stroke limit 2nd area OFF		O
G38	G38	G38	08	Stored stroke limit 3rd area ON		O
G39	G39	G39		Stored stroke limit 3rd area OFF		O
G40	G40	G40	06	Tool nose radius compensation cancel		O
G41	G41	G41		Tool nose radius compensation No. 1		O
G42	G42	G42		Tool nose radius compensation No. 2		O
G43	G43	G43		Tool nose radius compensation No. 3		O
G44	G44	G44		Tool nose radius compensation No. 4		O

Notes:

1. At power on and resetting, ◤-marked G code is automatically selected.
2. For G codes of group 04 and 03, G code at initial state when the control is energized with the power switch can be selected by parameter as listed on right side.

Group	G Code at Initial State	Parameter
04	G98 or G99	#6005 D_1
03	G90 or G91	#6005 D_0

Table 1. 6 List of G Codes (Cont'd)

B : Basic
O : Optional

G Code	Special G Code I	Special G Code II	Group	Function		Section
G 50	G 92	G 92	*	Coordinate system setup		B
				Maximum spindle revolution setup, work coordinate system setup		O
G 51	G 51	G 51	*	Return of current display value to origin		O
G 65	G 65	G 65		Macro program simple call		O
G 66	G 66	G 66	09	Macro program modal call		O
G 67	G 67	G 67		Macro program modal call cancel		O
G 68	G 68	G 68	10	Mirror image by programming ON		O
G 69	G 69	G 69		Mirror image by programming OFF		O
G 70	G 70	G 72	*	Finishing cycle	Multiple repetitive cycles	O
G 71	G 71	G 73		Stock removal in turning		O
G 72	G 72	G 74		Stock removal in facing		O
G 73	G 73	G 75		Pattern repeating		O
G 74	G 74	G 76		Peck drilling in Z-axis		O
G 75	G 75	G 77		Grooving in X-axis		O
G 76	G 76	G 78		Automatic threadcutting cycle		O
G 90	G 77	G 20	01	Turning cycle A		B
G 92	G 78	G 21		Threading cycle		B
G 94	G 79	G 24		Facing cycle B		B
G 96	G 96	G 96	02	Constant surface speed control		O
G 97	G 97	G 97		Constant surface speed control cancel		O
G 98	G 94	G 94	04	Feed per minute (mm/min)		B
G 99	G 95	G 95		Feed per revolution (mm/rev)		B
	G 90	G 90	03	Absolute programming		B
	G 91	G 91		Incremental programming		B
G 122	G 122	G 122	11	Tool registration start	Tool life control	O
G 123	G 123	G 123		Tool registration end		O
G 111	G 111	G 111	*	Taper multiple beveling/rounding		O
G 112	G 112	G 112		Arc multiple beveling/rounding		O

3. For G code of 01 group, G 00 or G 01 can be selected as initial state by setting parameter # 6005 D 6.

4. Radius programming for circular interpolation can be made by G 02, G 03 instead of G 22, G 23. See Para. 4.17.

5. G 01 code can be used instead of G 11 or G 12 for cornering. See Para. 4.20.

6. For G codes in groups 05, 07, and 08 group, their initial conditions at power on is determined by the corresponding data (# 6001 D_0, D_1, D_2).

APPENDIX 2 DIMENSIONS in mm (inch)

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

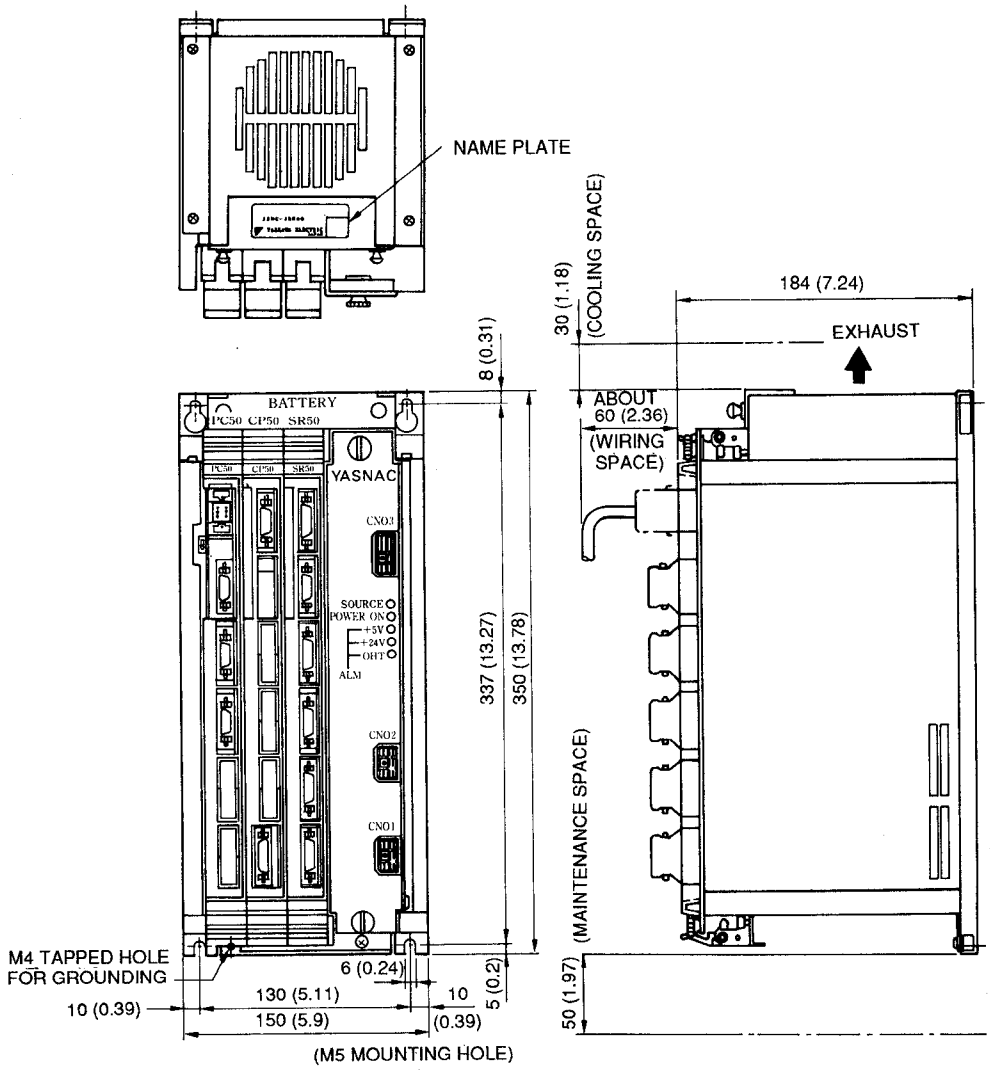


Fig. 2.1 Module Type CPU Rack

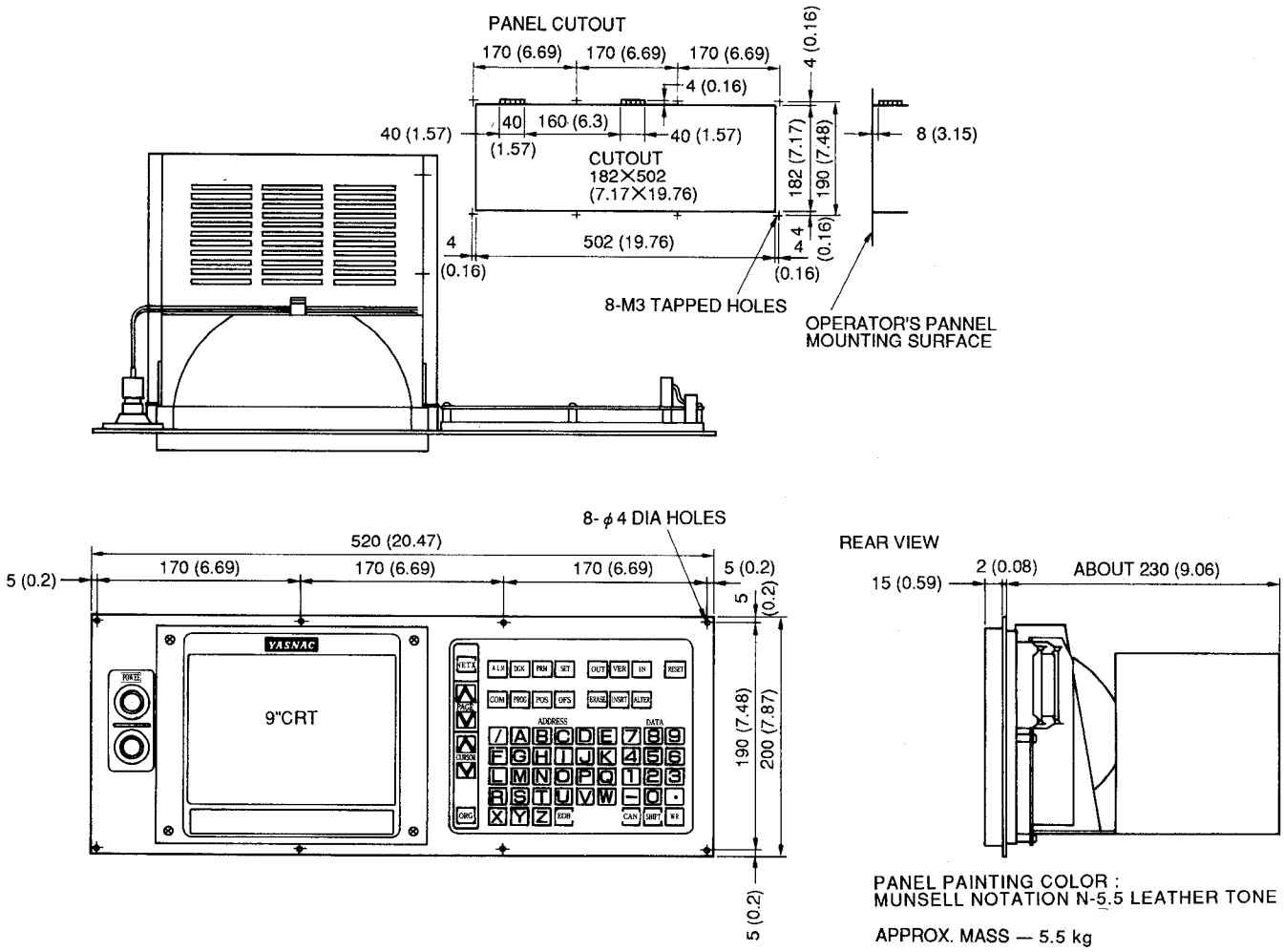


Fig. 2.2 NC Operator's Station with 9" Monochromatic CRT Display (Keyboard on right side of CRT) — with Power On/Off Pushbutton

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

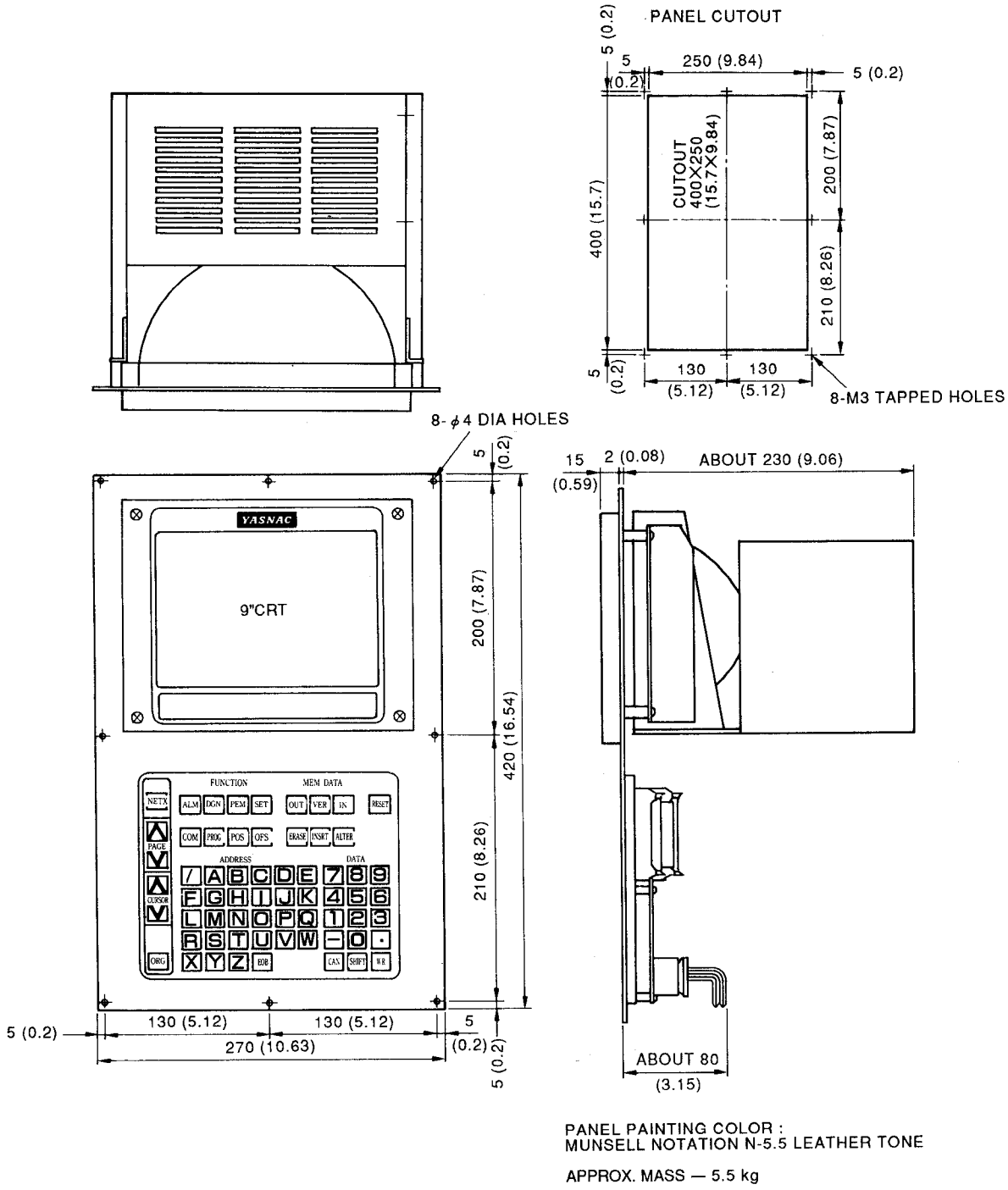


Fig. 2.3 NC Operator's Station with 9" Monochromatic CRT Display (Keyboard below CRT)

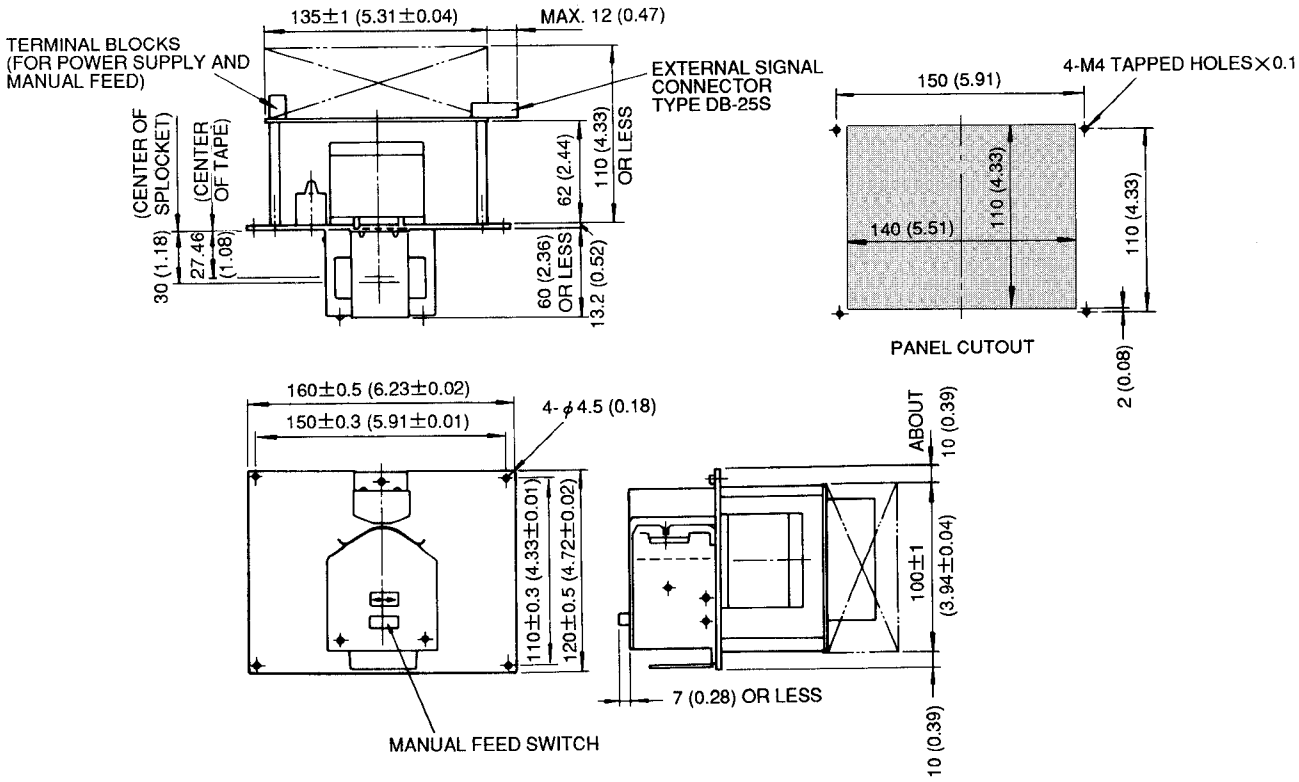


Fig. 2.4 Tape Reader Unit

YASNAC J50L
DESCRIPTIVE INFORMATION

TOKYO OFFICE New Pier Takesiba 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 1-513-847-6200 Fax 1-513-847-6277
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 46-486-10575 Fax 46-486-41410
Motoman Robotec GmbH
Kammerfeldstraße 1, 85391 Allershausen, Germany
Phone 49-8166-900 Fax 49-8166-9039
YASKAWA ELECTRIC UK LTD.
3 Drum Mains Park Orchardton Woods Cumbernauld, Scotland, G68 9LD U.K.
Phone 44-1236-735000 Fax 44-1236-458182
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 Room No. 8B Wan Zhong Building 1303 Yan An Road (West), Shanghai 200050, China
Phone 86-21-6212-1015 Fax 86-21-6212-1326
YASKAWA JASON (HK) COMPANY LIMITED
Rm.2916, Hong Kong Plaza, 186-191 Connaught Road West, Hong Kong
Phone 852-2858-3220 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 ELECTRIC CORPORATION

YASKAWA

SIE-C843-12.20

© Printed in Japan August 1996 95-2 0.3TA

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

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