SEIKI - SEICOS
∑10M/16M/18M
INSTRUCTION MANUAL

5 OPERATION

Edition 1.02 NO-0000-1-0221-E-1-02

Hitachi Seiki Deutschland
Werkzeugmaschinen GmbH

CONTENTS

1. OUTLINE

The SEIKI-SEICOS Σ 10M/16M/18M has realized a miniaturized high-reliability system by integrating up-to-date device technology. TheΣ10M/16M can perform high-speed, high-accuracy machining, using 64-bit RISC (Reduced Instruction Set Computer).
The operation system as a human interface is designed very user-friendly from a perspective of the user. For example, a canned cycle editing function is provided as an option in order to facilitate editing of canned cycles.
This manual describes how to operate the following models.
See “SEIKI-SEICOS Σ10M/16M/18M-PROGRAMMING” for programming, and “SEIKI-SEICOS Σ10/16/18/21” for alarms and maintenance.

Model	Designation
SEIKI-SEICOS Σ10M	S-Σ 10M
SEIKI-SEICOS Σ16M	S-Σ 16M
SEIKI-SEICOS Σ18M	S-Σ 18M

1-1 Precautions for Reading This Manual

(1) This manual and SEIKI-SEICOS SΣ10M/16M/18M-PROGRAMMING describe the entire functions of this NC unit, including the optional functions. The selected functions vary from one machine to another. As some of the functions described in the manual are not available, check the specifications of the machine beforehand.
(2) When there is any function not described “possible” in the manual, take it “impossible.”
(3) The information herein is subject to change without prior notice.

CAUTION

The programs, parameters, macro variables, and tool offset amounts have been stored in the internal memory of the NC unit. Generally, they are not lost by turning on/off the power. They may be erased by mistake or you are forced to erase the precious data saved in the memory in order to recover from a failure.
Make back-up copies of various data in advance so that you can quickly recover from such an unexpected incident.

CAUTION

Before starting machining, be sure to fully confirm proper operation of Machine by performing a trial run.
Before using work coordinate data and tool offset data, be sure to confirm that the data have been properly input.

2. SPECIFICATIONS

1. CONTROLLED AXES

1-1 Controlled Axes

The 3 axes, X, Y and Z, are controllable. Furthermore, up to 8 axes can be controlled by adding the optional 5 axex; 4th through 8th axes.

1-2 Simultaneous Controllable Axes

The 3 axes, X, Y and Z, can be controlled simultaneously regardless of rapid traverse or cutting feed. Furthermore, up to 8 axes can be controlled simultaneously (option). Table 1.2 shows the configuration of the controlled axes.

Table 1-2

	No. of Axes	Axis Name	Remarks
Standard controlled axes	3 axes	X, Y, Z	
Additional axes control	5 axes	Select of of U, V, W, A, B and C	
Simultaneous controllable axes	Standard 3 axes + additional axes	All the axes specified by the system (up to 8 axes)	

(Note 1) The number of controlled axes, and relations between the axis names and axes can be selected with prameters.

1-3 Increment System

There are two types of increment systems; IS-B and IS-C. You can select either of them by a parameter. (Is-A is not available for the moment.)
Millimeter/inch switching is set with a parameter. For detailed description of parameter, refer to “Parameters”.

1-4 Maximum Commandable Value

Table 1.4 shows the increment systems and commandable values.

1-5 10-Time Input

The input increment can be made 10 times larger by parameter setting. Table 1.5 shows the commandable values.

Table 1-4

Unit	Type	Linear Axis		Rotary Axis (deg.)
		(mm)	(inch)	
Input increment	IS-A IS-B IS-C	0.01 0.001 0.0001	0.001 0.0001 0.00001	0.01 0.001 0.0001
Least command increment	IS-A IS-B IS-C	0.01 0.001 0.0001	0.001 0.0001 0.00001	0.01 0.001 0.0001
Maximum stroke	IS-A IS-B IS-C	±999999.99 ±99999.999 ±9999.9999	±99999.999 ±9999.9999 ±999.99999	±999999.99 ±99999.999 ±9999.9999
Maximum commandable value	IS-A IS-B IS-C	±999999.99 ±99999.999 ±9999.9999	±99999.999 ±9999.9999 ±999.99999	±999999.99 ±99999.999 ±9999.9999

[10-Time Input Increment]

For the Types IS-B and IS-C, the input increment can be made 10 times larger by parameter setting.

Table 1-5

Unit	Type	Linear Axis		Rotary Axis (deg.)
		(mm)	(inch)	
Input increment	IS-B IS-C	0.01 0.001	0.001 0.0001	0.01 0.001
Least command increment	IS-B IS-C	0.01 0.0001	0.001 0.00001	0.01 0.0001
Maximum stroke	IS-B IS-C	±99999.999 ±9999.9999	±9999.9999 ±999.99999	±99999.999 ±9999.9999
Maximum commandable value	IS-B IS-C	±999999.99 ±99999.999	±99999.999 ±9999.9999	±999999.99 ±99999.999

(Note 1) For the Type IS-A, nothing is changed even if you set the parameter for 10-time input increment.

1-6 Position Detector

The pulse encoder is provided as a standard position detector. Optionally, the pulse scale or the Inductosyn detecting function can be selected. When the Inductosyn detecting function is selected, however, you need a converter which serves as an interface equivalent to the pulse scaler.

When adding an option for detecting an absolute position, an absolute encoder will be attached.

2. INTERPOLATING FUNCTIONS

2-1 Positioning (G00)

Each axis can be fed at a rapid traverse rate independently by specifying G00.

2-2 Linear Interpolation (G01)

Linear interpolation is performed at the feed rate specified by an F-code in a G01 command.

2-3 Single Direction Positioning (G60)

Since this function allows precise positioning with backlash excluded, positioning can be performed from only one direction.

2-4 Circular Interpolation (G02, G03)

Circular interpolation can be performed arbitrarily at 0 to 3600 at the feed rate specified by an F-code in a G02 or G03 command.

2-5 Radius Designation on Arc (G02, G03)

R can be directly specified as a circular arc radius value, assuming I, J, and K to be a vector amount from a start point to the center in circular interpolation.

2-6 Sine Curvilinear Interpolation <Virtual Axis Interpolation> (G07)

By performing interpolation without moving one axis within a circular arc plane (hypothetical axis) in a helical cutting command, sine curvilinear interpolation is performed between the remaining two axes.

2-7 Helical Cutting (G02,G03)

Another axis is linearly interpolated synchronously with circular interpolation.

2-8 Polar Coordinate Interpolation (G120,G121)

A command programmed in the orthogonal coordinate system is converted into a linear axis move (tool) and rotary axis move (work rotation) to control a profile.

2-9 Cylindrical Interpolation (G271)

If a linear axis stroke and rotary axis angle are specified by a program command, the rotary axis stroke internally specified in terms of angle is converted into a distance on the circumference. As the distance on the circumference can be regarded a linear axis stroke on the circumference, linear interpolation and cirfular interpolation can be performed in combination with other linear axis.

3. THREAD CUTTING

4. FEED FUNCTIONS

4-1 Rapid traverse Rate and Rapid Traverse Override

The speed in the axial direction is allowed up to 240,000 mm/min (IS-B).

An override can be applied to a rapid traverse rate by rapid traverse override.

4-2 Cutting Feed Rate and Feed Rate Override

A feed rate is allowed from 6 to 240,000 mm/min (IS-B).

Feed rate override allows you to apply an override in an increment of 10% from 0% to 200%.

Table 4-2 shows the feed rate command value range.

Table 4-2

		Type	F-command Range
Feed per minute(G94)	Metric input	F60	F1~F999999 mm/min
		F61	F0.1~3F999999.9 mm/minmin
		F62	F0.01~F999999.99 mm/min/min
	Inch input	F51	F0.1~F99999.9 inch/minn
		F52	F0.01~F99999.99 inch/min
Feed per revolution (G95)	Metric input	F32	F0.01~F999.99 mm/rev
		F33	F0.001~F999.999 mm/rev
	Inch input	F23	F0.001~F99.999 inch/rev
		F24	F0.0001~F99.9999 inch/rev
Threading (G33)	Metric input	F35	F0.00001~F999.99999 mm/rev
		F26	F0.000001~F99.999999 mm/rev
	Inch input	F26	F0.000001~F99.999999 inch/rev
		F17	F0.0000001~F9.9999999 inch/rev

(Note 1) The type can be selected with a parameter.

(Note 2) The cutting feed rate is a command given relative to the reference axis.

(Note 3) The maximum cutting feed rate is limited by the cutting feed clamp rate set with a parameter.

(Note 4) When an F1-digit feed option is added, F1-F9 have special meanings.

[Related Parameters]

No. 1003, #0, #1 — Sets the type IS-A, IS-B or IS-C.

#2 = 0 — Disables the input increment 10.
Enables the input increment 10.

No. 3401, #0 = 0 — Specifies F23 for feed per revolution in inches.
1 — Specifies F24 for feed per revolution in inches.

#1 = 0 — Specifies F32 for feed per revolution in millimeters.
1 — Specifies F33 for feed per revolution in millimeters.

#2 = 0 — Specifies F51 for feed per minute in inches.
1 — Specifies F52 for feed per minute in inches.

#3 = 0 — Specifies F60 for feed per minute in millimeters.
1 — Specifies F61 for feed per minute in millimeters.

#4 = 0 — Specifies F26 for threading lead in inches.
1 — Specifies F17 for threading lead in inches.

#5 = 0 — Specifies F35 for threading lead in millimeters.
1 — Specifies F26 for threading lead in millimeters.

#6 = 0 — Feed per minute of 0mm abides by #3.
1 — Feed per minute of 1mm for F62.

4-3 Override Cancel

A cutting feed override rate can be fixed at 100% by a signal from the machine.

4-4 Automatic Acceleration/Deceleration

Linear acceleration/deceleration is performed in case of rapid traverse, and exponential function type acceleration/deceleration is performed in case of cutting feed or jog feed.

4-5 Dwell (G04)

Migration to operation in the next program block can be delayed by a specified time by a G04 command. Use P, X, or U for an address.

4-6 Exact Stop Check (G09)

In the block where G09 is specified, an imposition check is made at the end of block execution.

4-7 Exact Stop Check Mode (G61)/Cutting Mode (G64)

Normally, the G64 mode is effected and the program proceeds to the next block immediately after interpolation is completed. If G61 is specified, the program will proceed to the next block after entering imposition at the end point of each block, in the subsequent move commands. The G61 mode is cancelled by specifying G64.

4-8 Automatic Corner Override (G62)

An override is applied automatically to a cutting feed rate at a corner during tool diameter compensation.

5. REFERENCE POINT

5-1 Reference Point Return A (G27 to G29)

Reference point return a includes the following:

(1) Manual reference point return
(2) Reference point return check (G27)
(3) Automatic reference point return (G28)
(4) Return from the reference point (G29)

5-2 Reference Point Return B (G30)

Second reference point return (G30) returns the axes to the position set in a parameter.

5-3 Third/Fourth Reference Point Return (G30)

The axes can be returned to the 3rd/4th reference point preset by a G30 command (P3, P4).

5-4 Floating Reference Point Return (G301)

The axes can be returned to the preset optional point of the machine.

6. COORDINATE SYSTEM

6-1 Coordinate System Setting (G92)

An axis command following G92 sets the coordinate system where a current tool coordinate value will be a specified value.

6-2 Machine Coordinate System Selection (G53)

A tool moves to a position in the machine coordinate system by a G53 command.

6-3 Plane Designation (G17, G18, G19)

A G-code is used to specify the plane where you want to perform circular interpolation, tool diameter compensation, and so on.
G17: X-Y plane, G18: Z-X plane, G19: Y-Z plane

6-4 Local Coordinate System Selection (G52)

With a G52 command, you can set a child coordinate system, that is, local coordinate system in all the work coordinate systems (G54 to G59).

6-5 Work Coordinate System setting (G54 to G59)

One of the preset coordinate systems is selected by a G-code, G54 through G59. The subsequent program is executed in that selected coordinate system. The number of additional pairs is 60.

7. COORDINATES AND DIMENSIONS

7-1 Absolute/Incremental Programming

Absolute/incremental programming is switched by a G-code.

G90 : Absolute programming

G91 : Incremental programming

7-2 Decimal Point Input

A decimal point can be input to the command data associated with a distance (angle), speed, and dwell. A decimal point position is after the millimetric or inch units digit. Decimal point usable addresses include X, Y, Z, A, B, C, U, V, W, I, J, K, R, P, and F. When P is a subprogram number, however, the decimal point is not available.

7-3 Inch/Metric Conversion (G20, G21)

You can select the inch system/metric system as units of input by specifying G20/G21.

8. SPINDLE FUNCTIONS

8-1 Spindle Function (8-digit S-code)

By specifying an address S following by up to 8-digit numerical command, you can send out an analog signal and gear signal corresponding to a binary code signal and spindle motor rpm.

8-2 Spindle Override

An override can be applied from 50 to 150% in an increment of 10% by an external signal.

9. TOOL FUNCTIONS

9-1 Tool Function (8-digit T-code)

An 8-digit BCD code signal is sent out by specifying an address T followed by up to 8-digit numerical command.

9-2 Addition of Tool Offsets

The number of tool offset or tool diameter compensation pairs can be expanded up to 400.

9-3 Tool Life Management Function

The tools are sorted into several groups and when the cutting time or integrated cutting times of a tool in each group reaches the specified life time or cutting times, this function selects the next tool in the preset order.

9-4 Programmable Data Input (G10)

With a G10 command, you can choose to set or change a tool offset amount and change the work coordinate system (G54 to G59).

10. MISCELLANEOUS FUNCTIONS

10-1 Miscellaneous Function 8-digit M-code)

The machine can be turned on/off by specifying an address M followed by up to 8-digit numerical value.

10-2 Second Miscellaneous Function (B-function)

An 8-digit BCD code signal is sent out by an address A, B, or C followed by up to 8-digit numerical command, based on parameter setting.

10-3 Miscellaneous Function Lock

The M, S, T, and B-function commands are disabled. No signal is sent out to the machine.

11. PROGRAM CONSTRUCTION

11-1 Command Tape

8-unit black paper tape (EIA RS-227, ISO 1154, JIS C6246)

11-2 Tape Format

EIA/ISO (At input: Automatic recognition, At output: Selected by a parameter)

11-3 Input Format

A variable-block, word-address format with decimal point (EIA RS-274C, ISO R1056/R1058) is used.

11-4 Command Tape Codes

Address	Description
A	Additional axis coordinate value
B	Additional axis coordinate value, 2nd miscellaneous function
C	Additional axis coordinate value
D	Tool offset number selection
E	
F	Feed functions
G	Preparatory functions
H	Tool offset number selection
I	X-axis component of the circular arc center
J	Y-axis component of the circular arc center
K	Z-axis component of the circular arc center
L	Canned cycle times designation, Repeat times in a subprogram call
M	Miscellaneous function
N	Sequence number
O	Program number
P	Dwell, Program number in a subprogram call
Q	Canned cycle
R	Radius command value for circular interpolation, True circular cutting, Canned cycle
S	Spindle functions
T	Tool functions
U	Additional axis coordinate value
V	Additional axis coordinate value
W	Additional axis coordinate value
X	X-axis coordinate value, Dwell
Y	Y-axis coordinate value
Z	Z-axis coordinate value

11-5 Command Words and Command Value Ranges

Function	Address	Metric Input.	Inch Input.
Program number #	O	1~99999999	1~99999999
Sequence number #	N	1~99999999	1~99999999
Preparatory function	G	0~999	0~999
Coordinate value	X, Y, Z, U, V, W, I, J, K, L Q, R,	±99999.999mm	±99999.999mm
	A, B, C,	±99999.999deg	±99999.999deg
Feed per minute	F	1~999999mm/min	0.1~99999.9inch/min
Spindle function	S		
Tool function	T		
Miscellaneous function	M		
Dwell	P, X	0~99999.999sec	0~99999.999sec
Call program number #	P	1~99999999	1~99999999
Repeat times	L	1~99999999	1~99999999
Offset number #	D, H	0~400	0~400

11-6 Subprogram (M98, M99)

A subprogram can be called in the MEMORY mode. A called subprogram can further call another subprogram. The subprogram can be called eightfold at maximum.

11-7 Programmable Mirror Image (G501, G511)

A mirror image can be applied to each axis by a program command.

11-8 Direct Tap (G741, G841)

In the G741/G841 tap cycle, high-speed, high-precision tapping can be performed by completely synchronizing spindle rotation with Z-axis feed.

11-9 Optional Block Skip

A program block containing a slash code, “/”, in its beginning is ignored by turning on the OPTIONAL BLOCK SKIP switch provided on the part of the machine. You can add “/2” through “/9” (optional block skip 2 through 9) as an option.

11-10 Control-in/-out

“(“: Control-out

“)”: Control-in

This function is used when giving a program name to a program number or giving a comment halfway a program. All the information between control-out and control-in is ignored within a significant information section.

11-11 Command Data Input Methods

(1) MDI (manual data input) through the keyboard
(2) Inputting from an external input/output device via an RS-232C interface (Reading the NC tape)

11-12 Internal Data Output Methods

(1) Displaying on the CRT
(2) Outputting to an external input/output device via an RS-232C interface (Punching out the NC tape)

12. HOW TO FACILITATE PROGRAMMING

12-1 Canned Cycle for Drilling (G73, G74, G76, G80 to G89)

Drilling, tapping, and boring cycles can be specified in one program block.

12-2 Drilling Pattern Cycle (G70, G71, G72, G77)

By specifying a radius and angle, a drilling position is calculated into the orthogonal coordinates to perform positioning. A canned cycle is used in combination.

12-3 ATC Canned Cycle (M06)

If M06 is specified upon completion of machining by the spindle tool, the machine operates as follows. This simplifies the program, ignoring a warming-up period for the ATC.
<Example of Cycle>

① M15 ..... Spindle stop (M05) and coolant stop (M09)
② Z-axis to the ATC position ..... 1st or 2nd reference point
③ X-and Y-axis to the ATC position 1st or 2nd reference point and spindle positioning (M19)
④ ATC activated (M06)

12-4 Optional Angle Chamfering Corner R (, C/, R)

Optional angle chamfering or corner R can be inserted automatically by adding C or R to the end of the program block where linear or circular interpolation is specified.

12-5 Screen-driven Special Canned Cycle

Machining profile patterns such as drilling, circle machining, square plane machining, square side machining, track machining, and pocket machining can be easily programmed through the screen, and complicated machinings can be performed in one program block.

13. TOOL OFFSET FUNCTIONS

13-1 Tool diameter Compensation (G40 to G42)

A tool diameter can be compensated by specifying a G-code command, G40 through G42. An offset number can be set by a D-code, set by the lower 4 digits of a T-code, or selected, depending on parameter setting.

13-2 Tool Length Compensation (G43, G44, G49)

A tool position can be offset (tool length compensation) by a G43/G44 command. An offset number can be set by an H-code, set by the lower 4 digits of a T-code, or selected, depending on parameter setting.

13-3 Tool Offset (G45 to G48)

A tool position can be offset by specifying a G-code command, G45 through G48. The tool position is extended or contracted to a move command in the axial direction by the offset amount specified by a D-code or H-code.

13-4 Three Dimensional Tool Offset (G40, G41)

When machining a three dimensional curved surface, the offset amount set in the tool offset memory is offset three dimensionally by specifying an offset component in the three dimensional direction.

14. ACCURACY COMPENSATING FUNCTIONS

14-1 Backlash Compensation

This function is to compensate the lost motions which the mechanical system has. A compensation amount can be set as a parameter in the least command increment for each axis within a range of up to 9,999 pulses.

14-2 Stored Pitch Error Compensation

This function is to compensate a pitch error for feed screws. Compensation data is set as a parameter. The number of compensation positions is 128 for each axis.

15. COORDINATE CONVERSION

15-1 Axis Switching

According to a selection of the machining plane, this function changes the program addresses, X, Y, and Z, specified in the program into the machine axis addresses, and changes the signs of the machine axes. This enables the program to use the right-handed orthogonal coordinate system for each machining surface.

15-2 Scaling (G50, G51)

The profile specified in the machining program can be expanded or contracted at your desired scale factor.

15-3 Coordinate Rotation (G68, G69)

The profile specified in the machining program can be rotated as mentioned in (A) or (B) below.

(A) When assuming the rotation center to the origin of the work coordinate system

(B) When specifying the rotation center in the program

16. MEASURING FUNCTIONS

16-1 Skip Function

If a skip signal is input from an external device in the midst of an X-, Y-, or Z-command following G31, the next block will be executed, cancelling the rest of this command.

16-2 Work Setter (Datum Level, Master Hole)

This function is to write a work coordinate system shift amount automatically by simple manual operation, using a touch sensor.

16-3 Tool Setter (Tool Length, Tool Diameter)

This function is to write a tool offset amount automatically by simple manual operation, using the touch sensor.

16-4 Safety Guard

A machining tool length is measured by starting the program for the first time. When it is started for the second time, a workpiece at an actual machining position is measured by the measuring device attached to the spindle. Putting these two information and the offset amount used together, a workpiece-tool interference is checked for by an approach command (G00).

17. CUSTOM MACRO

17-1 Custom Macro

A function peculiar to the user can be created. There are 100 common variables, but their number can be optionally extended up to 600.

18. AXIS CONTROL

18-1 Follow-up Function

In case of emergency stop or servo alarm, a machine travel amount is reflected on an NC unit internal position. For this reason, automatic operation is enabled after resetting the emergency stop or servo alarm, even if you do not have to perform zero point return.

In case of speed feedback or position feedback alarm, however, an actual machine position and the NC unit internal position do not match, because the follow-up function does not work properly.

18-2 Mirror Image

This function can reverse the sign of the travel direction specified by a program command or MDI command for the X-, Y-, Z-axis, or an additional axis. Make this setting in the setting screen.

18-3 Oscillation Function

This function is to reciprocate a positioning axis, which is not used for cutting a machining profile, over a width asynchronous with a cutting plane axis.

19. MANUAL OPERATION

19-1 Manual Continuous Feed

① Jog feed
A jog feed rate is the speed set in a parameter applied an override of 0 to 655.34% in an increment of 0.01%.

② Manual rapid traverse
Manual rapid traverse is also allowed. An override is applied to the rapid traverse rate set as a parameter.

19-2 Manual Pulse Generator

The machine is capable of fine feed by means of the pulse generator on the machine operation panel. One rotation of the pulse generator generates 100 pulses. You can select a scale factor of x1, x10, or xM (M=1 to 1,000 set in a parameter) by a signal from the machine.

20. AUTOMATIC OPERATION

20-1 DNC Operation

DNC operation can be performed from the host CPU equipped with an RS-232C interface.

20-2 Program Number Search

An 8-digit program number following 0 can be searched for from the data in the program screen.

20-3 Sequence Number Search

A sequence number can be searched for in the program currently selected from the data in the program screen.

20-4 Restart of Program

To restart a program, there are three ways; program restart, block restart, and machining break point return.

① Program restart is a function to restart from the block of a specified sequence number.
② Block restart is a function to restart from the beginning of or halfway the block.
③ Machining break point return is a function to position a tool to a break point by jog feed.

20-5 Sequence Number Comparison and Stop

If you encounter the block of a preset sequence number during program execution, the machine stops after executing that block.

20-6 Preread Buffer

In order to avoid discontinuation of the program blocks at the time of cutting due to the processing time of the NC unit, the preread buffer generally prereads one program block in case of automatic operation.

(1) The preread buffer prereads the different number of program blocks, depending on the function.

Function	No. of Preread Blocks
Multibuffer	14 blocks
High-accuracy profile control	Up to G05 P0 block
Tool diameter compensation	2 to 4 blocks
Others	0 or 1 block

(Note 1) In the tool diameter compensation mode, the preread buffer prereads up to 4 blocks if they contain the blocks free from an axis move command.

(Note 2) The following commands suppress the preread buffer.
Example: G28, G30, G31, G53, G05, G10, G20, G21, G22, G23, M00, M01

(2) When the SINGLE BLOCK switch is turned on
When the SINGLE BLOCK switch is turned on and the program blocks are executed sequentially by pressing the CYCLE START switch, the preread buffer does not exist. Because, when the CYCLE START switch is pressed, one program block is taken into the preread buffer and executed immediately. Therefore, the preread buffer does not exist during and after execution.

(Note 1) The preread buffer prereads the program blocks in the following modes.
(1) Tool diameter compensation mode (G41, G42)
(2) When optional angle chamfering corner R is specified (, C, R)
(3) Thread cutting (G33)
(4) Tapping mode (G63)
(5) High-accuracy profile control (G05 P10000)

(Note 2) When the SINGLE BLOCK switch is turned from OFF to ON during automatic operation to stop it, the preread buffer exists.

20-7 Feedhold

All axes can be stopped temporarily. Pressing the CYCLE START button restarts feeding the axes. Prior to restarting axis feed, you can allow intervention by manual operation in the manual mode.

20-8 External Reset and Reset Signal

The NC unit can be reset from the outside. A reset cancels all the commands and decelerates the machine to a stop. A reset signal is output to the machine while the RESET button of the MDI & CRT panel is being pressed, the machine is being reset by an external reset signal, or the EMERGENCY STOP button is being pressed.

20-9 Override Memory and Automatic Override

When performing trial cutting, an overrides (feed override/spindle override) according to the machining conditions is memorized for each tool. At the time of machining, the above-mentioned memorized override is reflected automatically, not the override rate selected at the operation panel.

20-10 Data Server

A large-capacity program can be processed at a high speed by means of the Ethernet controller and the hard disk of the auxiliary storage unit. (For details, see SEIKI-SEICOS Σ DATA SERVER INSTRUCTION MANUAL, FTP TYPE)

21. PROGRAM TEST FUNCTIONS

21-1 Machine Lock

In the machine lock mode, the machine does not move, but the position display is updated as if the machine were moving. Z-axis command cancel becomes equivalent to when a machine lock is applied only to the Z-axis.
This is effective when checking the contents of the machining data by pen-writing.
When the machine lock is turned ON→OFF in auto operation, Machine is shifted by the amount moved by the machine lock.

21-2 Dry Run

If the DRY RUN switch is turned on, the machine operates at a jog feed rate instead of a programmed cutting feed rate. This function can be also enabled in case of rapid traverse by parameter setting.

21-3 Single Block

Program commands can be executed block by block.

21-4 Pre-Machining Plotting

In pre-machining plotting, as machine performs synchronous plotting while in auto running in machine lock and dry run state, format failures and erroneous coordinate commands, if any, in a program can be easily detected.
When, on start of pre-machining plotting, an interlock signal, etc. to stop interpolation has been made effective, pre-machining plotting is stopped with the corresponding command.

22. DISPLAY AND SETTING

22-1 Machining End Notice

Input a scheduled program end time. When the machining time reaches the scheduled end time, a signal is output to an external device.

22-2 Run Hour Display

Machine run hours are displayed in the format of hours:minutes:seconds.
They are displayed for each of the functions such as scheduled end time, machining time, lap T, date and time.

23. PART PROGRAM STORAGE & EDITING

23-1 Part Program Storage & Editing

The contents of the NC tape can be stored and edited. They can be deleted, altered, and inserted, and an editing range can be set by expanded part program editing. Use of backgrounding allows you to edit another program during automatic operation.
Tape storage length: 80, 160, 320, 500, 1,000, 2,000, or 4,000m
Registered programs: 100, 200, 400, 800, or 1,000 programs

23-2 Part Program Comparison

The program registered in the memory is compared with the one in the tape.

24. DIAGNOSTIC FUNCTIONS

24-1 Self Diagnostic Function

This function makes various checks.

① Automatic operation starting condition
② Manual operation starting condition
③ Manual pulse generator starting condition
④ Speed setting status

24-2 Cutting Monitoring Function

This function monitors the cutting load of the spindle and feed axes to prevent abnormal cutting or defective cutting.

25. DATA INPUT AND OUTPUT

25-1 Input/Output Interface (RS-232C)

This function allows you to output the programs, tool offset amounts, parameters, etc. memorized in the memory to an external device, and input the data from the external device. A device equipped with the RS-232C interface is available as an external device.

26. SAFETY FUNCTIONS

26-1 Emergency Stop

An emergency stop cancels all the commands and stops the machine instantaneously.

26-2 Overtravel

When the machine reaches a stroke end, a relevant signal is received, the axes are stopped instantaneously, and an overtravel alarm is indicated.

26-3 Interlock

When an interlock is applied to any one of the operating axes, all the axes are decelerated to a stop. When an interlock signal is reset, they are accelerated to restart operation.

26-4 Stored Stroke Limit 1

Stored stroke limit 1 assumes the outside of the area set by a parameter to be a prohibited area.

26-5 Stored Stroke Limit 2 and 3 (G22, G23)

Use this function when you want to ensure that a tool will not enter a non-cutting area. Both stored stroke limit 2 and 3 assumes either inside or outside of the set area to be a prohibited area, based on parameter setting. This function is enabled or disabled by a G-code command.

G22 : Enabled

G23 : Disabled

26-6 Stroke Check Before Move

This function checks whether or not specified end point coordinates enter a stored stroke limit area before a move command in the program block.

27. STATUS OUTPUT

27-1 NC Ready Signal

When the NC unit is turned on and becomes ready to control, this signal is output to the machine, and when the NC unit is turned off, a signal output to the machine is called off.

27-2 Automatic Operation Running Signal

This is a signal to be output while automatic operation is under way.

27-3 Automatic Operation Stopping Signal

This is a signal to be output while the program is stopping due to feedhold.

27-4 Distribution Complete Signal

This is a signal to be output upon completion of distribution so that the M-, S-, T-, or B-function can be executed after completing a move command in the block where there were specified.

28. EXTERNAL DATA INPUT

28-1 External Data Input

The data are sent from a machine's external device to the NC unit to carry out required operation.

① External O- or N-number search
② External tool offset data read

29. HIGH-SPEED CUTTING

29-1 High-accuracy Profile Control

Machining errors due to the CNC includes the one resulting from acceleration/deceleration after interpolation. In order to eliminate this error, the following functions are realized at a high speed by the RISC processor.

(1) Acceleration/deceleration function before multiblock preread interpolation which does not cause any machining errors due to acceleration/deceleration.
(2) Automatic speed control function which can realize smooth acceleration/deceleration considering a change of profile and speed, and allowable acceleration of the machine by prereading multiple blocks.

3. OPERATION

I. Basic Machine Operation

II. Screen Operation

3. OPERATION

I. Basic Machine Operation

1. Manual Operation
2. Automatic Operation
3. Operation Related to Safety
4. NC Operation Keys
5. Operation Related to W Setter

(Note 1) The operation panel varies from one machine to another. See the Instruction Manual for your machine.

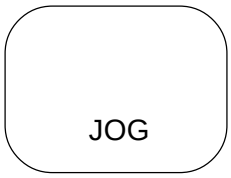
1. Manual Operation

The machine can be manually operated by using the swiches on the machine operation panel.

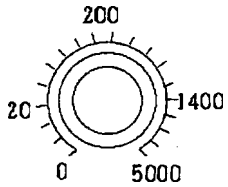
1-1 Jog Feed

The machine can be operated continuously by manual operation.

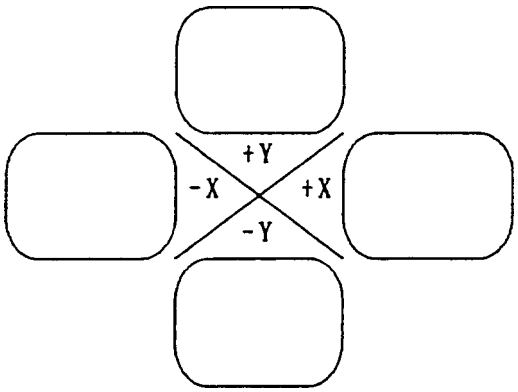
(1) Select the mode selector switch “JOG”.



(2) Select the jog feed rate.



(3) Select the axis you want to move.



The machine moves in the direction of the selected axis.

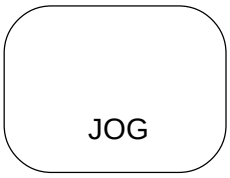
(Note 1) When multiple axes are selected, those axes move all simultaneously.

(Note 2) When the axis has been selected before selecting the JOG mode, the machine does not move even if the mode is changed to JOG.
Select the axis newly.

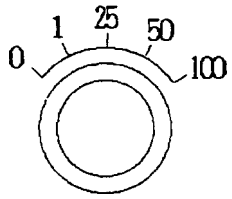
1-2 Manual Reference Point Return

The machine can be returned to the reference point by manual operation.

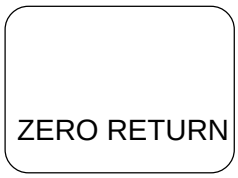
(1) Select the mode selector switch “JOG”.



(2) Select the rapid traverse rate.



(3) Select “ZERO RETURN”.



The machine moves at the rapid traverse rate toward the reference point for each axis.

(4) When the machine returns to the reference point, the reference point return lamp gets illuminated.

(Note 1) Once reference point return is completed and the reference point return lamp gets illuminated, manual reference point return operation cannot be performed again until the reference point return lamp is turned off.

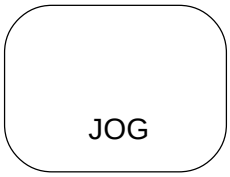
(Note 2) The reference point return lamp goes off in the following cases.
- (1) When the machine moved from the reference point
- (2) When an emergency stop resulted and the machine moved

(Note 3) When performing low-speed manual reference point return, do so from the position a little distant from the reference point.

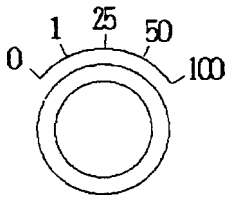
1-3 Rapid Traverse

The machine can be moved continuously at a rapid traverse rate by manual operation.

(1) Select the mode selector switch “JOG”.



(2) Select the rapid traverse override.



(3) Select the axis you want to move, and “RAPID” at the same time. The machine moves in the direction of the selected axis at the rapid traverse rate.

(Note 1) Same as Notes for Jog Feed

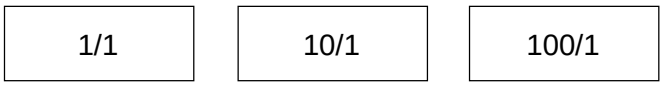
1-4 Manual Handle Feed

The machine can be finely fed by turning the manual pulse generator.

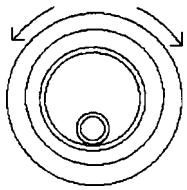
(1) Select the mode selector switch “HANDLE”.

(2) Select the handle axis.

(3) Select a handle magnification.



(4) Turn the handle.



Right turn: + direction
Left turn: - direction

(Note 1) Do not turn the manual pulse generator so quickly. If so done, the machine may not stop immediately after turning the handle, or the scale and the travel amount may not coincide with each other.

(Note 2) If the magnification “x 100” is selected and the handle is turned very quickly, the machine moves at a rate close to the rapid traverse rate. If you then stop the machine suddenly, it may be shocked.

(Note 3) In some cases, the mode select switch (1) is not provided with “handle”, where Handle mode is selected through (2) handle axis selection.

1-5 Manual Handle Interrupt (Optional)

This function serves to move an axis by combining a manual handle feed shift command with a shift command in auto running. With this, you can shift the tool path by an amount equal to the travel amount created by handle operation.

(1) Turn ON handle interrupt.

(2) Select a handle axis.

(3) Select a handle magnification.

(4) With the handle turned, travel amount by the handle is placed upon the travel amount by auto running, moving the axis. The travel amount by the handle is displayed in the manual interrupt position.

(5) Turn OFF handle interrupt.

(Note 1) The travel amount by handle interrupt is determined by the amount of turning of the manual pulse generator and the handle feed magnification (x1, x10, xn), to which, however, no acceleration/deceleration is applied. Therefore, handle interrupt, if effectuated with a large magnification, can be extremely dangerous. Avoid selecting a large magnification.
Travel amount per scale for x1, regardless of input unit, is 0.001mm (for IS-B).

(Note 2) No handle interrupt can function while in machine lock or in interlock.

(Note 3) Relations between shifting by handle interrupt and various position indications are as follows:
Work coordinate value : No change
Relative coordinate value : No change
Machine coordinate value : Interrupt amount alone is changed.

2. Automatic Operation

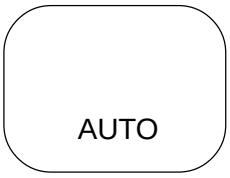
2-1 Automatic Operation

(1) Memory operation

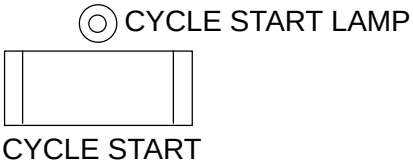
(a) Store the program in the memory in advance.

(b) Select the program you want to run.

(c) Select the mode selector switch “AUTO”.



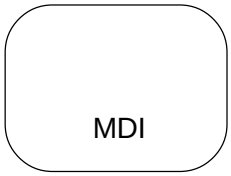
(d) Press the CYCLE START button.



Pressing this button starts automatic operation and turns on the CYCLE START lamp.

(2) MDI operation

(a) Select the mode selector switch “MDI”.



(b) Input the program into the MDI operation buffer memory.
The commands for multiple blocks can be input into the MDI operation buffer memory from the CRT/MDI panel. The program can be edited in the same manner as editing that stored in the memory.

(c) Press the CYCLE START button. Automatic operation starts and the CYCLE START lamp gets illuminated.

2-2 Selecting the Run Program

(1) Program No./sequence No. search

(a) Select the mode selector switch “AUTO”.

(b) Display the Overall screen. (The Program screen will also do.)
When any other screen than the Overall screen is displayed, press the RETURN key several times to display the Overall screen.

(c) Press the O key in case of program number search, and press the N key in case of the sequence number search.

(d) Then, enter the program number or sequence number you want to search for, and press the cursor move key.

(e) A program or sequence number search is executed.

(2) Rewind

(a) Select the mode selector switch “EDIT”.

(b) Display the Overall screen. (The Program screen will do.)

(c) Press the RESET key of the NC unit.

(d) Select the mode selector switch “AUTO”.

(3) Registerable programs

You can select one of the following numbers of registeralbe programs.

100, 200, 400, 800 or 1,000 programs

(Note 1) For 200 or more programs, the tape storage length of 320m or more is required.

(4) Tape storage length

You can select one of the following tape storage lengths.

80, 160, 320, 500, 1,000, 2,000 or 4,000m

(Note 1) The number of registerable programs for 80/160m is fixed at 100.

(Note 2) The number of registerable programs for 320/500m is 400 or less.

2-3 Stopping the Automatic Operation

There are two methods to stop automatic operation; one is to insert a stop instruction (M00, M01, M02 or M30) in the program in advance at which you want it to stop, and the other is to press the button (FEEDHOLD or RESET) on the operation panel.

(1) Program stop (M00)

If the block where M00 has been specified is executed, automatic operation stops and execution does not proceed to the next block. (the CYCLE START lamp remains turned on) The modal information so far is all saved.

Pressing the PROGRAM CYCLE START button restarts automatic operation.

(2) Optional stop (M01)

If the block where M01 has been specified is executed, automatic operation stops and execution does not proceed to the next block. (The CYCLE START lamp remains turned on) However, this is true only when the OPTIONAL STOP switch on the machine operation panel is turned on.

(3) Program end (M02, M30)

(a) Indicates the end of the main program.

(b) Stops automatic operation and places the machine in the reset mode.

(c) Normally, M02 moves the cursor to the next block, and M30 returns the program to its beginning.

2-4 Dry Run

This function enables a dry run speed, ignoring the feed rate specified with the program.

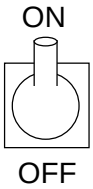
Normally, dry run is effective only during cutting feed.

To make it effective during rapid traverse as well, set “1” in the bit 6 (RDR) of the parameter 1401.

2-5 Single Block

The single block function stops the machine after executing one block.

Turn on the SINGLE BLOCK switch.

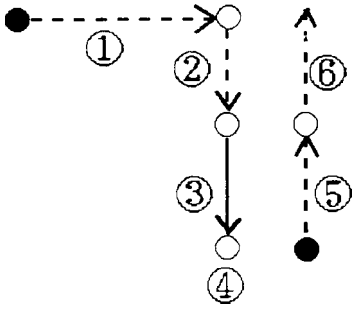


This causes the machine to stop after executing one block.

Pressing the PROGRAM CYCLE START button stops the machine after executing the next block.

(Note 1) Even at the middle point of the G28/G29/G30 command, the machine stops after executing one block.

(Note 2) The single block stop points in the canned cycle for drilling are ① , ② and ‡E shown in the figure below.



2-6 Override

For details related to the override function, refer to the instruction manual for the machine.

(1) Feed rate override

With the switch on the machine operation panel, an override of 0% to 254% (in an increment of 1%) can be applied to feed per minute (G94) and feed per revolution (G95).

(2) Rapid traverse override

The override selected with the switch on the machine operation panel can be applied to the rapid traverse rate.

(3) Feed rate override cancel

With a signal from the machine side, the feed rate override can be fixed at 100%.

2-7 Pre-Program Machining Time display Function (Optional)

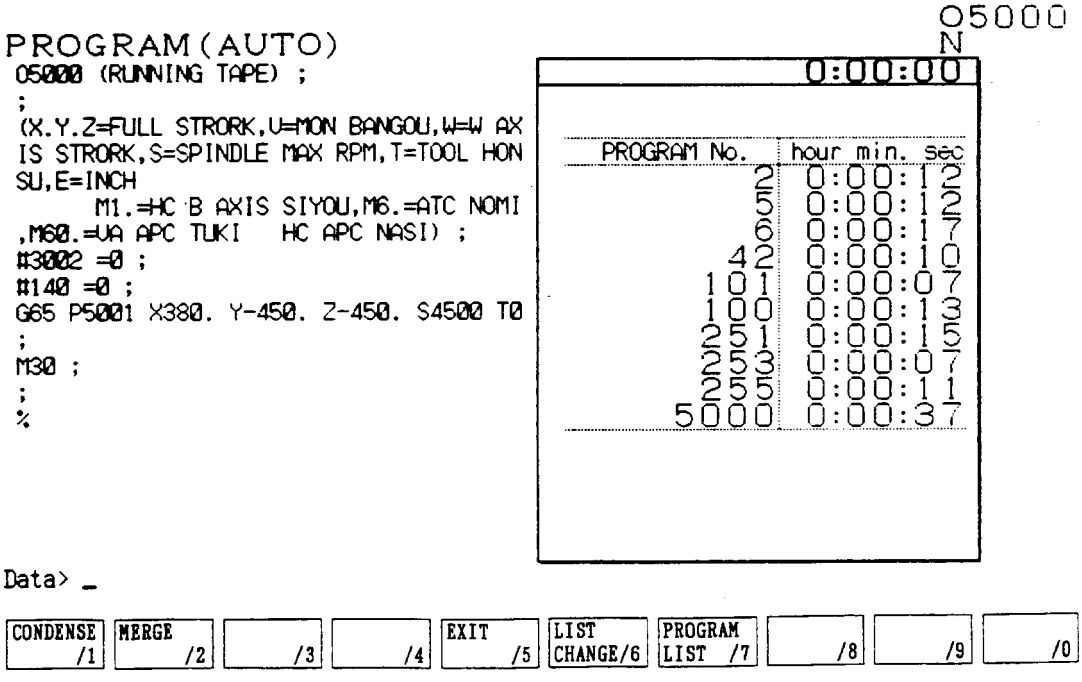
If it is executed to run the program, the program's machining time is displayed on the machining time display screen.

It is possible to display up to 10 main program numbers and the machining time. If more than 10 programs are run, programs are discarded the oldest one first. The machining time of the same program number is updated, however.

The time from start in the reset status to the next reset status being assumed is counted in the memory run mode. The "reset status" as referred to here is the status reset by the reset key, the external reset signal, M02, M30, etc.

Only the time of automatic run being activated is counted, so the stopped or single block stop time is not added.

Machining Time Calculation and Display



(1) Select the automatic (memory) run mode and press the reset key.
(2) Select the overall (program) screen and search for the program the machining time of which is to be calculated.
(3) Execute the program for actual machining.
(4) It is ended to count the machining time when the reset key is pressed or when M02 or M30 is executed machining.
(5) Select the program screen, press “7” (program list) of the menu and then press “5” (machining time) of the menu, and the machining time screen will be displayed.
(6) The program number and the time are displayed one by one on the machining time screen as the procedure from (1) to (5) is repeated.
It is possible to display the machining time of up to 10 programs.
If the number of programs exceeds 10, programs are discarded the oldest one first. The machining time of the same program number is updated, however.

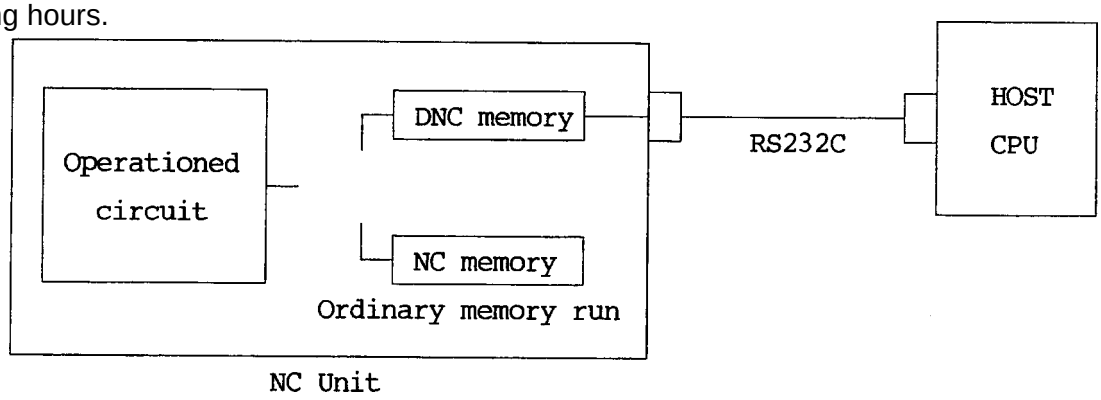
2-8 Tape Operation (Simple DNC Operation)

This function allows you to directly operate the machine with a program from the paper tape or external device, without using the internal memory of the NC unit.
From the tape operation program, you can also call a subprogram saved in the NC unit internal memory.
(Note) At the time of tape operation, be sure to disable DNC operation.

① In the Input/Output screen, select an input device and baud rate.
(When using the RS-232C of the operation panel, select RS-232C.
② With the mode selector switch, select TAPE.
③ Make setting on the part of the input device to be ready for operation.
(If a reader, set paper tape, and if the RS-232C, make a connected device ready for output.)
④ Press the PROGRAM START button.
→ One block worth of the program is read from the external device to start tape operation.
(Note) Since there is an internal processing buffer, the data for the buffer is preread.

2-9 DNC Operation (Optional)

This, by receiving NC command data from the host CPU, continuously performs machining for long hours.

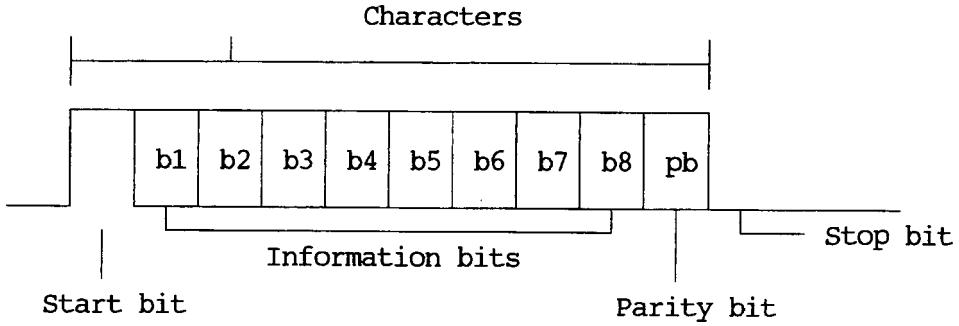


(1) Interface Between Host CPU and NC Unit

	RS232C Interface
Transmittal mode	Serial voltage interface (start/stop system)
Baud rate	1200 to 19200 (For details, see Parameters.)
Max. cable length	15m
Noise resistance	Inferior

Start/stop system

As in the following drawing, the start signal and the stop signal come preceding and following respectively in relation to the information bits.



CNC OPTION-1 BOARD
JD5C
(PCR-EV20MDT)

1	2	3	4	5	6	7	8	9	10
RD	OV	DR	OV	CS	OV	CD	OV		+24
11	12	13	14	15	16	17	18	19	20
SD	OV	ER	OV	RS	OV			+24	

When no using CS, short-circuit it with RS.
When using DR, short-circuit it with ER.
Be sure to short-circuit CD with ER.

(2) Parameter Setting

① No.0180 I/O channel when a remote buffer is used

Be sure to set 3.

② No.0181 The specification No. of the I/O unit when a remote buffer is used

Set Value	I/O unit
0	Control code (DC1 to DC4) is used.
4	Control code (DC1 to DC4) is not used.

③ No.0182 Baud rates when a remote buffer is used

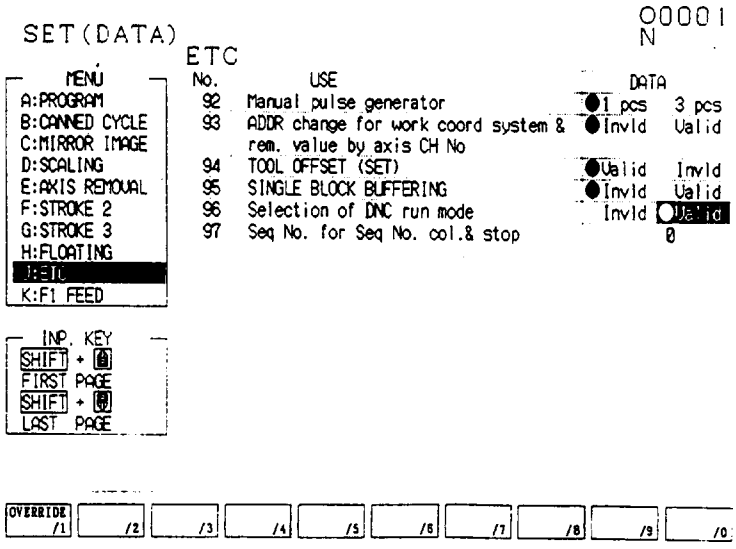
Set Value	Baud Rate	(bps)
1	50	
2	100	
3	110	
4	150	
5	200	
6	300	
7	600	
8	1200	

Set Value	Baud Rate	(bps)
9	1400	
10	4800	
11	9600	
12	19200	
13	38400	
14	76800	
15	86400	

The set values 13 to 15 are only useable for RS422.

(3) Operation

① Push F5/Set to enter Setting screen for other selections.



② Set DNC Operation Mode Selection to “YES”.

③ Pushing [RETURN] to enter Overall screen, select [Memory] mode.

④ Start the host CPU.

⑤ Pushing [START] , start running.

Program transfer into the DNC operation memory is started and, in a few seconds, running starts.

*Note) Steps ④ and ⑤ get reversed depending on the transfer conditions of the host CUP.

<Starting Following sequence No. Search>

You can start operation with the sequence No. having been assigned in the DNC program.

① Execute ① to ④ of the above item.

② Input, following “N”, a sequence No. where operation is to be started.

③ Push [↓] key.

Program transfer starts into DNC operation memory and searching is started.

On completion of searching, the sequence No. having been found is displayed on the screen at its beginning. When it has not been found, an alarm takes place.

④ Push [START] for running. Operation starts with the sequence No. thus having been found.

2.10 Submemory (Unavailable for Σ10)

2.10.1 Outline

The submemory is a feature to increase the capacity of the program memory by using a storage medium newly added into the NC unit(normally an ATA flash card *1) as the program memory.

The ATA flash card is called the submemory and the conventional memory(battery backup RAM) the main memory, respectively. Up to 10 directories can be created in the submemory and one of them is selected for operation as required.

*1 When a hard disk data server is attached, the submemory also uses a hard disk.

(1) Program storage length

You can select 1,000, 2,000, 4,000, or 10,000 m on the part of the submemory.

The total program storage length equals this length plus the capacity of the main memory.

(Example) When the main memory is 80 m long and the submemory 2,000 m long, respectively, the total program storage length is 2,080 m long.

(2) Number of programs

Up to 1,000 programs held in one directory plus the main memory

Up to 2,000 programs held in all the directories in the submemory

(Example 1) When no directories are created in the submemory;
The maximum number of programs is 1,000.

(Example 2) When the directories D1 and D2 are created in the submemory;
The following three conditions must be all satisfied:

- Number of programs in the main memory + Number of programs in D1 $\leqq$1,000
- Number of programs in the main memory + Number of programs in D2 $\leqq$1,000
- Number of programs in D1 + Number of programs in D2 $\leqq$ 2,000

(3) Directories in the submemory

Up to 10 directories can be created in the submemory.

(When the directories are not created at all, only one basic directory is to be used.)

Program operation and program editing apply to the programs in the main memory and those in the directory currently selected(to be referred to as a current directory hereinafter) in the submemory. The programs in the non-current directories in the submemory can be neither run nor edited.

The current directory is treated fully differently from the others in the submemory. You may create the program with the same program number in a different directory.

(Example 1) When no directories are created in the submemory;
All the programs are to be run/edited.

(Example 2) When the directories D1 and D2 are created in the submemory;

- When the current directory is D1;
  $\rightarrow$ The programs in the main memory + D1 are to be run/edited.
- When the current directory is D2;
  $\rightarrow$ The programs in the main memory + D2 are to be run/edited.

To create, alter, or delete the directories in the submemory, use the Directory Setting screen.

→ See “2.10.4 Submemory Directory Operation”

(4) Using the main memory and submemory properly

Whether the program is to be stored in the main memory or sumemory depends on its program number. Use a parameter to set which range of program numbers are to be assigned to the main memory side. Typical setting assigns O8000 to O9999 to the main memory and the rest to the submemory.

This is to treat the programs O8000 to O9999 as the subprograms which can be commonly used for any directory in the submemory. Thus, the macro programs provided by the machine manufacturer are made available no matter which directory in the submemory is selected.

(5) When the programs in the submemory are not available

The programs in the submemory are not available on the part of the lathe related NC unit’s 2nd system(loader control), simple 2nd system(feeder control), and loader control only machine(R21 version). Use and run only the programs in the main memory.

2.10.2 Program Editing

The program editing functions(back editing, extended editing, etc.) are available for both main memory and submemory the same as before.

Since the submemory uses the ATA flash card as a storage medium as described above, editing should be operated file by file like editing the files in the hard disk through a personal computer. As a result, you have to read the file before editing operation and save it after editing operation, causing entire editing operation to take more time than direct editing(directly altering part of the file) of the RAM-based main memory.

When editing the program on the part of the submemory, save it at the time of the following:

- When the screen is switched.
- When the mode is changed.
- When other program is called.
- When the machine is reset despite the STANDBY switch has been turned on.

(Note) If the power is turned on without saving the program after editing it, you will be returned to the conditions existing before editing when the power is turned on again.

If the power is turned off while saving the program, it may be corrupted. If the program has any errors, initialize the program memory.

2.10.3 Program Operation

The programs in the main memory and the current directory in the submemory can be run the same as before. For the programs in the submemory, however, there are the following restrictions to the number of programs which can be run simultaneously.

- When the data server is attached: Up to 5 programs
- When the data server is not attached: Up to 4 programs (Additional one program for the data server)

Specifically, multiplicity of subprogram(macro program) call to the programs in the submemory is limited by the above-mentioned numbers. There are no restrictions to the programs in the main memory.

The following describes this, using a specific example. The program on the part of the main memory is represented by Mn, that on the part of the submemory by Sn(n = 1, 2, 3, ...), and program call by →, respectively.

(Example 1) For the conventional specifications(without the submemory)

The following is allowed:

M1 → M2 → M3 → M4 → M5 → M6 → M7 → M8 → M9.

(Example 2) When the submemory is provided, but not data server; In the following state;

S1 → M1 → S2 → S3 → S4 → S5

Since five programs are used on the part of the submemory, it is possible to call only the main memory program after S5.

S1 → M1 → S2 → S3 → S4 → S5 → M2 Allowed

S1 → M1 → S2 → S3 → S4 → S5 → S6 Disallowed

(Example 3) When both submemory and data server are provided; In the following state;

M1 → S1 → S2 → S3 → S4

Since four programs are used on the part of the submemory, it is possible to call the main memory program or data server program after S4.

M1 → S1 → S2 → S3 → S4 → M2 Allowed

M1 → S1 → S2 → S3 → S4 → Data server Allowed

M1 → S1 → S2 → S3 → S4 → S5 Disallowed

(Note) It is possible to cancel by parameter setting the above-mentioned “uniform restriction on the number of programs on the part of the submemory, which can be simultaneously run.”

Whether this cancellation of restriction allows a program call beyond the limit, however, depends on the then running submemory program(generally program length).

This is briefly described below.

On the part of the submemory, up to five program read ports(called the channels) are available for operation. One channel is required to read one program; up to 5 programs can be run simultaneously.

Upon calling the program, however, if the program to be called has been read into the NC unit(about 10 m worth of read buffer) to the end, that channel can be closed and used to run other program.

Whether the program to be called can be read into the NC unit to the end upon calling it depends on he conditions such as the run time from the beginning to a call command, program length from the call command to the end, and so on. Basically, the number of simultaneously run submemory programs should be used within the above-mentioned limit.

2.10.4 Submemory Directory Operation

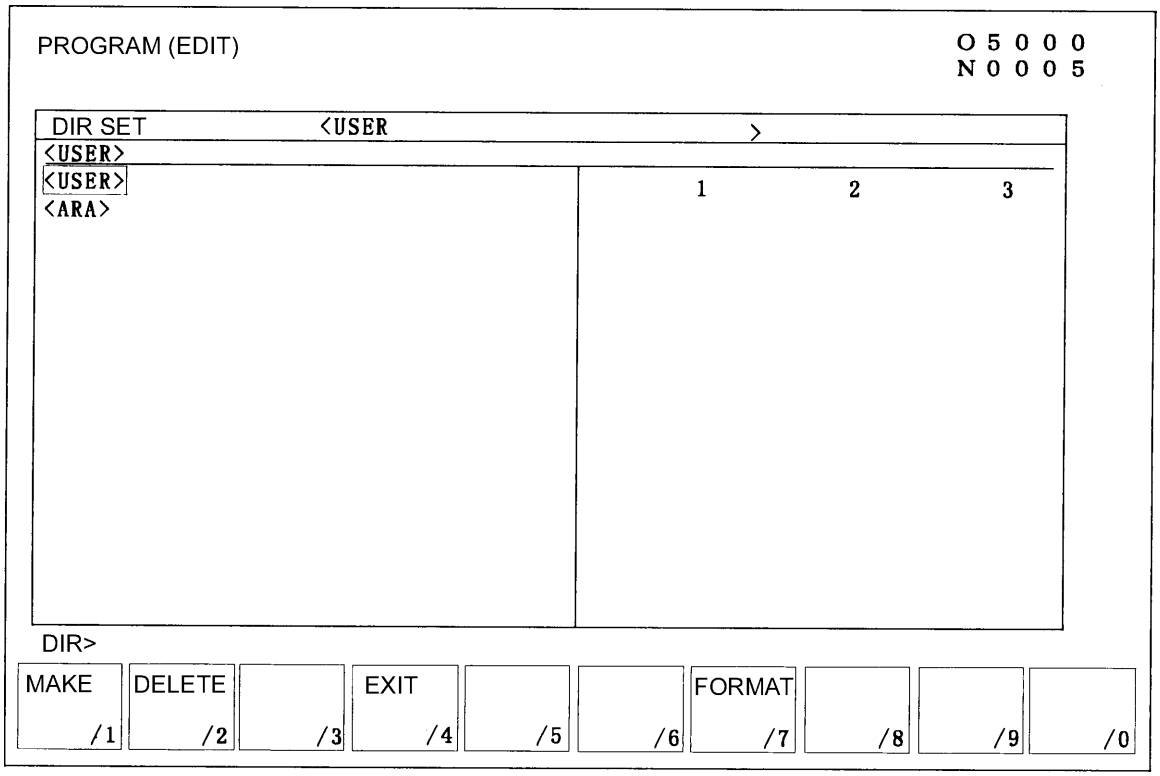
The program number can be used up to 8 digits, but you may want to use the same O-number to perform different machining in many cases. In such cases, it is very troublesome to replace the O-numbers included in the NC unit by means of input/output.

To avoid this trouble, the programs can be easily replaced by preparing several O-number storage areas(directories) and altering them. This is called directory setting.

Submemory directory operation(creation, alteration, deletion) is performed in the Directory Setting screen.

The Directory Setting screen is displayed by operating the following keys in the Overall screen.

F2/PROGRAM → F7/PROGRAM LIST → F4/DIR SET



This screen allows you to view the directory names and the program numbers registered in the respective directories. It also allows you to create new directories and delete unnecessary ones.

(Note) Directory related operation takes several to scores of seconds depending on the number of registered programs.

(1) Creating the directory

The following describes the procedure to create a directory.

① Input the up to 8-character name of the new directory you want to create into the key input area.

② Press F1/MAKE .

③ You are prompted, “O.K. ? Y-YES N-NO.”

④ Press Y to create or N to cancel.

(Note 1) In creating the directory, you simply name a storage area, but not change it. When using the created directory, execute the setting.
(Note 2) Up to ten directories are allowed.
(Note 3) The directory name is up to 8 characters.
(Note 4) Another directory cannot be created under one directory.
(Note 5) It is not allowed to give the directory name consisting of only numbers such as 1, 2, 3, 4. Its initial character must be an alphabet such as A1234. No decimal point(.) is available.

(2) Deleting the directory

The following describes the procedure to delete an unnecessary directory and the rograms contained in it.

① Move the cursor to the directory you want to delete.
② Press [F2/DELETE] .
③ You are prompted, “DO YOU WANT TO DELETE ? Y-YES N-NO.”
④ Press [Y] to delete or [N] to cancel.
If the directory being set is deleted, USER will be automatically set.

(Note 1) The deleted program or directory cannot be restored.
(Note 2) The USER directory is indispensable and cannot be deleted.

(3) Setting the directory

The following describes the procedure to alter the directory.

① Move the cursor to the directory you want to set.
② Press [INPUT] .
③ You are asked, “O.K. ? Y-YES N-NO.”
④ Press [Y] to set or [N] to cancel.
⑤ After the procedure is finished, you will be returned to the Programs List screen.

2.10.5 Associated Parameters

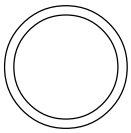
No. 3152 Minimum value of the program number to be stored in the main memory (Standard value = 0)
No. 3153 Maximum value of the program number to be stored in the main memory (Standard value = 0)
※ Regardless of this setting, O8000 to O9999 are stored in the main memory.
※ If this set value is altered, it is necessary to initialize the program memory(main memory/ submemory).
No. 8906, #0 = 0 Does not display the submemory access status.
= 1 Display the submemory access status in the center of the first line in the screen.
No. 6002, #0 = 0 Up to 5 submemory programs can be run simultaneously. (Up to 4 when the data server is attached)
= 1 Up to 9 submemory programs can be run simultaneously depending on the condition. (See Note in 2-10-3)

3. Operation Related to Safety

3-1 Emergency Stop

Pressing the EMERGENCY STOP button on the machine operation panel can stop the machine immediately.

EMERGENCY STOP



This switch is locked by pressing and released by turning.

(Note 1) Emergency stop shuts off a current to the motor.

(Note 2) The NC unit is reset to warn you of an emergency stop.

(Note 3) Before releasing the EMERGENCY STOP switch, it is necessary to eliminate a trouble cause.

3-2 Overtravel

When the tool moves over the stroke end set by the limit switch of the machine or enters the disabled area of the set stored stroke limit, an alarm is displayed and the moving axis is decelerated to a stop.

In this case, move the tool manually in the safe direction and press the RESET key to reset the alarm.

4. NC Operation Keys

No.	Name	Description
①	POWER ON/OFF button	Used to turn on/off the power for the CNC unit.
②	RESET key	Press this key when resetting the CNC unit in order to reset an alarm, and so on.
③	Function keys	When the function menu is displayed at the bottom of the CRT, there are the keys to select the menu. When the menu is not displayed, they serves as the keys to select the Position, Program, Tool, Work Coordinate, Setting, Plot and I/O screens. Press the F9/SELECT/FUNK key when displayed the function menu at the bottom of the CRT.
④	OPER/MAINTE key	Press this key when displaying the PC, Alarm or Maintenance screen. Pressing it once displays the function menu at the bottom of the CRT, and pressing it again erases the menu.
⑤	RETURN key	Press this key when you want to return to the Overall screen.
⑥	AUX. key	
⑦	HELP key	
⑧	Address and Numerical key	Used to input the alphabet, numbers, etc.
⑨	SHIFT keykeys	There are some address keys which have 2 characters marked on them. If you press the address key after the SHIFT key, upper left character is input.
⑩	INPUT key	If the address or numerical key is pressed, it is input into the key input buffer once, and then, displayed on the CRT. Press the INPUT key when actually setting the data input into the key input buffer.
⑪	CAN key	Press this key when deleting the characters or symbols input into the key input buffer.
⑫	ORIGIN key	This key is used to clear the Plot screen.
⑬	DELET, ALTER and INSERT keys	Used to perform deletion, alteration and insertion in editing the program.
⑭	Cursor move key	There are 4 keys which are used to move the cursor up/down and right/left.
⑮	Page key	There are 2 keys which are used to page in the forward and backward directions.
⑯	Operation Guide	Press this key when you want to display in the Operation Guide screen.

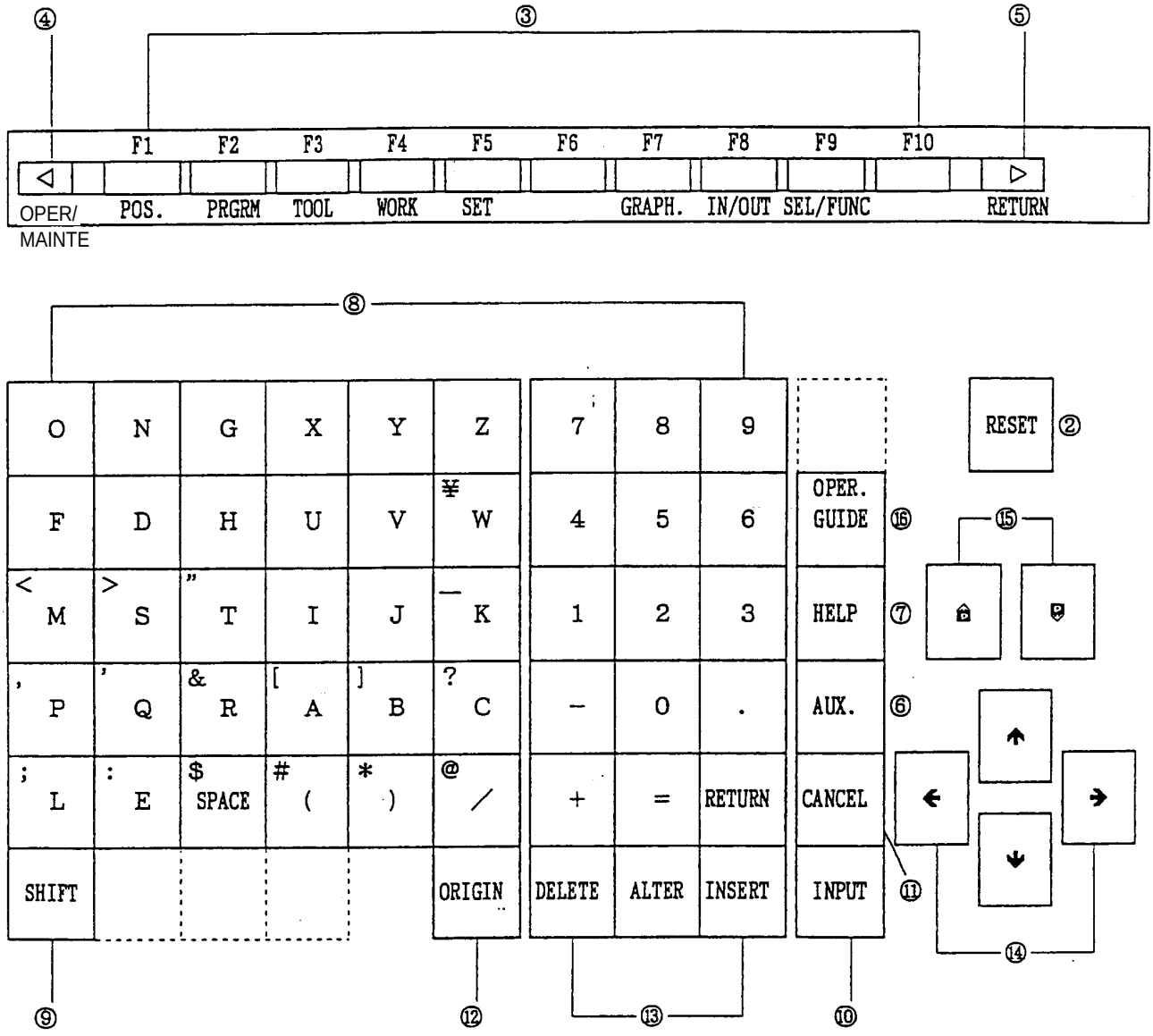


Fig. 4-1 NC Operation Panel

5. Operation Related to W Setter

5-1 Outline

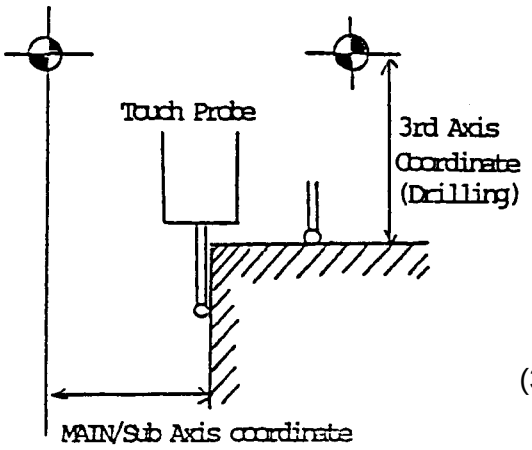
Using the touch sensor, the tool offset amount and work zero point offset amount can be written automatically by simple manual operation.

- Work setter (Work zero point offset amount setting)
  - Datum level
  - Master hole
  - Coordinate modification
- Tool setter (Tool offset amount setting)

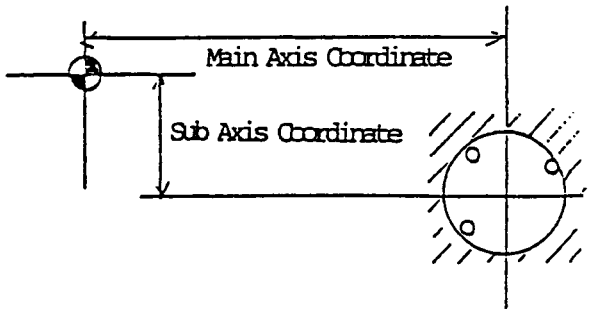
5-2 Input Signals and Various Operations

Work setter datum level selector button ..... WSS
Work setter master hole selector button ..... WSH
Work setter coordinate modification selector button ..... WSC
Tool setter selector button ..... OFH
Touch sensor signal ..... SKIP

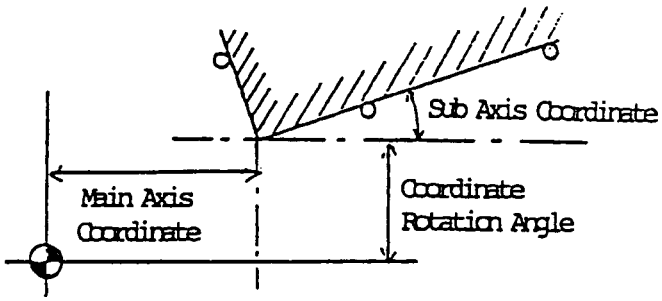
(1) Work setter datum level



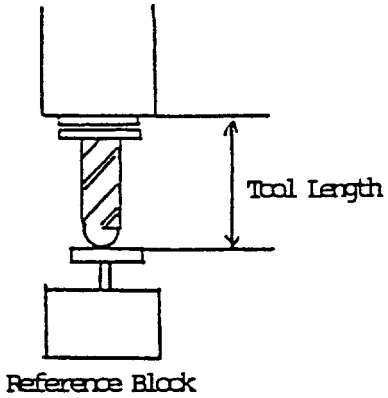
(2) Work setter master hole.



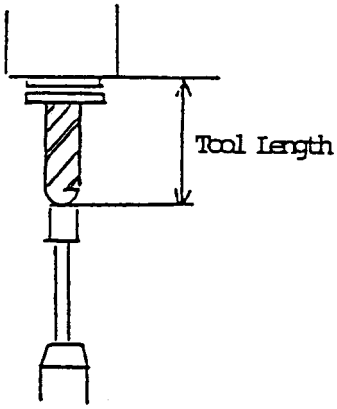
(3) Work setter coordinate modification



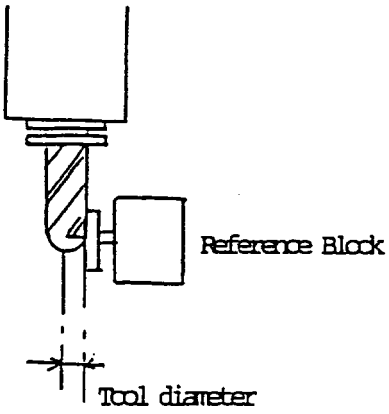
(4) Tool setter (Tool length)



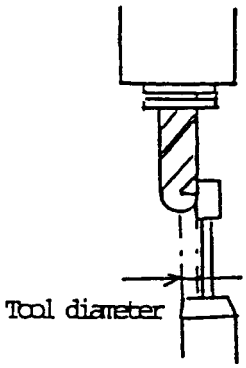
*When the touch sensor is used on the table



(5) Tool setter (Tool diameter)



*When the touch sensor is used on the table



5-3 Setting the Work Setter Datum Level

(1) Return the basic 3 axes manually to their reference points. Mount the touch probe to the spindle.
(2) In the manual mode, turn on the Work Setter Datum Level.
→The Work Coordinate (Offset) screen (Fig. 3-1) is selected automatically.
(3) Select the work coordinates you want to set with the page keys and cursor keys.
(4) Bring the touch probe into contact with the datum level by jog feed operation.
(5) The axis for which a touch sensor signal is turned on is judged automatically, and the work zero point offset amount is calculated and written automatically.
(6) After all the necessary work zero point offset amounts are set, turn off the Work Setter Datum Level.

(Note 1) The touch probe data used for offset amount calculation should be set in the parameters (No.6252 through No.6255) in advance.

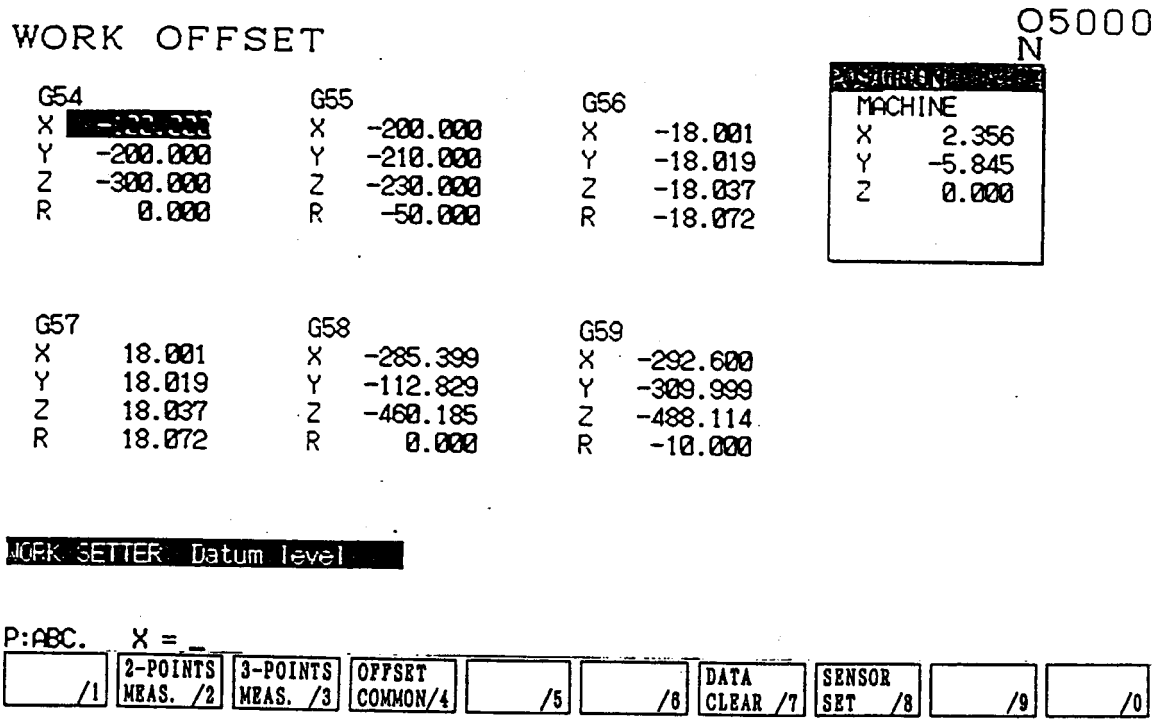


Fig. 3-1 Work Screen

5-4 Setting the Work Setter Master Hole

(1) Return the basic 3 axes manually to their reference points. Mount the touch probe to the spindle.

(2) In the manual mode, turn on the Work Setter Master Hole.
→ The Work Coordinate (Offset) screen (Fig. 3-1) is selected automatically.

(3) Select the work coordinates you want to set with the page keys and cursor keys.

(4) Bring the touch probe into contact with any one point of the master hole by jog feed operation.
→ The system stores the position of the main/sub axis for which a touch sensor signal is turned on, and displays a comment to prepare for measurement of the 2nd point.

(5) Bring the touch probe into contact with the 2nd measurement point of the master hole.
→ The system stores the position of the 2nd point and displays a comment to prepare for the measurement of the 3rd point.

(6) Bring the touch probe into contact with the 3rd point of the master hole.
→ The system stores the position of the 3rd point, calculates the center of the 3 points, and writes the work zero point offset amount automatically. A hole diameter is displayed to the right of the comment.
→ The system prepares for the measurement of the 1st point of the next master hole.

(7) After all the necessary work zero point offset amounts are set, turn off the Work Setter Master Hole.

* After (5), if you press the function keys F2:2-POINT MEAS., the center of the 2 points is calculated and the work zero point offset amount is written automatically. Also, the groove width is displayed to the right of the comment.

* If measurement is done in the 3rd-axis direction, the 3rd-axis work zero point offset amount is calculated and written automatically.

(Note 1) If reset operation is performed halfway in setting operation (while the 2nd or 3rd point is being measured), you are returned to the initial condition (condition to prepare for measurement of the 1st point).

(Note 2) When the distance among the 3 points is shorter than the parameter (N.6257), a comment is displayed, assuming it unmeasurable. Reset operation cancels this and returns you to the initial condition.

(Note 3) When you want to know only a hole diameter and a boss diameter without rewriting the work zero point offset amount, press the function keys F3:3-POINT MEAS., and bring the 3rd point into contact. The hole diameter and boss diameter are displayed to the right of the comment.
When you want to know only the groove width, press and F3, and then, F2:2-POINT MEAS. The groove width is displayed to the right of the comment.

(Note 4) Set in the parameters (N.6252-6255) in advance the touch probe data used for calculation of the work zero point offset amount.

5-5 Setting the Work Setter Coordinate Modification

(1) Return the basic 3 axes manually to their reference points. Mount the touch probe to the spindle.

(2) In the manual mode, turn on the Work Setter coordinate Modification.
→ The Work Coordinate (Offset) screen (Fig. 3-1) is selected automatically.

(3) Select the work coordinates you want to set with the page keys and cursor keys.

(4) Bring the touch probe into contact with any one point of the datum level by jog feed operation.
→The system stores the position of the main/sub axis for which a touch sensor signal is turned on, and displays a comment to prepare for measurement of the 2nd point.

(5) Bring the touch probe into contact with the 2nd measurement point of the master hole.
→The system stores the position of the 2nd point and displays a comment to prepare for the measurement of the 3rd points.

(6) Bring the touch probe into contact with the 3rd measurement point of the orthogonalized surface.
→The system stores the position of the 3rd point, calculates the corner position and coordinate rotation angle out of the values of the 3 points, and automatically writes the work zero point offset amount.
→The system prepares for the measurement of the 1st point of the next datum level.

(7) After all the necessary work zero point offset amounts are set, turn off the Work Setter Coordinate Modification.
* After (5), if you press the function keys F2:2-POINT MEAS., only the coordinate rotation angle is calculated and the work zero point offset amount is written automatically.
* If measurement is done in the 3rd-axis direction, the 3rd-axis work zero point offset amount is calculated and written automatically.

(Note 1) If reset operation is performed halfway in setting operation (while the 2nd or 3rd point is being measured), you are returned to the initial condition (condition to prepare for measurement of the 1st point).

(Note 2) When the distance among the 3 points is shorter than the parameter (N.6257), a comment is displayed, assuming it unmeasurable. Reset operation cancels this and returns you to the initial condition.

(Note 3) When machining by rotating the coordinates, specify a G-code (G68) for coordinate rotation in the program.

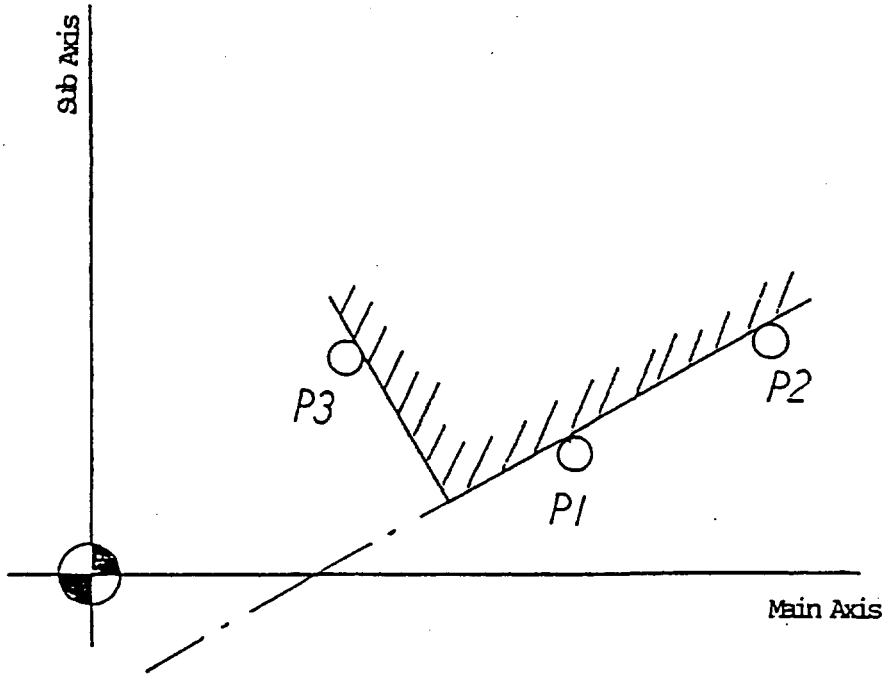


Fig. 5-1

The intersecting point of the straight line which passes through the points P1 and P2 and the one which is orthogonalized with this line and passes through the point P3 is assumed to be the corner position.

The coordinate rotation angle refers to the angle formed between the straight line passing through the points P1 and P2, and the main axis.

5-6 Rewriting the Work Zero Point Offset Amount Manually

(1) Pressing the function key F4: WORK selects the Work Coordinate (Offset) screen (Fig. 3-1).

(2) With the page keys and cursor move keys, select the work zero point offset amount you want to set.

(3) First, Press “P” (inputting a given value to the work zero point offset amount), “I” (additionally inputting a given value to the work zero point offset value) or “J” (inputting a given value plus the coordinate rotation angle to the work zero point offset amount).
Next, select the desired axis with the cursor move key or enter the desired axis name, and then, enter a “numerical value” and the INPUT key. This rewrites the specified work zero offset amount.

5-7 Setting the Tool Setter

(1) Return the basic 3 axes to their reference points manually. Mount the reference block or touch sensor on the table.

(2) In the manual mode, turn on the Tool Setter. → The Tool (Offset) screen (Fig. 7-1) is selected automatically.

(3) Set the distance from the origin to the reference block in the parameters (no.6249 to no.6251). When this value is a fixed one, this step is not required. When a reference gauge position is indefinite, it is necessary to set a correct value as follows.
Press [F2/REF.GAUGE]. The parameters (no.6249 to no.6251) are displayed on the lower right of the screen. Bring the touch probe into contact with the reference gauge. The distance from the origin to the reference block is set in the parameters (no.6249 to no.6251) by the same calculation as that for the work setter (datum level).
The safety guard parameters (no.6258 and no.6259) are also set simultaneously by measurement from the Z-axis direction. Be sure to press [F2/REF. GAUGE] to return to the original screen.

(4) With the page key and cursor move key, select the offset number you want to set.

(5) Bring the tool into contact with the reference gauge by jog feed operation. →The NC unit automatically discriminates the axis, for which a touch sensor signal was turned on, and writes the tool offset amount automatically.
Main/sub axis ..... The tool diameter is calculated and the tool profile offset amount for diameter compensation is rewritten.
3rd axis ..... The tool length is calculated and the tool profile offset amount for length compensation is rewritten.
At this time, the corresponding tool wear offset amount is cleared to 0.

(6) After all the necessary tool offset amounts are set, turn off the Tool Setter.

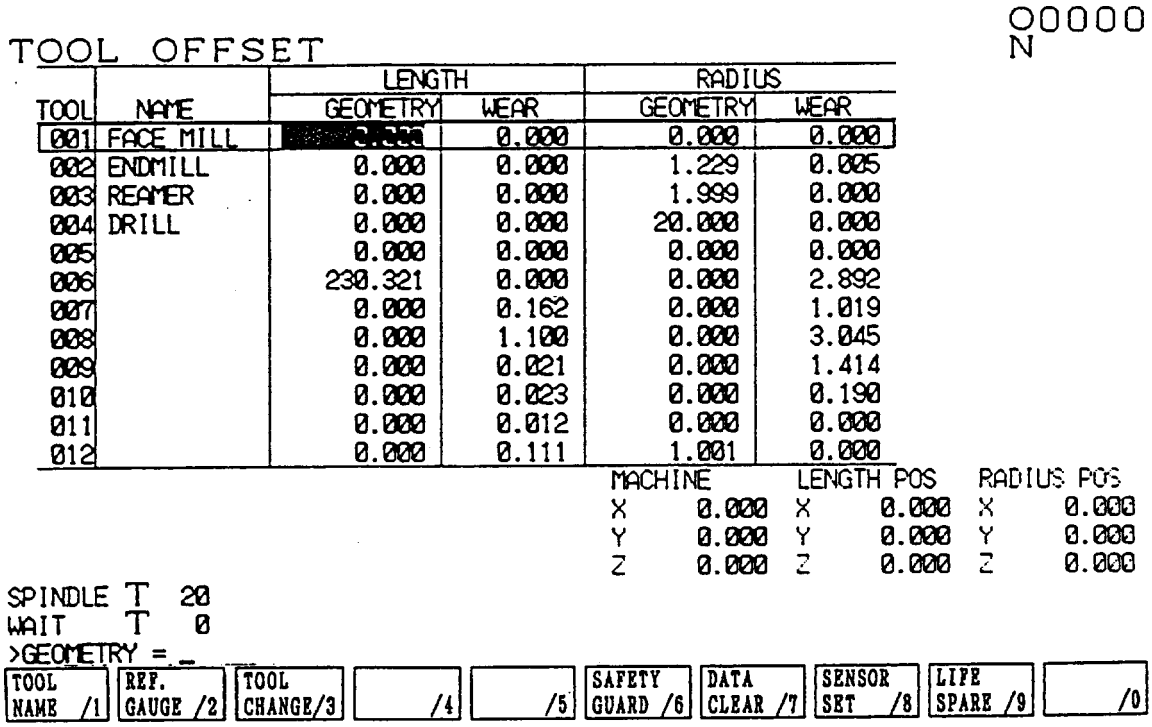


Fig. 7-1 Toll (Offset)

(Note 1) In writing the tool diameter, if you make measurement several times with the identical offset number set, the offset amount is updated only when a measured result is larger than the previous one.

However, if the offset number is respecified or after the TOOL SETTER button is repressed, the first measured result is written as the offset amount.

(Note 2) The reference gauge refers to the contact surface of the reference block. When using the on-table touch sensor other than the reference block, its contact surface is referred to.

(Note 3) The touch probe data used for measurement should be set in the parameters (N.6252-6255) in advance. To set it, there are two methods available; is to write with the parameters, and the other is to write through the Tool (Offset) screen. To use the latter method, press the function keys F8:SENSOR SET. The multi-window Setting screen (Fig. 7-2) appears and allows you to set the data.

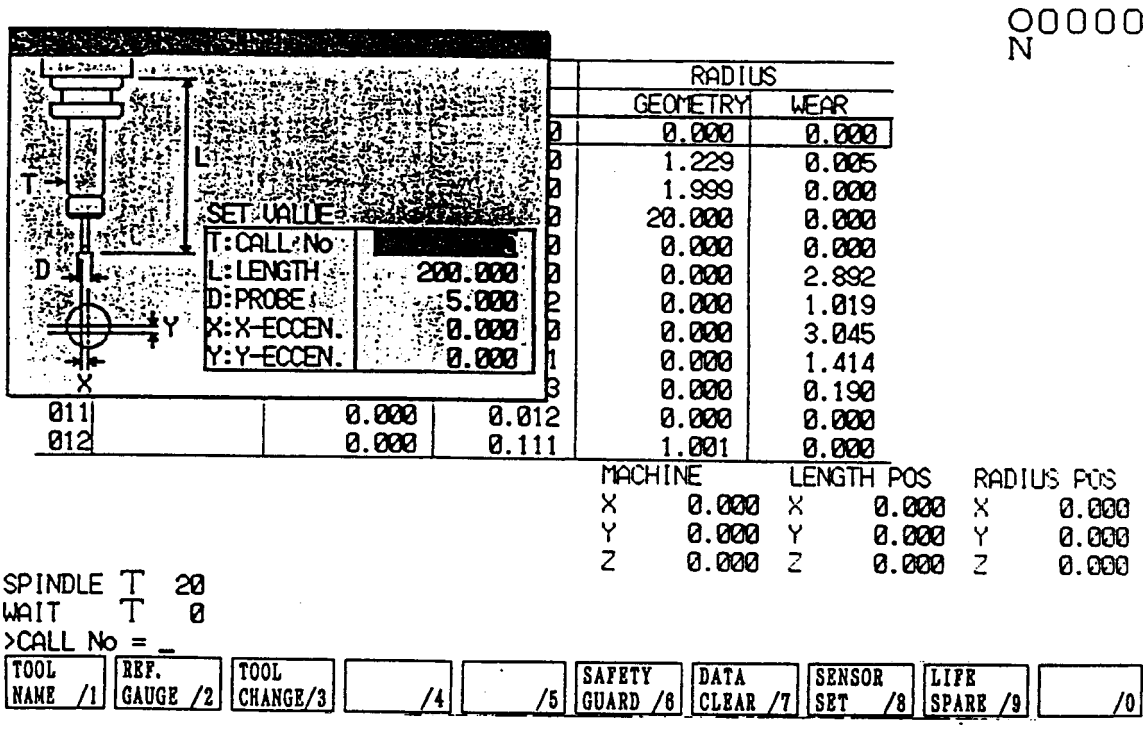


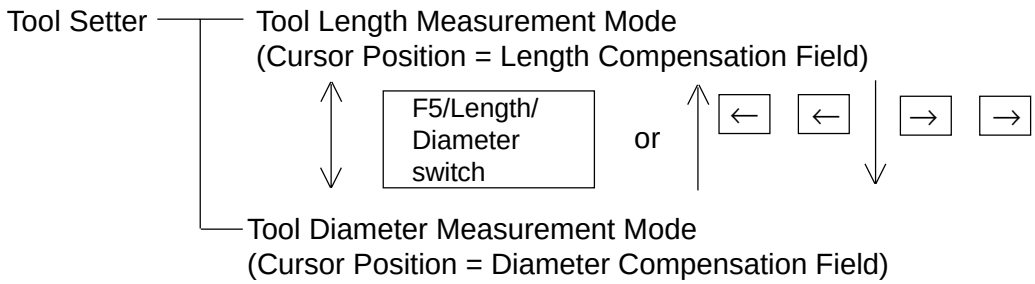
Fig. 7-2 Sensor Setting

The distance to the reference gauge can be measured, using the tool whose dimension is already known (called the reference tool) instead of the touch probe. In this case, set the parameters (N.6242, 5bit) to “1”, and the dimension of the reference tool in the parameters (N.6260-6263) in advance.

5-7-1 Tool Length Measurement Mode, Tool Diameter Measurement Mode

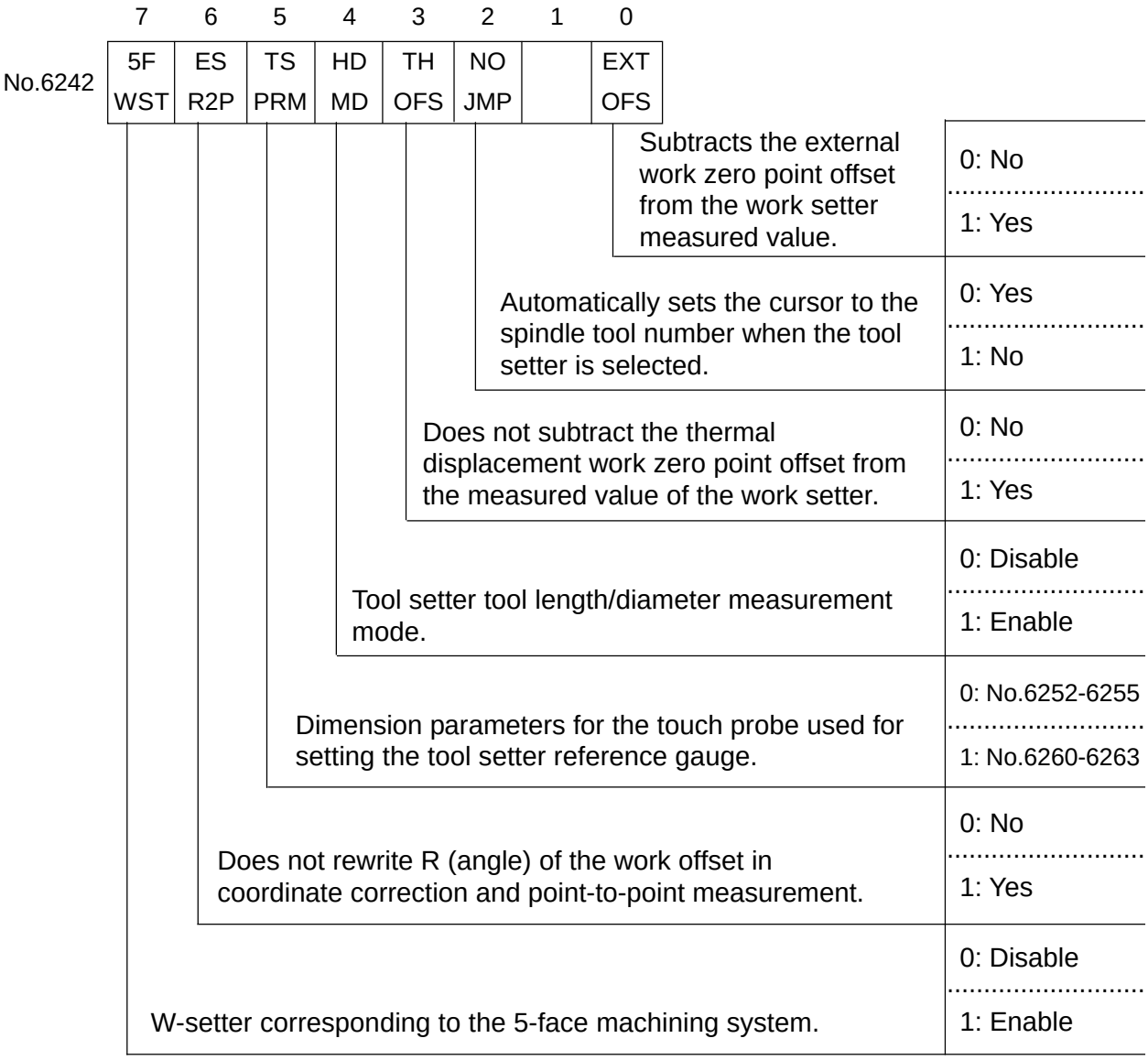
When the bit 4 of the parameter N.6242 is set to 1, the tool length/tool diameter measurement mode is enabled.

When this is done, the alarm No.716 results if a tool diameter is measured in the tool length measurement mode, or if a tool length is measured in the tool diameter measurement mode, or if measurement is done in other than the Tool (Offset) screen, thus enabling you to prevent an offset setting miss due to erroneous contact.



The tool length/tool diameter measurement mode is enabled even while setting the reference gauge. The X/Y position is set in the tool length measurement mode, and Z position is set in the tool diameter measurement mode.

5-8 Parameters



6249 Distance from the reference point to the 3rd-axis reference gauge

6250 Distance from the reference point to the sub-axis reference gauge

6251 Distance from the reference point to the main axis reference gauge

6252 Probe sphere diameter

6253 Eccentricity amount in the main axis direction from the center of the spindle shaft to that of the probe sphere

6254 Eccentricity amount in the sub-axis direction from the center of the spindle shaft to that of the probe sphere

6255 Distance from the gauge line to the end face of the 3rd-axis “-” direction of the probe sphere

6256 Return amount at retouch time

6257 Minimum inter-point distance value at 3-point measurement of the hole or corner

6260 Reference tool diameter

6261 Eccentricity amount in the main axis direction from the center of the spindle shaft to that of the reference tool

6262 Eccentricity amount in the sub-axis direction from the center of the spindle shaft to that of the reference tool

6263 Distance from the gauge lien to the end face of the 3rd-axis “-” direction of the reference tool

5-9 Others

(1) The work setter and tool setter are enabled in the manual mode.

(2) If a touch sensor signal is turned on while operating the work setter or tool setter, the axis is allowed to move only in the direction opposite to the direction in which the signal was turned on. The other axial moves are prohibited.

(3) If the touch sensor signal is turned on, the buzzer starts sounding.
If continues to sound until the axis has moved by a parameter (N.6256)-set amount in the direction opposite to the direction in which the then move axis was turned on. Even if the touch sensor is turned on while the buzzer is sounding, the work zero point offset amount and tool offset amount are not rewritten.

(4) Automatic rewriting of the work setter and tool setter are directed to the work zero point offset amount or tool offset amount indicated by the cursor when the touch sensor signal was turned on.
Even if the screen other than the Work Coordinate (Offset) screen and Tool (Offset) screen is displayed, the last place indicated by the cursor is stored.

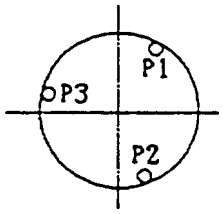
(5) Note that it sometimes becomes impossible to relieve the axis (impossible to move the axis) after touching if the sensor is made to touch by the following methods when measuring using the handle.

① Touch while turning the handle intermittently several memories at a time.

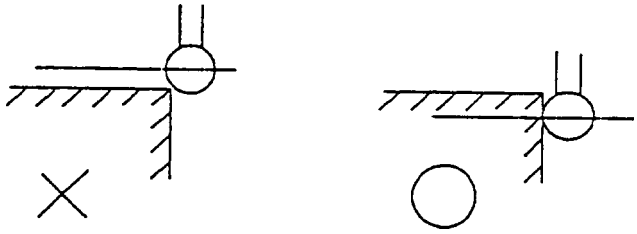
② Touch while moving the sensor close or far (turning the handle alternately in the “+” and “-” directions).

③ Touch at a very low speed.
If this trouble has occurred, relieve the axis after pressing “reset” and then re-measure.

(6) When making hole measurement, touch so that three measuring points will be as far as possible from each other.



(7) Make measurement near the center of the probe sphere.



(8) When the thermal displacement compensation function and work setter function are jointly used, note the following. Set the bit 3 of the parameter No.6242 to 1. Prior to measurement, specify the ATC (M06TXX) or thermal displacement compensation data enable M-code.

5-10 5-Face Working Machine's W Setter

In this item, the vertical machining center's W setter is denoted as the "vertical W setter" (explained in up to the preceding item) and the 5-face working machine's W setter as the "5-face W setter".

The 5-face W-setter performs measurement of the datum level, master hole, and coordinate modification, with the spindle attached vertically or horizontally (00, 900, 1800, 2700).

(1) Parameters

The parameters added in the 5-face W setter are shown below.

No.6307-6318	Side spindle's original position shifting amount (see 5-11 "Appendix")
No.6242, #7=0	5-face W setter disable (vertical W setter)
=1	5-face W setter enable
No.6319	Eccentricity in horizontal direction from horizontal spindle center to probe sphere center
No.6320	Eccentricity in vertical direction from horizontal spindle center to probe sphere center
No.6321	Distance from horizontal spindle gauge line to sensor tip

(2) Horizontal Spindle

When measuring by setting the spindle in the horizontal direction,the side axis face's original position shifting amount is to be set in arameter No.6307-6318 (For details, see 5-11 "Appendix".)

(3) Reference Plane Measurement

In the reference plane measurement, it is possible to measure 5 faces other than the one in the direct opposite direction as viewed from the touch sensor.

(Example) If the touch sensor is facing the X + direction, it is possible to measure from the 5 directions of X ±, Y ± and Z ±.

(4) Reference Hole Measurement

In the reference hole measurement, it is possible to measure the hole/boss of the plane which is vertical as viewed from the touch sensor.

(Example) If the touch sensor is facing the X + direction, it is possible to measure the hole/boss of the Y-Z plane.

(5) Coordinate Correction

In the coordinate correction, it is possible to measure the coordinate rotation of the plane which is vertical as viewed from the touch sensor.

(Example) If the touch sensor is facing the X + direction, it is possible to measure the coordinate rotation of the Y-Z plane.

(6) Tool Setter

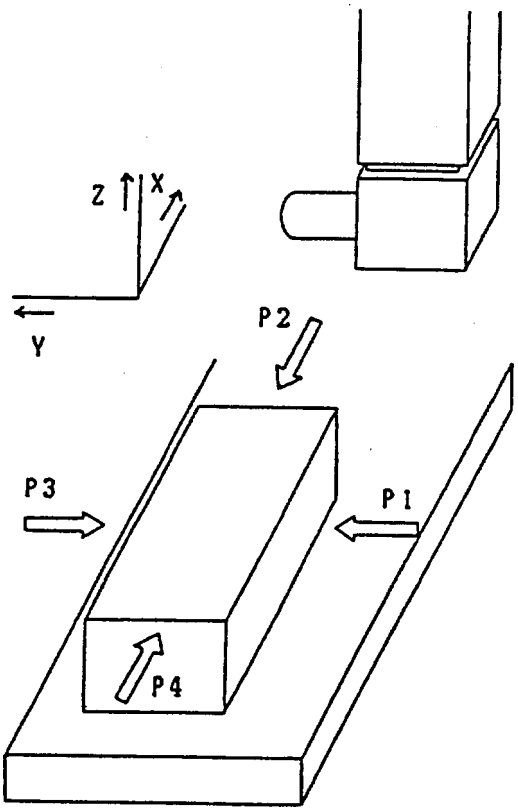
The tool setter is available only when the spindle is attached vertical.

(The method of use is the same as in the vertical W setter.)

5-11 Appendix

(1) Horizontal Spindle's Origin Shifting Amount (5-Face Working Machine)

Set the original position shifting amount when the spindle is attached horizontal in the parameters (No.6307-6318).



Parameter Setting

Shifting Amount When Spindle Was Set in Y (+) Direction (P1 Direction)

- No.6307 X shifting amount
- No.6308 Y shifting amount
- No.6309 Z shifting amount

Shifting Amount When Spindle Was Set in X (-) Direction (P2 Direction)

- No.6310 X shifting amount
- No.6311 Y shifting amount
- No.6312 Z shifting amount

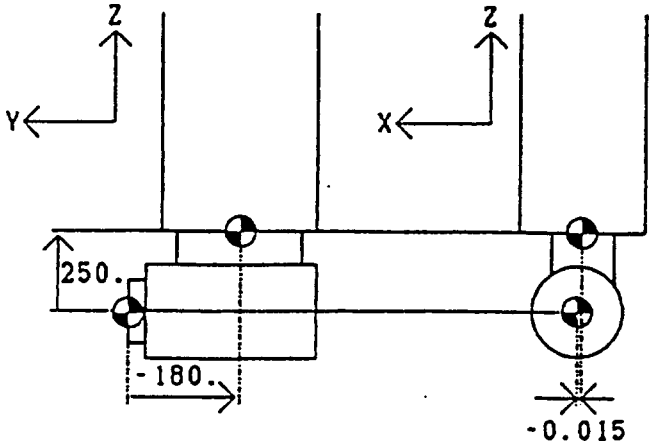
Shifting Amount When Spindle Was Set in Y (-) Direction (P3 Direction)

- No.6313 X shifting amount
- No.6314 Y shifting amount
- No.6315 Z shifting amount

Shifting Amount When Spindle Was Set in X (+) Direction (P4 Direction)

- No.6316 X shifting amount
- No.6317 Y shifting amount
- No.6318 Z shifting amount

・Example of Setting (If P1)

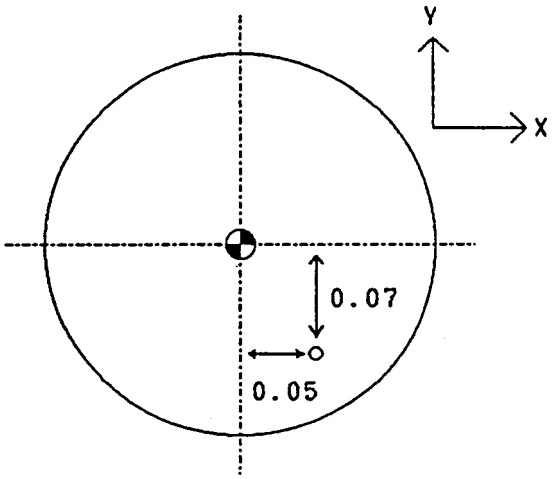


Parameter No.6307 = -0.015
No.6308 = -180
No.6309 = 250.

The coordinate system of the diagram takes the tool move for the work as reference.
Note that it may be different from in the JOG switch on the operator's console.

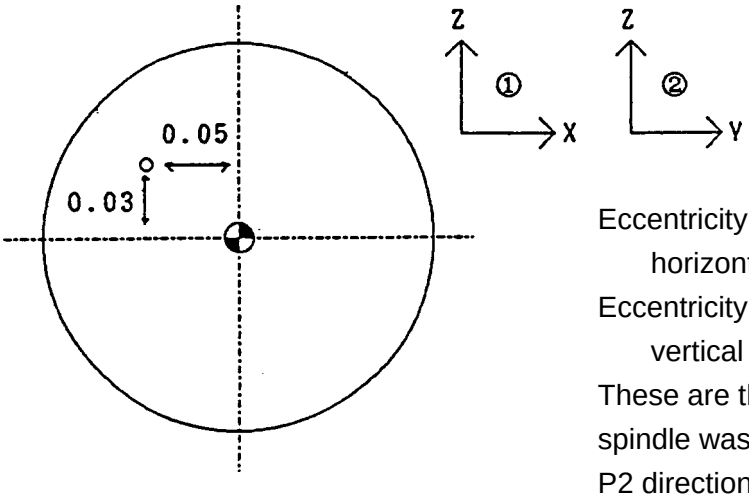
(2) Touch Probe Eccentricity Setting

If Spindle Was Attached Vertical in Vertical Machining Center or 5-Face Working Machine
(Example)



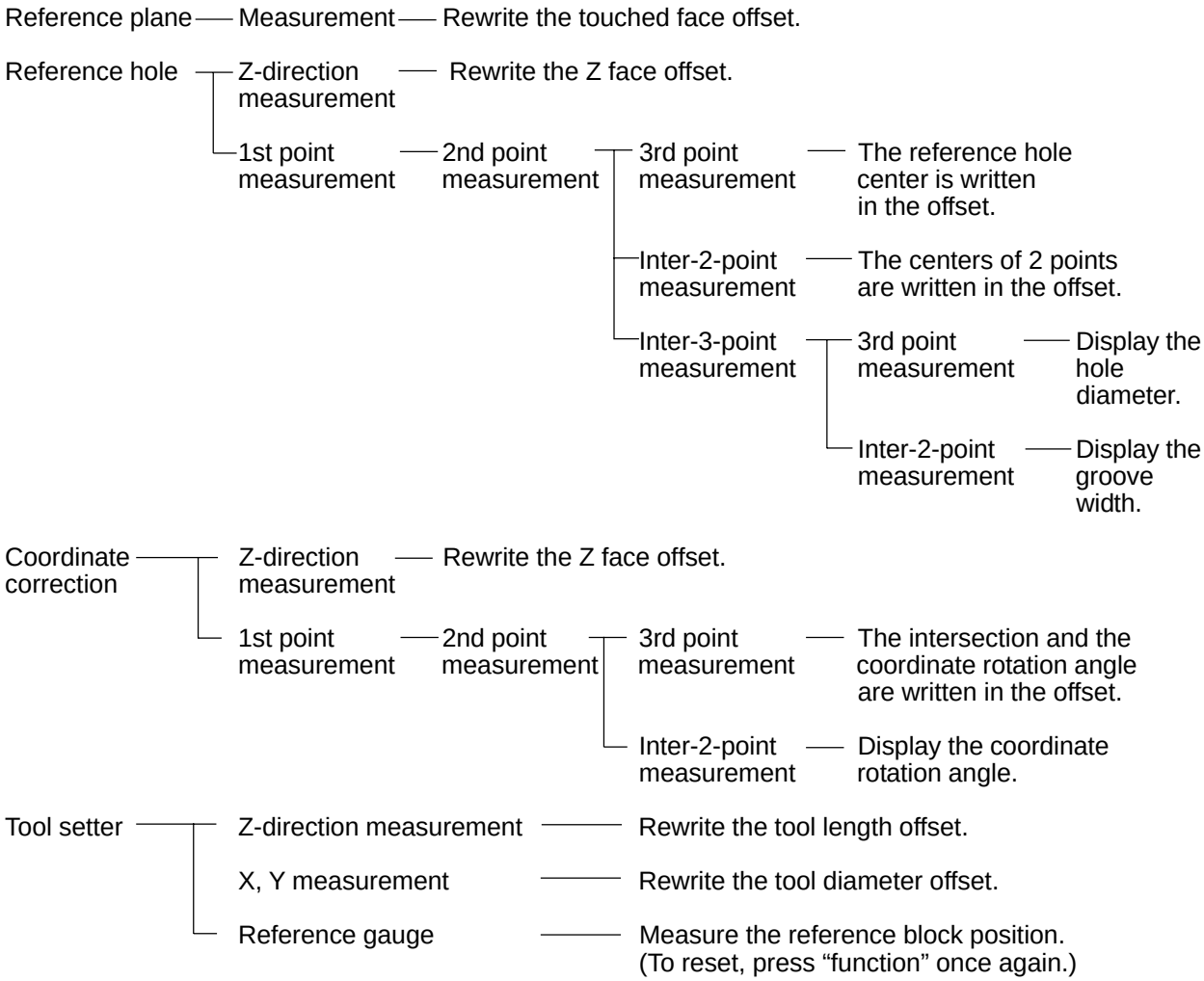
Eccentricity in X direction = 0.05
Eccentricity in Y direction = -0.07

If Spindle Was Attached Horizontal in 5-Face Working Machine
(Example)

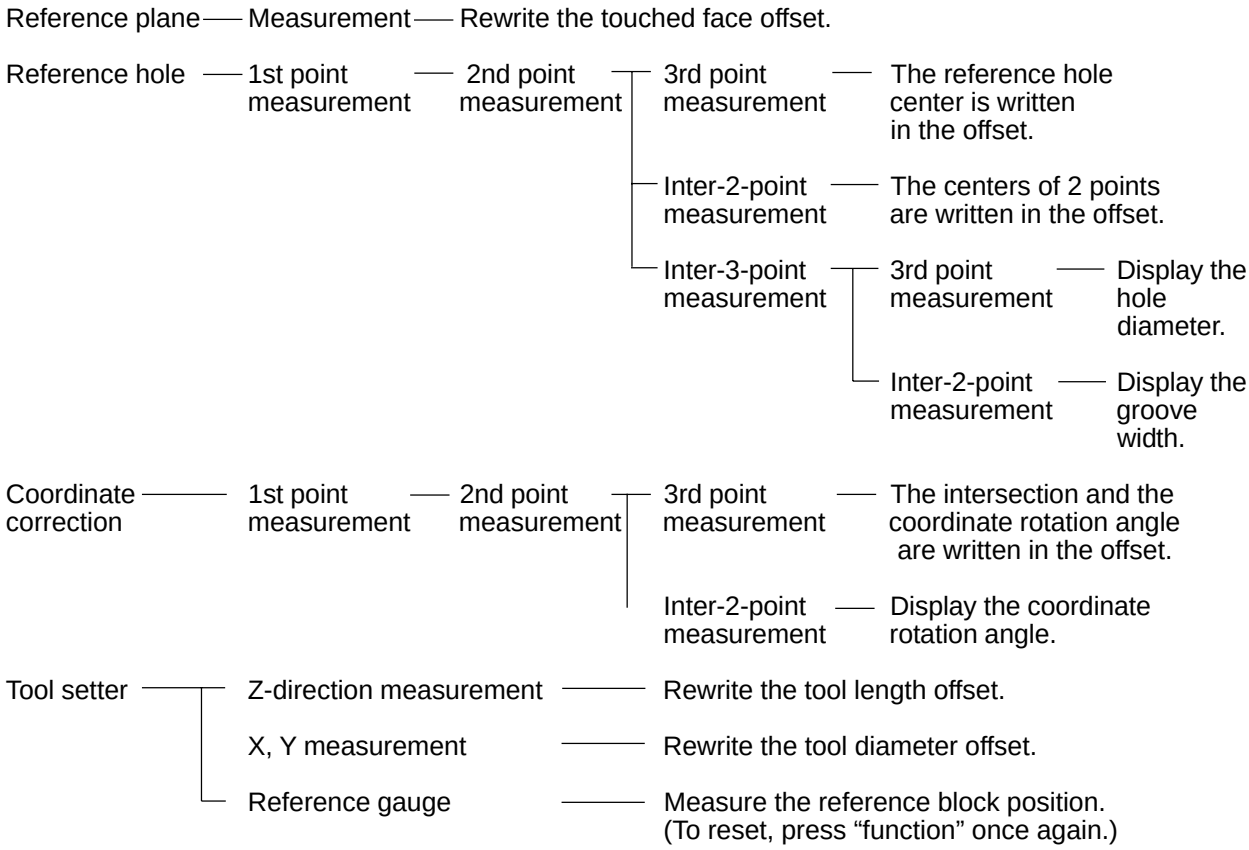


Eccentricity in
horizontal direction = -0.05
Eccentricity in
vertical direction = 0.03
These are the set values when the spindle was indexed in the (1) - P1, (2) - P2 direction.

(3) Operation Procedure Quick Reference Table (Vertical Machining Center)



(4) Operation Procedure Quick Reference Table (5-Face Working Machine)



II. Screen Operation

1. Screen Layout and Basic Operation
2. Operation of Overall Screen
3. Position
4. Work Coordinate (Offset)
5. Setting (Data)
6. Macro Variable
7. Plot
8. Plot Parameter
9. Tool (Offset)
10. Alarm Diagnosis
11. System
12. Program
13. Program List
14. Input/Output
15. Input/Output (With PC Card Used)

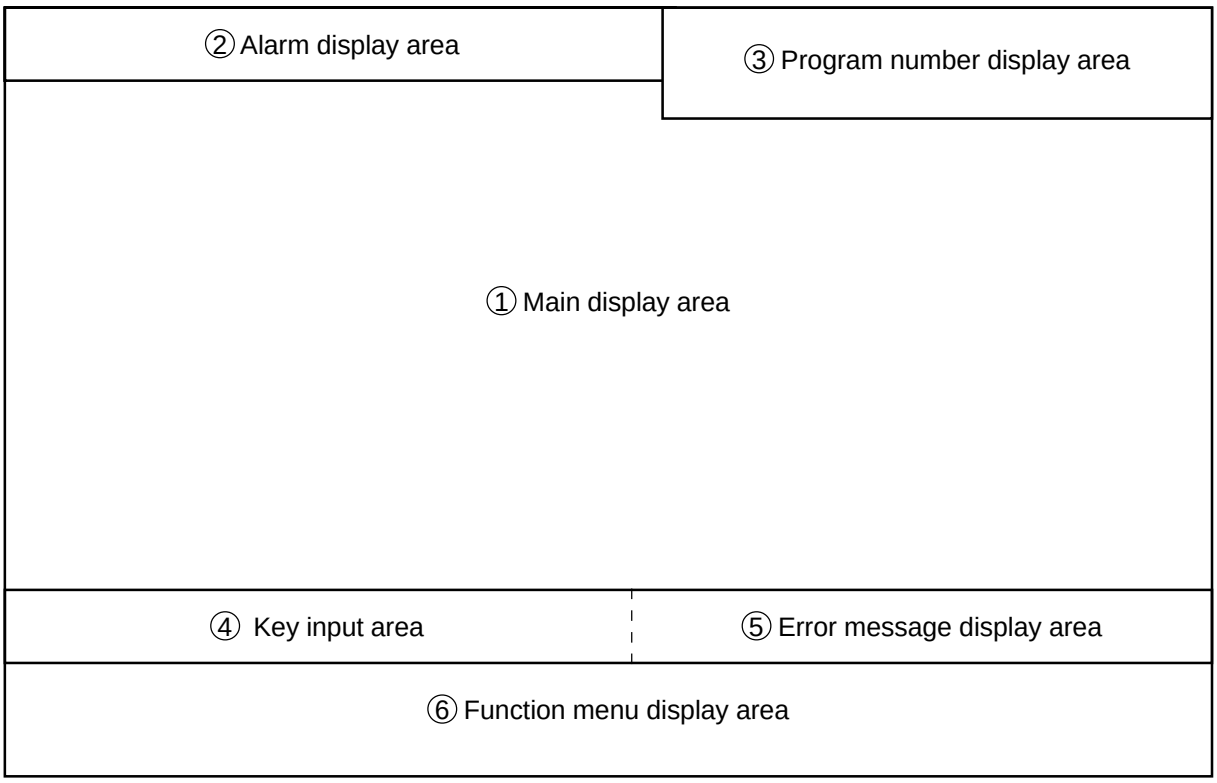
(Note) Some of the words bracketed in " " denote keys.
There are two types of CRT display screens; color and EL. When colors are specified, the color (EL) type is assumed.

1. Screen Layout and Basic Operation

One screen consists of;

① Main display area
② Alarm display area
③ Program No. display area
④ Key input area
⑤ Error message display area
⑥ Function menu display area

Fig. 1-1 shows the screen layout.



- In the overall screen, the tool data and command data are summarized.
- On certain screens, windows (small screens) may be overlapped and displayed, when a function menu key is pressed.
  (As a rule, if the same key is pressed again, the window disappears.)
- By setting the time to transition to the Good Night screen, the screen can be blanked (darkened) if you do not operate any keys within that time.

1-1 Main Display Area

Although the design differs from one screen to another, the following describes a basic operating method.

- [↑] / [↓] (Cursor move keys)

  Used to move the cursor up/down by one position. When the cursor is located at the top or bottom of the screen, the screen is scrolled by pressing them. The double-spread page such as parameters is not scrolled. They are also used to perform a search.

- [→] / [←] (Cursor move keys)

  Used to move the cursor to the right/left by one position.

- [PAGE↑] / [PAGE↓] (Page keys)

  Used to change pages when one page is not enough to display all. Here, the cursor refers to the character displayed in reverse video. Normally, it is surrounded by a yellow-frame guide line. The guide line is to indicate data connection; some screens do not have this. When the pages are changed, the cursor is moved to the beginning of the page.

In addition, the following key combinations are available for special operations.

- SHIFT + [↑] / SHIFT + [↓]

  Only the screen is scrolled without moving the cursor.

- SHIFT + [PAGE↑]

  Moves the page and cursor to the beginning of that screen.

- SHIFT + [PAGE↓]

  Moves the page and cursor to the end of that screen.

- SHIFT + [→] + [←]

  Moves the cursor character by character in editing the program.
  This allows you to edit character by character finely, not word by word.

1-2 Alarm Display Area

When an alarm occurs during execution of the program, a number and a simple message are displayed in one line. When multiple alarms occur simultaneously, only the first one is displayed. When you want to know its details, display the Alarm Diagnosis: screen.
When an alarm is displayed in white on the red ground (red blinking), it indicates a serious alarm. In that case, the machine stops and the Alarm Diagnosis screen is forced to apprer. (When you do not want to switching of the screen, set “1” in the bit 1 of the parameter 7000.)
When displayed in blue on the yellow ground, it is an alarm to awaken the operator. The machine does not stop and the screen is not switched.

1-3 Program Number Display Area

Displayed in this area are the program number and sequence number being currently executed, and multiplicity and repeat frequency at subprogram call time. The program number is 0 in case of MDI.

Example) O1000 → O2000 → O3000
L = 10 N1234

O3000, N1234 is being executed. O3000 repeats another 10 times. O3000 returns to O2000. O2000 returns to O1000.

1-4 Key Input Area

The entered characters are buffered in here once, and then, input.

Example) P: Absolute value = 1.234 (1.234 is entered)

Also, absolute/incremental value programming can be switched over and a number search can be performed by switching over the mode. To switch over the mode, press the following character keys when noting has been buffered.

- P (P: Absolute value =)
  Absolute value programming. The entered numerical value is input as it is.
- I (I: Incremental value =)
  Incremental value programming. The entered numerical value is added and input.
- N (N: Number =)
  Number search. The cursor is moved to the entered number.
  However, the cursor key is used for execution (not the INPUT key).
- J, K (J: X-axis rotation =) (K: Y-axis rotation =)
  Used only on the Work Coordinate (Offset) screen. Input is done with the rotation angle R taken into account.

In addition, the following functions are provided for input.

- CAN
  Deletes the character one before the cursor in the key input area.
- SHIFT + CAN
  Deletes the entire key input area.

1-5 Error Message

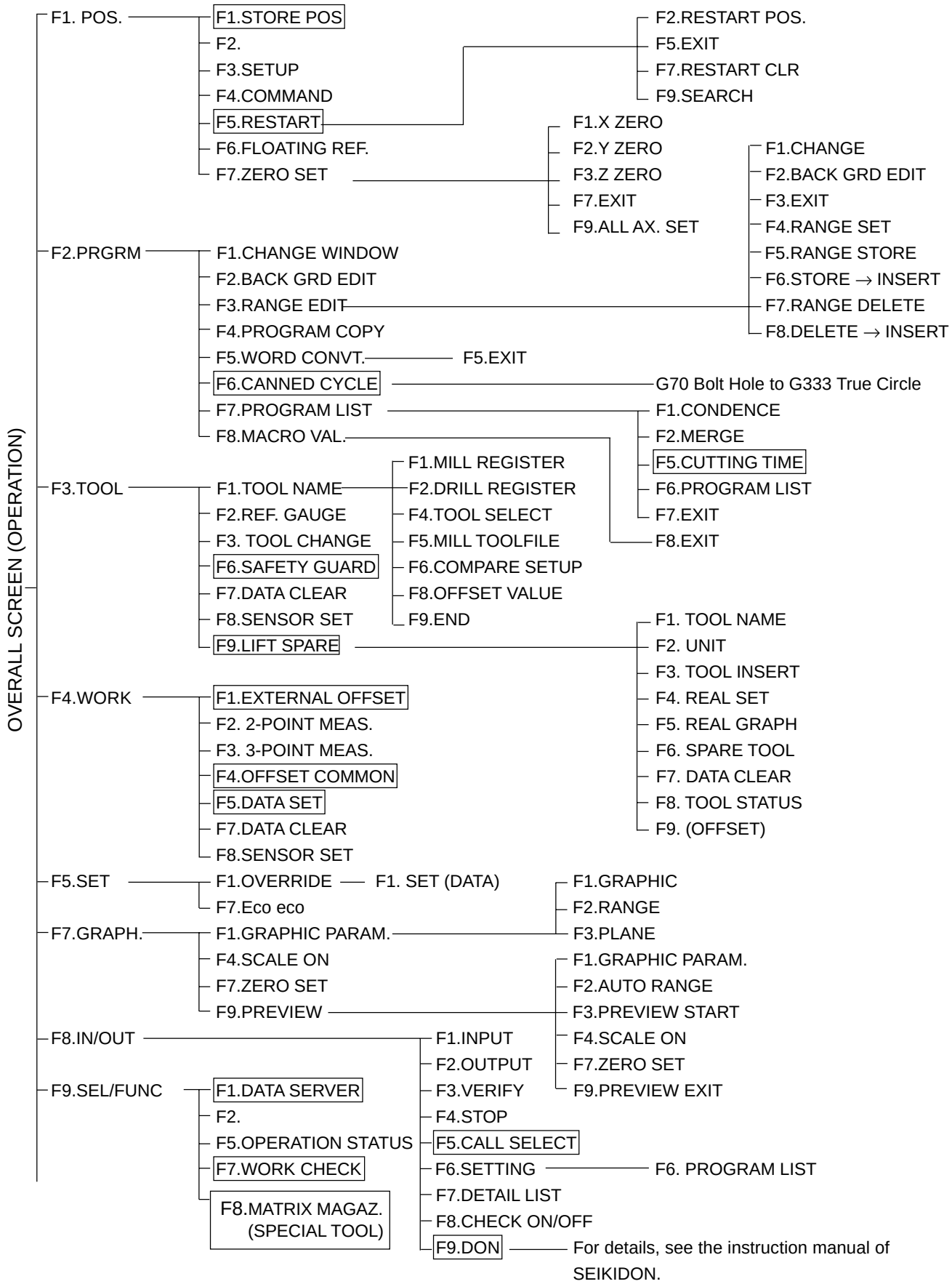
Different from the alarm, a message is displayed which notifies you of an operational mistake, etc.

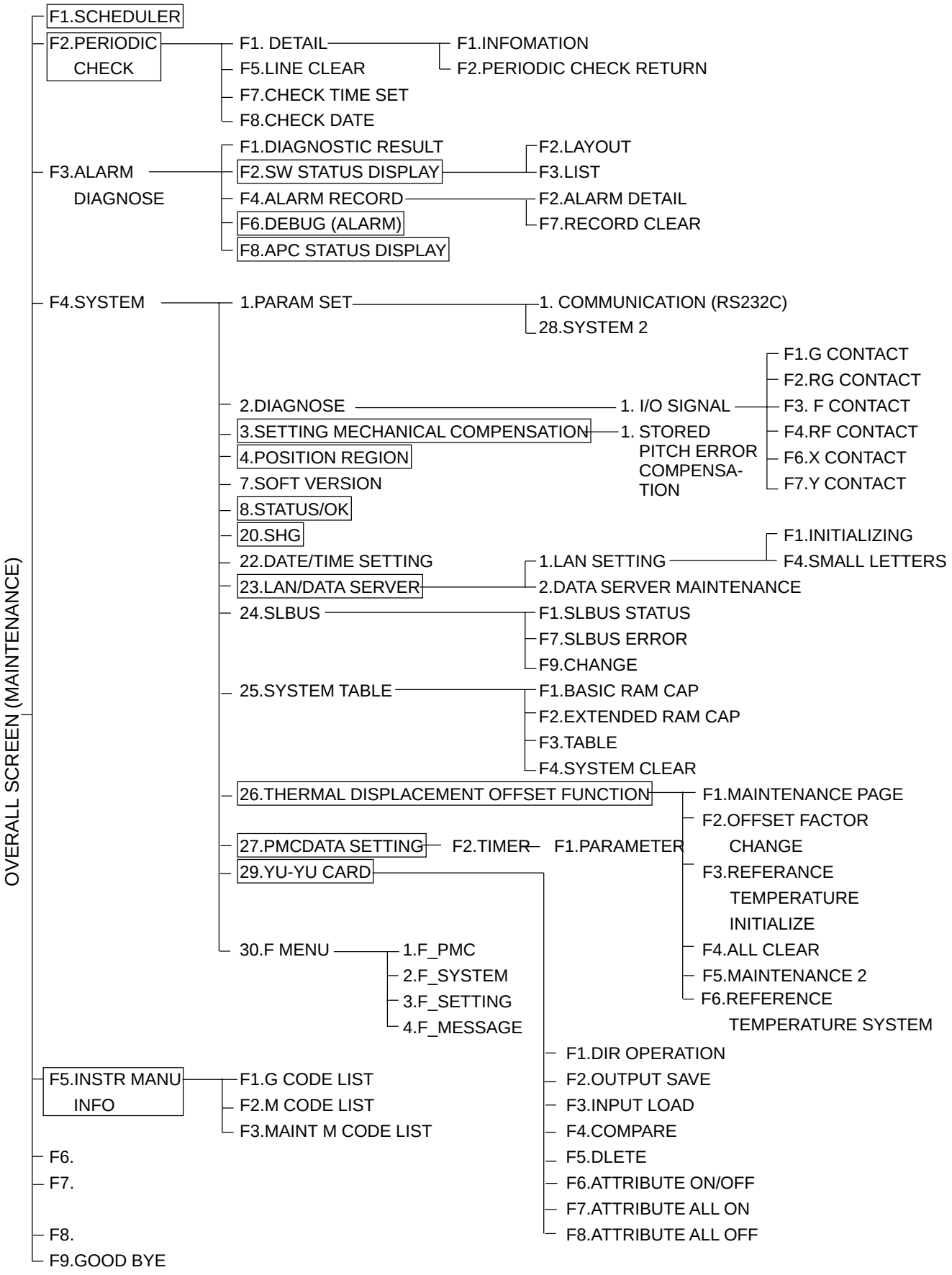
1-6 Function Menu

On the overall screen, a function key normally selects the screen written on the key. On the other screens, a (light) blue menu appears in the lower part of the screen in various manners according to the screen, so refer to the item of each screen.

- OPER/MAINTE key
  Pressing this key displays a green (black) menu. Using the functions in this menu, you can select some screens. Assuming the screen with the function keys is the operation side, this menu is the maintenance side, hence called the maintenance menu.

1-7 Tree Chart





(Note) □ denote an option.

2. Operation of Overall Screen

When the power is turned on or the RETURN key is pressed, the Overall screen (Fig. 2-0) is selected. It is very convenient because it displays many data required for operation. The data includes;

- Program
- Tool path plot
- Spindle tool number, standby tool number
- Next data of the spindle tool: Tool length compensation, cutter compensation, and tool life setting and use values
- H code and D code
- S, S % and F, F % values
- Remaining seconds of dwell
- Load meter of each axis
- Work coordinate system, remaining stroke
- G codes and M codes of the following groups
  01 (Positioning), 02 (Plane designation), 03 (Absolute/incremental), 07 (Cutter compensation), 08 (Tool length compensation), 09 (Canned cycle), 10 (Return point), 12 (Work coordinate system), and M codes of 5 groups
- Completion schedule time, machining time and tool lapping time
- Date and time

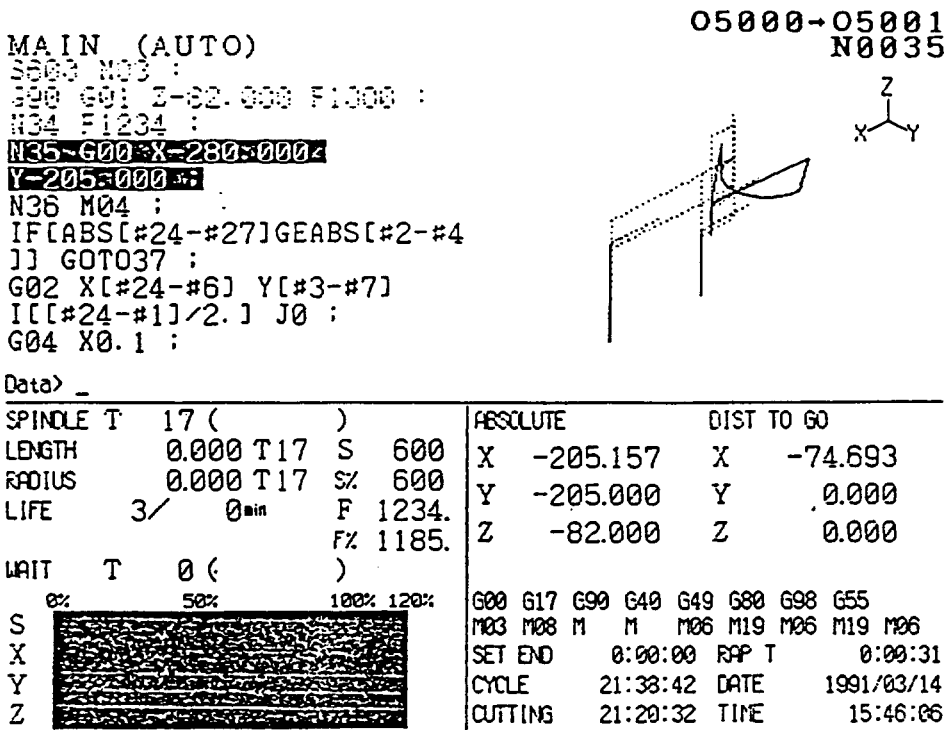


Fig. 2-0 Overall Screen

Also, the following is enabled in this screen.

- Program search
- Program editing

2-1 Program

This is located data the upper left part of the screen. It displays and edits the program. MDI input is also done here. However, expanded editing or background editing are not allowed.
When running the program, the display color changes as follows.

When running the program, the display.		Color (For monochrome)
Already executed block	:	Green
Block being executed or to be executed	:	Yellow reverse Video (light-colored reverse video with frame)
Pre-read block	:	Yellow (light-colored reverse video)
Previous block	:	White
Block to be executed next or part to be edited	:	White reverse video (reverse video)

The conditions for editing the program are given below.

- Write key has been pressed.
- In the edit mode and not in the automatic operation mode.

● For details of program operation, refer to section 12. Program.

● Here, one-fourth the program screen will be displayed in principle.

2-2 Tool Path Plot

This is located at the upper right part the screen and plots the tool path of the program being run. This is a miniature version of the Plot screen. Therefore, the plotting parameters are the same as those for the plot screen.
The Plot screen can be erased with the ORIGIN key.

2-3 Tool Data

This is located at the lower left part of the screen. It displays or inputs the spindle tool data. For the offset value, the tool offset profile plus wear are displayed. For the life value, the left is the use value and the right is the set value.

2-4 S, S % and F, F %

The spindle speed S, actual speed S %, feed rate F and actual feed rate F % are displayed to the right of the tool data. No input is done here.

2-5 Dwell

The remaining seconds of dwell operation are displayed above the standby tool number. When the remaining seconds come to 0, its display disappears.

2-6 Load Meter

The load condition of each axis is displayed under the tool data in the form of bar graph. When the load monitoring function is attached, the load data of each tool is used and a display color is changed so that you can tell the load condition at first glance.

Red : Overload or above
Yellow : Wear to overload
While : Below wear (reference load)

2-7 Position Data

The work coordinate system and remaining stroke and displayed at the lower right part of the screen from the work, relative, machine and remaining stroke coordinates.

2-8 G code and M code

Under the position data, 8 modal G codes are displayed in the following order.

- Group 01 (Positioning) : 00, 01, 02, 03
- Group 02 (Plane designation) : 17, 18, 19
- Group 03 (Absolute/incremental) : 90, 91
- Group 07 (Cutter compensation) : 40, 41, 42
- Group 08 (Tool length compensation) : 43, 44, 49
- Group 09 (Canned cycle) : 80, 81, 82
- Group 10 (Return point) : 98, 99
- Group 12 (Work coordinate) : 54, 55, 56

Under the G codes, M codes of 5 groups are displayed in the following order.

- Group 01 (Spindle)
  03, 04, 05, 13, 14, 15
- Group 02 (Coolant)
  07, 08, 09, 50
- Group 03 (Program stop)
  00, 01, 02, 30
- Group 04 (Monitoring unit)
  55, 56
- Group 05 (Other)
  Up to 5 past M codes of other than Groups 01 - 04 are displayed.

2-9 Time Measurement

For times are measured in the lower right corner of the screen. They can be measured within a range of 10,000 hours. To make input here, push F9/SEL/FUNC and, further, F5/OPERATION STATUS to display the operation status page (Fig. 2-9).
Shift the cursor to the appropriate place.

Example) (hours/minutes/seconds) = 0 (Set to 0)
(hours/minutes/seconds) = 1 (1 : 00 : 00 is entered)
(hours/minutes/seconds) = 1/2 (1 : 02 : 00 is entered)
(hours/minutes/seconds) = 1/2/3 (1 : 02 : 03 is entered)

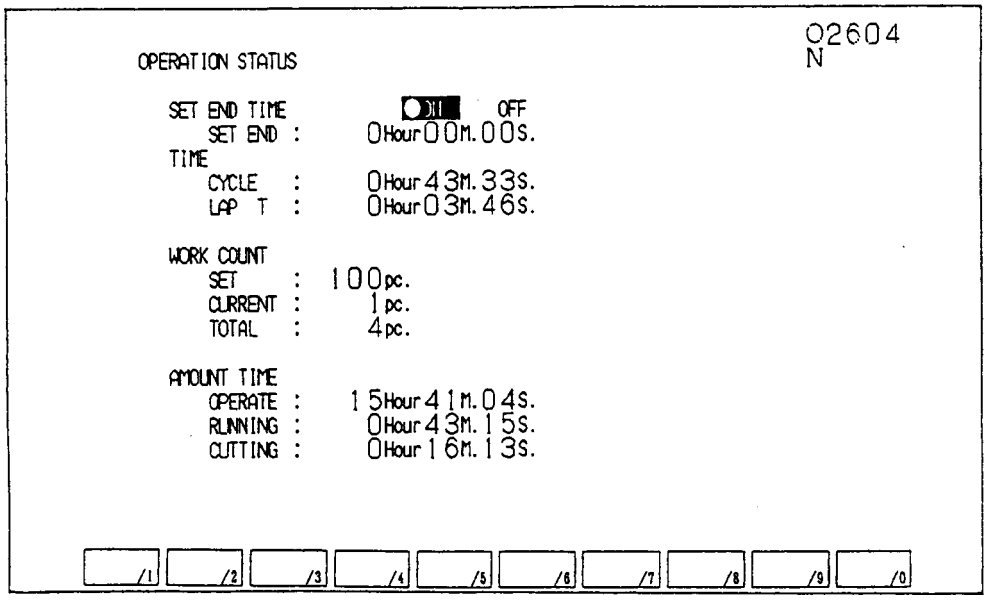


Fig. 2-9 OPERATION STATUS

(1) Scheduled ending
Notice of ending is made effective when End Notice on the page is held “valid”.
The scheduled ending time is equal to time length from the auto operation start to the ending notice.
With a scheduled ending time of a program having been input by an operator, call light functions when working time has reached the scheduled ending time.

(2) Working time
Time for working is summed up. (time during which the start lamp is held ON.)

(3) Lap T
Lap time among tools is measured. This is automatically set to 0 on tool changing. No measurement is conducted while NC is held in standby state.

(4) Work Count
NC, against M12 command, counts up the number of cutting and the total number of works cut. NC, when the set number is not equal to 0 and the number of cutting has exceeded the set number, outputs the work count correspondence signal into PC.

2-10 Date and Time

Date (calendar year) and time (13:00 for 1 p.m.) are displayed by the built-in clock. Do not change setting unnecessarily as it is used for periodic checking. Should it become incorrect due to any reason, be sure to reset properly.

To display Date/Time Set screen, push OPER/MAINTE , F4/SYSTEM , 2 , 2 , and INPUT . Move the cursor to Date or Time and perform setting.

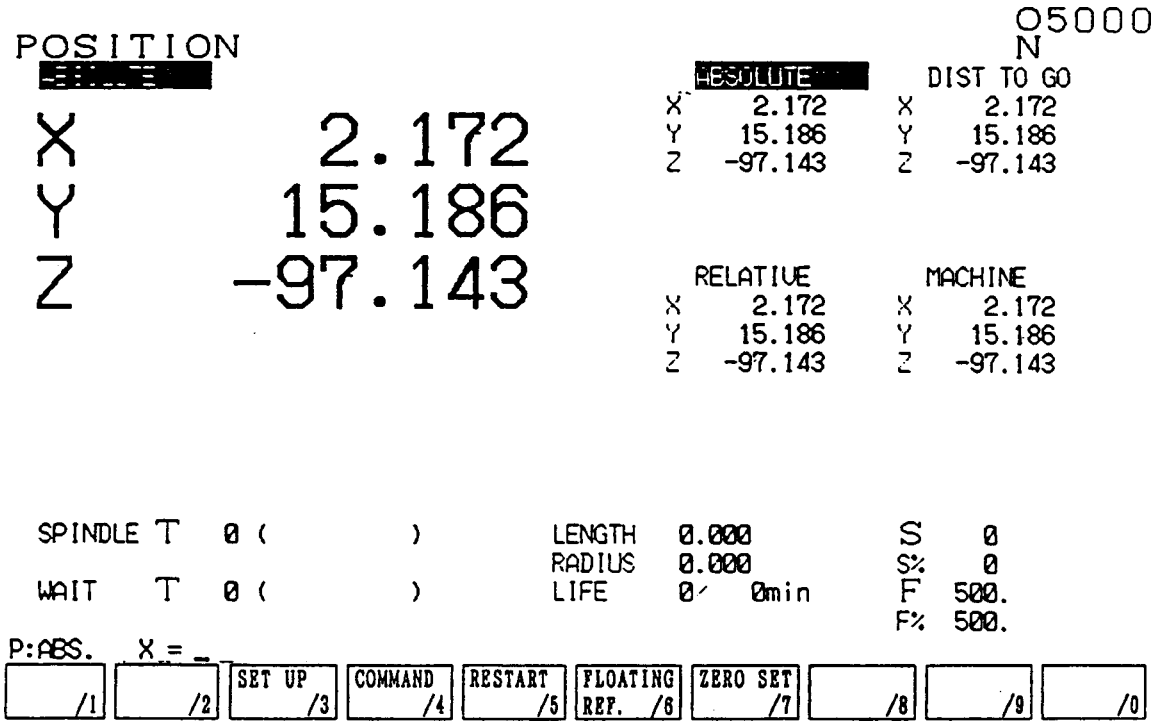
3. Position

Pressing the [F1/POSITION] key display the Position screen (Fig. 3-0).

This screen displays and allows you to rewrite those related to the position.

- Display of the work coordinate system, remaining stroke, relative coordinate system and machine coordinate system
- Rewriting of the relative coordinate system
- Display of the spindle tool number and standby tool number
- Display of the following data on the spindle tool
  Tool length compensation, cutter compensation, tool life setting and use value
- Display of the following data on the spindle tool
  Tool length compensation, cutter compensation, tool life setting and use value
- Remaining seconds of dwell
- Setup work

The left half of the screen shows an enlarged display of four right coordinate systems. You can use the Page keys, [PAGE↑] and [PAGE↓], to select any desired one. When an axis is changed by parameter setting, the work coordinate system where the axis is changed, and an axis address having a remaining stroke is displayed in reverse video.



<Fig. 3-0 Position Screen>

3-1 Rewriting the Relative Coordinate System

Of four coordinate systems, only the relative coordinate system can be rewritten. To set to zero, press [F7/ZERO SET] to change the function. Press the function for the axis which you want to set to zero. A value for that axis is set to zero. When you want to set all the axes to zero, press [F9/ALL ZERO SET].

To end zero set, press [F7/FUNCTION RETURN]. To input a numerical value, press a desired address key to display an input area. Input a numerical value and press [INPUT]. You can input either absolute/incremental value.

<Example> P: Absolute value X = 1.234 [INPUT] (1.234 is input to X)
I: Incremental value Z = 10. [INPUT] (10.0 is additionally input to Z)

3-2 Setup Work

Pressing the [F3/SET UP] allows you to input the data to the setup side.

For spindle speed, spindle positioning and table index selecting operation, use the cursor keys (↑ ↓) and input the data.

However, multiple data cannot be input at one time. To execute the input data, select the MDI mode, input the data, and press [INPUT].

You will be inquired, “OK ? Y-YES N-NO.” Pressing [Y] executes the input data. Pressing the [F3/SET UP] returns the cursor to the position side.

3-3 Command Value Display

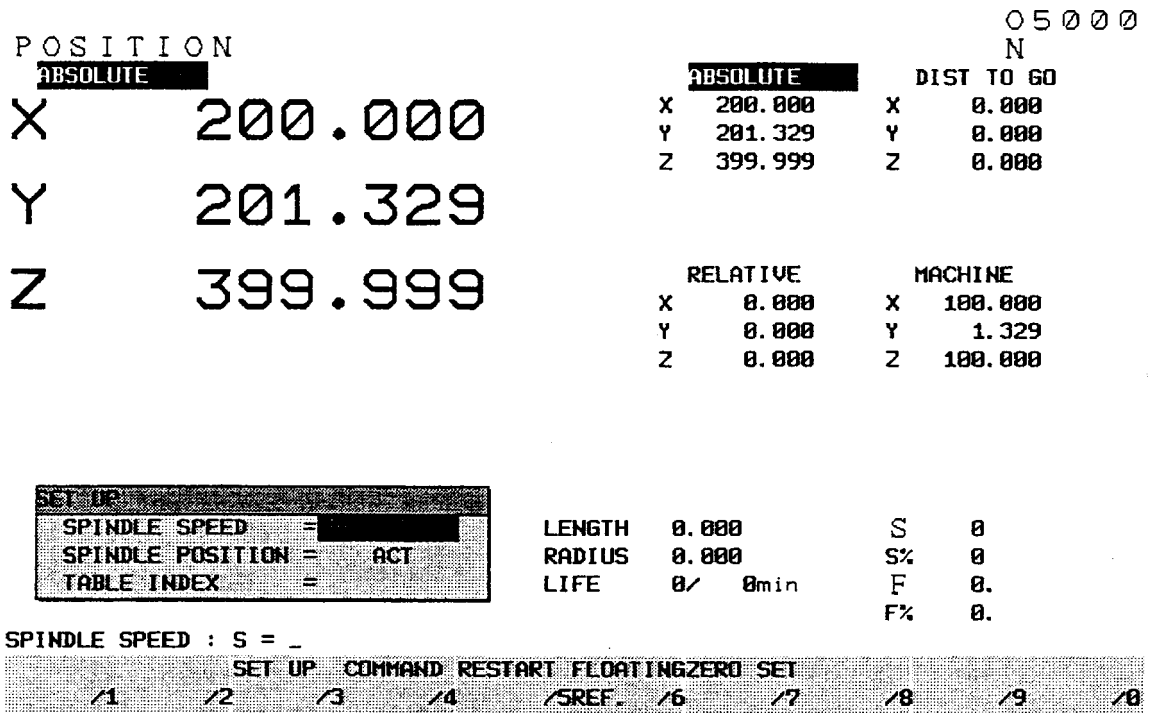
Pressing the [F4/COMMAND] displays the data specified by the program or MDI operation (Fig. 3-3).

3-4 Restart (Option)

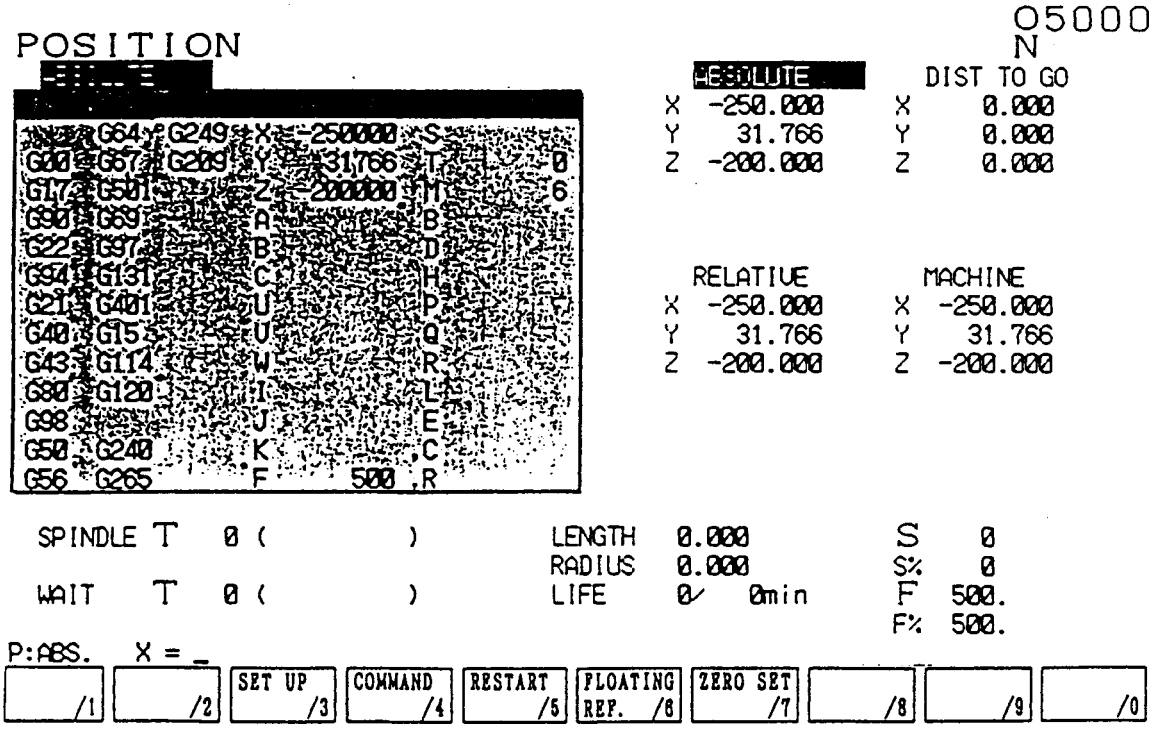
Pressing the [F5/RESTART] displays the screen required for program restart or block restart operation (Fig. 3-4).

3-5 Floating Reference

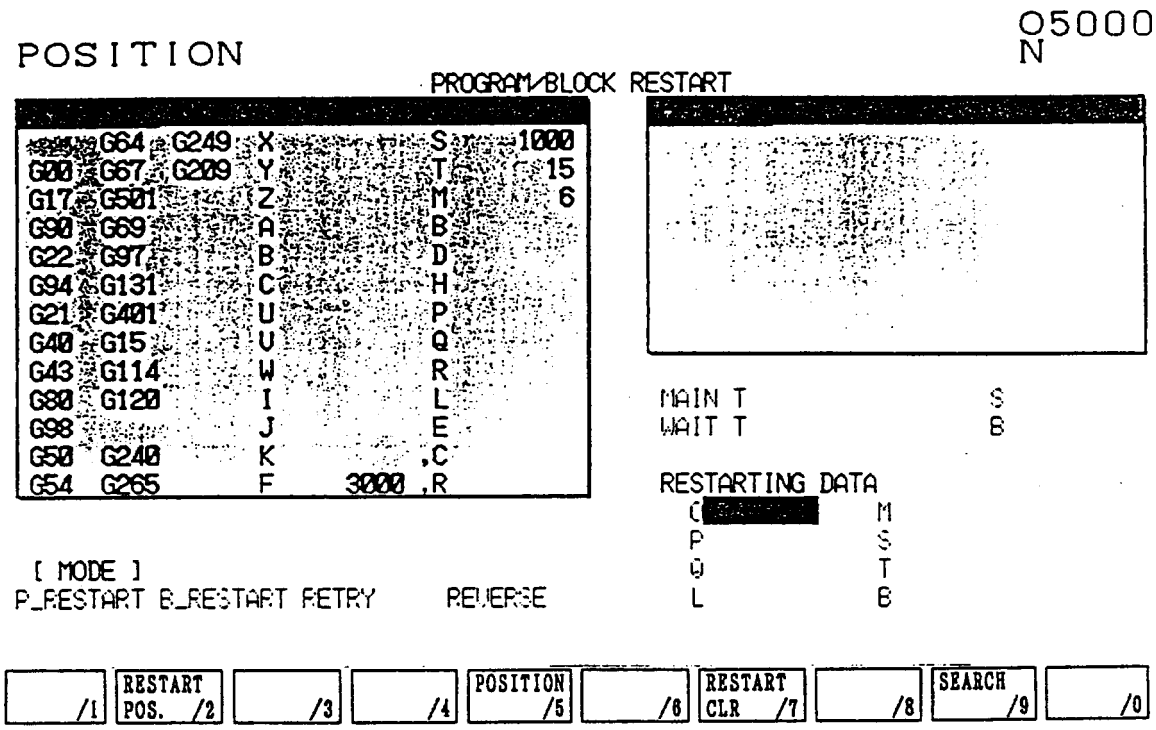
If you press [F6/FLOATING REF.], you will be asked “Do you set the floating reference point? Y-Yes N-No”, so press the “Y” key to set one and the “N” key not to set one. (Figure 3-5)



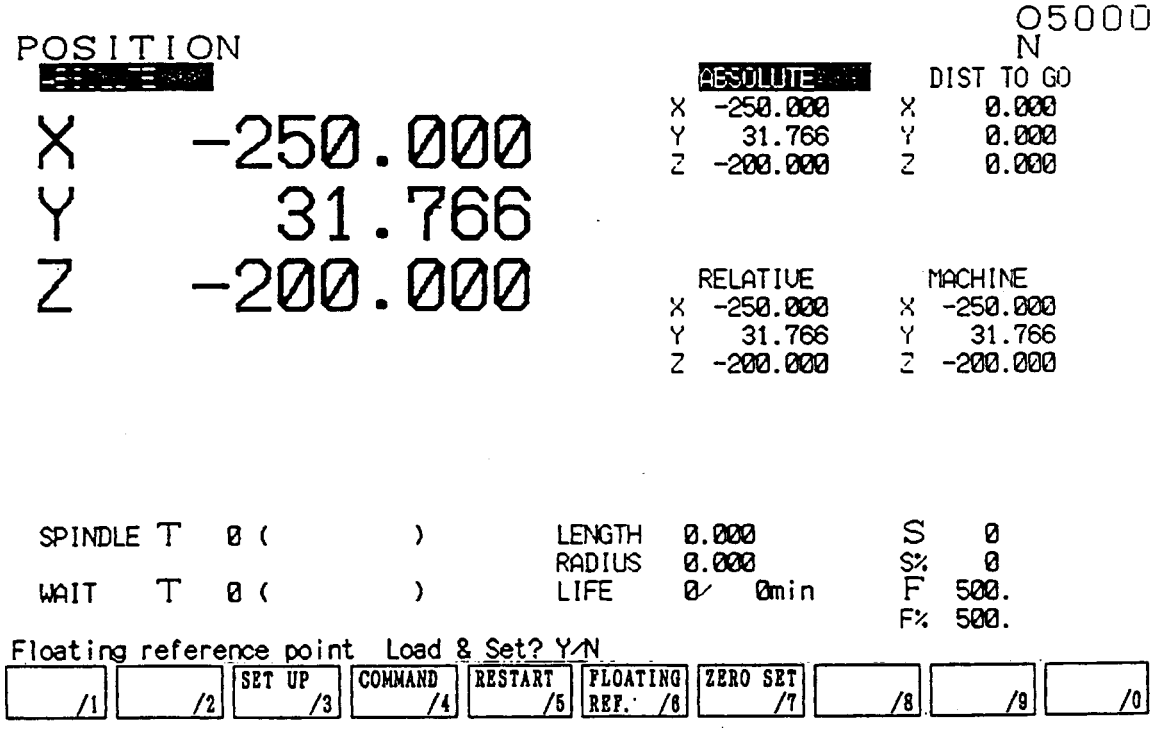
<Fig. 3-2 Setup Work>



<Fig. 3-3 Command Value Display>



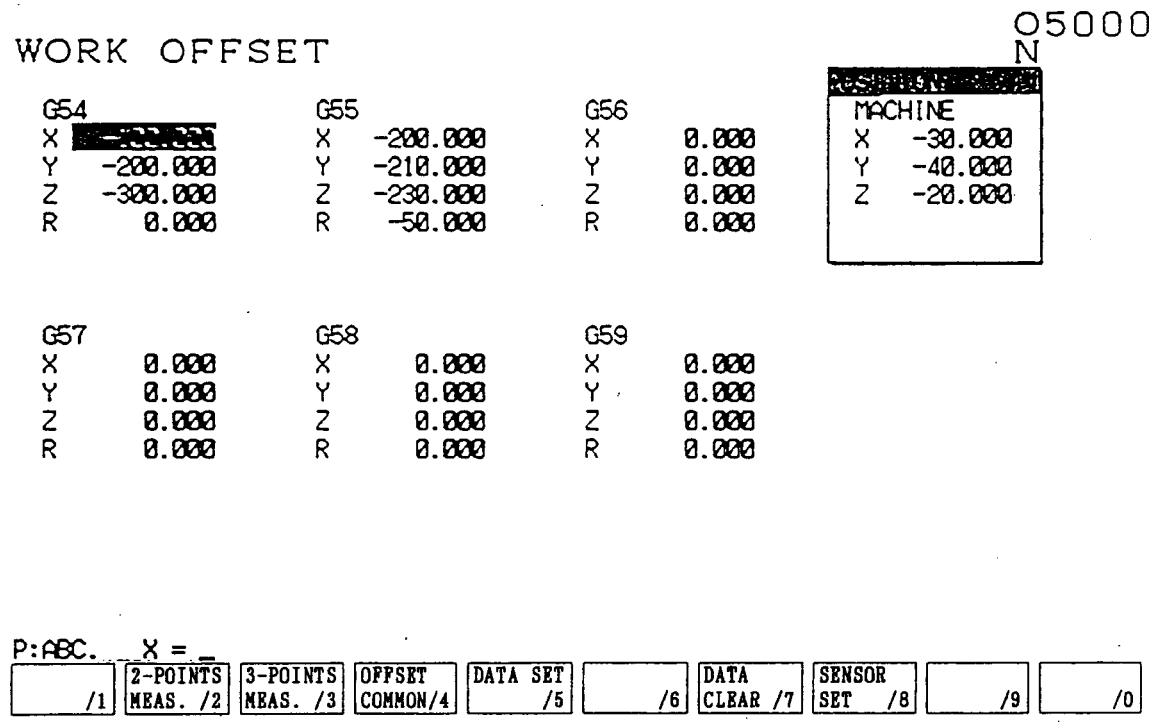
<Fig. 3-4 Restart>



<Fig. 3-5 Floating Reference>

4. Work Coordinate (Offset)

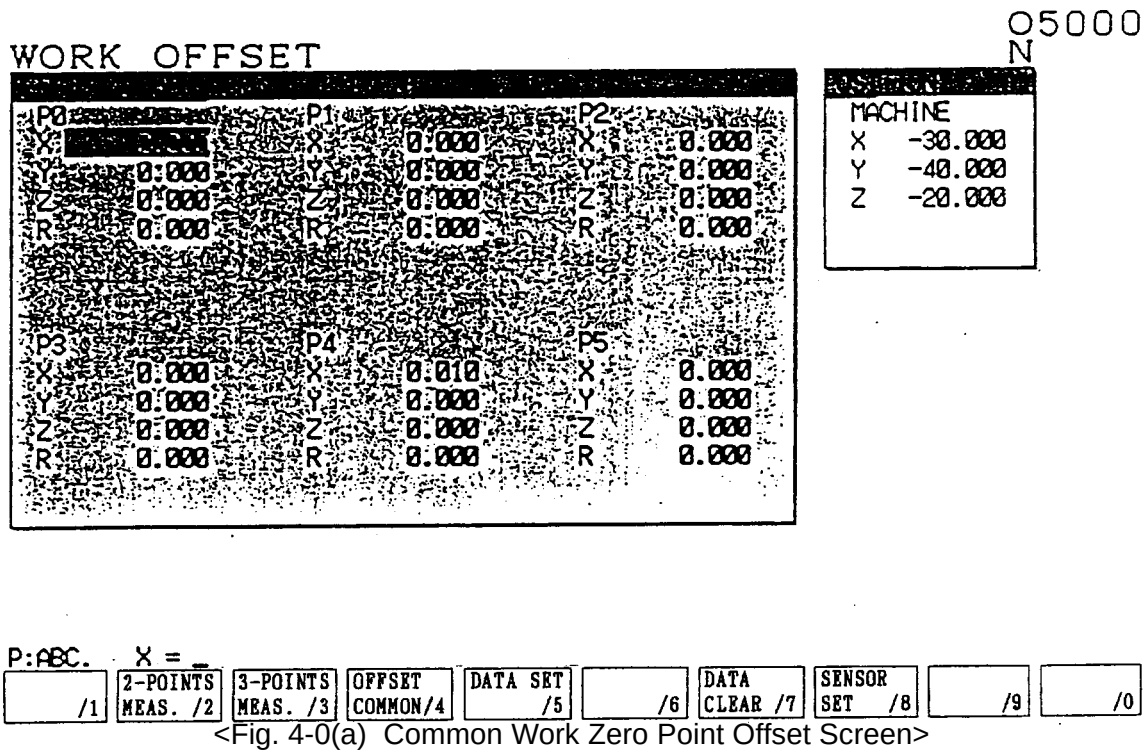
Pressing the [F4/WORK] key displays the Work Coordinate Offset Data screen (Fig. 4-0). Pressing the key twice displays the Common Work Zero Point Offset screen (Fig. 4-0(a)).



<Fig. 4-0 Work Coordinate (offset)>

It allows you to display and set the work offset value (G54-G59, G540-G599) and use the work setter.

- Display and setting of G54-G59
- Display and setting of G540-G599
- Display and setting of the external offset
- Work setter
- Display of the machine coordinate system



<Fig. 4-0(a) Common Work Zero Point Offset Screen>

4-1 Data Display and Setting

Six offset data of G54 through G59 are displayed and input. Optionally, the number of offset data can be expanded up to 66. When this is done, however, change pages with the page key (or) because all of them cannot be displayed in one page. The external offset and machine coordinate system exist in all pages.

To input the data, move the cursor with the cursor keys (↑ , ↓ , → , ←) or address keys. You can input the data, adding the rotating angle R as well as an absolute/incremental value. Input is allowed only when the WRITE switch 1 is turned on.

Example) P: Absolute value X = 1.234 INPUT
(1.234 is input to X)
I: Incremental value Z = 1.234 INPUT
(1.234 is added and input to Z)
J: X-axis rotation = 10 INPUT
(10 * COS[R] is added and input to X)
(10 * SIN[R] is added and input to Y)
K: Y-axis rotation = 10 INPUT
(10 * SIN[R] is subtracted and input to X)
(10 * COS[R] is added and input to Y)

4-2 Function Menu

(1) [F3/MEASURMENT AMONG 3 POINTS], [F2/MEASURMENT BETWEEN 2 POINTS]
Used for the work setter.

(2) [F7/DATA DELETION]
When you newly start, this is used to delete all the data to 0 at once.
Pressing this menu asks you at the center of the screen whether you want to delete all the data or the data of the page currently displayed. Select either of them with the cursor key ([↑] or [↓]) and press the INPUT key. Finally, the message “OK?” appears in the key input area for safety confirmation. Press the Y key.

(3) [F8/SENSOR SETTING]
Set the sensor for the work setter.

4-3 Work Setter

Refer to “Operation Related to W Setter”.

4-4 Number Search

The page and cursor can be moved at one time by searching the G code number. A search can be performed by pressing the cursor key ([↑] [↓] [→] [←]) after inputting N and a subsequent numerical value.

Example) N: No. G = 54 [↓] (Moves to G54)
N: No. G = 561 [↓] (Moves to G561)

4-5 Work Coordinates Setting Function

(1) Function : Sets the work coordinates based on the machine coordinates in the current position, by operating the function keys.

(2) Example of use : When a machining master hole is too small to measure with a measuring probe, attach a dial gauge to the spindle head and measure a machining master hole position by manual operation.
When this is done, display the Work screen, select a work coordinate number, and set the work coordinates with the function keys, leaving the machine in the very position.

(3) Operation

① Move the X-, Y-, and Z-axes of the machine to a position corresponding to the work zero point.

② In the Overall screen, press the function key [F4/Work Coordinates].

③ With the frame cursor, select the work coordinates to be set.

④ Select the axis for which the coordinates are to be set by;
・Moving the cursor.
・Entering the X/Y/Z key.

⑤ Press the function key [F5/SET WORK COORDINATES].
For the cursor X →Set the X-axis work coordinates ? Y: Yes, N: No
For the cursor Y →Set the Y-axis work coordinates ? Y: Yes, N: No
For the cursor Z → Set the Z-axis work coordinates ? Y: Yes, N: No

⑥ Pressing the Y key automatically sets the relevant work coordinates.

(4) Parameter

When using external work zero point offset as thermal displacement data on the machine equipped with thermal displacement offset function, set as:

No.6242, #0 = 1 (External work zero point offset is included in calculation.)

4-6 Setting the External Work Zero Point Offset

The entire work coordinate system can be shifted by a set value by setting an external work zero point offset amount. Pressing F1/EXT OFFSET opens the window shown in the right fugure. Use the cursor to select the axis you want to set, and input an offset value.

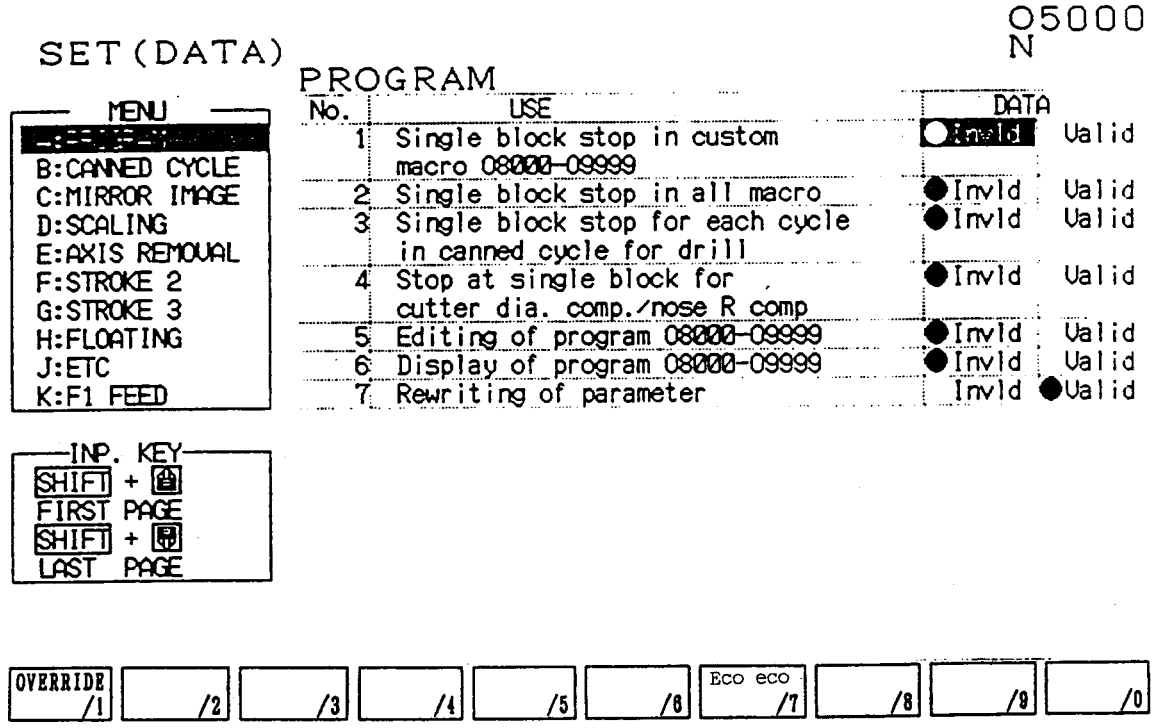
(Note) This is not useable when thermal displacement offset function has been made effective in external work zero point offset.

External Offset	
X	0.000
Y	0.000
Z	0.000
R	0.000

5. Set (Data)

If you press the “F5/set” key, the setting (data) screen (Figure 5-0) will be displayed. Frequently used parameters are collected by item and application here for easy setting. An optional item is not displayed if it is not attached.

The screen runs over several pages, therefore, scroll by using the page keys [P] [P] or by select the desired screen by pressing the alphabet key corresponding to the screens on the menu at the left side of the screen. Number search is also possible.



<Fig. 5-0 SET (Data)>

5-1 Display and Setting

For a selection such as “Yes/No”, the selected one is marked with ● .

When you want to alter, set the cursor with the cursor key ([→] or [←]) and press the INPUT key. When the cursor key ([↑] or [↓]) is pressed, the one marked with ● works for your safety. For your information, the HITACHI SEIKI’s standard is the left one.

For altering the numerical data, start from the key input area.

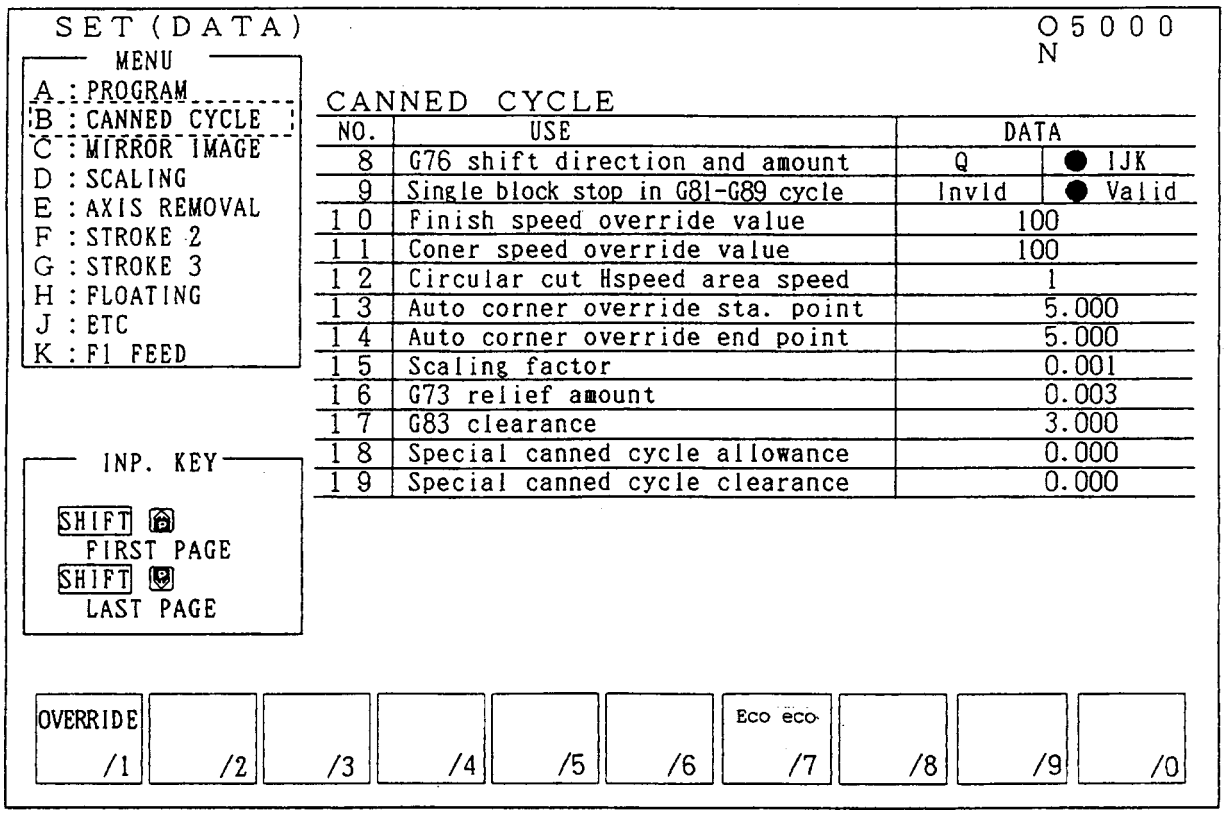
Naturally, calculation and absolute/incremental programming are also enabled.

Different from the Parameter screen, the Set (Data) screen does not have input limitations such as WRITE key. The operator should be fully careful to make alterration.

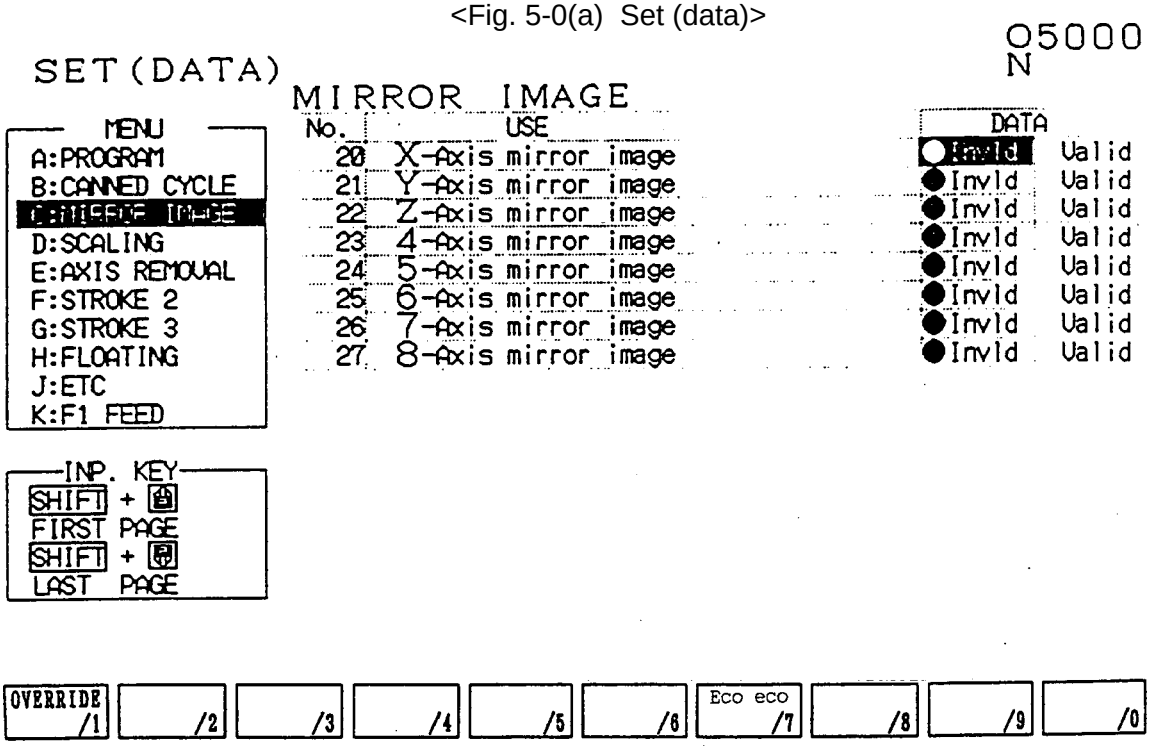
5-2 Number Search

Since all the pages are numbered serially, the page and cursor can be moved at one time by searching that number. A search can be performed by pressing the cursor key ([↑] [↓] [→] [←]) after inputting N and a subsequent numerical value.

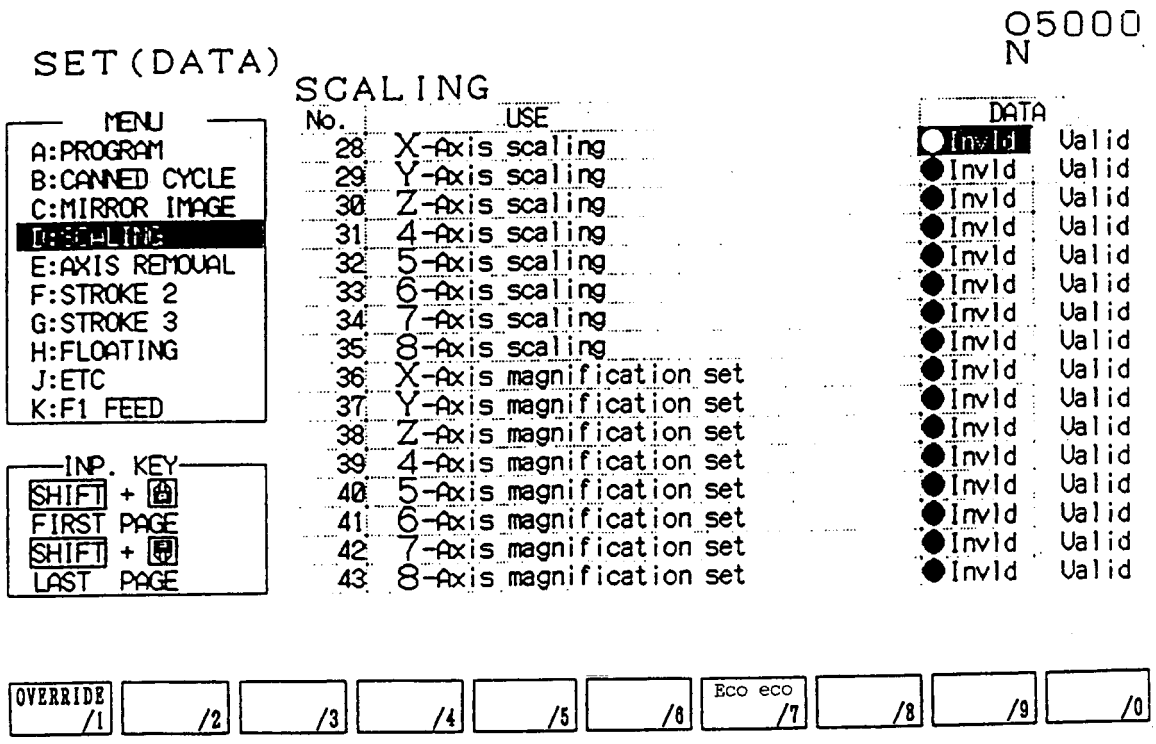
Example) N: No. = 12 [↓] (Moves to No.12)



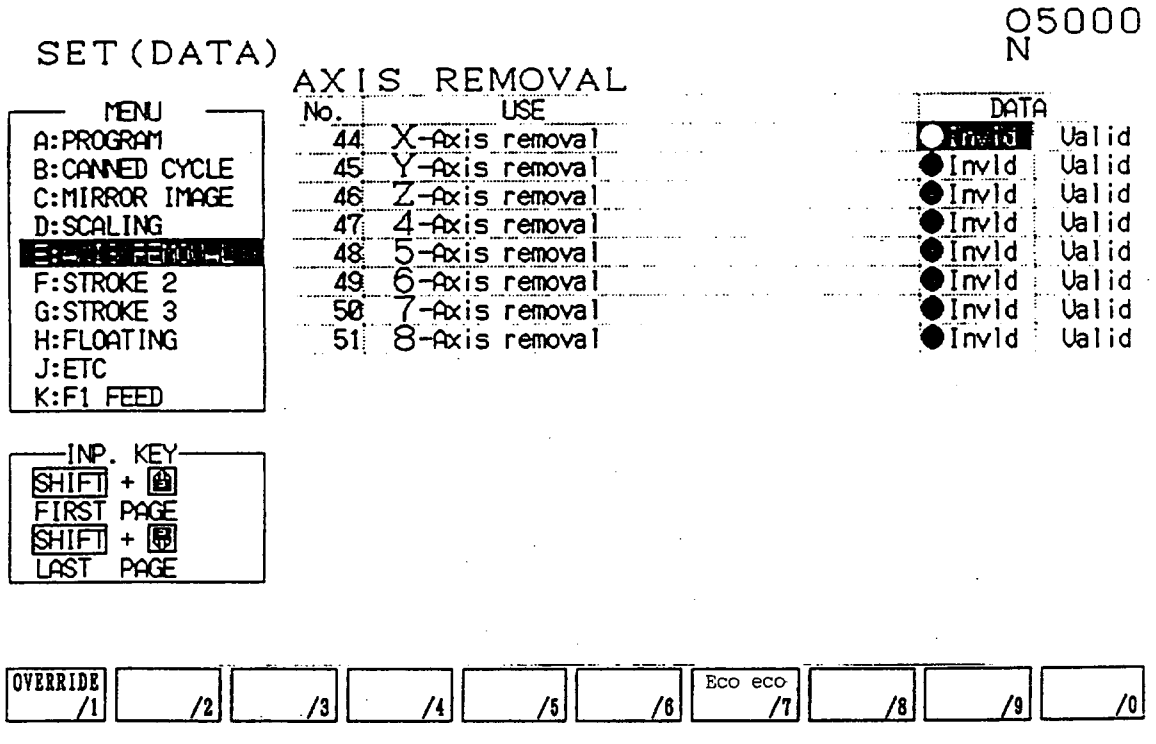
<Fig. 5-0(a) Set (data)>



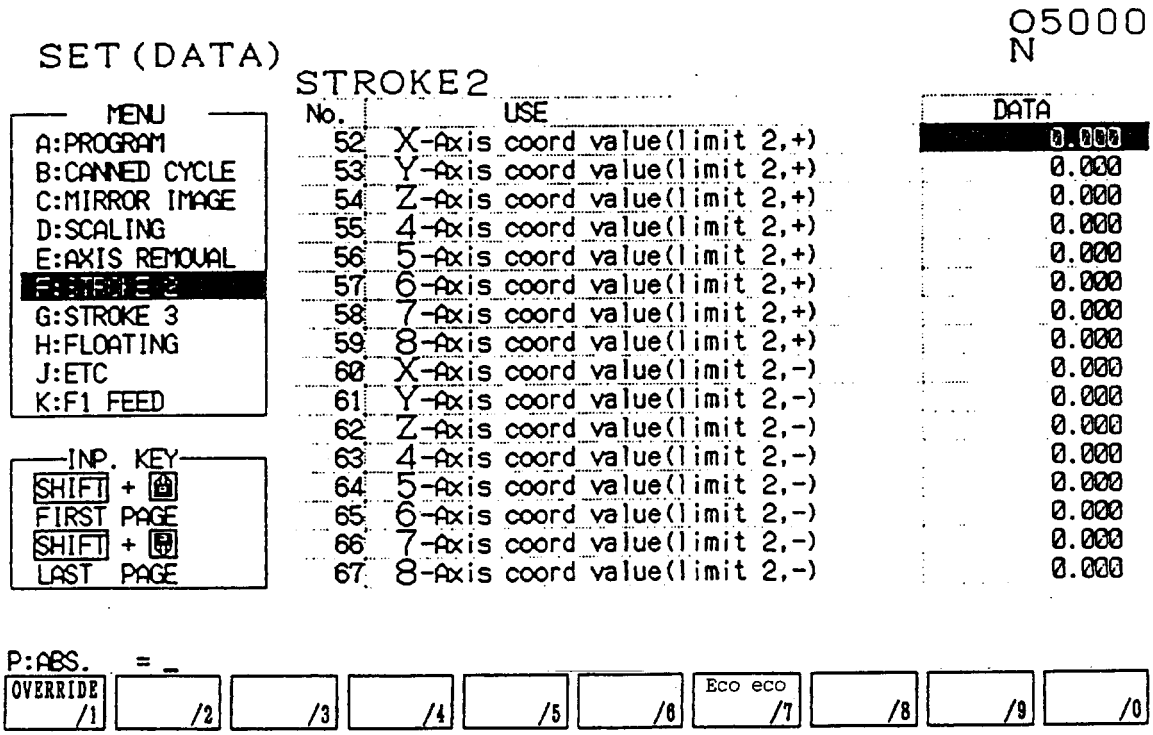
<Fig. 5-0(b) Set (data)>



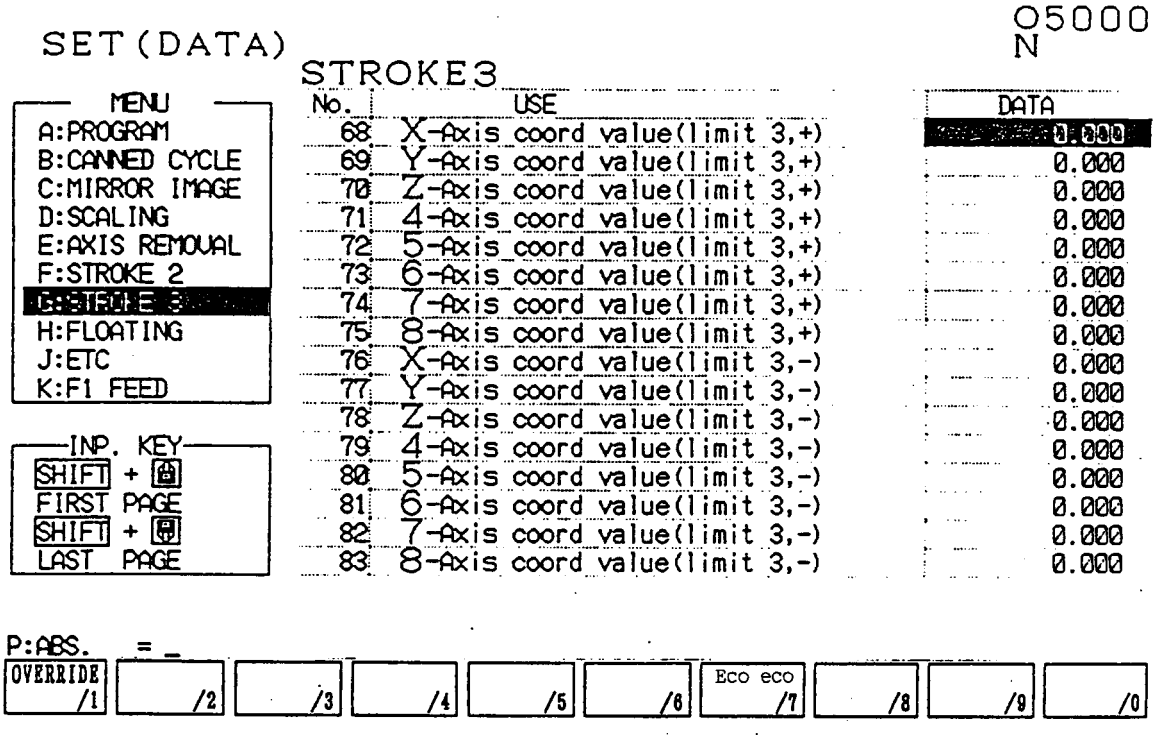
<Fig. 5-0(c) Set (data)>



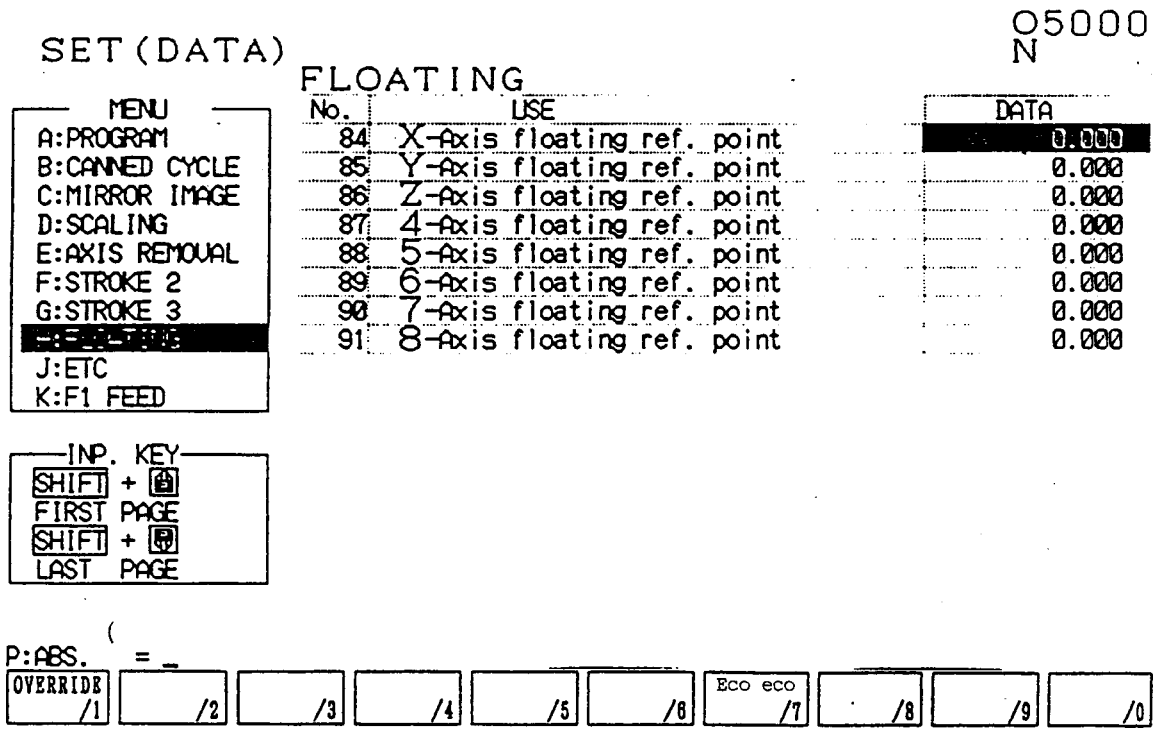
<Fig. 5-0(d) Set (data)>



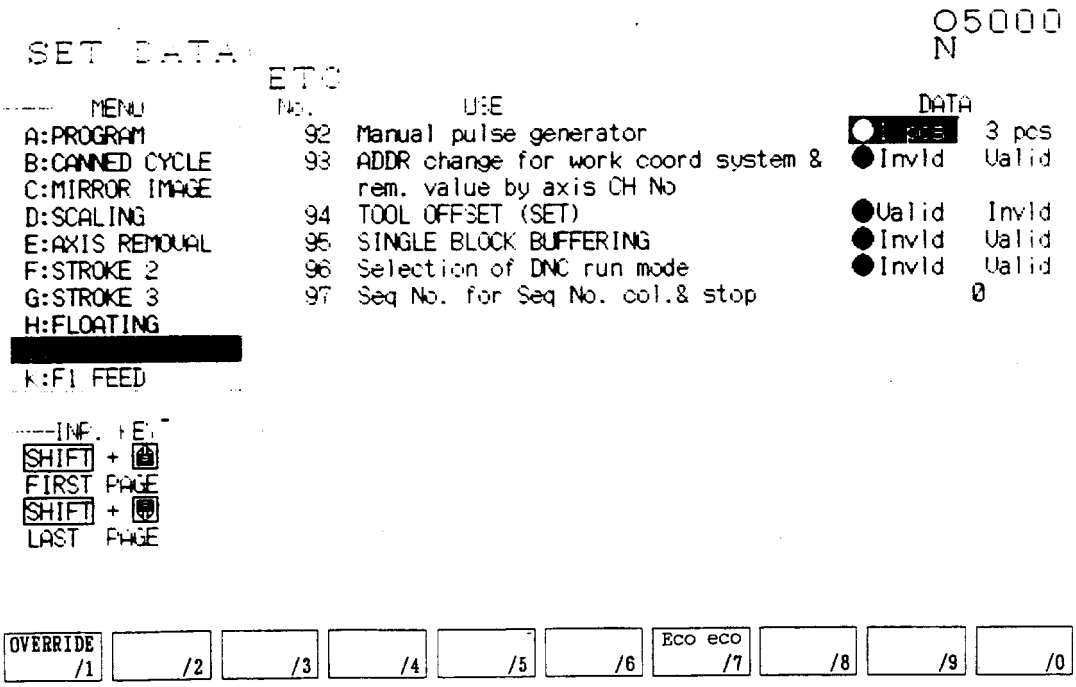
<Fig. 5-0(e) Set (data)>



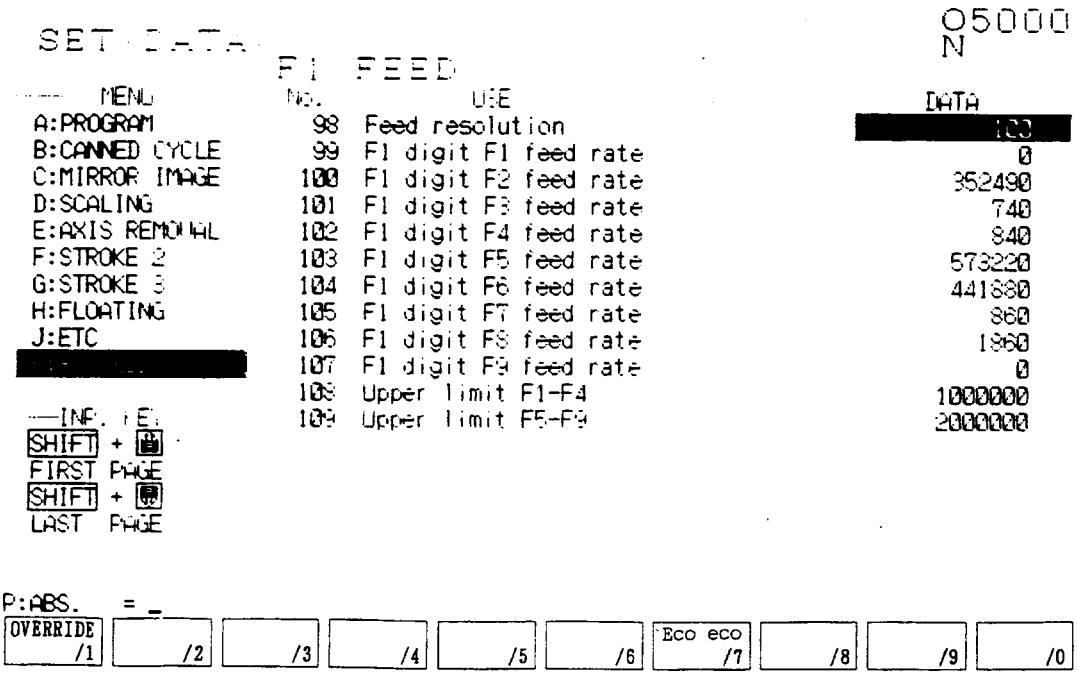
<Fig. 5-0(f) Set (data)>



<Fig. 5-0(g) Set (data)>



<Fig. 5-0(h) Set (data)>



<Fig. 5-0(i) Set (data)>

5-3 Automatic Override

A function to memorize the override according to the actual machining condition for the cutting feed rate and the spindle's rotational frequency specified in the machining program when test cutting and to make it be reflected automatically in the machining thereafter.

(1) Override Memory Display

If you press "F1/override" on the setting screen ("F5/set"), the override memory screen will appear overlapping the setting screen. (Figure 5-3)

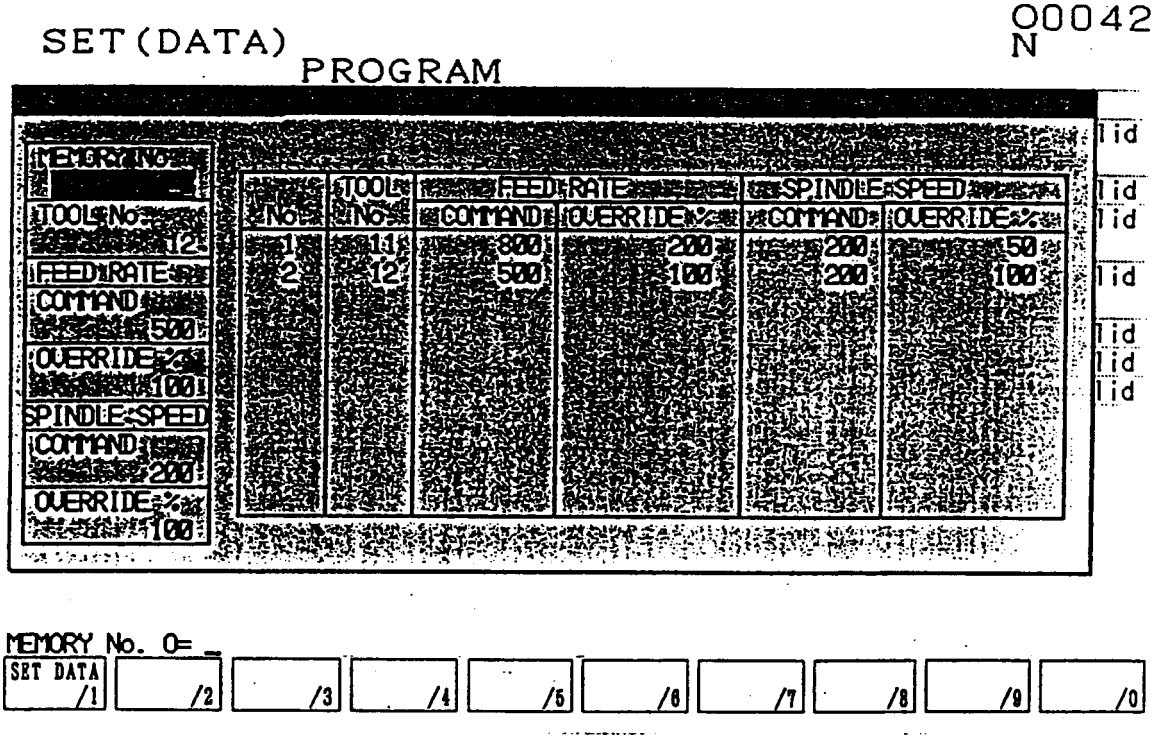


Figure 5-3 Override Memory

Content of Screen Display

Memory number (O)	Override memorized number (Machining program number O is specified.)
Tool number	Tool numbers are memory displayed in the order specified in the machining program.
Feed rate command value	The feed rate (F code) specified in the machining program is displayed.
Feed rate override	The feed rate override is displayed.
Spindle's rotational frequency command value	The spindle's rotational frequency (S code) specified in the machining program is displayed.
Spindle's rotational frequency override	The spindle's rotational frequency override is displayed.

The tool number, the feed rate command value, the feed rate override, the spindle's rotational frequency command value and the spindle's rotational frequency override displayed in the right part of the screen display the currently selected memory.

(2) Override Memory Function

(a) Display the override memory screen before starting test cutting.
What has been memorized is displayed.
To create anew, input the memory number (number 0) in the key input area and press "input". All the data on the screen is cleared and the number 0 just input is set.

(b) Press the "automatic" key of "override" on the operator's console to enable the memory function.

(c) Start machining.
The T code specified during machining is memorized in the tool number in the memory and the feed speed, the feed override, the spindle's rotational frequency and the spindle override at that time are memorized respectively and are displayed in the column where there is the frame cursor.

(d) Perform cutting. To rewrite due to the override having changed, press the "memory" key of "override".

(e) After finishing the test cutting, press the "automatic" key of "override" to reset the memory function.

(3) Automatic Override

(a) Before starting machining, press the "automatic" key of "override" on the operator's console to enable the automatic override function.

(b) Start machining.
Machining is performed according to the memorized override.
In that case, the operator's console's override is disabled.

(4) Override Memory Edit

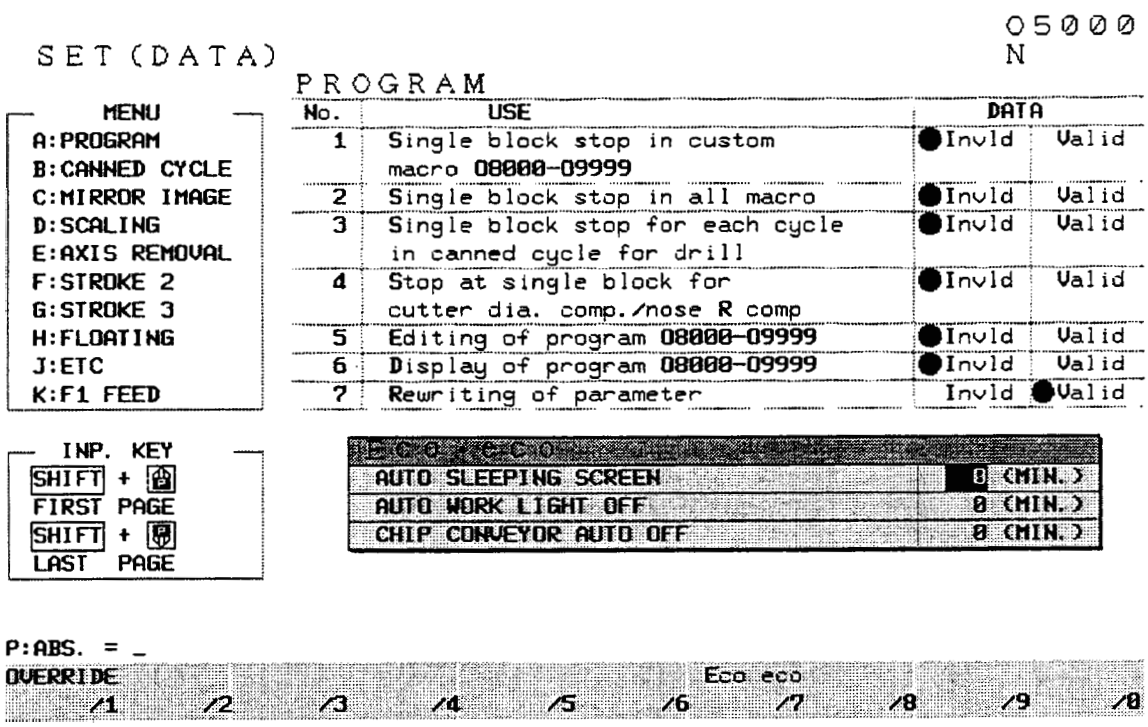
Override memory edit is impossible unless the following conditions are satisfied.

- The write key is on.
- The MDI mode is assumed.

Move the cursor to the data to be altered using the cursor key [←], [→], [↑], [↓], the override memory screen, input the alteration data in the key input area and alter by the “input” key. Override memory edit is possible on a maximum of 60 tools.

5-4 Eco・eco

Pressing [F7/Eco eco] displays the Eco eco screen (Fig. 5-4). In this screen, you set the auto sleeping screen, auto work light OFF, and chip conveyor auto OFF times (minutes)



<Fig. 5-4 Eco・eco>

6. Macro Variable

Press [F8/MACRO VARIABLE] in the PROGRAM screen (F2/PROGRAM). The MACRO VARIABLE screen appears (Fig. 6-2(a)).

6-1 Macro Variable and Data Display

The macro variable data is classified into the following:

①	1-33	Local variables (correspond to call multiplicity)
②	100-499	Common variables (turned to “null” at power-off time)
③	500 or above	Common variables (with name, remembered if the power is turned off)

They are changed over with the page keys, respectively. (Fig. 6-2 and Fig. 6-3)
The number of significant digits of data is 8 (± 99999999 to ± 0.0000001). When the numerical value exceeding this limit has been input, the message “OVER FLOW” is output. Nothing is displayed in case of the “null” data (empty, not 0). The name can be input up to 12 characters, but it is given to the common variables 500 or above.

(1) Data deletion
Since the data of No. 500 or above is remembered even if the power is turned off, a new function is provided, which can delete all.
If the function menu is displayed on Page No. 500 or above, the Menu 7 (DATA CLEAR) appears. Pressing this menu displays the question at the center of the screen whether to erase with 0 or “null”. Select either of them with the cursor key (↑ or ↓) and press the INPUT key. Finally, the message “OK?” appears in the key input area for safety confirmation. Press the Y key.

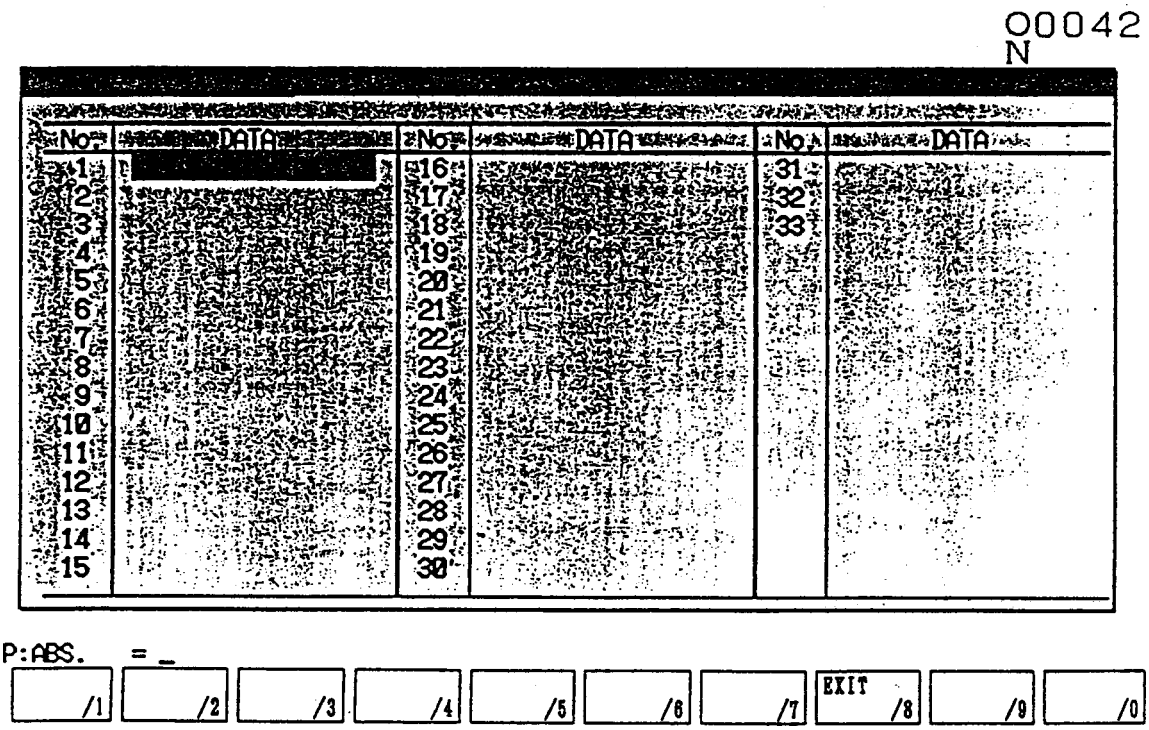
(2) “Null” input
If you press the CAN key when the key input area contains nothing, the “null” characters are displayed. Then, pressing the INPUT key turns the data at the cursor position to “null”.

(3) Deleting the name
Set the cursor to the name you want to delete (common variable 500 or above), press [SPACE], followed by [INPUT].

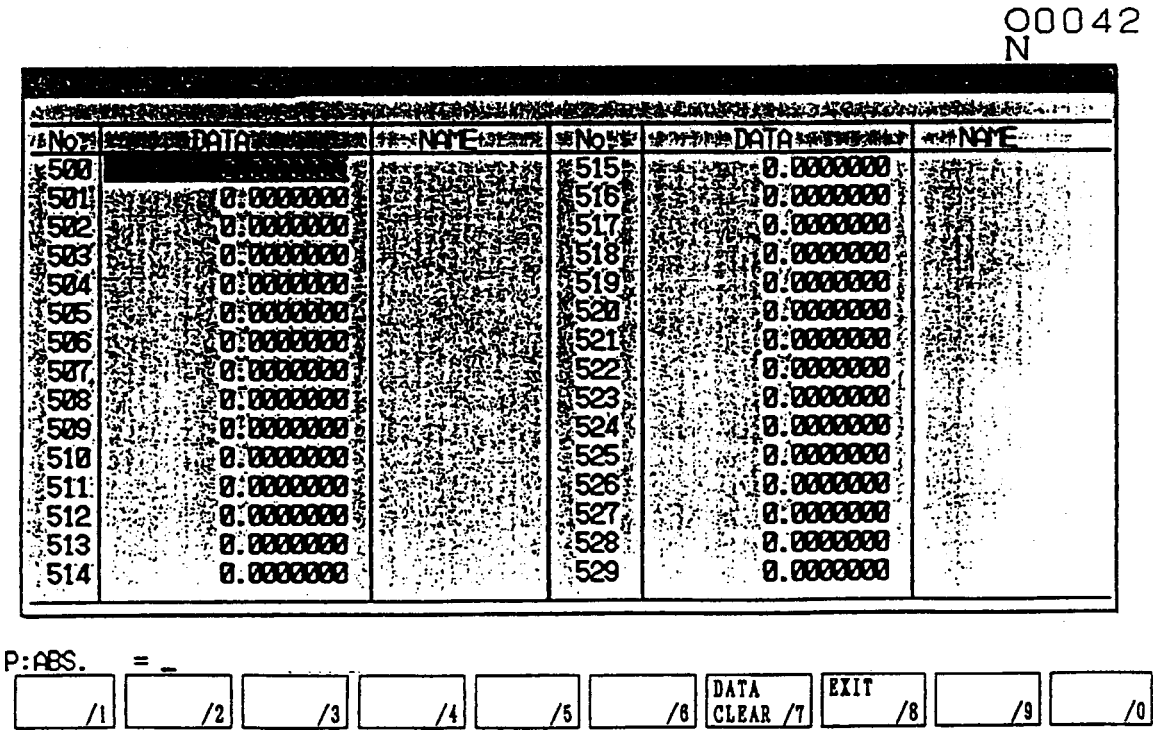
6-2 Number Search

The page and cursor can be moved at one time by searching the macro variable No. A search can be performed by pressing the cursor key (↑, ↓, →, ←) after inputting N and a subsequent numerical value.

Example) N: No. = 12 ↓ (Moves to No.12)



<Fig. 6-2(a) Local Variable>



<Fig. 6-2(b) Common Variable>

7.Plot

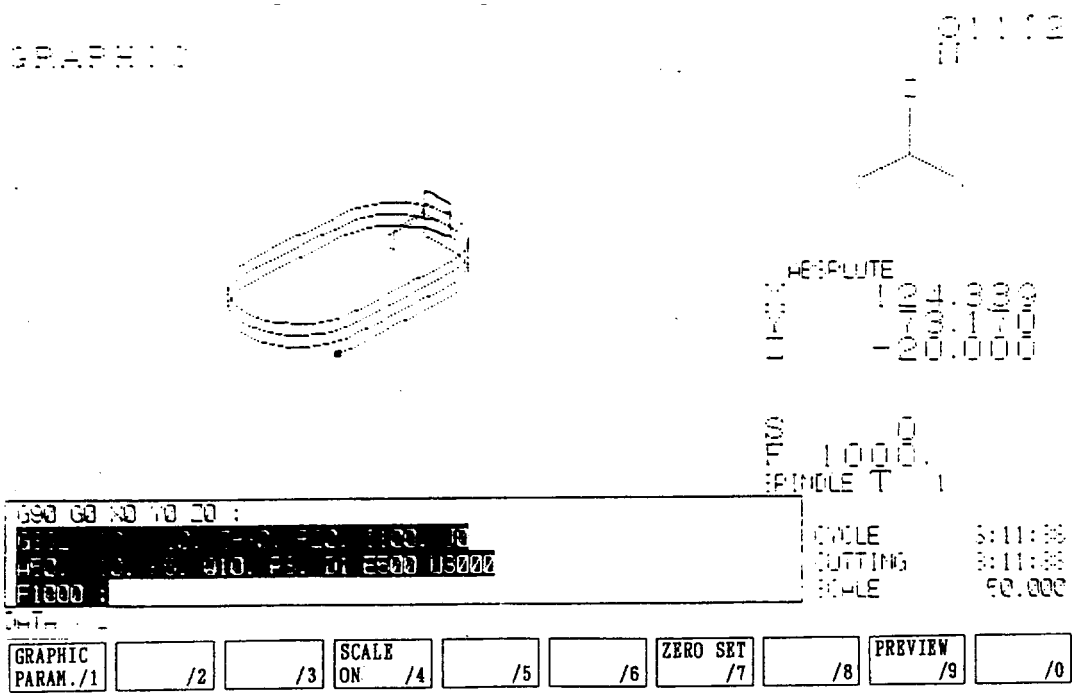
Pressing the [F7/GRAPH] key displays Plotting screen (Fig. 7-0)

This screen plots the tool path and also displays the following data for reference to plotting.

- Parameter plotting plane
- Work Coordinate System
- S, F, Spindle tool T
- Machining time, cutting time, Scale
- Program list in execution

A locus is drawn in the values of the work coordinate system. As too many lines can confuse movement, the tip is expressed in a small dot.

To erase the graphic page, push ORIGIN.



<Fig. 7-0 Plotting Screen> (Preview)

For plotting, it is possible to set the following. For details, refer to the next section.

- Selection of plotting plane
- Specification of angle of rotation (horizontal, vertical)
- Plotting range (maximum, minimum)
- Scale width indication
- Specification of plots per tool (color specification)
- Selection of cutting feed line
- Selection of rapid traverse line
- Color assignment for drawing point

In normal state, synchronous plotting is in display. When performing pre-machining plotting, therefore, press F9/PREVIEW to display the function key for pre-machining plotting. (Fig. 7-1)
Next, push F3/PREVIEW START key to start pre-machining plotting.
With F2/AUTO RANGE key pushed, auto range plotting starts.
When plotting range has been set so that a cutting locus falls within the screen scope on completion of plotting, pre-machining starts automatically.

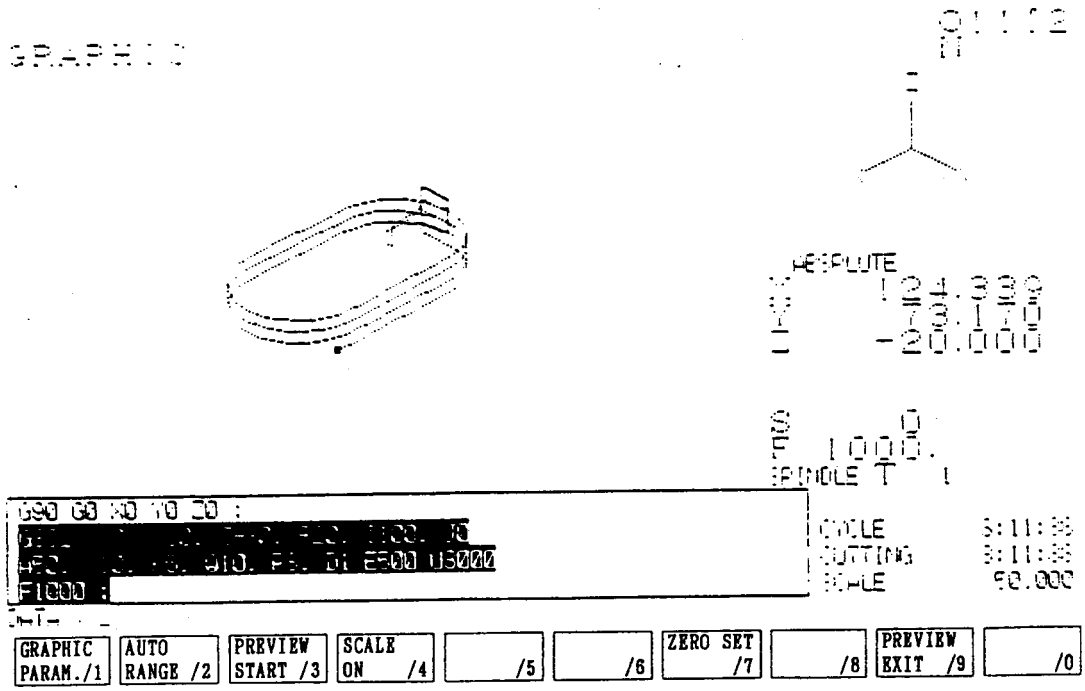
(Note 1) In order to start pre-machining plotting, first perform zero point return to initialize other states of Machine such as mechanical clamp.

(Note 2) When locus image is obscure in display, lower speed with the feed speed switch.

(Note 3) Those functions whose conditions change by machine operation (M60 cycle, macro program using skip function, oscillation function) cannot be executed in pre-machining plotting.

(Note 4) When the add axis gets as follows, either stopping or alarming takes place where pre-machining plotting cannot be executed.

- While interlocking of each axis is held ON, stopping takes place with the add axis command.
- On removal of axes, alarm for “Zero Return Not Completed” takes place with the added axis command.



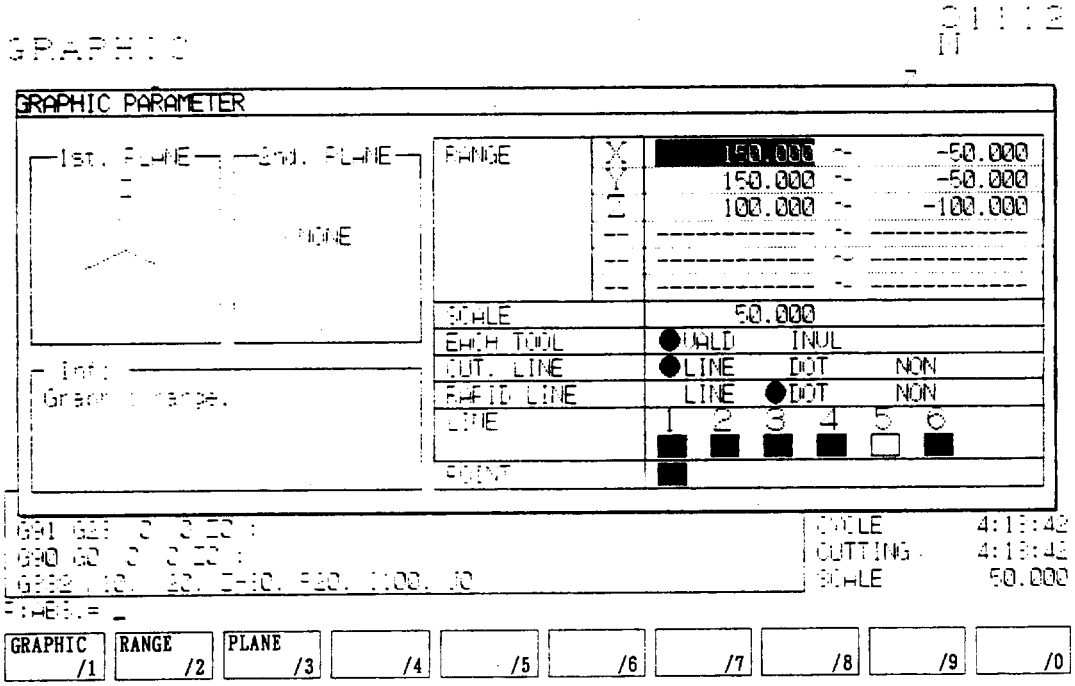
<Fig. 7-1 Plotting Screen> (Preview Exit)

8. Plotting Parameters

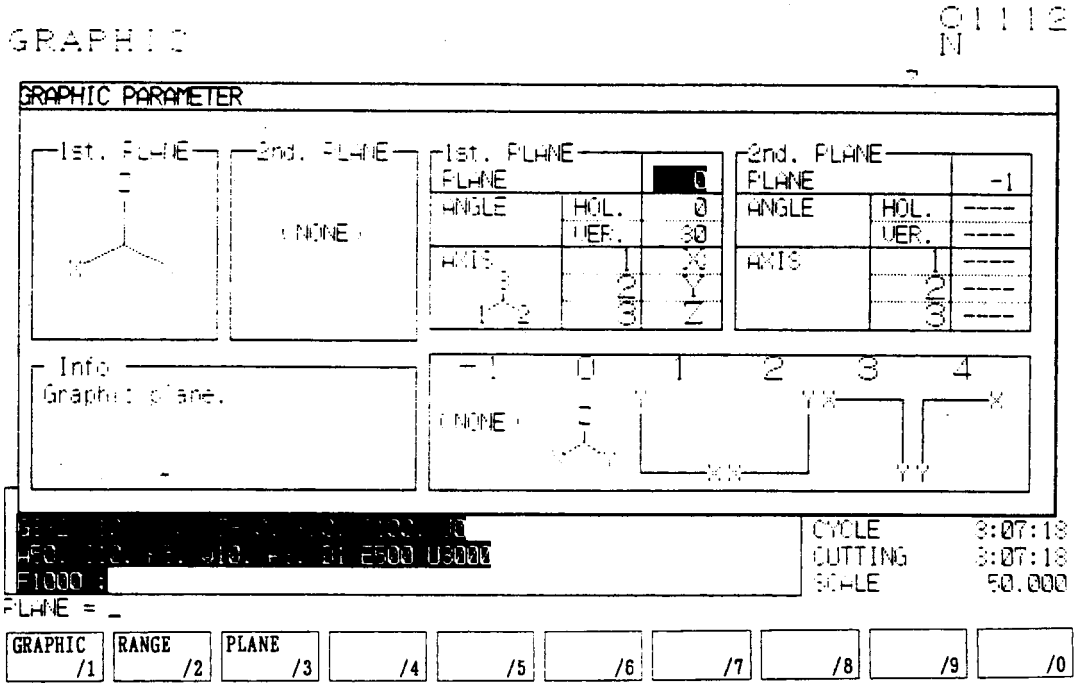
With the [F1/GRAPHIC PARAM.] menu of the [F7/GRAPH.], the Plotting Parameter screen (Fig. 8-0 or Fig. 8-1) appears over the Plot screen.

This screen allows you to set the plotting parameters.

Pressing the [F2/RANGE] key displays the Range Set screen (Fig. 8-0) and pressing the [F3/ PLANE] key displays the Plane Set screen (Fig. 8-1).



<Fig. 8-0 Plotting Parameter (Range Set)>



<Fig. 8-1 Plotting Parameter (Plane Set)>

- Setting plotting parameters

  First move the frame cursor to the column of the parameter to be set using the cursor key [↑] [↓].

  At that time, a simple explanation is displayed in the explanation column.

Range set

(1) Plotting Range

Set the maximum and minimum plotting values of each axis. The center coordinate (middle of the maximum and minimum values) and magnification factor (at which the maximum and minimum values stay within the screen) for plotting are decided. Input the coordinate value (work coordinate) in the key input area and press the INPUT key to decide. The maximum value and the minimum value can be set at left or right at that time.

(2) Scale

A scale width is set here.

(3) Plot per Tool

Every time the tool is changed, a plotting color is changed. However, this is inefective when plotting the specified tool. Up to 6 colors are available. After the 6th color, it returns to the first one.

Make setting at 'Feed Line Color'.

The set one is marked with • . To alter, select with the cursor key ([→] or [←]) and press the INPUT key to decide.

(4) Cutting Feed Line

Specify a type of cutting feed line. The set one is marked with • .

To alter, select with the cursor key ([→] or [←]) and press the INPUT key to decide.

(5) Rapid Traverse Line

Specify a type of rapid traverse lines The set one is marked with • .

To alter, select with the cursor key ([→] or [←]) and press the INPUT key to decide.

(6) Feed Line Color

When Plot per Tool is effective, set the changed-to color. Only this color designation can be altered later. To make setting, select color designation with the cursor key ([↑] or [↓]). Then, specify the tool with the cursor key ([→] or [←]).

In accordance with the description of color designation provided in the center of the screen, set a numerical value 0-7 and press the INPUT key to determine. The set color appears in [ ] on the display.

(7) Plotting Point Color

Specify a color of plotting point.

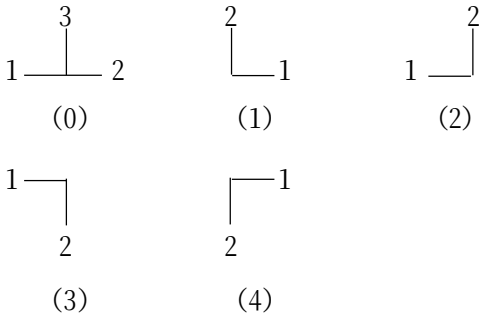
To make setting, select color designation with the cursor key ([↑] or [↓]).

In accordance with the description of color designation provided in the center of the screen, set a numerical value 0-7 and press the INPUT key to determine. The set color appears in [ ] on the display.

Plane Set

(1) Plotting Plane

Specify the plotting plane. You can select out of the following 5 types.



In accordance with the description of the plotting plane provided in the center of the screen, set a numerical value 0-4 and press the INPUT key to determine. The selected plane is displayed.

The plane setted a numerical value -1 is not displayed.

NOTE) No scale is displayed when the plotting plane is equal to 0 (in 3-axes display)

(2) Rotation Angle

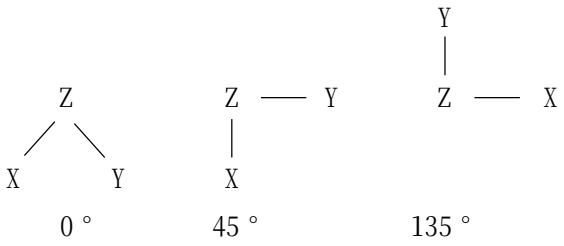
This is effective only when the plotting plane is 0 (3 axes display).

Input the angle in the key input area and press the INPUT key to decide.

(a) Horizontal rotation angle

Specify the horizontal plane rotation angle within a range of ±180 in an increment of 1 °

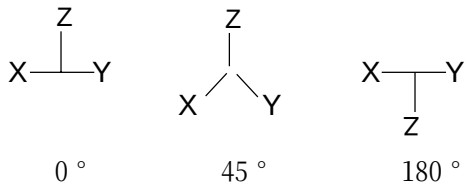
Example) When the plotting plane is (XYZ) and the vertical rotation angle is 90 °



(b) Vertical rotation angle

Adjust the slope angle of the vertical axis.

Example) When the plotting plane is (XYZ) and the horizontal rotation angle is 0 °



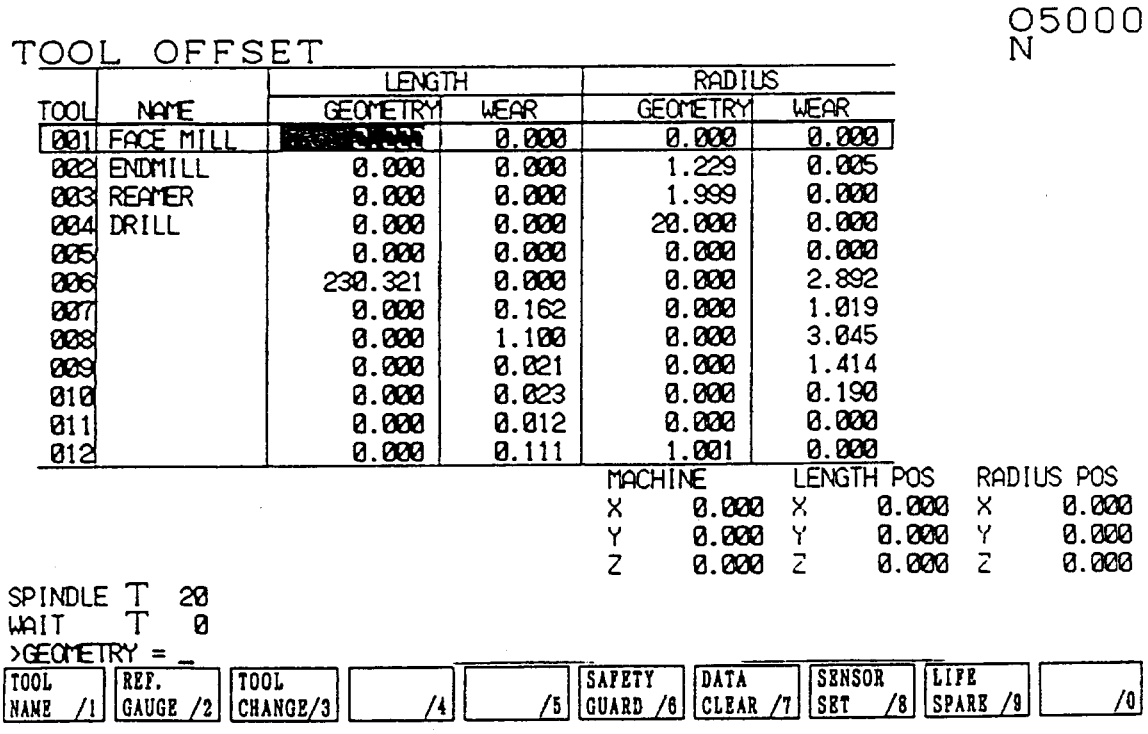
(3) Specified axis

Set the axis name which you want to specify. Minus (“-”) can be added to it.

9. Tool (Offset)

Pressing the function 3 (OFF SET) key displays the Tool screen (Fig. 9-0).
This screen is used to display and set the tool name, tool length compensation and cutter compensation for each tool number. In addition, the following function are provided.

- Display of the machine coordinate system
  (For parameter No.3108 #0 = 1, it is relative coordinate system.)
- Display of the spindle tool number and standby tool number
- Tool change
- Safety guard
- Setting of the sensor tool
- Tool setter
  Display of the length measurement position and diameter measurement position, reference gauge
- Data deletion



<Fig. 9-0 Tool (offset)>

- Handling of tool compensation data
  The tool length compensation data and the tool diameter compensation data are divided into profile and wear per item. Compensation using H, D in the program corresponds to the tool number on the tool (compensation) screen.
  For details, refer to the item of the H, D function in “Program”.

9-1 Compensation Value Setting

Compensation values are divided into profile and wear. Setting is made from the key input area together with the cursor. While it is possible to input wear at any time, it is possible to input profile only while the write key is on. Wear becomes 0 at the input of profile. It is possible to input an absolute value or an incremental value.

Input in absolute value : The key input data is just input.

Input in incremental value : The key input data is input as added to the original data.

- Decimal point input (minimum setting unit)

  Parameter 5004's bit 7

  0 : If '1' is input, the set value is 0.001.
  (If the number of places of decimals is 3)

  1 : If '1' is input, the set value is 1.000.

- Data inputting method

  There are the following 2 methods.

  Parameter 5004's bit 4

  0 : Input using the "input" key assuming the profile to be an absolute value and the wear to an incremental value.

  1 : When key input, the function menu shows "F1/absolute value" "F2/incremental value", so press either 1 to input it.

- Incremental value input clamp

  It is impossible to input a value which is greater than the value set in parameter 5027.

  A message "The clamp value is exceeded." Is given.

  Example Profile = 1.234 "F1/absolute value" (1.234 is input.)
  Wear = 0.234 "F2/incremental value" (0.234 is input as added.)

9-2 Name Setting

Use the function menu to set the name. First, press the [F1/TOOL NAME SETTING]. The window appears in the center of the screen.

Select with the cursor key (↑ or ↓) or page key (PAGE UP or PAGE DOWN) and press the INPUT key to determine. Selecting "00. Alphanumeral Input" allows you to input the data through the key input area.

When you want to prefix a dimension the tool name, determine a tool name position with the cursor. Then, set with the alphanumeral data "input" keys.

Since the name is allowed up to 10 characters, there are some abbreviated ones.

Example) D45. 0-3 / Chamfering → D45. 0-3 face
(Input) (Cursor Position) (Tool Name)

9-3 Data Delete

Used to delete 1 horizontal line of the data of the tool indicated by the cursor or to delete certain data collectively in the vertical direction to 0 for all tools when, for example, starting anew.

If you press "F7/data delete", a window will appear in the center of the screen to display the deletable data.

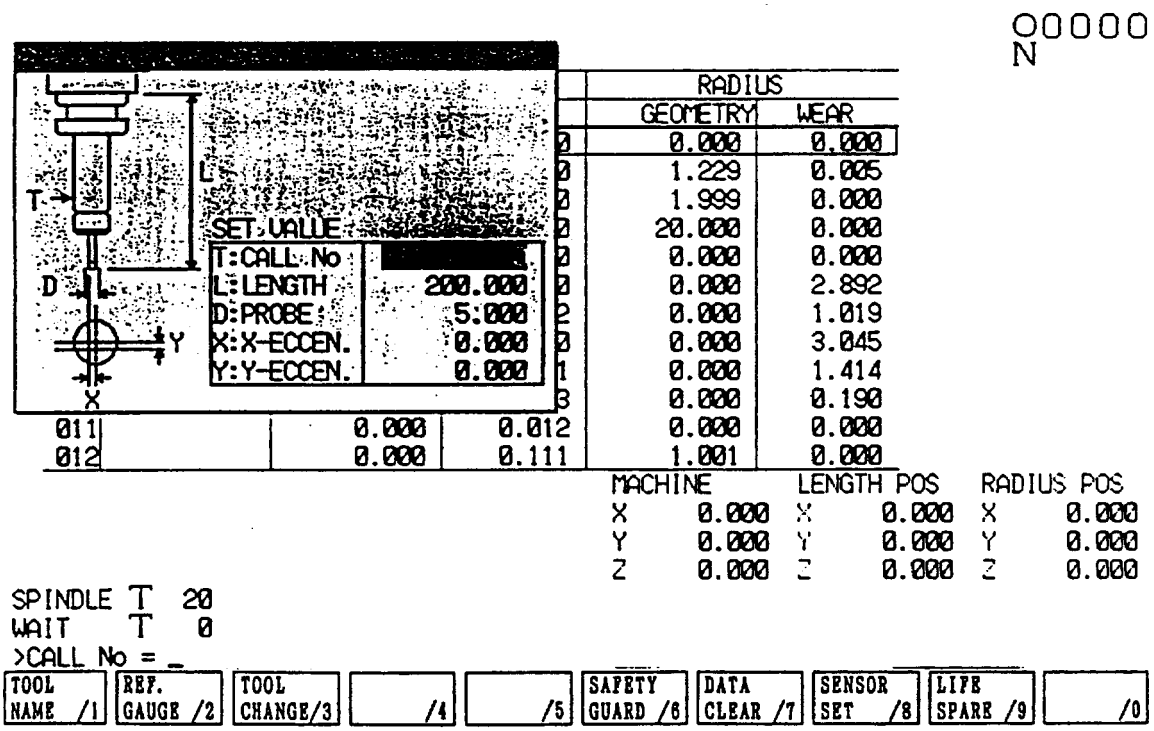
Select any using the cursor key X X and press the input key.

You will be asked "Are you sure? Y-Yes N-No", so press the "Y" key if so.

9-4 Sensor setting

If you press “F8/sensor set”, a window will appear in the center of the screen to enable you to set the sensor for the work setter. (Figure 9-4)

In “T: sensor call”, the same process as tool change is performed.



<Fig. 9-4 Sensor Set>

9-5 Reference Gauge

The reference gauge is the contact surface of the reference block.

In the manual mode, the distance from the zero point to the reference gauge is displayed. For details, refer to “Operation Related to W Setter”.

9-6 Tool Change

To change tools, enter with the keys the tool number you want to change in the MDI mode, and perform the same operation as the MDI operation by key input. (Fig. 9-6)

9-7 Tool Setter

Refer to “Operation Related to W Setter”.

9-8 Number Search

The page and cursor can be moved at once by searching the tool number.

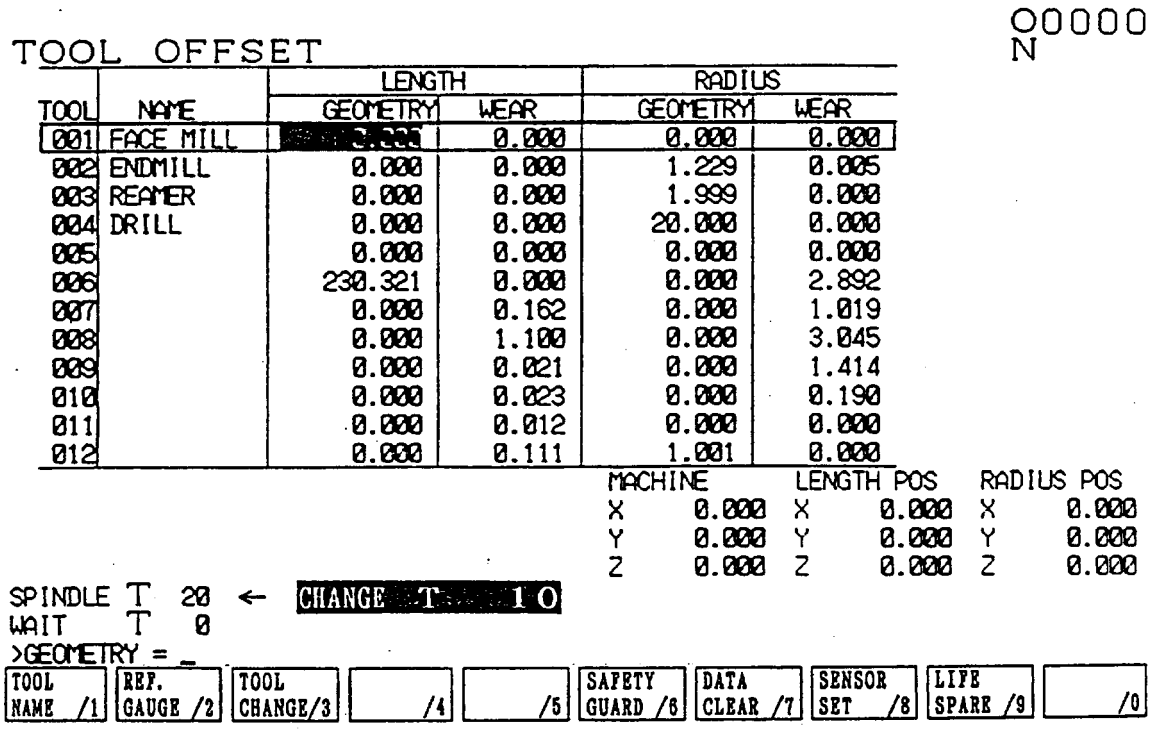
A search is performed by pressing the cursor key ([↑] [↓] [→] [←]) after inputting N and a subsequent numerical value.

Example) N: No. = 12 [↓] (Moves to No.12)

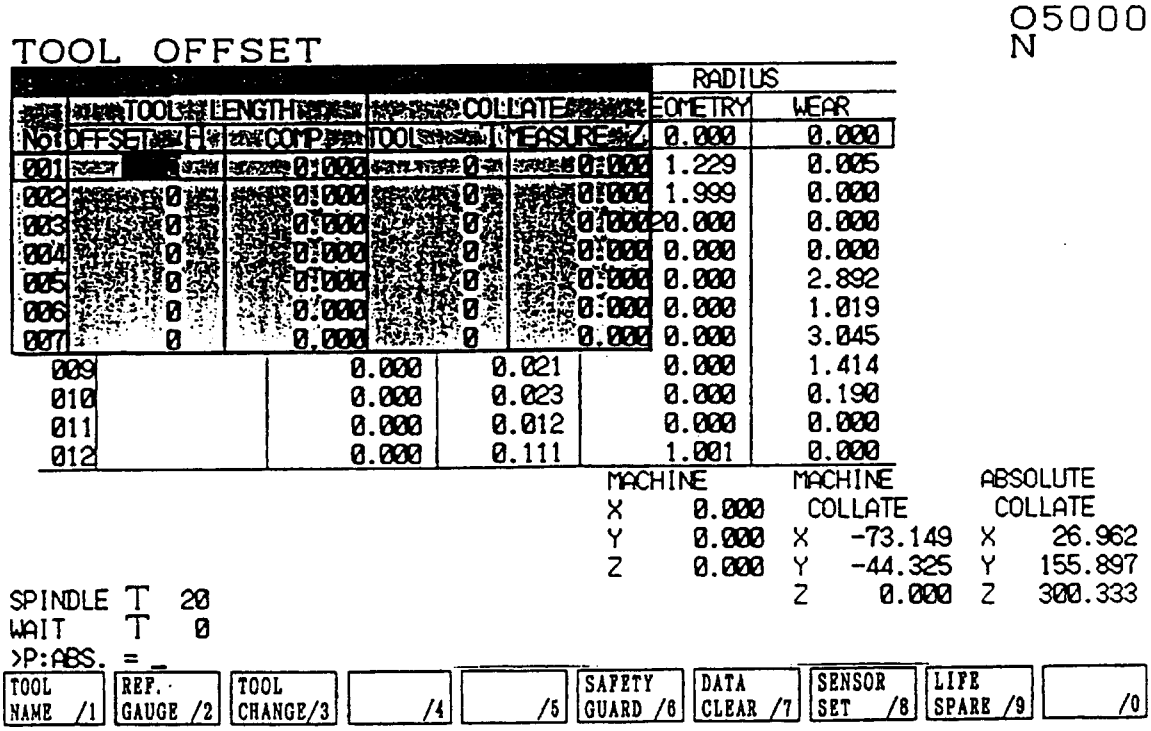
9-9 Safety Guard

If you press “F6/safety guard”, a window will appear in the center of the screen to enable you to set the safety guard. (Figure 9-9)

For details, refer to the item of the safety guard in “Program”.



<Fig. 9-6 Tool Change>



<Fig. 9-9 Safety Guard>

10. Alarm/Diagnosis

Press OPER/MAINTE , followed by F3/ALARM DIAGNOSIS in the BK Menu.

The Alarm Diagnosis screen (Fig. 10-0) appears, designed to display the machine status such as alarms.

It is automatically switched at the same time when an alarm occurs (it is also possible to prohibit switching with a parameter).

All the alarms taking place are divided, for display, into CNC alarms and PMC alarms (which are produced by the ladder sequence software of Machine).

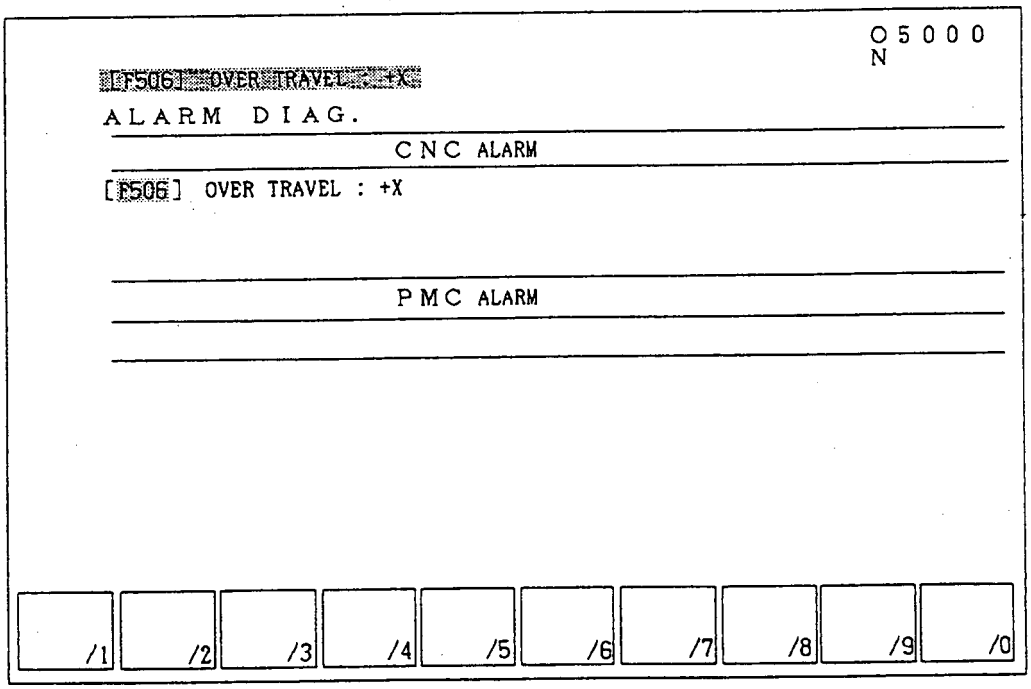
When they cannot fit within one page, they are carried on two pages or more.

Use the page keys for switching.

For CNC alarms displayed at the top left-hand corner of the screen, different colors are used for alarm and for warning.

- Red ..... Alarm (auto operation not available)
- Yellow ..... Warning (auto operation available)

While a PMC alarm is displayed in red color at the top left-hand corner of the screen, whether auto operation is available or not is controlled by the ladder sequence software irrespective of color of alarm display.



<Fig. 10-0 Alarm Diagnosis>

10-1 Alarm History

Press F4/ALARM LIST in the ALARM DIAGNOSE screen. The ALARM HISTORY screen appears (Fig. 10-1). It lists the occurrence data and time, type, and description of each alarm in order of more recent ones. The screen can store up to 32 CNC and PMC alarms. The ALARM HISTORY screen lists only one line of alarm description. When you want to know its details, move the frame cursor and press F2/MESSAGE . This will display a detailed alarm description.

When you want to clear all the records, press F7/HISTORY CLEAR .

A message, "DO YOU WANT TO CLEAR ? Y-Yes N-No," appears. If yes, press Y .

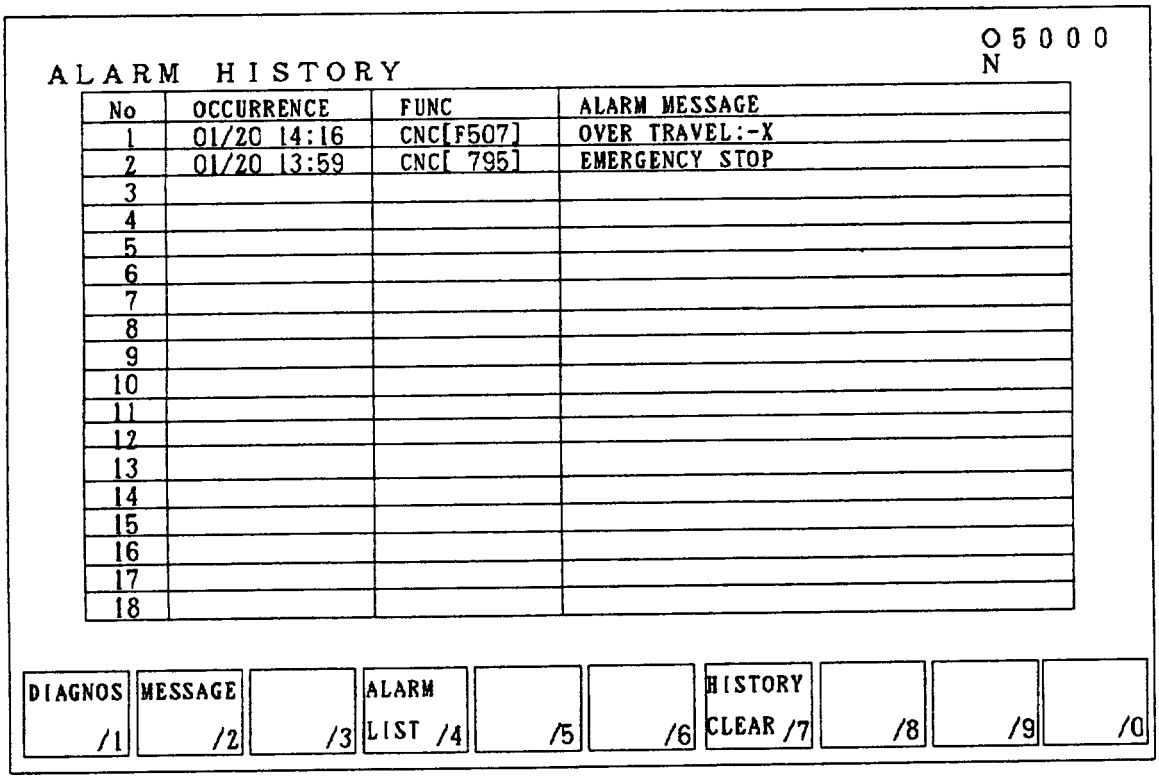


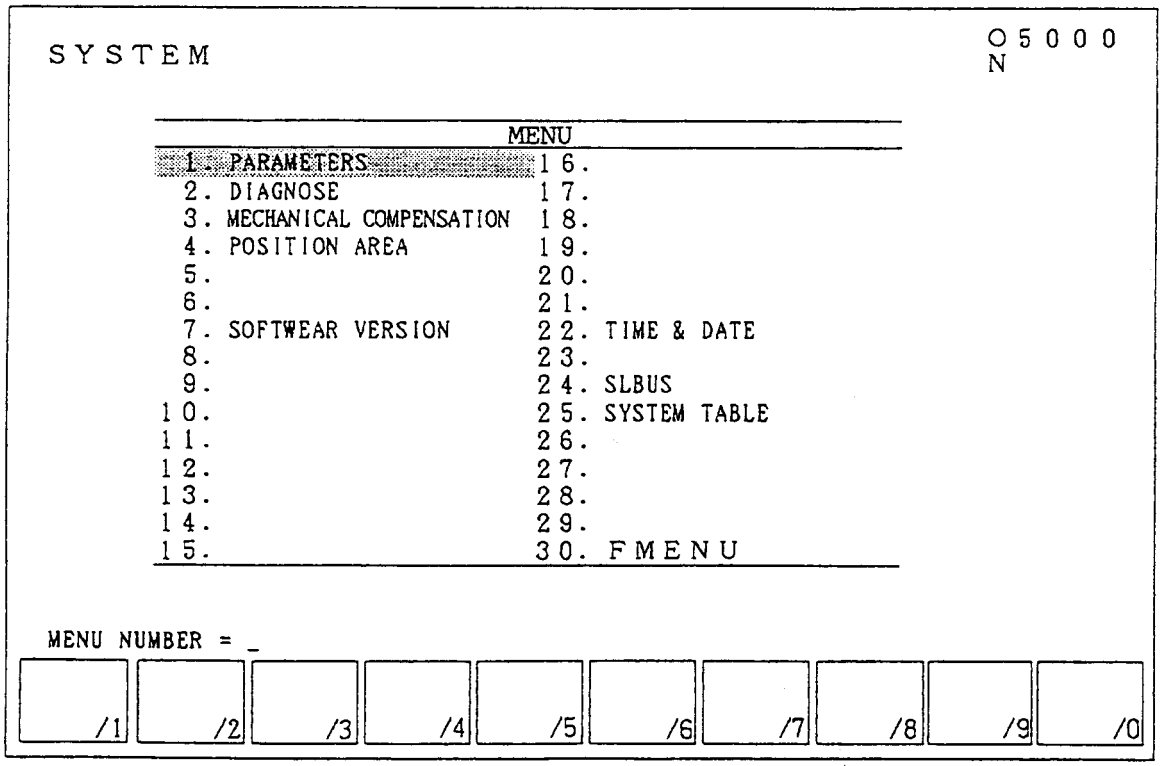
Fig. 10-1 Alarm History

11. System

Press the OPER/MAINTE key, and than, F4/SYSTEM the back menu.
The System screen (Fig. 11-0) appears. This is a collection of the screens used for maintenance by the maker. The following 5 screens exist.

- PARAMETERS
- DIAGNOSE
- MECHANICAL COMPENSATION
- POSITION AREA
- SOFTWEAR VERSION
- TIME & DATE
- SLBUS
- SYSTEM TABLE
- F MENU

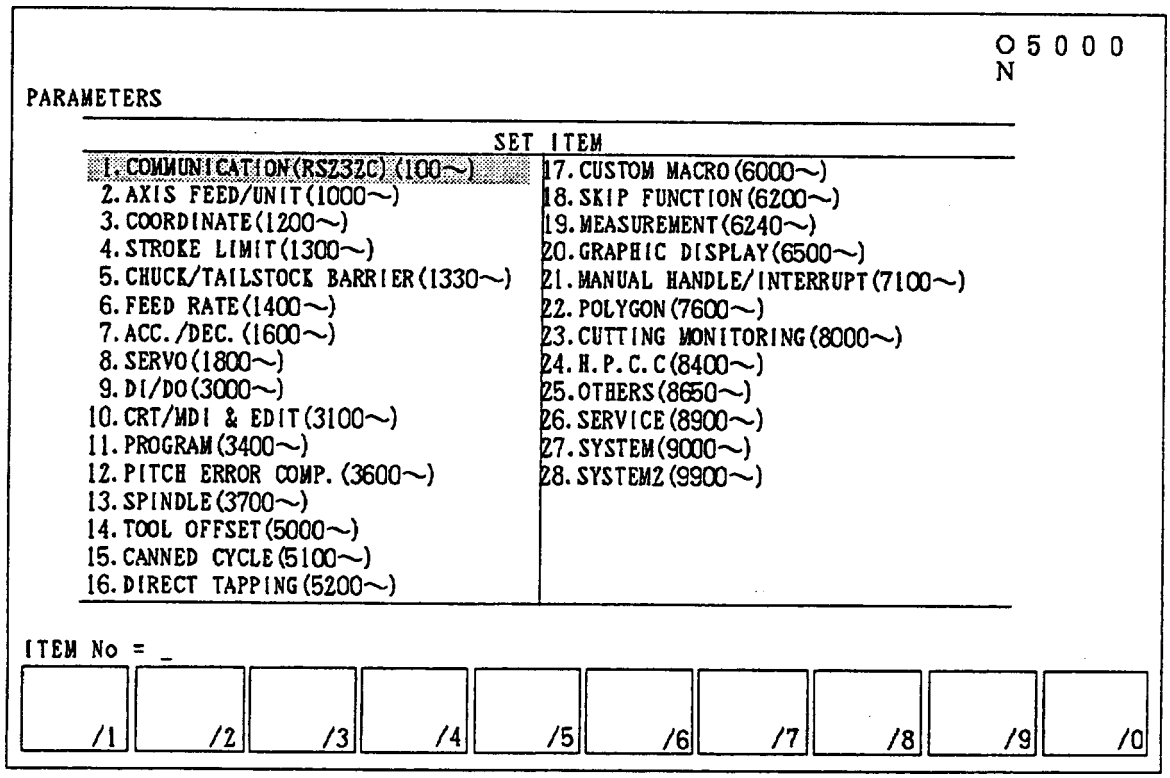
The menu is displayed on the screen. What is displayed in reverse video is the selected screen. To change over the menu, press the cursor keys.



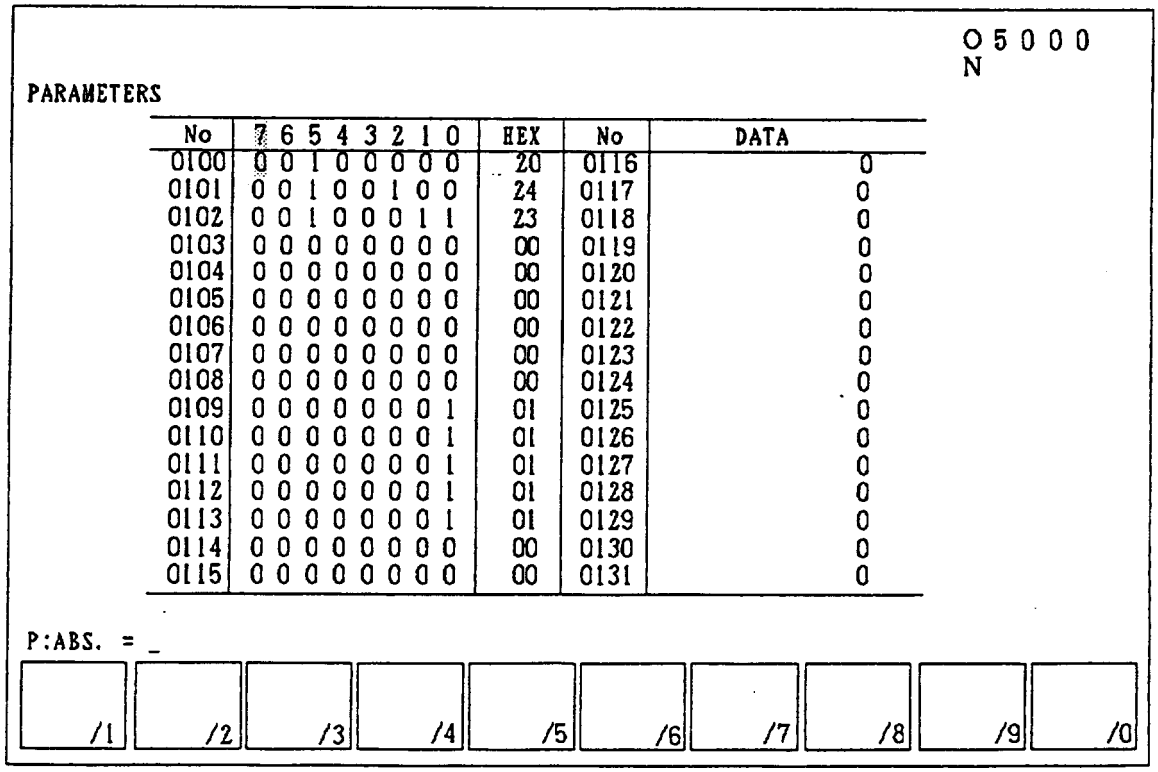
<Fig. 11-0 System>

11-1 Parameter Setting

Set the cursor to Parameter Setting on the System screen and press the INPUT key. The Parameters screen (Fig. 11-1) appears.
Display and set the NC parameters. The parameters are extended over many pages. The first page has the selection by Item screen to help display the page of your desired setting item directly. When a parameter number is known, the page and cursor can be scrolled at once by a number search.



<Fig. 11-1 Parameter>



No	7 6 5 4 3 2 1 0	HEX	No	DATA
0100	0 0 1 0 0 0 0 0	20	0116	0
0101	0 0 1 0 0 1 0 0	24	0117	0
0102	0 0 1 0 0 0 1 1	23	0118	0
0103	0 0 0 0 0 0 0 0	00	0119	0
0104	0 0 0 0 0 0 0 0	00	0120	0
0105	0 0 0 0 0 0 0 0	00	0121	0
0106	0 0 0 0 0 0 0 0	00	0122	0
0107	0 0 0 0 0 0 0 0	00	0123	0
0108	0 0 0 0 0 0 0 0	00	0124	0
0109	0 0 0 0 0 0 0 1	01	0125	0
0110	0 0 0 0 0 0 0 1	01	0126	0
0111	0 0 0 0 0 0 0 1	01	0127	0
0112	0 0 0 0 0 0 0 1	01	0128	0
0113	0 0 0 0 0 0 0 1	01	0129	0
0114	0 0 0 0 0 0 0 0	00	0130	0
0115	0 0 0 0 0 0 0 0	00	0131	0

<Fig. 11-1(a) Parameter>

(1) Item selection

Display the page off your desired setting item, using the page-up key ( ) or SHIFT .
Set the cursor to the desired setting item and press the INPUT key to select it.
Alternatively, since each item is numbered, enter the number of the desired setting item in the key input are and press the INPUT key to select it.

(2) Number search

The page and cursor can be moved at one time by searching for the parameter number. A search is performed by pressing the cursor key (↑, ↓, →, ←) and entering N and a subsequent numerical value.

Example) N: No. = 12 ↓ (Moves to No.12)

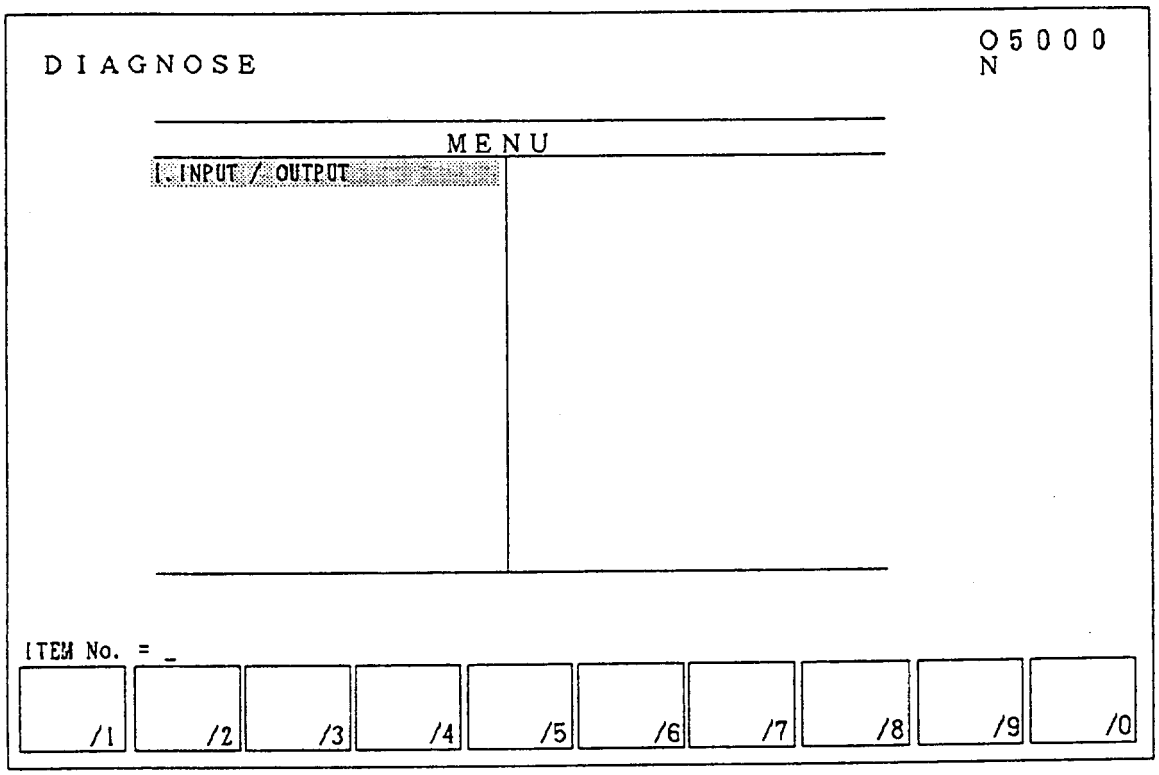
(3) Display and setting

There are two kinds of parameter data; one os represented by 0 or 1, and the other is represented by numerical values.
For data expressed by 0 or 1, display of hexadecimal characters and input are available.
Setting should be done through the key input area, with the cursor set.
To input the data, the following conditions must be met.

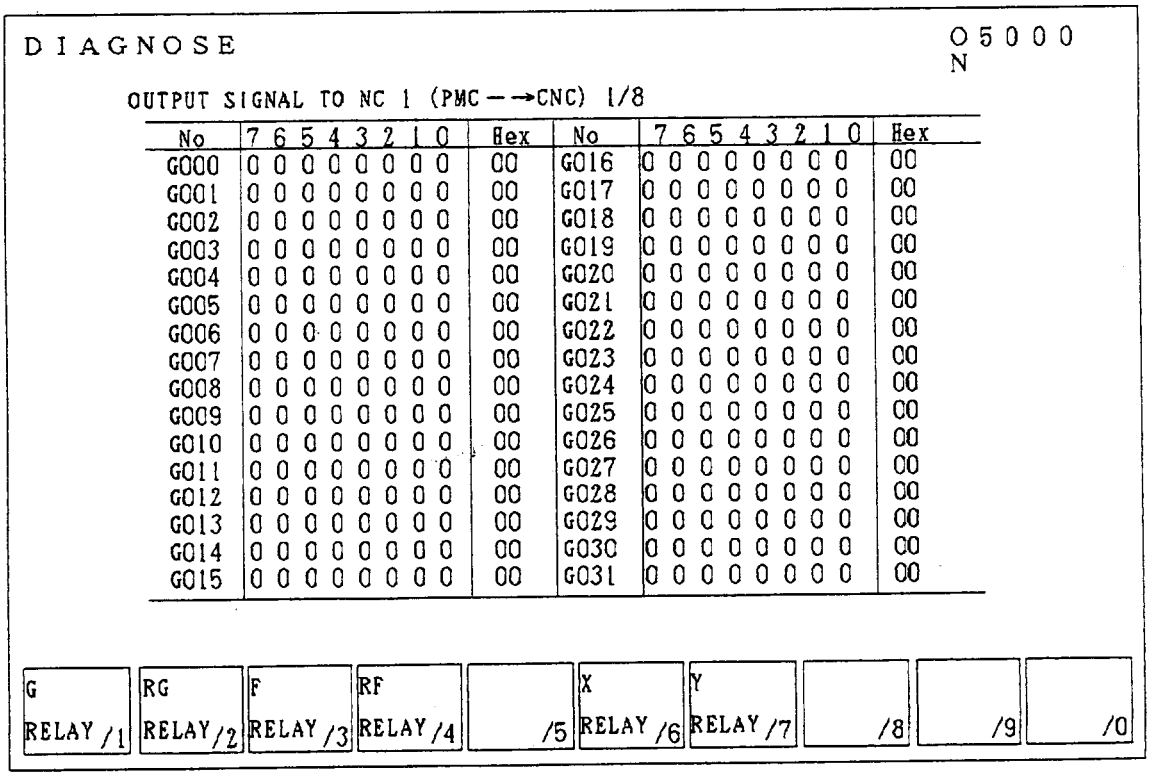
- ○ The NC unit must be reset.
- ○ The WRITE switch 2 must be turned on.
- ○ The MDI mode must be selected.
- ○ “Rewrite parameter” of the Set (Data) screen must be enabled.

11-2 Diagnosis

In the System screen, set the cursor to Diagnose and press the INPUT key.
The Diagnose screen (Fig. 11-2) appears.
Push INPUT the when cursor is held at Menu 1. I/O Signal.
The sequencer, servo, and internal status are displayed at real time. (Fig 11-2 (a))
However, the data cannot be input; it can only viewed.
(Use SHIFT for returning.)
Number search is not possible.



<Fig. 11-2 Diagnose>

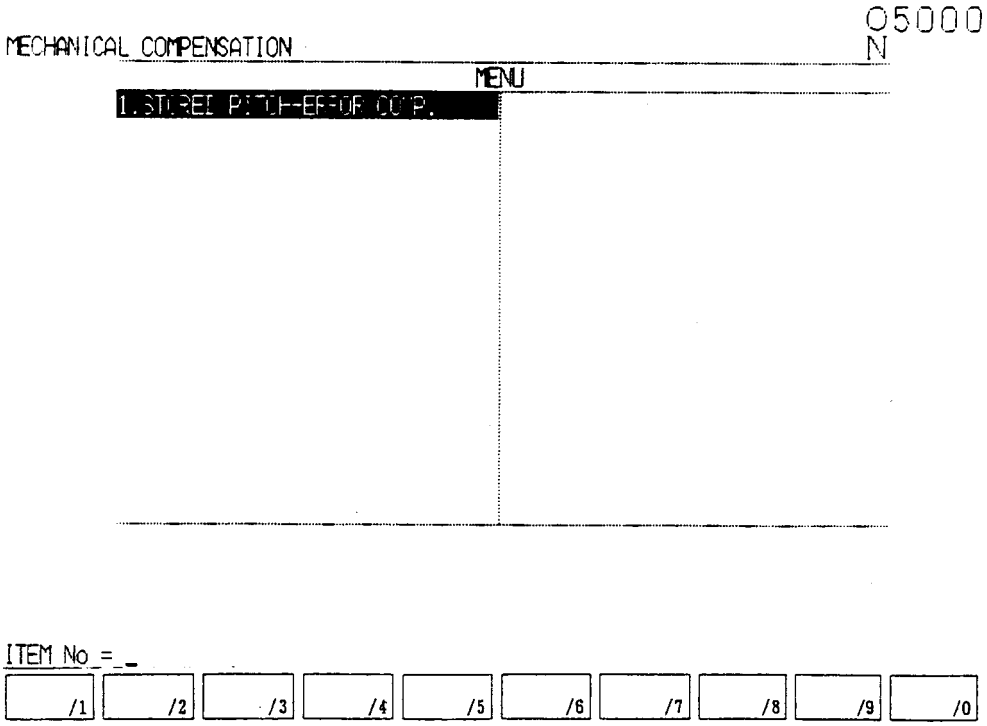


<Fig. 11-2(a) Diagnose>

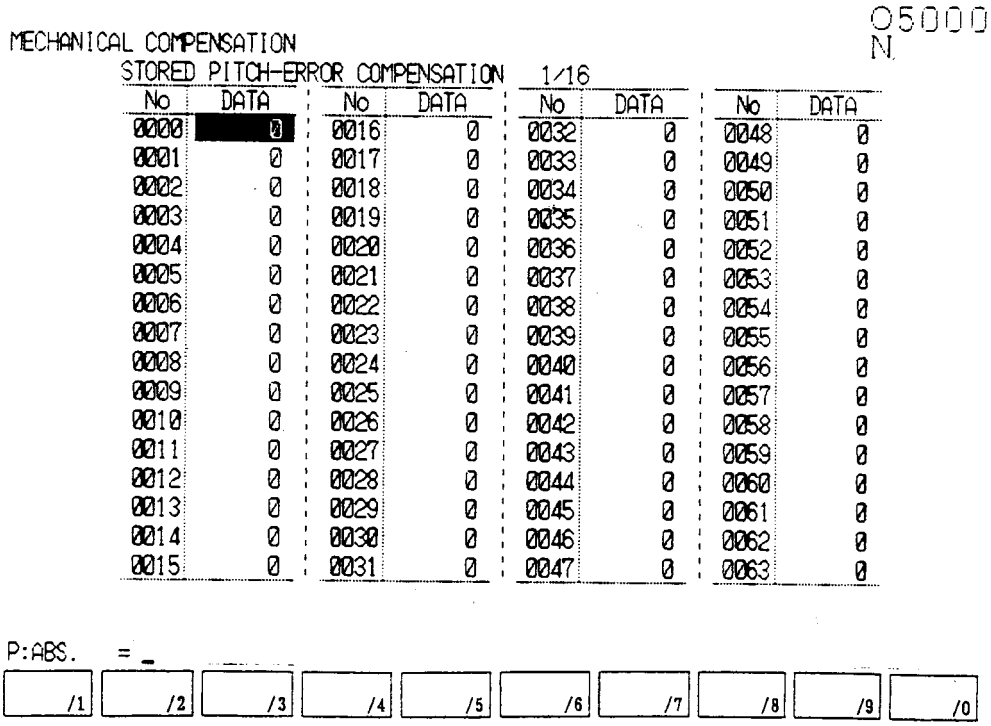
11-3 Mechanical Compensation

Set the cursor to “Mechanical Compensation” on System screen and push INPUT. Mechanical Compensation screen (Fig. 11-3) appears.

Set it to “Stored Pitch-Error Compensation” and push INPUT. Stored Pitch-Error Compensation screen (Fig. 11-3(a)) appears.



<Fig. 11-3 Mechanical Compensation>

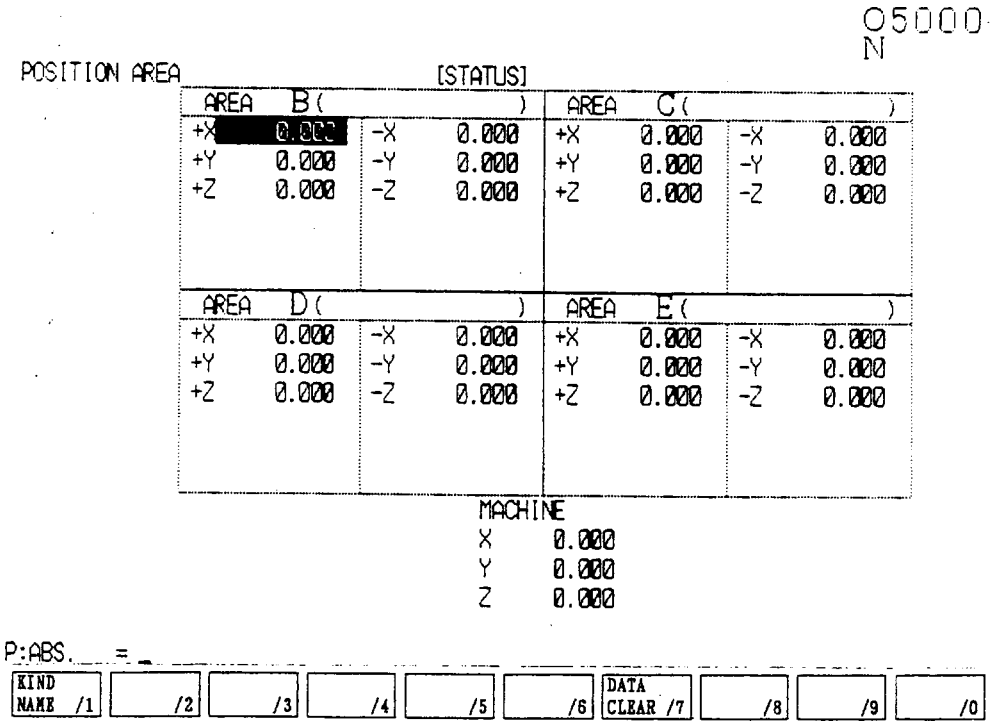


<Fig. 11-3(a) Stored Pitch-Error Compensation >

11-4 Position Region

Set the cursor to “Position Region” on System screen and push INPUT. Position Region screen appears.

Position region data are displayed and set.

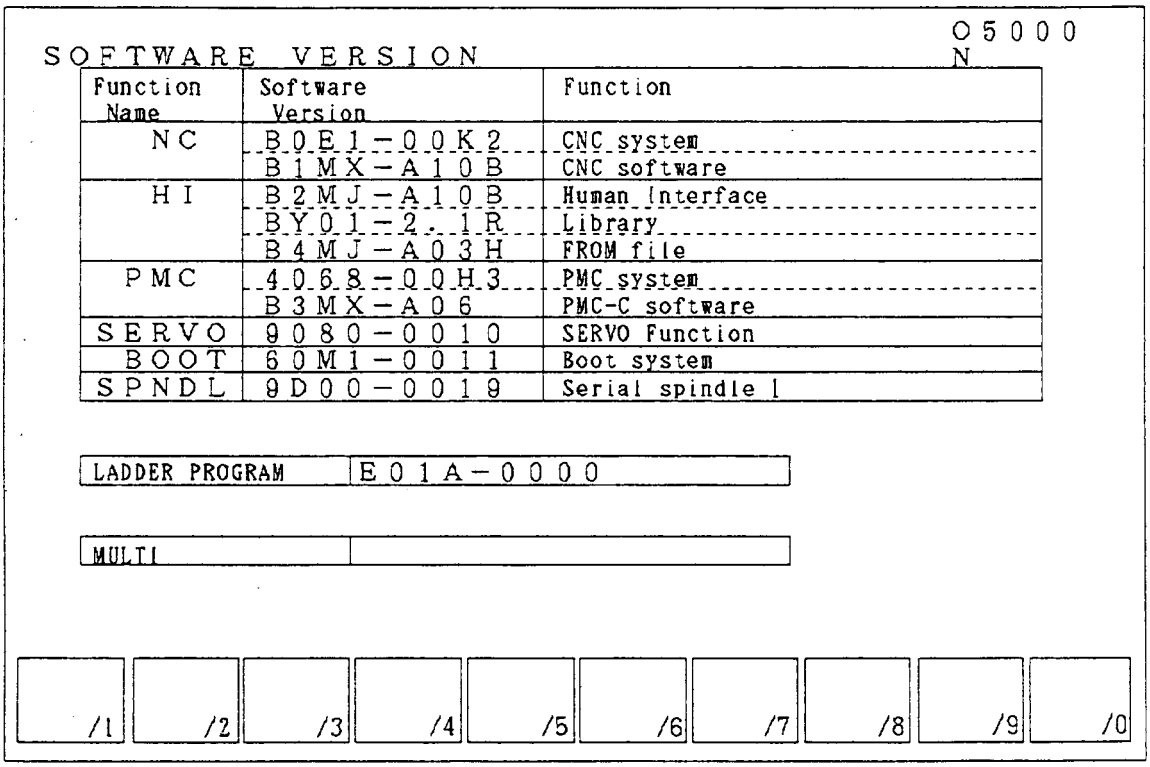


<Fig. 11-4 Position Region>

11-5 Software Version

Set the cursor to “Software Version” on System screen and push INPUT. Software Version screen appears.

Software versions are displayed.

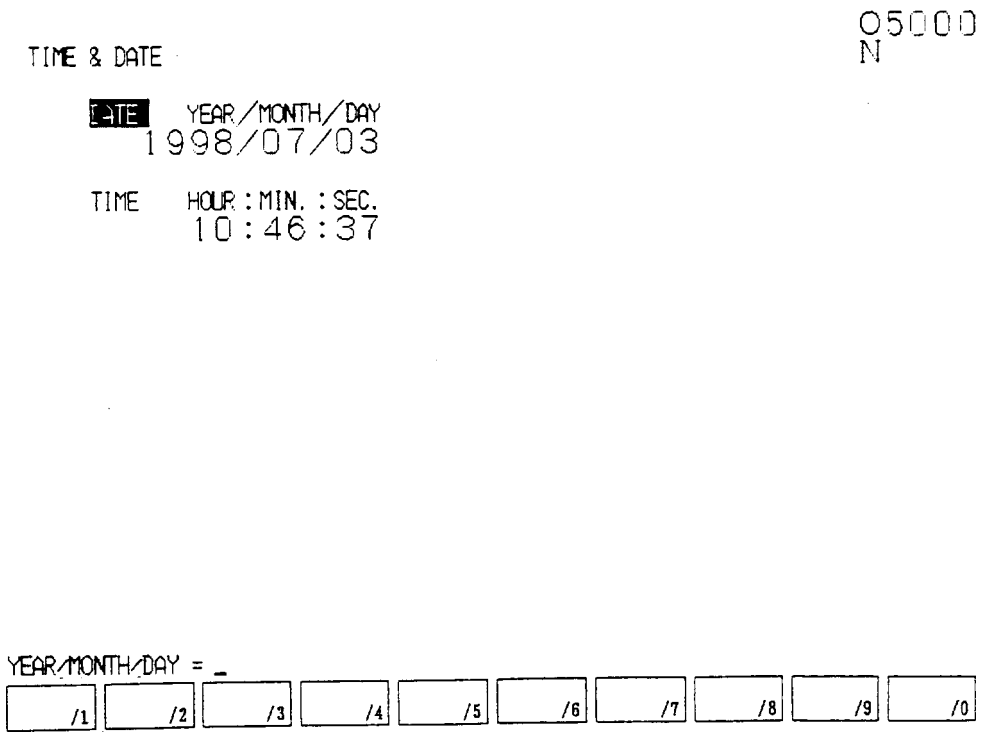


<Fig. 11-5 Software Version>

11-6 Date and Time

Set the cursor to “Date and Time” on System screen and push INPUT. Date and Time screen appears.

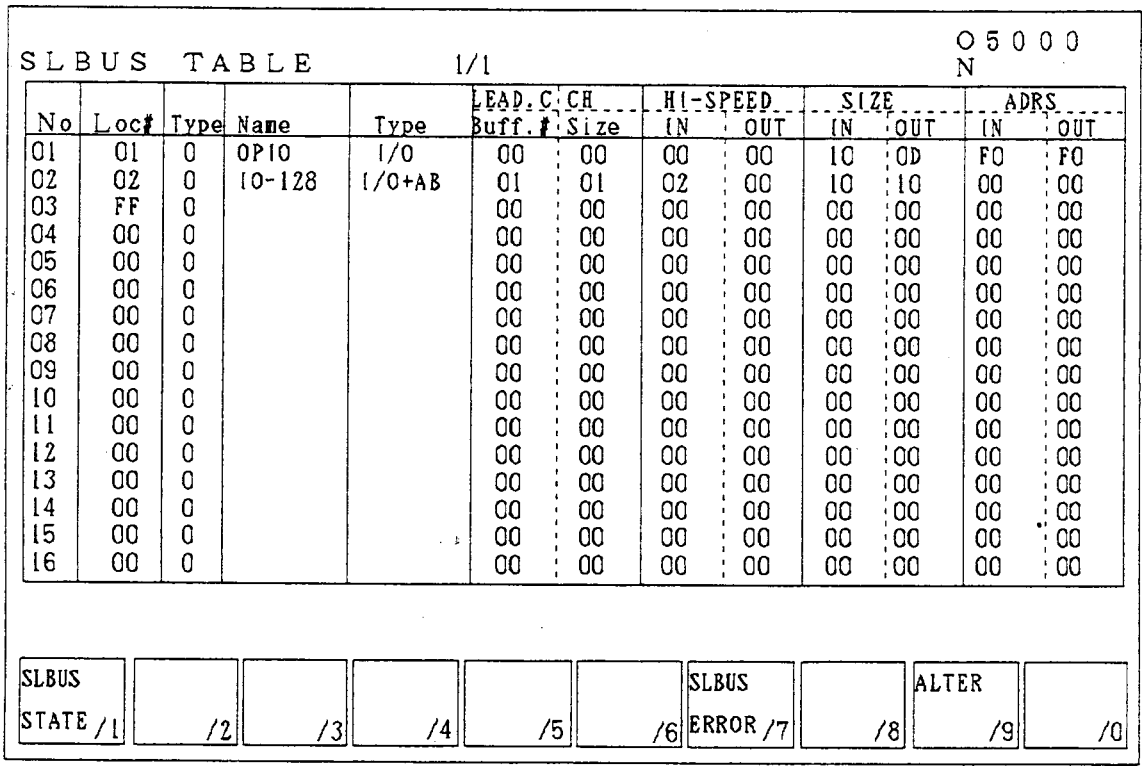
Date and time are displayed and set.



<Fig. 11-6 Date and Time Setting>

11-7 SLBUS

Set the cursor to “SLBUS” on System screen and push INPUT. SLBUS screen appears. SLBUS data are displayed and set.

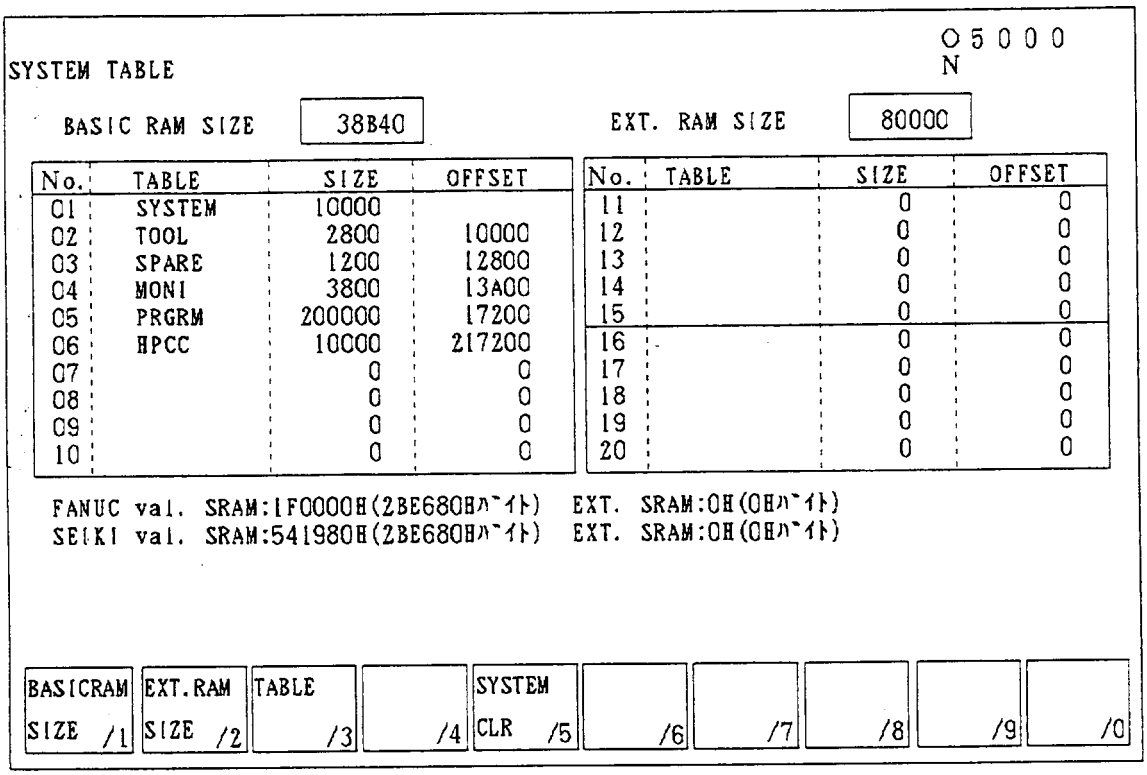


<Fig. 11-7 SLBUS Table>

11-8 System Table

Set the cursor to “System Table” on System screen and push INPUT. System Table screen appears.

System Table data are displayed and set.



<Fig. 11-8 System Table>

11-9 F Menu

Set the cursor to “F Menu” on System screen and push INPUT. FANUC System Menu screen appears.

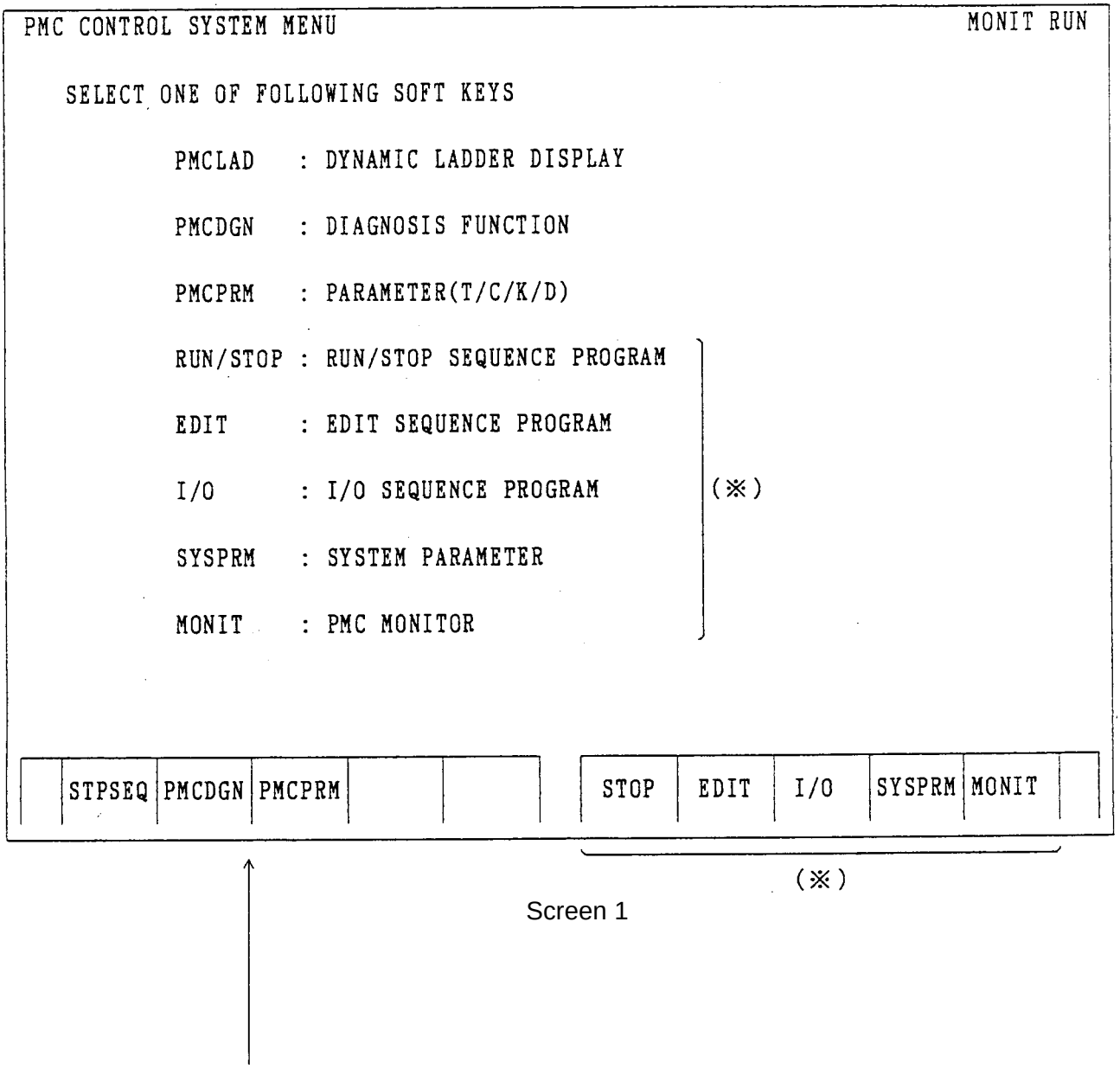
The menu consi ts of the following:

1. F-PMC (Sequencer)
2. F-SYSTEM (System)
3. F-SETTING (Setting)
4. F-MESSAGE (Alarm message)

Set the cursor to the right item and push INPUT. Once you enter screen selected on F Menu, you cannot go back with RETURN (▷) key. Use ALTER key to return to F Menu screen and, then, push RETURN key to go back to Overall screen.

11-9-1 Display Method of Dynamic Ladder Display Screen, Coil Search, Etc.

Input OPER/MAINTE, F4/SYSTEM, 30.F MENU, INPUT, 1.F_PMC and INPUT to display Screen 1.

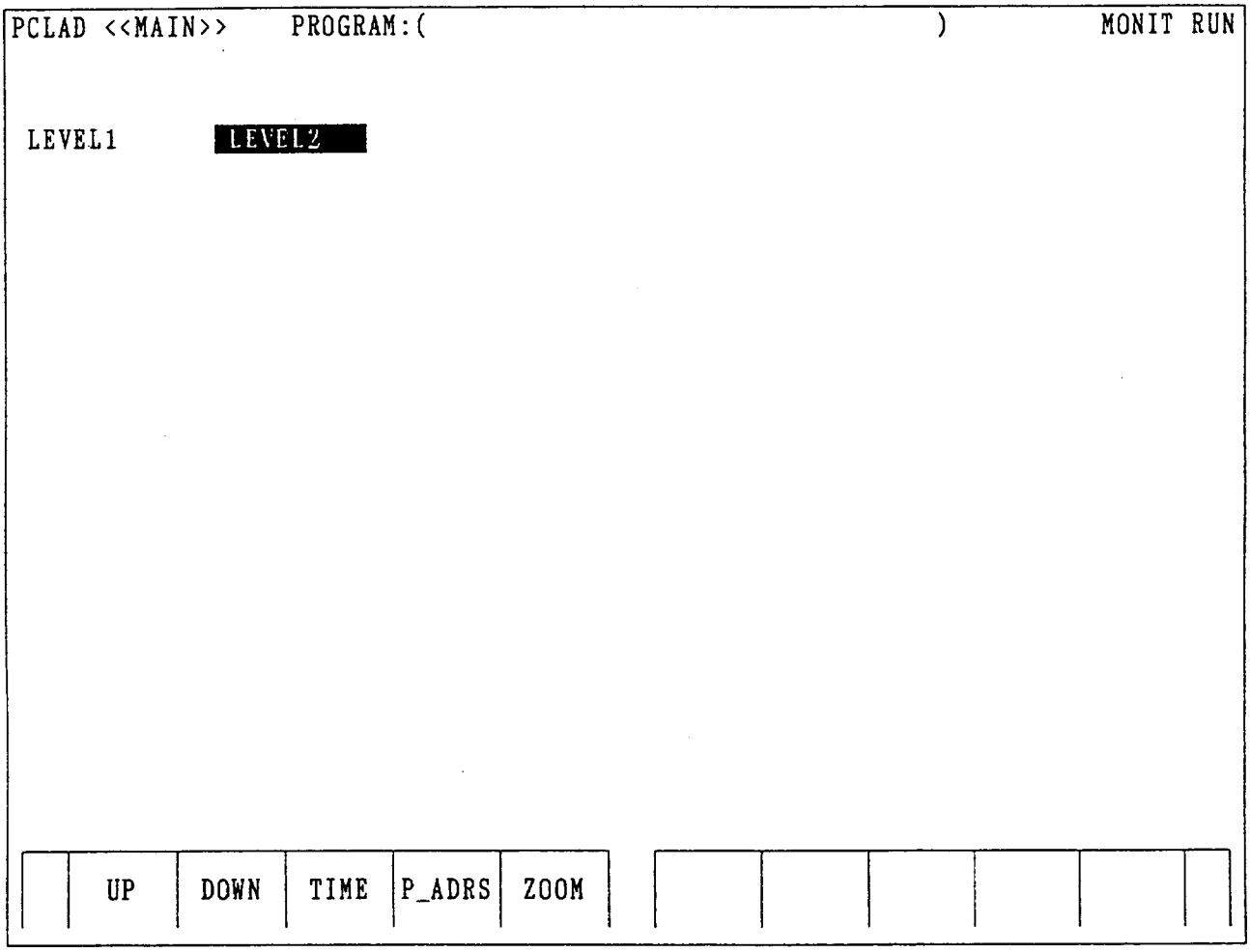


Screen 1

(Note 1) F1 may be displayed “PMCLAD.”

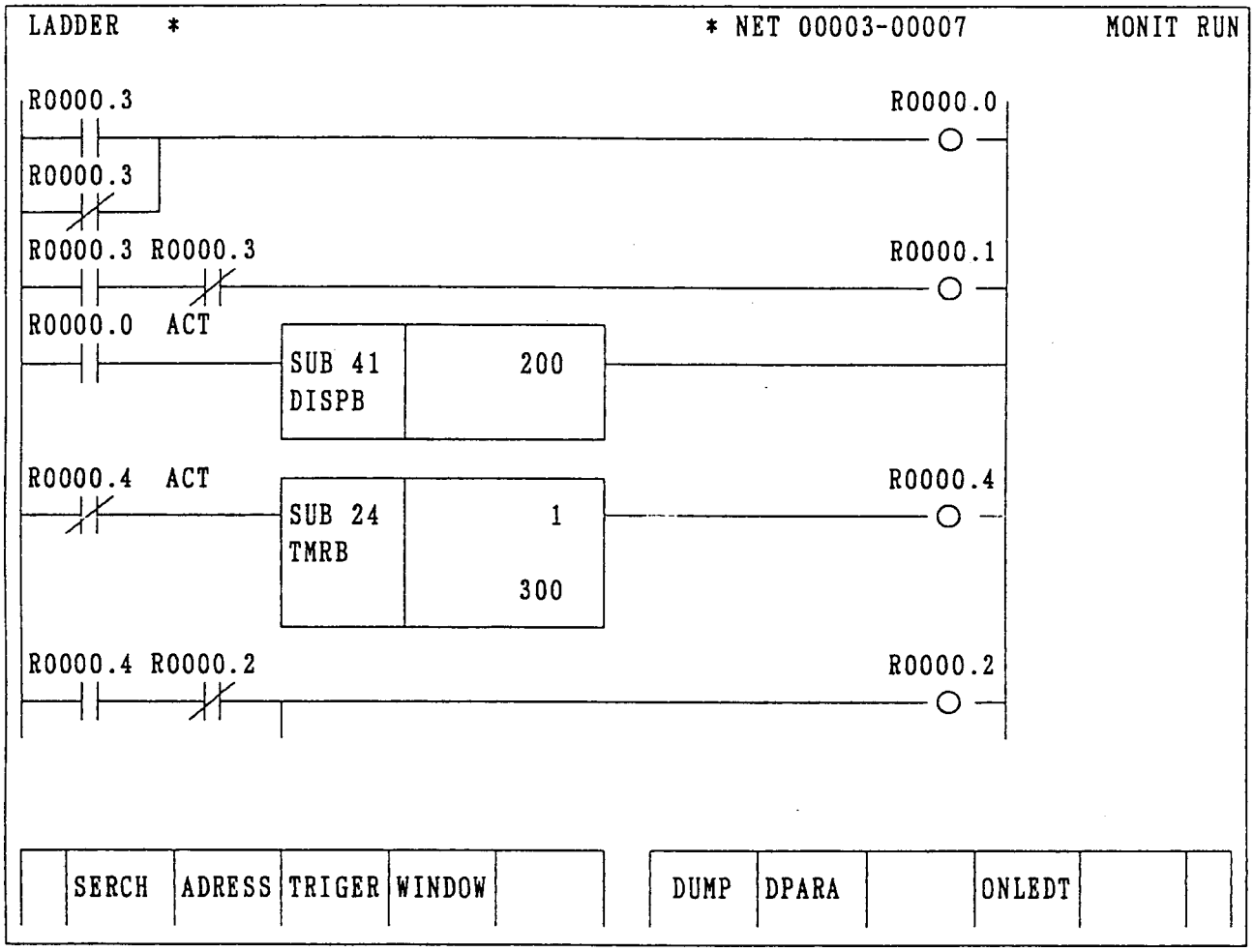
(Note 2) The portion marked ※ is not displayed depending on the setting of PMC.

Push F1/STPSEQ to display Screen 2.



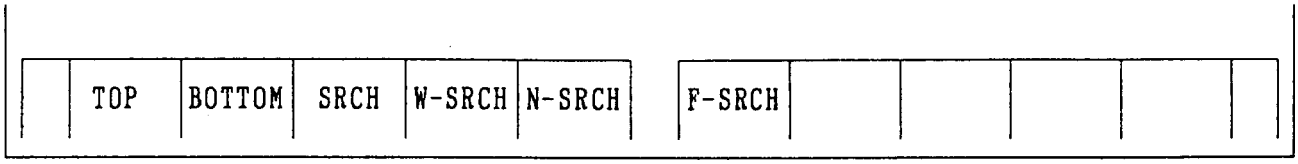
Screen 2

Move the cursor to LEVEL2 and push F5/ZOOM to display Screen 3 (Dynamic Ladder Display Screen).



Screen 3

Push F1/SERCH to display the functions at the screen bottom as in Screen 4:



Screen 4:

Description of Function Keys Relating to Search:

F1/TOP : Jumping takes place to the ladder beginning.

F2/BOTTOM : Jumping takes place to the ladder end.

F3/SRCH : Searching takes place concerning any specified coil or contact.
Input, with keys, an address and a bit number or a symbol name, and push F3/SRCH.

F4/W-SRCH : Searching takes place concerning any specified coil.
Input, with keys, an address and a bit number or a symbol name, and push F4/W-SRCH.

F5/N-SRCH : Jumping takes place to the ladder having the specified NET number.
The NET number generally represents the line number when counted from the beginning. Enter, with keys, an NET number to which jumping is made and push F5/N-SRCH.

F6/F-SRCH : Searching takes place concerning function instructions.
Enter, with keys, a function instruction name or function instruction number, and push F6/F-SRCH.

Of course, in addition to the above searching functions, the page key and the cursor key are always held effective.

To return to SEICOS screen, push CHANGE.

To return to the screen preceding, push ◁ OPER/MAINTE.

12. Program

With F2/PROGRAM being pushed, Program page (Fig. 12-0) appears.

Program search, editing, creation, deletion, display of an execution block, and background editing can be performed here.

・Most of the operations explained here are valid also in the overall screen programs.

On program execution, display is changed. Color (for monochrome)

Block having been executed (Fig. 12-0(1)): green

Block either in execution or to be executed (Fig. 12-0(2)): yellow inversion (inversion)

Block having been pre-read (Fig. 12-0(3)): yellow (light inversion)

Previous block (Fig. 12-0(4)): white

Block for next following executed or part for edition : white inversion (inversion)

For monochrome, depending on parameters, light/dark setting is useable for a block having been executed and for a pre-reading block.

Light/dark setting of pre-reading block : Parameter No.3119

Light/dark setting of block having been executed : Parameter No.3120

Program edit operation needs to satisfy the following conditions:

(1) Write key needs to be included. (Not required while in background editing.)

(2) Mode must be Edit. (Not necessary during background editing.)

(3) It must not be while auto operation start (or hold). (Not necessary during background editing.)

(4) No's O8000•'O9999 should not be included in Edit Inhibit by parameters.
(Edit Inhibit is made effective when parameter No.3107, #0 is equal to '0'.)

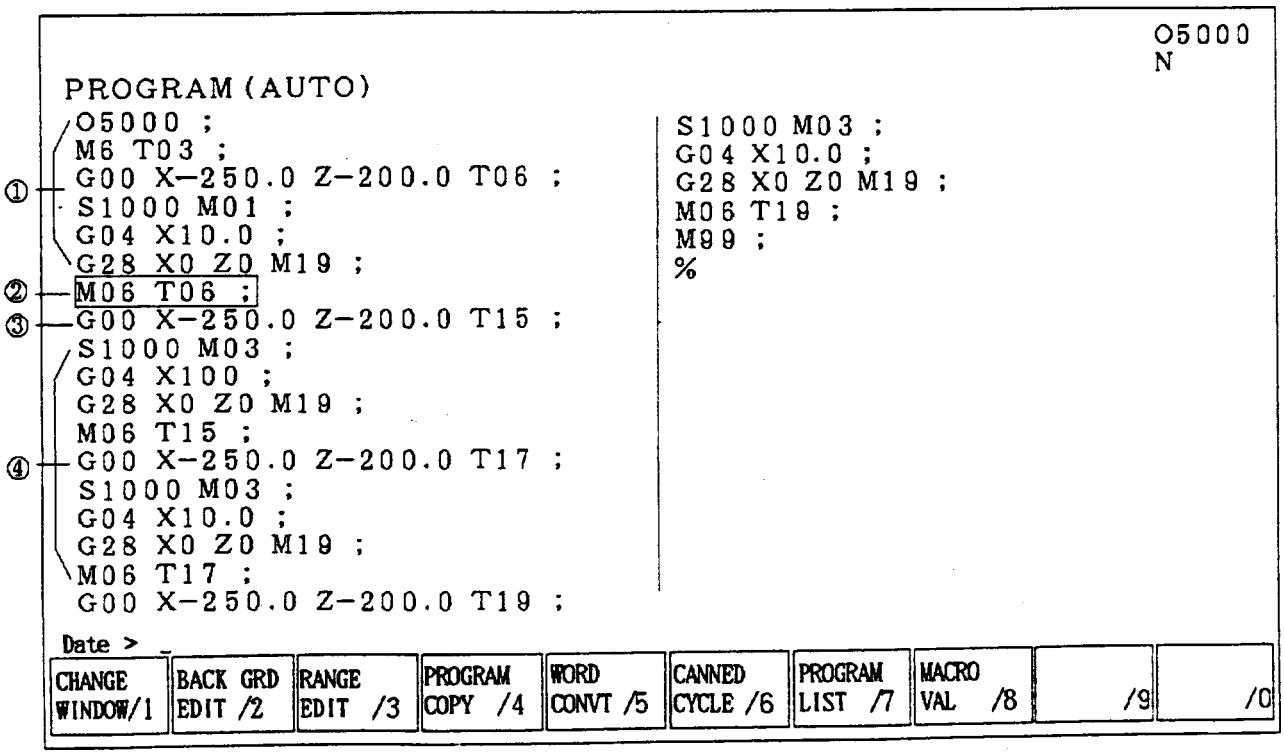


Fig. 12-0 Program

12-1 Operating Key Input Area On Screen

(1) Clearing Key Input Area

The key input area and the warning message are deleted.

1) Push [SHIFT] and, then, [CANCEL].

12-2 Displaying Program

(1) Changing Width of Program Display (Only applicable to Program page)

Display on the screen is switched from two-column display into one-column display (or vice versa).

1) Push [SHIFT] and, then, [ORIGIN].

(2) Changing Half-Em/Em Character Display

Display on the screen is switched from half-em into em (or vice versa).

1) Push [SHIFT] and, then, [HELP].

(a) Character type displayed on supply of power is determined by the parameter. When parameter No.3109, #0 is equal to '0', it becomes half-em.

12-3 Creating New Program

A new program is created.

A program immediately following creation consists of O, the input program No, ";" of line feed, and '%'.

1) Input [O].
2) Input the number for the program to be created.
3) Push [INSERT].

NOTE) When 0 exists at the beginning of the input number, the program number subject to creation and the program number displayed at the beginning of the program are not the same.

KEY INPUT	PROGRAM NO.	PROGRAM CONTENT
[O] [0] [0] [0] [1]	1	O0001;
[O] [2]	2	O2;

12-4 Deleting Program

(1) Deleting Program

One program is deleted. However, a program under Edit Inhibit or while in starting cannot be deleted.

1) Input [O].
2) Input the program number subject to deletion.
3) Push [DELETE].
4) "Delete? Y-Yes N-No" appears.
5) Push [Y] for deletion or any other key except [Y] not to delete.

(2) Deleting Two or More Programs

→ See 13-3 PROGRAM DELETION.

12-5 Initialization Of Program Memory

Deleting all the programs and initializing memory is called ODF.

1) Input O D F.
2) Push INPUT.
3) "ODF OK? Y-Yes N-No" appears.
4) Push Y to perform ODF or any other key except Y not to perform it.
   NOTE) Programs deleted in ODF cannot be recovered. Pay attention to its execution.

12-6 Program Search

(1) Searching Assigned Program

A program assigned is searched. However, a program under Display Inhibit cannot be searched.

1) Input O.
2) Input the program number subject to search.
3) Push any of ↑ ↓ → ←.
4) When it has been found, the program is displayed. When not, "NOT FOUND" message is displayed.

(2) Searching Before/After Displayed Program

Programs before/after the one in display are subject to search. When the program searched is under Display Inhibit, searching is continued until a program which can be displayed is finally found.

1) Input O .
2) To search a number larger than the one in display, push ↓.
   To search a number smaller than the one in display, push ↑.
3) A program found is displayed.

(3) Searching Out of Program list

→ See 13-2 PROGRAM SEARCH.

12-7 Program Copy

Contents of a program in display are copied into another program.

1) Input O.
2) Input the program number subject to copying.
3) Push INPUT.
4) The program copied is displayed.

12-8 Changing Program Number

A program in display is changed into another program.

1) Shift the cursor to the program number.
2) Input O.
3) Input the program number to be changed.
4) Push ALTER.
5) Delete the program in display and the new program is displayed.

12-9 Program Editing (Edit Mode or Background Editing)

12-9-1 Cursor Operation

The cursor normally displays a word in reverse video, which can move as follows:

(1) Moving in Word Unit

Cursor is moved with [→] or [←]. When it comes to the end of the screen, display is scrolled.

(2) Moving in Block Unit

With [↓] or [↑], the cursor is moved. When it is at the end of the screen, display is scrolled.

NOTE) When a character exists in the key input area, Word Search is judged.

→ See 12-10 WORD SEARCH.

(3) Switching Page

With [PAGE↓] or [PAGE↑], page is changed. The cursor appears at the beginning of the page.

(4) Moving to Beginning of Program

With [SHIFT] zand, then, [PAGE↑]pushed, the cursor is moved to the program head.

(5) Moving to End of Program

With [SHIFT] and, then, [PAGE↓]zpushed, the cursor is moved to the program end.

12-9-2 Edit Operation

Use, at the cursor position, [INSERT] , [ALTER] , and [DELETE] for editing.

(1) [INSERT]

Insert, after the cursor, the content of the key input area. When insertion takes place, the cursor moves to the last work having been inserted.

(2) [ALTER]

The part inversely displayed by the cursor is replaced by the content of the key input area. The cursor does not make a move.

(3) [DELETE]

The part inversely displayed by the cursor is deleted.

NOTE 1) Background editing cannot deal with the same program that can apply to ordinary editing. Choose ordinary editing.

→ See 12-11 BACKGROUND EDITING.

12-10 Word Search

An assigned number (numerical value) or work (character string) is searched.

Whether searching for a word or for a number is determined by input.

(1) Number Search

Number search includes searching with input data which consists of one alphabet (or '#') and numerals. Numerical values are recognized for identification as those with a decimal point and integers without it. Note, therefore, "X1." and "X1000" which indicate the same travel amount are recognized differently. However, "X0." and "X0." are not recognized differently.

(Example) When searching with “N10”:
N10, N010, N00010, etc.
(Example) When searching with “X0.1”:
X0.1, X0.100, X00.10, X.1, X.100, etc.

(2) Word Search

Character string search includes all the others except number search.

1) Input data for searching.
2) Assign the search direction with the cursor.
   [↑] Searching conducted before the cursor.
   [↓] Searching conducted after the cursor.
3) When the target word is found, the cursor moves there. When it is not found, “NOT FOUND” message is displayed.
   ・To halt searching, push any one key.

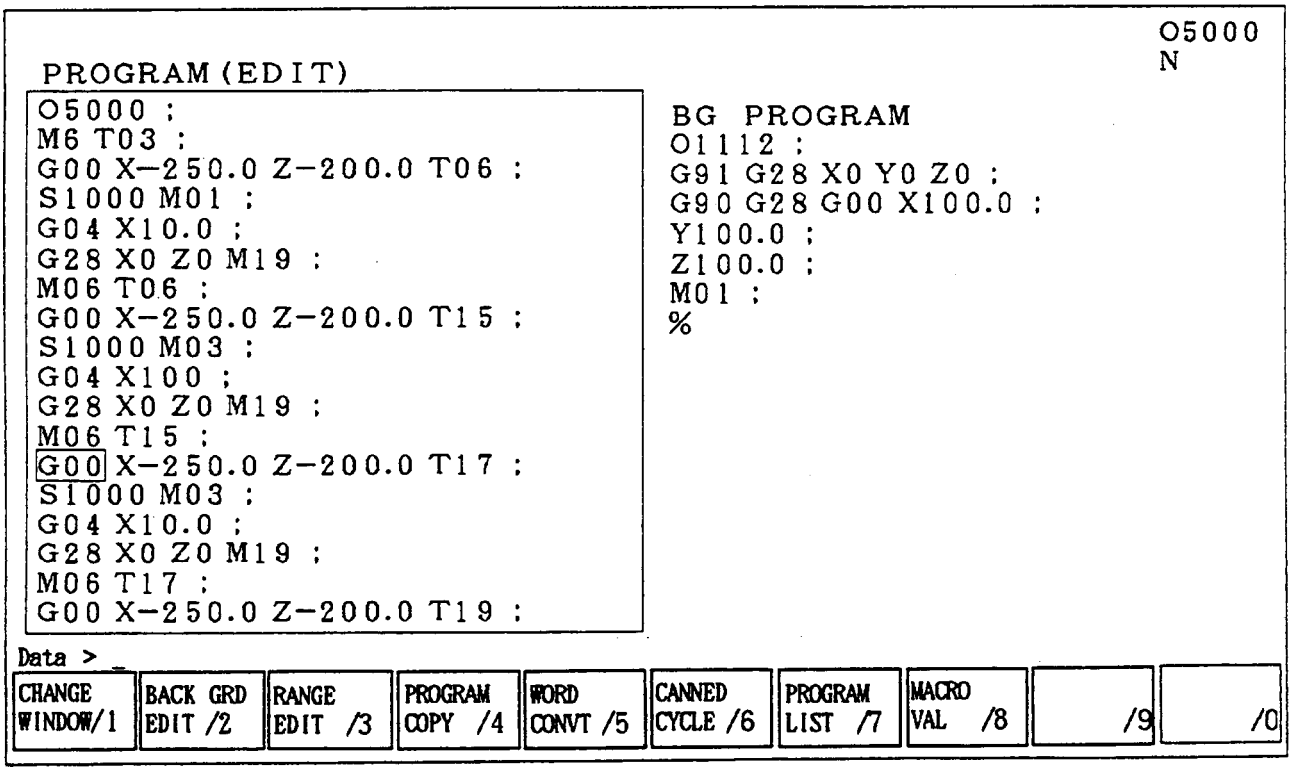
12-11 Background Edit (Program Screen Only)

Normally, it is impossible to observe a program while it is in execution. Background editing has enabled editing at this stage.

The background editing operation is completely the same as that of ordinary editing. NC execution is not affected when a program is searched, a cursor moved, or a page turned. The programs here become those dedicated to background edit where NC execution is not available. Therefore, it is possible neither to edit the program being executed nor to execute the program being background edited.

(1) Switching Screen

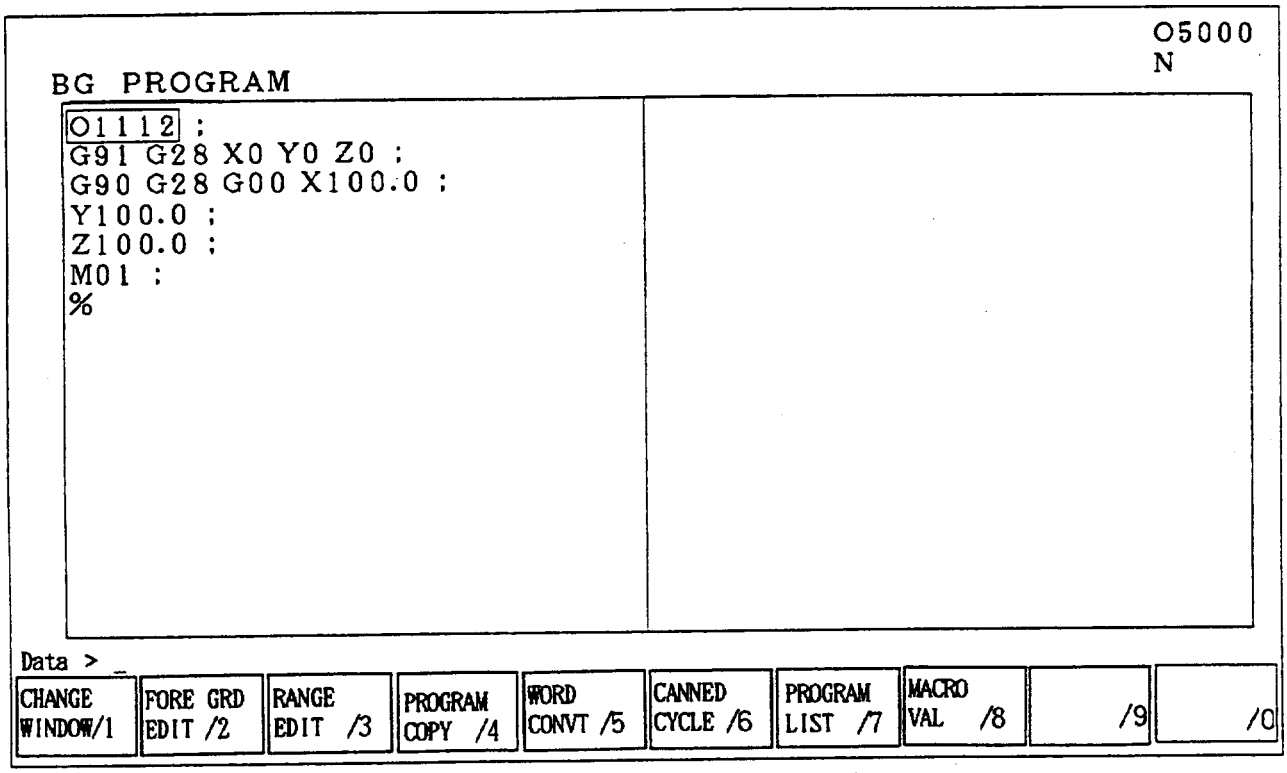
The ordinary edit programs are displayed along with the background edit progarams. With F1/CHANGE WINDOW being pushed, the ordinary edit programs are displayed on the left side and the background edit programs on the right side. The programs editable are, then, put in a frame. With one more push, the editable programs are displayed on the full screen.



NOTE) The programs put in a frame indicate those editable.

(2) Selecting Edit Program

With F2/BACK GRD EDIT pushed while in ordinary edit, background edit is selected. And, with F2/FORE GRT EDIT being pushed while in background edit, ordinary edit is selected. When F1/CHANGE WINDOW has been operated, the frame indicating available programs makes a move.



- Ordinary edit is differentiated from background edit by its title.
  Ordinary edit uses “Program” for its indication, while background edit uses “BG Program”.
  When either ordinary edit or background edit alone is in display, one without a frame is ordinary edit and the other with it is background edit.

12-12 Range Assignment Edit Operation (Program Screen Only)

Two or more blocks of a program in display are collectively deleted or copied into another program.

(1) Starting Range Edit

Range editing is started.

1) Push F3/RANGE EDIT.

Function menu changes into the one for range editing.

(2) Assigning Range

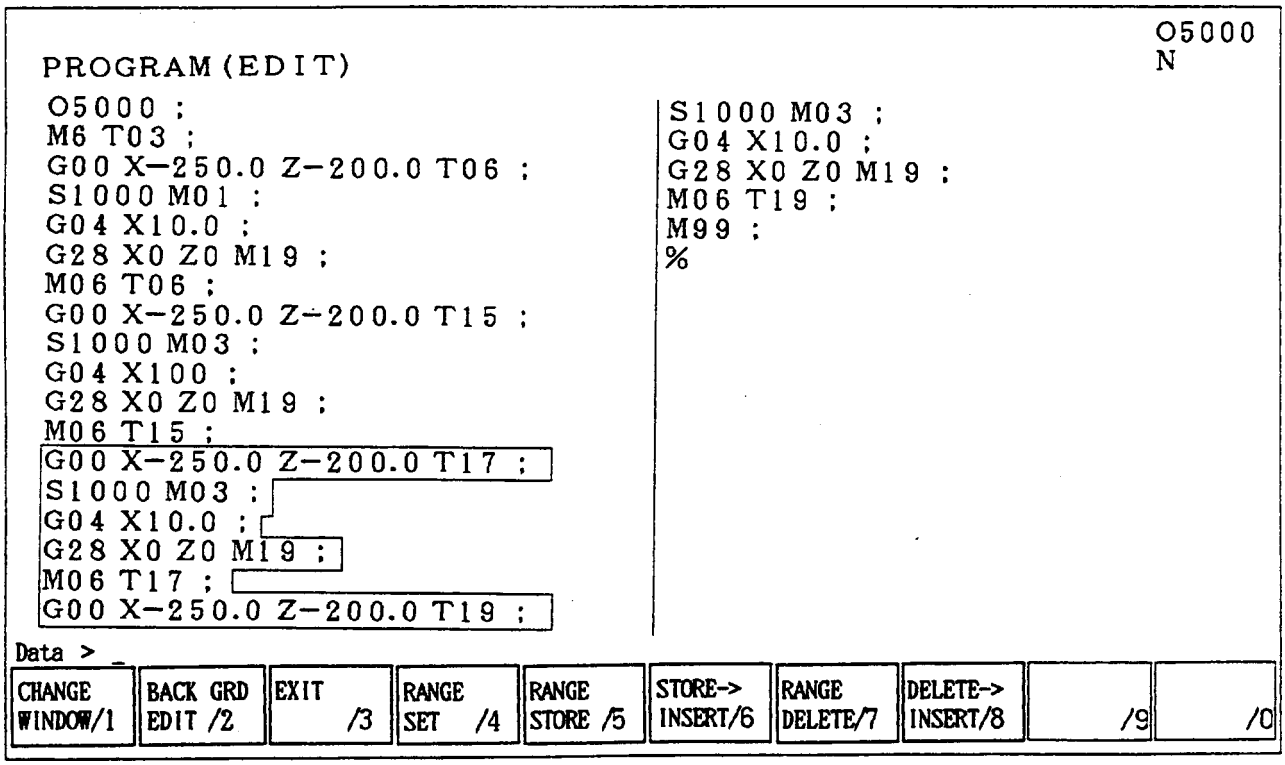
Assign a program for range editing.

1) Push F4/RANGE SET.

The cursor changes into the framed one.

2) Assign a range.

As in ordinary cursor shifting, a range for the framed cursor can be extended with the cursor key or the page key. The area inside the frame indicate programs under range assignment.



NOTE) The block inside the frame indicates the scope assigned. Assignment can extend over two or more pages.

(3) Cancelling Range Assignment

Range editing is interrupted. Range in assignment is made invalid.

1) Push F4/RANGE SET or F3/EXIT.

A framed cursor is changed into an ordinary cursor, cancelling a range.

(4) Storing Range

A program having been assigned in range is stored.

1) Push F5/RANGE STORE.

A program indicated by the framed cursor is put in memory, which is stored until power is cut off. However, only the last block subject to range assignment can be stored.

・Maximum 4096 characters (approx. 10m long) can be stored. When a range exceeds this, “Range Too Large” appears and no storage takes place. Re-assign a range.

(5) Inserting Range

A range stored program is inserted immediately after the cursor.

1) Shift the cursor to the insertion place.

2) Push F6/STORE → INSERT.

The part having been stored in Range Store is inserted after the cursor. The cursor position stays the same.

(6) Deleting Range

A range assigned program is deleted.

1) Push F7/RANGE DELETE.

A program enclosed with the framed cursor is deleted. A program having been deleted is stored until power is cut off. Only the block having been lastly subject to range deletion can be stored.

・Maximum 4096 characters (approx. 10m long) can be stored. When a range exceeds this, “Range Too Large” appears where no storage takes place. Re-assign a range.

(7) Recovering Deleted Data

A range deleted program is inserted immediately after the cursor.

1) Shift the cursor to the insertion place.

2) Push F8/DELETE → INSERT.

The part having been stored in Range Delete is inserted immediately after the cursor. The cursor position stays the same.

NOTE) Always remind that insertion takes place after the cursor.

12-13 Word Convert (Program Screen Only)

Words not yet being converted are searched in a program, which are rewritten into the converted words. Converting methods include the following two:

(1) The applicable words are searched one by one which are converted as being confirmed.

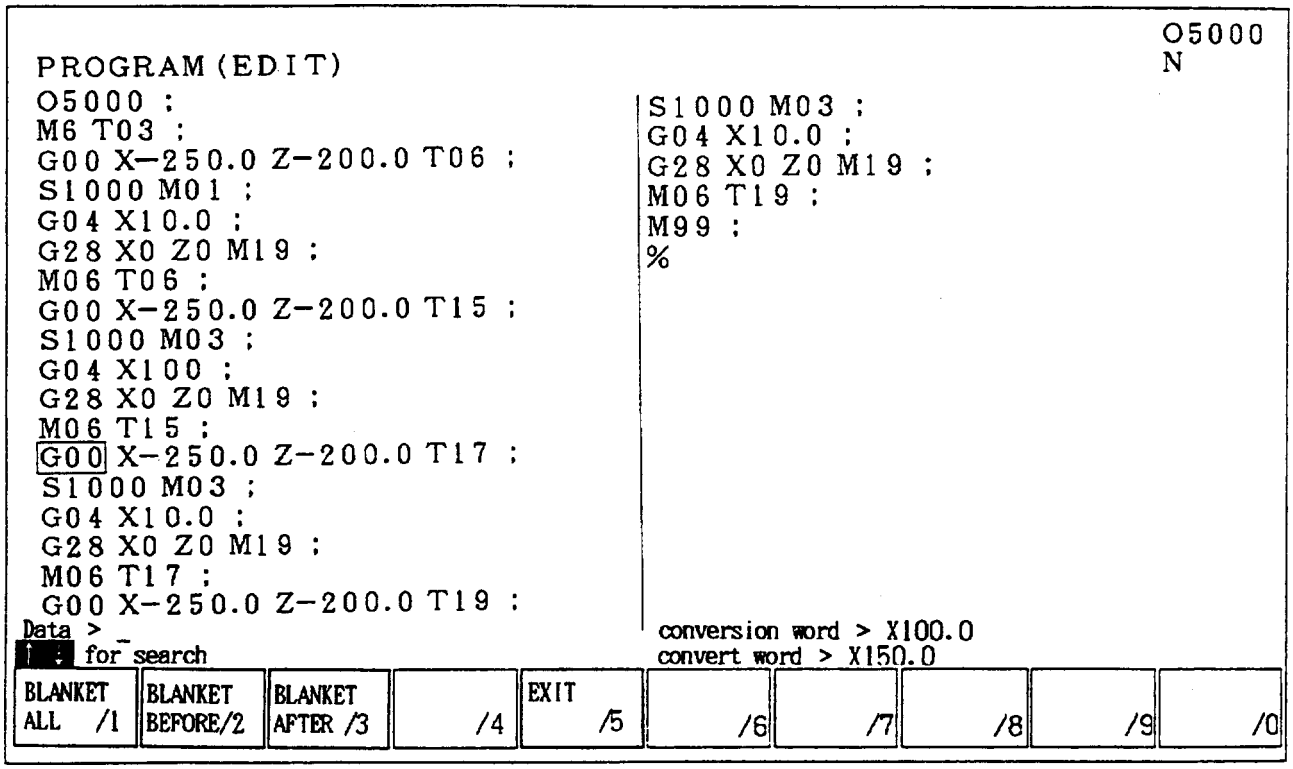
(2) The applicable words are converted collectively while the conversion state is displayed.

As the word searching system is the same as that for Word Search, it is either Number Search or Word Search depending on pre-converted words.

→ See 12-10 WORD SEARCH.

For example, conversion from “X.1” into “X.5” numerically corresponds each other. Those applicable to conversion includes character strings such as “C0.1”, “X0.100”, “X00.10”, “X.1”, “X.100”, etc., all of which are each converted into “X.5”.

1) Pushing [F5/WORD CONVT], start Word Conversion.
2) Input, with keys, the word before being converted and push [ENTER].
3) Input, with keys, the word after being converted and push [ENTER].
   If [INPUT] is pushed without any key input for the word after conversion, the word before conversion is deleted.



4) Assign a search method for conversion.
   (a) For individual search:
   - Indicate, with [↑] / [↓], the search direction from the cursor position.
   - When a pre-converted word has been found, “Convert? Y-Yes N-No” appears.
   - To convert, push [Y]. Not to convert, push any other except [Y].
   - Repeat the above steps until searching is ended.

   (b) For collective convert:
   - Indicate the search direction on Function menu.

F1/BLANKET ALL	Regardless of cursor position, searching starts with the program head.
F2/BLANKET BEFORE	Searching takes place in the forward part following the cursor including the word with the cursor.
F3/BLANKET AFTER	Searching takes place in the backward part preceding the cursor including the WORD WITH THE cursor.

- “OK? Y-Yes N-No” appears.
- To collectively convert here, push [Y]. If not, push any other except [Y].

5) On completion of conversion, the number of words having been converted is indicated in the message “x words have been converted.”
   - Word conversion, once started, lasts till it reaches the beginning or the end of a file. To stop halfway, push [F5/EXIT] .

12-14 Macro Variables

Macro variables are displayed.

→ See 6 MACRO VARIABLES.

13. Program List

The program list (Fig. 13-0) is displayed.

Not only reading the list, you can also perform searching and deletion of a program.

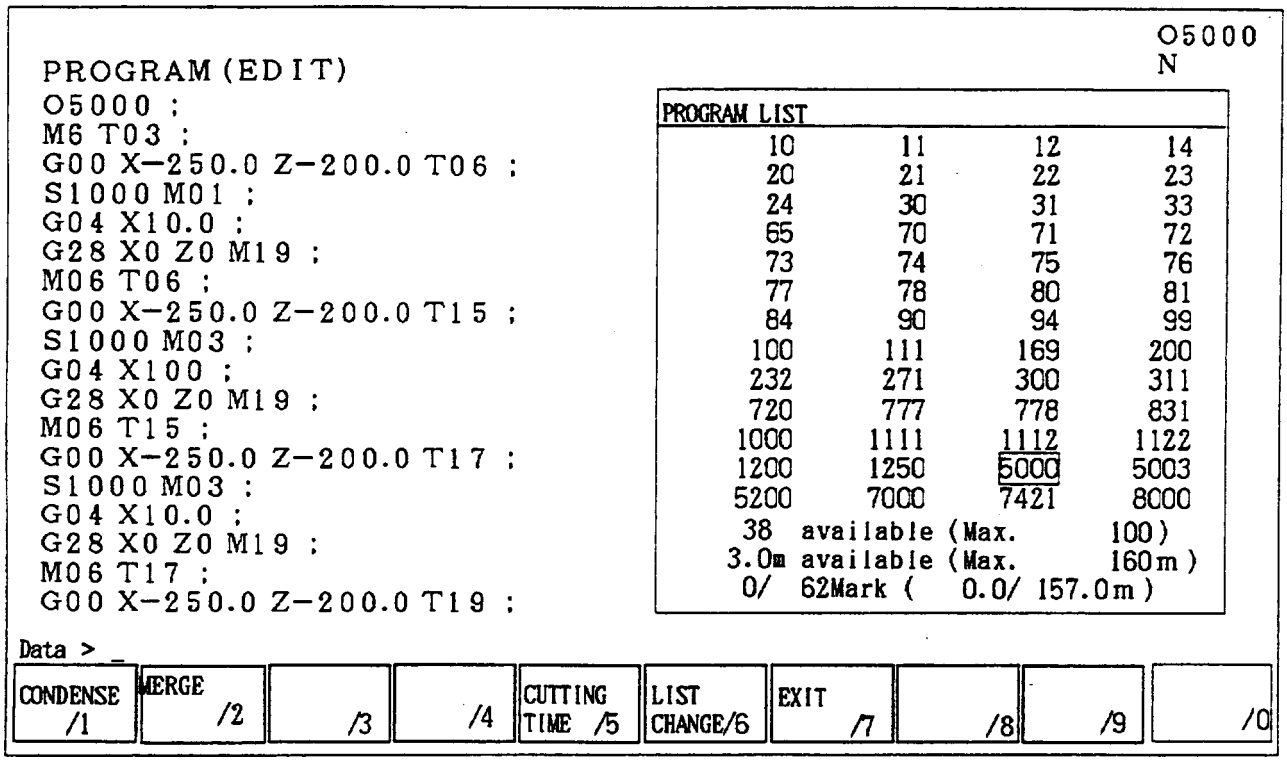


Fig. 13-0 Program List

13-1 Program List

With F7/ PROGRAM LIST pushed on Program page, the program list is displayed. With one more push, the list is erased.

As a display position differs by display method, the same operation applies.

When a list is displayed on the full screen, (Fig. 13-0) appears on the right side on the screen. When not, (Fig. 13-1) appears on the side opposite of the selected page.

Also, as the max. number of programs for registration and the memory amount still left unused are displayed, use them for program input and editing.

The cursor is moved with ↑ ↓ ← → Page Home.

Program List is available in two different kinds including the program list which displays only program numbers and Program Detailed Display (Fig. 13-2) which displays comment and size of each program. Push F6/LIST CHANGE to switch them.

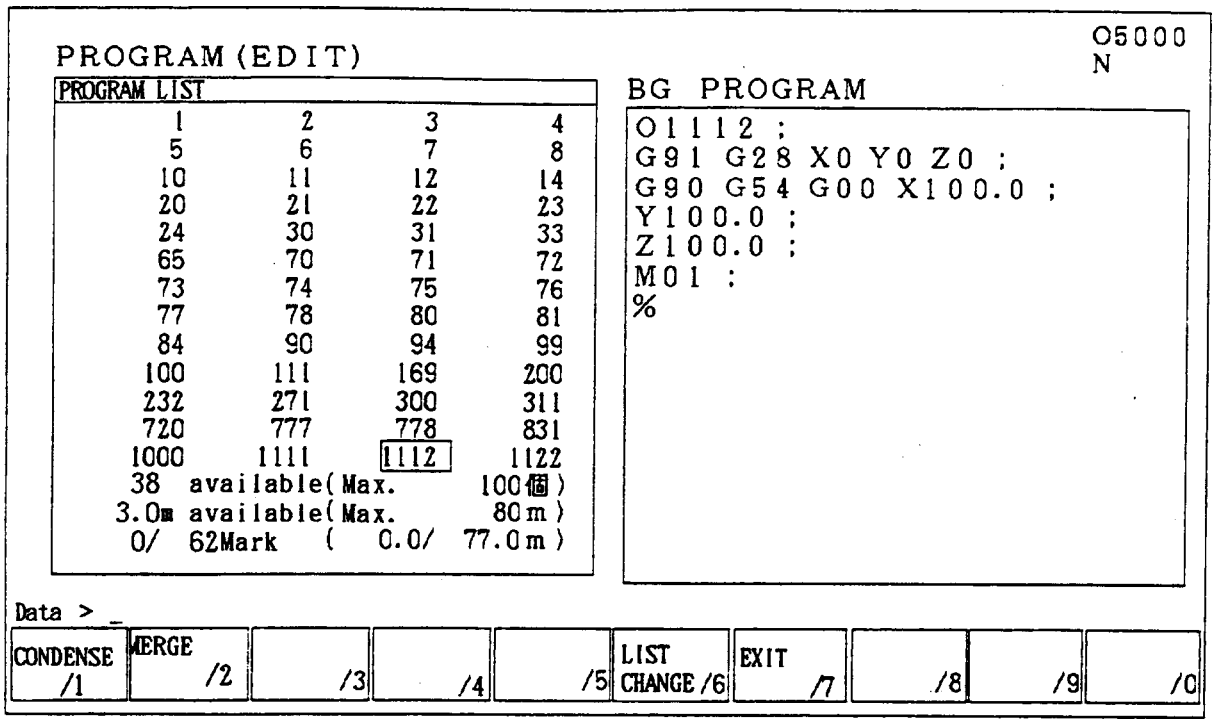


Fig. 13-1 Program List for Background Edit

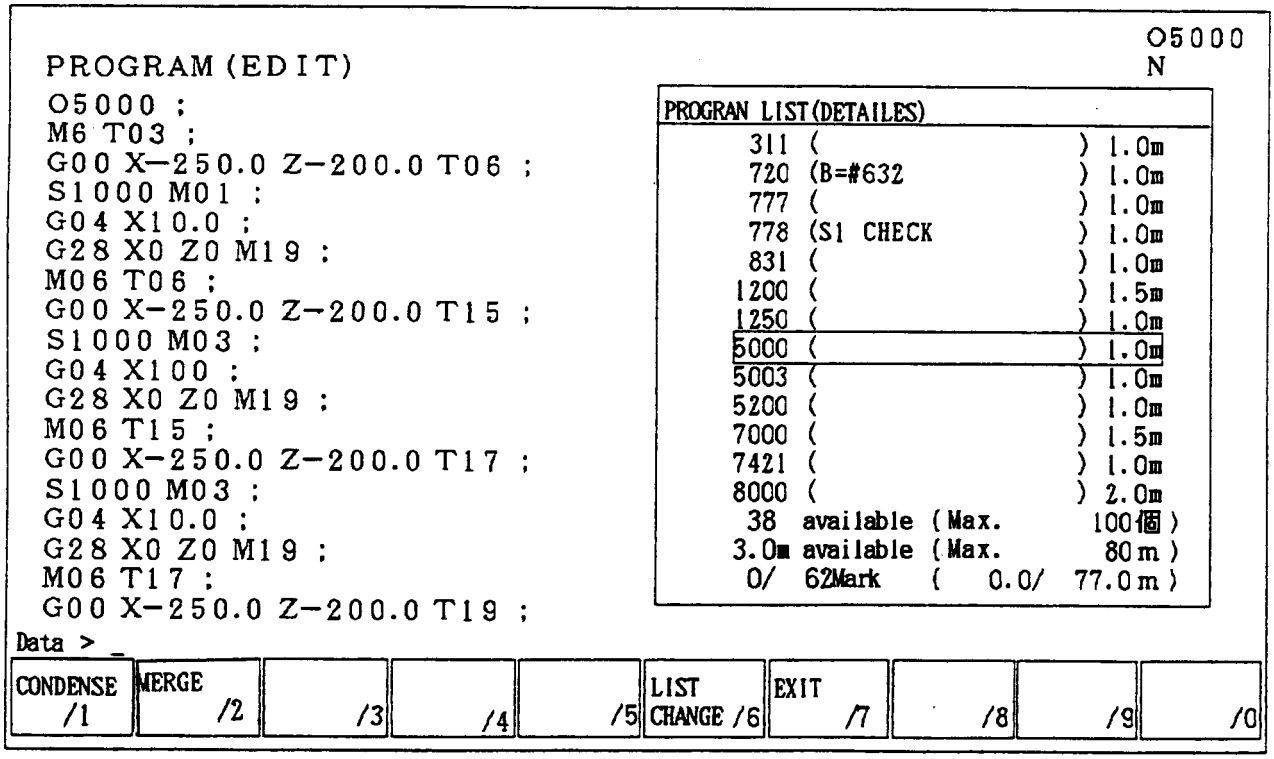


Fig. 13-2 Program Details

The comment for the program detailed display indicates the one immediately following O number in a program. If not existing, no display takes place. Max. 16 characters can be displayed. The size for output is 0.1m each. However, depending on memory control, it may slightly differ from the actual tape length.

13-2 Program Search

Program search can be performed as follows while Program List or Program Detail is in display.

(1) Searching with Program List.

1) Empty the key input area.
2) Move the cursor to the program number subject to search.
3) With [INPUT] pushed, a program is changed over.

(2) Searching by Key Input

1) Input "[O]".
2) Input the program number subject to search.
3) Push one of [↑] [↓] [→] [←].
4) On finding, the program is displayed. If not, "NOT FOUND" message appears.

13-3 Program Delete

Program deletion can be performed as follows while Program List or Program Detail is in display, provided that the following conditions are satisfied.

(a) Write key should be included.
(b) It is in Edit mode. (Not necessary while in background editing.)
(c) It is not in auto operation start (or halt). (Not necessary while in background editing.)
(d) For O8000•'O9999, it should not Edit Prohibit.

(1) Deletion from Program List

A program at the cursor position on the program list is deleted.

1) Empty the key input area.
2) Shift the cursor to the program number subject to deletion.
3) Push [DELETE].
4) "Delete? Y-Yes N-No" appears.
5) Push [Y] for deletion. If not, push any other key except [Y].

(2) Collective Deletion by Program Selection

A program selected on the program list is deleted.

1) Select a program to be deleted and put '*' in front of the number.
   To select the all programs, push [ORIGIN]. To select them one by one, move the cursor to a program to be selected and push [SPACE].
   To cancel it, push either ORIGIN or SPACE.
2) Push [DELETE].
3) "Delete? Y-Yes N-No" appears.
4) To delete, push [Y]. If not, push any other key except [Y].

NOTE 1) Before deleting, search the program and confirm program content.
A program having been deleted cannot be recovered.

NOTE 2) When a program in display has been deleted, the program of the next following program number is displayed.

13-4 Program Arrangement

In the course of program editing, program size may get larger than the actual size. When this occurs, a program can be adjusted so that useable memory is slightly increased. In this case, however, the following conditions need to be satisfied:

(a) It is not background editing.
(b) It is in Edit mode.
(c) NC is in reset state.
  1) Push F1/CONDENCE.
  2) "PROGRAM ADJUST IN EXECUTION" comment is displayed.
  3) On completion of program adjustment, "PROGRAM ARRANGE OVER" appears.

NOTE 1) Program adjusting takes several seconds~several minutes depending on the number of programs.

NOTE 2) When power is cut off while in "PROGRAM ADJUST IN EXECUTION", a program is destroyed. On occurrence of an abnormality in a program, initialize program memory and re-input the program.

13-5 Program Merging

Following the program displayed by the program list, another program indicated by the cursor on the program list can be merged.

1) Search the program to which a program is merged.
2) Shift the cursor to the program to be merged on the program list.
3) Push F2/MERGE.
4) "Merge? Y-Yes N-No" appears.
5) To merge, push Y. If not, push any other key except Y .
6) Start merging.
7) On ending, the program is displayed all over again from the beginning. At the merging section, the numbers in the O-number of the merged program alone are left behind.
8) "Delete? Y-Yes N-No" appears.
   To delete the program merged at the end, push Y. To leave it, push any other key except Y .

13-6 Initialization Of Program Memory

Deleting all the programs to initialize memory is called ODF.

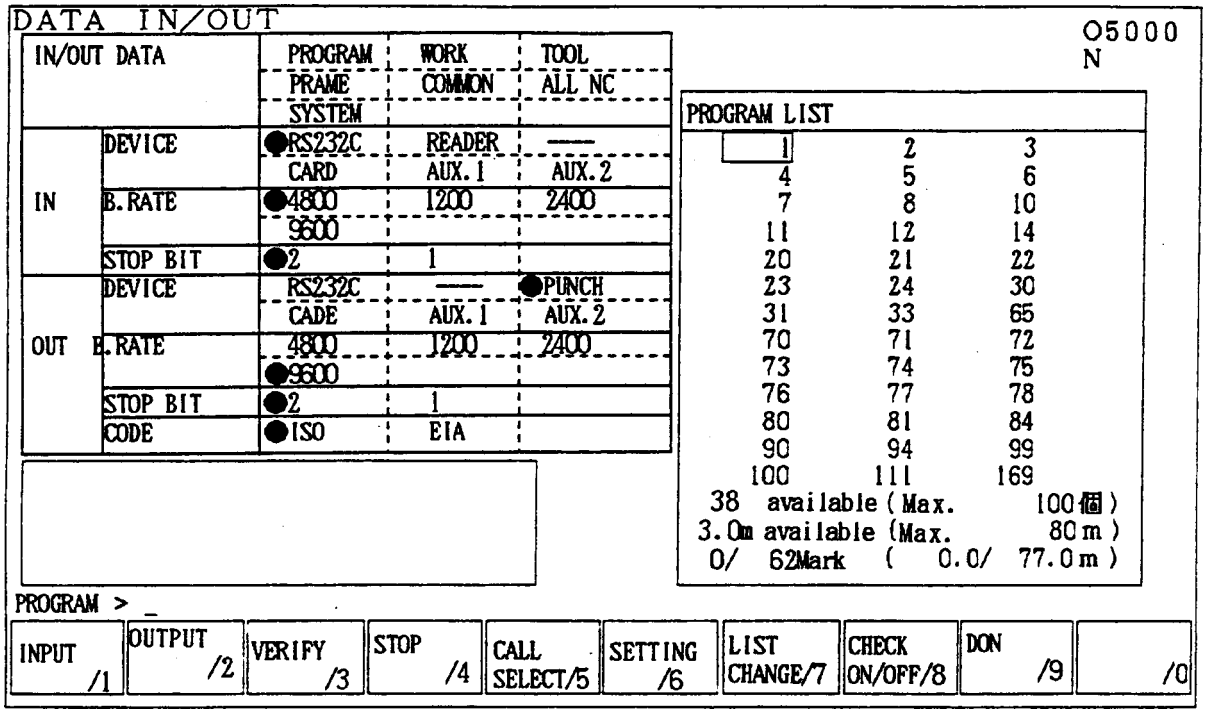
1) Input O D F.
2) Push INPUT.
3) "ODF? Y-Yes N-No" appears.
4) For ODF, push Y . If not, push any other key except Y .

(NOTE) A program having been deleted in ODF cannot be recovered. Be careful to execute correctly.

14. Input / Output

With F8/IN/OUT pushed, the data I/O screen is displayed.

All the operations relating to I/O such as tape input and punch-out are performed here. The program list is displayed on the right side of the page, the data setting table (various data setting) on the left, and the monitor screen at the bottom left-hand corner. As I/O state being displayed on the monitor screen, use it to confirm operation and to check collation errors.



(1) Input

Input the external unit data into NC main body. ISO/EIA information of the input data are automatically identified.

1) Connect the external unit.
2) Set “INPUT” item on the data setting table to the setting of the external unit.
3) Set I/O contents of the data setting table.
4) Set the external unit to Output Standby state.
5) With F1/INPUT pushed, input is started.

(2) Output

Data of the NC main body are output into the external unit.

1) Connect the external unit.
2) Set the “OUTPUT” item on the data setting table to the setting of the external unit.
3) Set I/O contents of the data setting table.
4) Set the external unit to Output Standby state.
5) With F2/OUTPUT pushed, input is started.

(3) Verification

Data are verified between the external unit and the NC main body.

Verification cannot be made depending on data. See applicable data.

1) Connect the external unit.
2) Set “INPUT” item on the data setting table to the setting of the external unit.
3) Set I/O contents of the NC main body.
4) Set the external unit to Output Standby state.
5) With F3/VERIFY , collation is started.

(4) Interrupt
Input, output, and collation are interrupted.

(5) Cursor Move Between Data Setting Table and Program List
Only one cursor exists on the screen. With F6/SETTING being pushed when the cursor is on the program list, it moves to the data setting table. With F6/PROGRAM LIST being, pushed when it exists on the data setting table, it moves to the program list.

14-1 Program List

This is basically the same as the program list on Program screen. With F7/LIST CHANGE being pushed, display is switched between Program List and Program Details.

(1) Selecting Program (Used to input/output a program.)
1) Move the cursor to the program to be selected.
2) Push SPACE.
The ‘*’ is attached at the beginning of the program having been selected. For Select/ Cancel all programs, push ORIGIN.

(2) Deleting Program on List
→ See 13-3 PROGRAM DELETE.

(3) Initializing Program Memory
→ See 13-6 INITIALIZATION OF PROGRAM MEMORY.

14-2 Data Setting Table

This serves to set I/O contents as well as to perform setting of an external unit on input/output.

14-2-1 NC Data

Input/output a program. Select [NC DATA]. Display in the key input area changes into (NC DATA)=.

(1) Input
Reading starts with EOB code on a tape, which continues up to % code. Use an O-number on a tape to register a program number.
- When the same number has already been registered or no O-number exists on a tape, no registration can be performed. However, input can be performed with parameter setting.

(a) With Parameter No.3106, #0 set to ‘1’, delete all the programs and input newly.
(b) With Parameter No.3106, #0 set to ‘1’, the formerly input number is deleted when the same number has entered and new input can be enabled.
- With a program No. having been input with keys followed by input operation, the initial program changes into the one having been input. The second programs on are as specified on the tape.

(2) Output

A program having been input with keys and one having been selected on File List are output. To output all the programs, input “O-9999”.

(3) Verification

On occurrence of any disagreement in verification, verification is stopped and the tape contents where disagreement has taken place is shown in reversing display.

14-2-2 Coordinate System

Work compensation data are input/output by the codes written in G10 format.

Select [WORK SYSTEM]. Indication in the key input area becomes (WORK SYSTEM)=.

All work compensation data are input/output.

14-2-3 Tool System

Tool compensation data are input/output by the codes written in G10 format.

Select [TOOL SYSTEM]. Indication in the key input area becomes (TOOL SYSTEM)=.

You can assign any tool compensation data at the time of input. When no assignment exists, all tool compensation data are output.

1) Input [T].
2) Input a tool compensation number subject to output.
   (a) To output one by one,
   Input [;].
   (b) To output continuously,
   Input [-] and input the tool compensation number to be output at the end.

[EX.] [T] [1] [2] [;] [T] [2] [0] [;]

No. 12 and 20 are assigned.

[T] [2] [0] [-] [T] [3] [0]

No. 20~30 are assigned.

NOTE) One-by-one output and continuous output cannot be assigned at the same time.

14-2-4 Parameters

Parameters are input/output with the codes written in G10 format.

Select [PARAMETER]. Display in the key input area becomes (PARAMETER)=.

With a parameter assigned, the SLBUS table is input/output together with the system table. The SLBUS table is expressed in No.20000 and the system table in No.30000.

You can assign any parameter for output. With no assignment, all the parameters are output.

1) Input [N].
2) Input a parameter number subject to output.
   (a) To output one bye one,
   Input [;].
   (b) To output continuously,
   Input [-] and, then, input a parameter number to be output at the end.

[EX.] [N][1][2][;][N][2][0] ;

Parameter No.12 and 20 are assigned.

[N][-][9][9][9] 9

Parameter No.0~9999 are assigned.

[N][2][0][0][0][0][-][N][2][9][9][9] 9

The SLBUS table are assigned.

NOTE) One-by-one output and continuous output cannot be assigned at the same time.

14-2-5 Common Variables

Common variables are input/output in macro language.

Select [COMMON VARIABLE]. Display in the key input area gets (COMMON VARIABLE)=. All the common variables are input/output.

As numerical values are rounded to 8 digits for output, collation may not be taken properly in some cases.

14-2-6 Setting External Units

Set this to inform the NC main body of setting of an external unit used for input/output. When the NC main body does not correspond to the external unit setting, input/output is not correctly performed. Set the NC main body by using parameters.

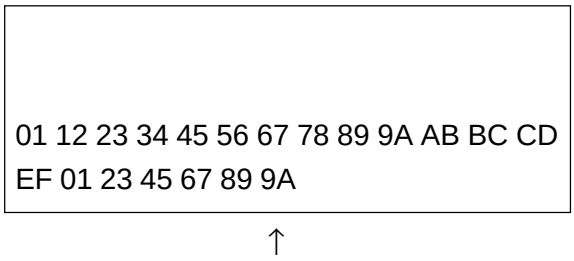
→ See 14-4 DETAILS OF SETTING DATA.

14-3 Check

With [F8/CHECK ON/OFF] pushed, communication status is displayed by the side of the monitor screen and a port number in connection below the input/output equipment of the data setting table.

Also, character data of the monitor screen are not converted for input in codes, but are displayed in hexadecimal.

Since processing speed is reduced, however, do not use this while in execution of input/output.



DR[ON] ← Circuit connection status

ER[FOP] ← Error status

F: framing erro

O: overrun error

P: parity error

↑

Receiving data are displayed without change. (input only)

14-4 Details Of Setting Data

These are details of data set in the data setting table and each parameter concerning I/O.

14-4-1 Name Of I/O Equipment

Names corresponding to I/O equipments are set in 1~6 of the following table.

PRA0132 : Input equipment

PRA0133 : Output equipment

PRA0135 : Macro external output command equipment

1	RS232C
2	Reader
3	Puncher
4	Card
5	Auxiliary 1
6	Auxiliary 2

Setting is RS232C for those other than the above.

14-4-2 Setting Of Each Equipment

Port numbers, baud rates, stop bits, presence or absence of DC code, data lengths, and parities are set.

(a) Setting of Baud Rate

Baud rates are set in 8~15 of the table.

8	1200
11	2400
13	4800
15	9600

Setting is 4800 for those other than the above.

(b) Setting of stop bits, presence or absence of DC code, data length, and parities.

0: Stop bit (0: 2 bits 1: 1 bit)

1: DC code (0: Used 1: Not used)

2: Data length (0: 8 bits 1: 7 bits)

3, 4: Parity (00: None 10: None 01: EVEN 11:ODD)

(1) Setting of RS232C (Equip. No.1)

Port No. : PRA0124

Baud rate : PRA0116

Parity, etc. : PRA0104

(2) Setting of Reader (Equip. No.2)

Port No. : PRA0125

Baud rate : PRA0117

Parity, etc. : PRA0105

(3) Setting of Puncher (Equip. No.3)

Port No. : PRA0126

Baud rate : PRA0118

Parity, etc. : PRA0106

(4) Setting of Card (Equip. No.4)
Port No. : PRA0127
Baud rate : PRA0119
Parity, etc. : PRA0107

(5) Setting of Auxiliary 1 (Equip. No.5)
Port No. : PRA0128
Baud rate : PRA0120
Parity, etc. : PRA0108

(6) Setting of Auxiliary 2 (Equip. No.6)
Port, etc. : PRA0129
Baud rate : PRA0121
Parity, etc. : PRA0109

14-4-3 DC Code

Values for DC codes 1~4 are set here.
DC1 : PRA0112 (11H)
DC2 : PRA0113 (12H)
DC3 : PRA0114 (93H)
DC4 : PRA0115 (14H)
When a set value is 0, use values in ( ).

14-4-4 I/O Parameters

Codes, etc. used for data output are set in Parameter No.0100.
Bit
0 ..... TV check (0: No 1: Yes)
1 ..... TV check of control out (0: Yes 1: No)
2 ..... ISO code parity (0: Assigned 1: Not assigned)
3 ..... Output code (0: ISO 1: EIA)
4,5 ..... Output format of EOB in ISO code
(00: LF, CR, CR 01: LF, CR 10: LF 11: LF)
6 ..... Delimiting each word by space (0: No 1: Yes)
7 ..... DC3 output on completion of input (0: Yes 1: No)

14-4-5 Special Characters Of EIA Code

Bit patterns of special characters in EIA code are set.
PRA6004 ([), PRA6005 (]), PRA6006 (#), PRA6007 (*)
PRA60048 (=), PRA6009 (?), PRA6010 (@), PRA6011 (&)

14-4-6 Feed

Feed length and program interval length during program output are set. Unit used here is centimaters.
PRA0138 : Feed length
PRA-139 : Program interval length

14-5 Call Select (Option)

A program called out on the program list is prefixed by '*'.

(1) Retrieval Method

The method of retrieving the call list can be selected by parameter.

Parameter No.3110, #5

0: Number in comment statement of program indicated by cursor.

1: Number in program indicated by cursor.

(a) Call Selection by Comment Statement

When the program number to be called is already known, the order of the call are loaded beforehand as comment statement according to the conditions.

1) Insert comment statement immediately after the O-number.

2) Comment statement consists of two characters 'S:' for call selection.

3) The delimiter of call O-number is ','.

4) To put comment along with, set the order as (Comment S: Oxx).

(EX) O123 (S: O1234, O12);

O123 (MAIN S: O1234, O12);

(NOTE) Only the comment statement in a program is checked.

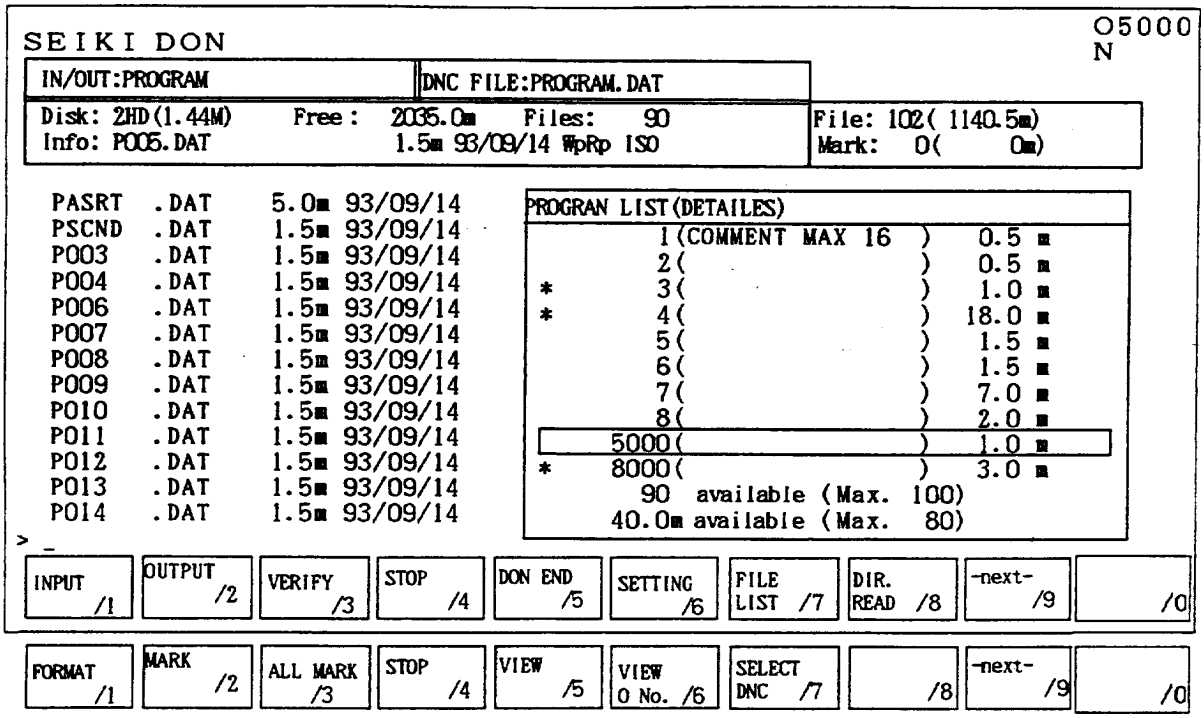
(b) Call Selection by Program

Programs called out are retrieved sequentially from the programs indicated by the cursor on the program list.

(There may be a slight delay before the display appears.)

14-6 Don-Don FD I/O Screen

This screen is made effective when the FD Handy type or the Built-in type is available. For details of operation, see the instruction manual of FD. When input/output are not performed on this screen, push F5/DON FD END and end the Don-Don FD I/O screen.



- DESCRIPTION OF FUNCTION KEYS

F1 (Input)	:	Reads from Don-Don FD.
F2 (Output)	:	Outputs in Don-Don FD.
F3 (Verify)	:	Reads files from Don-Don FD and verify with memory.
F4 (Stop)	:	Interrupts input/output.
F5 (Don End)	:	Ends Don-Don FD mode.
F6 (Setting)	:	Shows the window for setting parameters.
F7 (File List/ <Window>)	:	Shifts the cursor for the window and the file list.
F8 (Dir Read)	:	Reads the file list from Don-Don FD.
F9 (Next)	:	Changes over to the next menu.
Next F1 (Format)	:	Formats the floppy disk.
Next F2 (Mark), Space key	:	File selection (*) • cancellation (cancelled for the second time.) Without * mark, cursor position gets effective and the file in position is selected.
Next F3 (All Mark), Origin key	:	Selection • cancellation of all files (cancelled for the second time.)
Next F4 (Stop)	:	Input/output are interrupted.
Next F5 (View)	:	Content of the file at the cursor position is viewed.
Next F6 (View O No.)	:	The O-numbers contained in the file at cursor position are displayed in a list.
Next F7 (Select DNC)	:	The file at cursor position is registered as a file for DNC operation or simple DNC operation (tape mode).
Next F9 (Next)	:	Menu is switched into the next following one.

15. Input / Output (With PC Card Used)

In the Input/Output screen, you can also access the PC memory card. If the card is set in a slot beforehand, you can input/output the data to it in the same operating environment as before. Since the data is directly transferred to the memory card, not by way of serial communication, a transfer speed is improved.

It also allows simple directory management such as deleting a file from the PC memory card, altering a file name, or adding a subdirectory.

*The PC memory card is referred to as PC card hereinafter.

15-1 Available PC Cards

Available PC cards are 256 KB, 512 KB, 1 MB, 2 MB, and 4 MB S-RAM cards compliant with the PCMCIA or JEIDA Standards. The flush memory cards (FCs) and ATA Standards memory cards are not available.

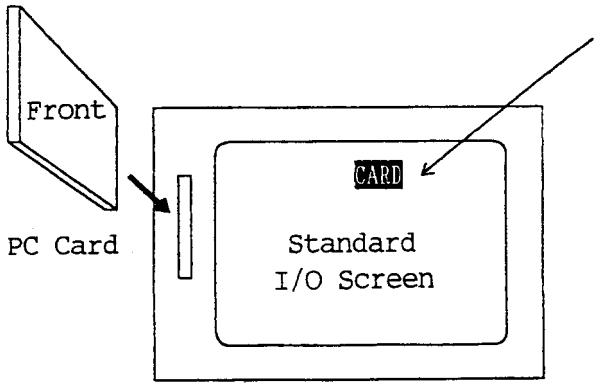
15-2 Initial Formatting of PC Card

Use a personal computer with a PC card slot to initially format the PC card. The card's format size may differ slightly depending on the model of the personal computer.

15-3 Starting PC Card Operation

① Press F8/IN/OUT in the Overall screen to display the conventional Input/Output screen. This is referred to as "Standard I/O screen" hereinafter.

② Insert the PC card into the card slot located next to the CRT screen as shown in the figure below. Beware of the inserting direction; the front side should face the CRT screen. Note that if it is inserted the other way around, the connection pins will be broken. If the CNC unit recognizes the PC card properly, a reverse video mark, CARD will be displayed on the screen.



The reverse video mark appears if the PC card is recognized, and it disappears if the card is removed.

Never touch the PC card while the CNC unit is accessing the PC card and data.

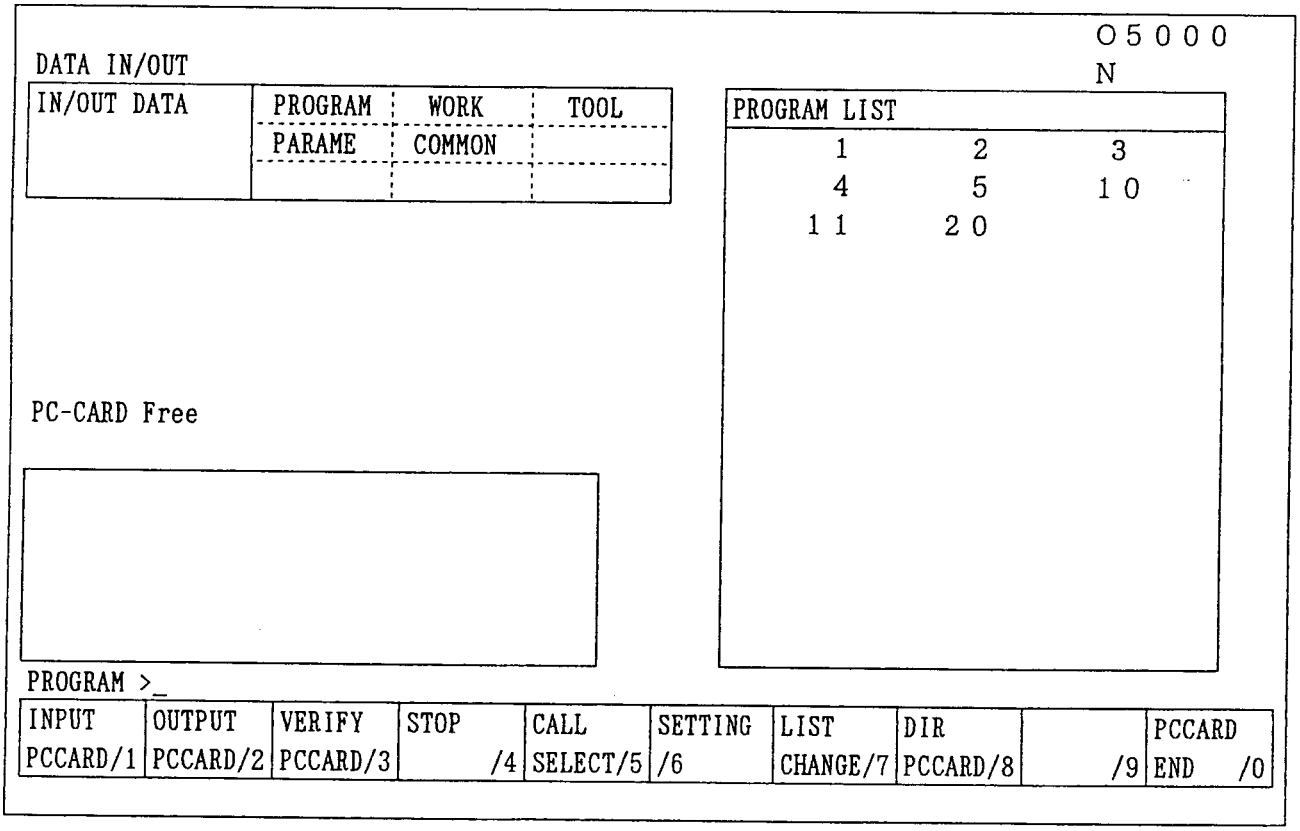
If the PC card is inserted, the function F10 will be changed to F10/PC CARD INOUT as shown in the figure below. If it has been inserted from the beginning, that function is already displayed.

INPUT /1	OUTPUT /2	VERIFY /3	STOP /4	CALL SELECT/5	SETTING /6	LIST CHANGE/7	CHECK ON/OFF/8	DON FD/9	/0

INPUT /1	OUTPUT /2	VERIFY /3	STOP /4	CALL SELECT/5	SETTING /6	LIST CHANGE/7	CHECK ON/OFF/8	DON /9	PCCARD INOUT /0

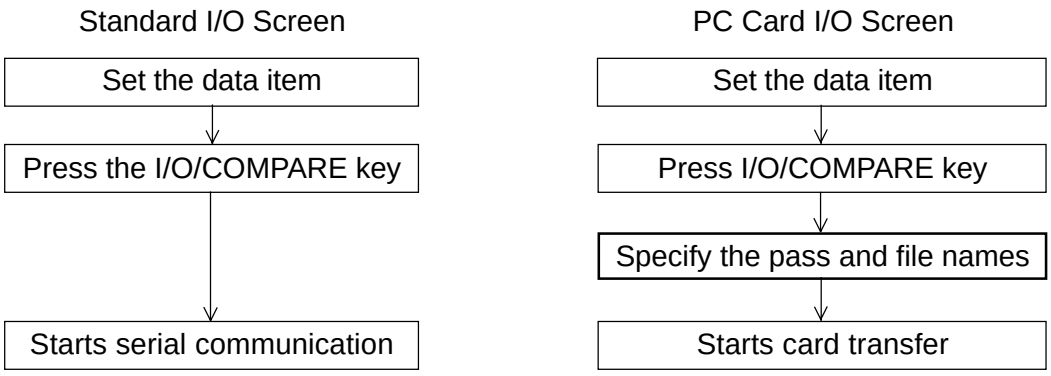
Inserting the PC card enables a card menu.

③ Pressing F10/PCCARD INOUT displays the PC Card I/O screen shown in the figure below. The setting items available are only “Data I/O Items.” There are no setting items associated with RS-232C communications.



PC Card I/O Screen

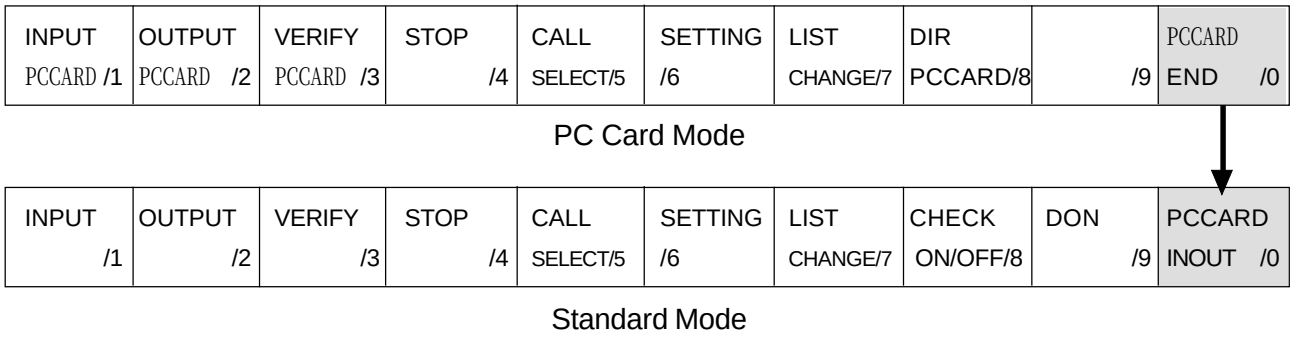
This PC Card I/O screen provides the same operating environment as usual. What has been newly added to the screen is the operation to specify the PC card path and file names.



15-4 Ending PC Card Operation

15-4-1 Transitioning to the Standard I/O Screen

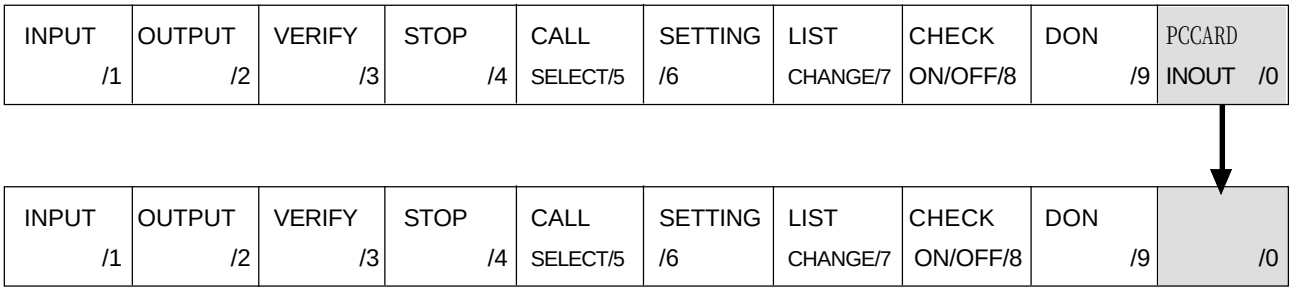
When exiting the PC card mode, press F10/PCCARD END. You will be returned to the Standard I/O screen of the RS-232C mode.



15-4-2 Removing the PC Card

The Standard I/O screen is always detecting the existence of the PC card. If the card is removed in the Standard I/O screen or if you revert to the Standard I/O screen with the card removed, the function key F10 will be blanked again.

Never remove the PC card while transferring the card data.



Removing the PC card turns off the PC card function.

15-5 PC Card Input/Output

15-5-1 Transfer Data Format

When operating the PC card, the transfer data format remains unchanged as shown below to facilitate editing through the personal computer. The transfer data format is restored when you finish PC card operation and return to the Standard Input/Output screen.

Exclusive PC Card Input/Output Format

- ASCII output (8-bit)
- Zero feed, zero space, LF mode, communication check OFF
- No TV check, no parenthesized TV check, no TH check

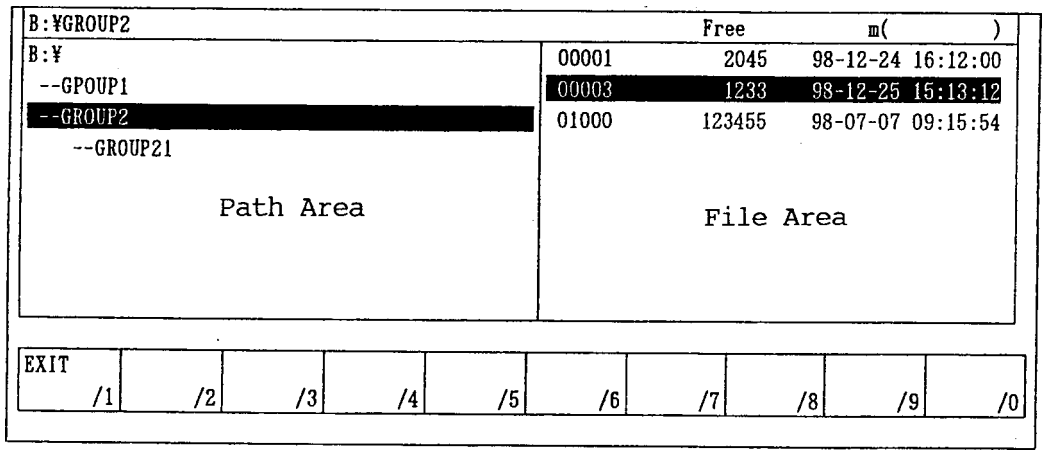
15-5-2 PC Card Input

This function allows you input the data from the PC card to the NC unit.

① Select the PC Card Operation screen.

② Set the input/output items in the data setting table. When setting a data range, describe it beforehand in the conventional method.

③ Press F1/INPUT PCCARD . The PC Card Directory screen appears as shown in the figure below.



PC Card Directory Screen

④ Set the cursor to the directory name in the path area and press the INPUT key to specify it.

⑤ Using the → key, move the cursor to the right file area. (If the available subdirectories in the PC card are only Root \, skip Steps ④ and ⑤)

⑥ Bring the cursor to the file name you want to input, and press the INPUT key to specify it. This will immediately start inputting the data from the PC card.
Never touch the card while transferring the data.

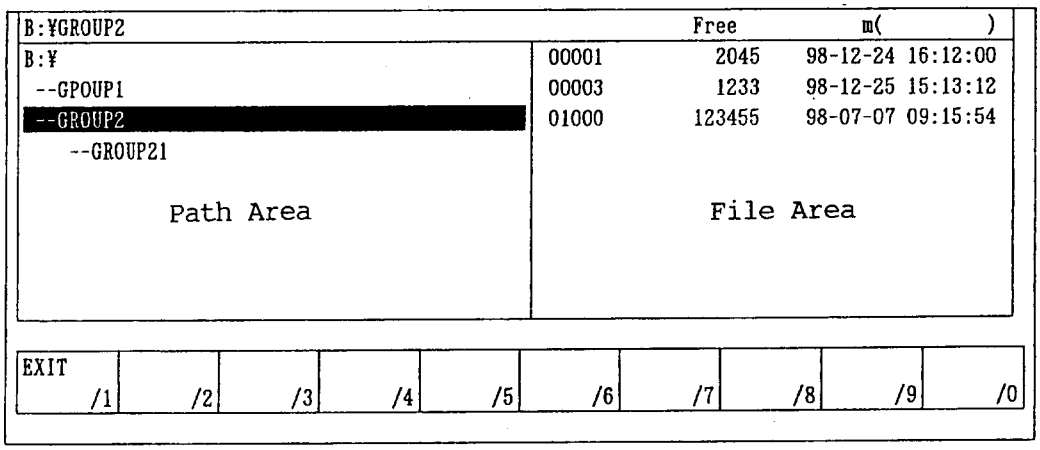
15-5-3 PC Card Output

This function allows you to output the data from the NC unit to the PC card.

① Select the PC Card Operation screen.

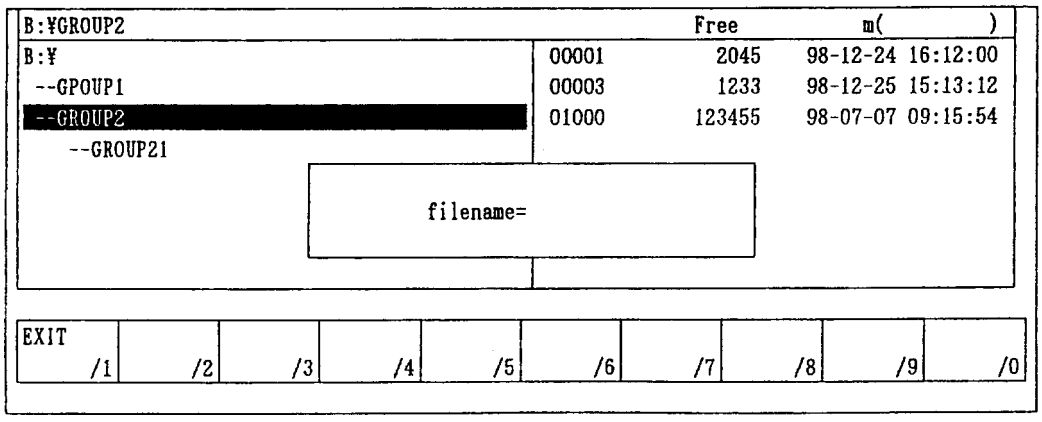
② Set the input/output items in the data setting table. When setting a data range, describe it beforehand in the conventional method.

③ Press F2/OUTPUT PCCARD. The PC Card Directory screen appears as shown in the figure below.



④ Set the cursor to the directory name in the path area and press the INPUT key to specify it. (If the available subdirectories in the PC card are only Root \, skip Step④)

⑤ Then, the window will appear, where you input an output file name. In this window, input a new file name and press the INPUT key. The file names are the same as the MS-DOS specific ones. An extension can be also input with a period.
File name (up to 8 characters) + Period "." + Extension (up to 3 characters)



If the PC card already contains the same file name, make sure for overwriting.
Once the output file name is set, the data will be output immediately.
Never touch the card while transferring the data.
If an error is displayed indicating that the PC card is running out of the memory capacity or otherwise, delete unnecessary files, seeing "15-6 Operating the PC Card Directory."

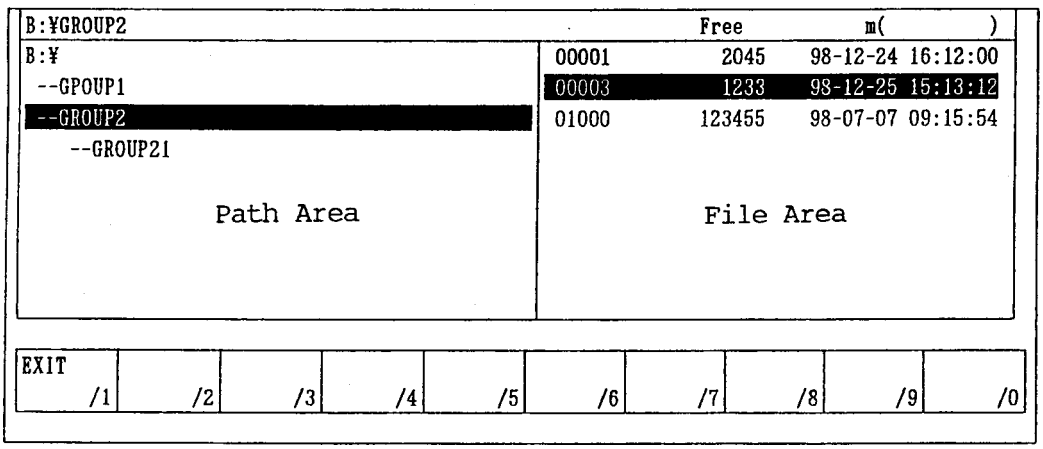
15-5-4 Comparing the Data

This function allows you to compare the PC card data with the NC unit data.

① Select the PC Card Operation screen.

② Set the input/output items in the data setting table. When setting a data range, describe it beforehand in the conventional method.

③ Press F3/VERIFY PCCARD. The PC Card Directory screen appears as shown in the figure below.



PC Card Directory Screen

④ Set the cursor to the directory name in the path area and press the INPUT key to specify it.

⑤ Then, using the → key, move the cursor to the right file area. (If the available subdirectories in the PC card are only Root \, skip Steps ④ and ⑤)

⑥ Bring the cursor to the file name you want to compare, and press the INPUT key to specify it. This will start data comparison immediately.
Never touch the card while transferring the data.

15-5-5 Canceling Operation

Press F4/STOP. This will cancel input, output, or comparison work. If data output is under way, the file being output will not remain in the PC card.

INPUT PCCARD/1	OUTPUT PCCARD/2	VERIFY /3	STOP /4	CALL SELECT/5	SETTIN G /6	LIST CHANGE/7	DIR PCCARD/8	DON /9	PCCARD END /0

15-6 Operating the PC Card Directory

The PC Card Operation screen has a simple directory management function. To operate the directory, select F8/DIR PCCARD in the following PC card mode menu.

INPUT /1	OUTPUT /2	VERIFY /3	STOP /4	CALL SELECT/5	SETTIN G /6	LIST CHANGE/7	DIR PCCARD/8	DON /9	PCCARD END /0

- The management function allows you to “create, rename, and delete the directory.” Select the directory or file with the cursor and press the function key to execute the function.
- Up to 32 subdirectories (Root ¥ included) can be displayed (2 pages worth).
- Up to 256 files in each directory can be displayed (16 pages worth).
- The subdirectories can be displayed nested up to 8 stages.

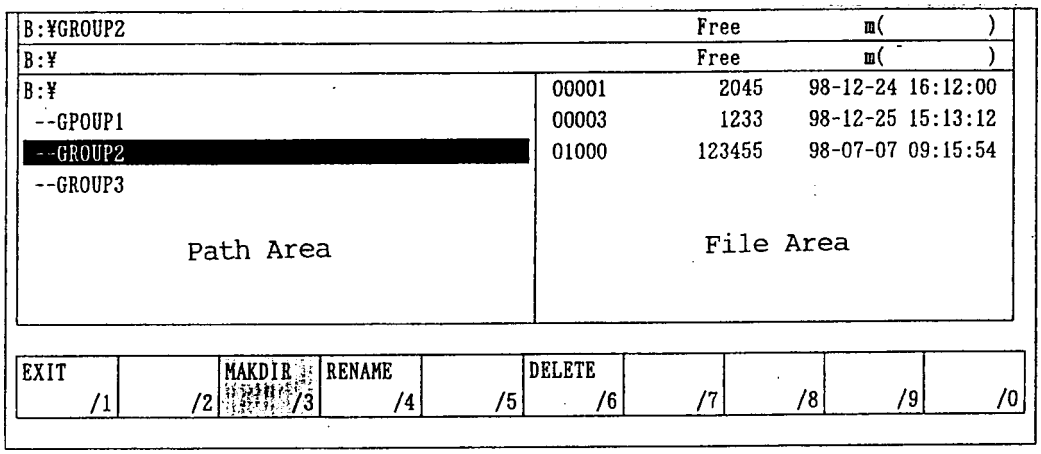
B:¥
-AAAAAAAA
-BBBBBBBB
-CCCCCCCC
-DDDDDDDD
-EEEEEEEE
-FFFFFFFF
-GGGGGGGG

If the PC card contains the subdirectories or files beyond the maximum number, only the maximum number of them can be displayed. If this is the case, the display priorities will change depending on the directory configuration of the PC card.

15-6-1 Creating the Directory

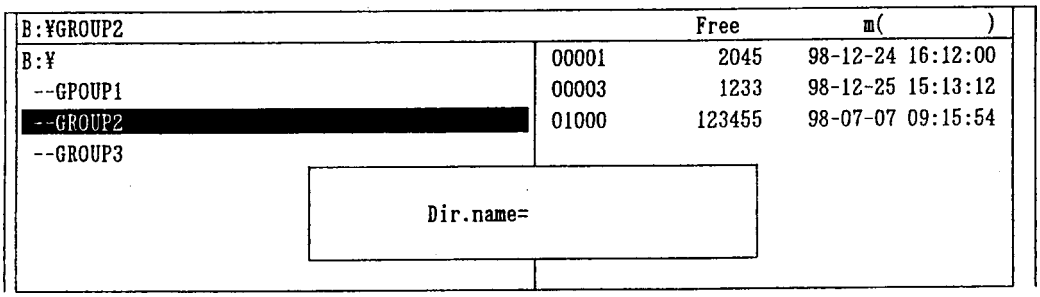
① Press F8/DIR PCCARD to display the Directory Operation screen.

② The subdirectory is added to below the cursor-indicated directory name. When adding it to below Root ¥, set the cursor to “B:¥.”



③ Press the F3/MAKDIR key.

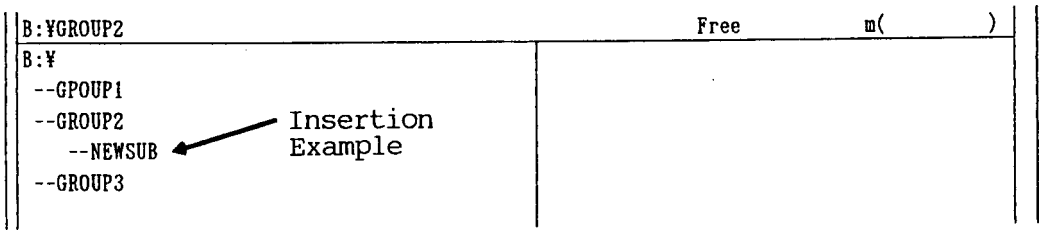
④ The window appears, where prompts you to input a new directory name.



Now, input the new directory name and press the INPUT key. The directory names are the same as the MS-DOS specific ones.

File name up to 8 characters (A subdirectory name has no extension)

If the PC card already contains the same name, you will be warned to that effect. Now, the subdirectory has been added to the PC card.



15-6-2 Deletion

(1) Deleting the subdirectory

Deletion of the subdirectory results in an error in the following cases:

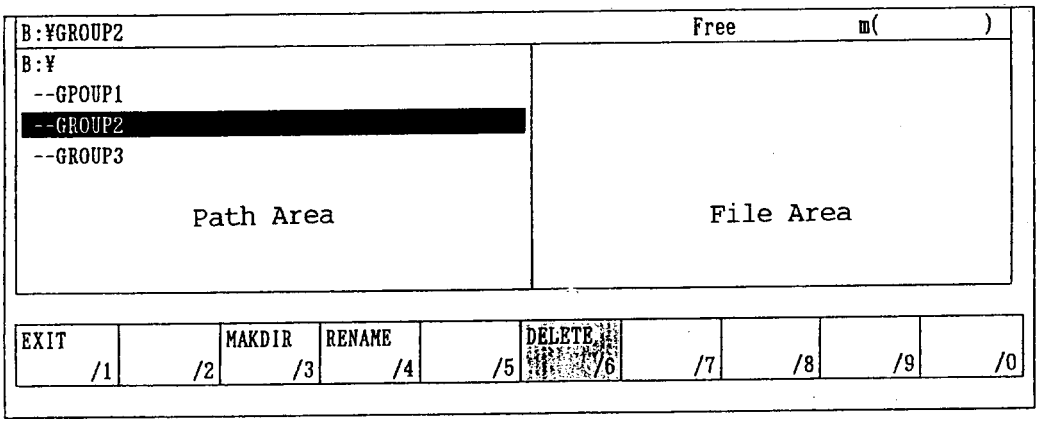
- When the directory to be deleted contains one or more files.
- When the subdirectory is remaining below the directory to be deleted.

If this is the case, delete all the subdirectories or files.

① Press F8/DIR PCCARD to display the Directory Operation screen.

② Set the cursor to the directory name you want to delete from the path area.

③ Press F6/DELETE.



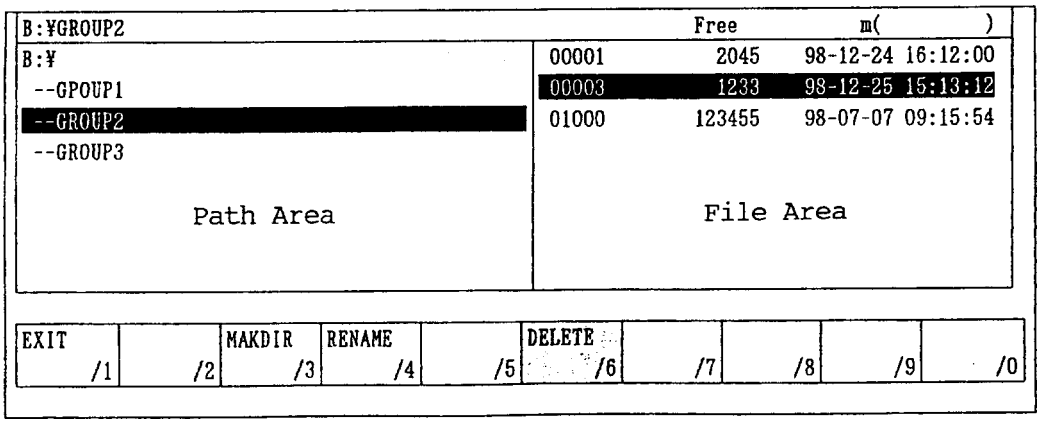
(2) Deleting the file

① Press F8/DIR PCCARD to display the Directory Operation screen.

② Set the reverse video cursor to the directory in the path area to display a files list.

③ Then, set the cursor to the file you want to delete.

④ Press F6/DELETE.

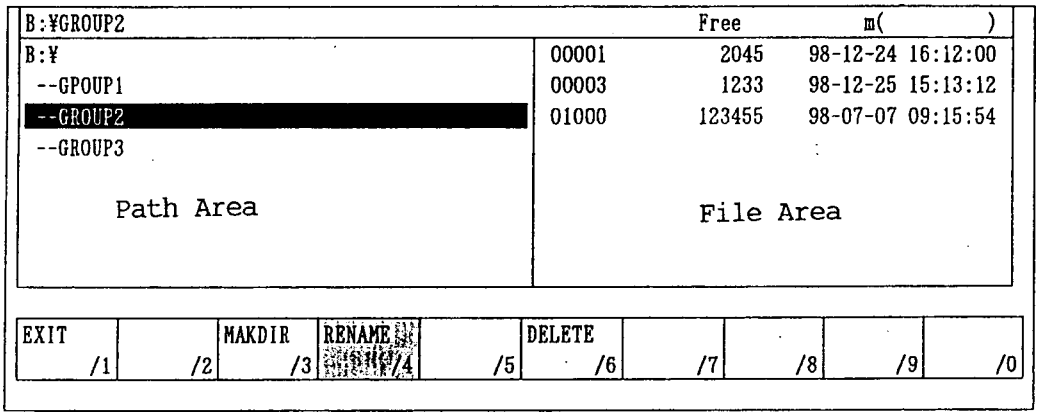


15-6-3 Renaming

(1) Renaming the subdirectory

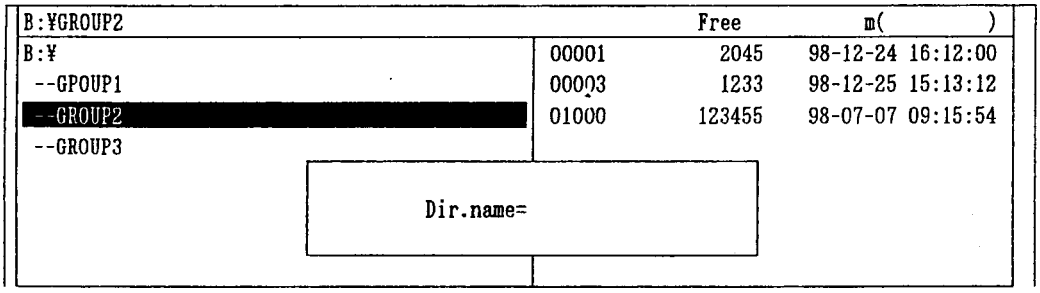
① Press F8/DIR PCCARD to display the Directory Operation screen.

② Set the cursor to the directory name you want to rename in the path area.

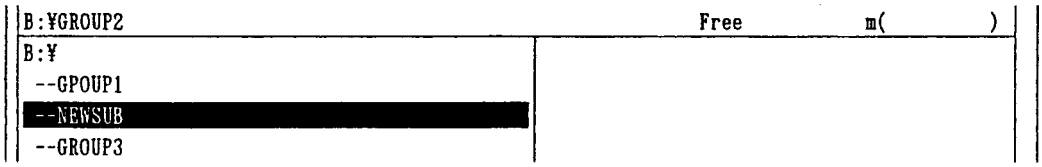


③ Press F4/RENAME.

④ The window appears, which prompts you to input a renewed directory name.



Now, input the renewed directory name and press the INPUT key. The directory names are the same as the MS-DOS specific ones.
Directory name up to 8 characters (A subdirectory name has no extension)
If the PC card already contains the same name, you will be warned to that effect.



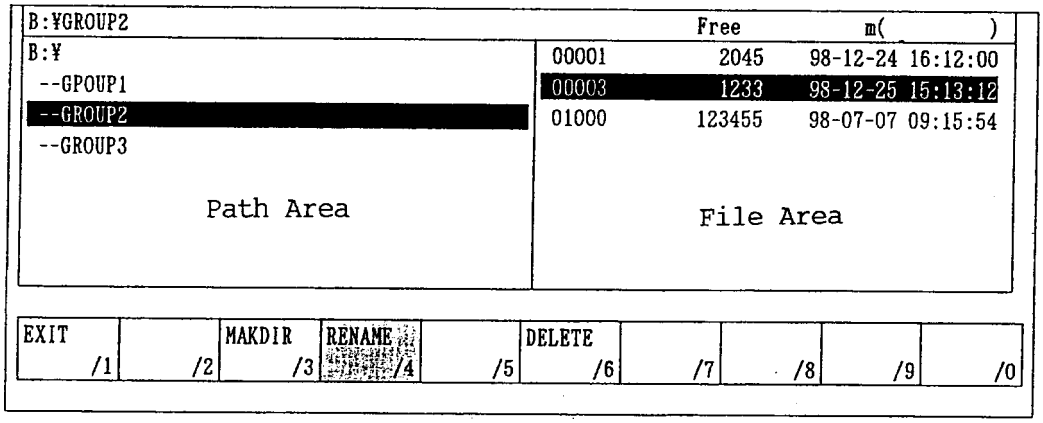
Now, the subdirectory name has been renamed in the PC card.

(2) Renaming the file

① Press F8/DIR PCCARD to display the Directory Operation screen.

② Set the reverse video cursor to the subdirectory in the path area to display a files list.

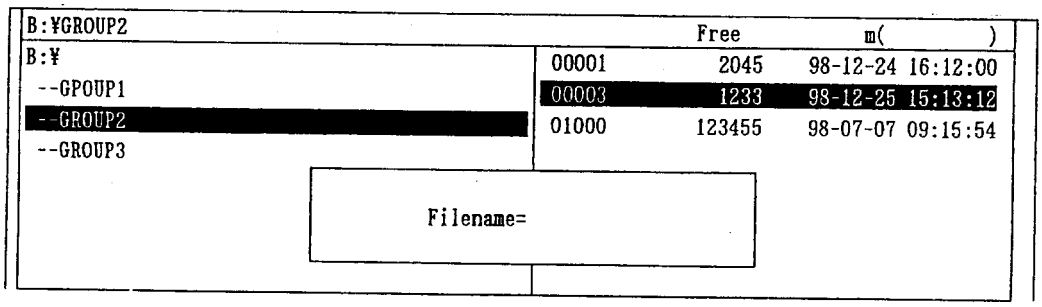
③ Set the reverse video cursor to the file you want to rename.



PC Card Directory Screen

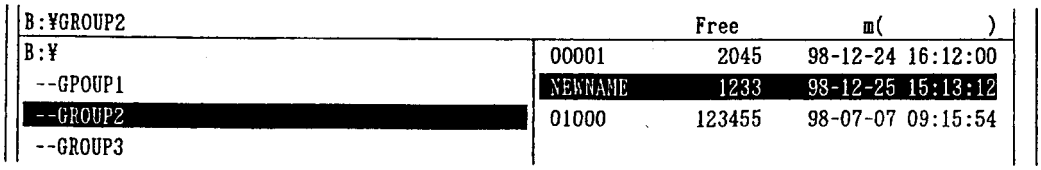
④ Press F4/RENAME.

⑤ The window appears, which prompts you to input a renamed file name.



Now, input a new file name and press the INPUT key. The file names are the same as the MS-DOS specific ones. An extension can be also in put with a period.

File name (up to 8 characters) + Period "." + Extension (up to 3 characters)
If the PC card already contains the same name, an error will be displayed.



15-7 Error Messages

Message	Description
“PC-CARD mount error.”	The PC card could not be opened. Generally, this error is not displayed.
“PC-CARD unmount error”	The PC card could not be closed. Generally, this error is not displayed.
“Non PC-CARD.”	The PC card has not been set in the slot. Set the formatted card into the slot.
“Low Battery.”	Replace the battery of the PC card.
“Illegal PC-CARD.”	The PC card has a wrong format. It may not be the PCMCIA card.
“Just a moments & Retry.”	It takes several seconds to recognize the PC card after setting it into the slot.
“Write protected.”	Turn off the PC card’s protect switch.
“Undefined PC-CARD error.”	The PC card may be faulty
“Undefined PC-CARD status.”	The PC card may be faulty
“CARD full error.”	Delete an unnecessary file from the PC card.
“Delete error (remain files,/\root)”	To delete the subdirectory, clear all the files in it. Root B:\ cannot be deleted.
“Delete error (read only)”	The file is of read only attribute. Cancel the attribute before deleting.
“makdir error (same name/CARD full)”	Another directory has the same name. This error also occurs when the PC card is out of capacity.
“rename error (same name/CARD full)”	Another file has the same name. This error also occurs when the PC card is out of capacity.

4. Individual Programs

I TOOL LIFE MANAGEMENT

I I CUTTING MONITORING UNIT

III TOOL PECULIAR NUMBER FUNCTION

IV STAF

V SHG

VI THERMAL DISPLACEMENT OFFSET FUNCTION

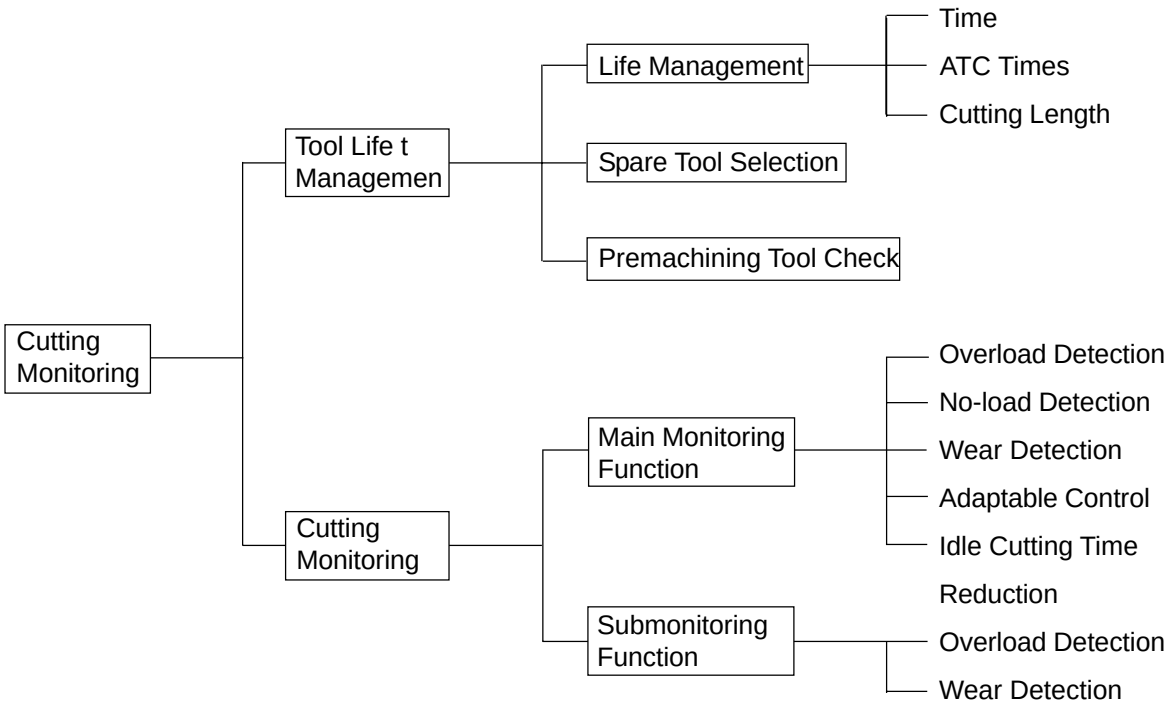
VII UUP FUNCTION

** OVERVIEW **

As labor-saving has been enhanced in the more factories, one worker takes charge of multiple NC machine tools and unmanned operation of the machines is going on all day long. Under these circumstances, the features of this system (tool life management, spare tool management, cutting monitoring) will effectively help prevent abnormal cutting and production of a large amount of defective products.

High reliability of the system itself is necessary in order to maintain these features. A high-reliability, efficient system has been realized by using up-to-data technologies. A plane display method is used for easier understanding and better operability. In cutting monitoring, the load state is displayed in the form of load graph and life graph to make the relations between the monitoring state and monitoring data more understandable.

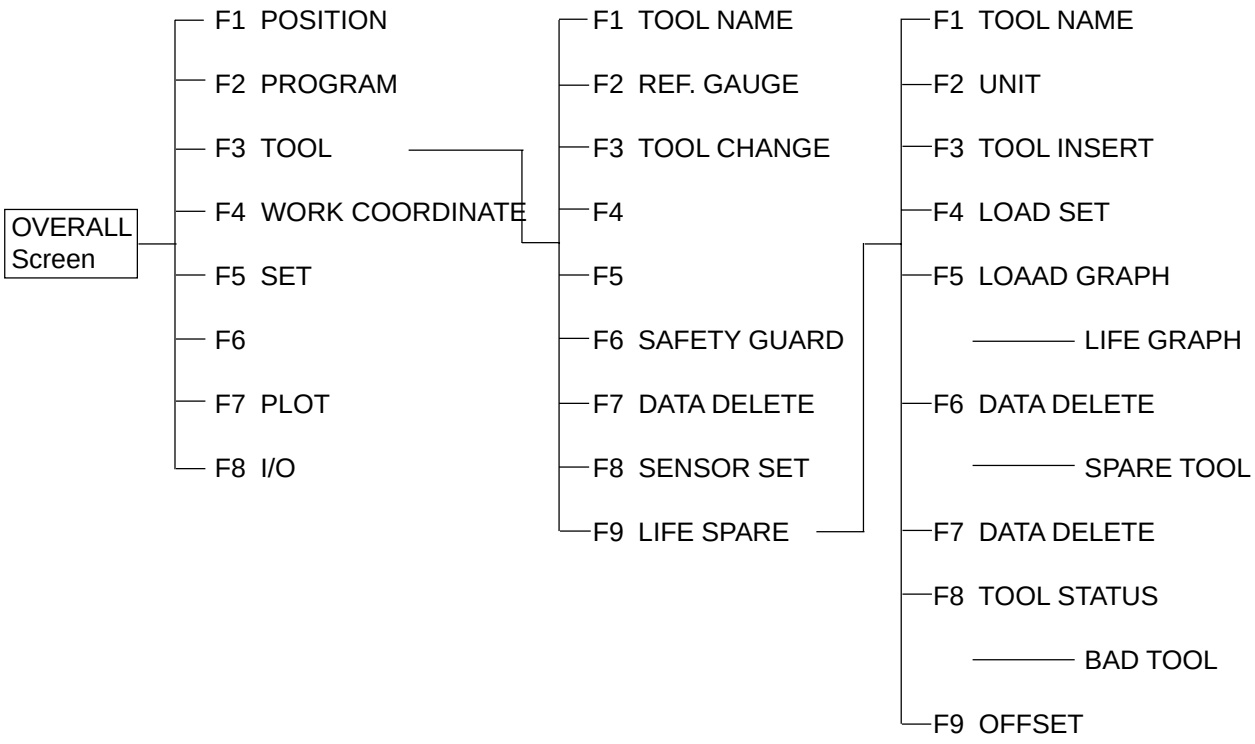
** CONFIGURATION CHART **



Unmanned machines such as pallet pool line are associated with the following functions.

- Tool breakage detection (Method by M52)
- Tool length measuring instrument (G100, G103)
- Automatic return function (M35, M36)

** SCREEN TREE CHART **



4. INDIVIDUAL PROGRAMS

I. Tool Life Management

1. Tool Life Management

1-1 Feature

The tool lefe management function counts the tool lefe use values of the tools used in the machining program according to the life units for each tool. It compares a counted tool use value with a tool lefe set value to control a tool life. When you do not want tool life management in the measuring program, control this function b a G-code. When the power is turned on or the NC unit is reset, the G131 mode is selected.

G130 ..... Tool life management disabled
G131 ..... Tool life management enabled

1-2 Functioning

Tool life management carries out the following actions sequentially.

- Selecting a tool for the spindle (when a Txx command in the program is executed)
- Checking the spindle tool status (when M06 in the program is completed)
- Counting the tool life (during machining by the program)

① Selecting the tool for the spindle

When a Txx command is executed in the machining program, the tool life management function selects a tool to be called to the spindle. If the Txx command is a spare tool command, a defectless tool will be automatic-ally selected out of a spare tool group.

* If the selected tool is an unused one, a new tool signal is output to the PC. (During TF)

② Checking the spindle tool status

The tool life management function checks the tool status of the tool called to the spindle by the M06 command in the machining program.

- When the tool is unused → Designates the tool status “USED” or “MONITOR ED.”
- When the tool is out of life → Displays a warning, “781 OUT-OF-LIFE TOOL WAS SELECTED.” In the M56 mode, however, the machine stops due to feedhold.
- When the tool is broken → Displays a warning, “780 BROKEN TOOL WAS SELECTED.” The machine stops due to feedhold.

③ Counting the tool life

When the following requirements are met, the tool life management function counts the tool life use value of the spindle tool according to the tool life units preset for each tool.

- Under automatic operation in the MEMORY mode
- Dry run, Z-axis cancel, and machine lock are not turned on
- In the M53 mode (M54 monitoring function is not stopping)
- In the G131 mode

When a counted use value reaches a preset life value, the tool status for that tool is designated OUT OF LIFE, but machining continues as it is. When the life set value is zero, only counting of the life use value is performed and out-of-life judgment is not made.

* When the tool status is designated OUT OF LINE and that tool or all the tools in the same tool group are defective, a tool change signal is output to the PC.

1-3 Life Units

As tool life units, set time, use times, length, and number of holes for each tool. The tool life is counted as follows according to the life units.

- Time ..... Counts cutting feed time (minutes) during spindle forward rotation. When the parameter 8002 #0 is “1”, it is counted even during spindle reverse rotation.
- Use times ..... Counts how many times the tool has been called to the spindle (at the time of M06).
- Length ..... Counts a cutting distance (meters) during spindle forward rotation.
- No. of holes ..... Counts the number of holes machined in a canned cycle for tapping or drilling. It is counted earlier than actual machining because it is counted by buffering.

(Note) The data below a minute and meter (seconds and millimeters) are internally counted and memorized.

TxxM06; ..... If the life units are use times, use times is counted here.
S100M03;
G54G90G00X0Y0;
G43Z50.0H12;
G01Z38.F500;
X50.F280;
Y10.;
X0.;
G00Z100.;
G91G28Z0M05;

If the lift units are time and distance, time or cutting distance is counted.

1-4 Tool Status

The TOOL screen lists the status of each tool as follows. It is displayed when the machine specification requires it. Typical specification is mentioned in parentheses, but it is not always displayed.

USED ..... The tool is used.
MONITORE ..... The tool is used with load monitoring enabled.
OUT OF LIFE ..... The tool is out of life.
WERN OUT ..... The tool is wern out. (Cutting monitoring)
SKIP ..... The tool is skipped by a skip signal.
TOOL NOSE MEAS. N.G. ..... The tool was found out no good in tool nose measurement. (Macro measurement, etc.)
WORK MEAS. N.G. ..... The tool was found out no good in work measurement. (Macro measurement, etc.)
OVERLOAD ERROR ..... An overload error resulted. (Cutting monitoring)
NO-LOAD ERROR ..... A no-load error resulted. (Cutting monitoring)
EXTERNAL ERROR ..... An external error resulted. (Breakage detection, etc.)

2. Spare Tool Management

2-1 Feature

If the Txx command in the machining program is a spare tool command, the spare tool management function will automatically select a defectless tool out of a preregistered spare tool group. The tool is selected out of the spare tool group in order of ① used tool (MONITORED included), ② unused tool of higher order registration, and ③ last selected tool if all the tools are out of life.

2-2 Functioning

Determine a group number for the tools of identical type which are specified as spare tools in the program, using an arbitrary number larger than the numerical value in the parameter no. 8078 (standard value:1.000), and register it as a call number used to specify a spare tool in the SPARE TOOL screen. Register the spare tools in the tool group in ORDER 1 onward. After setting the call number and spare tool number, specify a spare tool in the program. The spare tool management function automatically selects a defectless tool out of the spare tool group.

2-3 Specifying the Offset

2-3-1 Tool Offset by Tool Number

If a “tool offset by tool number” option is added to the NC unit for the SEICOS- Σ , tool offset can be applied by a spindle tool number without taking offset data “H” and “D” into account, even if the spare tools are used. (You only have to specify an offset instead of specifying H and D.)

2-3-2 Tool Offset by Optional Block Skip

An offset for the spare tool is specified by informing the NC unit of a spare tool registration position as an optional block skip position. (For two or more tools, an NC option is required)
If the OPTIONAL BLOCK SKIP switch on the operation panel is used by an NC program, machine operation becomes unstable because it is duplicated by offset designation for the tool life (optional block skip). When using offset designation for the spare tool, separate the OPTIONAL BLOCK SKIP switch on the operation panel from the program used.
Make a machining program by combining an offset number with a block skip number.

2-4 Spare Tool Data

Since the tool life set value and life units for the tool group are set under CALL NUMBER, it is not necessary to set them for each spare tool. The tools registered as spare tools are displayed less clearly in the TOOL LIFE screen and you cannot set the data for them. Set the data in the SPARE TOOL screen. If a tool number is registered in ORDER 1 in setting only the call number (the tool name and life set value of the call number are zero), the tool name and life data for the ORDER 1 tool will be set as the data for the call number. Cutting monitoring is performed
based on the data for the first spare tool.

3. Premachining Tool check

3-1 Feature

At the beginning of the machining program, specify the tools to be used in the subsequent machining. The premachining tool check function will checks whether all the tools used, including spare tools, are defectless or not. If there is even one defective tool, the machine stops, displaying a warning, “782 PREMACHINING TOOL CHECK ERROR.”

3-2 Functioning

If the tool used for machining is specified (spare tool) between M51 and M59 in the program, it will be checked if the specified tool is defectless or not. If there is even one defective tool, the machine stops upon completion of premachining tool check (M59), displaying “782 PREMACHINING TOOL CHECK ERROR.”

At this time, the tool numbers of defective tools are listed as defective tools in the TOOL STATUS screen. Interlocking this function with automatic return sets the next work and carries out next machining. (PPL system) This function helps prevent production of incomplete products.

4. Tool Skip And Tool Reset (Option)

Input a tool skip signal from an external device. The tool status of the then spindle tool is designated SKIPPED and that tool will be skipped (not used) from the next time.

* When the tool status is designated SKIPPED and that tool or all the tools in the same tool group are defective, a tool change signal is output to the PC.

If a tool reset signal is input from an external device, the tool status and tool life use value of the then spindle tool will be cleared. These signals are effective in any mode.

5. Tool Life Management Program

5-1 Special Purpose M-codes for Tool Life

5-1-1 Disabling/Enabling the Tool Life and Cutting Monitoring Functions (M54/M53)

M54 is used when disabling the tool life and cutting monitoring functions. After using M54, cancel it by M53.

M54 ..... Stops the tool life/cutting monitoring function. Although it disables a tool status check, life count, premachining tool check, and tool life suspension (M55), spare tool call, tool offset, and life data setting remains enabled.

M53 ..... Cancels M54. When the power is turned on, M53 is selected.

5-1-2 Machine Stop by Out-of-life Tool and its Reset (M56/M55)

M56 is used to stop the machine when a tool is out of life.

M56 ..... Stops the machine when an out-of-life tool comes to the spindle.

M55 ..... Cancels M56. When the power is turned on, M56 is selected.

5-1-3 Assisting/Resetting Spare Tool Offset (M45/M46)

Block skip and optional block skip are used to select a spare tool offset. Be sure to enclose an offset selection by M45 and M46 to program it.

(Note: Register M45 and M46 in the NC non-buffering M-codes.)

To select a tool offset, inform the NC unit of the registration position from “ORDER 1” registered as a spare tool by block skip.

Program Example (with Option)

T1100M06;
M45
G43Z_H12/2 H11/1 H10
M46

H12 ..... Offset for the tool no.1
H11 ..... Offset for the tool no.1
H10 ..... Offset for the tool no.1

CALL NO. T1100			BLOCK SKIP SIGNAL
ORDER	TOOL		
1	10		0
2	11		1
3	12		2
⟩	⟩		⟩

(Note 1) When optional block skip is not effected, offsets for ORDER 3 onward are adjusted to the tool no.10.

5-1-4 Premachining Tool Check (M51/M59)

This function checks whether or not the tools are defectless, including the spare tools. You specify the tool number you want to check between M51 and M59. A judgment result can be known when M59 is executed.

M51 ------ Starts a premachining tool check
M59 ------ Ends a premachining tool check

(Note) A command other than Txx is specified between M51 and M59, an error message, “783 MONITORING FORMAT ERROR,” appears. A T-code command for premachining tool check does not call a tool.

(Example)

M51;
T01;
T20;
T1100;
M59;

Checks whether or not all the tools specified here are defectless.
(Checks the tools T01 and T20, and spare tool T1100)

5-1-5 Life Data Automatic Setting (M57, M59)

This function downloads the tool life set value and use value from the NC program.

M57; ..... Starts downloading the life data.
Txx; ..... Specifies a tool number.
1st Sxxx; ..... Sets the life set value for Txx (Life set value for the group in case of call number)
2nd Sxxx; ..... Sets the life use value for Txx (Omissible, nothing is done in case of call number)
M59; ..... Ends downloading the life data

When the 1st S-code command is executed, it is determined by parameter setting to select the life units and clear the tool status and life use value.

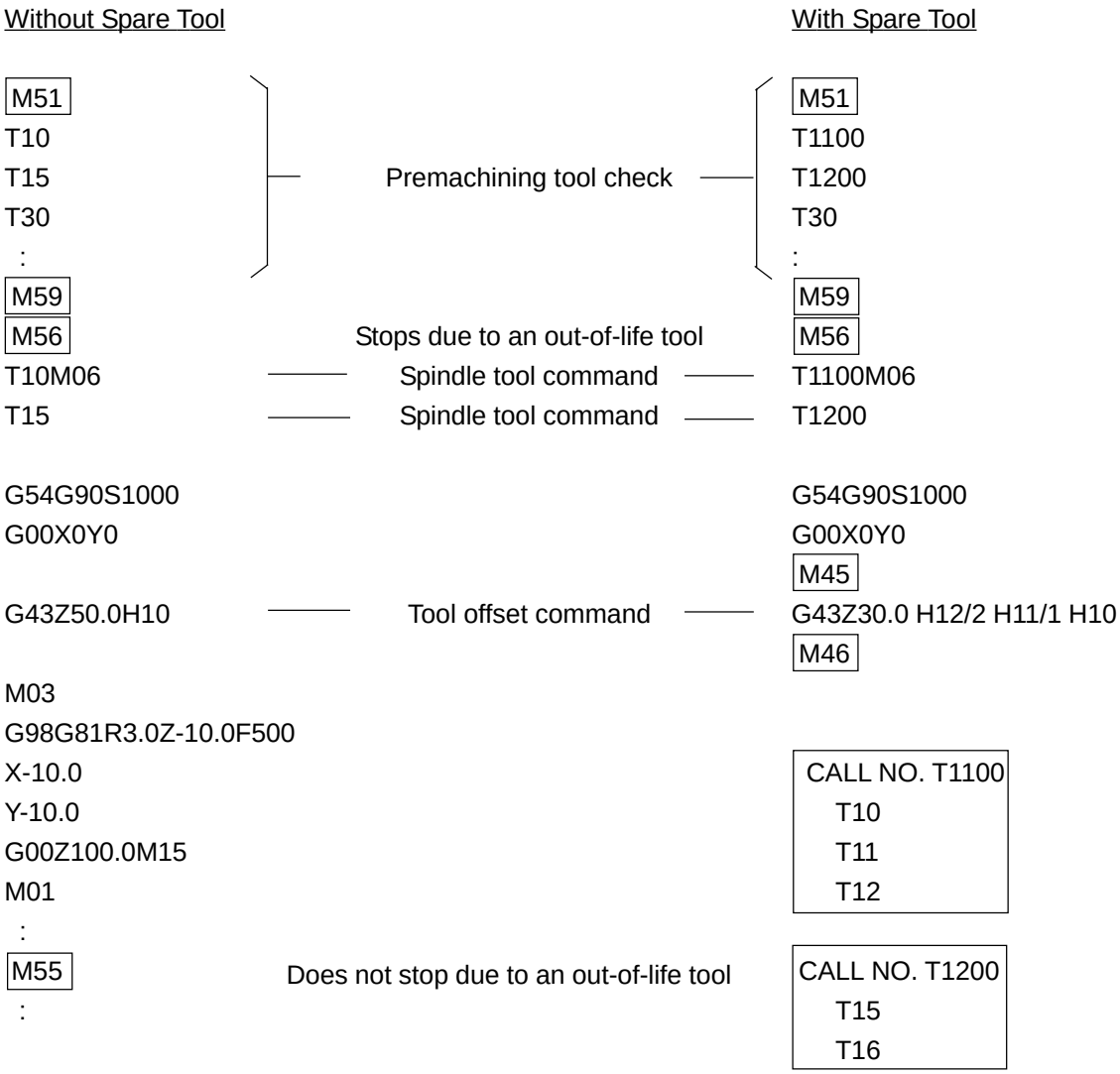
- Parameter no. 8001 #6 = 1 .....Times
  Parameter no. 8001 #7 = 1 ..... Minutes
- Parameter no. 8003 #7 = 0 .....Clears the tool status and tool life use value when the tool life is set.
  8003 #7 = 1 .....Does not clear the tool status and tool life use value when the tool life is set.

(Note) When a command format is not correct, an error message, “783 MONITORING FORMAT ERROR,” appears. After setting the data, be sure to specify a spindle rpm command, Sxxx.

(Example)

M57; ..... Starts downloading the life data
T05; ..... Specifies the tool no. 5
S20; ..... Sets 20 as a life set value for the tool no. 5
;
T01; ..... Specifies the tool no. 1
S230; ..... Sets 230 as a life set value for the tool no. 1
S80; ..... Sets 80 as a use value for the tool no. 1
;
T5321; ..... Specifies the call no. 5231
S150; ..... Sets 150 as a life set value for the call no. 5231
;
T30000; .... Specifies the call no. 30000
S250; ..... Sets 250 as a life set value for the call no. 30000
S80; ..... Does not set a use value for the spare tool of the call no. 30000
;
M59; ..... Ends downloading the life data

5-2 Sample Program



6. Tool Life Management Operation

6-1 Preparations for Operation

After changing a setup and setting in the tool magazine the tools you want to use for next machining, input their tool data (tool name, life units, life set value, life use value) in the TOOL LIFE screen.

To operate with a spare tool, set in the SPARE TOOL screen the call number used to specify the spare tools in the machining program, and input the life units and life set values for that tool group.

Set the spare tools in ORDER 1, ORDER 2, and so on with tool numbers and set the life use value for each tool.

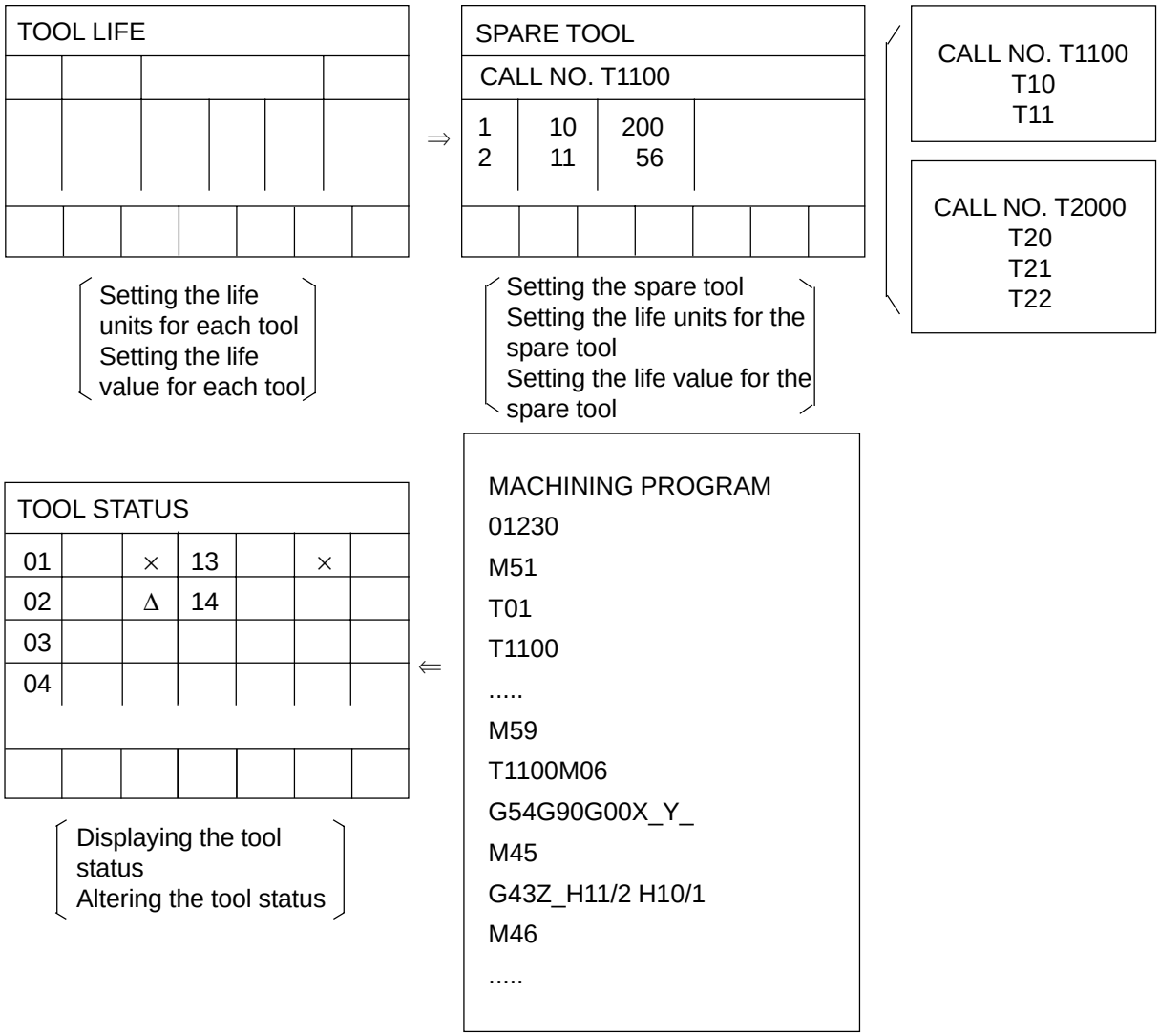
6-2 Machining Operation

Tool life management counts the tool lives of the tools used for machining by an NC program. When the tool runs out of its life, this function assumes that tool OUT OF LIFE and continues machining. When the out-of-life tool is called to the spindle by a tool command in next machining, an error message, “781 OUT-OF-LIFE TOOL WAS SELECTED,” appears and a call light is turned on. In the M56 mode, the machine stops feeding the axes. When the tool used is running out of its life, the tool status in the TOOL STATUS screen becomes “Δ”. Change the tool if early change is required.

6-3 Tool Change

When there is an out-of-life tool, stop the tool and change it to a new one. After changing to the new tool, delete the tool life use value and tool status, and then, restart machining.

6-4 Tool Life Management Operational Procedure



7. Tool Life Screen

Press F3/TOOL in the OVERALL screen. The following function screen is displayed. Although the function keys, TOOL NAME/1, REF. GAUGE/2, TOOL CHANGE/3, /4, /5,
SAFETY GUARD/6, DATA DELETE/7, SENSOR SET/8, LIFE SPARE/9, and /0, are also displayed on the right end of the screen, they will be omitted hereinafter.

Press F9/LIFE SPARE.

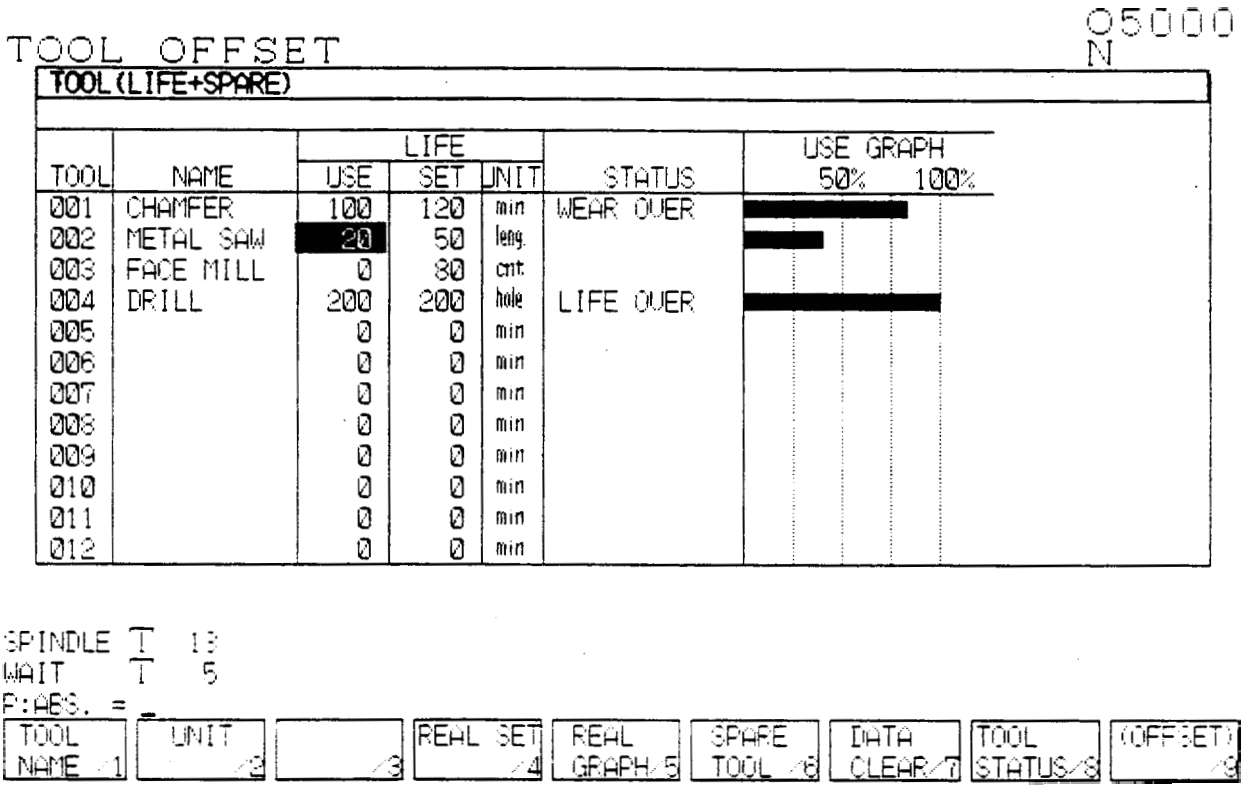


Fig. 1

(Note) The data for the tools registered as spare tools are displayed dimly. No entry is allowed through the TOOL LIFE screen.

7-1 Setting the Tool Name (A tool name can be set for each tool)

The tool life management function is not affected even if a tool name is not set.

① Using the Page keys and and cursor keys ↑ and ↓ , move the cursor to the tool number whose tool number you want to set. Or, enter N followed by a tool number (numerical value), and press ↓.

② Press F1/TOOL NAME → A Tool Name Setting window is displayed.

③ Input a tool diameter, etc. in the alphanumerical entry → The input data is displayed in the key input area.

④ Move the cursor with the ↓ key and select a tool name.

⑤ Press INPUT → The Tool Setting window disappears. Set the tool name.

⊙ When interrupting operation (turning off the Tool Setting window), press F1/TOOL NAME.

[Example] Setting [100 milling cutter] for T003

① Using the [↓] key, move the cursor to [003].
② Press [F1/TOOL NAME] → The cursor is located at an alphanumerical entry.
③ Press the keys, [1], [0], [0], and [SPACE] in that order.
④ Using the [↓], key, move the cursor to [MILL].
⑤ Press [INPUT].

	TOOL NAME
	ALPHANUMERAL INPUT
	DELETE TOOL NAME
	MILL
	SENSOR
	DRILL
	BURNISH
	TAP
	REAMER
	END MILL
	CHAMFEWR

	TOOL NAME
	CHAMFER
	BORE
	BACK BORE
	SPOT FACE
	BACK SPOT FACE
	CORNER MILL
	SHELL
	ANGULAR
	SIDE MILL
	METAL SAW

7-2 Setting the Life Units (Setting the life units for each tool)

Life units..... Minutes: Time,
Times: ATC times,
Length: Cutting length,
Hole: Number of holes (number of holes machined by a canned cycle for drilling/tapping)

	Unit Setting
	Minutes: Time
	Times: ATC times,
	Length: Cutting length,
	Hole: Number of holes

Initially, “Minutes” is selected. When you want to alter it;

① Using the [HOME], [PAGE DOWN], [↑], and [↓] keys, move the cursor to the tool number for which you want to set a life unit. Or, enter [N] followed by a tool number (numerical value), and press [↓].
② Press [F2/UNIT] → A Life Unit Setting window is displayed.
③ Using the [↓] key, move the cursor and select a life unit.
④ Press [INPUT] → The Life Unit Setting window disappears and the life unit is set.
⊙ When interrupting operation (turning off the window), press [F2/UNIT].

7-3 Setting the Life Value

① Using the [HOME], [PAGE DOWN], [↑], and [↓] keys, move the cursor to the tool number for which you want to set a life value. Or, enter [N] followed by a tool number (numerical value), and press [↓].
② Using the [→] key, move the cursor to the [SETTING] field.
③ Input a life value (numerical value) and press [INPUT].

For a tool life value, set the life value preset for each tool, When the tool with unknown life value is used, machining is performed assuming a set value zero. When the tool becomes unsharp, take note of the life use data. That data is used as a life set value. The tool is judge out of life in the following cases:

- Rough finishing surface
- Occurrence of burr
- High cutting sound
- Much chattering
- Adhesion of a constituent tool nose

7-4 Deleting the Use and Setting Data

- Deleting the Use, Setting, and Status Data for All the Tools
  When there are old data remaining, delete all the data first.

① Press [F7/DATA CLEAR] → A Delete Item window is displayed.
② Using the [↓] key, move the cursor to [DELETE (USE, SETTING, & STATUS) FOR ALL TOOLS].
③ Press [INPUT] → A message, "O.K. ? Y-YES N-NO," is displayed.
④ Press [Y] → The Delete Item window disappears and the use and setting data for all the tools are deleted.

⊙ When interrupting operation (turning off the Delete Item window), press [F7/DATA CLEAR] again or press [N] in ④.

	DELETE ITEM
	DELETE USE & STATUS
	DELETE (USE & STATUS) FOR ALL TOOLS
	DELETE (USE, SETTING & STATUS) FOR ALL TOOLS

- Clearing the use value when the out-of-life tool is changed

① Using the [HOME P], [P], [↑], and [↓] keys, move the cursor to the tool number whose tool was changed. Or, enter [N] followed by the tool number (numerical value), and press [↓].
② Move the cursor to [DELETE USE & STATUS].
③ Press [INPUT] → A message, "O.K. ? Y-YES N-NO," is displayed.
④ Press [Y] → The Delete Item window disappears and the tool use data and status are deleted. Or,

① Using the [HOME P], [P], [↑], and [↓] keys, move the cursor to the tool number whose tool was changed. Or, input the tool number (numerical value) and press [↓].
② Using the [→] key, move the cursor to [USE].
③ Press [Y] following by [INPUT] → The use value and status are deleted.

8. Spare Tool Screen

In the screen shown in Fig. 1, press F6/SPARE TOOL. The SPARE TOOL screen in Fig. 2 appears.

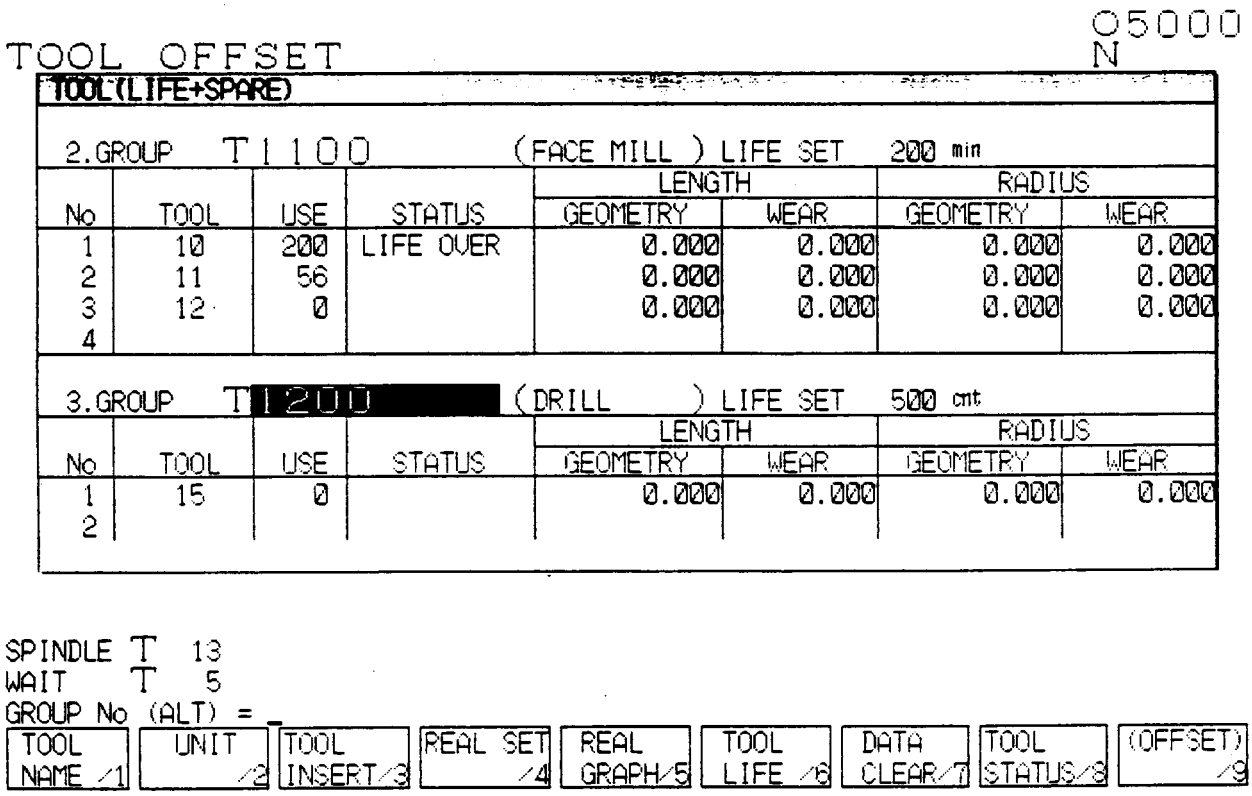


Fig. 2

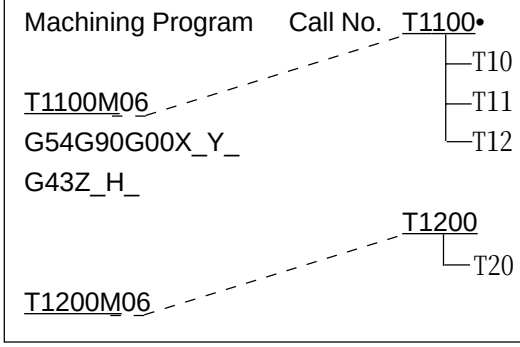
8-1 Setting the Call Number

① Move the cursor to a position of [CALL NO.]

② Input a call number (1.000 or greater) and press INPUT.

⊙ The call number must be greater than the value set in the monitor parameter No. 8078.
(Standard value: 1,000)

What Is the Call Number ?
This is a number used to change tools in the machining program. It is a tool number for each tool group.



8-2 Setting the Tool Name

Set a tool name for the tool group. The spare tool function is not affected even if the tool name is not set.

① See 7-2 in 7. TOOL LIFE SCREEN.

8-3 Setting the Life Units

Set the life units for the tool group.

① See 7-3 in 7. TOOL LIFE SCREEN.

8-4 Setting the Life Value

Set a life value for the tool group.

① Move the cursor to [LIFE SETTING].

② Input a life value (numerical value) for the tool group and press INPUT.

8-5 Registering the Call Number (Spare Group)

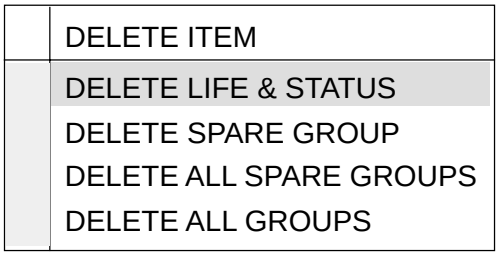
① Move the cursor to a call number (any number will do).

② Press F3/TOOL INSERT.

③ Input the call number (numerical value) you want to register additionally and press INPUT
→ With the cursor located at a position of the added call number, the call numbers are redisplayed in the numerical order.

8-6 Deleting the Call Number (Spare Group)

- Deleting the arbitrary call number (spare group)
  - ① Move the cursor to the call number you want to delete.
  - ② Press 0 followed by INPUT → The call number and spare tool are deleted.



- Deleting all the call numbers (spare group)
  - ① Press F7/DATA CLEAR → The Delete Item window is displayed.
  - ② Using the ↓ key, move the cursor to [DELETE ALL GROUPS].
  - ③ Press INPUT → A message, “O.K. ? Y-YES N-NO,” appears.
  - ④ Press Y → The Delete Item window disappears and all the call numbers and spare tools are deleted.
  - ⊙ When interrupting operation, press F7/DATA CLEAR again.

8-7 Registering the Spare Tool

- Registering the first spare tool
  - ① Move the cursor to TOOL for ORDER 1.
  - ② Input a tool number which you want to register as a spare tool, and press INPUT.

  (Note) When a spare tool is set in ORDER 1 with the tool name of the call number and its life setting data left unset, the tool name and life set value of the tool registered in ORDER 1 are also set for the call number.
- Additionally registering to the end of the spare tools
  - ① Move the cursor to TOOL for last ORDER X.
  - ② Input the number you want to register as a spare tool, and press INPUT.

- Registering halfway the spare tools
  - ① Move the cursor to the position where you want to add a spare tool.
  - ② Press F3/TOOL INSERT. (The spare tool is inserted into the line above the cursor)
  - ③ Input the number you want to add, and press INPUT.
  - ⊙ When interrupting operation, press F3/TOOL INSERT again.

8-8 Deleting the Spare Tool

- Deleting an arbitrary spare tool
  - ① Move the cursor to the tool you want to delete.
  - ② Press 0 (zero) followed by INPUT.
- Deleting the spare tool group of the call number
  - ① Move the cursor to any position of the call number group you want to delete.
  - ② Press F7/DATA CLEAR → The Delete Item window is displayed.
  - ③ Using the cursor, move the cursor to [DELETE SPARE GROUP].
  - ④ Press INPUT → A message, “O.K. ? Y-YES N-NO,” appears.
  - ⑤ Press Y → The Delete Item window disappears and the spare tools in the group are deleted.
  - ⊙ When interrupting operation, press F7/DATA CLEAR again.

9. Tool Status List Screen

In the screen shown in Fig. 1, press F8/TOOL STATUS.

⊙ When interrupting operation, press F8/TOOL CHECK to display the DEFECTIVE TOOL LIST screen, and then, press F8/TOOL STATUS.

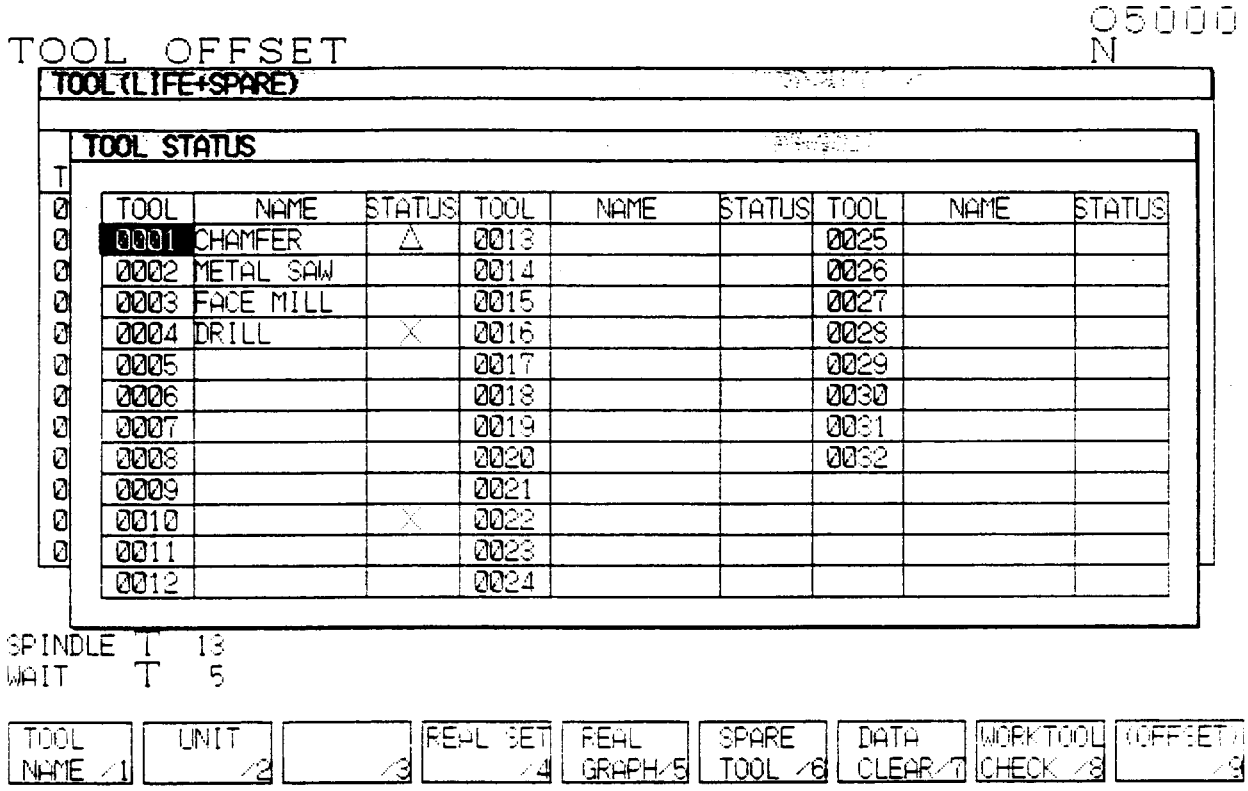


Fig. 3

<Tool Status> ×..... Defective tool (broken or out of life)

Δ ····· Running out of life (❍ ❍% or more of the parameter no. 8060)

Blank..... Usable (Less than ❍❍ % of the parameter no. 8060)

9-1 Deleting and Skipping the Tool Status

① Move the cursor to a tool number.

② Press F7/DATA CLEAR → The Delete Item window appears.

③ Move the cursor to your desired delete item to delete.

④ Press INPUT → A message, “O.K. ? Y-YES N-NO,” appears.

⑤ Press Y → The Delete Item window disappears and the selected item is deleted.

	Delete Item
	Delete the USE and STATUS data.
	Skip STATUS.

9-2 Listing the Defective Tools

Press F8/TOOL CHECK in the TOOL STATUS LIST screen. The DEFECTIVE TOOL LIST screen appears.

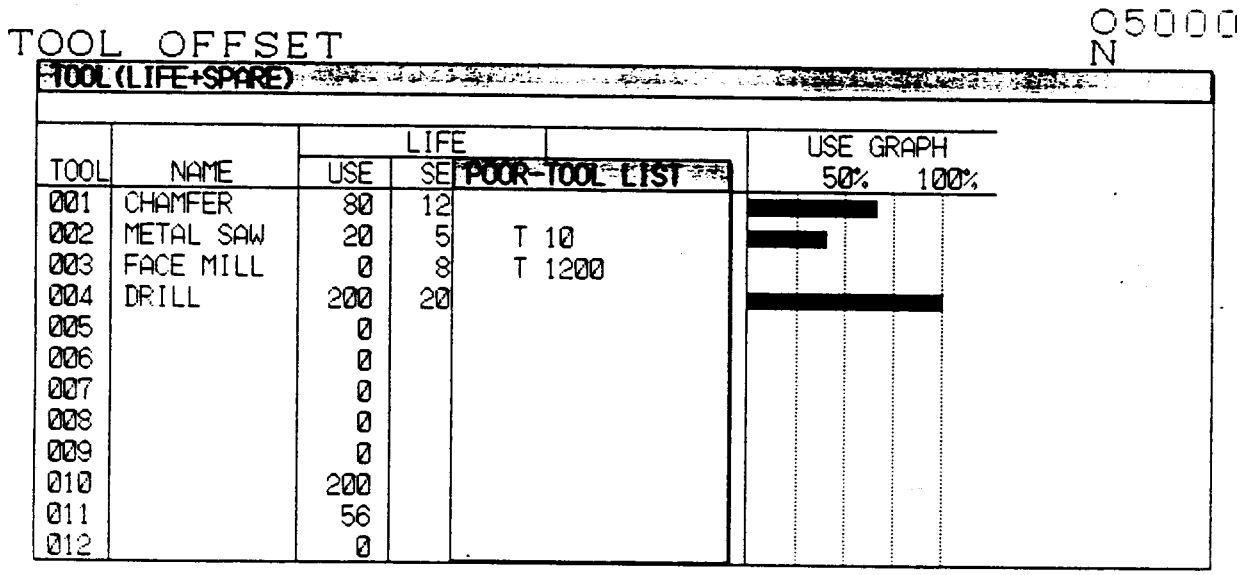


Fig. 4

This screen lists up to 10 tools found defective in one premachining tool check (M51 to M59). Since the defective tools are not displayed cumulatively, the screen displays results of the last premachining tool check.

10. Tool Life Management Parameters And Their Setting

10-1 Displaying the Parameters

Press F4/SYSTEM and enter “1”, INPUT, followed by “N8000.” With the ↓ arrow key, display a cutting monitoring parameter, no. 8000.

10-2 Setting the Parameter

① MDI mode
② Reset state
③ Turn on the WRITE switch.
④ Rewriting of parameters is enabled (SETTING screen)
⑤ Move the cursor to the number you want to set
⑥ Input the setting data with the keys
⑦ Press the INPUT key

10-3 Parameters Related to Tool Life

- No. 8001 #6 ..... Sets the tool life in terms of times when deleting all the data.
- No. 8001 #7 ..... Sets the tool life in terms of minutes when deleting all the modes.
- No. 8003 #0 ..... Spare tool position output 1 to 9
- No. 8003 #7 ..... Clears the tool status and life use value when downloading the life set value by M57.
- No. 8060 ..... % data marked by “• ¢” in the TOOL STATUS LIST screen (Standard value: 80)
- No. 8078 ..... In call number offset, tool numbers greater than this number will be call numbers for the spare tools. (Standard value: 1000)

11. Alarms

780 BROKEN TOOL WAS SELECTED
This alarm is issued when a broken tool is called to the spindle position. When manually resetting this alarm, check and change the tool and reset the alarm. After replacing the tool by a new one, clear the life use value and tool status for the old tool.

781 OUT-OF-LIFE TOOL WAS SELECTED
This alarm is issued when an out-of-life tool is called to the spindle position. When manually resetting this alarm, check and change the tool and reset the alarm. After replacing the tool by a new one, clear the life use value and tool status for the old tool.

782 PREMACHINING TOOL CHECK ERROR
One of the tools checked in a premachining tool check was found defective (out of life, broken, and so on) Replace the defective tool by a new one and clear the life use value and tool status for the defective tool.

783 CUTTING MONITORING FORMAT ERROR
There is a programming error in premachining tool check or tool life data automatic setting. Correct a program command and rerun.

II. Cutting Monitoring Unit

1. Outline

The cutting monitoring unit monitors the cutting loads of the spindle and feed axes of the NC machine tool to prevent abnormal cutting and manufacturing of defective products.
You can set monitoring data and monitoring functions for each tool to ensure detailed monitoring. The tool life management function is required because this unit is interlocked with it.

2. Monitoring Method

2-1 Monitoring Axes

The cutting monitoring unit monitors the loads of the spindle motor and NC feed axes (X-, Y-, and Z-axis) at real time. Monitoring is performed by a combination of a main monitoring axis, where all the monitoring functions work, and a sub-monitoring axes, where overload detection and wear detection work. You set for each tool which axis should monitor in what way.
The main monitoring axis is one of the spindle, X-axis, Y-axis, and Z-axis, and the sub-monitoring axes are the other axes.

- Main monitoring axis ..... Mainstay of cutting monitoring and provides overload detection, no-load detection, wear detection, adaptive control, and idle cutting time reduction.
- Sub-monitoring axes ..... Detects an overload and wear of the sub-monitoring axes in combination with the main monitoring axis.

Normally, monitoring by the main monitoring axis is enough, where the spindle motor has been set as the main monitoring axis. When you want to monitor by the feed axes simultaneously, however, turn on monitoring by the sub-monitoring axes. For a large-diameter drill, and so on whose cutting load is clearly detected in the thrust direction, the Z-axis can be used as the main monitoring axis.
When monitoring the load of the feed axes, monitor them without changing the cutting conditions. They may malfunction if you change a feed rate override, and so on halfway.
When the load of a cutting axis is not known or a cutting load is small, do not monitor by the feed axes because monitoring by them will result in erroneous detection.

2-2 Monitoring Data Unit

Monitoring data is displayed in terms of load factor, down to the first decimal place. A load factor of “100.0 %” has the following meaning.

- Spindle ..... 30-minute rated output (120 % at maximum)
- Feed axes ..... Maximum continuous rated output

2-3 Selecting the Monitoring Function

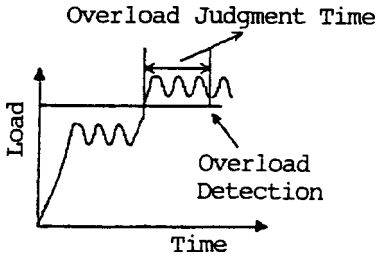
The monitoring functions for the main monitoring axis include overload detection, no-load detection, wear detection, adaptive control, and idle cutting time reduction, and those for the sub-monitoring axes include overload detection and wear detection. Select a desired function for each tool in the SET LOAD screen.

3. Monitoring Functions

3-1 Overload Detection

When a cutting load keeps exceeding an overload detection value for an overload judgment time, this function detects an overload, judging it a cutting error (tool-workpiece interference, improper machining condition, or defective tool nose).
When an overload is detected, an error message, “AXIS OVERLOAD ALARM,” is displayed and the machine stops feeding the axes and stops the spindle.

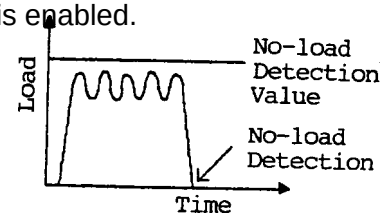
Spindle overload judgment time
..... Parameter 8031 x 0.1 second
X-axis overload judgment time
..... Parameter 8035 x 0.1 second
Y-axis overload judgment time
..... Parameter 8039 x 0.1 second
Z-axis overload judgment time
..... Parameter 8043 x 0.1 second



3-2 No-load Detection

When a cutting load does not exceed a no-load detection value in each cutting, this function detects a state of no load, judging it a cutting error (tool breakage, improper workpiece mounting, no tool setting, or program error), because there was no cutting state found.
When the state of no load is detected, an error message, “NO-LOAD ALARM,” is displayed and the machine stops feeding the axes and stops the spindle.
(Note) No-load detection is activated when overload detection is enabled.
Each cutting is;

- Spindle ..... Between M03 and M04/M05
  (M13 or M14 also allowed)
- Feed axes ..... Between start and end of cutting feed



3-3 Wear Detection

When a cutting load reaches a state of wear load, that tool’s status is regarded “worn out.” Machining continues even if wear is detected, but that tool will be treated in the same manner as an out-of-life tool in a tool command in next machining.
Since wear detection and adaptive control cannot be enabled simultaneously, disable adaptive control when you enable wear detection. There are two wear detecting methods; load time detection or load calculation detection.
(Note) Wear detection is activated when overload detection is enabled.

(a) Load time detection mode (Wear judgment time parameter ≠ 0)
When a cutting load keeps exceeding a wear detection value for wear judgment time, wear is detected and that tool is regarded worn out.

Spindle wear judgment time

..... Parameter 8032 x 0.1 second

X-axis wear judgment time

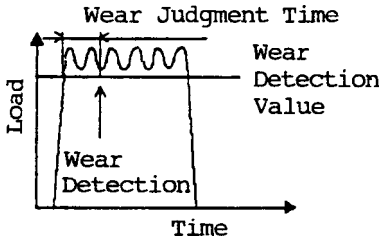
..... Parameter 8036 x 0.1 second

Y-axis wear judgment time

..... Parameter 8040 x 0.1 second

Z-axis wear judgment time

..... Parameter 8044 x 0.1 second



(b) Load calculation detection mode (Wear judgment time parameter = 0)

A cutting load value is calculated out of each cutting load, and when it exceeds a wear detection value, wear is detected and that tool is regarded worn out.

A cutting load value is calculated out of all the load data collected from one cutting by a data sampling and averaging method (sampling and averaging the load values in a certain range). A specific calculation range is specified by a parameter.

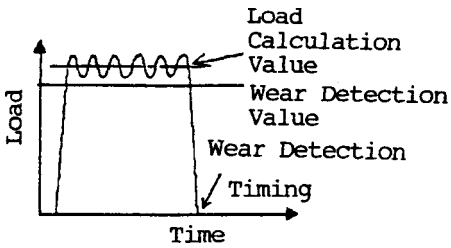
Spindle ..... Parameter 8010/8011

X-axis ..... Parameter 8016/8017

Y-axis ..... Parameter 8022/8023

Z-axis ..... Parameter 8028/8029

Unless the parameters are correct, a maximum load values is regarded a cutting load value.



3-4 Adaptive Control (Feed Rate Override Control)

When a cutting load exceeds a no-load value into a cutting state, a feed rate override is controlled so that the cutting load will stay between an upper-limit adaptive value and lower-limit adaptive value.

When the cutting load is still exceeding the upper-limit adaptive value for 2 seconds even if the override is minimized, an overload is detected, judging it a cutting error.

Since adaptive control and wear detection cannot be enabled simultaneously, disable wear detection when you enable adaptive control.

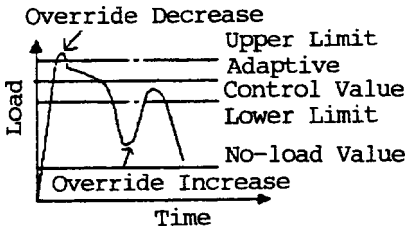
(Note) Overload detection must be working.

Upper-limit adaptive value .....

Adaptive control value x Parameter 8050

Lower-limit adaptive value .....

Adaptive control value x Parameter 8051



Decrease judgment time .....

Parameter 8055 x 0.1 second

Increase judgment time ..... Parameter 8056 x 0.1 second

Override Interval ..... Interval between override increase and decrease

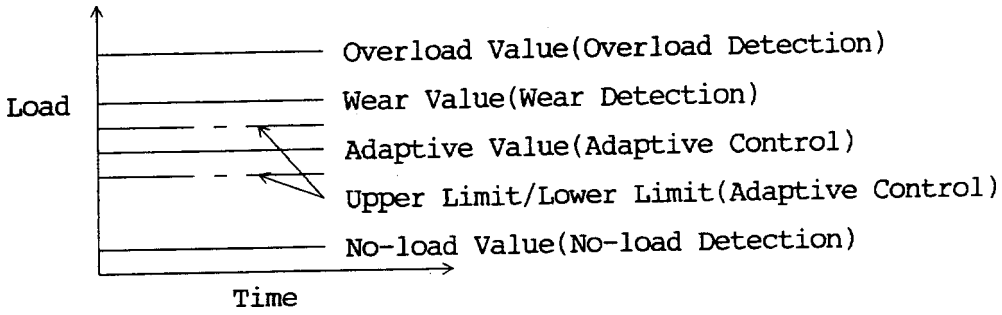
Maximum override ..... Maximum override value

Minimum override ..... Minimum override value

3-5 Idle Cutting Time Reduction (Feed Rate Override Control)

In an idle cutting state where a cutting load is lower than a no-load value, a feed rate override is set to 200 % to reduce an idle cutting time.

* The following figure lists each set value in terms of load versus time.



4. Cutting Monitoring Program

4-1 Cutting Monitoring Special M-code

4-1-1 Stopping/Resetting the Tool Life Management Function and Cutting Monitoring Function (M54/M53)

Since a cutting load does not take place up to an actual machining start point in remachining from halfway, M54 is executed to disable the tool life management function and cutting monitoring function when cutting monitoring makes erroneous detection. Operation is restarted after executing M53 at the actual machining start point.

M54 ..... Stops the tool life management function and cutting monitoring function. Each detecting function related to cutting monitoring does not work. Note that the tool life management related functions do not work.

M53 ..... Cancels M54. When the power is turned on, M53 is selected.

4-1-2 Monitoring Data Automatic Setting (M58/M59)

This function downloads the monitoring data from an NC program. Settable data include an overload value, adaptive value, and wear value for the main monitoring axis, and an overload value for the sub-monitoring axes.

M58; ..... Starts downloading the monitoring data
Txx; ..... Specifies a tool number
1st Sxxx; ..... Sets overload value data for the main monitoring axis
2ndSxxx; ..... Sets adaptive value data for the main monitoring axis
3rdSxxx; ..... Sets wear value data for the main monitoring axis
4th Sxxx; ..... Sets overload data for the X-axis position of the sub-monitoring axis
5th Sxxx; ..... Sets overload data for the Y-axis position of the sub-monitoring axis
6th Sxxx; ..... Sets overload data for the Z-axis position of the sub-monitoring axis
M59; ..... Ends downloading the monitoring data

<Notes>

- When a tool number is a call number, the data is set in the TOOL column of ORDER 1 registered in CALL NO.

CALL NO.1100			
ORDER	TOOL	USE	STATUS
1	25	0	
2	26	0	
3			

M58;
T1100; ..... Indicates a call no. 1100. A data setting area is TOOL 25 of ORDER 1.
S200; ..... Sets a main monitoring axis overload value for TOOL 25 to 20.0.
S150; ..... Sets a main monitoring axis adaptive value for TOOL 25 to 15.0.
M59;

- For Sxxxx;, set the data in the format of xxx.x.
- When the valid data is set, the monitoring functions are enabled.
- When an overload value for the main monitoring axis is set, other monitoring data are cleared.
- When an adaptive value is set, a no-load value is automatically calculated and set.
  No-load value = Adaptive value x Parameter 8009 ÷ Parameter 8008
- The 2nd S-code command onward are omitted unless there is setting data.
- When a data format is not correct, an error message, “783 MONITORING PROGRAM FORMAT ERROR,” appears.
- Specify a spindle rotation command again after downloading the data.

```
M58;
T01;
S125;   Sets a main monitoring axis overload value for the tool no. 01 to 12.5.

T05;
S450;   Sets a main monitoring axis overload value for the tool no. 05 to 45.0.

S370;   Sets a main monitoring axis adaptive value for the tool no. 05 to 37.0.

S0;     Sets a main monitoring axis adaptive value for the tool no. 05 to 37.0.

S176;   Sets a sub-monitoring X-axis overload value for the tool no. 05 to 17.6.

T1100;
S200;   Sets a main monitoring axis overload value for the tool no. 25 to 20.0.

S150;   Sets a main monitoring axis adaptive value for the tool no. 25 to 15.0.

M59;
```

4-2 Sample Program

```
0 7 7 ;
M51;
T01;                Checks the tool status of T01 and T1000 in a premachining tool check →
T1000;
M59;
M56; ..... Selects a feed hold stop because of an out-of-life tool.
G91G28X0Y0Z0;
G91G30X0Y0Z0M19;
T01M06; ..... Tool command (T01 to the spindle).
                The machine stops feeding the axes if the tool is out of life.
G54G90G00X0Y100.;
M03S1000T1000;
G43Z50.H01;         Spindle motor monitoring section
G01Y0.F620;                 ..... Feed monitoring section
G00X10.;
G01Y100.;                   ..... Feed monitoring section
G00Z100.M15;
G91G30Z0;
G30X0Y0M19;
;
T1100M06; ..... Spare tool command
G54G90G00X0Y100.; (T25/T26/T27 to the spindle)
M03S1000;
M45;
G43Z30. H27/2 H26/1 H25;    Selects a spare tool offset
M46;                        (See Tool Life Management)
G01Y0.F620;                 ..... Feed monitoring section
G00X10.;            Spindle motor
G01Y100.;           monitoring section ..... Feed monitoring section
H00Z100.M15;
G91G30Z0;
G30X0Y0M19;
M30;
```

CALL NO.1100			
ORDER	TOOL	USE	STATUS
1	25	0	
2	26	0	
3	27	0	

5. Data Setting

5-1 Selecting the Main Monitoring Axis

In the SET LOAD screen, select the spindle, X-axis, Y-axis, or Z-axis as the main monitoring axis for each tool. When machining is done with this tool, determine on which one of the spindle motor, X-axis, Y-axis, and Z-axis a change of the cutting load is reflected, and set that axis as the main monitoring axis.

Normally, select the spindle motor as the main monitoring axis to monitor, but it is possible to select one of the X-, Y-, and Z-axis as the main monitoring axis to monitor cutting.

The main monitoring axis monitors the loaded state of a selected axis to activate overload detection, no-load detection, wear detection, adaptive control, and idle cutting time reduction.

5-2 Selecting the Monitoring Functions

In the SET LOAD screen, you can turn on/off each function switch for each tool to select how you want to activate the cutting monitoring functions on which axis, such as overload detection, no-load detection, wear detection, adaptive control, and idle cutting time reduction.

5-3 Setting the Monitoring Data

5-3-1 Manual Setting

When the cutting monitoring data for the tool is known, set in the SET LOAD screen the monitoring data corresponding to a tool number (pot) at setup time.

① Setting the individual data

To set the individual data such as OVERLOAD, WEAR, ADAPTIVE, and NO LOAD, move the cursor to a data setting position, input the setting data, and press INPUT.

② Setting the data with a reference value

This is a setting method with a load reference value. When a cutting load reference value for that tool is input with the cursor located at REF. VALUE, the monitoring data for the function set to "● ON" is automatically set according to the parameter setting. The data for the function set to "● OFF" is cleared.

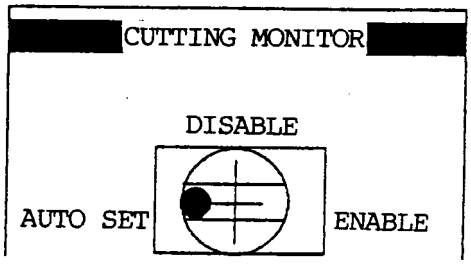
<Example> Calculated load values when the spindle motor's reference value is input

- OVERLOAD ..... (Reference value x Parameter no. 8006/100)
  (X: 8012, Y: 8018, Z: 8024)
- WEAR ..... (Reference value x Parameter no. 8007/100)
  (X: 8013, Y: 8019, Z: 8025)
- ADAPTIVE ..... (Reference value x Parameter no. 8008/100)
  (X: 8014, Y: 8020, Z: 8026)
- NO LOAD ..... (Reference value x Parameter no. 8009/100)
  (X: 8015, Y: 8021, Z: 8027)

③ Setting the data by downloading

Download the monitoring data from the NC program.

5-3-2 Automatic Setting



Set the CUTTING MONITOR mode selector switch on the operation panel to AUTO SET and perform teach cutting. The load reference value for the tool used for machining is calculated from the cutting data for the monitoring section and set. If the reference value is set, the monitoring data for the selected function is automatically set as with 2 Setting the data with the reference value in 5-3-1 Manual Setting.

Monitoring is performed with the automatically set data. When the monitoring data is low, however, erroneous detection may occur because the monitoring conditions become strict. If this is the case, alter the monitoring data to change the monitoring conditions, and monitor cutting. The data is altered as described in 1 Setting the individual data in 5-3-1 Manual Setting. When setting the data automatically, the standard data have been set in the parameters as reference data as shown in the table below. When you want to change them depending on the machining conditions or monitoring conditions, set the data according to the working conditions.

	Spindle		X-axis		Y-axis		Z-axis	
	Parameter No	Data	Parameter No	Data	Parameter No	Data	Parameter No	Data
OVERLOAD	8006	150	8012	200	8018	200	8024	200
WEAR	8007	120	8013	120	8019	120	8025	120
ADAPTIVE NO	8008	100	8014	100	8020	100	8026	100
LOAD	8009	60	8015	60	8021	60	8027	60
AV HIG	8010	95	8016	95	8022	95	8028	95
AV LOW	8011	90	8017	85	8023	85	8029	85

<Calculation Example> When the load reference value for teach cutting is 40.0 % in monitoring the spindle motor load = (40 x 150 (parameter 8006))/100 = 60.0 %

5-4 Monitoring Section

The spindle motor is monitored while the spindle is rotating in the forward direction (spindle reverse rotation also allowed), and the NC feed axes are monitored in the cutting feed section. During data unstable time at the time of start, the data for each axis is cancelled for the time specified by the respective parameters.

- Spindle ..... Parameter 8030 x 0.1 second
- X-axis ..... Parameter 8034 x 0.1 second
- Y-axis ..... Parameter 8038 x 0.1 second
- Z-axis ..... Parameter 8042 x 0.1 second

Every time the S-data is altered while the spindle is rotating, a cancellation time is activated. For the feed axes, the cancellation time is activated when cutting starts, but it is not between continuous cutting feed blocks. When the machine is operated in the SINGLE BLOCK mode, an axis is started at machine start time, and therefore, the cancellation time is activated every time. Note that the monitoring conditions are changed by turning on/off the SINGLE BLOCK switch. Monitor cutting at a constant feed rate override.

6. Cutting Monitoring Operation

Prior to starting monitoring operation, set the monitoring functions and monitoring data for the tool whose cutting you want to monitor. The monitoring data becomes valid when the tool is brought from a standby position to the spindle. If the spindle has the same tool since a start of machining after the power is turned on, cutting is not monitored until next tool change. The same applies when the monitoring data is changed halfway monitoring.

To enable the monitoring function, turn on a relevant function switch and set the monitoring data for the function. Overload detection must be working. The following table lists the conditions to enable each of the monitoring function.

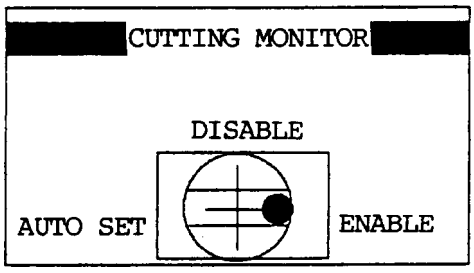
❍ . . . . . Condition required to activate the function

× . . . . . Condition you should not select to activate the function

	Overload Detection	Wear detection		Adaptive control	No-load Detection	Idle Cutting Time Reduction
		By Time	By Calculation			
Overload Detection ON	❍	❍	❍	❍	❍	❍
Overload	❍	❍	❍	❍	❍	❍
Wear Detection ON		❍	❍			
Wear Value		❍	❍			
Adaptive control ON		×	×	❍		
Adaptive Value				❍		
No-load Detection ON					❍	
No-load Value				○	❍	❍
Idle Cutting Reduction ALL/BEFORE/AFTER						❍

<Example> To enable this function in wear detection, set the overload detecting function to ON, an overload value to non-zero, wear detecting function to ON, and a wear value to non-zero in the SET LOAD screen in case of either wear detection by time or calculation. However, it will not work if the adaptive control function is set to ON.

6-1 MONITORING CONDITIONS



Basic monitoring conditions are as follows:

① Completion of tool change operation
(From the standby position to the spindle)

② In memory operation

③ The CUTTING MONITOR switch is set to ENABLE

④ M53 mode (M54 is not specified in the program)

⑤ Overload detection is working (Unless overload detection is working, the other monitoring functions will not)

When the above-mentioned conditions are met, the following monitoring sections are monitored according to the selected function and set monitoring data.

Spindle motor ..... During spindle forward rotation (reverse rotation also allowed by parameter setting)

Feed axis motor ..... During cutting feed

6-2 Overload Detection

Turn on the overload detecting function for a desired tool and set the overload detection data. Overload monitoring operation is activated in the monitoring section. When the monitoring conditions are met, an overload is monitored simultaneously with four axes.

To activate each of the functions such as no-load detection, wear detection, adaptive control, and idle cutting time reduction, be sure to activate overload detection.

When an overload is detected, an error message, “AXIS OVERLOAD ALARM,” is displayed and the machine stops the spindle after stopping axis feed.

(1) Manual return

Reset the NC unit to move each axis to a safe position, and replace a broken tool. After replacing it, clear the life use value and maximum load value for the broken tool.

(2) Automatic return (depending on the machine specification)

In case of automatic return, store the broken tool status. When an overload is detected, the NC unit is reset by a sequence and a return program is started by an NC unit external sequence search to perform next machining. After a broken tool is replaced, clear the tool status and life use value for the replaced tool, and the maximum load value.

6-3 No-load Detection

Turn on the no-load detecting function for a desired tool and set the no-load detection data. No-load detection is made every time cutting is completed.

When no-load detection is made, an error message, “NO-LOAD ALARM,” is displayed and the machine stops the spindle after stopping axis feed. A resetting method is the same as in 6-2 Overload Detection.

Each cutting is;

- Spindle ..... Between M03 and M04/M05 (M13 or M14 also allowed)
- Feed axes ..... Between start and end of cutting feed

6-4 Wear Detection

Turn on the wear detecting function for a desired tool and set the wear detection data. Wear detecting operation is activated. When wear is detected, that tool is regarded worn out, but cutting continues as it is. If there is a spare tool in next machining, it will be selected automatically. If a worn-out tool is brought to the spindle, an error message, “781 OUT-OF-LIFE TOOL WAS SELECTED,” is displayed. When M56 has been executed in advance, the machine stops after stopping axis feed. Replace a worn-out tool and clear the tool status and life use value for the replaced tool, and the maximum load value.

6-5 Adaptive Control

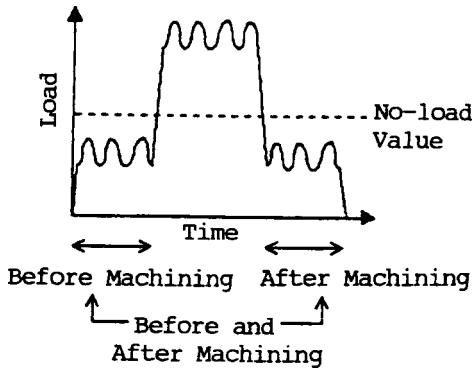
Turn on the adaptive control function for a desired tool and set the adaptive data and no-load data. Adaptive control is activated.

When a cutting load exceeds a no-load value into a cutting state, a feed rate override is controlled so that the cutting load will stay between an upper-limit adaptive value and lower-limit adaptive value.

When the cutting load keeps on exceeding the upper-limit adaptive value for a descent judgment time, the override is decreased, and when it keeps staying lower than the lower-limit adaptive value for an increase judgment time, it is increased.

Even if the override is minimized, an overload is detected if the cutting load keeps exceeding the upper-limit adaptive falue.

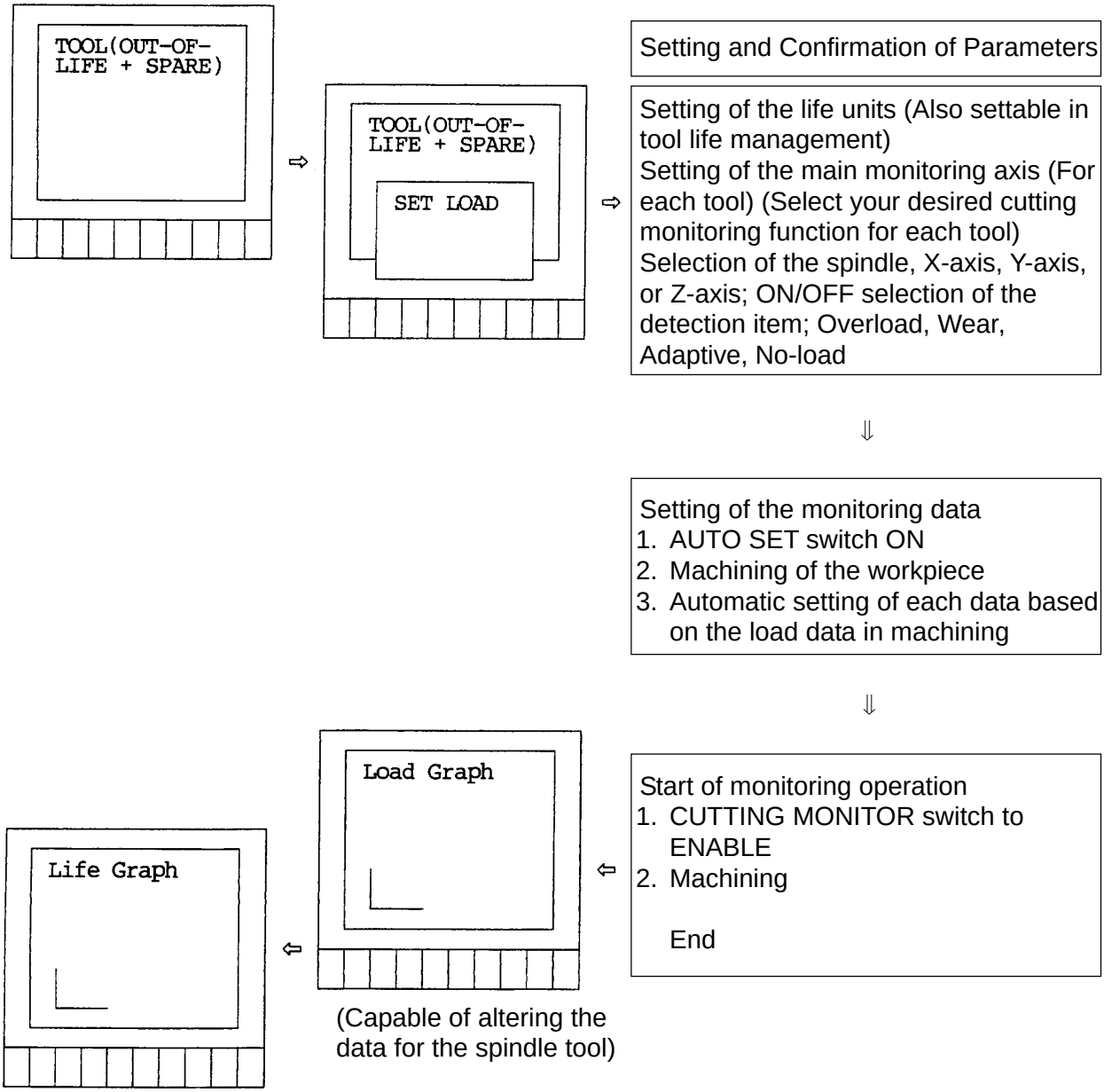
6-6 Idle Cutting Time Reduction



Select the idle cutting function to either “ALL,” “BEFORE,” or “AFTER” for the tool whose idle cutting time you want to reduce, and set the no-load data. A feed rate override is set to 200 % in the idle cutting state in which the cutting load does not exceed a no-load value. The idle cutting function can be effected before, after, or before and after machining. Select either one of them.

(Note) Do not use a tool whose loaded state is difficult to determine. As an undercut load is expected, select a proper tool.

6-7 Cutting Monitoring Operational Procedure



7. Set Load Screen

① Press F3/TOOL in the OVERALL screen → The TOOL screen appears and displays the following functions.
TOOL NAME/1 | REF. GAUGE/2 | TOOL CHANGE/3 | /4 | /5
SAFETY GUARD/6 | DATA CLEAR/7 | SENSOR SET/8 | LIFE SPARE/9

② Press F9/LIFE SPARE → The TOOL LIFE screen appears and displays the following functions.
TOOL NAME/1 | UNIT/2 | /3 | REAL SET/4 | REAL GRAPH/5
SPARE TOOL/6 | DATA CLEAR/7 | TOOL STATUS/8 | (OFFSET)/9

③ Press F4/REAL SET → The SET LOAD screen appears.

⊙ When interrupting operation (turning off the SET LOAD screen), pres F4/REAL SET zagain.

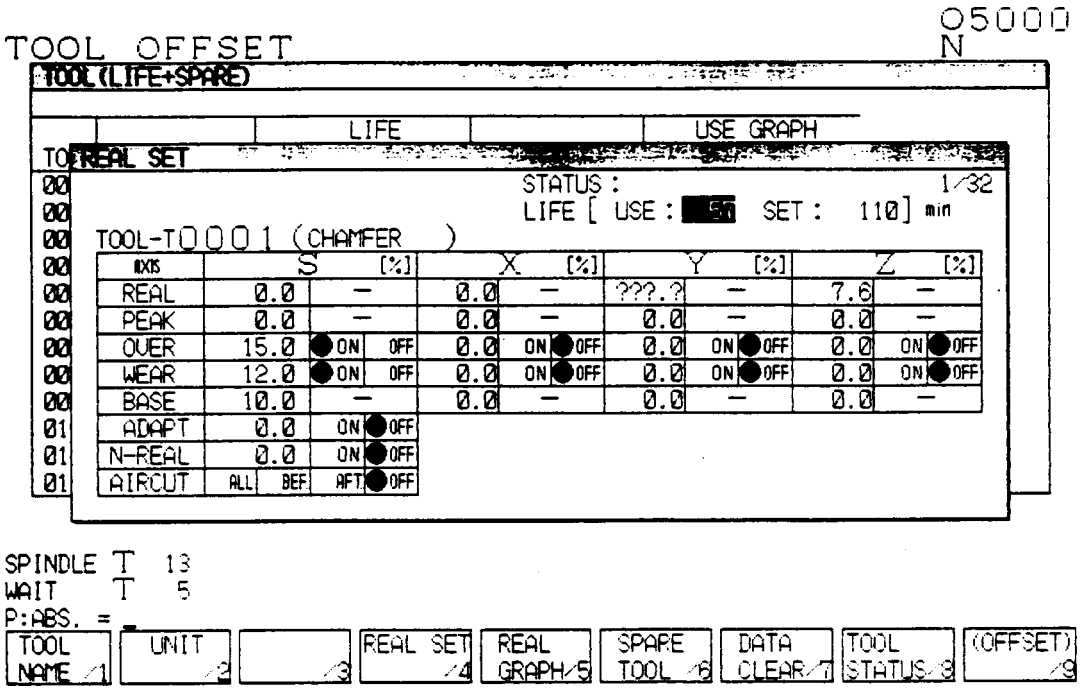


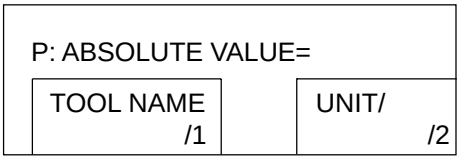
Fig. 5

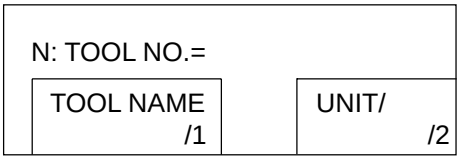
- Screen selecting method

① Press the ⇧ (previous page) key to display the previous TOOL NUMBER screen.

② Press the ⇩ (next page) key to display the next TOOL NUMBER screen.

③ Enter N following by a tool number (numerical value) and press the ↓ key. Pressing N zreplaces “P: ABSOLUTE VALUE=” on the lower left of the screen by “N: TOOL NO.=.”





- Input position selecting method
  Using the cursor move keys, [↑] [↓] [→] , and [←], move the cursor to the position where you want to alter or input the data.

7-1 Selecting the Main Monitoring Axis

Initially, the spindle(S) has been set as the main monitoring axis. When you want to change it;

① Using the [↓] key, move the cursor to "AXIS."

② Select and enter an axis name you want to change to; [S] , [X] , [Y] , or [Z].

③ Press [INPUT].

(Note) Changing the main monitoring axis means to replace it and one of the sub-monitoring axes with each other. The set data are also replaced, but ADAPTIVE, NO-LOAD, and IDLE CUT data are cleared.

7-2 Selecting the Monitoring Function

Select for each tool which monitoring functions should be activated for the main monitoring axis and sub-monitoring axis. Set to "● ON" a relevant selector switch for the function you want to select.

- Turning on the function
  ① Using the [↑], [↓], [→], and [←] keys, move the cursor to "ON" of your desired monitoring function.
  ② Press [INPUT] → A dot mark, "●", appears on the "ON" side.
- Turning off the function
  ① Using the [↑], [↓], [→], and [←], move the cursor to "OFF" of the monitoring function you want to deselect.
  ② Press [INPUT] → A dot mark, "●", appears on the "OFF" side.

7-3 Setting the Data

Set the monitoring data for each tool. There are two kinds of methods available; setting for individual items and setting with a load reference value.

- Setting the individual data
  To set the individual data such as OVERLOAD, WEAR, ADAPTIVE, and NO LOAD, move the cursor to a data setting position, input the setting data, and press [INPUT].
- Setting the data with the reference value
  This is a setting method with the load reference value. When a cutting load reference value for that tool is input with the cursor located at REF. VALUE, the monitoring data for the function set to "● ON" is automatically calculated and set according to the parameter setting. The data for the function set to "● OFF" is cleared.

<Example> Calculated load values when the spindle motor's reference value is input

- OVERLOAD ..... (Reference value x Parameter no. 8006/100)
  (X: 8012, Y: 8018, Z: 8024)
- WEAR ..... (Reference value x Parameter no. 8007/100)
  (X: 8013, Y: 8019, Z: 8025)
- ADAPTIVE..... (Reference value x Parameter no. 8008/100)
  (X: 8014, Y: 8020, Z: 8026)
- NO LOAD ..... (Reference value x Parameter no. 8009/100)
  (X: 8015, Y: 8021, Z: 8027)

7-4 Deleting the Data

① Press [F7/DATA DELETE] →
The Delete Screen window appears.

② Using the ↓ key, move the cursor to a selected item.

③ Press INPUT → A message, “O.K. ? Y-YES N-NO,” appears.

④ Press Y → The Delete Item window disappears and the data are deleted.

⊙ When interrupting operation (turning off the window), press F7/DATA CLEAR again or press N zin ④.

	DELETE ITEM
	DELETE MAXIMUM VALUE
	DELETE ALL DATA
	DELETE MAXIMUM VALUES OF ALL TOOLS
	DELETE ALL DATA OF ALL TOOLS

- DELETE MAXIMUM VALUE... Deletes the maximum value data of the tool displayed in the screen.
- DELETE ALL DATA ..... Deletes all the monitoring data of the tool displayed in the screen.
- DELETE MAXIMUM VALUES OF ALL TOOLS ..... Deletes the maximum value data of all the tools.
- DELETE ALL DATA OF ALL TOOLS ..... Deletes all the monitoring tools of all the tools. When this is done, the main monitoring axis is the spindle and overload detection is set to “ON.”

(Note) When you want to turn off overload detection with DELETE ALL DATA OF ALL TOOLS selected, change setting of the parameter no. 8001. (See 10-4)

8. Load Graph Screen

Press the CUT MONITOR key on the operation panel or press F5/REAL GRAPH.
The LOAD GRAPH screen appears.

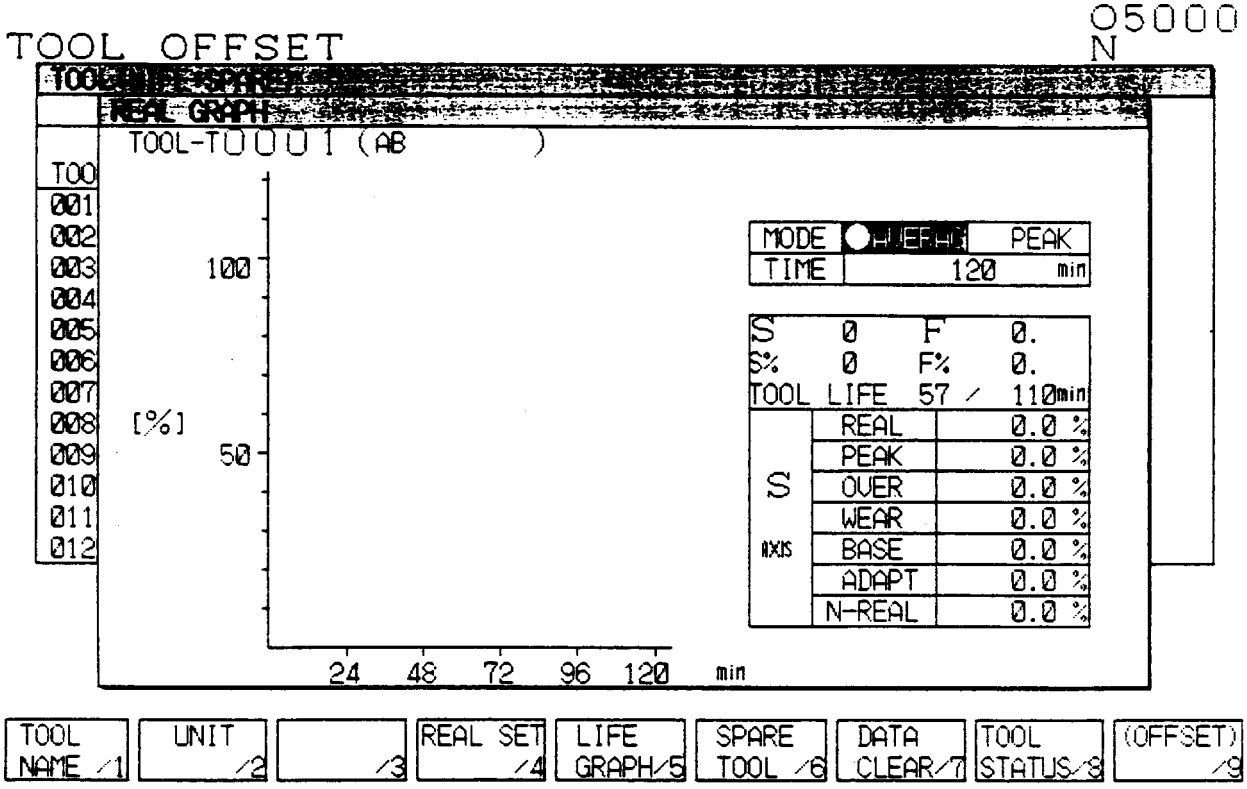


Fig. 6

The LOAD GRAPH screen is a display with respect to the spindle tool and graphs a loaded state after one of the spindle, X-axis, Y-axis, or Z-axis is placed under monitoring. This screen is displayed when the MONITOR mode is set to ENABLE.

- Display time ..... Set arbitrary data whose display time width is 1 minute or longer. Move the cursor to TIME, input display time data, and press INPUT.
- Display mode
  AVERAGE ..... Displays an average value of load data per unit time of display.
  PEAK ..... Displays the maximum and minimum values of load data per unit time of display in the form of bar graph.
- Altering the life data and monitoring data
  The life data and monitoring data being displayed in the load graph can be altered through this screen.
  ① Using the ↑ and ↓ keys, move the cursor to a setting position.
  ② Input the numerical data and press INPUT.
- Use the Page keys and to select the display for the spindle, X-axis, Y-axis, or Z-axis.

9. Life Graph Screen

Press the F5/LIFE GRAPH in the LOAD GRAPH screen. The LIFE GRAPH screen appears.

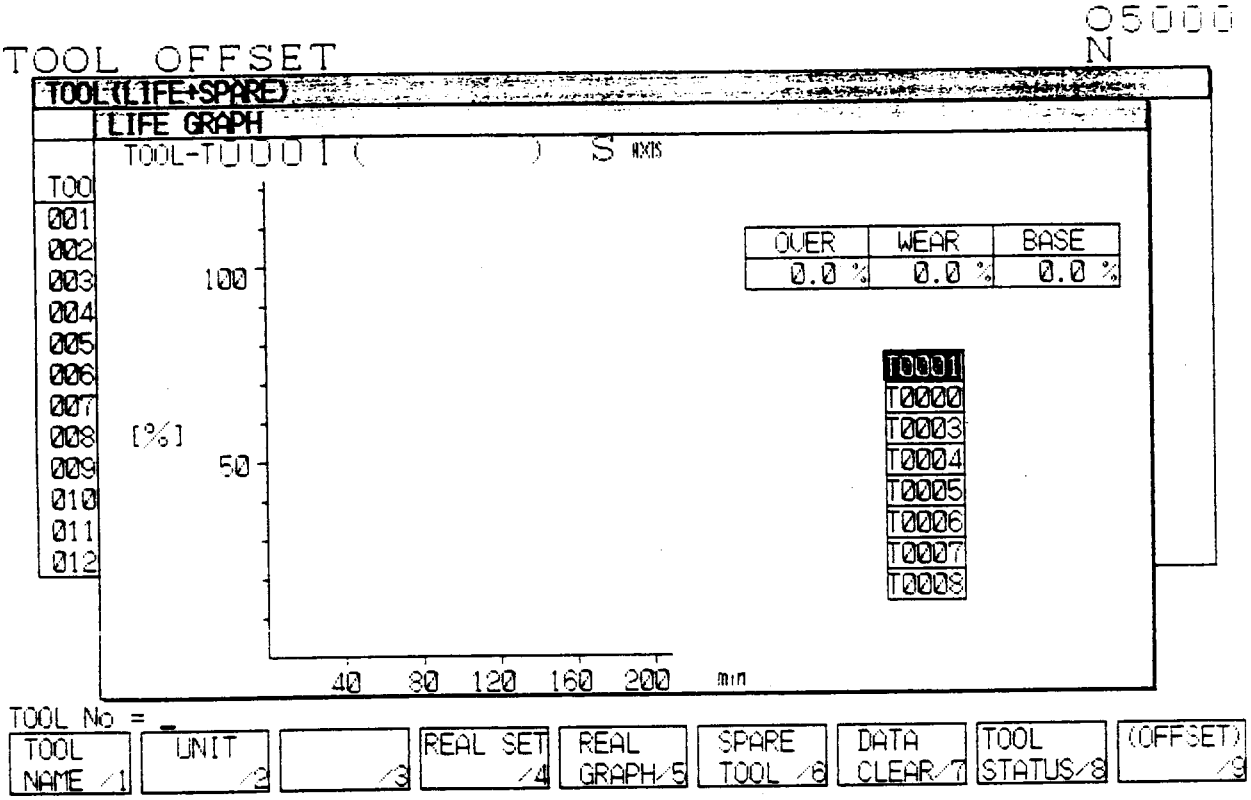


Fig. 7

The LIFE GRAPH screen graphs the cutting load data ranging from a start of using a life managed tool to its point of running out of life. You can see at a glance the loaded state by lapse of time until the tool runs out of life.

This screen can display the life graphs for preregistered 8 tools; it displays the post-registration cutting data. No graph is displayed unless monitoring operation is enabled.

- Selecting the display

  Select the life graph for the tool number you want to display.

  ① Using the ↑ and ↓ keys, move the cursor to the tool number whose life graph you want to display.

  ② Press INPUT → The life graph for the selected tool is displayed.

- Setting the tool number

  Set the tool number you want to display in the life graph.

  ① Using the ↑ and ↓ keys, move the cursor to a setting position.

  ② Input the tool number you want to set.

  ③ Press INPUT → The registered tool number is set.

  (Note) The life graph collects the cutting data on the eight registered tools. If a tool which is halfway its life is registered, the cutting data after that point will be displayed.

10. Display And Setting Of Parameters

10-1 Displaying the Parameter

Press F4/SYSTEM and enter “1”, INPUT, and “N8000.” Then, use the ↓ key to display the parameter no. 8000 for cutting monitoring.

10-2 Setting the Parameter

① Select the MDI mode
② Press the RESET switch
③ Turn on the WRITE switch
④ Enable rewriting of parameters (SETTING screen)
⑤ Move the cursor to the number you want to set
⑥ Input the setting data
⑦ Press the INPUT key

10-3 Selecting the Monitoring Data System (Parameter 8000)

Select whether the monitoring data should be of absolute value system or reference value system. The absolute value system assumes A/D input data “0” to be monitoring data “0”. The reference value system assumes A/D data just after cancellation time in the monitoring section to be “0”.

- Parameter 8000 (0: Absolute value system 1: Reference value system)
  Bit 0 ..... Spindle monitoring reference value system
  Bit 1 ..... X-axis monitoring reference value system
  Bit 2 ..... Y-axis monitoring reference value system
  Bit 3 ..... Z-axis monitoring reference value system

10-4 Selecting Initial Setting (Parameter 8001)

When you select and execute “DELETE ALL DATA OF ALL TOOLS” with the DATA DELETE function in the SET LOAD screen, the monitoring functions selected with this parameter are set to “ON.”

- Parameter 8001 (0: Not selected, 1: selected)
  Bit 0 ..... Overload of the main monitoring axis
  Bit 1 ..... Wear of the main monitoring axis
  Bit 2 ..... Adaptive control of the main monitoring axis
  Bit 3 .....
  Bit 4 ..... Overload of the sub-monitoring axis
  Bit 5 ..... Wear of the sub-monitoring axis
  Bit 6 ..... Initialization in the tool life times mode
  Bit 7 ..... Initialization in the tool life time mode

10-5 Selecting the Function (Parameter 8002)

- Parameter 8002 (0: Not selected, 1: selected)
  Bit 0 ..... Spindle reverse rotation

10-6 Displaying the Monitoring Data (Parameter 8004)

- Parameter 8004 (0: Displays the data in percentage(%), 1: Displays the data in ampere(A))
  Bit 6 = 1 ..... Displays the monitoring data in ampere(A)

A displayed data amount complies with the parameters no. 8070 to no. 8073.

10-7 Automatic Calculation Parameters (8006 to 8029)

You set the parameters which automatically calculate the monitoring data (detection values) for the spindle, X-axis, Y-axis, and Z-axis, and those which calculate cutting load reference values.

Unit of Setting Values: %

	Spindle	X-axis	Y-axis	Z-axis
OVERLOAD	8006	8012	8018	8024
WEAR	8007	8013	8019	8025
ADAPTIVE	8008	8014	8020	8026
NO LOAD	8009	8015	8021	8027
AV_HIG	8010	8016	8022	8028
AV_LOW	8011	8017	8023	8029

<Calculation Example> When the spindle is monitored and a machining load reference value is 40 %
An overload is set as follows:
40 x 150 (Parameter 8006)/100 = 60 %

(1) Detection value calculation parameters

- Spindle overload value = Key-input reference load (Calculated load value from OR cutting) x Parameter 8006/100
- X-axis overload value = Key-input reference load (Calculated load value from OR cutting) x Parameter 8012/100
- Y-axis overload value = Key-input reference load (Calculated load value from OR cutting) x Parameter 8018/100
- Z-axis overload value = Key-input reference load (Calculated load value from OR cutting) x Parameter 8024/100
- Wear value = Key-input reference load (Calculated load value from OR cutting) x Parameter 8007(8013, 8019, or 8025)/100
- Adaptive value = Key-input reference load (Calculated load value from OR cutting) x Parameter 8008(8014, 8020, or 8026)/100
- No-load value = Key-input reference load (Calculated load value from OR cutting) x Parameter 8009(8015, 8021, or 8027)/100

(2) Reference load calculation parameters

- Spindle load calculation value = $\frac{\Sigma \text{ Load data (Parameter 8010)}}{\Sigma \text{ No. of load data (Parameter 8010)}}$ / Parameter 8011

- X-axis load calculation value = $\frac{\Sigma \text{ Load data (Parameter 8016)}}{\Sigma \text{ No. of load data (Parameter 8016)}}$ / Parameter 8017

- Y-axis load calculation value = $\frac{\Sigma \text{ Load data (Parameter 8022)}}{\Sigma \text{ No. of load data (Parameter 8022)}}$ / Parameter 8023

- Y-axis load calculation value = $\frac{\Sigma \text{ Load data (Parameter 8028)}}{\Sigma \text{ No. of load data (Parameter 8028)}}$ / Parameter 8029

10-8 Timer Parameters (8030 to 8044)

You set the timer data used for monitoring. A timer setting increment is 0.1 second.

	Spindle	X-axis	Y-axis	Z-axis
Cancellation time	8030	8034	8038	8042
Overload judgment time	8031	8035	8039	8043
Wear judgment time	8032	8036	8040	8044

- The cancellation time refers to a data invalid time upon starting the spindle, changing a spindle speed, or starting cutting feed.
- The overload judgment time refers to a time required to determine an overload when a cutting load exceeds an overload value.
- The wear judgment time refers to a time required to determine wear when a cutting load exceeds a wear value.

When you need more than one spindle cancellation time, use the following parameters other than 8030.

Low-speed Gear (1st)		High-speed Gear (2nd)	
Switched rpm for spindle cancellation time	Spindle cancellation time 1-4	Switched rpm for spindle cancellation time	Spindle cancellation time 5-8
	Cancellation time 1 8030		Cancellation time 5 8066
1-2 switching point 8080	2 8063	5-6 switching point 8083	6 8067
2-3 8081	3 8064	6-7 8084	7 8068
3-4 8082	4 8065	7-8 8085	8 8069

(Notes)

- When none of the 1st through 4th gears has been selected (when the machine has no gear, see the parameter for the low-speed gear).
- When the 3rd or 4th gears is selected, see the parameter for the high-speed gear.
- Even if the high-speed gear has been selected, see 8030 when 8083 equals 0 and 8066 equals 0.
- See 8030 when 8080 equals 0.

10-9 Adaptive Parameters

(1) Upper-limit adaptive value (Parameter 8050)
A parameter to set a target load upper-limit value when activating adaptive control.
Upper-limit adaptive value = Adaptive value x Parameter 8050/100

(2) Lower-limit adaptive value (Parameter 8051)
A parameter to set a target load lower-limit value when activating adaptive control.
Lower-limit adaptive value = Adaptive value x Parameter 8051/100

(3) Adaptive maximum override (Parameter 8052)
Set a maximum override for adaptive control to “110 to 200.”

(4) Adaptive minimum override (Parameter 8053)
Set a minimum override for adaptive control to “10 to 90.”

(5) Adaptive override step (Parameter 8054)
Set to “10 to 50” feed rate override change intervals which are controlled by adaptive control.

(6) Adaptive decrease judgment time (Parameter 8055)
When a cutting load exceeds an upper-limit adaptive value in adaptive control, set a time to determine to decrease an override. A setting increment is 0.1 second.

(7) Adaptive decrease judgment time (Parameter 8056)
When a cutting load becomes lower than a lower-limit adaptive value in adaptive control, set a time to determine to increase an override. A setting increment is 0.1 second.

10-10 Data Parameters

(1) No. of spindle data averaging times (Parameter 8058)
Set to “0 to 40” the number of times to average the spindle load data form the A/D.

(2) No. of feed axis data averaging times (Parameter 8059)
Set to “0 to 40” the number of times to average the feed axis load data from the servo.

(3) Spindle load 100 % data (Parameter 8070)

(4) X-axis load 100 % data (Parameter 8071)

(5) Y-axis load 100 % data (Parameter 8072)

(6) Z-axis load 100 % data (Parameter 8073)

11. Alarms

770 SPINDLE OVERLOAD ALARM

A spindle load exceeded an overload value during cutting.

771 NO-LOAD ALARM

Since machining is completed without a no-load value being exceeded by a cutting load for the main monitoring axis, a no-load alarm results, judging it erroneous mounting or tool breakage.

772 X-AXIS OVERLOAD ALARM

An X-axis load exceeded an overload value during cutting.

773 Y-AXIS OVERLOAD ALARM

A X-axis load exceeded an overload value during cutting.

774 Z-AXIS OVERLOAD ALARM

A Z-axis load exceeded an overload value during cutting.

When the above-mentioned alarms take place, the machine stops feeding the axes and stops the spindle in 3 seconds. When manually returning the machine, move each axis to a safe position, check a tool, and replace it if necessary.

III. Tool Peculiar Number Function

1. Function

The tool peculiar number function is to assign a peculiar number to each tool to control it. There are two kinds of control methods depending on how a tool number is assigned; they are a tool peculiar number method or number method by functions. They cannot be mixedly used.

2. Tool Peculiar Number Method (Parameter 8004 #7 = 0)

The tool peculiar number method is to assign a peculiar number to each tool. The tool number can be up to 8 digits.

If the tool peculiar number, T3300, is assigned to a 5 mm end mill, T3300 will be always used to specify it in the machining program and the offset data and life data will be managed as the data for T3300.

2-1 Data Setting

After setting a tool into the magazine pot, set a tool number determined for that tool (called a tool peculiar number) by inputting it in “SPECIFIED TOOL” in the TOOL LIFE screen (TOOL OFFSET, TOOL STATUS, TOOL LIFE, SPARE TOOL).

2-2 Program Command

In the machining program, specify a tool command, using the tool peculiar number (specified tool) assigned to the tool. If specified with the tool peculiar number, a tool having a corresponding pot number will be selected out of the pot and tool command number set in the TOOL LIFE screen.

2-3 Spare Tool Management

Spare tool management by the tool peculiar number method differs from general spare tool management by call numbers because each tool has a peculiar number. You use the tool peculiar number (command number) to set which tool is to be used as a spare tool for the tool specified in the program, and manage the spare tool.

Select the tools in order of 1 tool used, 2 unused tool of higher registration order, and 3 last tool used.

(Note) When a number not set for the specified tools is given as a T-code command, an error message, “784 SPECIFIED TOOL REGISTRATION ERROR,” appears.

2-4 Spare Data Setting

Register a tool peculiar number for each pot in “SPECIFIED TOOL” in the TOOL LIFE screen, and then, register a spare tool. Set the tool number specified in the program in “CALL NO.” and set tool peculiar numbers for the spare tools in “ORDER 2” onward. (ORDER 1 automatically becomes the same number as CALL NO.)

In the SPARE TOOL screen in Fig. 2, the spare tools have been set as T1101 and T1102 for the tool peculiar number T1100 specified in the program.

(Note) When no tool command number has been set, an error message, “NO COMMAND NO. IS SET,” appears.

2-5 TOOL LIFE Screen

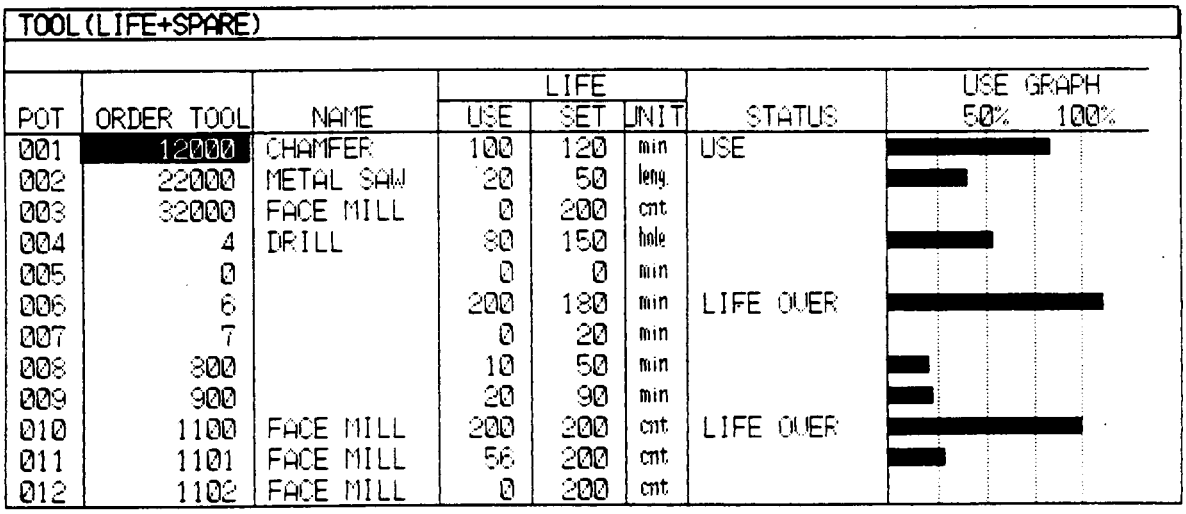


Fig. 8

- Setting the specified tool data
  Move the cursor to a setting position, input a tool command number, and press INPUT to set. An error message, “ALREADY REGISTERED,” appears in case of the tool number for which the data has been already registered.
- Displaying the tool data in the pot order/tool order
  You can use F3/POT/TOOL ORDER in the TOOL LIFE screen to choose to display the tool data in the pot order or tool order.

  Display in the pot order ..... Displays the tool data in the pot number order in the TOOL LIFE screen.

  Display in the tool order ..... Displays the tool data in the tool number order in the TOOL LIFE screen. For the tools with no tool number set for them, the tool data are displayed in the pot order after tool order.

2-6 SPARE TOOL Screen

TOOL(LIFE+SPARE)

1.GROUP T1100 (FACE MILL) LIFE SET 200 cnt

No	TOOL	USE	STATUS	LENGTH GEOMETRY	LENGTH WEAR	RADIUS GEOMETRY	RADIUS WEAR
1	1100	200	LIFE OVER	0.000	0.000	0.000	0.000
2	1101	56		0.000	0.000	0.000	0.000
3	1102	0		0.000	0.000	0.000	0.000
4							

2.GROUP T1200 ( ) LIFE SET 500 min

No	TOOL	USE	STATUS	LENGTH GEOMETRY	LENGTH WEAR	RADIUS GEOMETRY	RADIUS WEAR
1	1200	0		0.000	0.000	0.000	0.000
2	1201	6		0.000	0.000	0.000	0.000

Fig. 9

- Setting the specified tool data
  Move the cursor to a setting position, input a tool command number, and press INPUT to set. An error message, “ALREADY REGISTERED,” appears in case of the tool number for which the data has been already registered.

2-7 TOOL STATUS Screen

TOOL(LIFE+SPARE)

TOOL STATUS

POT	ORDER TOOL	STATUS	POT	ORDER TOOL	STATUS	POT	ORDER TOOL	STATUS
0001	12000	△	0013	0		0025	0	
0002	22000		0014	0		0026	0	
0003	32000		0015	0		0027	0	
0004	4	△	0016	1200		0028	0	
0005	0		0017	1201		0029	0	
0006	6	X	0018	0		0030	0	
0007	7		0019	0		0031	0	
0008	800		0020	0		0032	0	
0009	900		0021	0				
0010	1100	X	0022	0				
0011	1101		0023	0				
0012	1102		0024	0				

Fig. 10

- Setting the specified tool data
  Move the cursor to a setting position, input a tool command number, and press INPUT to set. An error message, “ALREADY REGISTERED,” appears in case of the tool number for which the data has been already registered.

3. Number Method By Functions (Parameter No. 8004 #7 = 1)

The number method by functions is to assign a peculiar tool function number to each tool to manage the tools. Since the tools with an identical function have the same function number, give reference numbers to manage them. The numbers consist of numbers by functions, and reference numbers. The numbers by functions are up to 6 digits, and the reference numbers have fixed digits of two.

Number by Function (6 Digits) + Reference Number (2 Digits)

Since a program command uses a number by functions, a T-code command may have up to 6 digits.
When a function number, T20000, is given to a 10 mm drill, a T20000 tool command is used whenever the 10 mm drill is used in the machining program. Since each tool has different offset data and life data, they are managed as the data for the tool specified by the number by functions and reference number.

3-1 Data Setting

After setting a tool into the magazine pot, set a number by functions determined for that tool by inputting it in “SPECIFIED TOOL” in the TOOL LIFE screen (TOOL OFFSET, TOOL STATUS, TOOL LIFE, SPARE TOOL).

3-2 Program Command

In the machining program, specify a tool command, using the number by functions assigned to the tool. If specified with the number by functions, a tool having a corresponding pot number will be selected out of the pot and tool command number set in the TOOL LIFE screen.

3-3 Spare Tool Management

Spare tool management by the number method by functions manages the tools by the numbers by functions. When the tools have the same function number, they are handled as a spare tool group because they are identical tools, even if their reference numbers differ.
To call the tools in the machining program, specify them with the numbers by functions. Select the tools in order of 1 tool used, 2 unused tool of smaller reference number, and 3 last tool used.

(Note) When a number not set for the specified tools is given as a T-code command, an error message, “784 SPECIFIED TOOL REGISTRATION ERROR,” appears. This method does not have the SPARE TOOL screen available.

3-4 TOOL LIFE Screen

TOOL (LIFE+SPARE)

POT	ORDER TOOL	NAME	LIFE USE	LIFE SET	LIFE UNIT	STATUS	USE GRAPH 50% 100%
001	12000-00	CHAMFER	100	120	min	USE	
002	22000-00	METAL SAW	20	50	leng.		
003	32000-00	FACE MILL	0	200	cnt		
004	04-00	DRILL	80	150	hole		
005	00-00		0	0	min		
006	06-00		200	180	min	LIFE OVER	
007	07-00		0	20	min		
008	800-00		10	50	min		
009	900-00		20	90	min		
010	1100-00		200	200	min	LIFE OVER	
011	1100-01		56	0	min		
012	1100-02		0	0	min		

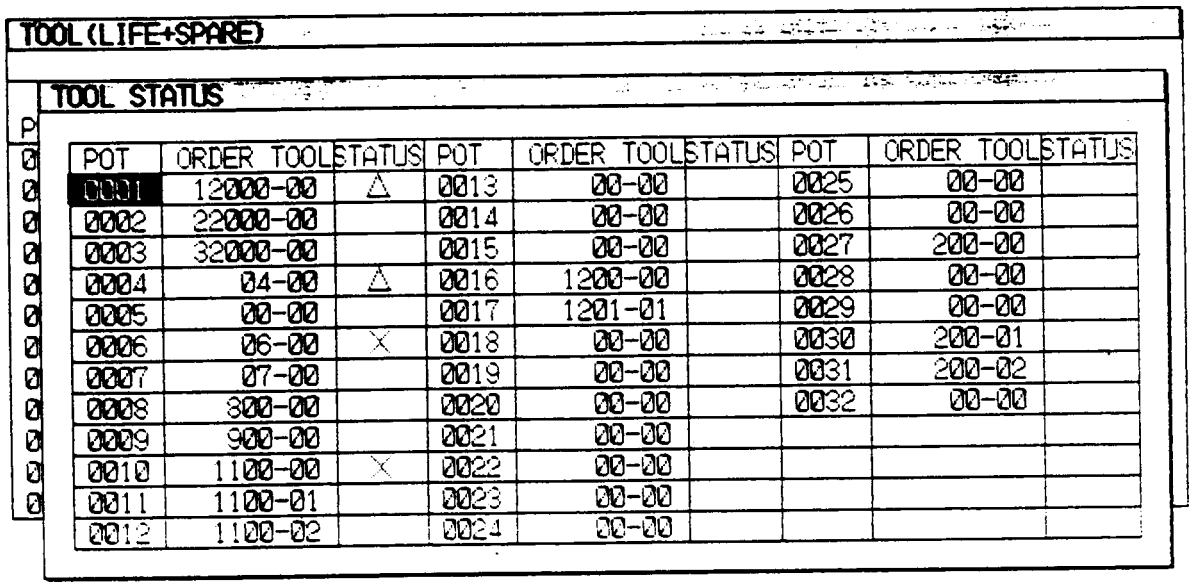
Fig. 11

- Setting the specified tool data
  Move the cursor to a setting position, input a tool number by functions, and press INPUT to set. An error message, “ALREADY REGISTERED,” appears in case of the tool number for which the data has been already registered.
- Displaying the tool data in the pot order/tool order
  You can use F3/POT/TOOL ORDER in the TOOL LIFE screen to choose to display the tool data in the pot order or tool order.

  Display in the pot order ..... Displays the tool data in the pot number order in the TOOL LIFE screen.

  Display in the tool order ..... Displays the tool data in order of numbers by functions in the TOOL LIFE screen. For the tools with no number by functions set for them, the tool data are displayed in the pot order after tool order. The reference numbers have nothing to do with the display order.

3-5 TOOL STATUS Screen



POT	ORDER TOOL	STATUS	POT	ORDER TOOL	STATUS	POT	ORDER TOOL	STATUS
0001	12000-00	△	0013	00-00		0025	00-00	
0002	22000-00		0014	00-00		0026	00-00	
0003	32000-00		0015	00-00		0027	200-00	
0004	04-00	△	0016	1200-00		0028	00-00	
0005	00-00		0017	1201-01		0029	00-00	
0006	06-00	×	0018	00-00		0030	200-01	
0007	07-00		0019	00-00		0031	200-02	
0008	800-00		0020	00-00		0032	00-00	
0009	900-00		0021	00-00				
0010	1100-00	×	0022	00-00				
0011	1100-01		0023	00-00				
0012	1100-02		0024	00-00				

Fig. 12

- Setting the specified tool data
  Move the cursor to a setting position, input a tool number by functions, and press INPUT to set. An error message, “ALREADY REGISTERED,” appears in case of the tool number for which the data has been already registered.

IV. Staf

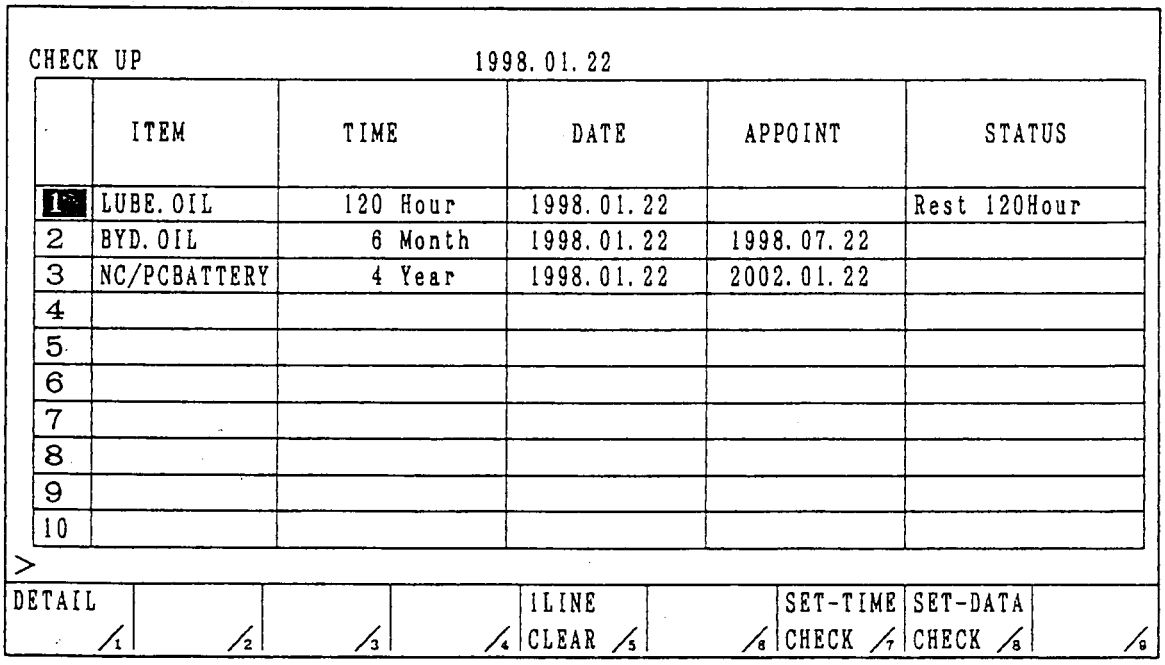
The production support system, STAF (SEIKI TECHNOLOGY ASSISTANCE FUNCTION), has the following screens:

1. Periodic Check
2. Status Display/OK Monitor
3. Instruction Manual Information
4. LSSOL Diagram

1. Periodic Check

The Period Check screen is intended for maintenance of various oils, battery, and so on. This screen is displayed by pressing OPRE/MAINTE and F2/CHECK UP in the Overall screen.
Note: The Periodic Check may not be available for some models.

1-1 Periodic Check - Main Screen



(1) Description of the items

- Item: Check item. No. 1 Item is only for the lubricant.
  Note : The details of items and the number of items differ depending on the model. Some models may not have the Lubricant item.
- Check Time : Check time interval. The unit is either year, month, day, or hour.
- Check Date : Actual date when you checked
- Next Check Date : Date when you want to check next time
- Status : Displays the remaining time only for the first Lubricant item. If Next Check Date has passed in each item, “Change” will be displayed.
- Reverse video Cursor : Indicates the item to view a data change or the Details screen.

(2) Functions in the main screen

- Reverse video Cursor ..... Move with the ↑ or ↓ key to select an item. Move the cursor before pressing any function key.
- F1/DETAIL ..... Displays the Details screen for the cursor-indicated item, if any. If there is no Details screen, “No Screen” will be displayed.
- F5/LINE CLEAR ..... Clears the cursor-indicated items, Check Time to Status. You are prompted, “Do You Want to Clear ?” Press Y to clear.
- F7/SET-TIME CHECK ..... Enters the time setting mode. Pressing again exits the mode.

[Time Setting Mode]

Move the reverse video cursor to an input item. Set the unit of time symbol (Y, M, D, or H), followed by an up to 3-digit number, and press the INPUT key to set. (Unit of time symbols: Y = Year, M = Month, D = Day, H = Hour)

<Example> 500 hours = H500
4 years = Y4
10 months = M10
70 days = D70

During the time setting mode, you can move the reverse video cursor to set as many times as you want.

After exiting the mode, be sure to press next F8/SET-DATE CHECK. Otherwise, time calculation will not be initiated.

Note: The first item is “Lubricant” and only requires you to set the unit of time.

- F8/SET-DATE CHECK ..... If you move the reverse video cursor to an actually checked item and press F8/SET-DATE CHECK, set the today’s data in the Check Date column and set the desired next check date in the Next Check Date column. Simultaneously, “Change” in the Status column will disappear.

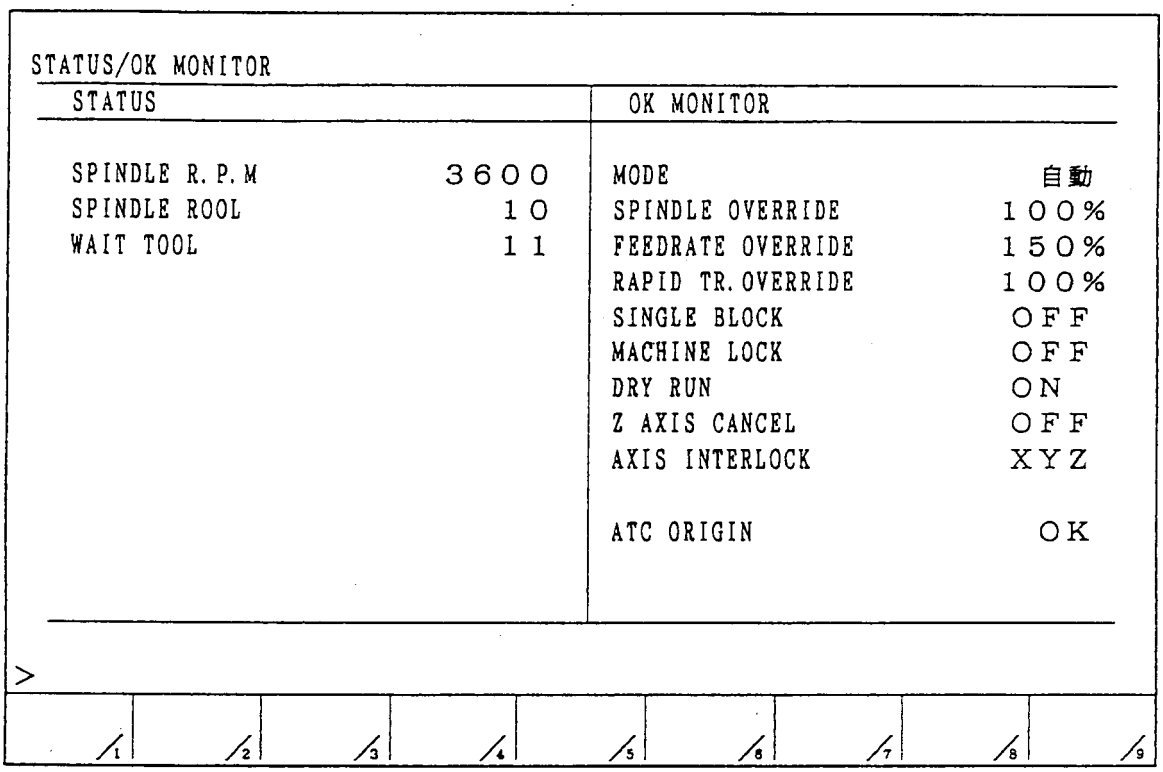
Note: Since the first item is “Lubricant” and generally requires you to set the unit of time, the Next Check Date data will not be displayed.

2. Status Display/OK Monitor

The Status Display/OK Monitor screen displays the machine status. It is displayed by pressing the “STATUS/OK” button on the operation panel.

Alternatively, it is displayed by selecting “8. STATUS/OK MONITOR” from the System Menu displayed by pressing OPRE/MAINTE and F4/SYSTEM.

Note: The displayed items vary from one model to another.



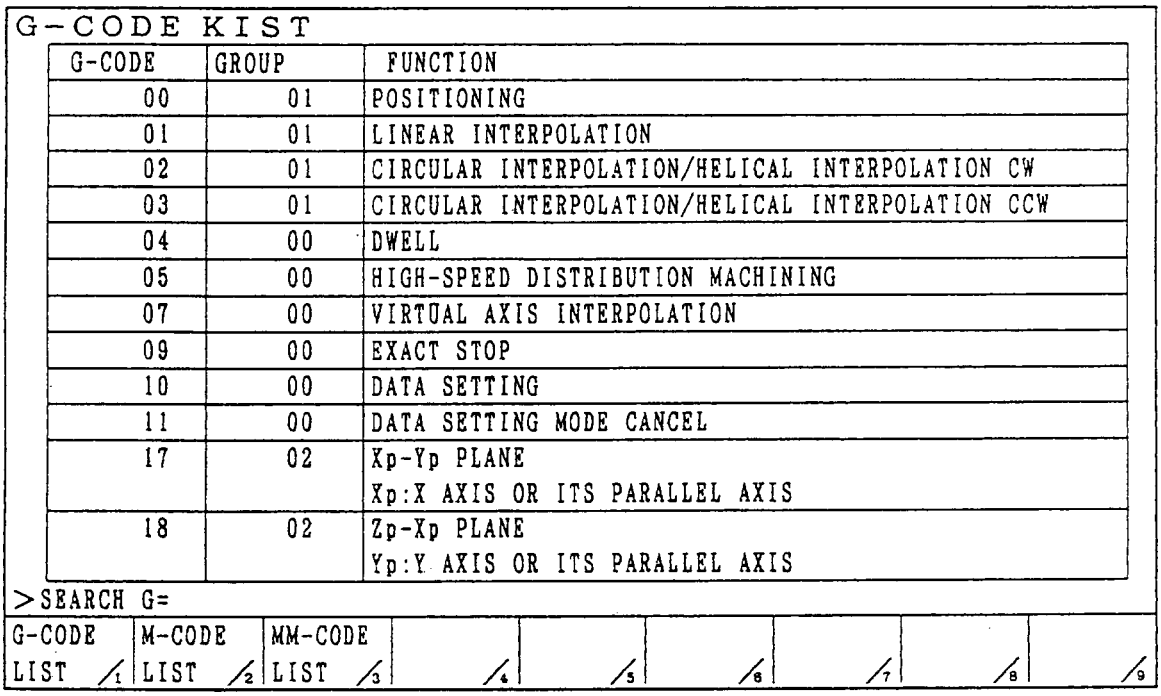
Example of Status Display Screen

3. Instruction Manual Information

This screen displays the G-codes, M-codes, and maintenance M-codes provided for the machine as standard. The optional specifications are not always displayed.

Note: The displayed details vary from one model to another. Some models may not have the maintenance M-codes.
All G codes including, also, optional ones are displayed.
Depending on machine type, however, some are inexecutable.

Select [OPRE/MAINTE] and [F5/INFORMATION] in the Overall screen to display the first page of the G-codes List.


G-CODE KIST

G-CODE	GROUP	FUNCTION
00	01	POSITIONING
01	01	LINEAR INTERPOLATION
02	01	CIRCULAR INTERPOLATION/HELICAL INTERPOLATION CW
03	01	CIRCULAR INTERPOLATION/HELICAL INTERPOLATION CCW
04	00	DWELL
05	00	HIGH-SPEED DISTRIBUTION MACHINING
07	00	VIRTUAL AXIS INTERPOLATION
09	00	EXACT STOP
10	00	DATA SETTING
11	00	DATA SETTING MODE CANCEL
17	02	Xp-Yp PLANE Xp:X AXIS OR ITS PARALLEL AXIS
18	02	Zp-Xp PLANE Yp:Y AXIS OR ITS PARALLEL AXIS

>SEARCH G=

G-CODE LIST /1 | M-CODE LIST /2 | MM-CODE LIST /3 | /4 | /5 | /6 | /7 | /8 | /9

Example of Instruction Manual Information Screen

Screen Common Operation

- [F1/G-CODE LIST].....Lists the G-codes.
- [F2/M-CODE LIST].....Lists the M-codes.
- [F3/MM-CODE LIST].....Lists the maintenance M-codes.
  Note: Some models may not have the list.
- [RETURN].....Returns you to the Overall screen.
- Page keys, , ...Scrolls up/down by one screen.
- [↑], [↓] keys.....Scrolls up/down by one line.
- To search for a code, use a number and the [↓] key.

<Example> Searching for the G-code 28

Press [2] and [8] •G = 28 is displayed at the lower left of the screen. In this state, pressing [↓] starts a search and G28 will be at the top of the G-codes list.

4. LSSOL Diagram (Switch Status)

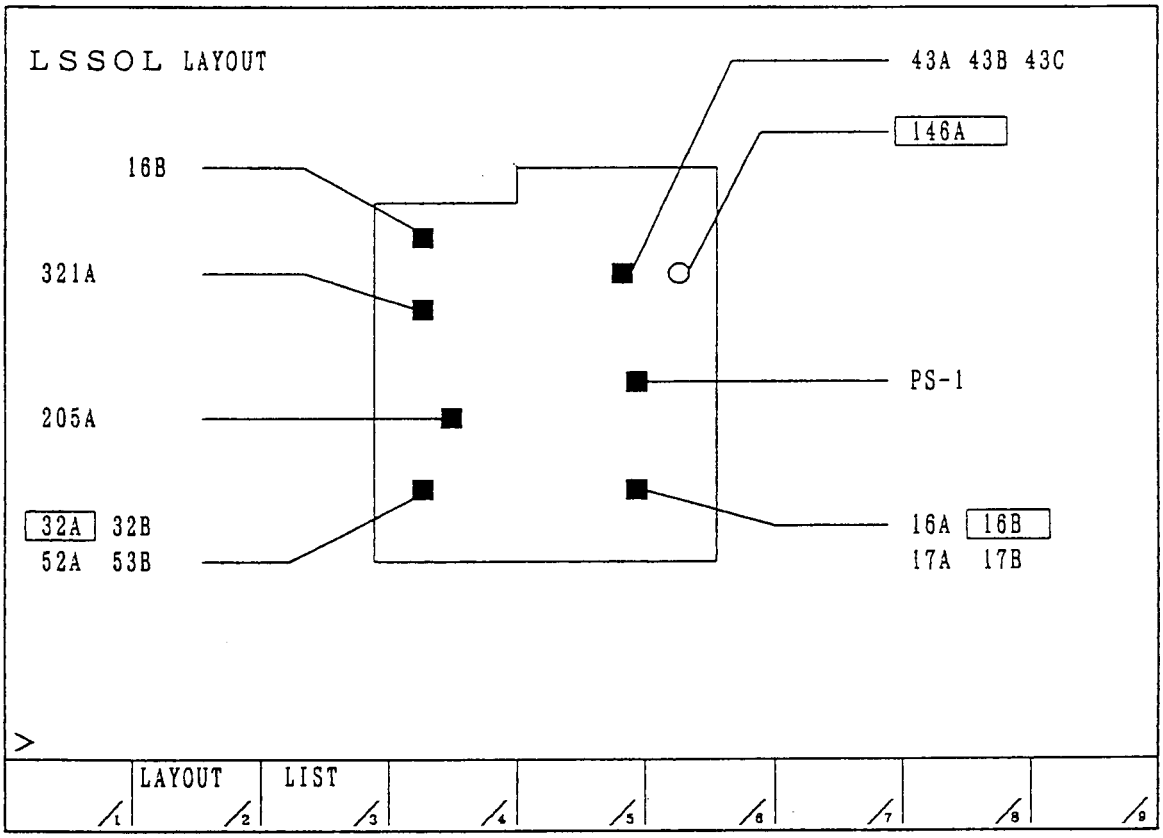
The LSSOL diagram shows a layout of the limit switches and solenoids, and their ON/OFF status. It has a layout drawing and list.

In the Overall screen, select OPRE/MAINTE, F3/ALARM DIAG., and F2/SWITCH STATUS to display the layout drawing first.

Note: If the LSSOL diagram is not provided, this key menu is not displayed.

4-1 Layout Drawing

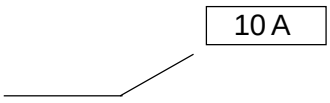
Note: The layout drawing and form vary from one model to another.



Example of LSSOL Layout Drawing Screen

If turned on, the limit switch or solenoid name is enclosed by a rectangle.

<Example>



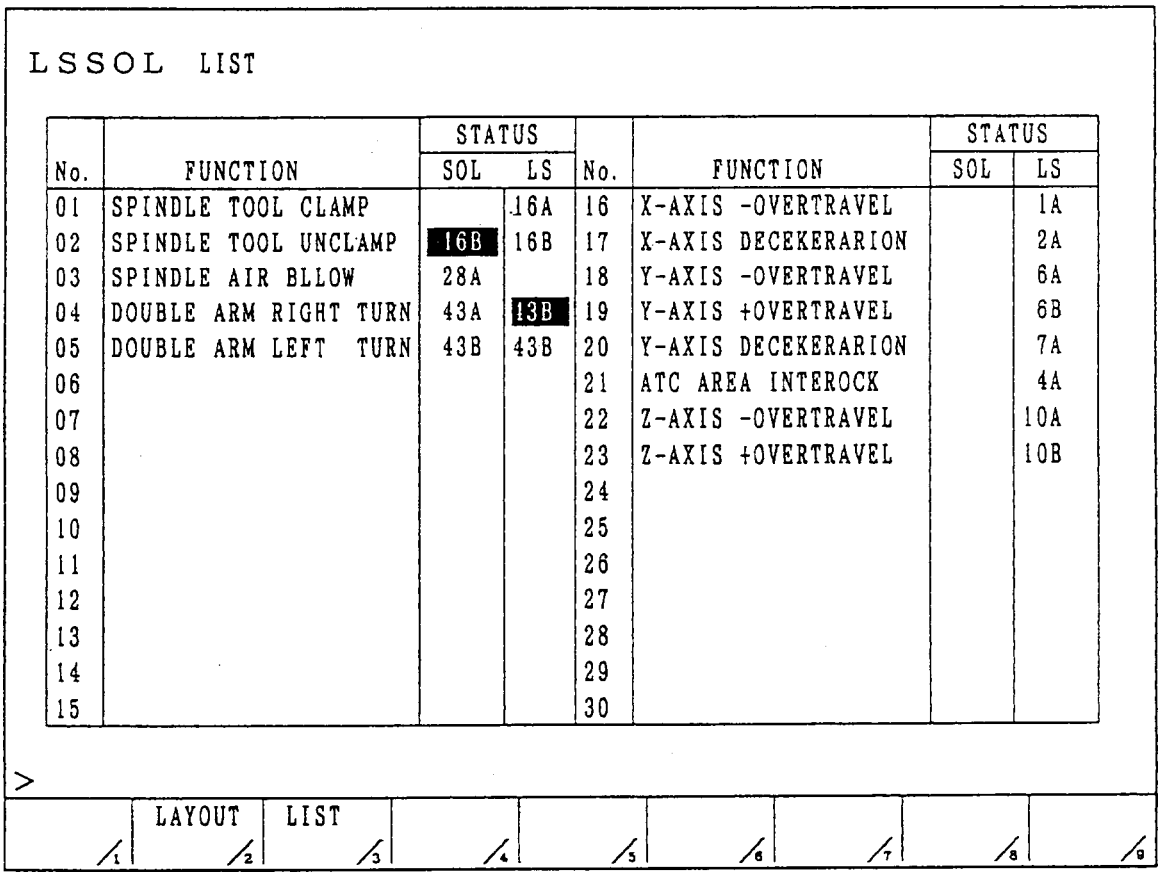
- F3/LIST..... Displays a list.
- RETURN..... Returns you to the Overall screen.

4-2 List

Pressing F3/LIST displays a list which describes the functions/applications.

Note: The details and form of the list vary from one model to another.

If the limit switch or solenoid is turned on, their names will be displayed in reverse video. If there are two or more pages, use the page keys (and) to change them.



LSSOL LIST

No.	FUNCTION	STATUS SOL	STATUS LS	No.	FUNCTION	STATUS SOL	STATUS LS
01	SPINDLE TOOL CLAMP		16A	16	X-AXIS -OVERTRAVEL		1A
02	SPINDLE TOOL UNCLAMP	16B	16B	17	X-AXIS DECEKERARION		2A
03	SPINDLE AIR BLLOW	28A		18	Y-AXIS -OVERTRAVEL		6A
04	DOUBLE ARM RIGHT TURN	43A	43B	19	Y-AXIS +OVERTRAVEL		6B
05	DOUBLE ARM LEFT TURN	43B	43B	20	Y-AXIS DECEKERARION		7A
06				21	ATC AREA INTEROCK		4A
07				22	Z-AXIS -OVERTRAVEL		10A
08				23	Z-AXIS +OVERTRAVEL		10B
09				24			
10				25			
11				26			
12				27			
13				28			
14				29			
15				30			

>

Example of List Screen

- F2/LAYOUT ..... Displays the layout drawing.
- RETURN ..... Returns you to the Overall screen.

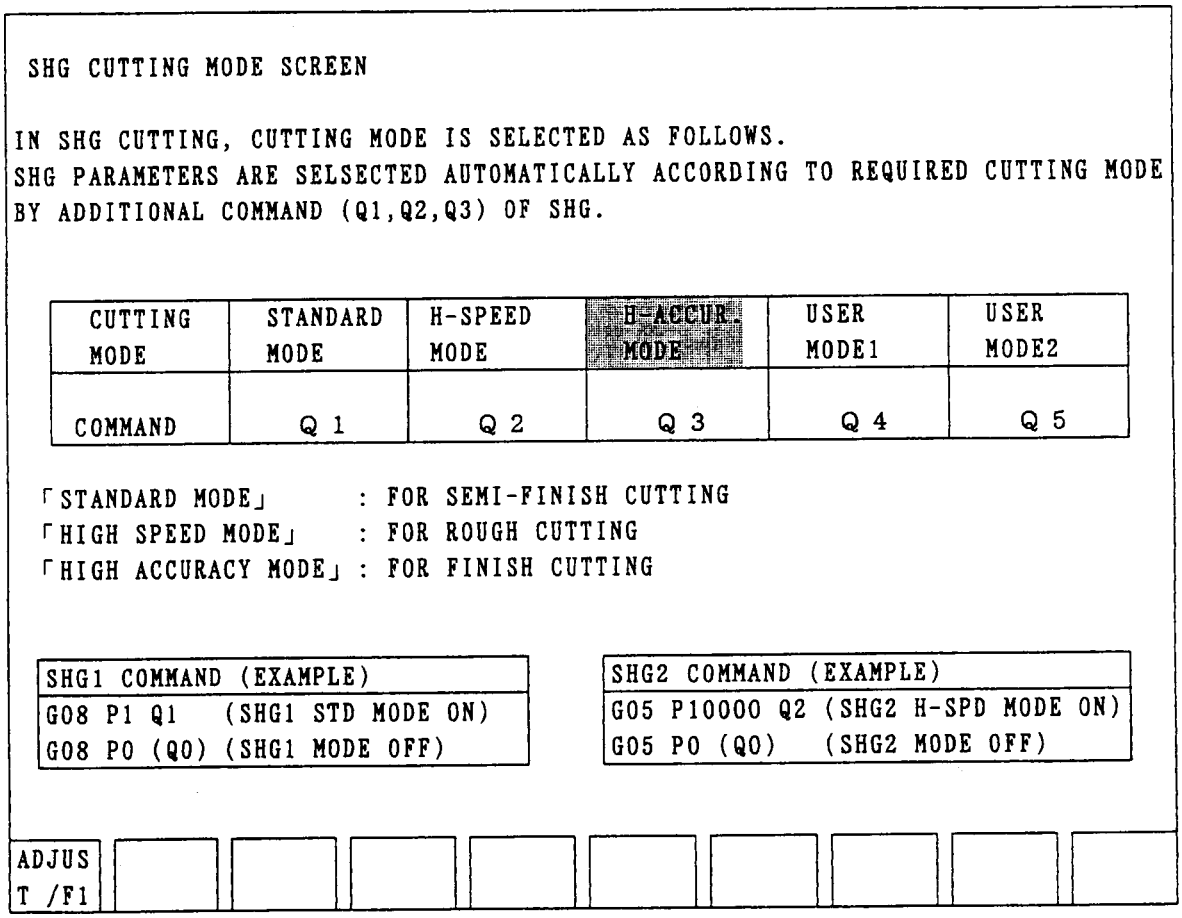
V. SHG

Key operation to get to SHG screen:
Overall screen → front/back → F4/System → 20.SHG

1. Outline

SHG Cutting Mode screen displays cutting mode selected in SHG cutting as well as SHG commanding method. Adjust screen displays and sets parameters to be used in SHG.

2. SHG Cutting Mode



- SHG Cutting Mode screen displays in reverse the mode currently selected.
- Commanding method examples are displayed when each of SHG1 and SHG2 options is held effective.
- In order to adjust parameters in each mode, push F1/ADJUST and change over into SHG Adjust screen.

3-1 SHG Adjust screen Σ10M, Σ16M

SHG ADJUST SCREEN

CONTENTS	Q 1	Q 2	Q 3	Q 4	Q 5
TIME CNST FOR ACC/DEC AFTER INTP (ms)	0	0	0	0	0
MAX FEEDRATE FOR ARC RADIUS R(mm/min)	0	0	0	0	0
STANDARD RADIUS R= ******* mm	0	0	0	0	0
MIN FEEDRATE FOR ARC RADIUS (mm/min)	0	0	0	0	0
SHG1:TIME CNST ACC/DEC BEFR INTP (ms)	0	0	0	0	0
STD CUT FEEDRATE F*****	0	0	0	0	0
SHG1:ALLOW SPD DIF FOR CORNER(mm/min)	0	0	0	0	0
SHG2:TIME CNST ACC/DEC BEFR INTP (ms)	0	0	0	0	0
STD CUT FEEDRATE F*****	0	0	0	0	0
SHG2:ALLOW SPD DIF FOR CORNER(mm/min)	0	0	0	0	0
SHG2:TIME CNST FOR ALLOWABLE ACC (ms)	0	0	0	0	0
STD CUT FEEDRATE F*****	0	0	0	0	0

>_

CUTT MODE

Use this to set constants for SHG Cutting Mode (Q1 to Q5).
The same conditions that were used for parameter writing, also, apply here for setting data.
Time constant acceleration/deceleration after interpolation
..... Time constant for linear acceleration/deceleration after interpolation in cutting feed
Maximum feed rate for arc radius R
..... Feed rate upper limit value required for passing standard radius arc block.
Minimum clamp feed rate for arc
..... Feed rate clamp lower limit value required for passing small radius arc block
SHG1: Pre-interpolation acceleration/deceleration time constant
..... Acceleration time to standard cutting feed rate to determine acceleration rate for pre-interpolation acceleration/deceleration
SHG1: Allowable speed difference for corner
..... Allowable speed difference for each axis for auto corner deceleration
SHG2: Acceleration/deceleration time constant for pre-interpolation
..... Acceleration time to standard cutting feed rate to determine acceleration rate for pre-read pre-interpolation
SHG2: Allowable speed difference for corner
..... Allowable speed difference to determine speed rate based on corner speed difference
SHG2: Time constant for allowable acceleration
..... Acceleration time to standard cutting feed speed to determine allowable acceleration
To return to SHG Cutting mode screen, push F1/CUTTING MODE.
(Note) For changing of a time constant, refer to “SHG Cutting” in “Program.”

3-2 SHG Adjust Screen Σ18M

SHG ADJUST SCREEN

CONTENTS	Q 1	Q 2	Q 3	Q 4	Q 5
TIME CNST FOR ACC/DEC AFTER INTP (ms)	0	0	0	0	0
MAX FEEDRATE FOR ARC RADIUS R(mm/min)	0	0	0	0	0
STANDARD RADIUS R= ******* mm	0	0	0	0	0
MIN FEEDRATE FOR ARC RADIUS (mm/min)	0	0	0	0	0
TIME CNST ACC/DEC BEFR INTP (ms)	0	0	0	0	0
STD CUT FEEDRATE F*****	0	0	0	0	0
ALLOW SPD DIF FOR CORNER(mm/min)	0	0	0	0	0
SHG2:TIME CNST ACC/DEC BEFR INTP (ms)	0	0	0	0	0
STD CUT FEEDRATE F*****	0	0	0	0	0

>_

CUTT MODE

Use this to set constants for SHG Cutting mode (Q1 to Q5).
The same conditions that were used for parameter writing, also, apply here for setting data.
Time constant acceleration/deceleration after interpolation
..... Time constant for linear acceleration/deceleration after interpolation in cutting feed
Maximum feed rate for arc radius R
..... Feed rate upper limit value required for passing standard radius arc block.
Minimum clamp feed rate for arc
..... Feed rate clamp lower limit value required for passing small radius arc block
SHG1: Pre-interpolation acceleration/deceleration time constant
..... Acceleration time to standard cutting feed rate to determine acceleration rate for pre-interpolation acceleration/deceleration
SHG1: Allowable speed difference for corner
..... Allowable speed difference for each axis for auto corner deceleration
SHG2: Time constant for allowable acceleration
..... Acceleration time to standard cutting feed speed to determine allowable acceleration
In order to return to SHG Cutting mode, push F1/CUTTING MODE.
(Note) Any condition which does not meet Machine properties
(Example: 0 as the constant for acceleration/deceleration after interpolation), if being set, can impede machine operation, creating a intense shock, etc.

VI. Thermal Displacement Offset Function

THERMAL DISP.(MAIN)

AXIS	COEF	VAL [um]
Z	10.0	10
X	-0.5	-100

SENSOR DATA

No.	NAME	CUR.TMP. [℃]	BAS.TMP. [℃]
1	ROOM	12.3	12.2
2	BASE	11.1	11.5
3	COLUMN	9.9	10.0
4	SADDLE	22.5	20.0
5	CROSS	0.0	0.0
6	RAM	0.0	0.0

エラー内容

Z Comp. Too Much
X Comp. Too Much
No.1 Wire Broken
No.2 Wire Broken
No.3 Wire Broken
No.4 Wire Broken
No.5 Wire Broken
No.6 Wire Broken

> _

MAINT. DISP. /1	COEF. CHANGE/2	TEMP INIT /3	ALL CLEAR/4	MAINT.2 DISP /5	BASE METHOD/6			

THERMAL DISP.(COEF.)

AXIS	COEF.	VAL [um]
Z	10.0	10
X	-0.5	-100

COEF	Z	X
a	10.0	-0.5
b	99.9	1.0
c	-99.9	2.0
d	0.0	3.0
e	0.0	4.0
f	0.0	5.0
g	0.0	6.0
h	0.0	7.0
i	1.0	8.0
SENS.	1	4

SENSOR DATA

No.	NAME	CUR.TMP. [℃]	BAS.TMP. [℃]
1	ROOM	12.3	12.2
2	BASE	11.1	11.5
3	COLUMN	9.9	10.0
4	CROSS	22.5	20.0
5	SADDLE	0.0	0.0
6	RAM	0.0	0.0

WAIT	INTERVAL	MINI.COMP	AXIS	SENSOR
30	10	0	2	6

> _

/1	/2	/3	MAIN /4	/5				

THEMAL DISP. (MAINT.)

	Z	X	1	2	3	4	5	6
0:00	0	0	20.0	30.0	40.0	10.0	10.0	10.0
0:15								
0:30								
0:45								
1:00								
1:15								
1:30								
1:45								
2:00								
2:15								
2:30								
2:45								
3:00								
3:15								
3:30								
3:45								
4:00								
4:15								

MAIN /1								

1. Thermal Displacement Offset Function (Main)

The Main screen displays the following data.

Table 1 (Offset Data)

Axis Name	:	Axis to be offset
Offset Factor	:	Magnification to correct an offset amount
Offset Amount	:	Offset amount to be output to the NC unit

Table 2 (Sensor Data)

Channel	:	Sensor channel number
Name	:	Name of a measuring point
Current Temperature	:	Temperature received from the sensor
Reference Temperature	:	Reference temperature (Not displayed if initialized at power-on)

Table 3 (Error Messages)

Displays error messages.

Table 4 (Internal Offset Factors and Threshold Values)

Displays only at the time of all clear

The Main screen has 5 function menus which have the following functions, respectively.

F1 (MAINTENANCE SCREEN)	Views a temperature transition table for the past 24 hours.
F2 (OFFSET FACTOR CHANGE)	Rewrites an offset factor.
F3 (REF. TEMP. INITIALIZE)	Adopts a current temperature as a reference one to clear an offset amount to 0.
F4 (ALL CLEAR)	Changes an internal offset factor, threshold value, reference temperature, and sensor name.
F5 (MAINTENANCE SCREEN 2)	For maintenance
F6 (REF. TEMP. SYSTEM)	Changes a reference temperature introduction system.

1-1 Offset Factor

The offset amount is calculated out of a temperature change of each sensor, but an error may occur depending on the user environment and machine operating condition. To avoid this, multiply by a factor to correct the offset amount, as required. The factor can be obtained by the following formula.

$$\text{Offset factor} = \frac{\text{Daily offset amount changed - Error}}{\text{Daily offset amount changed}}$$

Where; the error is defined as follows:

Error = Finish work dimension - Target work machining dimension

When there is no error, therefore, the factor is 1.0, and when the finish dimension is small, it is larger than 1.0.

The offset factor is rewritten in the following procedure.

① Press F2 (OFFSET FACTOR CHANGE).

② Use the cursor key to select the axis you want to change.

③ Input the data.

The offset factor can be set within a range of +/-99.9. If it is rewritten, the offset amount will be recalculated immediately against the previous one, not from the current temperature.

1-2 Initializing the Reference Temperature

When you have selected the system which does not initialize the reference temperature at power-on, you must manually initialize it (if you have selected the system which takes the temperature at power on as the reference temperature, this operation will change the system.) To change the system, see Section 1-4. The procedure is as follows:

① Press F3 (REF. TEMP. INITIALIZE).
"OK ? Y-YES N-NO" is displayed.

② Press the Y key.
Pressing any other key is equivalent to pressing the N key (does not initialize).

1-3 All Clear

Using the all clear function, set an internal temperature factor and sensor name.

① Press F4 (ALL CLEAR).
"INPUT PASSWORD" is displayed.

② Input a password (CLEAR).
A confirmation message is displayed together with a table of internal offset factors.

③ Using the cursor key, move the cursor to where you want to input the data.
The up, down, left, and right keys are available.

④ Input the data.

Once the password is confirmed, step②is omitted thereafter.

1-3-1 Sensor Name

You can select the sensor name by a number out of the following:
0. Delete name 1. Room temperature 2. Column 3. Cross 4. Saddle 5. Ram 6. Base 7. Nose 8. Cover 9. Head 10. Table 11. Bed 12. Spindle 13. Turret 14. Quill
Whichever name is selected, calculations are not affected. It is used as a guideline.

1-3-2 Internal Offset Factor

There are 9 factors, a through i, for each axis; a is the same as the offset factor in Table 1. All the data can be set within a range of +/-99.9.
For "Sensor Number," specify the number of sensor used to measure a surface temperature of the machine. This temperature data is used to calculate an offset by a delay response.
Effective numbers are 1 to the maximum sensor number. A number 0 is considered no. 1.

1-3-3 Reference Offset Amount

This is the least offset amount change value. When the calculated offset amount does not exceed this value, the previous offset amount is used.

1-3-4 Measurement Interval

Set in seconds the calculated thermal displacement offset interval. If it is 10 seconds or less, it will be calculated as 10 seconds.

1-3-5 Waiting Time

Enter in seconds a waiting time between power-on and when the thermal displacement offset function is initiated. This is used when a sensor stabilization time is required in a place where a room temperature is unstable.

1-3-6 Reference Temperature

The reference temperature currently used for offset calculation is displayed.

1-3-7 Controlled Axes

Select the axis (or axes) you want to offset out of the following and set with a relevant number. After inputting this data, turn on the power again.
"0: Z-axis only, 1: Z- and X-axis, 2: Z- and Y-axis"

1-3-8 Number of Sensors

Set the number of sensors. After inputting this data, turn on the power again.

1-4 Reference Temperature System

Thermal displacement offset detects a temperature change and calculates the offset amount, taking the temperature at power-on as reference.
Frequent power on/off may cause inconvenience because the offset is cleared to zero (0). For this reason, the system can be switched as to at which point the reference temperature should be updated.

System-1: Takes in the current temperature as the reference temperature at power-on.
System-2: Sets the reference temperature only by initializing the reference temperature.

With System-1, the reference temperature in the Main screen is displayed blank ("——"); it is displayed only when System-2 is selected.
The system can be switched as follows.

① Press F6 (REF. TEMP. SYSTEM).
"OK ? Y-YES N-NO" is displayed.

② Press the Y key.
Pressing any other key is equivalent to pressing the N key (does not change the system).

2. Thermal Displacement Offset Function (Maintenance)

Displays the temperature data and offset of each temperature sensor every 15 minutes up to 24 hours. The page keys and cursor up and down keys are available.

3. Thermal Displacement Offset Function (Maintenance 2)

Displays the interim calculation data. This is used only for adjustment of the machine.

VII. UUP Function

1. General Description

The internal information of the CNC unit can be read/written through the “UUP (Universal User Port)” interface connector of the machine operation panel.
Our “HS-MOS (Hitachi Seiki - Machine Operating System)” is the software for personal computers (to be referred to as PCs hereinafter), which allows you to read/write the internal information of the CNC unit from the PC. The HS-MOS is the API (Application Program Interface) function provided in the Microsoft’s DDL (Dynamic Linking Library) format.
The user applications such as Visual BASIC can easily read/write the internal data of the CNC unit from the PC side by calling this API function.
(Note) Microsoft, Windows, and Visual BASIC are the registered trademarks of U.S. Microsoft Corporation.
(Note) For the details of the API, etc., see a separate instruction manual.

1-1 Applicable CNC Models

CNC units of the SEICOS Σ 16/18/21 series, which has the UUP port on the machine operation panel

2. Connection Of UUP

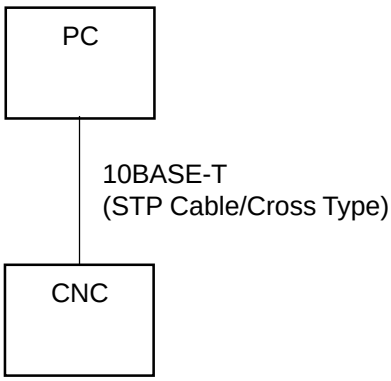
Connect the CNC unit to the PC via the “UUP (Universal User Port)” interface connector of the machine operation panel.

2-1 UUP Specifications

(1) Interface configuration: Ethernet LAN
Connector shape: 10BASE-T
Communication speed: 10 Mbps
(2) Communication system TCP/IP (Transmission Control Protocol/Internet Protocol)
Set up the TCP/IP driver for the PC system used in advance. If you do not know how to set up, inquire the network administrator.

2-2 Connecting between the PC and CNC Unit

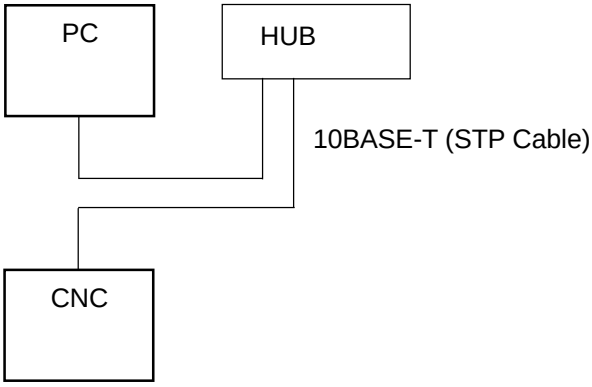
When the distance between the PC and CNC unit is short, they can be connected one to one without interposing the hub. If the communication conditions are not good, connect via the hub.
The connection cable used should be an overall shielded Category 5 twisted pair cable (STP cable).
(1) Connecting one PC to one CNC unit (Not via the hub/carry-about type)
When the distance is short (up to about 3 m), they can be connected without interposing the hub between them. If this is the case, be sure to use a cross type cable. If the communication conditions are not good, connect via the hub.



[One-to-one Connection]

(2) Connecting one PC to one CNC unit (Via the hub)

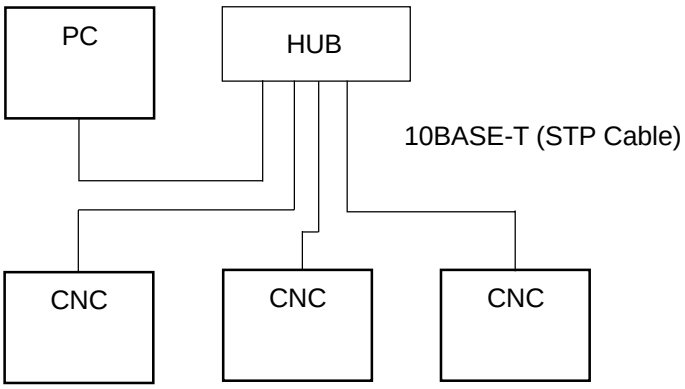
In one-to-one connection, when the distance is long or you want to enhance the communication conditions, interpose the hub between them. In this case, the cable used between the hub and CNC unit is a straight type.



[One-to-one Connection via Hub]

(3) Connecting one PC to multiple CNC units

When connecting one PC to multiple CNC units, be sure to interpose the hub. If this is the case, all the cables between the hub and CNC units are a straight type. You can connect from the PC to the multiple CNC units simultaneously.



[Connection of Multiple Units]

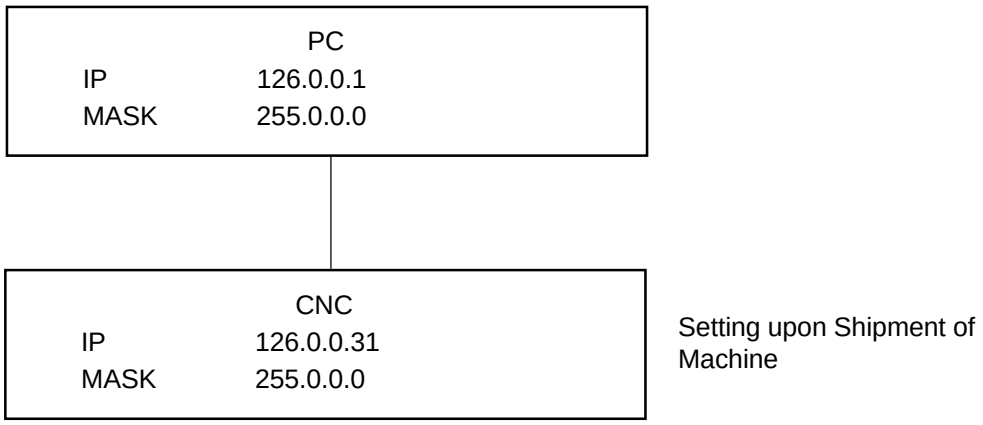
2-3 Setting the TCP/IP Addresses

Upon shipment of the CNC unit, the TCP/IP addresses are as follows:

Upon shipment of CNC Unit

IP address	126.0.0.31
Mask address	255.0.0.0
Port address (TCP/IP)	8193

(1) When connecting one PC to one CNC unit (When not connecting to the in-house LAN)
When connecting one PC to one CNC unit and neither of them is to be connected to the in-house LAN, set the IP address of the PC side as follows. If they may be connected to the in-house LAN, see (3) below.

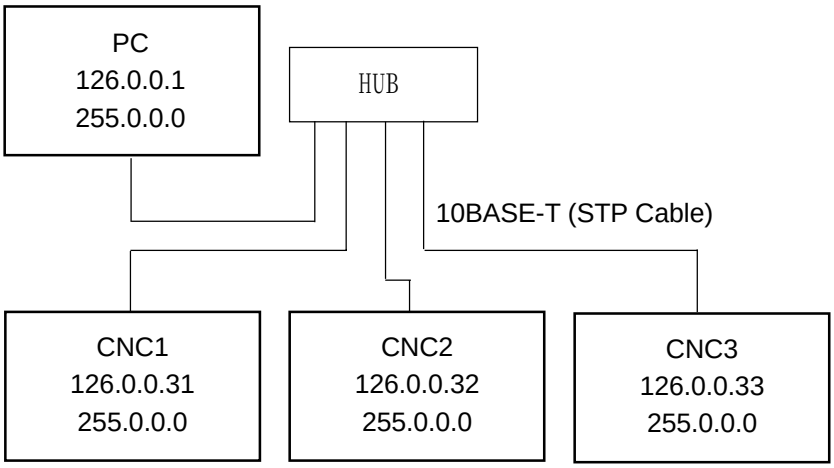


[One-to-one Connection]

(2) Connecting one PC to multiple CNC units (New network)

When connecting one PC to multiple CNC units and configuring a new network without connecting to the in-house LAN, take care not to duplicate the IP address of the 2nd CNC unit onward. For alteration of the IP address on the CNC side, see 2-4.

Range: 126.0.0.31 to 126.0.0.254



[Connection of Multiple Units]

(3) When incorporating into the in-house LAN

When the CNC unit or PC is to be incorporated into the in-house LAN, the IP addresses on the CNC and PC sides follow the IP address setting for the in-house LAN.

For the respective set IP addresses, inquire the network administrator of the in-house LAN.

For alteration of the IP address on the CNC side, see 2-4, and for that on the PC side, follow the instructions described in the PC operation manual.

Caution: When using for the first time, consult your network administrator to set the IP addresses, etc. carefully and conduct a full test. Note that if the IP addresses, etc. are set erroneously, the entire network may be affected by a communication failure, and so on.

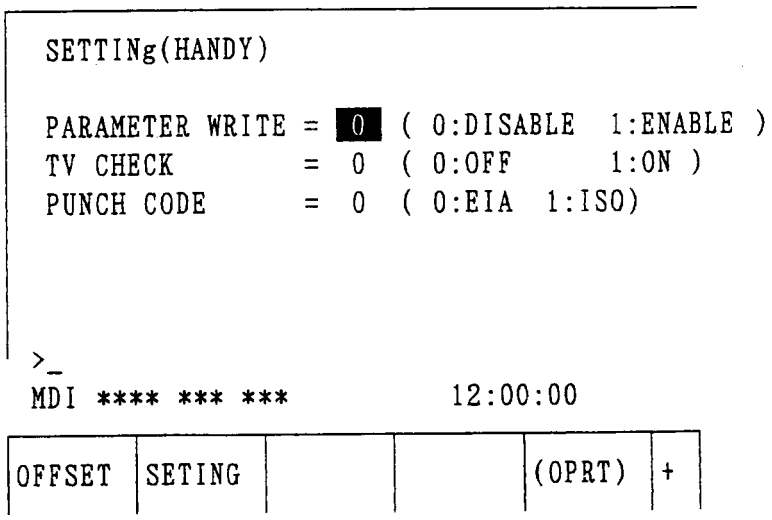
2-4 Setting/Referring to the IP Address on the CNC Side

The IP address is set or referred to in the F Menu screen of the CNC unit.

- When setting the IP address: Since the IP address is part of the system parameters, it is necessary to enable to write the parameters in advance. Once setting is finished, disable to write the parameters.
- When referring to the IP address: Omit the parameter write enabling/disabling procedure.

2-4-1 Enabling to Write the Parameters

(1) To write the parameters, press Ready and select the MDI mode.
(2) Select OPER/MAINT, followed by SYSTEM/F4 to display the System Menu.
(3) Select [30. F MENU].
(4) In F MENU, select [3. F_SETTING] to display the Setting screen.



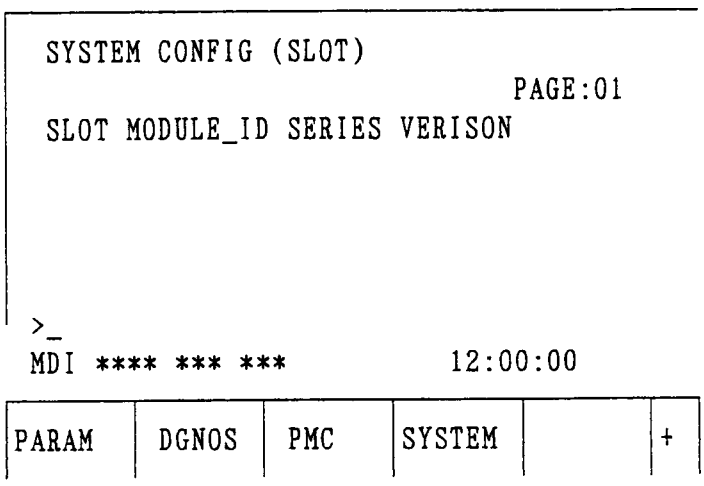
F_SETTING Screen

(5) In this Setting screen, press [1] and [INPUT] to set WRITE PARAMETER = 1. This allows you to write the parameters. Do not tamper with the other items.

(6) Press the [ALTER] key to return to the F MENU screen.

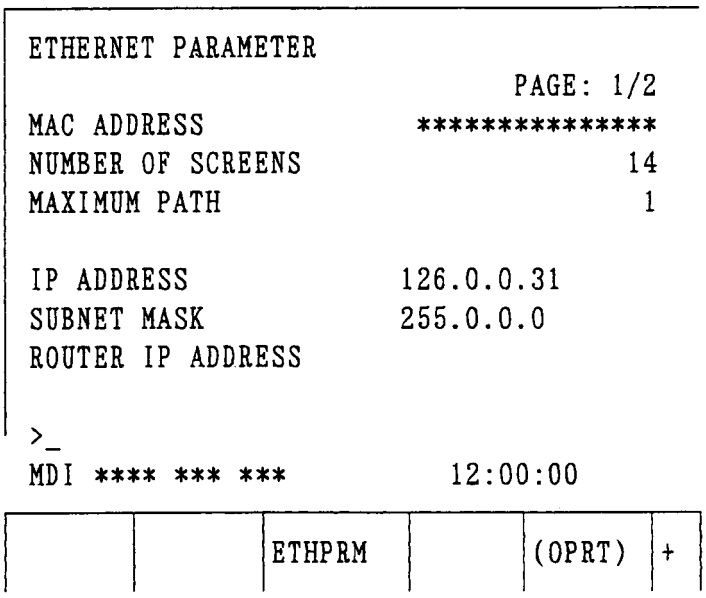
2-4-2 Setting/Referring to the IP Address

(1) Next, select [2. F_SYSTEM] in F MENU to display the F_SYSTEM screen.

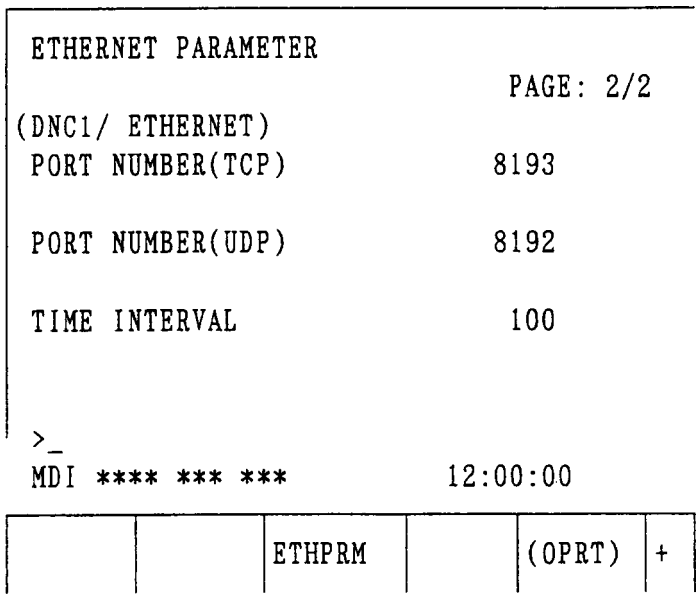


F_SYSTEM screen

(2) Turn several menu pages with the RETURN key to search for “ETHPRM.” Use the ETHPRM key to display the Ethernet parameter screen.



ETHERNET PARAMETER Screen (1st Page)



ETHERNET PARAMETER Screen (2nd Page)

(3) THE ETHERNET PARAMETER Screen has two pages.
You can use the page keys or [↑] and [↓] keys to scroll the screen. Upon shipment, the pages contain the following data.

1st Page	IP ADDRESS	126.0.0.31
	SUBNET MASK	255.0.0.0
2nd Page	PORT NUMBER (TPC)	8193
	PORT NUMBER (UDP)	8192
	TIME INTERVAL	100

(Note: The MAC does not require you to set it; it is automatically set.)

(4) When altering the IP address
- Correct the "IP ADDRESS" and "SUBNET MASK" data on the 1st page.
  Use the numerical keys and period key ([.]).
- Normally, the data on the 2nd page are left intact.

* If you have any question or inquiry about the IP address, inquire the network administrator.

(5) Press the ALTER key to return to the F MENU screen.

2-4-3 Disabling to Write the Parameter

Once setting of the IP address is finished, you need to disable to write the parameters.

(1) Take Steps (1) through (4) in 2-4-1 Enabling to Write the Parameters.
(2) In the Setting screen, press [0] and [INPUT] to set WRITE PARAMETER = 0. This disables you to write the parameters. Do not tamper with the other items.
(3) Press the [ALTER] key to return to the F MENU screen again.
(4) Turn on the NC power again.
Now, you are through with setting the IP address for the UUP.

2-4-4 PING Test

To see whether the TCP/IP address is correct, it is effective to conduct a test with the PING command from an external personal computer. For the Windows version, the PING command can be input in the DOS windows, "Start," "Program," and "DOS Prompt." For the details, inquire the network administrator.

Example) When the IP address of the CNC unit is 126.0.0.31, give PING 126.0.0.31 in the PC's command line. This allows you to determine whether the CNC unit's circuit is properly connected.

(1) When the CNC unit's circuit is properly connected
If properly connected, the time required for response will be displayed in 4 lines or so as shown below. If "Replay ..." is displayed as shown below, it indicates that the circuit is properly connected.

Pinging 126.0.0.31 with 32 bytes of data.
Replay from 126.0.0.31: bytes=32 time=5ms TTL=60
Replay from 126.0.0.31: bytes=32 time=5ms TTL=60
Replay from 126.0.0.31: bytes=32 time=5ms TTL=60
Replay from 126.0.0.31: bytes=32 time=5ms TTL=60
Pinging statistics for 126.0.0.31:
……

(2) When the CNC unit's circuit is improperly connected

If the following message is displayed, it indicates that the circuit is disconnected or the IP address is wrong.

Request timed out.
Request timed out.
Request timed out.
Request timed out.
or
Destinattion host unreachable.
Destinattion host unreachable.
Destinattion host unreachable.
Destinattion host unreachable.

2-5 Connecting Materials

For Ethernet connections, be sure to use overall shielded Category 5 twisted pair cables (STP cables) in order to improve noise resistance in the FA environment. The following table lists the recommended materials.

Name	Type	Manufacturer
10BASE-T STP cable (cross) with modular RS45s at both ends	HSTPTC-2P-*-shield (cross)	HITACHI CABLE
10BASE-T STP cable (straight) with modular RS45s at both ends	HSTPTC-2P-*-shield (straight)	HITACHI CABLE
HUB	HCN7500	HITACHI CABLE
10BASE5 cable with N-connectors at both ends	HBN-CX-100-*	HITACHI CABLE
Transceiver cable	HBN-TC-100-*	HITACHI CABLE
10BASE5 transceiver	HBN-200TZ	HITACHI CABLE

Note: * denotes a dimension in meters.

SEIKI-SEICOS Σ10M/16M/18M

INSTRUCTION MANUAL

OPERATION

Version 1.02

9-2000

First Edition 11-1999