

# SEIKI - SEICOS

## $\Sigma 10/\Sigma 18T/\Sigma 21L$

# INSTRUCTION MANUAL

OPERATION

**40** Edition 1.01 02-2001

NO-0000-1-0211-E-1-02



***Hitachi Seiki Deutschland  
Werkzeugmaschinen GmbH***

# CONTENTS

<b>1. OUTLINE .....</b>	<b>1 - 1</b>
1-1 Items requiring attention when reading this manual. ....	1 - 1
<b>2. SPECIFICATIONS .....</b>	<b>2 - 1</b>
<b>3. OPERATION .....</b>	<b>3 - 2</b>
I. Basic Machine Operation .....	3 - 2
1. Manual Operation .....	3 - 3
1-1 Jog Feed .....	3 - 3
1-2 Manual Reference Point Return .....	3 - 3
1-3 Rapid traverse .....	3 - 4
1-4 Manual Handle Feed .....	3 - 5
2. Automatic Operation .....	3 - 6
2-1 Automatic Mode .....	3 - 6
2-2 Selecting the Run Program .....	3 - 6
2-3 Stopping the Automatic Operation .....	3 - 7
2-4 Dry Run .....	3 - 8
2-5 Single Block .....	3 - 8
2-6 Override .....	3 - 8
2-7 Per-program Machining Time Display Function (Option) .....	3 - 9
2-8 Tape Operation (Simple DNC Operation) .....	3 - 10
2-9 DNC Operation (Optional) .....	3 - 10
2.10 Submemory .....	3 - 14
3. Operation Related to Safety .....	3 - 19
3-1 Emergency stop .....	3 - 19
3-2 Overtravel .....	3 - 19
4. NC Operation keys .....	3 - 20
5. Quick Tool Setter (Q Setter) .....	3 - 22
5-1 Outline .....	3 - 22
5-2 Operation Method .....	3 - 22
5-3 Tool Nose Point Interlock .....	3 - 25
5-4 Related Parameters .....	3 - 25
5-5 Related Alarms .....	3 - 26
5-6 Compensation Input Checking .....	3 - 26
6. Q setter barrier .....	3 - 27
6-1 Outline .....	3 - 27
6-2 Description of the function .....	3 - 27
6-3 Precautions .....	3 - 28
6-4 Related parameters .....	3 - 28

7. Q Setter Repeat Function .....	3 - 29
7-1 Outline .....	3 - 29
7-2 Operation .....	3 - 29
7-3 Working .....	3 - 30
7-4 Precaution .....	3 - 33
7-5 Relevant Parameters .....	3 - 33
7-6 Relevant Alarms .....	3 - 34
8. Simple Jaw Edge Forming Function .....	3 - 34
8-1 Outline .....	3 - 34
8-2 Operation .....	3 - 34
8-3 Function Keys .....	3 - 36
8-4 Jaw Configuration And Machining Conditions .....	3 - 36
8-5 Precaution .....	3 - 37
8-6 Relevant Parameters .....	3 - 37
8-7 Relevant Alarms .....	3 - 37
8-8 Set Values for Standard Soft Jaws .....	3 - 39
8-9 Operation Of Simple Soft Jaw Forming Function .....	3 - 45
II. Screen Operation .....	3 - 49
1. Screen Layout and Basic Operation .....	3 - 50
1-1 Main Display Area .....	3 - 50
1-2 Alarm Display Area .....	3 - 51
1-3 Program Number Display Area .....	3 - 51
1-4 Key Input Area .....	3 - 52
1-5 Error Message .....	3 - 52
1-6 Function Menu .....	3 - 52
1-7 Tree Chart .....	3 - 53
2. Operation of Overall Screen .....	3 - 55
2-1 Program .....	3 - 56
2-2 Tool Path Plot .....	3 - 57
2-3 Tool Data .....	3 - 57
2-4 Spindle Data .....	3 - 57
2-5 Feed Speed .....	3 - 57
2-6 Rotary Tool Speed .....	3 - 57
2-7 Spindle Clamp Speed .....	3 - 57
2-8 Dwell .....	3 - 57
2-9 Load Meter .....	3 - 57
2-10 Positional Data .....	3 - 57
2-11 G Codes .....	3 - 58
2-12 M codes .....	3 - 58
2-13 Time Measurement .....	3 - 58
2-14 Date And Time .....	3 - 59

3. Position .....	3 - 60
3-1 Reloading of Relative Coordinate System .....	3 - 61
3-2 Command Value Display .....	3 - 61
3-3 Restart (Option) .....	3 - 61
3-4 Floating reference .....	3 - 61
3-5 Set up .....	3 - 61
4. Work Coordinates (Offset) .....	3 - 64
4-1 Removal Amount .....	3 - 65
4-2 Machining Reference Point Shift .....	3 - 65
4-3 Work Length .....	3 - 65
4-4 External Work Zero Point Offset .....	3 - 65
4-5 Setting of 2nd Reference Point .....	3 - 66
5. Set (Data) .....	3 - 67
5-1 Display and Setting .....	3 - 67
5-2 Number Search .....	3 - 68
5-3 Eco-eco .....	3 - 71
6. Macro Variable .....	3 - 72
6-1 Macro Variable and Data Display .....	3 - 72
6-2 Number Search .....	3 - 72
7. Plot .....	3 - 74
8. Plotting Parameters .....	3 - 76
9. Tool (offset) .....	3 - 80
9-1 Setting of Offset Value .....	3 - 81
9-2 Tool list .....	3 - 81
9-3 Setting of Type .....	3 - 82
9-4 Setting of Name .....	3 - 83
9-5 Data Clear .....	3 - 83
9-6 Tool life Management .....	3 - 84
9-7 Number Search .....	3 - 84
9-8 Double Cutting (Option) .....	3 - 84
10. Alarm Diagnosis .....	3 - 85
10-1 Alarm History .....	3 - 86
11. System .....	3 - 87
11-1 Parameter Setting .....	3 - 87
11-2 Diagnosis .....	3 - 89
11-3 Mechanical Compensation .....	3 - 91
11-4 Position Region .....	3 - 92
11-5 Software Version .....	3 - 93
11-6 Date and Time .....	3 - 94
11-7 SLBUS .....	3 - 95
11-8 System Table .....	3 - 96
11-9 F Menu .....	3 - 96

12. Program .....	3 -101
12-1 Operating Key Input Area On Screen .....	3 -102
12-2 Displaying Program .....	3 -102
12-3 Creating New Program .....	3 -102
12-4 Deleting Program .....	3 -103
12-5 Initialization Of Program Memory .....	3 -103
12-6 PROGRAM SEARCH .....	3 -103
12-7 Program Copy .....	3 -104
12-8 Changing Program Number .....	3 -104
12-9 Program Editing (Edit Mode or Background Editing) .....	3 -104
12-10 Word Search .....	3 -105
12-11 Background Edit (Program Screen Only) .....	3 -106
12-12 Range Assignment Edit Operation (Program Screen Only) .....	3 -108
12-13 Word Convert (Program Screen Only) .....	3 -110
12-14 Macro Variables .....	3 -111
13. Program List .....	3 -112
13-1 Program List .....	3 -112
13-2 Program Search .....	3 -114
13-3 Program Delete .....	3 -114
13-4 Program Arrangement .....	3 -115
13-5 Program Merging .....	3 -115
13-6 Initialization Of Program Memory .....	3 -115
14. Input / Output .....	3 -116
14-1 Program List .....	3 -117
14-2 Data Setting Table .....	3 -118
14-3 Check .....	3 -120
14-4 Details Of Setting Data .....	3 -120
14-5 Call Select (Optional) .....	3 -122
14-6 Don-Don FD I/O Screen .....	3 -123
15. Input/Output (With PC Card Used) .....	3 -125
15-1 Available PC Cards .....	3 -125
15-2 Initial Formatting of PC Card .....	3 -125
15-3 Starting PC Card Operation .....	3 -125
15-4 Ending PC Card Operation .....	3 -127
15-5 PC Card Input/Output .....	3 -128
15-6 Operating the PC Card Directory .....	3 -131
15-7 Error Messages .....	3 -136

## **4. INDIVIDUAL PROGRAMS ..... 4 - 2**

I. Tool Life Management .....	4 - 2
1. Outline of tool life management .....	4 - 2
2. Operation of tool life management function .....	4 - 2
3. Screen explanations .....	4 - 2
4. Alarm .....	4 - 3

5.	Spare tool .....	4 - 3
6.	Registering and deleting the spare tool .....	4 - 3
7.	Selection order of spare tools .....	4 - 4
8.	Sample Program .....	4 - 4
8-1	Details of Tool Life by Times .....	4 - 5
II.	Cutting Monitor .....	4 - 6
1.	Outline of Cutting Monitor .....	4 - 6
2.	Monitoring Function .....	4 - 6
3.	Description of Monitoring Function .....	4 - 6
4.	Load Settings .....	4 - 8
5.	Monitoring Operation .....	4 - 9
6.	Load Graph .....	4 - 10
7.	Life Graph .....	4 - 11
8.	Alarms .....	4 - 11
9.	Sample Program .....	4 - 12
10.	Parameters .....	4 - 12
10-1	Selection of Monitored Data System (Parameter 8000) .....	4 - 12
10-2	Selection of Initial Setting (Parameter 8001) .....	4 - 12
10-3	Automatic Calculation Parameters (Parameters 8010-8029) .....	4 - 13
10-4	Timer Parameters (8030-8044) .....	4 - 13
10-5	Adaptive Parameters .....	4 - 14
III.	Staf .....	4 - 15
1.	Periodic Check .....	4 - 15
1-1	Periodic Check - Main Screen .....	4 - 15
2.	Status Display/OK Monitor .....	4 - 17
3.	Instruction Manual Information .....	4 - 18
4.	LSSOL diagram (Switch Status) .....	4 - 19
4-1	Layout Drawing .....	4 - 19
4-2	List .....	4 - 20
IV.	UUP Function .....	4 - 21
1.	General Description .....	4 - 21
1-1	Applicable CNC Models .....	4 - 21
2.	Connection of UUP .....	4 - 21
2-1	UUP Specifications .....	4 - 21
2-2	Connecting between the PC and CNC Unit .....	4 - 21
2-3	Setting the TCP/IP addresses .....	4 - 23
2-4	Setting/Referring to the IP Address on the CNC Side .....	4 - 24
2-5	Connecting Materials .....	4 - 28
V.	External Measuring Function .....	4 - 29
1.	Outline .....	4 - 29
2.	Tool Data .....	4 - 30
3.	External Measurement .....	4 - 31
4.	Qualitative Compensation .....	4 - 32
5.	Wear Management .....	4 - 33



# 1. OUTLINE

SEICOS-Σ10L, Σ16T, Σ18T and Σ21L are products that have integrated the latest device technology and realized down-sizing with high reliability.

The machine is designed by giving consideration to users standpoint in the operating system as the man-machine interface, thus offers a most easy-to-operate machine.

This manual contains explanation on the operating method of the following models. As for the subject of programming, refer to “SEIKI-SEICOS Σ10L/16T/18T/21L program part” and for subjects on alarms and maintenance to “SEIKI-SEICOS Σ10/16/18/21 maintenance part” respectively.

Model Name	Abbreviation
SEIKI-SEICOS Σ10L	S-Σ10L
SEIKI-SEICOS Σ16T	S-Σ16T
SEIKI-SEICOS Σ18T	S-Σ18T
SEIKI-SEICOS Σ21L	S-Σ21L

## 1-1 Items requiring attention when reading this manual.

- (1) In this manual and the reference manual “program part”, explanations are made on all the functions that are applicable to these NC machines, including optional functions. The option function selected for adoption are different for each machine. Please confirm the specification of the machine beforehand, as there may be some functions referred to in the manual are not usable depending on the machine.
- (2) In this manual, those functions not specifically remarked “able” should be understood as “unable”.
- (3) The contents of this manual may be changed without notice to meet a future machine improvement.

Note) Programs, parameters, macro variables and tool compensation amounts, etc. are stored in the memory of NC unit. Generally, these data are not lost by switching the power ON/OFF.

Nevertheless, data could be lost inadvertently or by erroneous operation. Also a case may occur when you are compelled to have valuable data in the memory cleared for restoring the system from a trouble.

To cope with such an unexpected situation, we suggest that you take note of the important data and keep them separately. It facilitates quick restoration of working condition of the machine by re-entering the data.

Note) When processing a work, do not start the operation suddenly. Make a trial warm-up run first for fully confirming that the machine acts correctly, then proceed with subsequent operating procedures.





## 2. SPECIFICATIONS

### 1. CONTROLLED AXES

#### 1-1 Controlled Axes

The 2 axes, X and Z, are controllable. Optionally, additional axes can be appended. The  $\Sigma 10L$  can control up to 6 axes, including 4 additional axes, the 3rd through 6th axes. The  $\Sigma 21L$  can control up to 4 axes, including 2 additional axes, the 3rd and 4th axes.

#### 1-2 Simultaneous Controllable Axes

The 2 axes, X and Z, can be controlled simultaneously regardless of rapid traverse or cutting feed. Table 1-2 shows the configuration of the controlled axes.

Table 1-2

	No. of Axes	Axis Name	Remarks
Standard controlled axes	2 axes	X, Z	
Additional axes control	$\Sigma 10L$ : 4 axes $\Sigma 21L$ : 2 axes	Select out of Y, A, B and C	
Simultaneous controllable axes	Standard 2 axes + additional axes	All the axes specified by the system (up to 4 axes)	

Note 1) The number of controlled axes, and relations between the axis name and axes can be selected with parameters.

#### 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 parameters, refer to "Parameters".

#### 1-4 Maximum Commandable Value

Table 1-4 (next page) shows the increment systems and commandable values.

#### 1-5 10-Time Minimum Setting Unit

The input increment can be made 10 times larger by parameter setting. Table 1-5 (next page) shows the commandable values.

Table 1-4

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

[10-Time Minimum setting unit]

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

Table 1-5

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

Note 1) For the Type IS-A, nothing is changed even if you set the parameter for 10-time minimum setting unit.

## 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 scale.

Absolute encoder will be installed for adding the optional absolute position detection function.

## **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 ANGLE DESIGNATION LINEAR INTERPOLATION (G01)**

With G01, an angle from Z axis is specified where linear interpolation is commanded.

### **2-4 Circular Interpolation (G02, G03)**

Circular interpolation can be performed arbitrarily at 0 to 360° 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)**

**Not available with  $\Sigma 21L$ .**

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.....Not available with $\Sigma 21L$ .**

Another axis is linearly interpolated synchronously with circular interpolation.

### **2-8 Polar Coordinate Interpolation**

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**

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 circular interpolation can be performed in combination with other linear axis.

### 3. THREADING

#### 3-1 THREADING (G32)

With F or E code, a thread lead is directly commanded. With E code, you can assign the number of thread ridges per inch can be assigned by a parameter.

#### 3-2 MULTIPLE THREADIGN (G32)

Use this to perform multiple thread cutting which has two or more thread ridges in a lead. With Address Q, command a threading start shift angle. be performed.

#### 3-3 VARIABLE LEAD THREADING

By commanding the increase or decrease amount in lead per one screw thread turning, you can perform variable lead threading. With Address K, command a lead changing amount.

### 4. FEED FUNCTION

#### 4-1 RAPID TRAVERSE RATE AND RAPID OVERRIDE

The maximum speed available in the axial direction is 240,000mm/min (IS-B). Further, override can be applied to rapid traverse by rapid override.

#### 4-2 CUTTING FEED RATE AND FEED OVERRIDE

The maximum feed rate range available for setting is 6~240,000mm/min (IS-B).  
Override can be applied, by feed rate override, within a range of 0~200% by every 10%.

Table 4-2 shows the ranges of feed rate command values:

		TYPE	F COMMAND RANGE
Feed per minute (G98)	mm	F60	F1~F999999 (mm/min)
		F61	F0.1~F999999.9 (mm/min)
	inch	F51	F0.1~F99999.9 (inch/min)
		F52	F0.01~F99999.99 (inch/min)
Feed per revolution (G99)	mm	F32	F0.01~F999.99 (mm/rev)
		F33	F0.001~F999.999 (mm/rev)
	inch	F23	F0.0001~F99.999 (inch/rev)
		F24	F0.001~F99.9999 (inch/rev)
Threading (G32)	mm	F35	F0.00001~F999.99999 (mm/rev)
		F26	F0.000001~F99.999999 (mm/rev)
	inch	F26	F0.000001~F99.999999 (inch/rev)
		F17	F0.0000001~F9.9999999 (inch/rev)

Note 1) The above types are selected with parameters. (Parameter no. 3401)

Note 2) The maximum cutting feed rate is limited by the cutting feed clamp rate set with a parameter.

Note 3) Address E may be used as the feed rate, where the command value ranges for E and F are the same.

### **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 command. The G61 mode is cancelled by specifying G64.

### **4-8 Automatic Corner Override (G62)..... Not available with $\Sigma 21L$ .**

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) ..... Not available with $\Sigma 21L$ .**

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

## **6. COORDINATE SYSTEM**

### **6-1 Tool Nose Coordinate System**

At the time of turret indexing or manual zero point return, the tool nose position assuming the machining reference point to be zero (0) is automatically set in the coordinate system.

### **6-2 Coordinate System Setting (G50)**

An axis command following G50 sets the coordinate system where a current tool coordinate value will be a specified value.

### **6-3 Machine Coordinate System Selection (G53)**

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

### **6-4 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

## **7. COORDINATES AND DIMENSIONS**

### **7-1 Absolute/Incremental Programming**

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

Absolute : X Y Z B C

Incremental : U V W D H

### **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.

The addresses which can use decimal points are X, Y, Z, A, B, C, D, H, U, V, W, I, J, K, R, P, Q, E, and F.

Depending on conditions applies, however, a decimal point may not be usable.

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

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

- G20: Inch input
- G21: Metric input

## **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.

### **8-3 Constant Surface Speed Control (G96, G97)**

With a surface speed directly assigned with S code, this function serves to continuously control the spindle motor rpm so that the circumferential speed is held constant to changes in tool position. Command to make this function valid or invalid is performed with G code.

G96: Constant surface speed control is performed.

G97: Constant surface speed control is not performed.

## **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.

When ATC is mounted, use T6 digits. When not, use T4 digits.

Use T code to perform ATC operation, tool rest indexing, setting of a work coordinate system (tip coordinate system), combined compensation, etc..

### **9-2 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.

Note) For the  $\Sigma 21L$ , only a life count by M12 is allowed.

## **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

Table 11-4

Address	Description
A	Additional axis coordinate value
B	Additional axis coordinate value, 2nd miscellaneous function
C	Additional axis coordinate value
D	Incremental coordinate value of B axis
E	Feed function (threading)
F	Feed functions
G	Preparatory functions
H	Incremental coordinate value of C axis
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	Fixed cycle, Multiple threading starting angle
R	Radius command value for circular interpolation, Canned cycle
S	Spindle functions
T	Tool functions
U	Incremental coordinate value of X axis, Dwell
V	Incremental coordinate value of Y axis
W	Incremental coordinate value of Z axis
X	X-axis coordinate value, Dwell
Y	Y-axis coordinate value
Z	Z-axis coordinate value

## 11-5 Command Words and Command Value Ranges

Table 11-5

Function	Address	Metric input.	Inch Input.
Program number #	O	1~999999999	1~999999999
Sequence number #	N	1~999999999	1~999999999
Preparatory function	G	0~999	0~999
Coordinate value	X, Y, Z, U, V, W, I, J, K, Q, R, A, B, C,	$\pm 99999.999(\text{mm})$ $\pm 99999.999(\text{deg})$	$\pm 9999.9999(\text{inch})$ $\pm 9999.9999(\text{deg})$
Feed function	F	See the table 4-2.	See the table 4-2.
Spindle function	S	0~32767	0~32767
Tool function	T		
Miscellaneous function	M	0~999999999	0~999999999
Dwell	P, X, U	0~99999.999(sec)	0~99999.999(sec)
Call program number	P	1~999999999	1~999999999
Repeat times	L	1~999999999	1~999999999

## 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 (G842, G843)

You can perform highly accurate tapping at high speed in G842/G843 tap cycles, by fully synchronizing a rotary tool rotation with the feed of X or Z axis.

## 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 (G80~G89, G831, G841, G861)**

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

## **12-2 Fixed Cycle (G90, G92, G94)**

The following 3 kinds of fixed cycles can be commanded:

- (1) Cutting cycle A (G90) → outside/inside diameter cutting
- (2) Threading cycle (G92)
- (3) Cutting cycle B (G94) → end face cutting

## **12-3 Maltipul Fixed Cycle**

Several kinds of fixed cycles are prepared beforehand to facilitate a program. With information of the finished configuration alone being given, the tool passage for rough cutting to the end is automatically fixed. A fixed cycle for threading is also available. They are divided roughly into the following three:

Maltipul type fixed cycle A : G70, [(G71, G72) Type 1], G76

Maltipul type fixed cycle B : A + G73, G74, G75

Maltipul type fixed cycle C : B + [(G71, G72) Type 2]

## **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.

## **13. TOOL OFFSET FUNCTIONS**

### **13-1 Automatic Tip R Compensation and Cutter Compensation**

(1) Automatic Nose R Compensation (G143)

It is normally held at G143 (Automatic Nose R Compensation Valid Mode). Therefore, without G code not being commanded, tip R compensation is automatically executed.

Nose R is set to tool compensation R and virtual nose points (1~8) to T.

(2) Cutter Compensation (G145 : G40~G42)

With G145 command, Cutter Compensation mode is created.

With G40~G42 commands while in this mode, cutter compensation can be carried out.

A tool diameter is set to tool compensation R and a virtual nose point (9) to T.

### **13-2 Groove Width Compensation (G150 to G152)**

When a grooving tool is used, one virtual tool nose (for example, 3) is used to run the program to apply compensation. It is also necessary to compensate the other virtual tool nose (for example, 4) side. When this is done, this function compensates the groove width by specifying a G-code.

### **13-3 Addition of Tool Offsets**

The number of tool offset pairs can be expanded up to 200.

Expansion of tool offset pairs: 64/99/200 (Up to 99 pairs for the  $\Sigma 21L$ )

Note) The number of tool offset pairs, 200, is only allowed when the ATC is attached.

## **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 9999 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. MEASURING FUNCTIONS**

### **15-1 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, canceling the rest of this command. A skip signal input position can be read with a system macro variable.

### **15-2 Q SETTER**

Through simple manual operation with the touch sensor, tool offset volume is automatically written.

### **15-3 Q-setter Repeat Function**

By measuring the tool with the Q-setter once, automatic measurement with the Q-setter is activated by simple operation when changing tips.

## **16. CUSTOM MACRO**

### **16-1 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.

## **17. AXIS CONTROL**

### **17-1 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.

### **17-2 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. MANUAL OPERATION**

### **18-1 Manual Continuous Feed**

With the axial feed switch being pressed in Jog mode, manual continuous feed is performed.

Feed rates are as follows:

① Jog feed

Jog feed rate can be changed over into 24 stages through use of the switch. Speed in 24 stages is set with parameters.

② Manual rapid

When jog feed is executed as the rapid traverse button being pushed, manual rapid traverse is available.

### **18-2 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 x 1, x 10, or x M (M=1 to 100 set in a parameter) by a signal from the machine.

## **19. AUTOMATIC OPERATION**

### **19-1 DNC Operation**

With the optional board being added, DNC operation from the host CPU is made available.

### **19-2 Program Number Search**

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

### **19-3 Sequence Number Search**

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

### **19-4 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 Machining break point return is a function to position a tool to a break point by jog feed.

### **19-5 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.

### **19-6 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 pre-reading blocks
Automatic tip R compensation	4 blocks
Multi buffer	12 blocks
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,  
G10, G20, G21,  
M00, M01, M02, M30  
T Command

Note 3) As the automatic tip R compensation is normally held valid (G143 mode), pre-reading of 4 blocks is carried out.

- (2) When the SINGLE BLOCK switch is turned on

(State in which automatic tip R compensation is held invalid : G140, G145 mode)

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) Pre-reading is performed when any of the following applies:

- (a) In Multi Buffer mode (G251)
- (b) Tool radius compensation mode (G41, G42)
- (c) When optional angle chamfering corner R is specified (, C/, R)
- (d) Thread cutting (G32)
- (e) Tapping mode (G63)

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

## 19-7 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.

### **19-8 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.

### **19-9 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  $\Sigma$  DATA SERVER INSTRUCTION MANUAL, NT DOMAIN TYPE)

### **19-10 Scheduler**

The program number per work and the number of machining works are managed to operate different types of works continuously, using the robot or the bar feeder.

(For details, see SCHEDULER AJC FUNCTION INSTRUCTION MANUAL (SEIKI-SEICOS $\Sigma$ ))

## **20. PROGRAM TEST FUNCTIONS**

### **20-1 Machine Lock**

In the machine lock mode, the machine does not move, but the position display is updated as if the machine were moving.

When the machine lock is turned ON→OFF in auto operation, Machine is shifted by the amount moved by the machine lock.

### **20-2 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.

### **20-3 Single Block**

Program commands can be executed block by block.

### **20-4 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.



## **21. DISPLAY AND SETTING**

### **21-1 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.

### **21-2 Run Hour Display**

Machine run hours are displayed in the format of hours:minutes:seconds. Display is made by each of the functions including the expected ending, working time, cutting time, lap T, and date/time.

### **21-3 Work Count Function**

The number of machined workpieces can be counted by the M12 command. By setting the number of workpieces beforehand, a signal is output to the machine when reaching the prescribed number of workpieces.

## **22. PART PROGRAM STORAGE & EDITING**

### **22-1 Part Program Storage & Editing**

The contents of the NC tape can be stored and edited. Relevant operations include deletion, alteration, and insertion. Range editing is allowed by expanded part program editing. Use of backgrounding allows you to edit another program during automatic operation.

Tape storage length : 80, 160, 320, 500 or 1,000 m

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

Note) For the  $\Sigma$ 21L, the maximum part program storage length and registered programs are 320 m and 400 programs, respectively.

### **22-2 Part Program Comparison**

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

## **23. DIAGNOSTIC FUNCTIONS**

### **23-1 Self Diagnostic Function**

This function makes various checks.

The signals exchanged between the machine and NC unit can be confirmed on the screen.

- G contact : From PMC to CNC
- RG contact: From PMC to CNC
- F contact : From CNC to PMC
- RF contact : From CNC to PMC
- X contact : From machine to PMC
- Y contact : From PMC to machine

## **23-2 Alarm Diagnosis**

If the CNC has any error, it displays its corresponding alarm number and message.

## **23-3 Cutting Monitoring Function**

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

# **24. DATA INPUT AND OUTPUT**

## **24-1 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.

# **25. SAFETY FUNCTIONS**

## **25-1 Emergency Stop**

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

## **25-2 Overtravel**

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

## **25-3 Interlock**

There are three kinds of axis interlocks; all axes interlock, individual axis interlock, and axis directional interlock.

If the interlock is applied while moving the axis, the machine will be decelerated to stop. If an interlock signal is canceled, the machine will be accelerated to resume operation.

## **25-4 Stored Stroke Limit 1**

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

## **25-5 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.

Set stored stroke limit-2 with a parameter whether the inside or outside of the set area should be prohibited. Stored stroke limit-3 assumes the inside of the set area to be a prohibited area.

Use a G-code to enable/disable stored stroke limit-2.

- G22: Enable
- G23: Disable

(Stored stroke limit-3 is enabled regardless of the G-code)

## **25-6 Stroke Check Before Move ..... Not available with $\Sigma 21L$ .**

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

## **26. STATUS OUTPUT**

### **26-1 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.

### **26-2 Automatic Operation Running Signal**

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

### **26-3 Automatic Operation Stopping Signal**

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

### **26-4 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.

## **27. EXTERNAL DATA INPUT**

### **27-1 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

### **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. Quick Tool Setter (Q Setter)**
- 6. Q Setter Barrier**
- 7. Q Setter Repeat Function**
- 8. Simple Jaw Edge Forming Function**

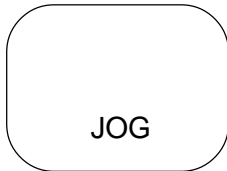
## 1. Manual Operation

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

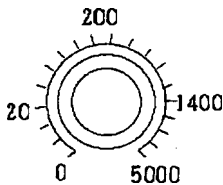
### 1-1 Jog Feed

The machine can be operated continuously by manual operation.

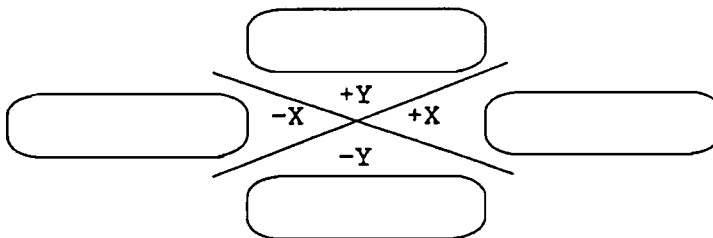
- (1) Select the mode selector switch "JOG".



- (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.

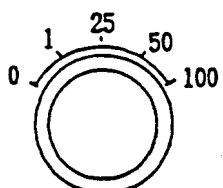
### 1-2 Manual Reference Point Return

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.

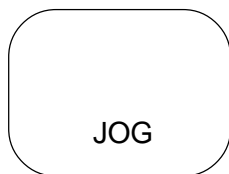
(2) When an emergency stop resulted and the machine moved.

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

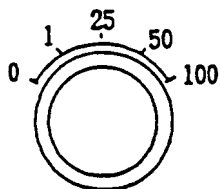
### 1-3 Rapid traverse

The machine can be rapid traversed continuously by manual operation.

- (1) Select the mode selector switch "JOG".



- (2) Select an appropriate rapid traverse override.



- (3) Select the axis you want to move, and "RAPID" simultaneously. 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

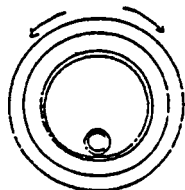
The machine can be finely fed by turning the manual pulse generator.

- (1) Select the mode selector switch "HANDLE".

- (2) Select the handle axis.

- (3) Select a handle magnification.

- (4) Turn the handle.



Clockwise: "+" direction

Counterclockwise: "-" 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.

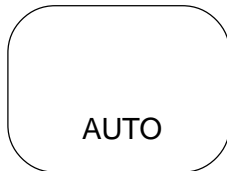
Note 3) In some cases, the mode selector switch "HANDLE" of (1) may not be provided, where Handle mode is selected by handle axis selection of (2).



## 2. Automatic Operation

### 2-1 Automatic Mode

- (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.

⊙ CYCLE START Lamp



CYCLE START

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 "EIDT".
  - (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".

Part Program Storage Length	Registered Programs
80 m	100 programs
160 m	100 or 200 programs
320 m	100, 200 or 400 programs
500 m	100, 200, or 400 programs
1000 m	100, 200, 800, or 1,000 programs

- (3) Part program storage length and registered programs

Note) For the  $\Sigma 21L$ , the maximum part program storage length and registered programs are 320 m and 400 programs, respectively.

## 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 model 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.

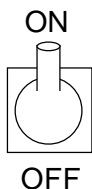


Normally, dry run is effective only during cutting feed.

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

## 2-5 Single Block

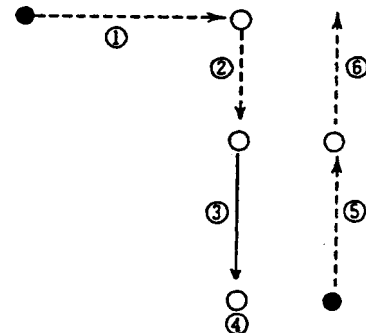
The single block function stops the machine after executing one block. Turn on the SINGLE BLOCK switch.



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 ⑥ 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 (G98) and feed per revolution (G99).

### (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 and automatic rapid traverse override can be fixed at 100 %.

## 2-7 Per-program Machining Time Display Function (Option)

When you run the program, the programmed machining time is displayed on the machining time display screen.

Up to 10 main program numbers and machining times can be displayed. If you run more than 10 programs, the oldest programs will be discarded sequentially. However, the machining time of the same program number will be updated.

The time from the start of reset condition to the next reset condition in the memory operation mode will be counted. The reset condition at this stage is the condition obtained by pressing the reset key, external reset signal, M02, M30, etc.

The time counted will be only the time automatic operation is executed, therefore, the machining time will not be counted during hold and during single block stoppage.

Calculation and display of machining time

PROGRAM (AUTO)

```
O1 ;
G0 X100. ;
X0 ;
Z100. ;
Z0 ;
M30 ;
%
```

00001  
N

PROGRAM No.		hour	min.	sec
	3	0	00	00
	4	0	00	18
2000	0	0	01	15
3000	1	0	00	10
3000	3	0	00	59
4000	0	0	00	29
5400	0	0	00	09
194		0	00	01
4989		0	00	09
	1	0	00	03

Data> \_

CONDENSE /1	MERGE /2	/3	/4	EXIT /5	LIST CHANGE/6	PROGRAM LIST /7	/8	/9	/0
----------------	-------------	----	----	------------	------------------	--------------------	----	----	----

- ① Select the automatic (memory) operation mode and press the reset key.
- ② Select the overall (program) screen and search for the program that calculates the machining time.
- ③ Run the program and execute machining.
- ④ If the reset key is pressed or M02, M30 is executed, the count for machining time will be terminated.
- ⑤ Select the program screen, press F7/Program list and then press F5/Machinig time. The machining time screen will be displayed.
- ⑥ If you repeat the procedures from ① to ⑤ the program number and time will be sequentially displayed on the machining time screen.

The maximum number of programs for which the machining time can be displayed is 10. If the number of programs exceeds 10, the oldest programs will be discarded sequentially. However, the machining time of the same program number will be updated.

## 2-8 Tape Operation (Simple DNC Operation)

The machine is directly operated by the 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 stored in the internal memory of the NC unit.

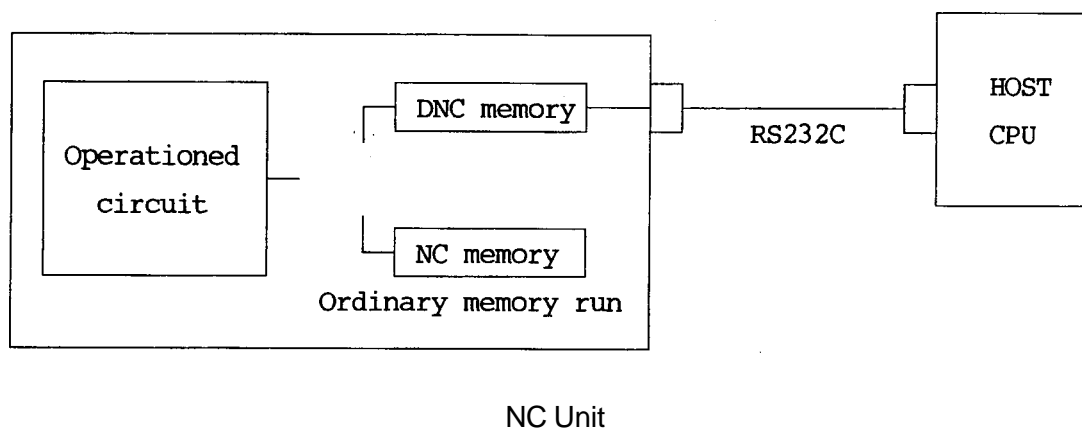
Note) Be sure to disable DNC operation during tape operation.

- ① Select an input device and baud rate in the I/O screen.  
(When using the RS-232C of the operation panel, select the RS-232C.)
- ② Select the mode selector switch, TAPE.
- ③ Make setting on the part of the input device to make the system ready.  
(Set the paper tape when the reader is used, and make the connected device ready for output when the RS-232C is used.)
- ④ Press the PROGRAM START button.  
→ This reads one block worth of the program from the external device and start tape operation.

Note) Since an internal processing buffer is provided, the buffer worth of the data are pre-read.

## 2-9 DNC Operation (Optional)

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

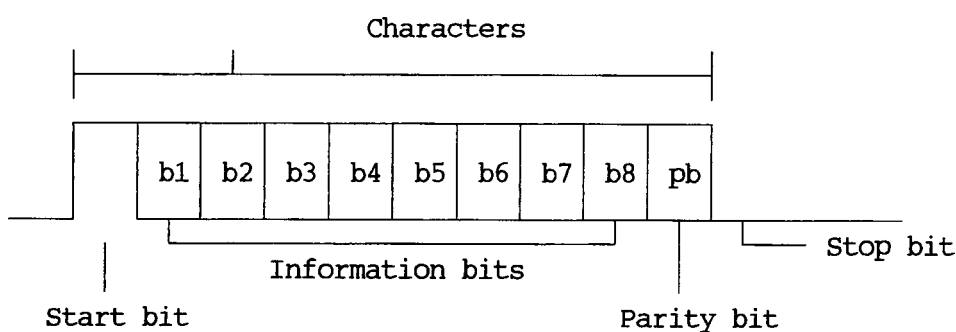


(1) Interface Between Host CPU and NC Unit

	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.



JD5C ( $\Sigma 10L$ )/JD28A ( $\Sigma 21L$ )  
(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.

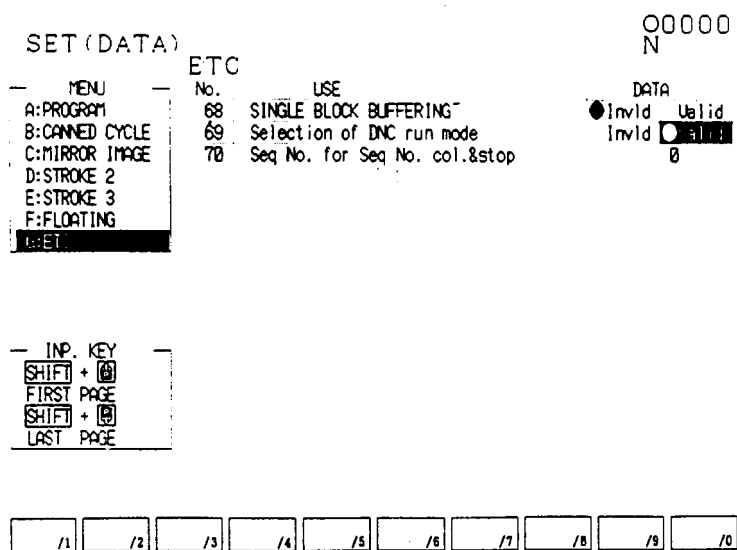
③ No.0182 Baud rates when a remote buffer is used.

Set Value	Baud Rate (bps)	Set Value	Baud Rate (bps)
1	50	9	2400
2	100	10	4800
3	110	11	9600
4	150	12	19200
5	200	13	38400
6	300	14	76800
7	600	15	86400
8	1200		

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 CPU.

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

#### (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;  
→ The programs in the main memory + D1 are to be run/edited.
- When the current directory is D2;  
→ 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 submemory 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  $M_n$ , that on the part of the submemory by  $S_n$  ( $n = 1, 2, 3, \dots$ ), and program call by  $\rightarrow$ , respectively.

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

The following is allowed:

$M1 \rightarrow M2 \rightarrow M3 \rightarrow M4 \rightarrow M5 \rightarrow M6 \rightarrow M7 \rightarrow M8 \rightarrow M9$ .

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

$S1 \rightarrow M1 \rightarrow S2 \rightarrow S3 \rightarrow S4 \rightarrow S5$

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

$S1 \rightarrow M1 \rightarrow S2 \rightarrow S3 \rightarrow S4 \rightarrow S5 \rightarrow M2$  Allowed

$S1 \rightarrow M1 \rightarrow S2 \rightarrow S3 \rightarrow S4 \rightarrow S5 \rightarrow S6$  Disallowed

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

$M1 \rightarrow S1 \rightarrow S2 \rightarrow S3 \rightarrow S4$

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  $S4$ .

$M1 \rightarrow S1 \rightarrow S2 \rightarrow S3 \rightarrow S4 \rightarrow M2$  Allowed

$M1 \rightarrow S1 \rightarrow S2 \rightarrow S3 \rightarrow S4 \rightarrow \text{Data server}$  Allowed

$M1 \rightarrow S1 \rightarrow S2 \rightarrow S3 \rightarrow S4 \rightarrow S5$  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 the 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**

PROGRAM (EDIT)		O 5 0 0 0 N 0 0 0 5		
DIR SET		<USER>		
<USER>				
<USER>		1 2 3		
<ARA>				
DIR>				
MAKE	DELETE		EXIT	
/1	/2	/3	/4	/5
			FORMAT	
			/7	/8
				/9
				/0

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.

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

Press **F1/MAKE**.

You are prompted, "O.K. ? Y-YES N-NO."

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 programs contained in it.

Move the cursor to the directory you want to delete.

Press **F2/DELETE** .

You are prompted, "DO YOU WANT TO DELETE ? Y-YES N-NO."

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.

Move the cursor to the directory you want to set.

Press **INPUT** .

You are asked, "O.K. ? Y-YES N-NO."

Press **Y** to set or **N** to cancel.

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, 08000 to 09999 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)

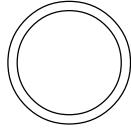
= 1 Up to 9 submemory programs can be run simultaneously depending on the condition. (See Note in 2-10-3)

### **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 a safe direction, press the reset key, and cancel 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	RESEDT 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 PosDition, 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 oDf 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 keys	Used to input the alphabet, numbers, etc.
⑨	SHIFT key	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.

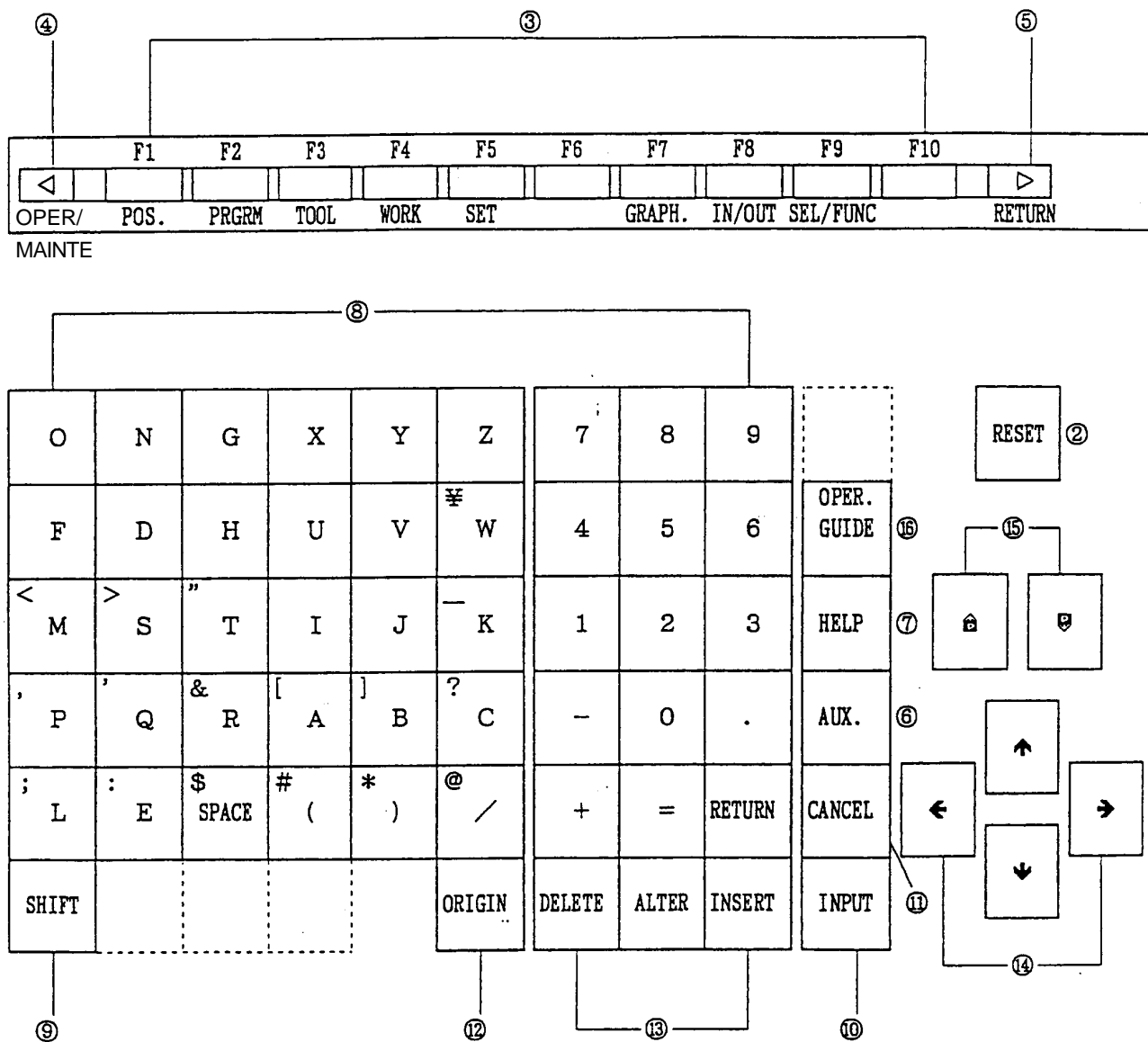


Fig. 4-1 NC Operation Panel



## 5. Quick Tool Setter (Q Setter)

### 5-1 Outline

A tool offset amount can be automatically written by simply bringing the tool nose into contact with the touch sensor manually.

### 5-2 Operation Method

- (1) Set the quick tool setter.
  - (a) Set the tool setter. (Fig. 5-2)
  - (b) Select the MANUAL mode.
  - (c) A screen display is automatically switched to the Tool (Offset) screen; "Q SETTER" is displayed to inform you that the system is ready. (Fig. 5-1) When initial reference point return has not been completed, setting the tool setter results in an alarm.
- (2) Call the tool for which you want tool offset.
- (3) Select and confirm the tool offset number.

The tool offset number is consistent with the called tool number, and the cursor moves automatically. When the cursor is moved with the key, the cursor-indicated tool offset number is selected.
- (4) Bring the tool nose closer to the tool setter.
  - (a) When bringing it closer by jog feed, select the JOG mode and press the X- or Z-axis direction selector switch.
  - (b) When bringing it closer by handle feed, select the HANDLE mode, and then, the X or Z axis. Next, set a handle magnification to 100/1 and turn the handle.
- (5) Position the tool nose to the center of the sensor by handle feed. (Fig. 5-3)
- (6) Apply the tool tip to the touch sensor in jog feed.

The jog feed rate is held at the constant rate set with a parameter. Maintain the tool tip in continuous jog feed by not bringing it close to or keeping it away from the touch sensor.
- (7) Bringing the tool nose into contact with the touch sensor inputs a tool offset amount automatically. When the tool nose comes into contact with the sensor, the axis stops, emitting a "beep" sound. (Fig. 5-4)
- (8) Put away the tool nose from the touch sensor by more than a retouch return amount (parameter), and bring it into contact again. Repeating this, confirm the tool offset amount shown on the screen. (Fig. 5-5)
- (9) Put away the tool nose to a safety position from the touch sensor. (Fig. 5-6)
- (10) Put away the tool nose to the position where the tool can be called.
- (11) Obtain the tool offset amount of the next tool.

Repeating the above steps (2) through (10), obtain the tool offset amount of each tool sequentially.
- (12) House, the tool setter. (Fig. 5-7) The display of Q SETTER disappears.

While in axis shifting, do not store the tool setter.

# TOOL OFFSET

00001  
N

GEOM.		WEAR	
01 (TURN) (RGH-FACE)			
X	0.000	U	0.035
Y	0.000	U	0.000
Z	0.000	W	0.000
R	0.000	Q	0.000
T	0		
H	0.000	J	0.000
04 ( ) ( )			
X	0.000	U	0.000
Y	0.000	U	0.000
Z	0.000	W	0.000
R	0.000	Q	0.000
T	0		
H	0.000	J	0.000

GEOM.		WEAR	
02 ( ) (RGH-FACE)			
X	0.000	U	-0.005
Y	0.000	U	0.000
Z	0.000	W	0.000
R	0.000	Q	0.000
T	0		
H	0.000	J	0.000
05 ( ) ( )			
X	0.000	U	0.000
Y	0.000	U	0.000
Z	0.000	W	0.000
R	0.000	Q	0.000
T	0		
H	0.000	J	0.000

GEOM.		WEAR	
03 ( ) ( )			
X	0.000	U	0.000
Y	0.000	U	0.000
Z	0.000	W	-0.015
R	0.000	Q	0.000
T	0		
H	0.000	J	0.000
06 ( ) ( )			
X	0.000	U	0.000
Y	0.000	U	0.000
Z	0.000	W	0.000
R	0.000	Q	0.000
T	0		
H	0.000	J	0.000

ABSOLUTE		MACHINE	
X	0.000	X	-198.964
Y	-0.013	Y	0.000
Z	0.000	Z	-99.995

Q-SETTER

GEOMETRY : X = \_

TOOL LIST /1	TYPE /2	NAME - /3	/4	/5	Q-SETTER REPEAT/6	DATA CLEAR /7	/8	LIFE SPARE /9	/0
-----------------	------------	--------------	----	----	----------------------	------------------	----	------------------	----

Fig. 5-1

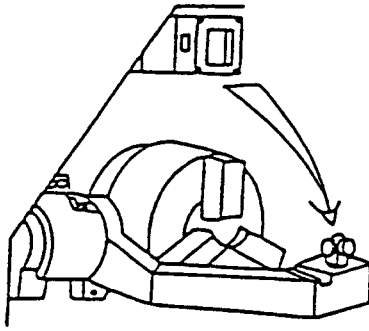


Fig. 5-2

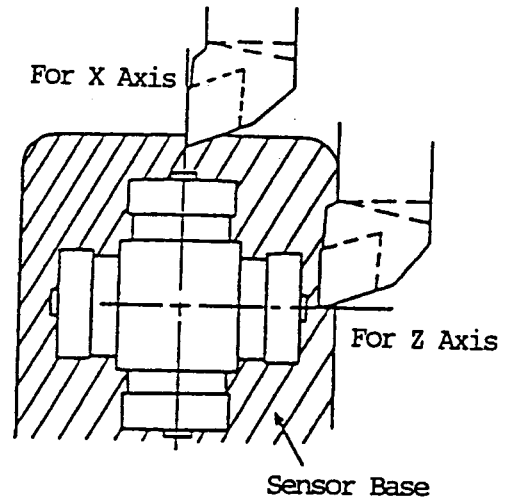


Fig. 5-3

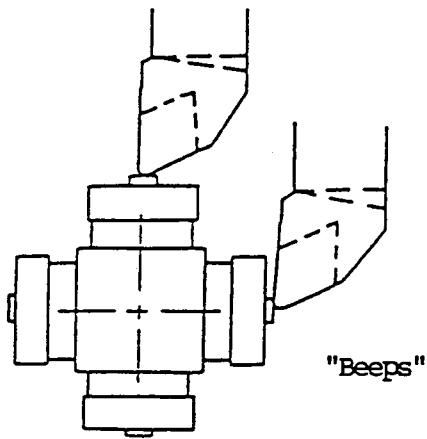


Fig. 5-4

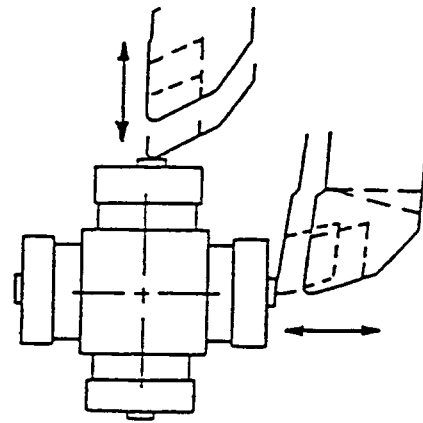


Fig. 5-5

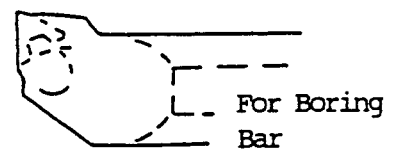
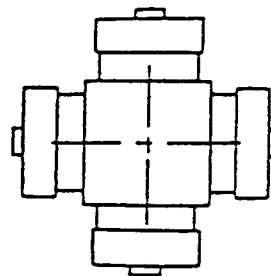
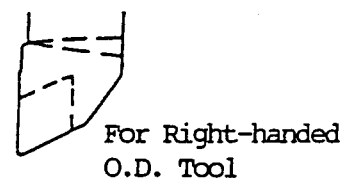
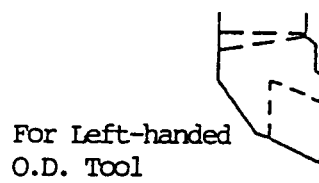


Fig. 5-6

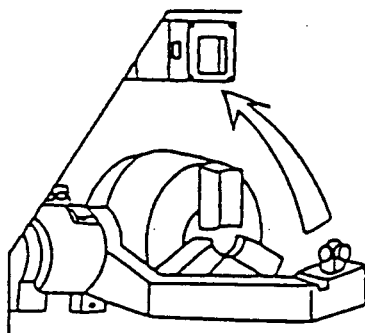


Fig. 5-7

### 5-3 Tool Nose Point Interlock

Bringing the tool into contact with the sensor stops the axis and checks the tool nose point. When the contact directions and tool nose point conditions shown in the table below are not met, an interlock is applied to an input of tool offset amount to disable a write.

The tool nose point for tool nose point interlock is that for the offset number currently selected.

Tool Nose Point Contact Direction	1	2	3	4	5	6	7	8	9	0
+ X	×	×	○	○	×	×	×	○	○	○
- X	○	○	×	×	×	○	×	×	○	○
+ Z	×	○	○	×	×	×	○	×	○	○
- Z	○	×	×	○	○	×	×	×	○	○

The contact and move

○ : Write enabled

× : Write disabled

Note) To enable this interlock, it is necessary to set "1" in the bit 0 (QTL) of the parameter No.6244.

### 5-4 Related Parameters

#### (1) Parameters for the tool (offset) memory

With signed parameters, set the distances from the reference tool nose position to the respective contact surfaces at the machine origin.

No.6281 Q setter contact surface amount on the "+" side (inside)

No.6282 Q setter contact surface amount on the "-" side (outside)

Note 1) Axis type parameters provided for each axis

Note 2) For the diameter designated axis, set a diameter.

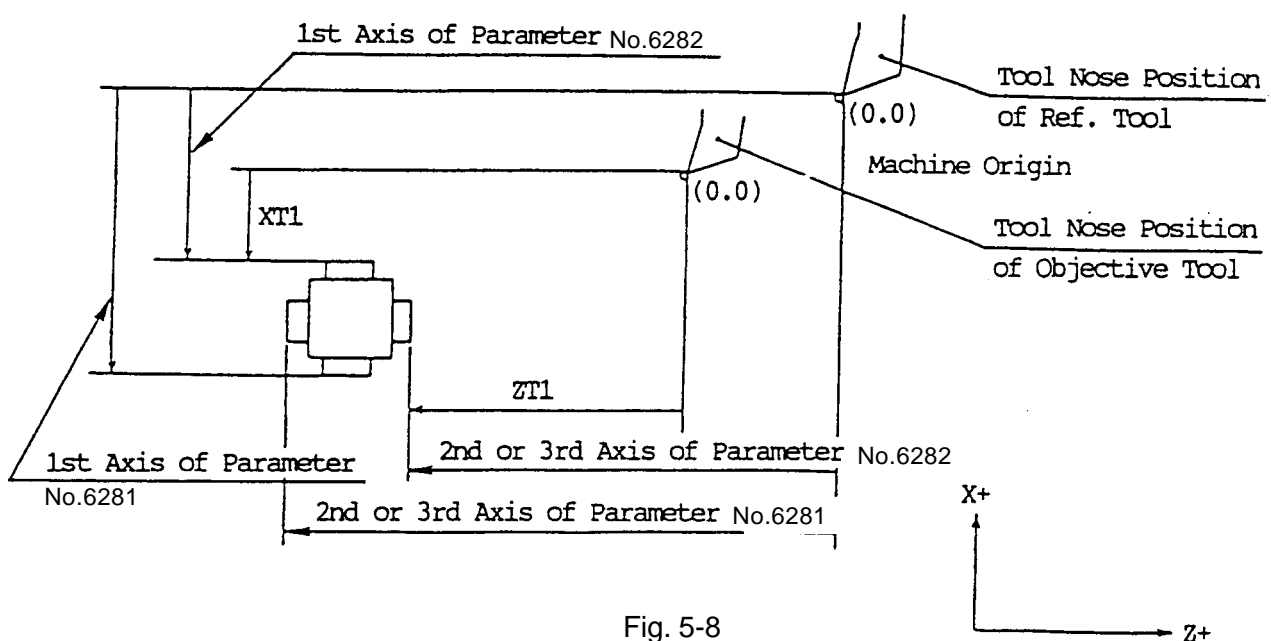


Fig. 5-8

Calculate the tool offset amount by the following formula.

Tool offset amount = Skipped machine coordinate value – Q-setter contact surface amount parameter (6281 or 6282)

(2) Tool nose point interlock

No.6244 bit 0 (QTL)

QTL Tool nose point interlock

0: Disabled, 1: Enabled

(3) Retouch return amount

No.6256 Retouch return amount of the Q, Z, and B setters

## 5-5 Related Alarms

No.712 The Q setter mode was selected without performing manual zero point return.

No.713 Measurement is being made with the Q setter, using 2 or more simultaneous axes

No.714 A tool nose point interlock is being applied with the Q setter.

## 5-6 Compensation Input Checking

When rewriting compensation in Q setter and Q setter repeat, you can perform the following checking:

- ① When the difference between previous tool compensation volume and measured compensation volume does not fall within the set range (parameter), an alarm takes place. The compensation volume, however, is going to be written. Therefore, an alarm takes place, also, when compensation volume is newly taken in Q setter or when erroneous compensation volume is taken as proper data.

When this takes place, ignore it.

- ② When a tool No. does not correspond to the cursor No., an alarm takes place. The compensation volume, however, is going to be written. "Enabled/ Disabled" is switched for both ① and ② through parameter setting.

(1) Related Parameters

No.6244, #6 = 0	Checking of Q setter compensation number	Disabled
= 1		Enabled

No.6244, #7 = 0	Checking of Q setter compensation volume	Disabled
= 1		Enabled

No.6264 Clamp value for Q setter compensation input checking

(2) Related Alarms

No. 709 Data measured in Q setter is abnormal.

No. 719 Q setter carries an abnormal number.

## 6. Q setter barrier

### 6-1 Outline

After measuring the tool compensation amount using the quick tool setter function, if the tool is made to escape in the direction of the reference point, it may touch the touch sensor, depending on the tool diameter.

For instance, if the tool shape is as shown in Fig. 6-1, after measuring the tool compensation amount by means of the sensor in the X+ direction, if the tool is immediately returned to the reference point from this position, the tool nose may touch the touch sensor.

To prevent such occurrences, there is a function that sets up a barrier area near the touch sensor and prohibits motions such as return to the reference point when the nose position has entered this area. Thus, this function prevents damage to the touch sensor.

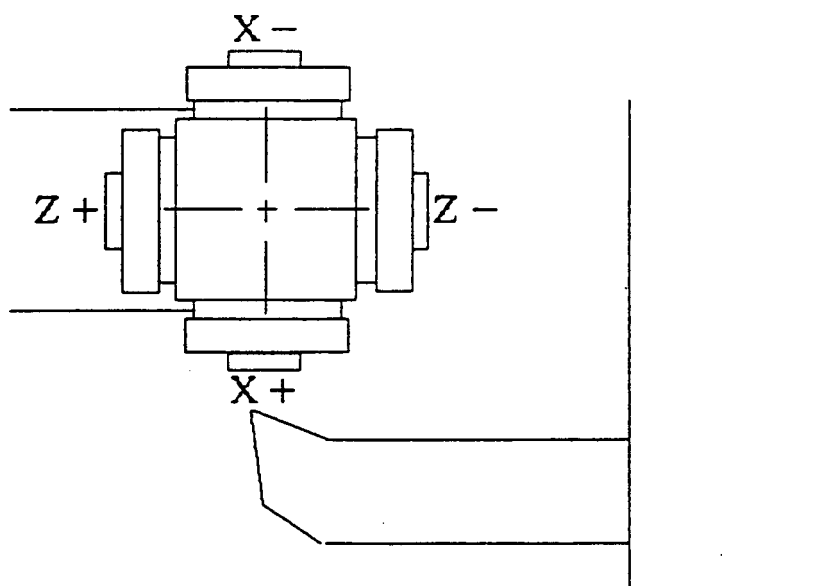


Fig. 6-1

### 6-2 Description of the function

- (1) When the tool setter arm descends and enters the Q setter mode, the control equipment checks whether the tool is within the Q setter barrier area by performing the calculations given below.

X value of machine coordinate system  $\leq$  Parameter 6283 (X) + Tool compensation amount X  
AND

Z value of machine coordinate system  $\leq$  Parameter 6283 (Z) + Tool compensation amount Z  
when the above equations are satisfied, the control equipment judges the tool to be within the barrier area. (Fig. 6-2)

The tool compensation amount is calculated as the sum of the shape of compensation number currently selected and the wear.

- (2) When the tool is in the barrier area, the signal given below is output to the PMC.

PMC address

Q setter barrier area in-signal QBRI (RF24, #1)

Using this signal, process the reference point return interlock and message display with in the barrier area.

Return the arm of the tool setter. When the Q setter mode is terminated, this signal becomes "0" even if the tool is within the barrier area.

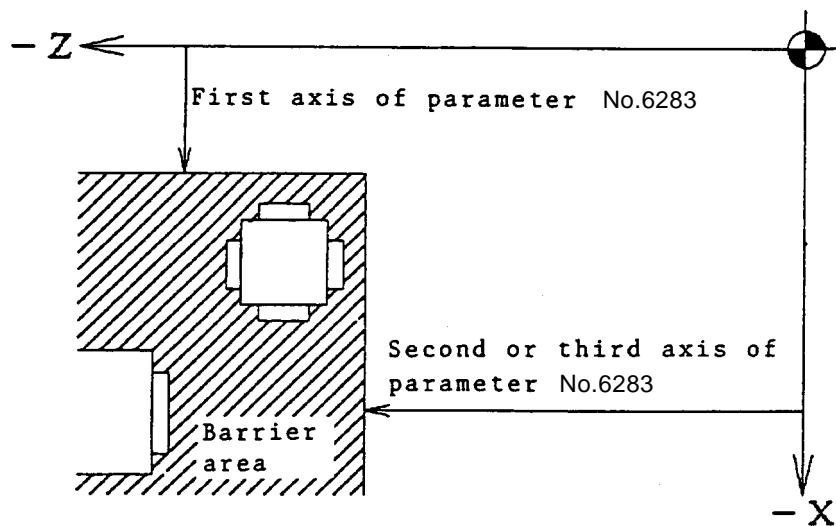


Fig. 6-2

### 6-3 Precautions

- (1) Positions above the barrier line is taken as positions within the barrier area.
- (2) After tool replacement, check will be performed using the previous tool compensation amount, until the new tool compensation amount is written.

### 6-4 Related parameters

- (1) Q setter barrier value  
No. 6283 Q setter barrier value (per axis)
- (2) Q setter barrier check  
Bit 1 of No. 6244 (QBR)  
Q setter barrier check is an given below.  
0: Ineffective  
1: Effective

## 7. Q Setter Repeat Function

### 7-1 Outline

Through a simple operation, you can repeatedly perform measurement on tip changing of a tool which has already been subject to measurement in Q setter.

### 7-2 Operation

- (1) Push **F6/Q SETTER REPEAT** on Tool (Compensation) page, Window page for Q setter repeat is displayed.
- (2) Set Feed mode.
- (3) Changing a tip, perform indexing of the turret to a measurement position. At this time both the cursor inside Turret State on the left side of the page and the compensation data at the top right-hand corner of the page change as linking to the turret face.
- (4) Bring out the Q setter arm.
- (5) Push **F4/REPEAT START**.

If the tool has been already subject to measurement in Q setter (with the data already existing in the measurement position on the page), measurement by Q setter repeat is started. When the tool touches the sensor, a measured value is written in compensation data. The touch mark is displayed on the picture indicating a measurement position.

If measurement has not yet been performed in Q setter, alarm takes place.

**TOOL OFFSET**

**Q-SETTER REPEAT**

TURRET STATUS

TURRET	TOOL	TOOL NAME	MEASURE
No	No		
* 1	1	RGH-FACE	-
2	2	RGH-FACE	
3	3		
4	4		
5	5		
6	6		
7	7		
8	8		
9	9		
10	10		
11	11		
12	12		

[F6 RETURN]

00001  
N

GEOM.		WEAR	
Ø1	(TURN)	(RGH-FACE)	
X	0.000	U	0.035
Z	-198.793	W	0.000
R	0.000	Q	0.000
T	0		
H	0.000	J	0.000

**Q SETTER**

ABSOLUTE

X 198.964  
Y -0.013  
Z 298.788

MACHINE

X 0.000  
Y 0.000  
Z 0.000

TOOL LIST /1

/2

/3

REPEAT START /4

REPEAT HALT /5

EXIT /6

DATA CLEAR /7

/8

LIFE SPARE /9

/0



### 7-2-1 Function Key

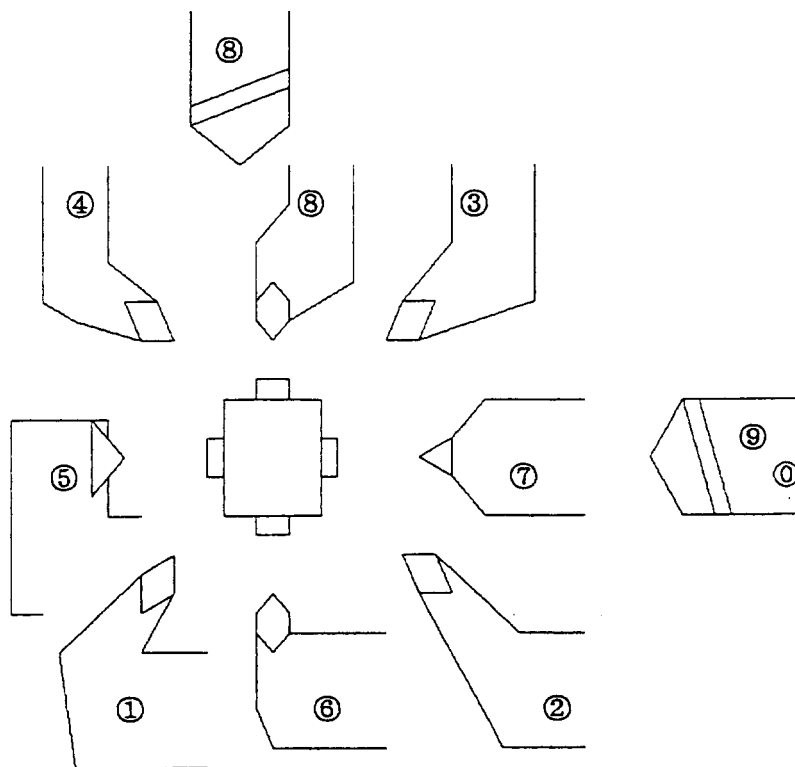
- F4/ Repeat Measurement Start : “OK? Y- Yes N- No” message appears. With  , measurement starts.
- F5/ Repeat Measurement Halt : Measurement operation is stopped.
- F6/ Function Return : Screen display returns to Tool Compensation.
- F7/ Data Delete : On appearance of a window, select either “One Meas. Spot” or “Whole Meas. Spot” and push  . “OK? Y- Yes, N- No” message appears. With  , the measurement spots are deleted.

### 7-3 Working

The working patterns, as shown in Fig. 3, can be divided into 8 kinds according to the virtual nose points.

When Virtual Point 1, 2, 3, or 4 being assigned, both X and Z axes are subject to measurement. The order for measurement, in that case, is always Z to X axis.

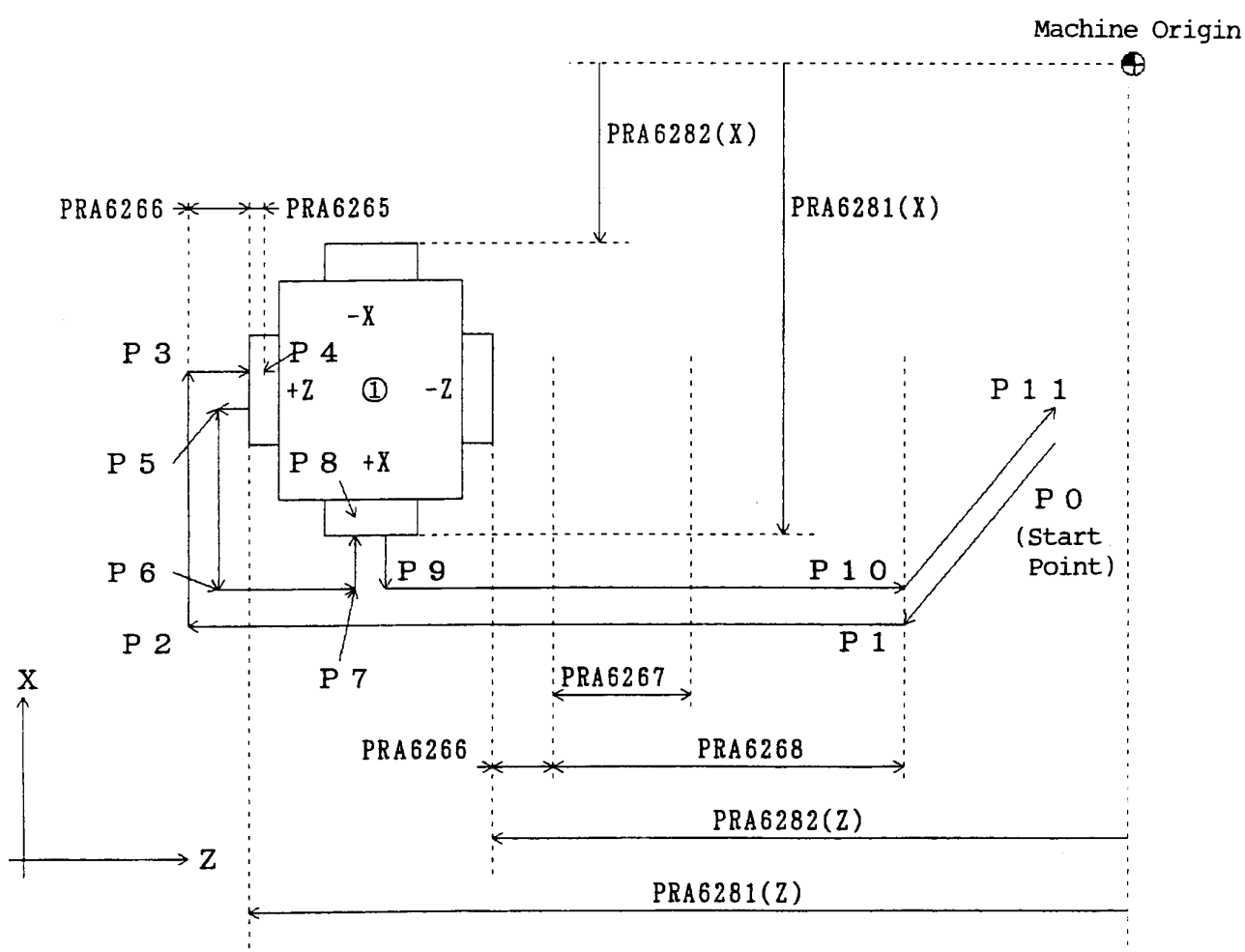
When Virtual Point 0, 5, 6, 7, 8, or 9 being assigned, X or Z axis alone is subject to measurement. In this case, for compensation of the axis not being measured, compensation needs to be input manually beforehand. As for virtual nose points of tools, see Drawing 1.



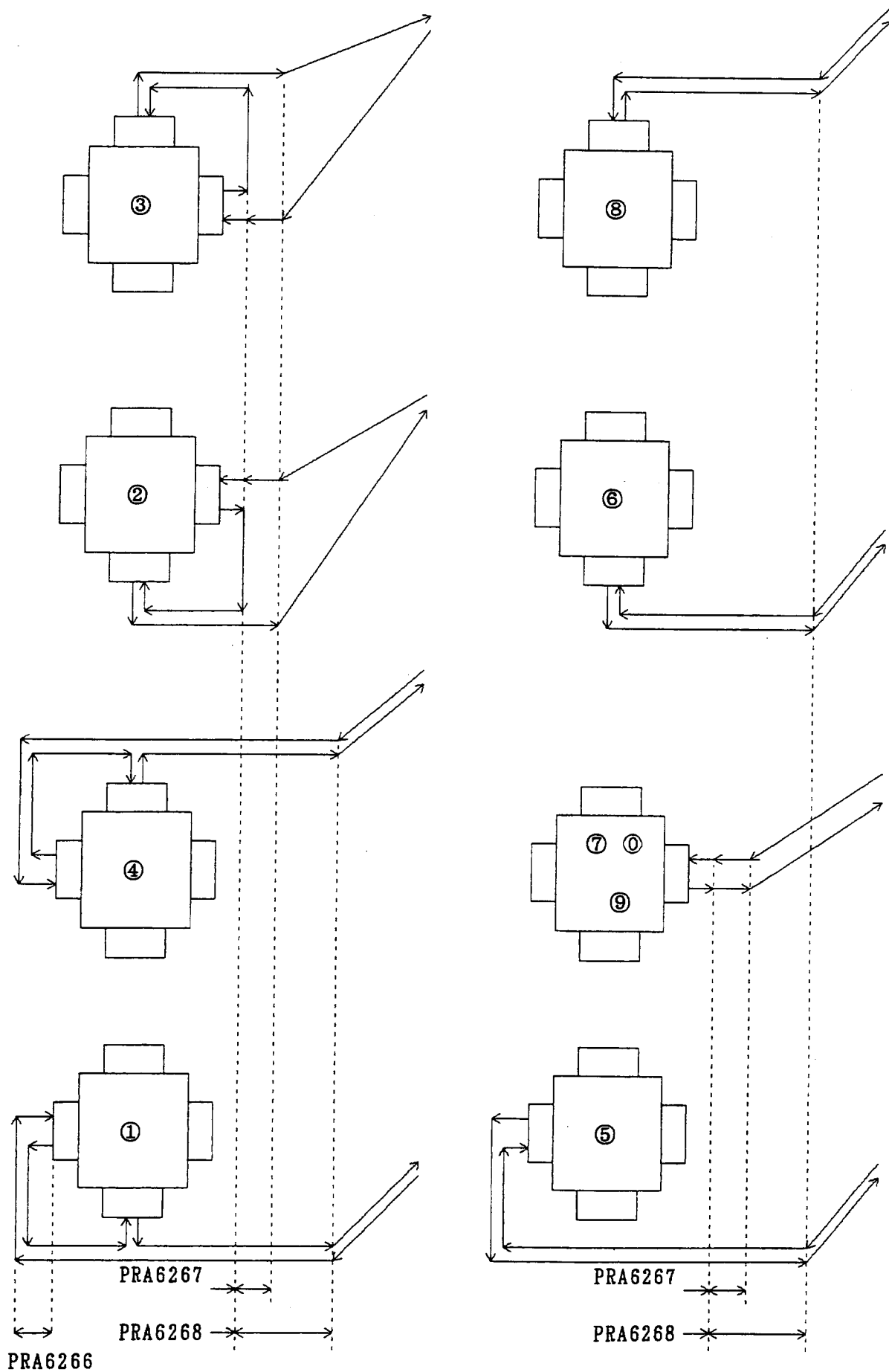
Drawing 1 Virtual Nose Points of Tools

The following example describes the case when the virtual nose point is 1: (See Drawing 2.)

- (1) Shifted in rapid traverse from Start Point (P0) to Point 1 (P1).
- (2) Shifted in rapid traverse from P1 to P2.
- (3) Shifted in rapid traverse from P2 to P3.
- (4) Shifted in jog feed speed from P3 to the end position (P4) to be touched.
- (5) Shifted in jog feed speed from the point where touching and stopping have taken place.
- (6) Shifted from P5 to P6 in rapid traverse.
- (7) Shifted from P6 to P7 in rapid traverse.
- (8) Shifted in jog feed speed from P7 to the end position (P8) to be touched.
- (9) Shifted in jog feed speed from the touch-and-stop position to P9.
- (10) Shifted from P9 to P10 in rapid traverse.
- (11) Shifted from P10 to the start point (P11) in rapid traverse.



Drawing 2 Working with Virtual Nose Point 1



Drawing 3 Working Patterns

## 7-4 Precaution

- (1) When a tool has been changed, be sure to manually apply it to Q setter.
- (2) Apply it to Q setter only once.
- (3) Approaching, in rapid traverse, from the sensor to the place just before the clearance amount on approach (PRA6266), touch the sensor in the jog feed rate.
- (4) Single block is held valid while in measurement operation.  
(Re-started with F4/REPEAT START.)
- (5) Machine lock, when having occurred in Q setter mode, produces an alarm.
- (6) When a compensation no. having been selected in Q Setter mode has been shifted with the cursor and the page key, measurement is carried out with the compensation no. data.
- (7) When the measurement start point can touch the sensor, alarm takes place.
- (8) When the sensor has not been touched in measurement, alarm takes place after operation is over.
- (9) Threading tools, tip changing type drills, end mills, and other tools which are similar to these cannot be subject to measurement.
- (10) The measurement start point is any one position free from danger or touching the sensor.
- (11) When, in some operation patterns, an interference exists with a work, remove the work from the chuck.
- (12) When a large difference exists between the Q setter measurement and the work measurement dimension, adjust Q setter position on Maintenance page.
- (13) When the machine is provided with the Y-axis, it must be in the following conditions when starting Q-setter repeat.
  - The Y-axis is at the origin. (The ORIGIN lamp is ON)
  - The tool offset amount for the Y-axis is 0. (Both shape and Wear are 0)
- (14) The measuring spots are deleted in the following cases:
  - When Data Delete on the screen is effectuated.
  - When shape data for tool compensation volume and nose T have been input.

## 7-5 Relevant Parameters

(Reference Value)

No.6244, #3	= 0 : Does not make a door close check. (0)	
	= 1 : Makes a door close check.	
No.6244, #4	= 0 : Does not operate with the tool nose radius compensation amount taken into account. (1)	
	= 1 : Operates with the tool nose radius compensation amount taken into account.	
No.6244, #5	= 0 : Cannot measure the rotary tool. (Alarm) (0)	
	= 1 : Can measure the rotary tool	
No.6265	Distance from touch position to end position for Q setter repeat	(1.500mm)
No.6266	Clearance amount on approaching for Q setter repeat	(3.000mm)
No.6267	Clearance amount for right-handed machine tool for Q setter repeat	(5.000mm)
No.6268	Clearance amount for left-handed machine tool for Q setter repeat	(50.000mm)

## 7-6 Relevant Alarms

No.715	Machine lock has been used in measurement.
No.724	No measurement has been performed in Q setter.
No.738	Touch signals of Q setter repeat is abnormal.
No.739	Measurement start point of Q setter repeat is not correct.
No.749	Cannot measure the rotary tool with Q-setter repeat.

## 8. Simple Jaw Edge Forming Function

### 8-1 Outline

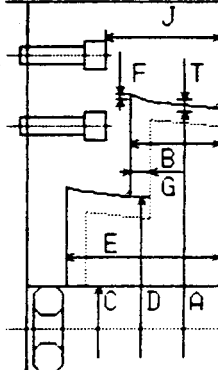
With “Edge shape” “Working Conditions” being input according to the guidance on the page and the start button being pushed, raw edge machining starts.

### 8-2 Operation

- (1) Index the tool used for raw edge forming to the machining position.
- (2) Pushing **F6/SOFT JAW** on the page of the work coordinate system (General → **F4/WORK COORDINATE** ), display Window page for raw edge forming.
- (3) Select either outer jaw or inner jaw forming by using **F4/OUTER JAW** or **F5/INNER JAW**.
- (4) Prepare the section where a core bar is fitted and mount it properly.
- (5) Input an edge configuration and working conditions.
- (6) Turning the spindle in Manual mode, shift the tool to the edge position.  
With **F3/JAW END SURFACE** pushed, “Jaw end surface position setting? Y-Yes/N-No” is displayed. With **Y** , set the jaw end position.
- (7) Pushing **F9/CUT PATH CHECK** in MDI mode, check the locus of a working program. If not in MDI mode, “Set MDI mode” message is displayed. While in locus checking, Dry-Run and Machine Lock ON state are held effective. The operation panel lamps, however, do not lit up. Also, no MST code is output.  
The coordinate system (tip position) is brought to presetting on completion of locus checking or by resetting while in checking.
- (8) Perform zero returning as paying attention not to cause interference.
- (9) Pushing the start button in MDI mode, execute a working program of raw edge forming.

00000  
N

# SOFT JAW FORMING (OUTER)



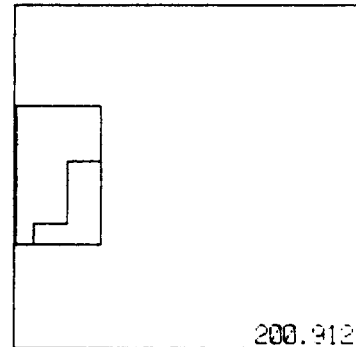
DIMENSION	
A	200.000
B	40.000
C	100.000
D	150.000
E	80.000
T	0.500
F	1.000
G	0.000
H	3.000
I	10.000
J	50.000
K	30.000

MACHINING	
CUTTING R.	60
SPEED F.	120
SPINDLE R.	6366
SPEED F.	12732
FEED R.	1.000
SPEED F.	0.500
CUT-IN R.	5.000
F.AMOUNT F.	2.000

CHUCK OUT	400.000
SOFT IDIA	50.000
HIGH	100.000

## DESCRIPTION

The inner diameter of 1st step.



TOOL NOSE POSITION	
X	198.964
Y	-0.013
Z	200.912

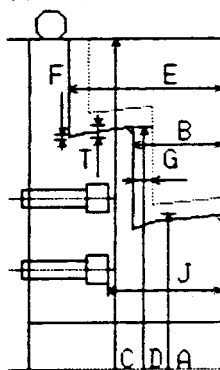
Jaw end surface Tool T0100.

A =

ZOOM IN GRAPH./1	SET UP /2	JAW END SURFAC/3	OUTER JAW /4	INNER JAW /5	EXIT /6	DATA CLEAR /7	RESET /8	CUT PATH CHECK /9	/0
---------------------	--------------	---------------------	-----------------	-----------------	------------	------------------	----------	----------------------	----

00000  
N

# SOFT JAW FORMING (INNER)



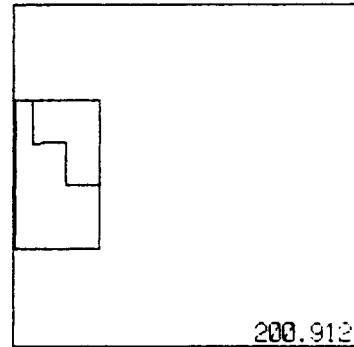
DIMENSION	
A	200.000
B	40.000
C	400.000
D	300.000
E	80.000
T	0.500
F	1.000
G	0.000
H	3.000
I	10.000
J	50.000
K	30.000

MACHINING	
CUTTING R.	60
SPEED F.	120
SPINDLE R.	6366
SPEED F.	12732
FEED R.	1.000
SPEED F.	0.500
CUT-IN R.	5.000
F.AMOUNT F.	2.000

CHUCK OUT	400.000
SOFT IDIA	50.000
HIGH	100.000

## DESCRIPTION

The outer diameter of 1st step.



TOOL NOSE POSITION	
X	198.964
Y	-0.013
Z	200.912

Jaw end surface Tool T0100.

A =

ZOOM IN GRAPH./1	SET UP /2	JAW END SURFAC/3	OUTER JAW /4	INNER JAW /5	EXIT /6	DATA CLEAR /7	RESET /8	CUT PATH CHECK /9	/0
---------------------	--------------	---------------------	-----------------	-----------------	------------	------------------	----------	----------------------	----

### 8-3 Function Keys

- F1/ZOOM IN GRAPH. : A cut part of an edge is displayed magnified.
- F2/SET UP : The Setup window appears and allows you to specify the spindle speed and turret indexing.
- F3/JAW END SURFAC : "Jaw End Surface Position Setting? Y-Yes/N-No" message is displayed. Set with  .  
The Z coordinate value of the (soft jaw) tool nose position is cleared to 0. (The Z coordinate value for the relative coordinate system.)
- F4/OUTER JAW : Outer Jaw Forming page is selected.
- F5/INNER JAW : Inner Jaw Forming page is selected.
- F6/EXIT : Work Coordinate System page is returned.
- F7/DATA CLEAR : "Edge Configuration and Working Condition Erased? Y-Yes/N-No" message appears. Select  to clear.
- F8/RESET : Soft jaw forming is ended. Push this to return to normal work. The jaw end surface setting position is cleared.
- F9/CUT PATH CHECK : A locus of a working program is drawn.

### 8-4 Jaw Configuration And Machining Conditions

#### (1) JAW CONFIGURATION

The symbols of jaw configurations indicate the following:

- A : Inner diameter for 1st stage
- B : Depth for 1st stage
- C : Core metal diameter
- D : Inner diameter for 2nd stage
- E : Depth for 2nd stage (With 0 setting, one-stage jaw is obtained.)
- T : Grip part taper amount
- F : Necking depth
- G : Max. Value of stock allowance
- H : Setting of chamfering amount
- I : Setting of necking width (clearance)
- J : Bolt position 1
- K : Jaw extrusion from chuck outer face is input in +/- . (Chuck outer diameter taken as reference)

#### (2) MACHINING CONDITIONS

- |                      |                                                                                                       |
|----------------------|-------------------------------------------------------------------------------------------------------|
| Cutting speed        | Rough: Cutting speed for rough cutting (m/min)                                                        |
| Cutting speed        | Finish: Cutting speed for finishing (m/min)                                                           |
| Number of revolution | Rough: Number of revolution for rough cutting (rpm)<br>(Automatically calculated from cutting speed.) |
| Number of revolution | Finish: Number or revolution for finishing (rpm)<br>(Automatically calculated from cutting speed.)    |
| Feed speed           | Rough: Feed speed for rough cutting (mm/rev)                                                          |
| Feed speed           | Finish: Feed speed for finishing (mm/rev)                                                             |

Cut depth	Rough: Depth of cut for rough cutting (mm) (With 0 setting, finishing alone is performed.)
Finishing Allowance	Finish: finishing allowance for finishing (mm) (With 0 setting, finishing alone is performed.)

## 8-5 Precaution

- (1) Before executing soft jaw forming, set a jaw end surface position. When locus checking or soft jaw forming is executed without a edge position being set, warning takes place.
- (2) Clamp max. spindle rpm is Soft Jaw Forming mode to the parameter set value (No. 5160).
- (3) When E value is 0, it gets 1-stage edge.
- (4) When a locus is not clearly displayed, reduce speed with the feed speed switch.
- (5) In order to perform necking, pay attention to the tip shape.
- (6) When the machine is provided with the Y-axis, it must be in the following conditions when starting soft jaw forming.
  - The Y-axis is at the origin. (The ORIGIN lamp is ON)
  - The Y-axis tool offset amount for the Y-axis is 0. (Both shape and Wear are 0)

## 8-6 Relevant Parameters

	(Reference Values)
No.5130 Feed rate magnification in unload cutting of soft jaw forming	
No.5160 Max. spindle rpm of soft jaw forming	(1000rpm)
No.5155 Approach amount on finishing in soft jaw forming	(10.000mm)
No.5156 Soft jaw forming clearance	(2.000mm)

## 8-7 Relevant Alarms

No.270 Soft jaw error

The following display appears:

[270] (#001) Soft jaw error

↑

Numerals following “#” tells details of the alarm as follows:



#001	A $\leq$ 0 (A: 1st stage inner dia./outer dia.)
#002	B $\leq$ 0 (B: 1st stage depth)
#003	C $\leq$ 0 (C: Core bar dia.)
#004	D $\leq$ 0 (D: 2nd stage inner dia./outer dia.)
#005	E $\leq$ 0 (E: 2nd stage depth)
#006	T < 0 (T: Grip part taper amount)
#007	F < 0 (F: Necking depth)
#008	G < 0 (G: Max. value of stock allowance)
#009	H < 0 (H: Chamfering amount)
#00A	I < 0 (I: Necking width)
#010	C $\geq$ A (Core bar dia. $\geq$ 1st stage inner dia.)
#011	C $\leq$ A (Core bar dia. $\leq$ 1st stage outer dia.)
#012	D $\geq$ A (2nd stage $\geq$ 1st stage inner dia.)
#013	D $\leq$ A (2nd stage $\leq$ 1st stage outer dia.)
#014	C $\geq$ D (Core bar dia. $\geq$ 2nd stage inner dia.)
#015	C $\leq$ D (Core bar dia. $\leq$ 2nd stage outer dia.)
#016	B $\geq$ E (1st stage $\geq$ 2nd stage depth)
#017	Chamfering is too large.
#018	Necking width is too large.
#019	Bolt interference has occurred.
#020	cutting rate (rough) $\leq$ 0
#021	cutting rate (finish) $\leq$ 0
#022	Feed rate (rough) $\leq$ 0
#023	Feed rate (finish) $\leq$ 0
#024	Depth of cut
#030	Outer 1-stage edge configuration 1
#031	Outer 1-stage edge configuration 2
#032	Outer 1-stage edge configuration 3
#040	Inner 1-stage edge configuration 1
#041	Inner 1-stage edge configuration 2
#042	Inner 1-stage edge configuration 3
#050	Outer 2-stage edge configuration 1
#051	Outer 2-stage edge configuration 2
#052	Outer 2-stage edge configuration 3
#053	Outer 2-stage edge configuration 4
#054	Outer 2-stage edge configuration 5
#060	Inner 2-stage edge configuration 1
#061	Inner 2-stage edge configuration 2
#062	Inner 2-stage edge configuration 3
#063	Inner 2-stage edge configuration 4
#064	Inner 2-stage edge configuration 5
#070	Combined undercut error
#071	Combined cut depth error
#072	Combined above-cut error
#073	Error in increase to combined peak
#074	A block exists where no calculation is available when nose R is applied to configuration data.
#0F0	Configuration can not be specified.
#0F1	Intersecting point calculation has entered infinite loop.
#0FF	Soft Jaw Forming page has not been displayed.

No.729 Jaw end surface position setting not completed (Warning)

## 8-8 Set Values for Standard Soft Jaws

### 8-8-1 Metric Specifications

- (1) TS15 ..... Model A  
 NR20 Ⅲ , HT20R Ⅲ , CA20 Ⅱ .....Model B  
 NR23 Ⅲ , HT23R Ⅲ , TG25, HT25G, TF25, CA23 Ⅱ .....Model C

Model			Model A		Model B		Model C	
Item			Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw
Jaw Configuration	A		80.0 mm	80.0 mm	130.0 mm	130.0 mm	150.0 mm	150.0 mm
	B		10.0 mm	10.0 mm	10.0 mm	10.0 mm	10.0 mm	10.0 mm
	C		15.0 mm	165.0 mm	52.0 mm	210.0 mm	48.0 mm	254.0 mm
	D		50.0 mm	160.0 mm	90.0 mm	205.0 mm	100.0 mm	240.0 mm
	E		20.0 mm	20.0 mm	20.0 mm	20.0 mm	20.0 mm	20.0 mm
	T		0.03 mm	0.03 mm	0.03 mm	0.03 mm	0.03 mm	0.03 mm
	F		0.5 mm	0.5 mm	0.5 mm	0.5 mm	0.5 mm	0.5 mm
	G		0.0 mm	0.0 mm	0.0 mm	0.0 mm	0.0 mm	0.0 mm
	H		1.0 mm	1.0 mm	1.0 mm	1.0 mm	1.0 mm	1.0 mm
	I		1.5 mm	1.5 mm	1.5 mm	1.5 mm	1.5 mm	1.5 mm
	J		13.0 mm	13.0 mm	12.0 mm	12.0 mm	18.0 mm	18.0 mm
	K							
Working conditions	Cutting Speed	Rough	100m/min	100m/min	100m/min	100m/min	100m/min	100m/min
		Finish	120m/min	120m/min	120m/min	120m/min	120m/min	120m/min
	RPM	Rough	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.
		Finish	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.
	Feed Speed	Rough	0.15mm/rev	0.15mm/rev	0.15mm/rev	0.15mm/rev	0.15mm/rev	0.15mm/rev
Soft Jaw		Finish	0.10mm/rev	0.10mm/rev	0.10mm/rev	0.10mm/rev	0.10mm/rev	0.10mm/rev
	Depth of Cut	Rough	2.0 mm	2.0 mm	2.0 mm	2.0 mm	2.0 mm	2.0 mm
		Finish	0.2 mm	0.2 mm	0.2 mm	0.2 mm	0.2 mm	0.2 mm
	Stock Allowance							
Soft Jaw	Outer Dia.		165.0 mm	165.0 mm	210.0 mm	210.0 mm	254.0 mm	254.0 mm
	Inner Dia.		15.0 mm	15.0 mm	52.0 mm	52.0 mm	48.0 mm	48.0 mm
	Height		30.0 mm	30.0 mm	40.0 mm	40.0 mm	48.0 mm	48.0 mm
Jaw Bolt Hole Pitch PRM5163			25.0 mm		25.0 mm		32.0 mm	
Bolt Center Pos. from Jaw Surface PRM5164			12.0 mm		15.0 mm		21.0 mm	
Bolt Spot Facing Dia. PRM5165			14.5 mm		19.0 mm		20.0 mm	

- (2) TG30, HT30G, CA30 **Ⅱ** .....Model D  
 TG40, HT40G, CA40 **Ⅱ** .....Model E  
 TG50, HT50G ..... Model F

Model Item			Model D		Model E		Model F	
			Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw
Jaw Configuration		A	160.0 mm	160.0 mm	220.0 mm	220.0 mm	300.0 mm	300.0 mm
		B	15.0 mm	15.0 mm	15.0 mm	15.0 mm	15.0 mm	15.0 mm
		C	25.0 mm	305.0 mm	81.0 mm	381.0 mm	157.0 mm	457.0 mm
		D	100.0 mm	260.0 mm	150.0 mm	360.0 mm	200.0 mm	440.0 mm
		E	30.0 mm	30.0 mm	30.0 mm	30.0 mm	30.0 mm	30.0 mm
		T	0.03 mm	0.03 mm	0.03 mm	0.03 mm	0.03 mm	0.03 mm
		F	0.5 mm	0.5 mm	0.5 mm	0.5 mm	0.5 mm	0.5 mm
		G	0.0 mm	0.0 mm	0.0 mm	0.0 mm	0.0 mm	0.0 mm
		H	1.0 mm	1.0 mm	1.0 mm	1.0 mm	1.0 mm	1.0 mm
		I	1.5 mm	1.5 mm	1.5 mm	1.5 mm	1.5 mm	1.5 mm
		J	18.0 mm	18.0 mm	22.0 mm	22.0 mm	22.0 mm	22.0 mm
		K						
Working conditions	Cutting Speed		Rough Finish	100m/min 120m/min	100m/min 120m/min	100m/min 120m/min	100m/min 120m/min	100m/min 120m/min
	RPM		Rough  Finish	Automatically Calculated.  Automatically Calculated.	Automatically Calculated.  Automatically Calculated.	Automatically Calculated.  Automatically Calculated.	Automatically Calculated.  Automatically Calculated.	Automatically Calculated.  Automatically Calculated.
	Feed Speed		Rough Finish	0.15mm/rev 0.10mm/rev	0.15mm/rev 0.10mm/rev	0.15mm/rev 0.10mm/rev	0.15mm/rev 0.10mm/rev	0.15mm/rev 0.10mm/rev
	Depth of Cut		Rough	2.0 mm	2.0 mm	2.0 mm	2.0 mm	2.0 mm
	Stock Allowance		Finish	0.2 mm	0.2 mm	0.2 mm	0.2 mm	0.2 mm
	Soft Jaw		Outer Dia. Inner Dia. Height	305.0 mm 25.0 mm 60.0 mm	305.0 mm 25.0 mm 60.0 mm	381.0 mm 81.0 mm 61.0 mm	381.0 mm 81.0 mm 61.0 mm	457.0 mm 157.0 mm 61.0 mm
Jaw Bolt Hole Pitch PRM5163			32.0 mm		50.0 mm		50.0 mm	
Bolt Center Pos. from Jaw Surface PRM5164			40.0 mm		30.0 mm		30.0 mm	
Bolt Spot Facing Dia. PRM5165			23.5 mm		32.0 mm		32.0 mm	

(3) CS20 ..... Model G

Model Item			Model G		Model		Model	
			Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw
Jaw Configuration		A	130.0 mm	130.0 mm	mm	mm	mm	mm
		B	10.0 mm	10.0 mm	mm	mm	mm	mm
		C	21.0 mm	210.0 mm	mm	mm	mm	mm
		D	90.0 mm	205.0 mm	mm	mm	mm	mm
		E	20.0 mm	20.0 mm	mm	mm	mm	mm
		T	0.03 mm	0.03 mm	mm	mm	mm	mm
		F	0.5 mm	0.5 mm	mm	mm	mm	mm
		G	0.0 mm	0.0 mm	mm	mm	mm	mm
		H	1.0 mm	1.0 mm	mm	mm	mm	mm
		I	1.5 mm	1.5 mm	mm	mm	mm	mm
		J	11.0 mm	11.0 mm	mm	mm	mm	mm
		K						
Working conditions	Cutting Speed		Rough Finish	100m/min 120m/min	100m/min 120m/min	m/min m/min	m/min m/min	m/min m/min
	RPM		Rough  Finish	Automatically Calculated. Automatically Calculated.	Automatically Calculated. Automatically Calculated.			
	Feed Speed		Rough Finish	0.15mm/rev 0.10mm/rev	0.15mm/rev 0.10mm/rev	mm/rev mm/rev	mm/rev mm/rev	mm/rev mm/rev
	Depth of Cut		Rough	2.0 mm	2.0 mm	mm	mm	mm
	Stock Allowance		Finish	0.2 mm	0.2 mm	mm	mm	mm
	Soft Jaw		Outer Dia.	210.0 mm	210.0 mm	mm	mm	mm
Inner Dia.			21.0 mm	21.0 mm	mm	mm	mm	
Height			38.0 mm	38.0 mm	mm	mm	mm	
Jaw Bolt Hole Pitch PRM5163			25.0 mm		mm		mm	
Bolt Center Pos. from Jaw Surface PRM5164			24.0 mm		mm		mm	
Bolt Spot Facing Dia. PRM5165			19.0 mm		mm		mm	

## 8-8-2 Inch Specifications

- (1) S15 ..... Model A  
 HT20R III , CA20 I , CS20 ..... Model B  
 HT23R III , HT25G, TF25, CA23 I ..... Model A

Model Item			Model A		Model B		Model C	
			Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw
Jaw Configuration	A		3.00 inch	3.00 inch	5.00 inch	5.00 inch	6.00 inch	6.00 inch
	B		0.40 inch	0.40 inch	0.40 inch	0.40 inch	0.40 inch	0.40 inch
	C		0.60 inch	0.60 inch	2.00 inch	8.25 inch	1.90 inch	10.00 inch
	D		2.00 inch	2.00 inch	3.50 inch	8.00 inch	4.00 inch	9.50 inch
	E		0.80 inch	0.80 inch	0.80 inch	0.80 inch	0.80 inch	0.80 inch
	T		0.001 inch	0.001 inch	0.001 inch	0.001 inch	0.001 inch	0.001 inch
	F		0.02 inch	0.02 inch	0.02 inch	0.02 inch	0.02 inch	0.02 inch
	G		0.00 inch	0.00 inch	0.00 inch	0.00 inch	0.00 inch	0.00 inch
	H		0.04 inch	0.04 inch	0.04 inch	0.04 inch	0.04 inch	0.04 inch
	I		0.06 inch	0.06 inch	0.06 inch	0.06 inch	0.06 inch	0.06 inch
	J		0.51 inch	0.51 inch	0.47 inch	0.47 inch	0.71 inch	0.71 inch
	K							
Working conditions	Cutting Speed	Rough	300ft/min	300ft/min	300ft/min	300ft/min	300ft/min	300ft/min
		Finish	400ft/min	400ft/min	400ft/min	400ft/min	400ft/min	400ft/min
	RPM	Rough	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.
		Finish	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.
	Feed Speed	Rough	0.006in/rev	0.006in/rev	0.006in/rev	0.006in/rev	0.006in/rev	0.006in/rev
Soft Jaw		Finish	0.004in/rev	0.004in/rev	0.004in/rev	0.004in/rev	0.004in/rev	0.004in/rev
	Depth of Cut	Rough	0.08 inch	0.08 inch	0.08 inch	0.08 inch	0.08 inch	0.08 inch
		Finish	0.008 inch	0.008 inch	0.008 inch	0.008 inch	0.008 inch	0.008 inch
	Stock Allowance	Finish	0.008 inch	0.008 inch	0.008 inch	0.008 inch	0.008 inch	0.008 inch
Jaw Bolt Hole Pitch PRM5163	Outer Dia.		6.50 inch	6.50 inch	8.25 inch	8.25 inch	10.00 inch	10.00 inch
	Inner Dia.		0.60 inch	0.60 inch	2.00 inch	2.00 inch	1.90 inch	1.90 inch
	Height		1.20 inch	1.20 inch	1.60 inch	1.60 inch	1.90 inch	1.90 inch
Bolt Center Pos. from Jaw Surface PRM5164			1.00 inch		1.00 inch		1.26 inch	
Bolt Spot Facing Dia. PRM5165			0.47 inch		0.60 inch		0.82 inch	
			0.57 inch		0.75 inch		0.80 inch	

- (2) HT30G, CA30 II .....Model D  
HT40G, CA40 II .....Model E  
HT50G ..... Model D

Model Item			Model D		Model E		Model F	
			Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw
Jaw Configuration		A	6.30 inch	6.30 inch	8.70 inch	8.70 inch	11.80 inch	11.80 inch
		B	0.60 inch	0.60 inch	0.60 inch	0.60 inch	0.60 inch	0.60 inch
		C	1.00 inch	12.00 inch	3.20 inch	15.00 inch	6.20 inch	18.00 inch
		D	4.00 inch	10.20 inch	6.00 inch	14.20 inch	7.90 inch	17.30 inch
		E	1.20 inch	1.20 inch	1.20 inch	1.20 inch	1.20 inch	1.20 inch
		T	0.001 inch	0.001 inch	0.001 inch	0.001 inch	0.001 inch	0.001 inch
		F	0.02 inch	0.02 inch	0.02 inch	0.02 inch	0.02 inch	0.02 inch
		G	0.00 inch	0.00 inch	0.00 inch	0.00 inch	0.00 inch	0.00 inch
		H	0.04 inch	0.04 inch	0.04 inch	0.04 inch	0.04 inch	0.04 inch
		I	0.06 inch	0.06 inch	0.06 inch	0.06 inch	0.06 inch	0.06 inch
		J	0.71 inch	0.71 inch	0.87 inch	0.87 inch	0.87 inch	0.87 inch
		K						
Working conditions	Cutting Speed		Rough Finish	300ft/min 400ft/min	300ft/min 400ft/min	300ft/min 400ft/min	300ft/min 400ft/min	300ft/min 400ft/min
	RPM		Rough	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.
			Finish	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.
	Feed Speed		Rough Finish	0.006in/rev 0.004in/rev	0.006in/rev 0.004in/rev	0.006in/rev 0.004in/rev	0.006in/rev 0.004in/rev	0.006in/rev 0.004in/rev
	Depth of Cut		Rough	0.08 inch	0.08 inch	0.08 inch	0.08 inch	0.08 inch
	Stock Allowance		Finish	0.008 inch	0.008 inch	0.008 inch	0.008 inch	0.008 inch
Soft Jaw		Outer Dia.	12.00 inch	12.00 inch	15.00 inch	15.00 inch	18.00 inch	18.00 inch
		Inner Dia.	1.00 inch	1.00 inch	3.20 inch	3.20 inch	6.20 inch	6.20 inch
		Height	2.35 inch	2.35 inch	2.40 inch	2.40 inch	2.40 inch	2.40 inch
Jaw Bolt Hole Pitch PRM5163			1.26 inch		2.00 inch		2.00 inch	
Bolt Center Pos. from Jaw Surface PRM5164			1.60 inch		1.20 inch		1.20 inch	
Bolt Spot Facing Dia. PRM5165			0.92 inch		1.26 inch		1.26 inch	

## (3) CS20 ..... Model G

Model Item			Model G		Model		Model	
			Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw	Outer Jaw	Inner Jaw
Jaw Configuration	A		5.00 inch	5.00 inch	inch	inch	inch	inch
	B		0.40 inch	0.40 inch	inch	inch	inch	inch
	C		2.00 inch	8.25 inch	inch	inch	inch	inch
	D		3.50 inch	8.00 inch	inch	inch	inch	inch
	E		0.80 inch	0.80 inch	inch	inch	inch	inch
	T		0.001 inch	0.001 inch	inch	inch	inch	inch
	F		0.02 inch	0.02 inch	inch	inch	inch	inch
	G		0.00 inch	0.00 inch	inch	inch	inch	inch
	H		0.04 inch	0.04 inch	inch	inch	inch	inch
	I		0.06 inch	0.06 inch	inch	inch	inch	inch
	J		0.43 inch	0.43 inch	inch	inch	inch	inch
	K							
Working conditions	Cutting Speed	Rough Finish	300ft/min 400ft/min	300ft/min 400ft/min	ft/min ft/min	ft/min ft/min	ft/min ft/min	ft/min ft/min
	RPM	Rough	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.
		Finish	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.	Automatically Calculated.
	Feed Speed	Rough Finish	0.006in/rev 0.004in/rev	0.006in/rev 0.004in/rev	in/rev in/rev	in/rev in/rev	in/rev in/rev	in/rev in/rev
	Depth of Cut	Rough	0.08 inch	0.08 inch	inch	inch	inch	inch
	Stock Allowance	Finish	0.008 inch	0.008 inch	inch	inch	inch	inch
	Soft Jaw	Outer Dia. Inner Dia. Height	8.25 inch 0.80 inch 1.50 inch	8.25 inch 0.80 inch 1.50 inch	inch inch inch	inch inch inch	inch inch inch	inch inch inch
Jaw Bolt Hole Pitch PRM5163			1.00 inch		inch		inch	
Bolt Center Pos. from Jaw Surface PRM5164			0.95 inch		inch		inch	
Bolt Spot Facing Dia. PRM5165			0.75 inch		inch		inch	

## 8-9 Operation Of Simple Soft Jaw Forming Function

### (1) OUTER JAW

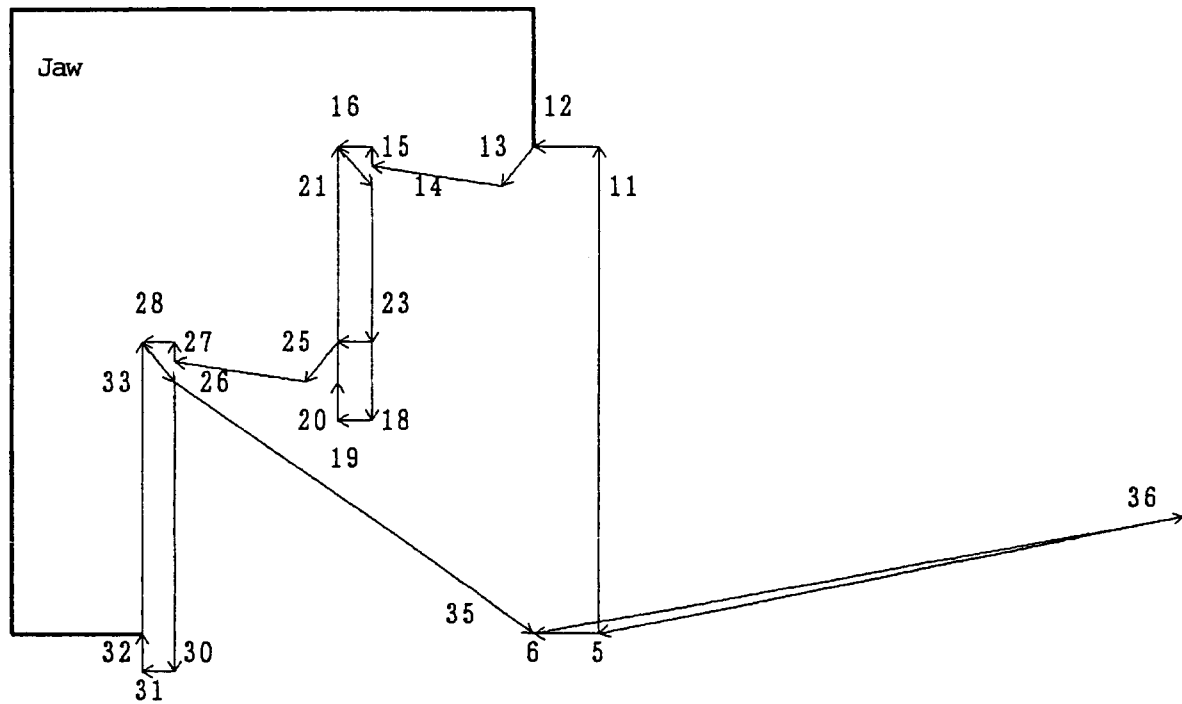
Process	Command	Description
1	G99 Fxx	Every revolution, feed speed for roughing
2	G96 Sxx	Constant surface speed, cutting speed for roughing
3	G50 Sxx	Spindle rpm clamp, soft jaw forming spindle max. rpm
4	M03	Spindle forward rotation
5	G00 Xxx Zxx	Approach to the point immediately before jaw end surface (parameter No.5155)
6	G01 Zxx	Machining start position (Jaw end surface)
7	See the attached table.	Soft jaw forming rough cutting
8	G00 Xxx Zxx	Approach to the point immediately before jaw end surface (parameter No.5155)
9	G99 Fxx	Every revolution, feed speed for finishing
10	G96 Sxx	Constant surface speed, cutting speed for finishing
11	G01 Xxx Fxx	X start point of 1st stage chamfering, unload cutting feed speed
12	G99 G01 Zxx Fxx	Z start point of 1st stage chamfering, finishing feed speed
13	G01 Xxx Zxx	1st stage chamfering end point
14	G01 Xxx Zxx	1st stage necking start point
15	G96 G01 Xxx Sxx	1st stage necking X cut depth, rough cutting speed
16	G01 Zxx	1st stage necking Z cut depth
17	G96 G01 Xxx Sxx	Clearance (Parameter No.5156), finishing cutting speed
18	G01 Xxx Fxx	X (1st stage inner dia. – Parameter No.5156), unload cutting feed speed cutting
19	G01 Zxx	1st stage necking Z end point
20	G99 G01 Xxx Fxx	X (1st stage inner dia.), finishing feed speed
21	G01 Xxx	1st stage necking X cut depth
22	G96 G01 Xxx Zxx Sxx	Clearance (Parameter No.5156), finishing cutting speed
23	G01 Xxx Fxx	2nd stage chamfering X start point, unload cutting feed speed
24	G99 G01 Zxx Fxx	2nd stage chamfering Z start point, finishing feed speed
25	G01 Xxx Zxx	2nd stage chamfering end point
26	G01 Xxx Zxx	2nd stage necking start point
27	G96 G01 Xxx Sxx	2nd stage necking X cut depth, rough cutting speed
28	G01 Zxx	2nd stage necking Z cut depth
29	G96 G01 Xxx Zxx Sxx	Clearance (Parameter No.5156), finishing cutting speed
30	G01 Xxx Fxx	X (2nd stage inner dia. – Parameter No.5156), unload cutting feed speed
31	G01 Zxx	2nd stage necking Z end point
32	G99 G01 Xxx Fxx	X (2nd stage inner dia.), finishing feed speed
33	G01 Xxx	2nd stage necking X cut depth
34	G96 G01 Xxx Zxx Sxx	Clearance (Parameter No.5156), finishing cutting speed
35	G00 Xxx Zxx	Machining start position (jaw end surface)
36	G00 Xxx Zxx	Starting point
37	M05	Spindle stop
38	G97	Constant surface speed cancelled
39	M30	Soft jaw forming ended



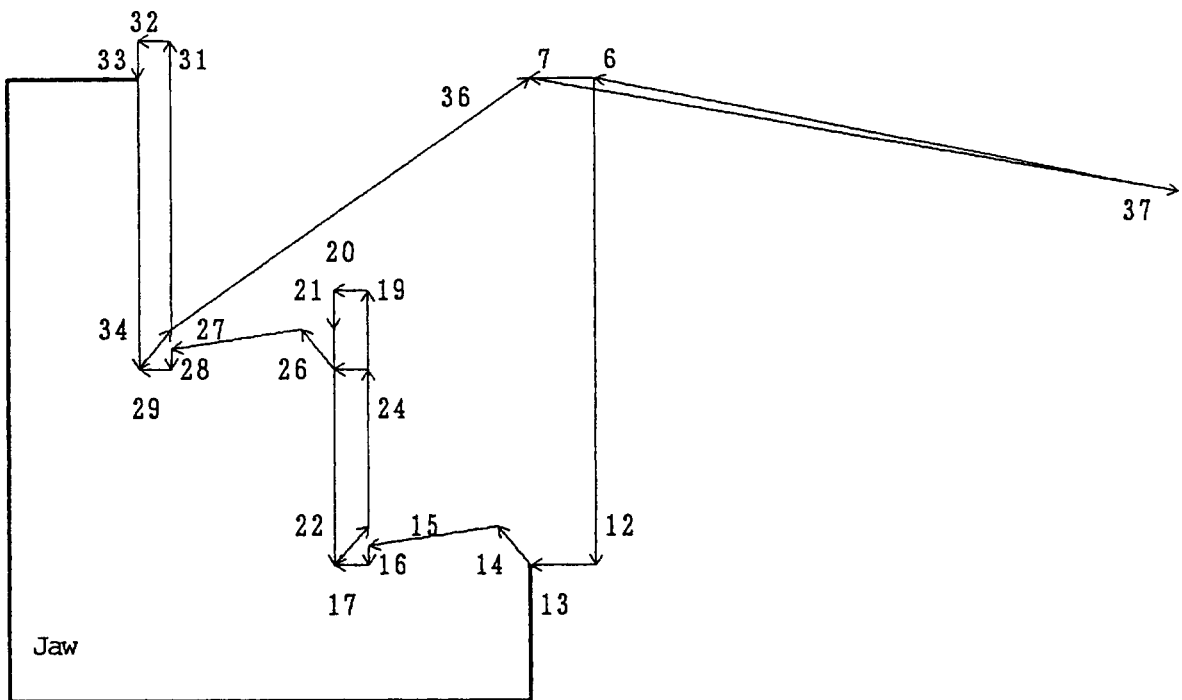
## (2) INNER JAW

Process	Command	Description
1	G99 Fxx	Every revolution, feed speed for roughing
2	G96 Sxx	Constant surface speed, cutting speed for roughing
3	G50 Sxx	Spindle rpm clamped, soft jaw forming spindle max. rpm
4	M03	Spindle forward rotation
5	G23	Stroke limit check 2/3 OFF
6	G00 Xxx Zxx	Approach to the point immediately before jaw end surface (parameter No.5155)
7	G01 Zxx	Machining start position (Jaw end surface)
8	See the attached table.	Soft jaw forming rough cutting
9	G00 Xxx Zxx	Approach to the point immediately before jaw end surface (parameter No.5155)
10	G99 Fxx	Every revolution, finishing feed speed
11	G96 Sxx	Constant surface speed, finishing cutting speed
12	G01 Xxx Fxx	1st stage chamfering X start point, unload cutting feed speed
13	G99 G01 Zxx Fxx	1st stage chamfering Z start point, finishing feed speed
14	G01 Xxx Zxx	1st stage chamfering end point
15	G01 Xxx Zxx	1st stage necking start point
16	G96 G01 Xxx Zxx	1st stage necking X cut depth, rough cutting speed
17	G01 Zxx	1st stage necking Z cut depth
18	G96 G01 Xxx Zxx Sxx	Clearance (Parameter No.5156), finishing cutting speed
19	G01 Xxx Fxx	X (1st stage inner dia. – Parameter No.5156), unload cutting feed speed cutting
20	G01 Zxx	1st stage necking Z end point
21	G99 G01 Xxx Fxx	X (1st stage inner dia.), finishing feed speed
22	G01 Xxx	1st stage necking X cut depth
23	G96 G01 Xxx Zxx Sxx	Clearance (Parameter No.5156), finishing cutting speed
24	G01 Xxx Fxx	2nd stage chamfering X start point, unload cutting feed speed
25	G99 G01 Zxx Fxx	2nd stage chamfering Z start point, finishing feed speed
26	G01 Xxx Zxx	2nd stage chamfering end point
27	G01 Xxx Zxx	2nd stage necking start point
28	G96 G01 Xxx Sxx	2nd stage necking X cut depth, rough cutting speed
29	G01 Zxx	2nd stage necking Z cut depth
30	G96 G01 Xxx Zxx Sxx	Clearance (Parameter No.5156), finishing cutting speed
31	G01 Xxx Fxx	X (2nd stage inner dia. + Parameter No.5156), unload cutting feedspeed
32	G01 Zxx	2nd stage necking Z end point
33	G99 G01 Xxx Fxx	X (2nd stage inner dia.), finishing feed speed
34	G01 Xxx	2nd stage necking X cut depth
35	G96 G01 Xxx Zxx Sxx	Clearance (Parameter No.5156), finishing cutting speed
36	G00 Xxx Zxx	Machining start position (jaw end surface)
37	G00 Xxx Zxx	Start point
38	G22	Stroke limit check 2/3 ON
39	M05	Spindle stop
40	G97	Constant surface speed cancelled
41	M30	Soft jaw forming ended

(3) PERFORMANCE CHART (OUTER JAW)

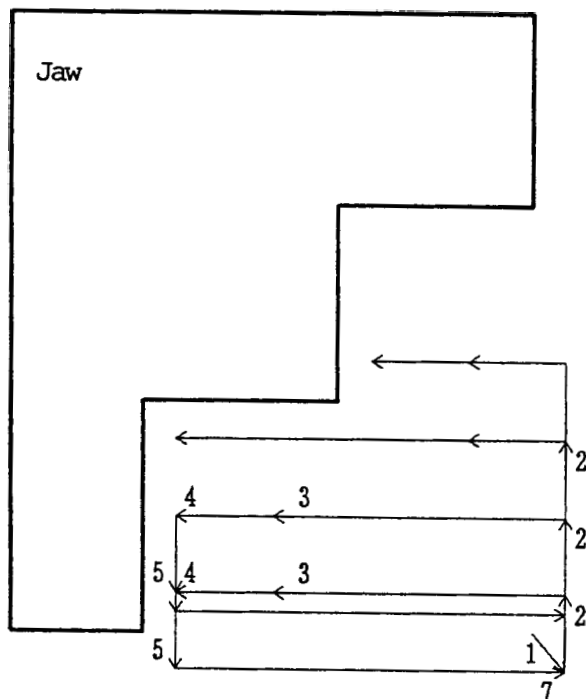


(4) PERFORMANCE CHART (INNER JAW)



## (5) ROUGH CUTTING OF SOFT JAW FORMING

Process	Command	Description
1	G00 Xxx Zxx	X is shifted by the amount of finishing allowance and Z by 3mm.
2	G01 Xxx	Cut depth in X direction is increased by the amount of one time.
3	G01 Zxx Fxx	Feeding takes place to the point 3mm before the stroke allowance. (feed speed for unloaded cutting)
4	G01 Zxx Fxx	Roughing in Z direction (feed speed for roughing)
5	G01 Xxx	Returning takes place in X direction by the amount of one time of cut depth.
6	G01 Xxx	Shifting takes place in X direction. (Parameter No.5156)
7	G00 Zxx	Returning takes place to the point which is 3mm from the jaw end.
8	G01 Xxx Fxx	X is shifted to the cutting place of the preceding time. (feed speed for unloaded cutting)
9	G01 Xxx	To the point which has been shifted by the amount of the stock allowance from the starting position.
10	G00 Zxx	To Z start position
11	G00 Xxx	To X start position



When the handle is not provided with a taper, the path becomes as above. When the handle is provided with taper or in order to know details of the path, refer to G71 Type 2 of Combined Canned Cycle of “Program” version of the instruction manual.

## **II. Screen Operation**

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

Note) There are two types of display screen – color and monochrome. If color is specified, the item in the brackets “( )” after the color name, shows the condition during monochrome display, as color (monochrome).

## 1. Screen Layout and Basic Operation

One screen fundamentally consists of the following (See Fig. 1-0):

- ① Main display area
- ② Alarm display area
- ③ Program No. display area (No. O, No. N)
- ④ Key input area
- ⑤ Error message display area
- ⑥ Function menu display area

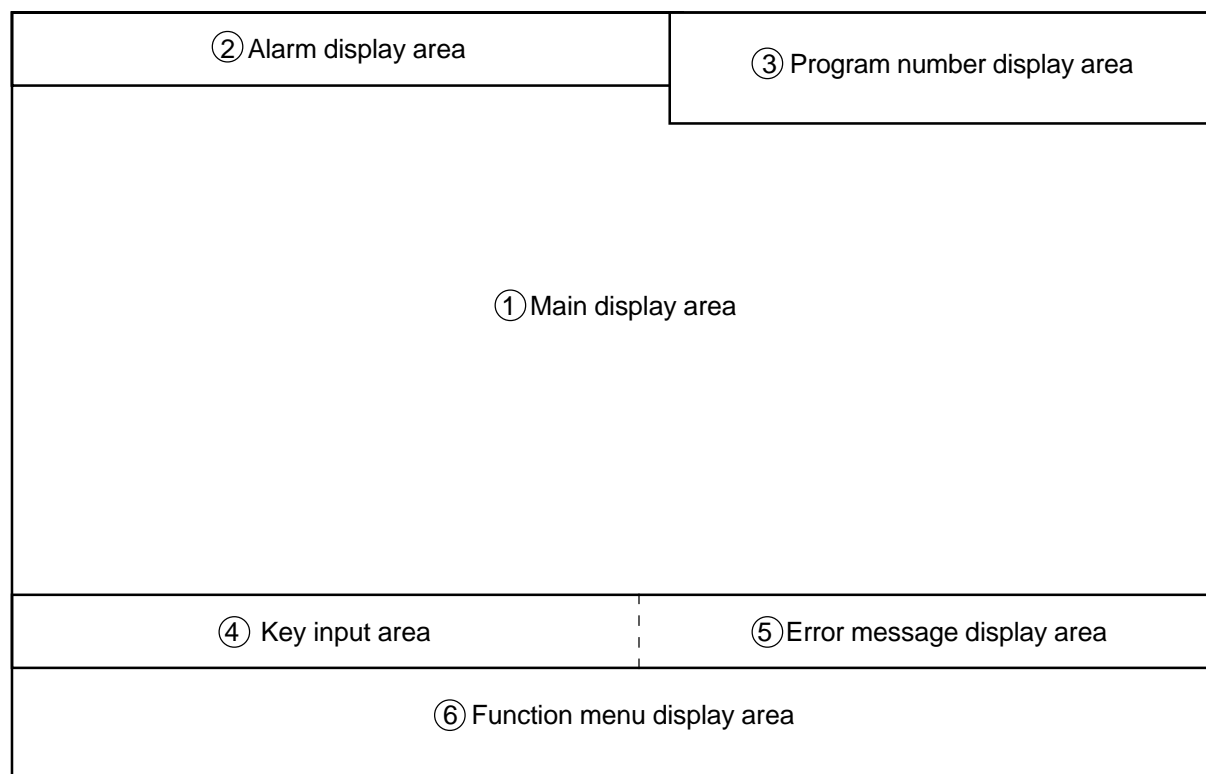




Fig. 1-0 Configuration of basic screen











- 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 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 +   
Moves the page and cursor to the beginning of that screen.
- SHIFT +   
Moves the page and cursor to the end of that screen.
- SHIFT +  / SHIFT +   
Moves the cursor character by character in editing the program. This allows you to edit 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. For more details, refer to the alarm diagnostic 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 Maintenance screen is forced to appear. (When you do not want 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 0000 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 nothing 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). 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

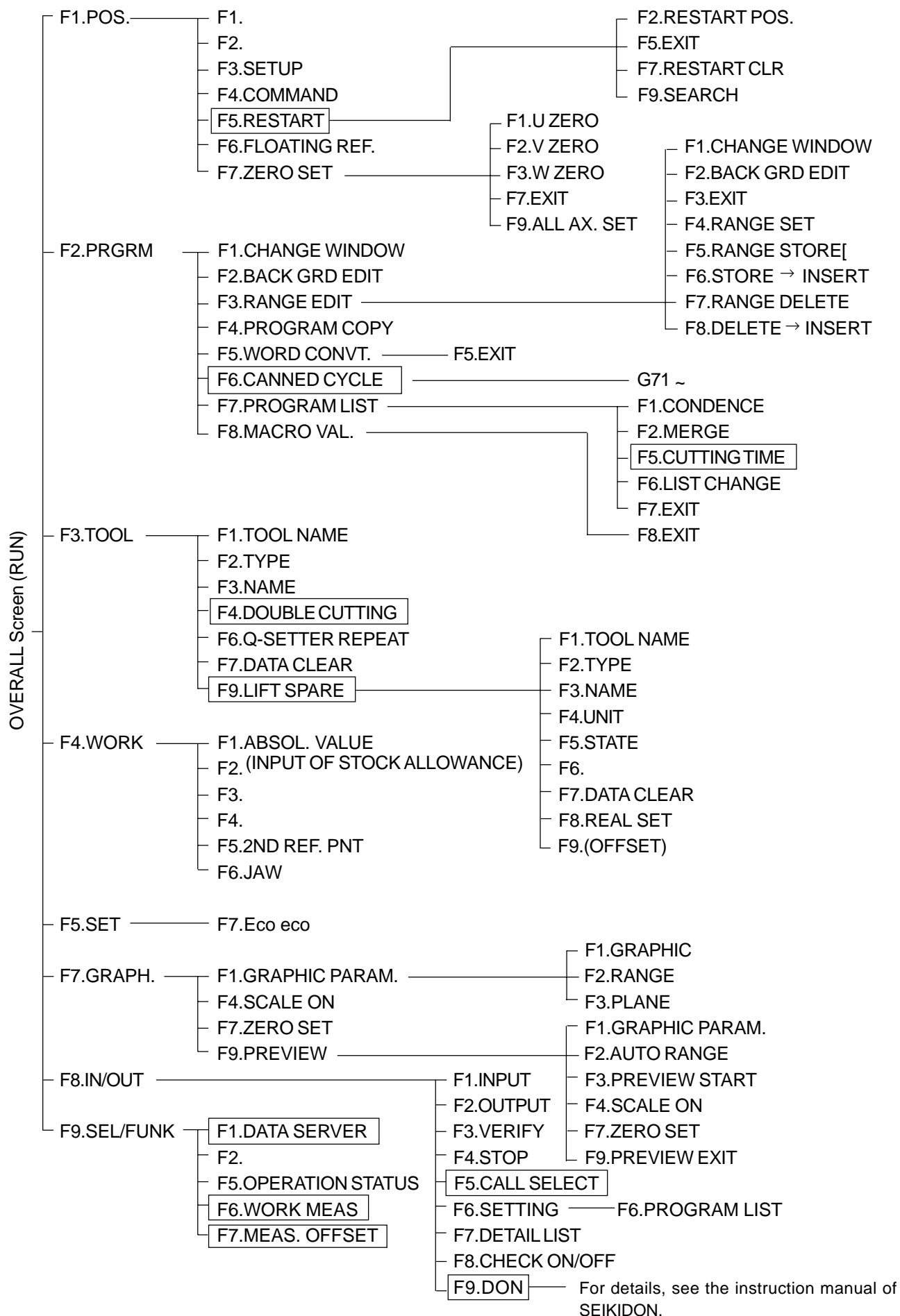
Different from the alarm, a message is displayed which notifies you of an operational mistake, etc. Message are listed.

## 1-6 Function Menu

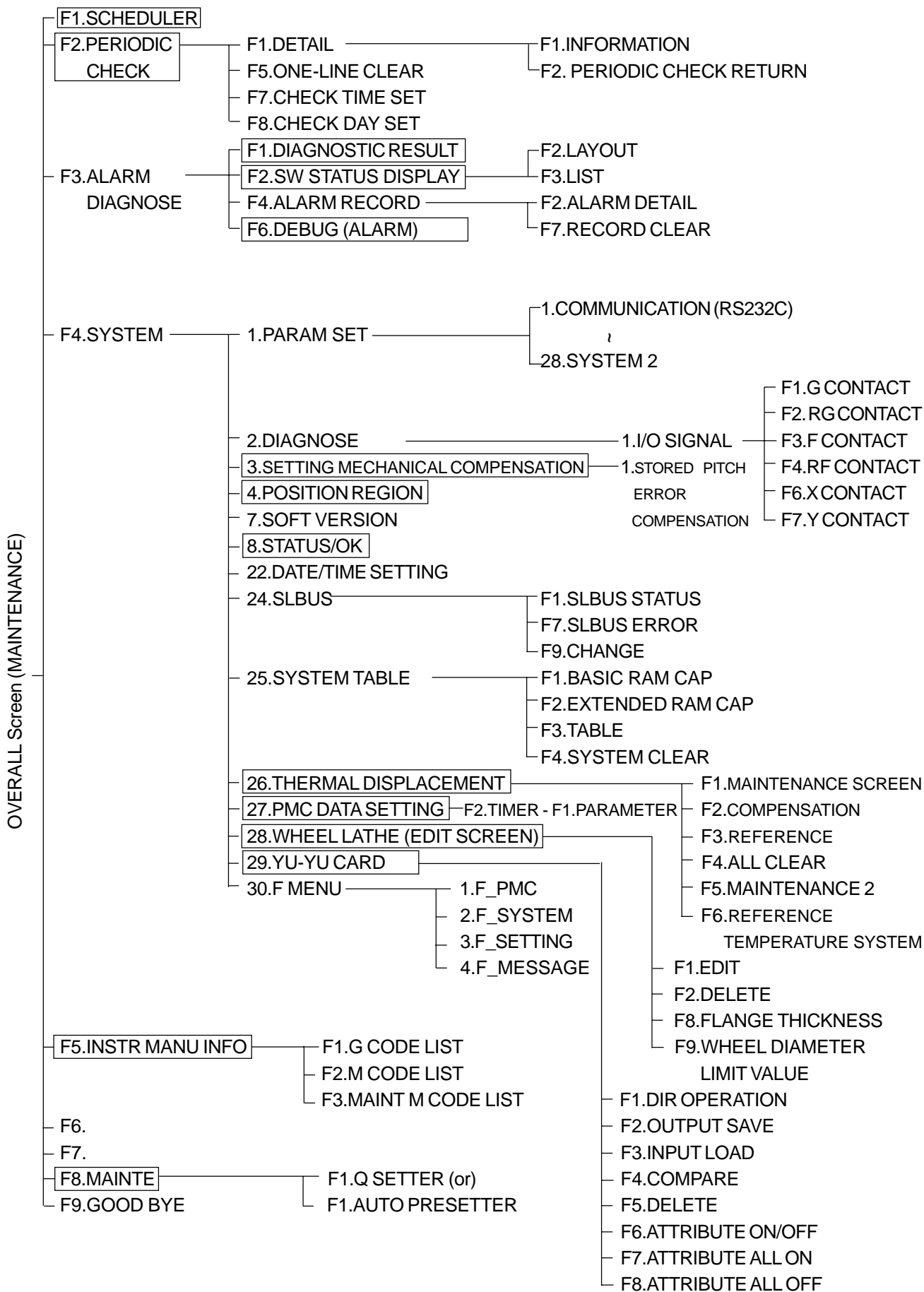
On the overall (program) screen, a screen is normally selected by a function key on which the name of the screen is inscribed. On other screens, the lower part of the screen in blue (light color) is the function menu. This menu differs depending on the screen. For details, refer to descriptions of individual screens.

- ☐ OPER/MAINTE key  
Pressing this key displays a green (black) menu.  
The function of this menu allows you to select some screens. Generally, the screen with the function keys is called a operation menu and this one a maintenance menu, respectively.

## 1-7 Tree Chart







(Note) □ denotes 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 data (tool number, tool compensation, nose radius, tool width, nose point)
- Spindle data (rpm command/surface speed command, actual rpm)
- Feed rate (feed per minute command/feed per revolution command)
- Rotary tool rpm
- Spindle clamp rpm
- Dwell remainder in seconds
- Axis load meters
- Operation state (Operating time, Running time, Cutting time, No. of set units, No. of works, Total no. of works)
- Nose position, remaining shift
- G code of the following groups  
00 (non-mode), 01 (positioning/linear or circular interpolation), 02 (plane specification), 05 (feed per minute/feed per revolution) 07 (tool diameter compensation), 09 (canned cycle for drilling), 10 (return point of canned cycle for drilling), 17 (constant surface speed control).
- M codes of five group
- Scheduled end time, machining time, tool lapping time
- Date and time

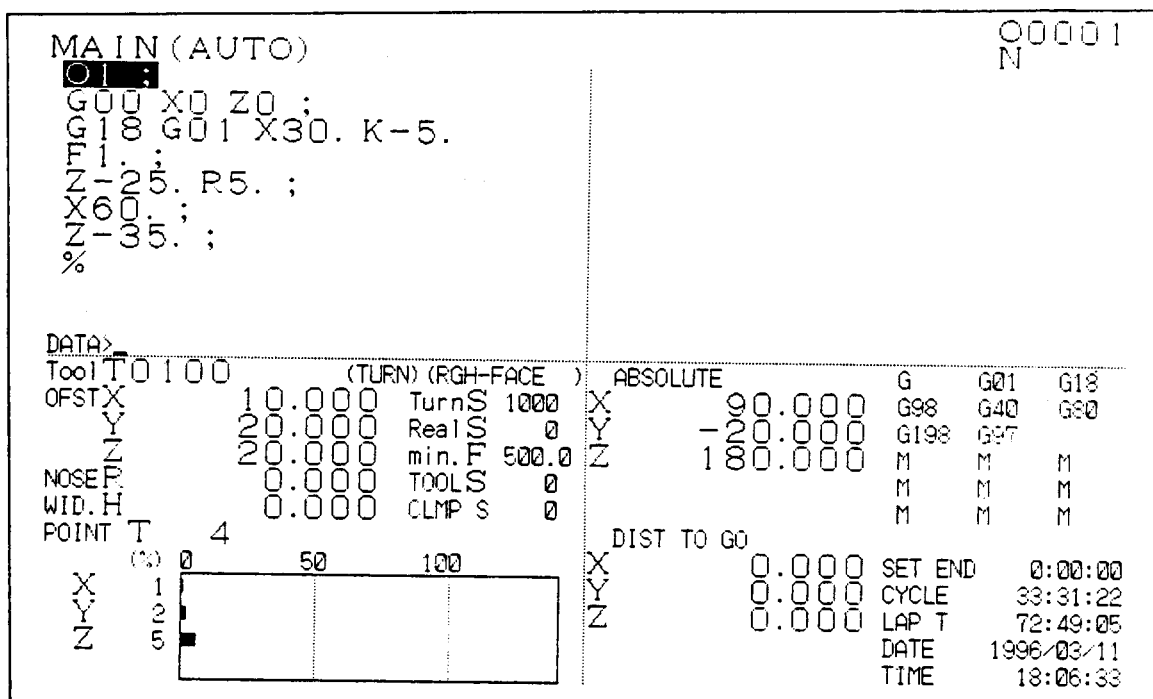


Fig. 2-0 Overall Screen

Also, the following is enabled 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<br>(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 part of 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**

Indication appears at the bottom left-hand corner of the page, displaying tool data (no., type, name, compensation, nose R, tip width, and tip point) of cut section of the tool rest.

## **2-4 Spindle Data**

On the right side of the tool data, the spindle command rpm S and actual rpm of the spindle are displayed. While in constant circumferential speed control, "Rpm S" is indicated as "Circumferential Speed V".

Further, while in Back mode, data for the back spindle are displayed.

## **2-5 Feed Speed**

Feed speed F is displayed under the spindle data.

It is indicated in "F per minute" for feeding per minute, and in "Rotation F" in feeding per rotation.

## **2-6 Rotary Tool Speed**

Rotary tool rpm is indicated under the feed speed.

With a rotary tool being fitted, "Tool S" is indicated.

## **2-7 Spindle Clamp Speed**

Spindle clamp rpm is displayed under the rotary tool.

When the spindle clamp rpm indication is valid, "Clamp S" is indicated.

Further, while in Back mode, clamp rpm of the back spindle is displayed.

## **2-8 Dwell**

While in dwell execution, dwell remainder is displayed in seconds.

When it reaches zero, the display disappears.

## **2-9 Load Meter**

State of load of each axis is indicated in a bar graph under the tool data.

## **2-10 Positional Data**

The tip position and the remainder shift volume are indicated at the bottom right-hand corner of the page concerning 4 coordinates including Tip, relative, Machine, Remainder Shift Volume.

## 2-11 G Codes

On the right side of the positional data, 8 G codes are displayed horizontally from the top left in the order below:

- \* 00 Group (Non-model)  
04, 09, 27~31, 70~76, etc.
- \* 01 Group (Positioning, Linear interpolation, Circular interpolation, Threading, Single type fixed cycle)  
00, 01, 02, 03, 32, 90, 92, 94
- \* 02 Group (Plane assignment)  
17, 18, 19
- \* 05 Group (Per-minute feed /Per-rotation feed)  
98, 99
- \* 07 Group (Tool diameter compensation)  
40, 41, 42
- \* 09 Group (Canned cycle for drilling)  
80, 81, 82, 83, 84, etc.
- \* 10 Group (Return point of canned cycle for drilling)  
198, 199
- \* 17 Group (Constant surface speed control)  
96, 97

## 2-12 M codes

M codes blonging to 5 groups are displayed horizontally from the top left in the order elow:

- \* 01 Group (Sprindle)  
03, 04, 05, 13, 14, 15
- \* 02 Group (Coolant)  
08, 09
- \* 03 Group (Program stop)  
00, 01, 02, 30
- \* 04 Group (Spindle gear)  
40, 41, 42, 43
- \* 05 Group (Others)  
Maximum 5 newest M codes other than those of 01~04 Groups are displayed.

## 2-13 Time Measurement

At the bottom right-hand corner of the page, 3 kinds of time measurement appear. Measurement is carried out within a scope not exceeding 10000 hours. To make input here, push **F9/ SEL/ FUNC** and, further, **F5 / OPERATION STATUS** to display the operation status page (Fig. 2-13). Shift the cursor to the appropriate place.

- EX. (hour/minute/second)=0 (It is set to 0.)  
(hour/minute/second)=1 (1 : 00 : 00 is input.)  
(hour/minute/second)=1/2 (1 : 02 : 00 is input.)  
(hour/minute/second)=1/2/3 (1 : 02 : 03 is input.)

OPERATION STATUS		02604 N
SET END TIME	<input checked="" type="checkbox"/> ON <input type="checkbox"/> OFF	
SET END :	0 Hour 00 M. 00 S.	
TIME		
CYCLE :	0 Hour 43 M. 33 S.	
LAP T :	0 Hour 03 M. 46 S.	
WORK COUNT		
SET :	100 pc.	
CURRENT :	1 pc.	
TOTAL :	4 pc.	
AMOUNT TIME		
OPERATE :	15 Hour 41 M. 04 S.	
RUNNING :	0 Hour 43 M. 15 S.	
CUTTING :	0 Hour 16 M. 13 S.	
<input type="text"/> /1 <input type="text"/> /2 <input type="text"/> /3 <input type="text"/> /4 <input type="text"/> /5 <input type="text"/> /6 <input type="text"/> /7 <input type="text"/> /8 <input type="text"/> /9 <input type="text"/> /0		

Fig. 2-13 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.

## 2-14 Date And Time

Date (in A.D.) and time (13:00 for 1 p.m.) are displayed based on the built-in clock.

Avoid changing it unnecessarily as it is used for periodic checking, etc. Should it become incorrect due to any reason, be sure to reset properly. To display Date/Time Set screen, push



,  ,  ,  , and  . Move the cursor to Date or Time and perform setting.

### 3. Position

If you press the F1/Position key, the position screen (Fig. 3-1) will be displayed.

Items related to position are displayed and these items can be re-written when this screen is displayed.

- Display of nose position, remaining shift amount, relative coordinate system, and machine coordinate system
- Rewriting of relative coordinate system
- Display of tool data
- Display of spindle data
- Display of feed rate

The tool data, spindle data, and feed rate are the same as the overall screen. The left half of the screen is a view of the four coordinate systems on the right that has been expanded for better understanding. The page keys   can be used for arbitrary selection.

POSITION		00000 N	
<b>ABSOLUTE</b>		<b>ABSOLUTE</b>	
X	90.0000	X	0.000
Y	-20.0000	Y	0.000
Z	180.0000	Z	0.000
C	0.0000	C	0.000
B	0.0000	B	0.000
		<b>RELATIVE</b>	
		X	0.000
		Y	0.000
		Z	0.000
		C	0.000
		B	0.000
		<b>MACHINE</b>	
		X	0.000
		Y	0.000
		Z	0.000
		C	0.000
		B	0.000
Tool T 0100			
OFST X	10.000	(TURN) (RGH-FACE)	
Y	20.000	NOSE R	0.000
Z	20.000	WID. H	0.000
		POINT T	4
P:ABS. U = -		Turn S	1500
		Real S	0
		min. F	0.3
		TOOLS	0
<div style="display: flex; justify-content: space-between; padding: 0 10px;"> <span>/1</span> <span>/2</span> <span>SET UP /3</span> <span>COMMAND /4</span> <span>RESTART /5</span> <span>FLOATING REF. /6</span> <span>ZERO SET /7</span> <span>/8</span> <span>/9</span> <span>/0</span> </div>			

Fig. 3-1 Position Screen

### 3-1 Reloading of Relative Coordinate System

Among the 4 coordinate systems, only the relative coordinate system is reloadable.  
For 0 setting, first push (F7/Zero Set) to change over the function.  
When the function for the axis for zero setting being pushed, the value of the axis gets 0.  
To zero-set all axes, push (F9/All Axes Zero Set).  
To end zero setting, push (F7/Function Return).  
To input numerals, push the key for the address to be input and display the input area.  
Input, then, numerals and push (Input). Absolute value/increment value input is available.  
EX P: absolute value X = 1.234 (Input) (1.234 is placed in X)  
I : increment value Z = 10. (Input) (10.0 is added and placed in Z)

### 3-2 Command Value Display

Pressing the Menu 4 (Command Value) displays the data specified by the program or MDI operation (Fig. 3-2).

### 3-3 Restart (Option)

Pressing  displays the screen (Fig. 3-3) required for restarting the program.

### 3-4 Floating reference

If you press F6 / Floating reference, the following message is displayed: "Set floating reference point? Y-Yes, N-No"

To set the floating reference point, press the Y key otherwise press the N key.

(See Fig. 3-4)

### 3-5 Set up

Pressing  allows you to input the data on the Setup side. (Fig. 3-5)

Use the cursor keys  and  to select the spindle speed and turret indexing, and input the data. You cannot input multiple data at one time. To execute the input data, select the MDI mode, input the data, and press . You will be inquired, "OK ? Y-YES N-NO." Pressing  executes the input data. Pressing  returns the cursor to the Position side.



00000  
N

### POSITION

ABSOLUTE

COMMAND					
G64	G110	X	S	1500	
G00	G67	G143	Y	T	100
G18	G501	Z	M	3	
G91	G150	A	B		
G22	G97	B	D		
G94	G131	C	H		
G21		U	P		
G40		W	Q		
		I	R		
G80	G120	J	L		
G198	G25	K	E		
		F	.C		
			.R		

ABSOLUTE

X	90.000	X	0.000
Y	-20.000	Y	0.000
Z	180.000	Z	0.000
C	0.000	C	0.000
B	0.000	B	0.000

RELATIVE

U	0.000
U	0.000
W	0.000
H	0.000
D	0.000

MACHINE

X	0.000
Y	0.000
Z	0.000
C	0.000
B	0.000

Tool T U U U (TURN) (RGH-FACE )  
 OFST X 10.000 NOSE R 0.000  
       Y 20.000 WID. H 0.000  
       Z 20.000 POINT T 4

Turn S 1500  
 Real S 0  
 min. F 0.3  
 TOOLS 0

P: ABS. U = -
 

/1

/2

SET UP /3

COMMAND /4

RESTART /5

FLOATING REF. /6

ZERO SET /7

/8

/9

/0

Fig. 3-2 Command Value Screen

00001  
N

### POSITION

ABSOLUTE

COMMAND					
G64	G110	X	S		
G01	G67	G143	Y	T	
G18	G501	Z	M		
	G150	A	B		
G22	G97	B	D		
G98	G131	C	H		
G21		U	P		
G40		W	Q		
		I	R		
G80	G120	J	L		
G198	G25	K	E		
		F	.C		
			.R		

### PROGRAM RESTART

TOOL T

S

E

RESTARTING DATA  

C  
P  
Q  
L

M  
S  
T  
B

[ MODE ]  
P\_RESTART

/1

RESTART POS. /2

/3

/4

POSITION /5

/6

RESTART CLR /7

/8

SEARCH /9

/0

Fig. 3-3 Restart

3 - 62

POSITION		00000 N	
ABSOLUTE		ABSOLUTE	
X	90.000	X	90.000
Y	-20.000	Y	-20.000
Z	180.000	Z	180.000
C	0.000	C	0.000
B	0.000	B	0.000

RELATIVE		MACHINE	
U	0.000	X	0.000
V	0.000	Y	0.000
W	0.000	Z	0.000
H	0.000	C	0.000
D	0.000	B	0.000

Tool T 0100		(TURN) (ROUGH-FACE)	
OFST X	10.000	NOSE R	0.000
Y	20.000	WID. H	0.000
Z	20.000	POINT T	4

Floating reference point Load & Set? Y/N

/1	/2	SET UP /3	COMMAND /4	RESTART /5	FLOATING REF. /6	ZERO SET /7	/8	/9	/0
----	----	-----------	------------	------------	------------------	-------------	----	----	----

Fig. 3-4 Floating reference

POSITION		05000 N0004	
ABSOLUTE		ABSOLUTE	
X	-1.000	X	-1.000
Y	-33.001	Y	-33.001
Z	-3.000	Z	-3.000
C	0.000	C	0.000
B	0.000	B	0.000

RELATIVE		MACHINE	
U	62.269	X	0.000
V	-24.947	Y	-31.001
W	4.166	Z	0.000
H	8.634	C	0.000
D	8.634	B	0.000

Tool T 0100		(TURN) (ROUGH O. D.)	
OFST X	1.000	NOSE R	0.000
Y	2.000	WID. H	0.000
Z	3.000	POINT T	0

SPINDLE SPEED : S = \_

/1	/2	SET UP /3	COMMAND /4	RESTART /5	FORTING REF. /6	ZERO SET /7	/8	/9	/0
----	----	-----------	------------	------------	-----------------	-------------	----	----	----

Fig. 3-5 Set up

## 4. Work Coordinates (Offset)

Pressing the F4 (WORK COORD) key displays the Work Coordinates screen (Fig. 4-0). This screen operation is used to shift the machining origin.

WORK OFFSET		O 5 0 0 0	N 0 0 0 4
WORK SHIFT			
X	0 . 0 0 0		
Y	0 . 0 0 0		
Z	0 . 0 0 0		
C	0 . 0 0 0		
B	0 . 0 0 0		
WORK LENGTH			
Z	0 . 0 0 0		
COORD. P Z=			
ABS. WRITE/1	/2	EXT. OFFSET/4	2ND.REF. PNT /5
	/3	JAW /6	/7
			/8
			/9
			/0

Fig. 4.0 Work Coordinates (Machining Reference Point Shift)

As described in the instruction manual (PROGRAMMING), Section 7-2 "Tool Nose Coordinate System", the position of the tool nose serves as the reference position of the coordinate system. Shifting of the machining origin, there are three methods – machining reference point shift, machining allowance, and work length.

Pressing the **F1** key alternately changes the input mode as follows. In each input mode, F1 and the key input area look as follows.

- <Display of Key input Area> <display of F1>
- Removal amount input mode → "COORD.P Z=" F1/ABS. WRITE
  - ↑
  - Absolute value input mode → "P: ABS. Z=" F1/COORD.P

Note) When this screen is displayed, the removal amount input mode takes effect.

#### 4-1 Removal Amount

Machining allowance = Length from the chuck jaw face to work end face – work length.

First cut the work end face within the limits of cutting allowance, so that the black layer on the end face is removed and escape in the same condition in the direction of the positive X axis. (Do not move the Z axis.)

Next, stop the rotation of the spindle. Measure the overall length of the work and find the cutting allowance. Enter this measured value as the machining allowance.

(If the cut is excessive, the value becomes negative.)

#### 4-2 Machining Reference Point Shift

The machining reference point is located at a length between the end faces of the chuck jaws (work shift amount) from the machine zero point. Press **F1/ABS. WRITE** to display

“P:ABSOL. VALUE Z=” in the key input area, and press the curs or key **↑** or **↓** to adjust the cursor to the axis you want to change. Then, input a machining reference point shift amount.

An X-axis shift amount cannot be changed.

#### 4-3 Work Length

The work length is the length between the end face of the chuck jaw and the machining zero point. Press **F1/ABS. WRITE** to display “P: ABSOL. VALUE Z =” in the key input area, and press the cursor key **↑** or **↓** to adjust the cursor to Work Length. Then, input the work length.

#### 4-4 External Work Zero Point Offset

When an external work zero point offset value is set, the entire work coordinate system can be shifted by the set value.

However, the external work zero point offset is not enabled unless the coordinates are set (T-command, manual Turret indexing, etc.).

Pressing **F4/EXT. OFFSET** opens the window shown in the right figure.

Select the axis you want to select, and input an offset value.

Note) When Equipment is equipped with heat displacement correction function, external work zero point offset is not useable.

EZTERNAL OFFSET	
X	0.000
X	0.000
Z	0.000

#### 4-5 Setting of 2nd Reference Point

With **F5/2ND REF. PNT** pushed, 2nd Reference Point Set page (Fig. 4-1) appears. Use this page to set X and Z axes of the 2nd reference point (G30). When the machine X/Z axes have been positioned to the 2nd reference point and with **F2/ENTER** having been pushed, "Is this OK? Y-Yes, N-No" appears for confirmation. For OK, push Y key.

With **F5/EXIT**, the original page is recovered.

WORK OFFSET

O 5 0 0 0  
N 0 0 0 4

WO 2ND. REF. PNT (G30) SET

Y  
Z  
C  
B  
WO  
Z  
B

2ND. REF. PNT (INDEX)

POSITION

MACHINE

X	0.000
Y	0.000
Z	0.000
C	0.000
B	0.000

X 2 1 . 0 0 0

Z 2 3 . 0 0 0

ENTER /1

EXT. OFFSET /2

EXIT /3

JAW /4

/5

/6

/7

/8



/9

/0

Fig. 4-1 2nd Reference Point

## 5. Set (Data)

If you press the F5/Set key, the setting (data) screen (see Fig. 5-0) is displayed. This screen is arranged so that frequently used parameters can be set according to item and application. If there are no optional items, none will be displayed.

The screen runs over several pages, therefore, scroll by using the page keys   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.

**SET (DATA)**

MENU

A: PROGRAM

B: CANNED CYCLE

C: MIRROR IMAGE


D: STROKE 2

E: STROKE 3


F: FLOTTING

G: ETC

INP. KEY

SHIFT + 

FIRST PAGE

SHIFT + 

LAST PAGE

00000  
N

No.	USE	DATA	
1	Single block stop in custom macro 08000-09999	<input checked="" type="radio"/> Invid	Valid
2	Single block stop in all macro	<input checked="" type="radio"/> Invid	Valid
3	Single block stop for each cycle in canned cycle for drill	<input checked="" type="radio"/> Invid	Valid
4	Stop at single block for cutter dia. comp./nose R comp	<input checked="" type="radio"/> Invid	Valid
5	Editing of program 08000-09999	<input checked="" type="radio"/> Invid	Valid
6	Display of program 08000-09999	<input checked="" type="radio"/> Invid	Valid
7	Rewriting of parameter	<input type="radio"/> Invid	<input checked="" type="radio"/> Valid

/1

/2

/3

/4

/5

/6

Eco eco /7





/8

/9

/0

Fig. 5-0 Set (Data) Screen

### 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.

Set the numeric value data from the key input area, 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 alteration.

## 5-2 Number Search

All the pages are numbered serially. The page and cursor can be moved all at once by searching for that number. A search is Performed by pressing the cursor key (  ,  ,  ,  ) after inputting N, followed by a numerical value.

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

SET (DATA)

MENU

A:PROGRAM

**B:CANNED CYCLE**

C:MIRROR IMAGE

D:STROKE 2

E:STROKE 3

F:FLOATING

G:ETC

INP. KEY

+

FIRST PAGE

+

LAST PAGE

CANNED CYCLE

No.	USE	DATA
8	G76 shift direction and amount	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
9	Single block stop in ATC cycle	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
10	Single block stop in G81-G89 cycle	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
11	G73 relief amount	0.000
12	G83 clearance	0.000
13	Chamfering amount in thread cycle	0
14	Chamfering angle for thread cycle	0
15	Return amount in roughing cycle	1001
16	Clearance amount in roughing cycle	1001
17	Return amount in cutting-off cycle	1001
18	Depth of minimum cut in thread cycle	99
19	Finishing amount in thread cycle	99

O5000  
N0004

P:ABS. = \_

/1

/2

/3

/4

/5

/6

Eco eco /7

/8

/9

/0

Fig. 5-0 (a) Set (Data) Screen

SET (DATA)

MENU

A:PROGRAM

B:CANNED CYCLE

**C:MIRROR IMAGE**

D:STROKE 2

E:STROKE 3

F:FLOTING

G:ETC

INP. KEY

+

FIRST PAGE

+

LAST PAGE

MIRROR IMAGE

No.	USE	DATA
20	X-Axis mirror image	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
21	Y-Axis mirror image	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
22	Z-Axis mirror image	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
23	A-Axis mirror image	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
24	C-Axis mirror image	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
25	C-Axis mirror image	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
26	T-Axis mirror image	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid
27	S-Axis mirror image	<input checked="" type="radio"/> Invalid <input type="radio"/> Valid

00000  
N

Eco eco

/1

/2

/3

/4

/5

/6

Eco eco /7

/8

/9

/0

Fig. 5-0 (b) Set (Data) Screen

SET (DATA)

00000  
N

MENU

A:PROGRAM

B:CANNED CYCLE

C:MIRROR IMAGE

**D:STROKE 2**

E:STROKE 3

F:FLOTTING

G:ETC

STROKE2

No.	USE	DATA
28	X-Axis coord value (limit 2,+)	-100.000
29	Y-Axis coord value (limit 2,+)	100.000
30	Z-Axis coord value (limit 2,+)	-300.000
31	A-Axis coord value (limit 2,+)	0.000
32	C-Axis coord value (limit 2,+)	0.000
33	C-Axis coord value (limit 2,+)	0.000
34	Y-Axis coord value (limit 2,+)	0.000
35	S-Axis coord value (limit 2,+)	0.000
36	X-Axis coord value (limit 2,-)	-200.000
37	Y-Axis coord value (limit 2,-)	-100.000
38	Z-Axis coord value (limit 2,-)	-400.000
39	A-Axis coord value (limit 2,-)	0.000
40	C-Axis coord value (limit 2,-)	0.000
41	C-Axis coord value (limit 2,-)	0.000
42	Y-Axis coord value (limit 2,-)	0.000
43	S-Axis coord value (limit 2,-)	0.000

INP. KEY

SHIFT +

FIRST PAGE

SHIFT +

LAST PAGE

P:ABS. = \_

/1	/2	/3	/4	/5	/6	Eco eco /7	/8	/9	/0
----	----	----	----	----	----	------------	----	----	----

Fig. 5-0 (c) Set (Data) Screen

SET (DATA)

00000  
N

MENU

A:PROGRAM

B:CANNED CYCLE

C:MIRROR IMAGE

D:STROKE 2

**E:STROKE 3**

F:FLOTTING

G:ETC

STROKE3

No.	USE	DATA
44	X-Axis coord value (limit 3,+)	-100.000
45	Y-Axis coord value (limit 3,+)	0.000
46	Z-Axis coord value (limit 3,+)	-100.000
47	A-Axis coord value (limit 3,+)	0.000
48	C-Axis coord value (limit 3,+)	0.000
49	C-Axis coord value (limit 3,+)	0.000
50	Y-Axis coord value (limit 3,+)	0.000
51	S-Axis coord value (limit 3,+)	0.000
52	X-Axis coord value (limit 3,-)	-200.000
53	Y-Axis coord value (limit 3,-)	0.000
54	Z-Axis coord value (limit 3,-)	-200.000
55	A-Axis coord value (limit 3,-)	0.000
56	C-Axis coord value (limit 3,-)	0.000
57	C-Axis coord value (limit 3,-)	0.000
58	Y-Axis coord value (limit 3,-)	0.000
59	S-Axis coord value (limit 3,-)	0.000

INP. KEY

SHIFT +

FIRST PAGE

SHIFT +

LAST PAGE

P:ABS. = \_

/1	/2	/3	/4	/5	/6	Eco eco /7	/8	/9	/0
----	----	----	----	----	----	------------	----	----	----

Fig. 5-0 (d) Set (Data) Screen



SET (DATA)

MENU

A:PROGRAM

B:CANNED CYCLE

C:MIRROR IMAGE

D:STROKE 2

E:STROKE 3

**F:FLOTING**

G:ETC

INP. KEY

SHIFT +

FIRST PAGE

SHIFT +

LAST PAGE

FLOATING

No.	USE	DATA
60	X-Axis floating ref. point	10.000
61	Y-Axis floating ref. point	-250.000
62	Z-Axis floating ref. point	-244.000
63	A-Axis floating ref. point	0.000
64	C-Axis floating ref. point	0.000
65	C-Axis floating ref. point	0.000
66	7-Axis floating ref. point	0.000
67	8-Axis floating ref. point	0.000

00000  
N

P:ABS. = -

/1

/2

/3

/4

/5

/6

Eco eco /7

/8

/9

/0

Fig. 5-0 (e) Set (Data) Screen

SET (DATA)

MENU

A:PROGRAM

B:CANNED CYCLE

C:MIRROR IMAGE

D:STROKE 2

E:STROKE 3

F:FLOTING

**G:ETC**

INP. KEY

SHIFT +

FIRST PAGE

SHIFT +

LAST PAGE

ETC

No.	USE	DATA
68	SINGLE BLOCK BUFFERING	<input type="radio"/> Invlid <input checked="" type="radio"/> Valid
69	Selection of DNC run mode	<input checked="" type="radio"/> Invlid <input type="radio"/> Valid
70	Seq No. for Seq No. col.&stop	0

00000  
N

/1

/2

/3

/4

/5

/6

Eco eco /7

/8

/9

/0

Fig. 5-0 (f) Set (Data) Screen

### 5-3 Eco-eco

Pressing [F7/Eco eco] displays the Eco-eco screen (Fig. 5-3). In this screen, you set the auto sleeping screen, auto work light OFF, and chip conveyor auto OFF times (minutes)

SET (DATA)

MENU

A: PROGRAM

B: CANNED CYCLE

C: MIRROR IMAGE

D: STROKE 2

E: STROKE 3

F: FLOATING

G: ETC

INP. KEY

[SHIFT] + [F7] FIRST PAGE

[SHIFT] + [F8] LAST PAGE

O5000  
N0004

PROGRAM

No.	USE	DATA	
1	Single block stop in custom macro 08000-09999	<input checked="" type="radio"/> Invl	<input type="radio"/> Valid
2	Single block stop in all macro	<input checked="" type="radio"/> Invl	<input type="radio"/> Valid
3	Single block stop for each cycle in canned cycle for drill	<input checked="" type="radio"/> Invl	<input type="radio"/> Valid
4	Stop at single block for cutter dia. comp./nose R comp	<input checked="" type="radio"/> Invl	<input type="radio"/> Valid
5	Editing of program 08000-09999	<input checked="" type="radio"/> Invl	<input type="radio"/> Valid
6	Display of program 08000-09999	<input checked="" type="radio"/> Invl	<input type="radio"/> Valid
7	Rewriting of parameter	<input type="radio"/> Invl	<input checked="" type="radio"/> Valid

E C O - E C O	
AUTO SLEEPING SCREEN	0 (MIN.)
AUTO WORK LIGHT OFF	8 (MIN.)
CHIP CONVEYOR AUTO OFF	8 (MIN.)

P: ABS. = \_

										Eco eco
/1	/2	/3	/4	/5	/6	/7	/8	/9	/0	

Fig. 5-3 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 ( $\pm 99999999$  to  $\pm 0.0000001$ ). 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)

00001  
N

PROGRAM(AUTO)  
LOCAL VAL. 1/1

No.	DATA	No.	DATA	No.	DATA
1		16		31	
2		17		32	
3		18		33	
4		19			
5		20			
6		21			
7		22			
8		23			
9		24			
10		25			
11		26			
12		27			
13		28			
14		29			
15		30			

P:ABS. = \_

/1	/2	/3	/4	/5	/6	/7	EXIT /8	/9	/0
----	----	----	----	----	----	----	---------	----	----

Fig. 6-2 (a) Local Variable

00001  
N

PROGRAM(AUTO)  
COMMON VALLE 3/4

No.	DATA	NAME	No.	DATA	NAME
500	1.0000000	SIN[30]	515	0.0000000	ADF[#24]
501	0.0000000	COS[60]	516	1.0000000	
502	1.0000000	TAN[45]	517	0.0000000	
503	1.0000000	ASIN[0.5]	518	0.0000000	
504	1.0000000	ACOS[0.5]	519	0.0000000	
505	0.0000000	ATAN[1]/[1]	520	1.0000000	
506	1.2340000	ABS[-1.234]	521	0.0000000	
507	1.7320508	SQRT[3]	522	0.0000000	
508	2.7182818	EXP[1]	523	0.0000000	
509	4.6051702	LN[100]	524	0.0000000	
510	4.0000000	ROUND[3.5]	525	0.0000000	
511	123.0000000	FIX[123.5]	526	0.0000000	
512	-124.0000000	FUP[-123.4]	527	0.0000000	
513	123.0000000	BIN[291]	528	0.0000000	
514	291.0000000	BCD[123]	529	0.0000000	

P:ABS. = \_

/1	/2	/3	/4	/5	/6	DATA CLEAR /7	EXIT /8	/9	/0
----	----	----	----	----	----	---------------	---------	----	----

Fig. 6-2 (b) Common Variable

## 7. Plot

Pressing the **F7/GRAPHIC** key displays the Graphic screen (PREVIEW EXIT) (Fig. 7-0) This screen plots the tool path and also displays the following data for reference to plotting.

- ☐ Parameter plotting plane
- ☐ Tool nose position
- ☐ Spindle speed S, feed rate F, tool command T
- ☐ Machining time, cutting time
- ☐ 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**.

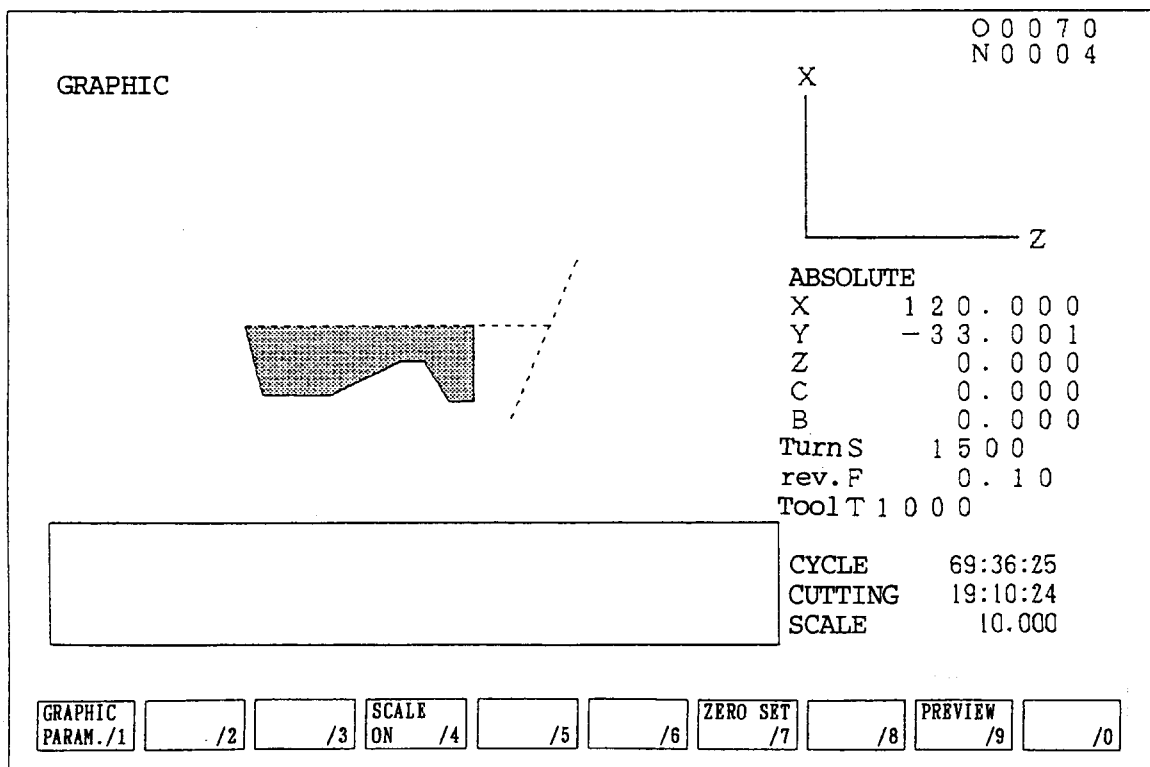


Fig. 7-0 Graphic Screen (PREVIEW EXIT)

The following settings are allowed for plotting. For details, refer to the next section.

- ☐ Selection of plotting plane
- ☐ Specification of angle or 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-matching 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 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) Any function whose conditions change by machine operation (macro program using skip 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.

Note 5) For the model equipped with the built-in type main spindle with Main Spindle/C spindle switching, when zero point return is not completed for C spindle in Main Spindle mode or in C-Axis mode, an alarm takes place in Polar Coordinate Interpolation/Cylinder Interpolation mode. Other C spindle commands are ignore.

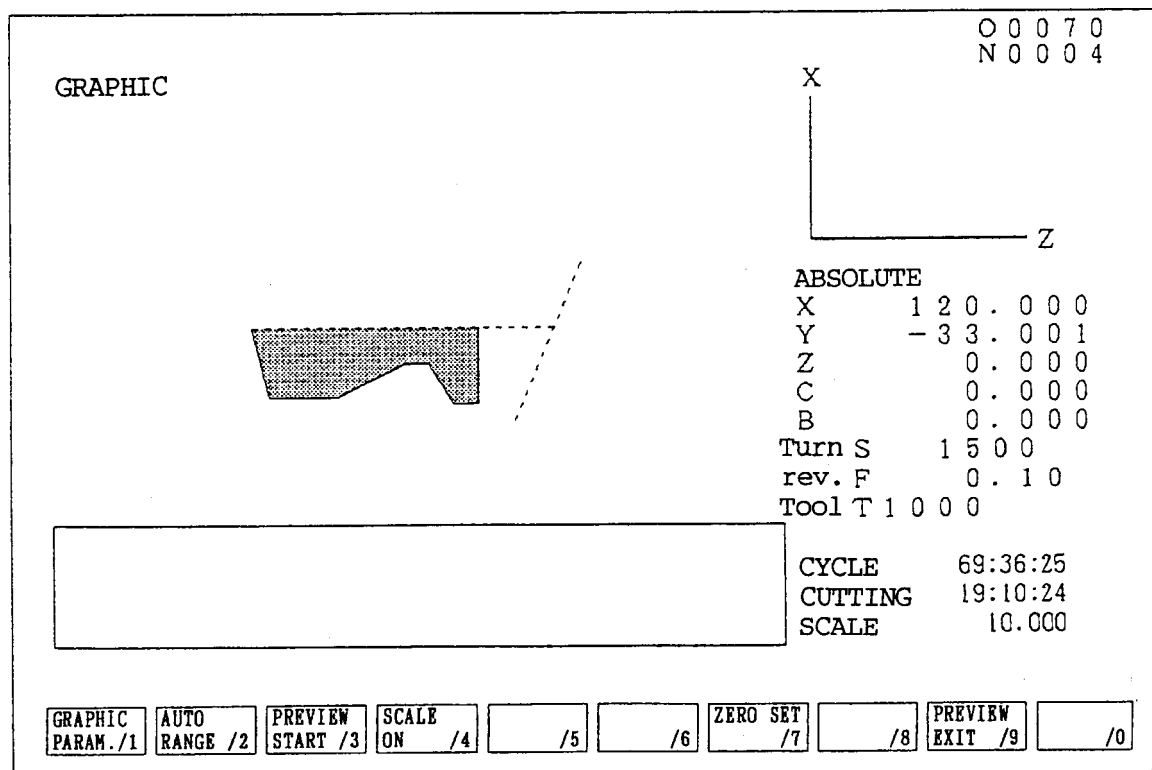


Fig. 7-1 Graphic Screen (Preview)

## 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.

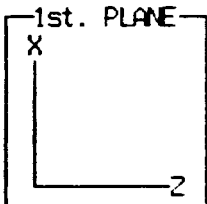
This screen allows you to set the plotting parameters.

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).

00100  
 N0005

GRAPHIC
X

1st. PLANE



2nd. PLANE

(NONE)

RANGE	Z	X	~	
		20.500	~	26.500
		30.500	~	19.500
			~	
			~	
			~	
			~	

SCALE
10.000

EACH TOOL
☒ UALD ☐ INUL

CUT. LINE
☒ LINE ☐ DOT ☐ NON

RAPID LINE
☐ LINE ☒ DOT ☐ NON

LINE

123456

POINT

☐ ☐ ☐ ☐ ☐ ☐

Info

Graphic range.

G50 X100. Z100. ;

G98 G00 X60.0 Z100. ;

G71 P1 Q10 D1.0 F1000 ;

P:ABS.=

CYCLE 3:12:02

CUTTING 1:17:11

SCALE 10.000

GRAPHIC /1

RANGE /2

PLANE /3

/0

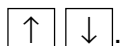
Fig. 8-0 Plotting Parameter (Range Set)

GRAPHIC PARAMETER									
1st. PLANE		2nd. PLANE		1st. PLANE		2nd. PLANE			
X		(NONE)		PLANE		PLANE		-1	
Z				ANGLE		HOL.			
				UER.		UER.			
				AXIS		1		Z	
				2		2		X	
				3		3			
Info				-1 0 1 2 3 4					
Graphic plane.				(NONE) 1 2 1 2 1 2 1					
G50 X100. Z100. ;						CYCLE 3:12:02			
G98 G00 X60.0 Z100. ;						CUTTING 1:17:11			
G71 P1 Q10 D1.0 F1000 ;						SCALE 10.000			
PLANE =									
GRAPHIC	RANGE	PLANE							
/1	/2	/3	/4	/5	/6	/7	/8	/9	/0

Fig. 8-1 Plotting Parameter (Plans 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 ineffective 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 after, 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 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.

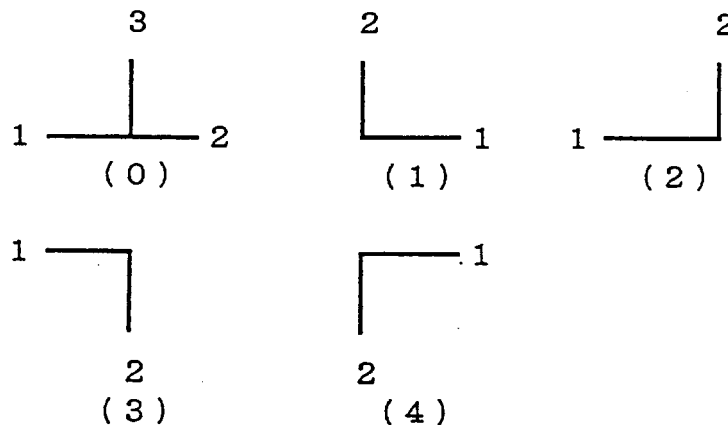
To make setting, select color designation with the cursor key (  or  ).

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

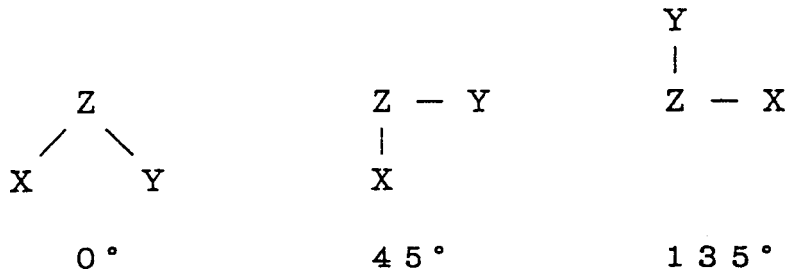
This is effective only when the plotting plane is 0 (3 axes display).

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  $\pm 180$  in an increment of  $1^\circ$

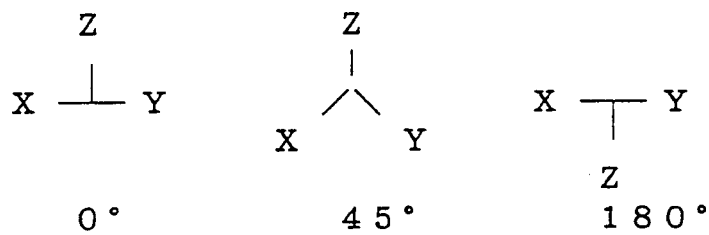
Example) When the plotting plane is (XYZ) and the vertical rotation angle is  $90^\circ$



### (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  $0^\circ$



## (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 (OFFSET) key displays the Tool screen (Fig. 9-0).

The tool screen (Fig. 9-0) is displayed when you press the F3/ compensation key.

The compensation data for each tool number is displayed and set as shown below.

	Tool Size	Tool Nose Radius	Tool Nose Point	Tool width
Form	X Y Z	R	T	H
Wear	U V W	Q		J

In addition, the following functions are provided.

- Display of the tool nose position and machine coordinate system
- Data deletion
- Display and setting of the tool list data

O 5 0 0 0  
N 0 0 0 4

TOOL OFFSET											
GEOM.		WEAR		GEOM.		WEAR		GEOM.		WEAR	
01 (TURN)		(ROUGH FACE)		02 ( )		( )		03 (TURN)		(FINISH OD)	
X	1.000	U	0.000	X	1.999	U	0.000	X	100.000	U	0.000
Y	2.000	V	0.000	Y	20.000	V	0.000	Y	200.000	V	0.000
Z	3.000	W	0.000	Z	30.000	W	0.000	Z	299.999	W	0.000
R	0.000	Q	0.000	R	2.499	Q	0.000	R	0.000	Q	0.000
T	3			T	2			T	3		
H	0.000	J	0.000	H	0.000	J	0.000	H	0.000	J	0.000
04 ( )				05 ( )				06 ( )			
X	20.000	U	0.000	X	30.000	U	0.000	X	0.000	U	0.000
Y	30.000	V	0.000	Y	40.000	V	0.000	Y	23.000	V	0.000
Z	40.000	W	0.000	Z	50.000	W	0.000	Z	299.999	W	0.000
R	3.000	Q	0.000	R	1.999	Q	0.000	R	0.000	Q	0.000
T	4			T	5			T	6		
H	0.000	J	0.000	H	0.000	J	0.000	H	0.000	J	0.0000
						ABSOLUTE		MACHINE			
						X 120.000		X 121.000			
						Y -33.001		Y -31.000			
						Z 0.000		Z 3.000			
						C 0.000		C 0.000			
						B 0.000		B 0.000			

GEOMETRY : X = \_

TOOL	TYPE	NAME			Q-SETTER	DATA		LIFE	
LIST /1	/2	/3	/4	/5	REPEAT/6	CLEAR/7	/8	SPARE/9	/0

Fig. 9-0 Tool (offset) Screen

### ● Using tool compensation data

Tool compensation data is divided into form and wear data for each item.

Compensation using T in the program corresponds to the tool number of the tool (compensation) screen. For details refer to the section on Tool Function (T function) in Program Editing.

## 9-1 Setting of Offset Value

There are 2 kinds of offset value; profile and wear. To make setting, adjust the cursor and set through the key input area.

Wear can be input any time but for input of form data, the manual mode must be selected and the write switch must be enabled.

When form is input, the wear will become 0.

### ○ Data input method

Form is input as absolute value, wear as increment value using the INPUT key.

### ○ Wear input clamping

A value greater than the value set in parameter 5027 cannot be input. The message "Clamped value is exceeded" will be output.

Note) If wear is input during automatic operation, the wear data of the relevant tool will be reflected from the next T-command onward, because the coordinate system will not be set. (The coordinate system will be set in the manual mode.)

## 9-2 Tool list

Pressing **F1/TOOL LIST** displays the TOOL LIST screen (Fig. 9-2). This screen is mainly used when the machine is provided with the ATC function. Since the ATC's tool number is handled by the address rewriting method, it is necessary to associate the turret face numbers with the tool numbers, and the magazine pot numbers with the tool numbers.

When the ATC function is not provided, the turret face numbers and the tool numbers have been set one to one upon shipment. Do not alter them. Register in the TOOL LIST screen as follows:

- ① Move the cursor to the field you want to input the data, and input it.
- ② Press the **INPUT** key.
  - If duplicate data is input, a message, "DUPLICATE (Y-ALTER, N-CANCEL)," will appear. Pressing **Y** clears the other same number and registers in the cursor-indicated position. Pressing **N** does not alter the data.
  - When the machine is provided with the ATC function, there must be two or more empty areas, including the ATC arm.

You can also set a tool type and a tool name in the TOOL LIST screen. (See the following two sections) If **F7/DATA CLEAR** is pressed in the TOOL LIST screen, a window will appear in the center of screen. Then, you clear all the data in the TOOL LIST screen or set the turret numbers and the tool number to one to one.

Select either one with the cursor key **↑** or **↓** and press the **INPUT** key.

You will be prompted, "OK ? Y-YES, N-NO." If OK, press the **Y** key.

TOOL OFFSET										O 5 0 0 0 N 0 0 0 4
GEOM.		TOOL LIST								
01 (TURN		ARM TURRET - ( ) ( )								
X 1.		ARM MAGAZINE - 14 (MI-X) ( )								
Y 2.										
Z 3.										
R 0.										
T 3										
H 0.										
04 (TURN										
X 20.										
Y 30.										
Z 40.										
R 3.										
T 4										
H 0.										

TRRT No.	TOOL No.	TYPE	NAME	MGZN POT	TOOL No.	TYPE	NAME
1	1	TURN	ROUGH OD	11	11	TURN	
2	2	MI-X		12	12	MI-X	
3	3	TURN	ROUGH FACE	13	13	TURN	
4	4	TURN	FNSH FACE	14			
5	5	TURN		15	15	TURN	
6	6	MI-X		16	16	TURN	
7	7	TURN	FINISH OD	17	17	TURN	
8	8	MI-Z		18	18	TURN	
9	9	TURN		19	19	TURN	
10	10	MI-X		20	20	MI-Z	
				21	21	TURN	
				22	22	MI-X	

TOOL No. = \_

EXIT	TYPE	NAME			Q-SETTER	DATA		LIFE	
/1	/2	/3	/4	/5	REPEAT/6	CLEAR/7	/8	SPARE/9	/0

Fig. 9-2 Tool List

Note) The above-mentioned screen is for the machine with the ATC function. If the ATC function is not provided, the arm and the magazine will not be displayed.

### 9-3 Setting of Type

Setting of the type is done through the function menu. First, Press the Menu 2 (TYPE). Since the window appears in the center of the screen, select with the cursor key (  or  ) and press the INPUT key to determine.

Note) When the tool type is changed by parameter setting, an offset amount for that tool can be automatically set.

Parameter no. 5005, #6 = 0 : Enables automatic setting of the offset amount when changing the tool type.

= 1 : Disables automatic setting of the offset amount when changing the tool type.





Breakdown of automatic setting is as follows. (When the tool type group (\*1) is not changed or the tool type is deleted, the offset amount will not be set.)

Tool Type Group (*1)		Automatic Setting of Offset Amount
Before Change	After Change	
Turning tool, Z rotary tool	X rotary tool	Shape Z = PRM5031, Wear Z = 0, Tool Nose Point = 9, Others (X, Y, Tool Nose Radius, Tool Width: Shape and Wear) = 0
Turning tool, X rotary tool	Z rotary tool	Shape X = PRM5032, Wear X = 0, Tool Nose Point = 9, Others (Y, Z, Tool Nose Radius, Tool Width: Shape and Wear) = 0
X rotary tool, Z rotary tool	Turning tool	All (X, Y, Z, Tool Nose Radius, Tool Width: Shape and Wear) = 0

(\*1) The tool type groups are classified into three categories:

- X rotary tools: Rotary X, Stationary X
- Z rotary tools: Rotary Z, Stationary Z
- Turning tools: Other than X rotary tools and Z rotary tools (Turning tools, stationary tools, others)

#### 9-4 Setting of Name

Use the function menu to set the name. First, press the Menu 3 (TOOL NAME). The window appears in the center of the screen. Select with the cursor key (  or  ) or page key (P  or  ) 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 to 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)

#### 9-5 Data Clear

The F7/Data Clear key may be used to clear the tool data at the cursor or one line of characters when starting afresh. It is used also to clear specific data of all the tools and replace the data by zero. If you press the F7/Data Clear key, a window appears at the center of the screen, showing the data to be cleared. Select the data to be deleted using the cursor keys \*\*\* and press the INPUT key.

The message "OK? Y-Yes, N-No" appears. If OK, press the Y key.

## 9-6 Tool life Management

To set the tool life, press the F9/Life Spare key. For this functional operation, refer to the instruction manual for tool life management.

## 9-7 Number Search

The page and cursor can be moved at once by searching for the tool number. A search is performed by pressing the cursor key (  ,  ,  ,  ) after inputting N, followed by a numerical value.

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

## 9-8 Double Cutting (Option)

Pressing (F4/DOUBLE CUT) displays the Double Cut screen (Fig. 9-8). The double cutting function is designed to prevent chattering by changing the spindle speed. In this screen, set a speed change ratio for each tool.

CUTDOUBLER

TOOL No.	TYPE	TOOL NAME	COMMAND A
1	TURN	ROUGH FACE	50
2			100
3	TURN	ROUGH O. D.	
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			
15			
16			

05000  
N0004

Tool T 0 1 0 0

Turn S      0  
Real S      0

DATA A = \_



EXIT			DATA					
/1	/2	/3	/4	/5	/6 CLEAR/7	/8	/9	/0

Fig. 9-8 Double Cutting

## 10. Alarm Diagnosis

Press the **OPER/MAINT** key, and then, **F3/ALARM DIAG** key back menu. The Alarm Diagnosis screen (Fig. 10-0) appears, which shows machine conditions such as alarm, etc. It is automatically switched at the same time when an alarm occurs (it is also possible to prohibit switching with a parameter).

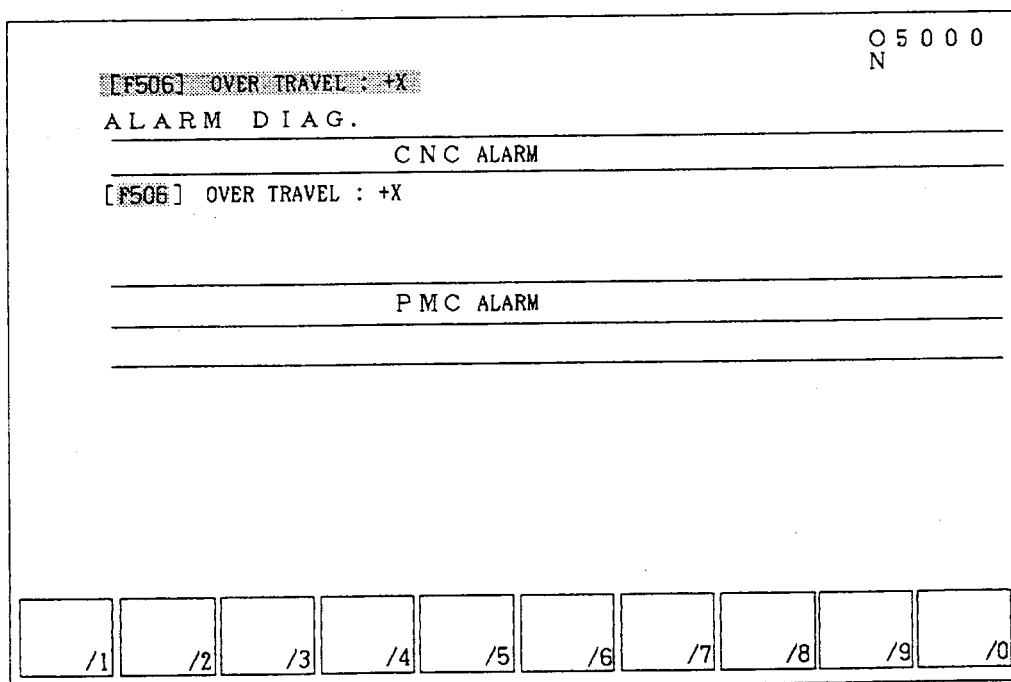
All the generated alarms are displayed, sorted into the CNC alarms and the PMC alarms (issued by ladder sequence software of the machine).

When one screen is not enough to display all the alarms, they will be extended over multiple pages. Use the page key  or  to change over the screen. The CNC alarms are displayed in different colors at the upper left part of the screen to distinguish between an alarm and a warning.

- ☐ Red ..... Alarm (Automatic operation disallowed)
- ☐ Yellow ..... Warning (Automatic operation allowed)

The PMC alarms are displayed in red at the upper left part of the screen.

Whether or not automatic operation is allowed is controlled by ladder sequence software, regardless of the alarm display color.



The screen displays the following information:

- Top right corner: **O 5 0 0 0**  
**N**
- Alarm message: **[F506] OVER TRAVEL : +X**
- Section header: **ALARM DIAG.**
- Section separator: **CNC ALARM**
- Alarm message: **[F506] OVER TRAVEL : +X**
- Section separator: **PMC ALARM**
- Status bar at the bottom with 10 slots labeled **/1** through **/10**.

Fig. 10-0 Alarm Diagnosis Screen



## 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 date 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.

Note) Simplified display and detailed display may be the same for some alarms.

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**.

ALARM HISTORY				O 5 0 0 0 N				
No	OCCURRENCE	FUNC	ALARM MESSAGE					
1	01/20 14:16	CNC[F507]	OVER TRAVEL:-X					
2	01/20 13:59	CNC[ 795]	EMERGENCY STOP					
3								
4								
5								
6								
7								
8								
9								
10								
11								
12								
13								
14								
15								
16								
17								
18								

DIAGNOS /1	MESSAGE /2	/3	ALARM LIST /4	/5	/6	HISTORY CLEAR /7	/8	/9	/0
---------------	---------------	----	------------------	----	----	---------------------	----	----	----

Fig. 10-1 Alarm History



O 5 0 0 0  
N

PARAMETERS

SET ITEM	
1. COMMUNICATION(RS232C) (100~)	17. CUSTOM MACRO (6000~)
2. AXIS FEED/UNIT (1000~)	18. SKIP FUNCTION (6200~)
3. COORDINATE (1200~)	19. MEASUREMENT (6240~)
4. STROKE LIMIT (1300~)	20. GRAPHIC DISPLAY (6500~)
5. CHUCK/TAILSTOCK BARRIER (1330~)	21. MANUAL HANDLE/INTERRUPT (7100~)
6. FEED RATE (1400~)	22. POLYGON (7600~)
7. ACC./DEC. (1600~)	23. CUTTING MONITORING (8000~)
8. SERVO (1800~)	24. H. P. C. C (8400~)
9. DI/DO (3000~)	25. OTHERS (8650~)
10. CRT/MDI & EDIT (3100~)	26. SERVICE (8900~)
11. PROGRAM (3400~)	27. SYSTEM (9000~)
12. PITCH ERROR COMP. (3600~)	28. SYSTEM2 (9900~)
13. SPINDLE (3700~)	
14. TOOL OFFSET (5000~)	
15. CANNED CYCLE (5100~)	
16. DIRECT TAPPING (5200~)	

ITEM No = \_

/1	/2	/3	/4	/5	/6	/7	/8	/9	/0
----	----	----	----	----	----	----	----	----	----

Fig. 11-1 Parameter

O 5 0 0 0  
N

PARAMETERS



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

P:ABS. = \_

/1	/2	/3	/4	/5	/6	/7	/8	/9	/0
----	----	----	----	----	----	----	----	----	----

Fig. 11-1 (a) Parameter





(1) Item selection


Display the page of 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 area 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 is 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** when the 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 be viewed.

(Use **SHIFT**  for returning.)

Number search is not possible.

DIAGNOSE
O 5 0 0 0  
N

MENU

I. INPUT / OUTPUT

ITEM No. =

/1

/2

/3

/4

/5

/6

/7

/8

/9

/0

Fig. 11-2 Diagnose

DIAGNOSE
O 5 0 0 0  
N

OUTPUT SIGNAL TO NC 1 (PMC → CNC) 1/8

No	7	6	5	4	3	2	1	0	Hex	No	7	6	5	4	3	2	1	0	Hex
G000	0	0	0	0	0	0	0	0	00	G016	0	0	0	0	0	0	0	0	00
G001	0	0	0	0	0	0	0	0	00	G017	0	0	0	0	0	0	0	0	00
G002	0	0	0	0	0	0	0	0	00	G018	0	0	0	0	0	0	0	0	00
G003	0	0	0	0	0	0	0	0	00	G019	0	0	0	0	0	0	0	0	00
G004	0	0	0	0	0	0	0	0	00	G020	0	0	0	0	0	0	0	0	00
G005	0	0	0	0	0	0	0	0	00	G021	0	0	0	0	0	0	0	0	00
G006	0	0	0	0	0	0	0	0	00	G022	0	0	0	0	0	0	0	0	00
G007	0	0	0	0	0	0	0	0	00	G023	0	0	0	0	0	0	0	0	00
G008	0	0	0	0	0	0	0	0	00	G024	0	0	0	0	0	0	0	0	00
G009	0	0	0	0	0	0	0	0	00	G025	0	0	0	0	0	0	0	0	00
G010	0	0	0	0	0	0	0	0	00	G026	0	0	0	0	0	0	0	0	00
G011	0	0	0	0	0	0	0	0	00	G027	0	0	0	0	0	0	0	0	00
G012	0	0	0	0	0	0	0	0	00	G028	0	0	0	0	0	0	0	0	00
G013	0	0	0	0	0	0	0	0	00	G029	0	0	0	0	0	0	0	0	00
G014	0	0	0	0	0	0	0	0	00	G030	0	0	0	0	0	0	0	0	00
G015	0	0	0	0	0	0	0	0	00	G031	0	0	0	0	0	0	0	0	00

G  
RELAY /1

RG  
RELAY /2

F  
RELAY /3

RF  
RELAY /4

/5

X  
RELAY /6

Y  
RELAY /7

/8

/9

/0

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.

05000  
N

MECHANICAL COMPENSATION
 

MENU

1. STORED PITCH-ERROR COMP.

ITEM No = 
 

/1

/2

/3

/4

/5

/6

/7

/8

/9

/0

Fig. 11-3 Mechanical Compensation

05000  
N

MECHANICAL COMPENSATION
 

1/16

STORED PITCH-ERROR COMPENSATION

No	DATA	No	DATA	No	DATA	No	DATA
0000	0	0016	0	0032	0	0048	0
0001	0	0017	0	0033	0	0049	0
0002	0	0018	0	0034	0	0050	0
0003	0	0019	0	0035	0	0051	0
0004	0	0020	0	0036	0	0052	0
0005	0	0021	0	0037	0	0053	0
0006	0	0022	0	0038	0	0054	0
0007	0	0023	0	0039	0	0055	0
0008	0	0024	0	0040	0	0056	0
0009	0	0025	0	0041	0	0057	0
0010	0	0026	0	0042	0	0058	0
0011	0	0027	0	0043	0	0059	0
0012	0	0028	0	0044	0	0060	0
0013	0	0029	0	0045	0	0061	0
0014	0	0030	0	0046	0	0062	0
0015	0	0031	0	0047	0	0063	0

P:ABS. = 
 

/1

/2

/3

/4

/5

/6

/7

/8

/9

/0

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

## 11-4 Position Region

Set the cursor to "Position Region" on System screen and push .

Position Region screen appears.

Position region data are displayed and set.

O5000  
N

POSITION AREA
[STATUS]

AREA B ( )			AREA C ( )		
+X	0.000	-X 0.000	+X	0.000	-X 0.000
+Y	0.000	-Y 0.000	+Y	0.000	-Y 0.000
+Z	0.000	-Z 0.000	+Z	0.000	-Z 0.000

AREA D ( )			AREA E ( )		
+X	0.000	-X 0.000	+X	0.000	-X 0.000
+Y	0.000	-Y 0.000	+Y	0.000	-Y 0.000
+Z	0.000	-Z 0.000	+Z	0.000	-Z 0.000

MACHINE  
 X 0.000  
 Y 0.000  
 Z 0.000

P:ABS. =

KIND NAME /1	<span style="border: 1px solid black; padding: 2px 10px;"> </span> /2	<span style="border: 1px solid black; padding: 2px 10px;"> </span> /3	<span style="border: 1px solid black; padding: 2px 10px;"> </span> /4	<span style="border: 1px solid black; padding: 2px 10px;"> </span> /5	<span style="border: 1px solid black; padding: 2px 10px;"> </span> /6	DATA CLEAR /7	<span style="border: 1px solid black; padding: 2px 10px;"> </span> /8	<span style="border: 1px solid black; padding: 2px 10px;"> </span> /9	<span style="border: 1px solid black; padding: 2px 10px;"> </span> /0
-----------------	-----------------------------------------------------------------------	-----------------------------------------------------------------------	-----------------------------------------------------------------------	-----------------------------------------------------------------------	-----------------------------------------------------------------------	------------------	-----------------------------------------------------------------------	-----------------------------------------------------------------------	-----------------------------------------------------------------------

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.

O 5 0 0 0  
N

Function Name	Software Version	Function
NC	B E E 1 - 0 0 0 8	CNC system
	B 1 L X - A 0 9	CNC software
H I	B 2 L J - A 0 8	Human interface
	B Y 0 1 - 2 . 1 A	Library
	B 4 L J - A 0 3 H	FROM file
PMC	4 0 . 6 . 8 - 0 0 H 3	PMC system
	B 3 L X - A 0 3 A	PMC-C software
SERVO	9 0 8 0 - 0 0 1 0	SERVO Function
BOOT	6 0 M 1 - 0 0 1 1	Boot system
SPNDL	9 D 0 0 - 0 0 1 9	Serial spindle 1

LADDER PROGRAM	E 0 1 A - 0 0 0 0
----------------	-------------------

MULTI	
-------	--

/1	/2	/3	/4	/5	/6	/7	/8	/9	/0

Fig. 11-5 Software Version



## 11-6 Date and Time

Set the cursor to "Date and Time" on System screen and push .

Date and Time screen appears.

Date and time are displayed and set.

05000  
N

TIME & DATE

DATE

YEAR/MONTH/DAY

1998/07/03

TIME

HOUR:MIN.:SEC.

10:46:37

YEAR/MONTH/DAY =

/1

/2

/3

/4

/5

/6

/7

/8

/9

/0

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.

SLBUS TABLE

1/1

O 5 0 0 0  
N

No	Loc#	Type	Name	Type	LEAD.C/CH		HI-SPEED		SIZE		ADRS	
					Buff.#	Size	IN	OUT	IN	OUT	IN	OUT
01	01	0	OP10	I/O	00	00	00	00	10	0D	F0	F0
02	02	0	10-128	I/O+AB	01	01	02	00	10	10	00	00
03	FF	0			00	00	00	00	00	00	00	00
04	00	0			00	00	00	00	00	00	00	00
05	00	0			00	00	00	00	00	00	00	00
06	00	0			00	00	00	00	00	00	00	00
07	00	0			00	00	00	00	00	00	00	00
08	00	0			00	00	00	00	00	00	00	00
09	00	0			00	00	00	00	00	00	00	00
10	00	0			00	00	00	00	00	00	00	00
11	00	0			00	00	00	00	00	00	00	00
12	00	0			00	00	00	00	00	00	00	00
13	00	0			00	00	00	00	00	00	00	00
14	00	0			00	00	00	00	00	00	00	00
15	00	0			00	00	00	00	00	00	00	00
16	00	0			00	00	00	00	00	00	00	00

SLBUS

STATE /1

/2

/3

/4

/5

/6

SLBUS

ERROR /7

/8

ALTER

/9

/0

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.

05000  
N

SYSTEM TABLE

BASIC RAM SIZE 38B40

No.	TABLE	SIZE	OFFSET
01	SYSTEM	10000	
02	TOOL	2800	10000
03	SPARE	1200	12800
04	MONI	3800	13A00
05	PRGRM	200000	17200
06	HPCC	10000	217200
07		0	0
08		0	0
09		0	0
10		0	0

EXT. RAM SIZE 80000

No.	TABLE	SIZE	OFFSET
11		0	0
12		0	0
13		0	0
14		0	0
15		0	0
16		0	0
17		0	0
18		0	0
19		0	0
20		0	0

FANUC val. SRAM:1F0000H(2BE680H~1F) EXT. SRAM:0H(0H~1F)

SEIKI val. SRAM:541980H(2BE680H~1F) EXT. SRAM:0H(0H~1F)

BASICRAM

SIZE /1

EXT. RAM

SIZE /2

TABLE

/3

/4

SYSTEM

/5

/6

/7

/8

/9

/0

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 consist 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 got 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  ,  ,  ,  ,  and  to display Screen 1.

PMC CONTROL SYSTEM MENU										MONIT RUN												
SELECT ONE OF FOLLOWING SOFT KEYS																						
<div style="display: flex; justify-content: space-between;"> <div> <p>PMCLAD : DYNAMIC LADDER DISPLAY</p> <p>PMCDGN : DIAGNOSIS FUNCTION</p> <p>PMCPRM : PARAMETER(T/C/K/D)</p> <p>RUN/STOP : RUN/STOP SEQUENCE PROGRAM</p> <p>EDIT : EDIT SEQUENCE PROGRAM</p> <p>I/O : I/O SEQUENCE PROGRAM</p> <p>SYSPRM : SYSTEM PARAMETER</p> <p>MONIT : PMC MONITOR</p> </div> <div style="border-left: 1px solid black; padding-left: 10px; margin-left: 20px;"> <p>(※)</p> </div> </div>																						
<table border="1" style="width: 100%; border-collapse: collapse; margin-top: 10px;"> <tr> <td style="width: 10%;"></td> <td style="width: 10%;">STPSEQ</td> <td style="width: 10%;">PMCDGN</td> <td style="width: 10%;">PMCPRM</td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;">STOP</td> <td style="width: 10%;">EDIT</td> <td style="width: 10%;">I/O</td> <td style="width: 10%;">SYSPRM</td> <td style="width: 10%;">MONIT</td> <td style="width: 10%;"></td> </tr> </table>												STPSEQ	PMCDGN	PMCPRM			STOP	EDIT	I/O	SYSPRM	MONIT	
	STPSEQ	PMCDGN	PMCPRM			STOP	EDIT	I/O	SYSPRM	MONIT												



Screen 1

(※)

Note 1) F1 may be displayed "PMCLAD."

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

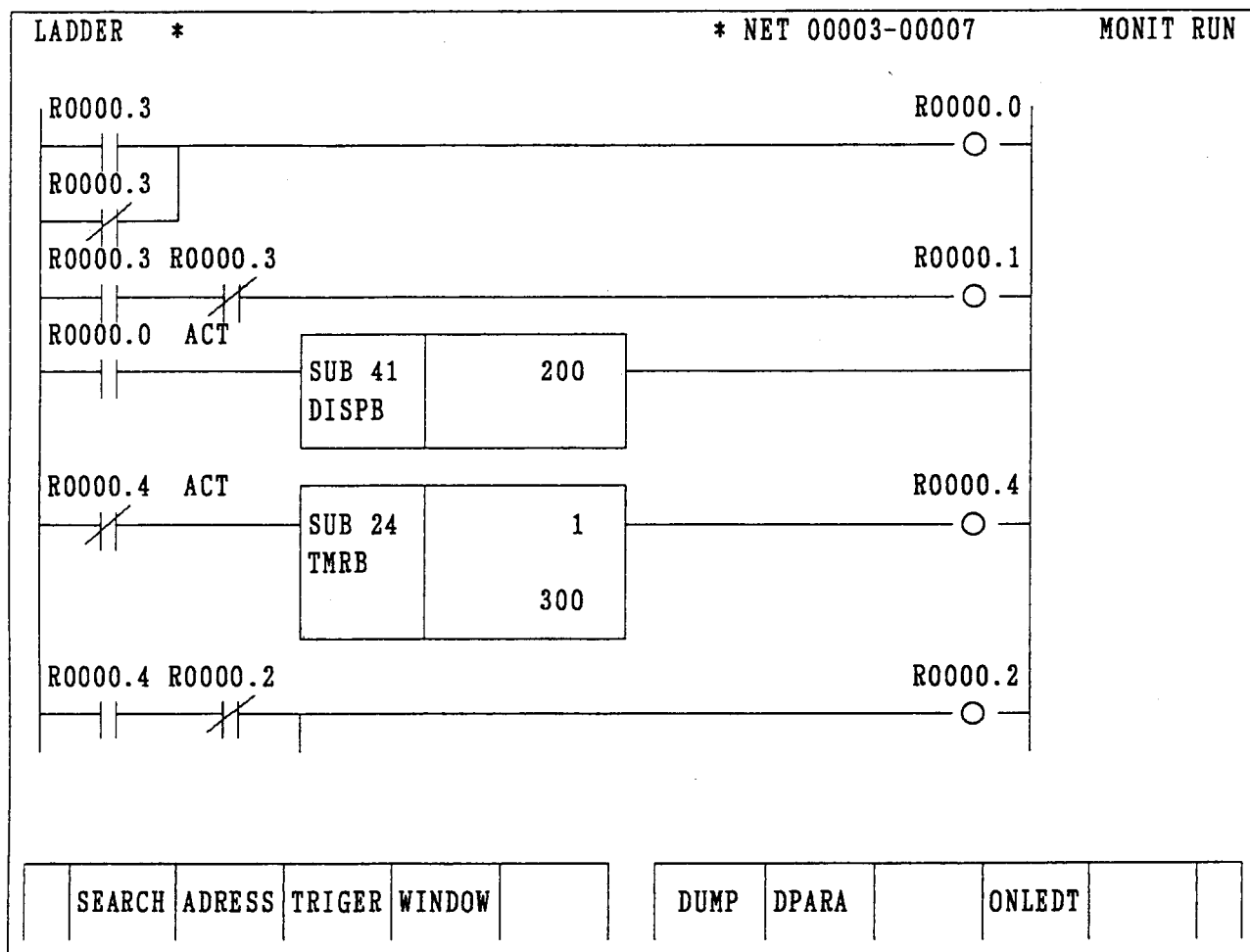
Push F1/STPSEQ to display Screen 2.

PCLAD <<MAIN>>		PROGRAM: (				)				MONIT RUN	
LEVEL1		<div style="background-color: black; color: black; display: inline-block; padding: 0 10px;">LEVEL2</div>									
	UP	DOWN	TIME	P_ADRS	ZOOM						

Screen 2

If you select LEVEL or LEVEL2 with the cursor and press **F5/ZOOM** , Screen 3 (Ladder Live Wire Display screen) will be displayed.

Note) LEVEL is a high-speed ladder and LEVEL2 is a normal ladder.



Screen 3

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

	TOP	BOTTOM	SRCH	W-SRCH	N-SRCH	F-SRCH						
--	-----	--------	------	--------	--------	--------	--	--	--	--	--	--

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 **ALTER** .

To return to the screen preceding, push **OPER/MAINT** .

## 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:

- Write key needs to be included. (Not required while in background editing.)
- Mode must be Edit. (Not necessary during background editing.)
- It must not be while auto operation start (or hold). (Not necessary during background editing.)
- 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'.)

PROGRAM (AUTO)										O5000 N
①	O5000 ;									M05 ;
	G140 G130 ;									N2 T1011100 ;
	(RUNNING PROGRAM MAIN) ;									M03 ;
	G28 U0 ;									G00 Z150.0 ;
	G28 V0 ;									X70.0 S15000 ;
	G28 W0 ;									G02 X130.0 Z120.0 I30.0
②	M50 S360 ;									F0.5 ;
③	M98 P5003 L3 ;									G03 X190.0 Z90.0 R30.0
	N1 T010100 M40 S20 ;									G00 G28 U0 W0 ;
	M08 ;									G30 U0 V0 P3 ;
	G00 Z200.0 M03 ;									M05 ;
	G00 X120.0 Z150.0 ;									T010100 ;
④	G01 Z100.0 F12.0 ;									M30 ;
	G00 Z200.0 M41 S3600 ;									N3 ;
	G01 X150.0 Z100.0 F0.3 ;									G01 X170.0 Z100.0 F0.3 ;
	G00 G28 U0 W0 ;									G01 X70.0 ;
	G30 U0 V0 W0 P3 ;									G00 G28 U0 ;
Date >										
CHANGE WINDOW/1	BACK GRD EDIT /2	RANGE EDIT /3	PROGRAM COPY /4	WORD CONVT /5	CANNED CYCLE /6	PROGRAM LIST /7	MACRO VAL /8	/9	/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
<b>O</b> <b>0</b> <b>0</b> <b>0</b> <b>1</b>	1	O 0 0 0 1 ;
<b>O</b> <b>2</b>	2	O 2 ;

## 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 .
- 2) Input the program number subject to deletion.
- 3) Push .
- 4) "Delete? Y-Yes N-No" appears.
- 5) Push  for deletion or any other key except  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 .
- 2) Push .
- 3) "ODF OK? Y-Yes N-No" appears.
- 4) Push  to perform ODF or any other key except  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 p program under Display Inhibit cannot be searched.

- 1) Input .
- 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 .
- 2) To search a number larger than the one in display, push .
- To search a number larger 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 .
- 2) Input the program number subject to copying.
- 3) Push .
- 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 .
- 3) Input the program number to be changed.
- 4) Push .
- 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  or , page is changed. The cursor appears at the beginning of the page.
- (4) Moving to Beginning of Program  
With  and, then,  pushed, the cursor is moved to the program head.
- (5) Moving to End of Program  
With  and, then,  pushed, the cursor is moved to the program end.

### 12-9-2 Edit Operation

Use, at the cursor position, , , and  for editing.

- (1)   
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)   
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)   
The part inversely displayed by the cursor is deleted.

Note 1) When power is cut off while the warning alarm "[742] IN PROGRAM EDIT" is

displayed, editing is made invalid.

→ See 13-10 PROGRAM STORE.

Note 2) 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

As 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 identified as being differentiated between those with a decimal point and those without. Note, therefore, that "X1." and "X1000" both of which indicating the same shifting amount, are differentiated. "X0." and "X0", however, are not separated one from the other.

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 programs. 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.

PROGRAM (EDIT)										O5000 N									
<div style="border: 1px solid black; padding: 5px;"><pre>O5000 ; G140 G130 ; (RUNNING PROGRAM MAIN) ; G28 U0 ; G28 V0 ; G28 W0 ; M50 S360 ; /M98 P5003 L3 ; N1 T010100 M40 S20 ; M08 ; G00 Z200.0 M03 ; G00 X120.0 Z150.0 ; G01 Z100.0 F12.0 ; G00 Z200.0 M41 S3600 ; G01 X150.0 Z100.0 F0.3 ; G00 G28 U0 W0 ; G30 U0 V0 W0 P3 ;</pre></div>										<pre>BG PROGRAM O1112 ; G28 U0 V0 W0 H0 ; G00 X100.0 ; Y100.0 ; Z100.0 ; M01 ; %</pre>									
Data > _																			
CHANGE WINDOW /1	BACK GRD EDIT /2	RANGE EDIT /3	PROGRAM COPY /4	WORD CONVT /5	CANNED CYCLE /6	PROGRAM LIST /7	MACRO VAL /8	/9	/0										

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/FOR GRD 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.

BG PROGRAM								O5000 N	
<pre>O1112 ; G28 U0 V0 W0 H0 ; G00 X100.0 ; Y100.0 ; Z100.0 ; M01 ; %</pre>									
Data > _									
CHANGE WINDOW/1	FORE GRD EDIT /2	RANGE EDIT /3	PROGRAM COPY /4	WORD CONVT /5	CANNED CYCLE /6	PROGRAM LIST /7	MACRO VAL /8	/9	/0

- Ordinary edit is differentiated from background edit by its title.  
Ordinary edit used "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.

O5000 N									
PROGRAM (EDIT)									
O5000 ; G140 G130 ; (RUNNING PROGRAM MAIN) ; G28 U0 ; G28 V0 ; G28 W0 ; M50 S360 ; /M98 P5003 L3 ; N1 T010100 M40 S20 ; M08 ; G00 Z200.0 M03 ; G00 X120.0 Z150.0 ; G01 Z100.0 F12.0 ; G00 Z200.0 M41 S3600 ; G01 X150.0 Z100.0 F0.3 ; G00 G28 U0 W0 ; G30 U0 V0 W0 P3 ;					M05 ; N2 T1011100 ; M03 ; G00 Z150.0 ; X70.0 S15000 ; G02 X130.0 Z120.0 I30.0 F0.5 ; G03 X190.0 Z90.0 R30.0 G00 G28 U0 W0 ; G30 U0 V0 P3 ; M05 ; T010100 ; M30 ; N3 ; G01 X170.0 Z100.0 F0.3 ; G01 X70.0 ; G00 G28 U0 ;				
Data > _									
CHANGE WINDOW/1	BACK GRD EDIT /2	EXIT /3	RANGE SET /4	RANGE STORE /5	STORE-> INSERT/6	RANGE DELETE/7	DELETE-> INSERT/8	/9	/0

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

(3) Canceling 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, canceling 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 CONV**, 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.

O5000 N									
<div style="display: flex; justify-content: space-between;"> <div style="width: 48%;"> <p>PROGRAM (EDIT)</p> <p>O5000 ;</p> <p>G140 G130 ;</p> <p>(RUNNING PROGRAM MAIN) ;</p> <p>G28 U0 ;</p> <p>G28 V0 ;</p> <p>G28 W0 ;</p> <p>M50 S360 ;</p> <p>/M98 P5003 L3 ;</p> <p>N1 T010100 M40 S20 ;</p> <p>M08 ;</p> <p><b>G00</b> Z200.0 M03 ;</p> <p>G00 X120.0 Z150.0 ;</p> <p>G01 Z100.0 F12.0 ;</p> <p>G00 Z200.0 M41 S3600 ;</p> <p>G01 X150.0 Z100.0 F0.3 ;</p> <p>G00 G28 U0 W0 ;</p> <p>G30 U0 V0 W0 P3 ;</p> </div> <div style="width: 48%;"> <p>M05 ;</p> <p>N2 T1011100 ;</p> <p>M03 ;</p> <p>G00 Z150.0 ;</p> <p>X70.0 S15000 ;</p> <p>G02 X130.0 Z120.0 I30.0</p> <p>F0.5 ;</p> <p>G03 X190.0 Z90.0 R30.0</p> <p>G00 G28 U0 W0 ;</p> <p>G30 U0 V0 P3 ;</p> <p>M05 ;</p> <p>T010100 ;</p> <p>M30 ;</p> <p>N3 ;</p> <p>G01 X170.0 Z100.0 F0.3 ;</p> <p>G01 X70.0 ;</p> <p>G00 G28 U0 ;</p> </div> </div>									
Data > _					conversion word > X100.0				
for search					convert word > X150.0				
BLANKET ALL /1	BLANKET BEFORE/2	BLANKET AFTER /3	/4	EXIT /5	/6	/7	/8	/9	/0

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 . Not to convert, push any other except .
- 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 . If not, push any other except .

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 .

## 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.

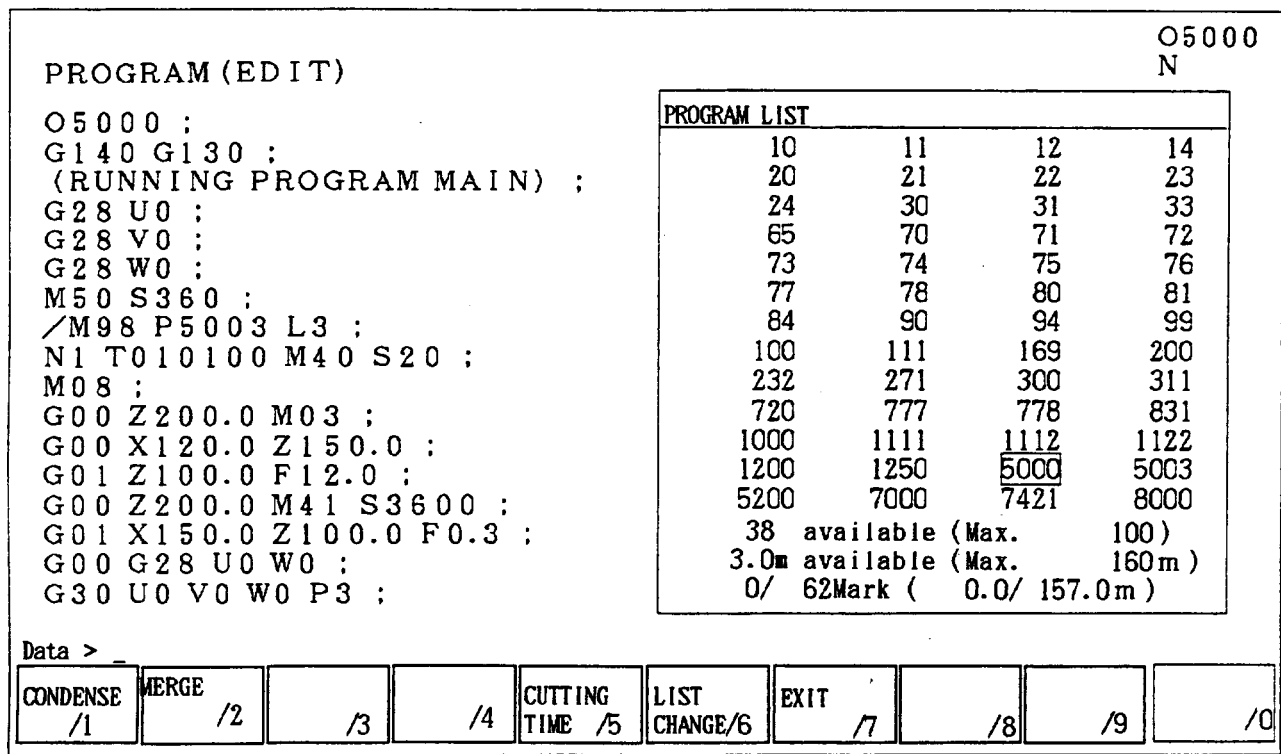


Fig. 13-0 Program List

#### 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 .

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.

PROGRAM (EDIT)

PROGRAM LIST			
1	2	3	4
5	6	7	8
10	11	12	14
20	21	22	23
24	30	31	33
65	70	71	72
73	74	75	76
77	78	80	81
84	90	94	99
100	111	169	200
232	271	300	311
720	777	778	831
1000	1111	1112	1122
38 available (Max.		100個)	
3.0m available (Max.		80 m)	
0/ 62Mark (		0.0/ 77.0 m)	

05000  
N

BG PROGRAM

```

O1112 ;
G28 U0 V0 W0 H0 ;
G00 X100.0 ;
Y100.0 ;
Z100.0 ;
M01 ;
%
```

Data > \_

CONDENSE	MERGE				LIST	EXIT			
/1	/2	/3	/4	/5	CHANGE /6	/7	/8	/9	/0

Fig. 13-1 Program List for Background Edit.

PROGRAM (EDIT)

```

O5000 ;
G140 G130 ;
(RUNNING PROGRAM MAIN) ;
G28 U0 ;
G28 V0 ;
G28 W0 ;
M50 S360 ;
/M98 P5003 L3 ;
N1 T010100 M40 S20 ;
M08 ;
G00 Z200.0 M03 ;
G00 X120.0 Z150.0 ;
G01 Z100.0 F12.0 ;
G00 Z200.0 M41 S3600 ;
G01 X150.0 Z100.0 F0.3 ;
G00 G28 U0 W0 ;
G30 U0 V0 W0 P3 ;
```

05000  
N

PROGRAM LIST (DETAILS)	
311 (	) 1.0m
720 (B=#632	) 1.0m
777 (	) 1.0m
778 (S1 CHECK	) 1.0m
831 (	) 1.0m
1200 (	) 1.5m
1250 (	) 1.0m
5000 (	) 1.0m
5003 (	) 1.0m
5200 (	) 1.0m
7000 (	) 1.5m
7421 (	) 1.0m
8000 (	) 2.0m
38 available (Max.	100個)
3.0m available (Max.	80 m)
0/ 62Mark (	0.0/ 77.0 m)

Data > \_

CONDENSE	MERGE				LIST	EXIT			
/1	/2	/3	/4	/5	CHANGE /6	/7	/8	/9	/0

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/CONDENSE** .
  - 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.

DATA IN/OUT					O5000 N																																							
IN/OUT DATA		PROGRAM PRAME SYSTEM	WORK COMMON	TOOL ALL NC	<b>PROGRAM LIST</b> <table style="width: 100%; border-collapse: collapse;"> <tr><td style="width: 33%;">1</td><td style="width: 33%;">2</td><td style="width: 33%;">3</td></tr> <tr><td>4</td><td>5</td><td>6</td></tr> <tr><td>7</td><td>8</td><td>10</td></tr> <tr><td>11</td><td>12</td><td>14</td></tr> <tr><td>20</td><td>21</td><td>22</td></tr> <tr><td>23</td><td>24</td><td>30</td></tr> <tr><td>31</td><td>33</td><td>65</td></tr> <tr><td>70</td><td>71</td><td>72</td></tr> <tr><td>73</td><td>74</td><td>75</td></tr> <tr><td>76</td><td>77</td><td>78</td></tr> <tr><td>80</td><td>81</td><td>84</td></tr> <tr><td>90</td><td>94</td><td>99</td></tr> <tr><td>100</td><td>111</td><td>169</td></tr> </table> <div style="font-size: small; margin-top: 5px;"> 38 available (Max. 100個)  3.0m available (Max. 80m)  0/ 62Mark ( 0.0/ 77.0m ) </div>	1	2	3	4	5	6	7	8	10	11	12	14	20	21	22	23	24	30	31	33	65	70	71	72	73	74	75	76	77	78	80	81	84	90	94	99	100	111	169
1	2	3																																										
4	5	6																																										
7	8	10																																										
11	12	14																																										
20	21	22																																										
23	24	30																																										
31	33	65																																										
70	71	72																																										
73	74	75																																										
76	77	78																																										
80	81	84																																										
90	94	99																																										
100	111	169																																										
IN	DEVICE	● RS232C CARD	READER AUX. 1	— AUX. 2																																								
	B. RATE	● 4800 9600	1200	2400																																								
	STOP BIT	● 2	1																																									
OUT	DEVICE	RS232C CADE	— AUX. 1	● PUNCH AUX. 2																																								
	B. RATE	● 4800 9600	1200	2400																																								
	STOP BIT	● 2	1																																									
	CODE	● ISO	EIA																																									
PROGRAM > _																																												
INPUT /1	OUTPUT /2	VERIFY /3	STOP /4	CALL SELECT/5	SETTING /6	LIST CHANGE/7	CHECK ON/OFF/8	DON /9																																				

### (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 of 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, output 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 of 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 .

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 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 by 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 2 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.

01	12	23	34	45	56	67	78	89	9A	AB	BC	CD
EF	01	23	45	67	89	9A						

↑

Receiving data are displayed without change. (input only)

DR[ON] ← Circuit connection status

ER[FOP] ← Error status

F: framing error

O: overrun error

P: parity error

### 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

PRA0134 : DNC input 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	1 2 0 0
11	2 4 0 0
13	4 8 0 0
15	9 6 0 0

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 No. : 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 centimeters.

PRA0138 : Feed length

PRA-139 : Program interval length

### 14-5 Call Select (Optional)

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) 0123 (S: 01234, 012) ;  
0123 (MAIN S: 01234, 012) ;

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 END** and end the Don-Don FD I/O screen.

SEIKI DON				O5000 N																																								
IN/OUT:PROGRAM		DNC FILE:PROGRAM.DAT																																										
Disk: 2HD(1.44M)		Free : 2035.0m		Files: 90																																								
Info: PC05.DAT		1.5m 93/09/14 WpRp ISO		File: 102( 1140.5m)																																								
				Mark: 0( 0m)																																								
PASRT .DAT 5.0m 93/09/14 PSCND .DAT 1.5m 93/09/14 P003 .DAT 1.5m 93/09/14 P004 .DAT 1.5m 93/09/14 P006 .DAT 1.5m 93/09/14 P007 .DAT 1.5m 93/09/14 P008 .DAT 1.5m 93/09/14 P009 .DAT 1.5m 93/09/14 P010 .DAT 1.5m 93/09/14 P011 .DAT 1.5m 93/09/14 P012 .DAT 1.5m 93/09/14 P013 .DAT 1.5m 93/09/14 P014 .DAT 1.5m 93/09/14			<table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th colspan="3" style="text-align: left;">PROGRAM LIST(DETAILS)</th> </tr> </thead> <tbody> <tr><td></td><td>1 (COMMENT MAX 16 )</td><td>0.5 m</td></tr> <tr><td></td><td>2 ( )</td><td>0.5 m</td></tr> <tr><td>*</td><td>3 ( )</td><td>1.0 m</td></tr> <tr><td>*</td><td>4 ( )</td><td>18.0 m</td></tr> <tr><td></td><td>5 ( )</td><td>1.5 m</td></tr> <tr><td></td><td>6 ( )</td><td>1.5 m</td></tr> <tr><td></td><td>7 ( )</td><td>7.0 m</td></tr> <tr><td></td><td>8 ( )</td><td>2.0 m</td></tr> <tr style="border: 2px solid black;"><td></td><td>5000 ( )</td><td>1.0 m</td></tr> <tr><td>*</td><td>8000 ( )</td><td>3.0 m</td></tr> <tr><td colspan="3">90 available (Max. 100)</td></tr> <tr><td colspan="3">40.0m available (Max. 80)</td></tr> </tbody> </table>			PROGRAM LIST(DETAILS)				1 (COMMENT MAX 16 )	0.5 m		2 ( )	0.5 m	*	3 ( )	1.0 m	*	4 ( )	18.0 m		5 ( )	1.5 m		6 ( )	1.5 m		7 ( )	7.0 m		8 ( )	2.0 m		5000 ( )	1.0 m	*	8000 ( )	3.0 m	90 available (Max. 100)			40.0m available (Max. 80)		
PROGRAM LIST(DETAILS)																																												
	1 (COMMENT MAX 16 )	0.5 m																																										
	2 ( )	0.5 m																																										
*	3 ( )	1.0 m																																										
*	4 ( )	18.0 m																																										
	5 ( )	1.5 m																																										
	6 ( )	1.5 m																																										
	7 ( )	7.0 m																																										
	8 ( )	2.0 m																																										
	5000 ( )	1.0 m																																										
*	8000 ( )	3.0 m																																										
90 available (Max. 100)																																												
40.0m available (Max. 80)																																												
INPUT /1	OUTPUT /2	VERIFY /3	STOP /4	DON END /5	SETTING /6																																							
		FILE LIST /7	DIR. READ /8	-next- /9	/0																																							
FORMAT /1	MARK /2	ALL MARK /3	STOP /4	VIEW /5	VIEW 0 No. /6																																							
		SELECT DNC /7	/8	-next- /9	/0																																							

- 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)

(Not available with  $\Sigma 10$ )

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

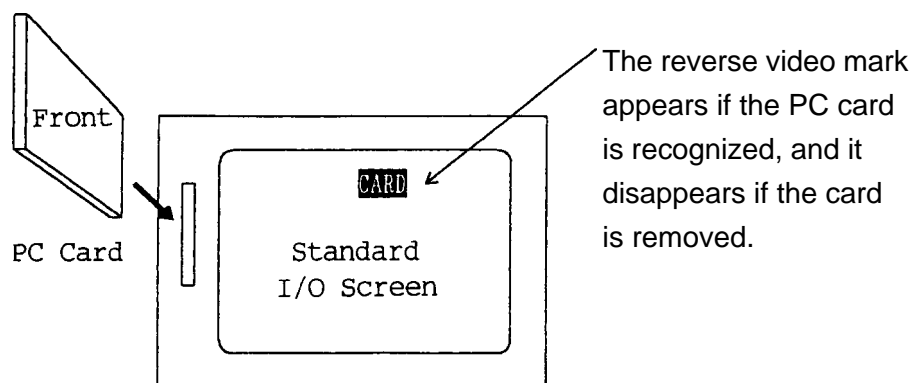
Available PC cards are 256 KB, 512 KB, 1 MB, 2 MB, and 4 MB S-RAM cards compliant with the PCMCIA or JEIDA Standards. The flush memory cards (FCs) and ATA Standards memory cards are not available.

### 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 order 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.



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	OUTPUT	VERIFY	STOP	CALL	SETTING	LIST	CHECK	DON	
/1	/2	/3	/4	SELECT/5	/6	CHANGE/7	ON/OFF/8	/9	/0



INPUT	OUTPUT	VERIFY	STOP	CALL	SETTING	LIST	CHECK	DON	PCCARD
/1	/2	/3	/4	SELECT/5	/6	CHANGE/7	ON/OFF/8	/9	INOUT /0

Inserting the PC card enables a card menu.

- ① 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-323C communications.

O 5 0 0 0  
N

DATA IN/OUT

IN/OUT DATA	PROGRAM	WORK	TOOL
	PARAME	COMMON	

PC-CARD Free

PROGRAM LIST

1	2	3
4	5	1 0
1 1	2 0	

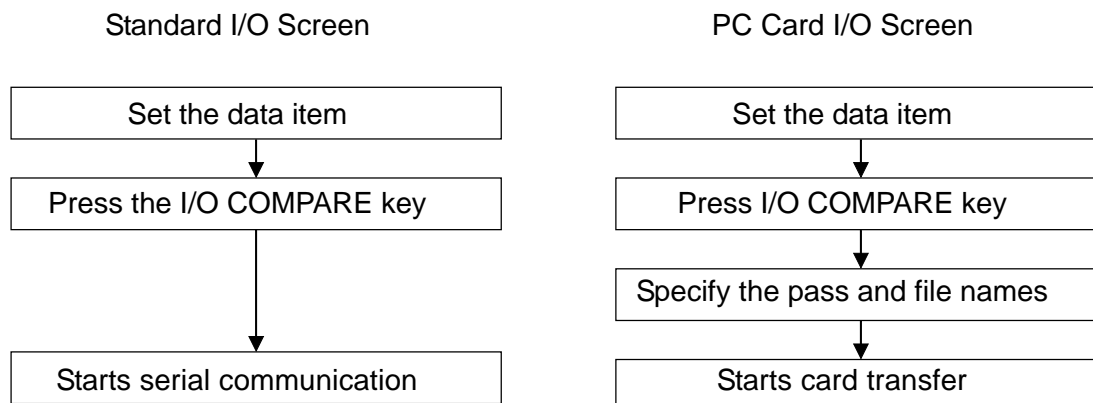
PROGRAM > \_

INPUT	OUTPUT	VERIFY	STOP	CALL	SETTING	LIST	DIR		PCCARD
PCCARD/1	PCCARD/2	PCCARD/3	/4	SELECT/5	/6	CHANGE/7	PCCARD/8	/9	END /0

PC Card I/O Screen

This PC Card I/O screen provides the same operating environment as usual.

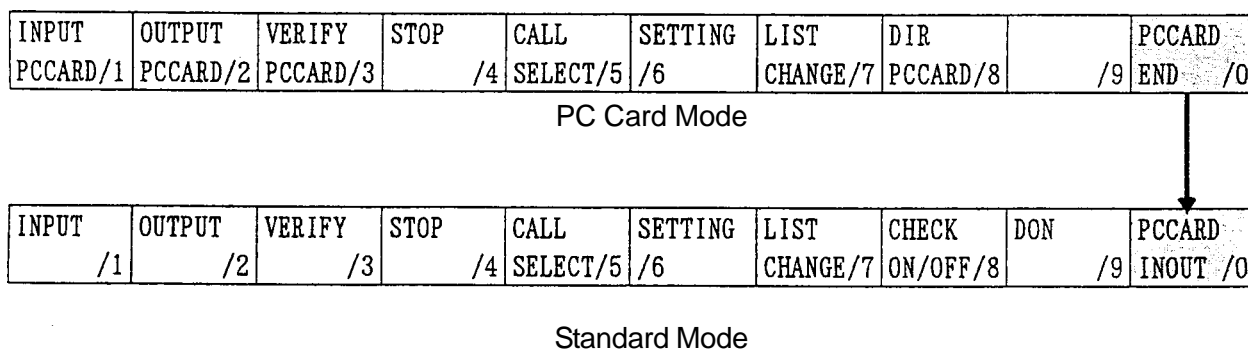
What has been newly added to the screen is the operation to specify the PC card path and file names.



## 15-4 Ending PC Card Operation

### 15-4-1 transitioning to the Standard I/O Screen

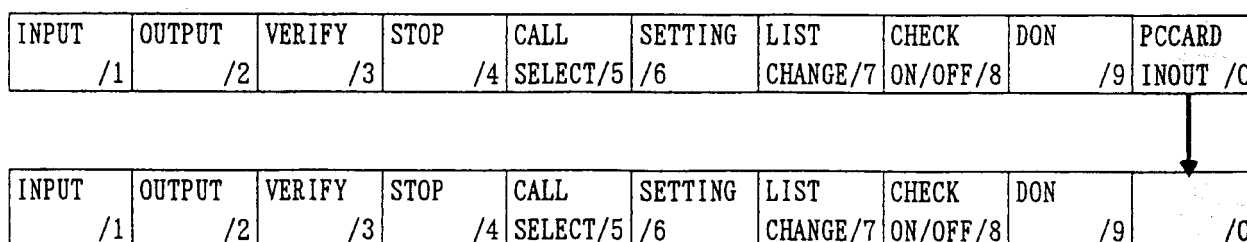
When exiting the PC card mode, press F10/PCCARD END . You will be returned to the Standard I/O screen of the RS-232C mode.



### 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.

B:¥GROUP2		Free	m( )
B:¥		00001	2045 98-12-24 16:12:00
--GPOUP1		00003	1233 98-12-25 15:13:12
--GROUP2		01000	123455 98-07-07 09:15:54
--GROUP21			
Path Area		File Area	

EXIT	/1	/2	/3	/4	/5	/6	/7	/8	/9	/0
------	----	----	----	----	----	----	----	----	----	----

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.

B:¥GROUP2		Free	m( )
B:¥		00001	2045 98-12-24 16:12:00
--GPOUP1		00003	1233 98-12-25 15:13:12
--GROUP2		01000	123455 98-07-07 09:15:54
--GROUP21			
Path Area		File Area	

EXIT	/1	/2	/3	/4	/5	/6	/7	/8	/9	/0
------	----	----	----	----	----	----	----	----	----	----

- 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 Step ④)
- 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.

B:¥GROUP2		Free	m( )
B:¥		00001	2045 98-12-24 16:12:00
--GPOUP1		00003	1233 98-12-25 15:13:12
--GROUP2		01000	123455 98-07-07 09:15:54
--GROUP21			
<div style="border: 1px solid black; width: 200px; height: 30px; margin: 0 auto; display: flex; align-items: center; justify-content: center;">             filename=           </div>			

EXIT	/1	/2	/3	/4	/5	/6	/7	/8	/9	/0
------	----	----	----	----	----	----	----	----	----	----

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.

B:¥GROUP2		Free	m( )
B:¥		00001	2045 98-12-24 16:12:00
--GPOUP1		00003	1233 98-12-25 15:13:12
--GROUP2		01000	123455 98-07-07 09:15:54
--GROUP21			
Path Area		File Area	
EXIT		/1	/2
		/3	/4
		/5	/6
		/7	/8
		/9	/0

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	OUTPUT	VERIFY	STOP	CALL	SETTING	LIST	DIR	DON	PCCARD
PCCARD/1	PCCARD/2	/3	/4	SELECT/5	/6	CHANGE/7	PCCARD/8	/9	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	OUTPUT	VERIFY	STOP	CALL	SETTING	LIST	DIR	DON	PCCARD
/1	/2	/3	/4	SELECT/5	/6	CHANGE/7	PCCARD/8	/9	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:¥

-AAAAAAA

-BBBBBBB

-CCCCCCC

-DDDDDDD

-EEEEEEE

-FFFFFFF

-GGGGGGG

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:¥."

B:¥GROUP2	Free	m( )
B:¥	Free	m( )
B:¥	00001	2045 98-12-24 16:12:00
--GPOUP1	00003	1233 98-12-25 15:13:12
--GROUP2	01000	123455 98-07-07 09:15:54
--GROUP3		
Path Area	File Area	
EXIT /1	MAKDIR /2	RENAME /3
	DELETE /6	/0

- ③ Press the **F3/MAKDIR** key.
- ④ The window appears, where prompts you to input a new directory name.

B:¥GROUP2	Free	m( )
B:¥	00001	2045 98-12-24 16:12:00
--GPOUP1	00003	1233 98-12-25 15:13:12
--GROUP2	01000	123455 98-07-07 09:15:54
--GROUP3		
Dir.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.

B:¥GROUP2	Free	m( )
B:¥		
--GPOUP1		
--GROUP2		
--NEWSUB		
--GROUP3		

Insertion  
Example

## 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**.

B:¥GROUP2				Free		m( )	
B:¥							
--GPOUP1							
--GROUP2							
--GROUP3							
Path Area				File Area			
EXIT	/1	/2	MAKDIR	RENAME	DELETE	/7	/8
	/1	/2	/3	/4	/5	/6	/0

### (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**.

B:¥GROUP2				Free		m( )	
B:¥				00001 2045 98-12-24 16:12:00			
--GPOUP1				00003 1233 98-12-25 15:13:12			
--GROUP2				01000 123455 98-07-07 09:15:54			
--GROUP3							
Path Area				File Area			
EXIT	/1	/2	MAKDIR	RENAME	DELETE	/7	/8
	/1	/2	/3	/4	/5	/6	/0



### 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.

B:¥GROUP2		Free	m( )
B:¥	00001	2045	98-12-24 16:12:00
--GPOUP1	00003	1233	98-12-25 15:13:12
--GROUP2	01000	123455	98-07-07 09:15:54
--GROUP3			
Path Area		File Area	

EXIT	MAKDIR	RENAME	DELETE
/1	/2	/3	/4
/5	/6	/7	/8
/9	/0		

- ③ Press **F4/RENAME**.
- ④ The window appears, which prompts you to input a renewed directory name.

B:¥GROUP2		Free	m( )
B:¥	00001	2045	98-12-24 16:12:00
--GPOUP1	00003	1233	98-12-25 15:13:12
--GROUP2	01000	123455	98-07-07 09:15:54
--GROUP3			
Dir.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.

B:¥GROUP2		Free	m( )
B:¥			
--GPOUP1			
--NEWSUB			
--GROUP3			

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.

B:¥GROUP2		Free	m( )
B:¥		00001	2045 98-12-24 16:12:00
--GPOUP1		00003	1233 98-12-25 15:13:12
--GROUP2		01000	123455 98-07-07 09:15:54
--GROUP3			
Path Area		File Area	

EXIT	/1	/2	MAKDIR	/3	RENAME	/4	/5	DELETE	/6	/7	/8	/9	/0
------	----	----	--------	----	--------	----	----	--------	----	----	----	----	----

PC Card Directory Screen

- ④ Press **F4/RENAME**.
- ⑤ The window appears, which prompts you to input a renamed file name.

B:¥GROUP2		Free	m( )
B:¥		00001	2045 98-12-24 16:12:00
--GPOUP1		00003	1233 98-12-25 15:13:12
--GROUP2		01000	123455 98-07-07 09:15:54
--GROUP3			
Filename=			

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.

B:¥GROUP2		Free	m( )
B:¥		00001	2045 98-12-24 16:12:00
--GPOUP1		NEWNAME	1233 98-12-25 15:13:12
--GROUP2		01000	123455 98-07-07 09:15:54
--GROUP3			

## 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.
"mkdir 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 Functions**

**I. Tool Life Management**

**II. Cutting Monitor**

**III. STAF**

**IV. UUP Function**

**V. External Measuring Function**

## 4. INDIVIDUAL PROGRAMS

### I. Tool Life Management

#### 1. Outline of tool life management

Tool life management counts the tool use value in accordance with the tool life unit set per tool. This function compares the tool use value and the setting value (I.1.2) and manages the tool life. It also meets with measurement, load monitoring, skip, and tool status setting from the external device.

#### 2. Operation of tool life management function

- Counts the life according to the life units per tool. If the use value coincides with the set value, the tool used will be set as the life tool and the machining continues.
- If a spare tool exists in the previous case, then the spare tool is selected automatically by the T command in the next machining process.
- When worn-out tool is detected during monitoring of the cutting process, the tool is set as worn-out tool and machining is continued.

In the next machining, if a spare tool exists, the spare tool is selected by the T command.

- When tool life management is not necessary, the function is controlled with the G codes. G131 is selected at power-on and reset time.

G130 ..... Tool life management disable

G131 ..... Tool life management enable

#### 3. Screen explanations

Pressing F3/TOOL or F9/LIFE SPARE displays the "Tool (Life + Spare)" screen.

If the tool is defective (life is reached), a '\*' mark appears before the tool T. If the tool is registered as a priority tool, then 'P' appears before the tool T.

Tool Life							SpareTool	
Tool	Type	Tool name	Use	Setting	Units	Status	Reference	Spare
T01		Face rough	12	20	min.	Use	T0100	T0700 T0900
T02								T1100
T03		Face Finish	43	100	length	Use	*T0400	T1200 PT1400
*T04		Drill 1	110	80	hole	Out of life	T0200	

- Type ..... Determine the type of tool by pressing the F2/Type setting key in the type setting window.
- Tool name ..... Display and setting of tool name (does not affect the tool life management function)  
Press the F3/Tool Name key, and determine the name and settings in the tool name setting window.

- Use / Setting ..... Display and setting of life setting value and use value of each tool. Move the cursor, input the numeric value and set using the INPUT key.
- Units ..... Set the units of life management for each tool.
 

Minutes	: Cutting rate time (minutes)	} (Note) Only Times is available for other then the 10L.
Times	: Number of calls	
Length	: Cutting distance (M)	
Holes	: Number of drilling fixes cycles	
Press the F4/Units key and determine the settings in the units setting window.		
- Status ..... Display the status of each tool.
 

To clear the status, 0 set the use value or press the F7/Data Clear key.

  - Use ..... Tool being used
  - Monitor in use .... Cutting monitor is effective and tool is being used
  - Out of life ..... The tool is out of life.
  - Wear life ..... The tool is cut of wear life (While monitoring the cutting)
  - Skip ..... The tool is skipped by a skip signal.
  - Tool nose measurement NG .... The tool is found no good in tool nose measurement.
  - Work measurement NG ..... The tool is found no good in work measurement.
  - Overload error ..... An overload error occurred (load monitor).
  - No-load error ..... A no-load error occurred.
  - External error ..... An external error occurred (external breakage, etc.).
- Reference ..... Reference tool
- Spare ..... Set the same type as the reference tool. If the reference tool becomes a defective tool, tool will be selected from this type.

#### 4. Alarm

- No.780 damaged tool selected ..... Measurement NG, load NG, skip, externally defective tool recalled in the spindle.
- No.781 life tool selected ..... Life tool recalled in the spindle.

#### 5. Spare tool

- Set the spare tool group in the reference and spare tools in the Tool (Life + Spare) screen.

#### 6. Registering and deleting the spare tool

- Reference tool and spare tool have tool numbers that include the offset number.
- Registration of 1 spare tool ..... Bring the cursor by the side of the reference tool and press TOOL NUMBER INPUT.
- Registration of more than 2 tools... In the Spare Tool column, press TOOL NUMBER INPUT.
- Spare tools will be registered continuously.
- If spare tools are more than 3 in number, set the reference tool in an empty column.

- Canceling a tool ..... Set the cursor on the tool to be cancelled and press 0 INPUT.
- Registration of priority tool ..... Set the cursor on the tool to be given priority and press P INPUT. (Priority tool)
- Canceling the priority tool ..... Set the cursor on the priority tool and press C INPUT.

## 7. Selection order of spare tools

①Used tool ②Priority tool ③Unused tool at the top of the registration list④Last selected tool

\*\*\* Tool Number Specification Method \*\*\*

Method A ..... Move the cursor to the corresponding number using      .

Method B ..... Input N tool number (numeric value) and press .

## 8. Sample Program

G28U0W0;

T○○△△□□;

```

    }
G99 G01 Z50. F0.2; _____ EX1
G00 X55,;

```

```

    }
G81 Z-30. R20., R-18. _____ EX2
    F500 E3000 L0;

```

```

    }
G28U0W0M05;
M12 _____ EX3

```

Tool Life						Spare tool		
Tool	Type	Tool name	Use	Setting	Units	Status	Reference	Spare
T01		Face Rough	12	20	min.	Use	T0100	T0700 T0900
T02								T1100
*T03		Face Finish	100	100	length	Use	*T0400	T1200 PT1400
*T04		DRILL1	80	80	hole	Out of Life	T0200	
T05		O.D.Rough	0	50	times		*T0300	T1500 T1600
T06								

Fig. 1

(1) Time life ..... Available only for the  $\Sigma 10L$ .

Since the tool life of the tool 01 in Fig. 1 is set to 20 min., the sample program counts the time at “\*1” where cutting feed match. The unit of time is minutes. Since the data below the minute is stored inside, however, the fractional time is not cut off. When the tool use time comes to 20 min., the tool 01 is assumed to be out of life.

- (2) Length life ..... Available only for the  $\Sigma 10L$ .

Since the tool life of the tool 02 in Fig. 1 is set to 200 m, the sample program counts the cutting distance at “\*1” where cutting feed matches. The unit of distance is meters.

Since the data below the meter is stored inside, however, the fractional distance is not cut off. When the cutting distance comes to 200 m, the tool 02 is assumed to be out of life.

- (3) Frequency life

Since the tool life of the tool 03 in Fig. 1 is set to 100 times, the sample program counts the frequency at “\*2” of M12.

When the frequency comes to 100 times, the tool 03 is assumed to be out of life.

- (4) No.-of-holes life ..... Available only for the  $\Sigma 10L$ .

Since the tool life of the tool 04 in Fig. 1 is set to 300 holes, the number of holes to be machined is counted in the canned cycle for tapping/drilling. When the number of holes to be machined comes to 80, the tool 04 is assumed to be out of life.

## 8-1 Details of Tool Life by Times

The following describes the tool life by Times.

Select and set Times as the unit of tool life for T0100 and T0500.

[Sample Program 2]

```
T0100;
{
M12;                ← Increments the use count of T0100 by 1
T0500;
{
T0100;
{
M12;                ← Increments the use count of T0100 and T0500 by 1
```

If the tool life management function stores the used tool and detects M12, it will increment the use count of each tool by 1.

[Sample Program 3]

```
T0100;
{
T0500;
{
T0100
{
M12;                ← Increments the use count of T0100 and T0500 by 1
```

Since T0100 has been already stored, the use count for the second T0100 is incremented only by 1.



## II. Cutting Monitor

### 1. Outline of Cutting Monitor

This unit monitors the cutting load of the spindle and feed axes to prevent abnormal cutting and defective machining. Since various kinds of monitored data can be set using the auxiliary functions (M code), it can provide monitoring corresponding to machining.

### 2. Monitoring Function

- The cutting monitor monitors a program for the section between Auxiliary Function M720 and M770.
- The cutting monitor monitors the loading condition of the spindle motor and the feed axis motor, based on the setting values of M720 to M769. It detects abnormalities and controls the load.
- The main monitor in which all the monitoring functions are operative, and the sub-monitor, monitor each item. Normally, the monitoring is done at the spindle of the main monitor.
- Although the spindle and rotary tool are always monitored, the feed axis is monitored only during cutting feed.

### 3. Description of Monitoring Function

- (1) Overload detection ..... The overload detects if the overload detection value exceeds the overload detection judgement time continuously.

Spindle, rotary tool overload judgement

time ..... Parameter, 8031×0.1 sec.

X axis overload judgement time .....

Parameter, 8035×0.1 sec.

Y axis overload judgement time .....

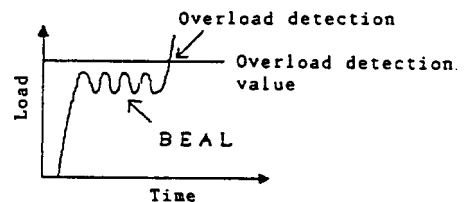
Parameter, 8039×0.1 sec.

Z axis overload judgement time .....

Parameter, 8043×0.1 sec.

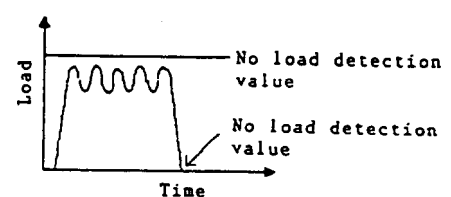
(Example)

Assume that the spindle is monitored at the load setting (See 4), overload value is 100, and the parameter 8031 is set at 30. Overload alarm is produced when state with real load of 100 or above has lasted 3 seconds or more while in working.



- (2) No load detection .....

Unless cutting load inside the monitored section exceeds the no-load value, it is judged that no cutting state has existed, where no-load is detected.



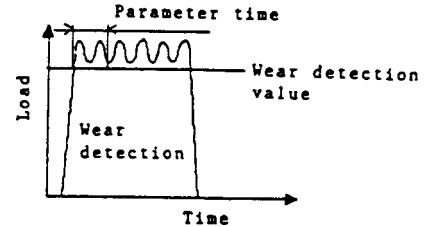
(Example)

Assume that the X axis is monitored by the load setting (See 4) and the no load value has been set at 80.

When real load is held constantly below 79 inside the monitored section, no-load alarm is produced.

### (3) Wear Detection

- (a) Loading time detection mode (Wear judgement time parameter is not equal to 0)  
The cutting load detects wear if the wear detection value exceeds the wear judgement time continuously.



Spindle, rotary tool overload judgement time .... Parameter, 8032×0.1 sec.

X axis overload judgement time ..... Parameter, 8036×0.1 sec.

Y axis overload judgement time ..... Parameter, 8040×0.1 sec.

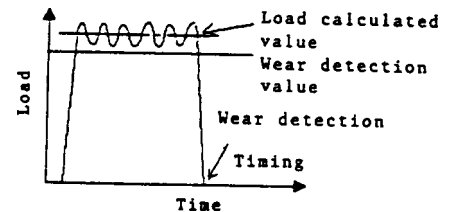
Z axis overload judgement time ..... Parameter, 8044×0.1 sec.

(Example)

Suppose that the Z axis is monitored by the load setting (See 4), wear value is 95, and the parameter 8044 is set at 50.

A tool gets worn state if real load of 96 or above is applied for 5 seconds or more while in cutting.

- (b) Load calculation detection mode  
With the wear judgment time parameter being set to 0, this mode is created.  
The cutting load value is calculated from the monitoring space.



If this value exceeds the wear detection value, then wear is detected.

(Note) If the parameter is incorrect, the maximum load value becomes the cutting load.

Parameters that specify the range of data used for calculations

Spindle / rotating tool calculation range .....Parameter 8010, 8011

X axis calculation range ..... Parameter, 8016, 8017

Y axis calculation range ..... Parameter, 8022, 8023

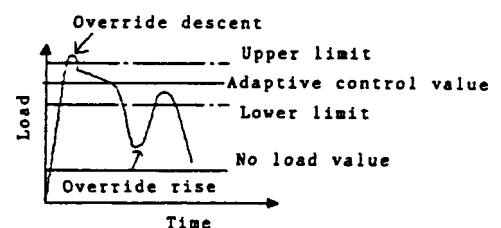
Z axis calculation range ..... Parameter, 8028, 8029

### (4) Adaptive control (Feed override control)

When the cutting load exceeds the no load value to reach the cutting condition, the feed override is controlled so that the cutting load falls between the adaptive upper limit value and the adaptive lower limit value.

If the cutting load exceeds the adaptive upper limit value (reference value) for 2 sec.

- Adaptive upper limit value ..... Reference value
- Decrease judgment time ..... Parameter, 8055×0.1 sec.
- Increase judgment time ..... Parameter, 8056×0.1 sec.
- Override distance ..... Distance of override increase/Decrease  
Parameter, 8054



- Maximum override ..... Maximum override value Parameter, 8052
- Minimum override ..... Minimum override value Parameter, 8053

(Example)

Reference value	90
No load detection value	50
Adaptive lower limit value	80
Parameter 8055	20
8056	20
8054	30
8052	150
8053	80

The settings are made as shown in the table on the left.

When the NC program is started, the cutting load exceeds 50 (no load value) and reaches the cutting condition. When this state lasts 2 seconds or more (Parameter 8056), 30% of override (Parameter 8054) is added. When further 2 seconds has passed, 30% of override is added. In this way, the cutting load exceeds the reference value or the override continues to rise until 150 (Parameter 8052) is reached.

#### 4. Load Settings

Load settings						
(M720)				1/50		
Spindle	Axis		[%]	[%]		
Actual load	0.0	-		0.0	-	
Maximum value	11.6	-		0.0	-	
Overload	15.0 (150%)	● ON	OFF	0.0 (0%)	ON	● OFF
Wear	12.0 (120%)	● ON	OFF	0.0 (0%)	ON	● OFF
Reference value	10.0	-		0.0	-	
Adaptive lower limit	9.0 (90%)	ON	OFF	ON = Monitors OFF= Does not monitor		
No load	4.0 (40%)	ON	OFF			

- Monitoring axis ..... Display and set the main monitoring axis. Move the cursor to the axis, set by {S.X.Y.Z.R} INPUT, and cancel by pressing C.  
The same axis cannot be selected in the main monitoring axis and sub-monitoring axis.
- Function selection ..... Display and set the monitoring function. Move the cursor to “ON” or “OFF” and set by pressing INPUT.
- Monitoring data ..... Display and set the data for each monitoring function.  
Move the cursor, input data, and set by pressing INPUT.
- Reference value ..... Display and set the cutting load reference value of tool.  
Ampere display can be set by setting “1” for PRA8004 #6 and a 100 % current value for PRA8070 through PRA8074.

(1) Conditions for executing each function

		Wear detection function			
	Over load detection	Time	Calculation	Adaptive control	No load detection
Overload detection function Setting value	● ●	● ●	● ●	● ●	● ●
Wear detection function Setting value		● ●	● ●		
Adaptive control function Setting value		×	×	● ●	
No load detection function Setting value				●	● ●

- Wear detection and adaptive control cannot be executed simultaneously.
- If both the detection functions are effective, adaptive control takes priority.

(Example)

When adaptive control is used ..... Overload detection value setting is 'ON.'

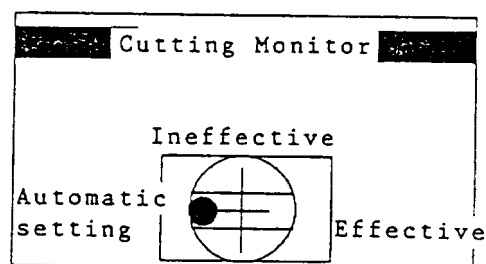
Adaptive control value setting is 'ON.'

No load detection value setting

## 5. Monitoring Operation

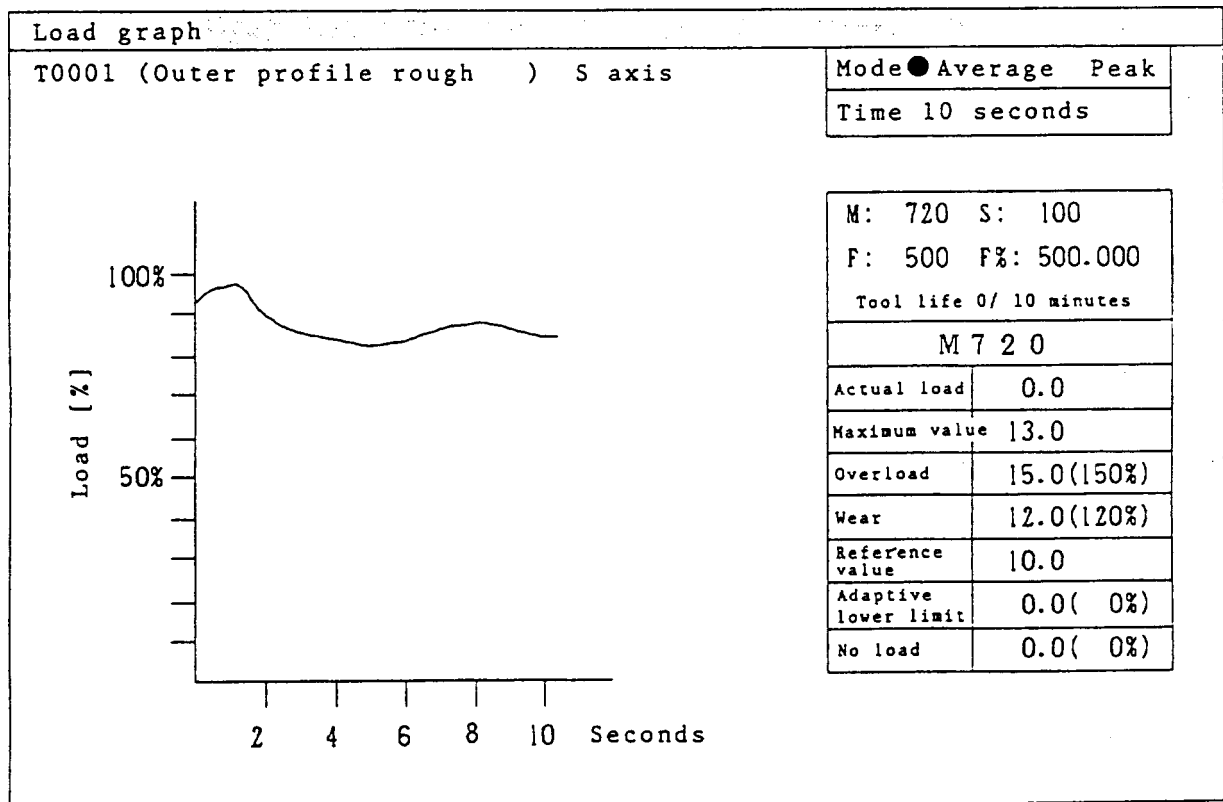
Basic conditions

- Memory operation started
- If there is a monitoring switch on the machine operation panel, it is set to "Cutting Monitor ON."
- Monitoring space is specified using M720-769, M770 in the program.



## 6. Load Graph

(Note) For other than the  $\Sigma 10L$ , the load graph display is only for main monitoring.

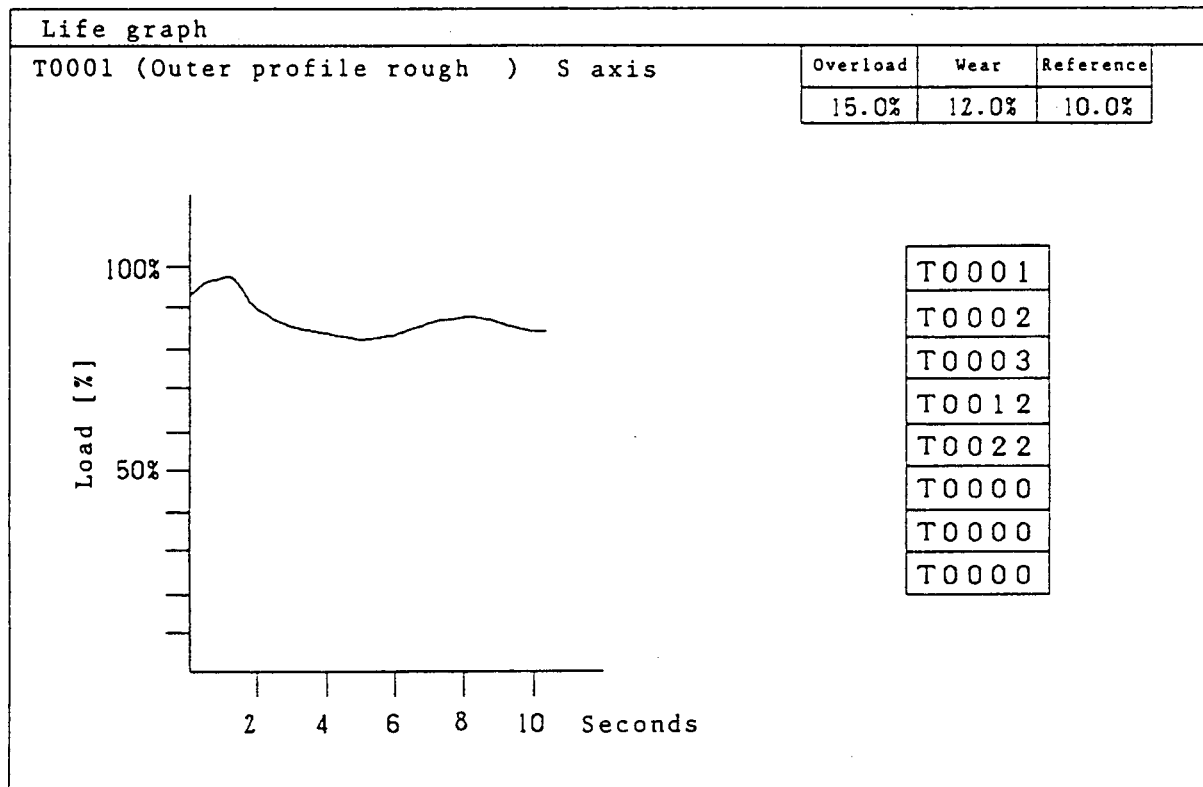


Displays the cutting load corresponding to the monitoring axis time by a graph in realtime.

- Display mode
  - Average ..... Displays the average value of load data per unit time
  - Peak ..... Displays the maximum and minimum values of load data per unit time
- Display time ..... Set arbitrary data greater than 1 minute
  - Move the cursor on time, input data, and press INPUT.
- Life data and monitoring data can be changed also.

## 7. Life Graph

(Note) There is no life graph display for other than the  $\Sigma 10L$ .



Displays the loading condition from the start of usage of 8 registered tools in the form of a graph. The loading condition with time can be seen at a glance.

- Display selection ..... Move the cursor on the number to be displayed and press INPUT.
- Change in tool number ..... Move the cursor, input the number and press INPUT.

## 8. Alarms

- 770 Spindle overload alarm ..... Spindle load has exceeded the overload value during cutting
  - 772 X axis overload alarm
  - 773 Y axis overload alarm
  - 774 Z axis overload alarm
  - 771 No load alarm ..... Machining has terminated without the cutting load exceeding the no load value.
- Feed axis load has exceeded the overload value during cutting.

When the alarms mentioned above occur, the machine makes a feed-hold stop, and 3 seconds later the spindle rotation stops. To restore the operation manually, first move each axis to a safe position, and then perform inspection / replacement of the tool.

## 9. Sample Program

In cutting monitoring, a program for the section between Auxiliary Function M720 and M770 is monitored. Monitoring start commands are issued by M720-M769, monitoring end by M770. There are 50 types of monitoring data corresponding to M720-M769.

[Programming Example]

```
077;  
G131;  
G00X0Y0Z0;  
T010100;  
M03S100;  
M720; ..... (1)  
G98G01X50.Y50.F100;  
X0Y0; ..... (2)  
M721; ..... (3)  
Y50.Z50.;  
Y0Z0; ..... (4)  
M770; ..... (5)  
M05;  
M12;      (1)~(2)    Monitoring by data of M720  
M30;      (3)~(4)    Monitoring by data of M721  
%         (5)        Monitoring completed by M770.
```

## 10. Parameters

### 10-1 Selection of Monitored Data System (Parameter 8000)

Select whether the monitored data is of absolute value system or of reference value system. The absolute value system assumes the A/D input “0” to be the monitored data “0”. The reference value system treats as “0” the A/D data immediately after the cancellation time in the monitoring section.

- Parameter 8000 (0: Absolute value system, 1: Reference value system)
  - Bit 0 ..... Spindle monitored reference value system
  - Bit 1 ..... X-axis monitored reference value system
  - Bit 2 ..... Y-axis monitored reference value system
  - Bit 3 ..... Z-axis monitored reference value system
  - Bit 4 ..... Rotary tool reference value system

### 10-2 Selection of Initial Setting (Parameter 8001)

When all the data of all the tools are deleted with the data delete function in the Load screen, turn on the monitoring function selected with this parameter.

- Parameter 8001 (0: Note selected, 1: Selected)
  - Bit 0 ..... Overload of the main monitoring axis
  - Bit 1 ..... Wear of the main monitoring axis

- Bit 2 ..... Adaption 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 frequency mode
- Bit 7 ..... Initialization in the tool life time mode

### 10-3 Automatic Calculation Parameters (Parameters 8010-8029)

Set the parameters which automatically calculates the cutting load values of the spindle, rotary tool, X axis, Y axis, and Z axis. The parameters for the rotary axis are the same as those for the spindle.

	Spindle	X-axis	Y-axis	Z-axis
AV_HIG	8010	8016	8022	8028
AV_LOW	8011	8017	8023	8029

- Spindle load calculation value  
(Rotate Tool)

$$= \frac{\text{Parameter 8010} \quad \text{Parameter 8010}}{\sum \text{Load data} / \sum \text{No. of load data}}$$

$$\text{Parameter 8011} \quad \text{Parameter 8011}$$
- X-axis load calculation value

$$= \frac{\text{Parameter 8016} \quad \text{Parameter 8016}}{\sum \text{Load data} / \sum \text{No. of load data}}$$

$$\text{Parameter 8017} \quad \text{Parameter 8017}$$
- Y-axis load calculation value

$$= \frac{\text{Parameter 8022} \quad \text{Parameter 8022}}{\sum \text{Load data} / \sum \text{No. of load data}}$$

$$\text{Parameter 8023} \quad \text{Parameter 8023}$$
- z-axis load calculation value

$$= \frac{\text{Parameter 8028} \quad \text{Parameter 8028}}{\sum \text{Load data} / \sum \text{No. of load data}}$$

$$\text{Parameter 8029} \quad \text{Parameter 8029}$$

### 10-4 Timer Parameters (8030-8044)

Set the timer data used for monitoring. The input increment of the timer is “0.1 second”. The parameters for the rotary tool are the same as those for the spindle.

	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 is a data disabled time at spindle start, spindle speed change, and cutting feed start time.
- The overload judgment time is a time required to judge as an overload when the cutting load exceeds the overload value.
- The wear judgment time is a time required to judge as wear when the cutting load exceeds the wear value.

## 10-5 Adaptive Parameters

- (1) Adaptive upper limit value (no parameter)  
The adaptive upper limit value is the reference value set in the Load Set screen.  
Adaptive upper limit value = Reference value
- (2) Adaptive lower limit value (no parameter)  
The adaptive lower limit value is the value set in the Load Set screen.  
Adaptive lower limit value = key input value or automatic calculated value
- (3) Adaptive maximum override (parameter 8052)  
Set the maximum override in adaptive control to "110-200".
- (4) Adaptive minimum override (parameter 8053)  
Sets the minimum override in adaptive control to "10-90".
- (5) Adaptive override step (parameter 8054)  
Specifies an feed rate override change interval, which is controlled in adaptive control, to "10-50".
- (6) Adaptive decrease judgment time (parameter 8055)  
Specifies the time to judge that an override is to be decreased, when the cutting load exceeds the adaptive upper limit value in adaptive control.  
Its input increment is "0.1" second.
- (7) Adaptive increase judgment time (parameter 8056)  
Specifies the time to judge that an override is to be increased, when the cutting load is below the adaptive upper limit value in adaptive control. Its input increment is "0.1" second.
- (8) spindle/rotary tool data averaging frequency (parameter 8058)  
Set the averaging frequency of the spindle load data to be input from the A/D to "0-40".
- (9) Feed axis data averaging frequency (parameter 8059)  
Set the averaging frequency of the feed axis load data to be input from the servo to "0-40".

### III. 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

(Note) For  $\Sigma$ L, periodic checking does not exist.

The Period Check screen is intended for maintenance of various oils, battery, and so on.

This screen is displayed by pressing **OPER/MAINT** 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

CHECK UP		1998.01.22			
	ITEM	TIME	DATE	APPOINT	STATUS
1	LUBE.OIL	120 Hour	1998.01.22		Rest 120Hour
2	BYD.OIL	6 Month	1998.01.22	1998.07.22	
3	NC/PCBATTERY	4 Year	1998.01.22	2002.01.22	
4					
5					
6					
7					
8					
9					
10					
>					
DETAIL			1LINE	SET-TIME	SET-DATA
/1	/2	/3	/4 CLEAR /5	/6 CHECK /7	CHECK /8 /9

##### (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 items, "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.
- ..... Displays the Details screen for the cursor-indicated item, if any. If there is no Details screen, "No Screen" will be displayed.
- ..... Clears the cursor-indicated items, Check Time to Status. You are prompted, "Do You Want to Clear ?" Press  to clear.
- ..... 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  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  .

Otherwise, time calculation will not be initiated.

Note: The first item is "Lubricant" and only requires you to set the unit of time.

- ..... If you move the reverse video cursor to an actually checked item and press  , 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

(Note) The  $\Sigma$ L has not 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 **OPER/MAINT** and **F4/SYSTEM**.

Note: The displayed items vary from one model to another.

STATUS/OK MONITOR	
STATUS	OK MONITOR
SPINDLE R. P. M                      3 6 0 0	MODE                                      自動
	SPINDLE OVERRIDE                      1 0 0 %
	FEEDRATE OVERRIDE                      1 5 0 %
	RAPID TR. OVERRIDE                      1 0 0 %
	SINGLE BLOCK                              OFF
	MACHINE LOCK                              OFF
	DRY RUN                                      ON
	Z AXIS CANCEL                              OFF
	AXIS INTERLOCK                              XYZ
	ATC ORIGIN                                      OK
>	
/1	/2
/3	/4
/5	/6
/7	/8
/9	/9

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.

Select **OPER/MAINT** and **F5/INFORMATION** in the Overall screen to display the first page of the G-codes List.

G - CODE K I S T		
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	M-CODE	MM-CODE							
LIST /1	LIST /2	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)

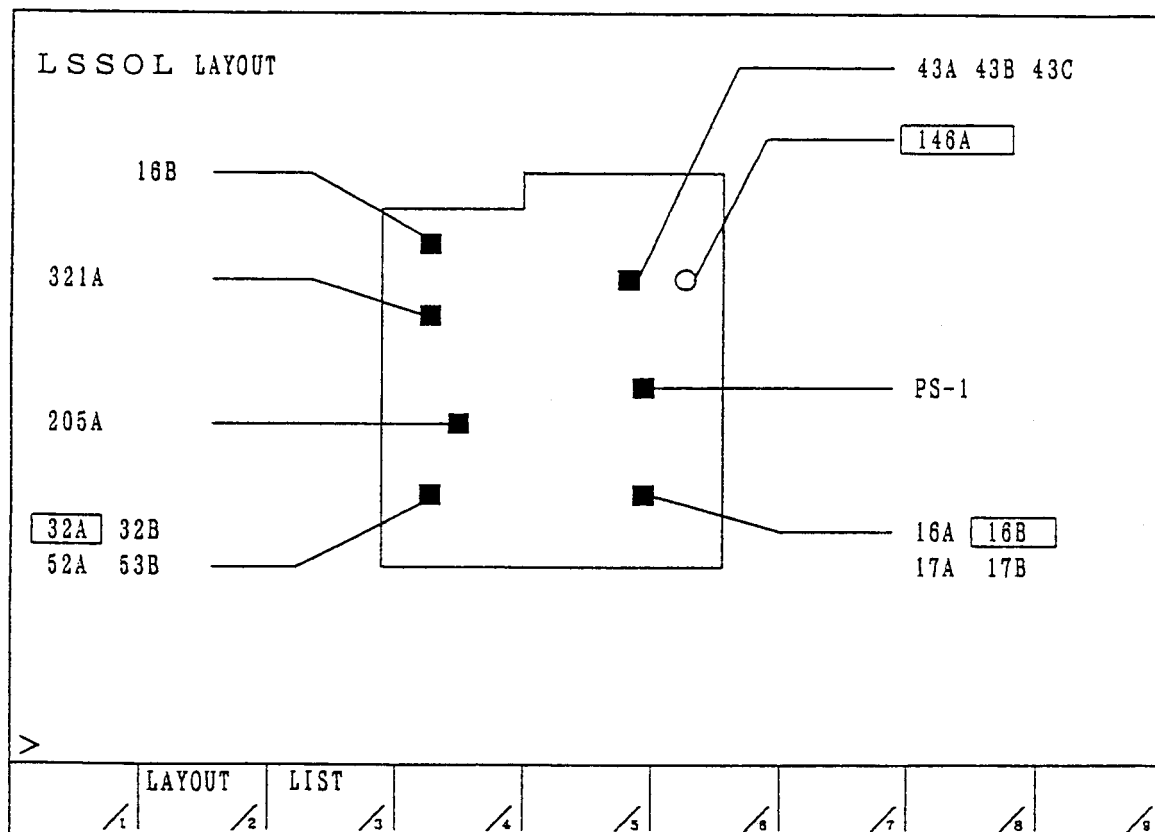
The LSSOL diagram shows a layout of the limit switches and solenoids, and their ON/OFF status. It has a layout drawing and list.

In the Overall screen, select **OPER/MAINT** , **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

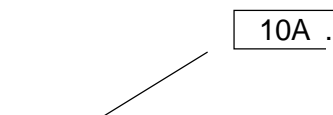
Note: The layout drawing and form vary from one model to another.



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 from 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.

L S S O L   L I S T							
No.	FUNCTION	STATUS		No.	FUNCTION	STATUS	
		SOL	LS			SOL	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 BLOW	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			

>

/1	LAYOUT	LIST	/2	/3	/4	/5	/6	/7	/8	/9
----	--------	------	----	----	----	----	----	----	----	----

Example of List Screen

- **F2/LAYOUT** ..... Displays the layout drawing.
- **RETURN** ..... Returns you to the Overall screen.

## IV. 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 DLL (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 SEICO  $\Sigma$ 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 : 10 BASE-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.

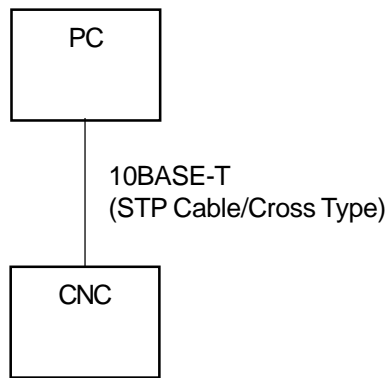
### 2-2 Connecting between the PC and CNC Unit

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.

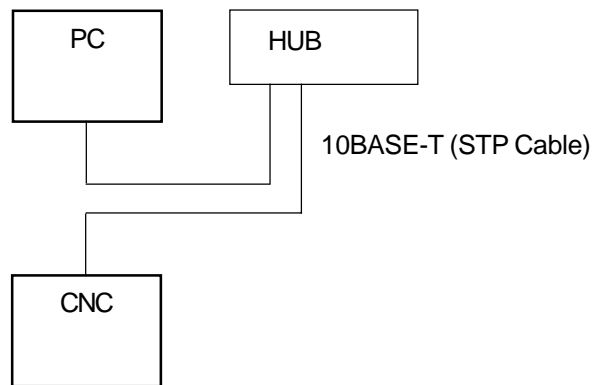




[One-to-one Connection]

(2) Connecting one PC to one CNC unit (Via the hub)

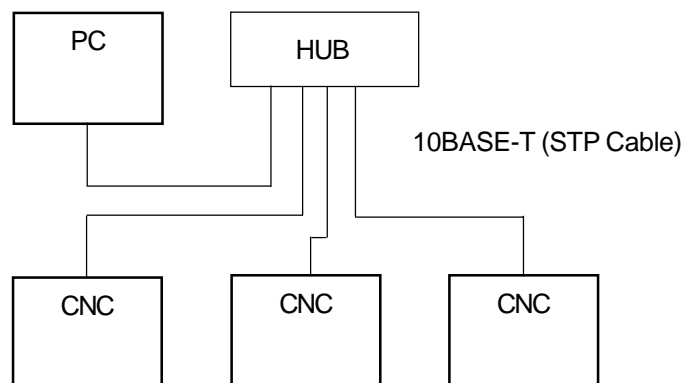
In one-to-one connection, when the distance is long or you want to enhance the communication conditions, interpose the hub between them. In this case, the cable used between the hub and CNC unit is a straight type.



[One-to-one Connection Via Hub]

(3) Connection 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:

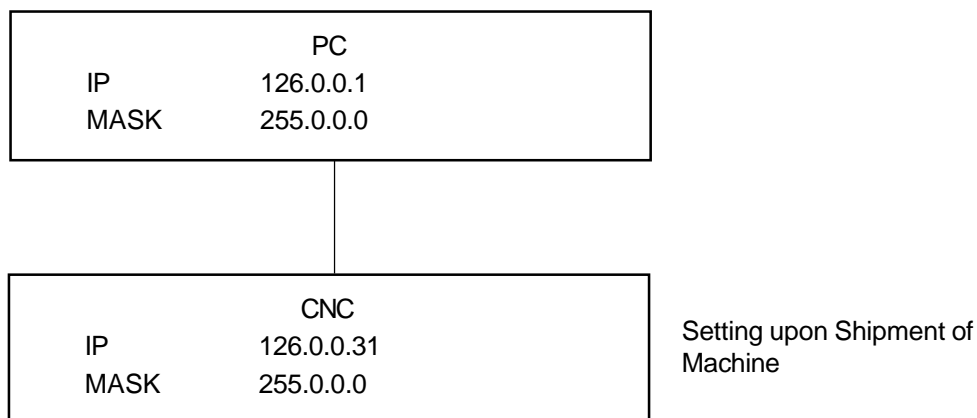
Upon Shipment of CNC Unit

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.

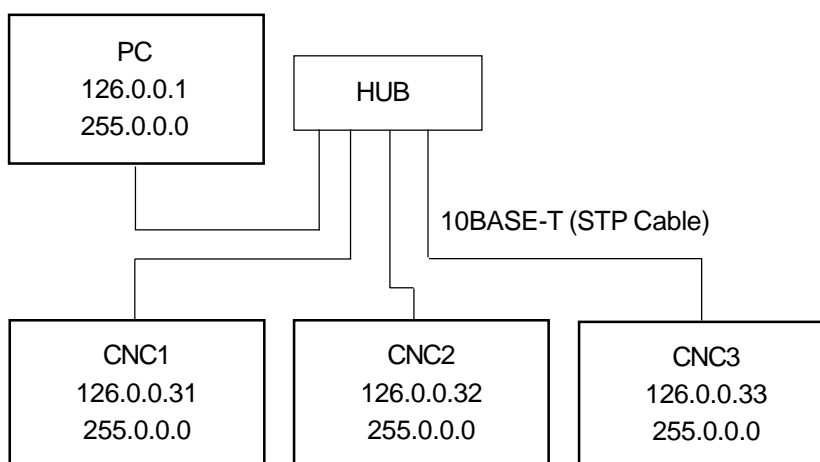


[One-to-one Connection]

- (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.

```
SETTING(HANDY)

PARAMETER WRITE = 0 ( 0:DISABLE  1:ENABLE )
TV CHECK        = 0 ( 0:OFF      1:ON  )
PUNCH CODE      = 0 ( 0:EIA     1:ISO )

>_
MDI **** *** ***                12:00:00
```

OFFSET	SETTING			(OPRT)	+
--------	---------	--	--	--------	---

#### 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.

SYSTEM CONFIG (SLOT)					
					PAGE:01
SLOT MODULE_ID SERIES VERISON					
>_					
MDI **** * * * * 12:00:00					
PARAM	DGNOS	PMC	SYSTEM		+

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					
					PAGE: 1/2
MAC ADDRESS *****					
NUMBER OF SCREENS 14					
MAXIMUM PATH 1					
IP ADDRESS 126.0.0.31					
SUBNET MASK 255.0.0.0					
ROUTER IP ADDRESS					
>_					
MDI **** * * * * 12:00:00					
		ETHPRM		(OPRT)	+

ETHERNET PARAMETER screen (1st Page)


ETHERNET PARAMETER		PAGE: 2/2
(DNC1/ ETHERNET)		
PORT NUMBER(TCP)		8193
PORT NUMBER(UDP)		8192
TIME INTERVAL		100
>_		
MDI **** * * *		12:00:00
	ETHPRM	(OPRT) +

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  and  to set WRITE PARAMETER = 0.  
This disables you to write the parameters. Do not tamper with the other items.
- (3) Press the  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

Destination host unreachable.

Destination host unreachable.

Destination host unreachable.

Destination 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.

## V. External Measuring Function

### 1. Outline

This function is designed to compensate a tool offset amount(wear amount) by feeding back the measurement results obtained by an external measuring instrument to the NC unit through a sequencer.

It is optimum for long-time continuous operation with identical work.

It is executed by setting the conditions such as the tool used at the measuring position, measuring point in the Setting screen and commanding G194 TXXXX(XXXX = set tool) in the program. Up to 8 points can be compensated. G194 processing can be largely divided into the following three:

- (1) Compensation of the external measuring instrument data
- (2) Qualitative compensation
- (3) Wear management

The following describes their details.

- ① Compensation of the external measuring instrument data  
Every time a measurement is done by the measuring instrument, its result is counted. The G194 program block compares their cumulative results with the preset value to compensate a wear amount as required.
- ② Qualitative compensation  
By setting a wear amount for each work, the tool's wear amount can be calculated out of the usage of the tool to compensate an offset.  
It is necessary to jointly use a tool life management function(setting a quantity) in order to learn the usage of the tool.
- ③ Wear management  
The tool life can be determined by setting the allowable wear amount of that tool. If the wear amount obtained in ① and ② exceeds the set value, the tool will be determined out of life.

#### ASSOCIATED PARAMETERS

Parameter 6240: 3 1 Qualitative compensation enabled

Parameter 6240: 2 1 Wear management enabled



## 2. Tool Data

Set the measuring position data for the measuring instrument; the data can be set for up to 8 points.

No.	Tool	Meas. Point	Axis	Ref.Value	Error	Judgment
1	100	O.D.	- X	10.000	3	OK
2	200	I.D.	+ X	5.000	5	+OK
3						
4						
5						
6						
7						
8						

NO.1 to 8 : Setting number.

Corresponds to the measuring position of the external measuring instrument.

TOOL : Tool number(4 digits).

Set the tool number(machining tool programmed: lower 4 digits) used at the measuring position.

MEAS. POINT : Profile of the work for which the tool is to be used.

Set the profile of the work. You can set an inner diameter, outer diameter, end face, and back end face.

AXIS : Tool nose direction + cutting axis at the time of cutting.

This is automatically set when inputting a measuring point, and can be altered forcibly.

(Example) Inner diameter --- +X

Outer diameter --- -X

REF. VALUE : Work reference value.

Set the work reference value. This is meaningless when the data from the external measuring instrument is a judgment value or error value.

ERROR : Error from the reference value.

Displays an error from the reference value when the data from the external measuring instrument is an error value or measured value.

JUDGMENT : Work judgment result.

Displays a work judgment result.

Set the data for the items marked with  .

### ASSOCIATED PARAMETERS

- Parameter 6240 : 1, 0    00   Outputs a judgment value as the measured data.  
                                  01   Outputs an error value.  
                                  10   Outputs a measured value.

### 3. External Measurement

Set the compensating conditions for the tool set in 2. TOOL DATA.

No.	Judg-ment	Comp. Amount	Counter(repeat) Current Value Set Value	Tolerance	Meas. Pass
1	++NG	Out Of Life		++NG Limit	Measurement Pass
	+NG	10		+NG Limit	After Altering An Offset
	+OK	5		+OK	2
	OK			-OK	Measurement Pass
	-OK	5		-NG Limit	After The Tool Being
	-NG	10		--NG Limit	2
	--NG	Out Of Life			

NO.1 to 8 : Setting number.

Corresponds to the measuring position of the external measuring instrument.

JUDGMENT : Judges the work in 7 grades; ++NG, +NG, +OK, OK, -OK, -NG, --NG

COMP. AMOUNT : Compensation amount for each judgment result. Set a compensation amount commensurate with the judgment result. ++NG and --NG are always OUT OF LIFE.

COUNTER CURRENT VALUE: Displays the number of works which have been judged.

This is cleared when compensation is executed(G194).

COUNTER SET VALUE: Work counter to determine the compensation amount.

The compensation amount is determined at CURRENT VALUE = SET VALUE.

TOLERANCE : Allowable range of errors for approving the judgment result.

Input the allowable range of errors in terms of absolute value. This is not necessary for a judgment value output method.

MEAS. PASS : Number of times to pass the measurement result after determining the compensation amount or determining that the tool is out of life.

Set the data for the items marked with .

#### JUDGMENT and TOLERANCE

There are the following relations between Judgment and Tolerance.

Judgment

++NG	.	ERROR >	++NG LIMIT
+NG	++NG LIMIT	>= ERROR >	+NG LIMIT
+OK	+NG LIMIT	>= ERROR >	+OK
OK	+OK	>= ERROR >=	-OK
-OK	-OK	> ERROR >=	-NG LIMIT
-NG	--NG LIMIT	> ERROR >=	--NG LIMIT
--NG	--NG LIMIT	> ERROR	

## ASSOCIATED PARAMETERS

Parameter 6240:4 1 Only --NG and ++NG are determined a defective work.

At this time, the display of +NG and -NG are changed to ++OK and --OK, respectively.

## COUNT OF COUNTER CURRENT VALUE

Once a measurement is done, COUNTER CURRENT VALUE is counted up as follows:

- ① When JUDGMENT is OK  
Clears the counter for all the areas.
- ② When JUDGMENT is +OK/-OK  
Counts up +OK/-OK
- ③ When JUDGMENT is +NG/-NG  
Counts up +NG/-NG and +OK/-OK
- ④ When JUDGMENT is ++NG/--NG  
Counts up ++NG/--NG, +NG/-NG, and +OK/-OK
- ⑤ When JUDGMENT changes from "+" to "-" area or from "-" to "+" area  
Clears the counter for the previous signed area.

## MEAS. PASS

The work fed out of the machine is carried to the measuring instrument, and the measured data are fed back to the NC unit to perform compensation. Assuming that several works(including the one being machined) are always standing between the machine and measuring instrument, it is necessary to override the compensation amount for them. Set that surplus compensation amount.

## 4. Qualitative Compensation

No.	Tool Usage	Comp. Amount 1/100um	Work Counter
1	0		
	1000	2	1
	2000	3	
	∞	4	

NO.1 to 8 : Setting number.

Set a qualitative compensation amount by tool.

**TOOL USAGE** :The compensation amount can be set in 3 stages according to tool usage.

**COMP. AMOUNT** : Wear amount resulting from each use of the tool.

**WORK COUNTER** : Displays the number of works.

Set the data for the items marked with  .

For the tools which do not need qualitative compensation, clear all the compensation amounts.

## 5. Wear Management

Tool	Wear Amount	Set Value	Status
1			
2			
3			
4			
5			
6			
7			
8			

TOOL : Offset number.

Available as many as the number of tool sets.

WEAR AMOUNT : Current wear amount.

Clear this when replacing with a new tool.

SET VALUE : Allowable wear amount of the tool.

If the wear amount reaches the set value, that tool will be determined out of life.

No wear management is provided for the offset number whose set value is 0(zero).





SEIKI-SEICOS  $\Sigma$ 10L/ $\Sigma$ 16T/ $\Sigma$ 18T/ $\Sigma$ 21L  
INSTRUCTION MANUAL  
OPERATION  
Version 1.02  
9-2000

First Edition 11-1999