VERTICAL MACHINING CENTER

VS50/60

INSTRUCTION MANUAL

OPERATION

SEIKI-SEICOS Σ16M/18M

Version 1.01

SEIKI Hitachi Seiki Co.,Ltd.

BO-2782-1-0221-E-1-01

Introduction

Thank you for your having purchased the machine, favoring our product lines for your use.
This manual contains fundamental information on the machine operation. Please read and fully understand the contents for your safe machine operation.
In particular, the contents of the items concerning safety in this manual and the descriptions on the “caution plates” attached to the machine are important. Please follow the instructions contained and keep them always in mind to ensure safe operation.
The reference record papers on adjusting setting values such as a parameter list are attached to the machine unit and enclosed in the packing. These are necessary for maintenance and adjustment of the machine later on. Please keep them safely not to be mislaid.
The design and specifications of this machine may be changed to meet any future improvement. As the result, there may arise some cases where explanations in this manual could become partly inconsistent with the actual machine. Please note this point in advance.
In this manual, items on the standard and optional specifications are handled indiscriminately. Please refer to the “delivery note” for the detailed specification of your machine confirmation.

CONTENTS

OPERATION OF MACHINE ..... 1 -1

1. Daily Maintenance and Inspection ..... 1 -1
2. Outline of Main Operation Panel and NC Operation Panel ..... 1 -1
   2-1 Operation by Using the Main Operation Panel and Sub-operation Panel ..... 1 -1
   2-2 NC Operation Panel ..... 1 -15
3. Operational Procedures ..... 1 -18
   3-1 When Starting the Machine ..... 1 -18
   3-2 Warming-up Operation of Spindle ..... 1 -19
   3-3 Zero Return Procedures ..... 1 -20
   3-4 When Completing the Daily Operation ..... 1 -21
4. Manual Operation ..... 1 -22
   4-1 Each Axial Feed ..... 1 -22
   4-2 Rapid Traverse ..... 1 -22
   4-3 Mounting and Dismounting the Tool ..... 1 -23
5. Manual Data Input (MDI) Operation ..... 1 -24
   5-1 Each Axial Feed ..... 1 -24
   5-2 Coolant ..... 1 -24
   5-3 Spindle Speed Change and Spindle Revolution ..... 1 -24
   5-4 Spindle Positioning ..... 1 -25
6. Registration of Program 1-26
   6-1 Registration from an External Device 1-26
   6-2 Manual Registration by the Address/Numeral Keys 1-27
7. Program No. Search 1-29
   7-1 Search by Key in a Program No. 1-29
   7-2 Search to Utilize the Program List. 1-30
8. Edition of Program 1-31
   8-1 Preparation in Advance at the Time of the Edition of Program. 1-31
   8-2 Search of Word 1-32
   8-3 Edition of Program 1-33
   8-4 Back Ground Editing 1-39
   8-5 Copy of Program 1-41
   8-6 Editing Procedure of Range Designation (Expanded Tape Editing/Option) 1-42
   8-7 Alteration of Word (Expanded Tape Editing) 1-43
   8-8 Deletion of Program 1-45
   8-9 Arrangement of Program 1-46
   8-10 Process After Edition 1-47
9. Output of Program 1-48

10. Automatic Operation 1-50
10-1 When Machining the First Workpiece, Checking the Newly Created Program 1-50
10-2 Program Memory Operation 1-50
10-3 When Inserting Manual Operation in Automatic Operation 1-51
10-4 When Performing MDI Operation in Automatic Operation 1-51
11. Operation of Automatic Tool Changer (ATC) ..... 1 -52
11-1 Names of ATC Each Section ..... 1 -52
11-2 Explanation of Actions ..... 1 -53
11-3 Rotation of Tool Storage Magazine (at Single Mode) ..... 1 -56
11-4 Attachment and Detachment of Tools ..... 1 -57
11-5 Relation Between Spindle Nose Key and ATC ..... 1 -58
11-6 Attention When Using Oil Hole Tool ..... 1 -58
11-7 ATC Program Operation ..... 1 -59
11-8 Condition Check During ATC Execution ..... 1 -60
12. Setting (Data) 1-62
12-1 Outline 1-62
13. Time Measuring, Data 1-64
13-1 Time Measuring 1-64
13-2 Date and Time 1-65
14. Animated Drawing 1-66
14-1 Outline 1-66
14-2 Drawing Parameter 1-67
15. Parameter Setting 1-70
16. Operations Concerning W-Setter ..... 1 -74
16-1 Outline ..... 1 -74
16-2 Input Signal and Various Operations ..... 1 -74
16-3 Operation for Setting of Work Setter Reference Surface ..... 1 -75
16-4 Operation for Setting of Work Setter Reference Hole ..... 1 -76
16-5 Operation for Setting of Work Setter Coordinate Modification ..... 1 -77
16-6 Manual Overwriting of Work Zero Point Offset Amount. ..... 1 -78
16-7 Setting Operation of Tool Setter ..... 1 -78
16-8 Tool Length Measuring Mode, Tool Diameter Measuring Mode ..... 1 -80
16-9 Parameter ..... 1 -81
16-10 Others ..... 1 -82
17. Safety Guard 1-84
17-1 Safety Guard (Tool Length) 1-84
17-2 Safety Guard (Comparison) 1-88
18. Tool Life Management 1-92
18-1 Tool Life Management 1-92
18-2 Spare Tool Management 1-94
18-3 Premachining Tool Check 1-95
18-4 Tool Skip and Tool Reset 1-96
18-5 Tool Life Management Program 1-96
18-6 Tool Life Management Operation 1-100

18-7 Tool Life Screen 1-101
18-8 Spare Tool Screen 1-105
18-9 Tool Status List Screen 1-108
18-10 Tool Life Management Parameters and Their Setting 1-109
18-11 Alarms 1-110
19. Cutting Monitoring Unit 1-111
19-1 Outline 1-111
19-2 Monitoring Method 1-111
19-3 Monitoring Functions 1-112
19-4 Cutting Monitoring Program 1-115
19-5 Data Setting 1-118
19-6 Cutting Monitoring Operation 1-120
19-7 Set Load Screen 1-124
19-8 Load Graph Screen 1-127
19-9 Life Graph Screen 1-128
19-10 Display and Setting of Parameters 1-129
19-11 Alarms 1-133

OPERATION OF MACHINE

1. Daily Maintenance and Inspection

To prevent trouble with the machine and keep it always in a good operating condition, daily check is as important as the periodic maintenance and inspection. As for the daily maintenance, refer to Chapter 2 “Daily Inspection Item List” of the “Maintenance Manual” and carry out necessary inspections accordingly.

2. Outline of Main Operation Panel and NC Operation Panel

2-1 Operation by Using the Main Operation Panel and Sub-operation Panel

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
Note *: When you operated a function which the specifications do not allow, the lamp of that push button switch blinks, and the alarm lamp [8] and call light are turned on. Also, “NO OPTION” is displayed on the CRT.*						
[1]	Mode selector switches	These are used for selecting operation modes: “MDI”, “PROGRAM EDITING”, “MEMORY AUTOMATIC OPERATION” and “TAPE AUTOMATIC OPERATION”.	O	O	O	
[2]	Function selector switches (Push a necessary function switch with the SELECT” switch pressed.)	CANCEL Z... A machine lock condition is applied to the Z axis only. It is convenient if used for a program test run.	O	O	O	O
		MACHINE LOCK... Only a display proceeds omitting axial moves of the machine. This function enables checking of program coordinate values without moving the axes.	O	O	O	O
		DRY RUN... A manual jog feed rate become valid instead of a feed rate specified by the program	O	O	O	

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
[3]	Spindle rotation effective key	These push button switches start and stop the spindle. (It is necessary to specify a spindle speed in advance.)	O	O		O
[4]	Mode selector switches	These are used for selecting operation modes: “FEED”, and “ZERO RETURN”.	O	O		O
		FEED... This mode enables a manual continuous feed. A feed axis and its direction are selected by the axis move push button switches [6], and a feed rate by the FEEDRATE [19] switches. Rapid traverse is available by the rapid traverse button.	O	O		O
		RAPID... This mode enables a rapid traverse. A rapid traverse axis and its direction are selected by the push button switches [6] and a rapid traverse rate (override) by the RAPID OVERRIDE push button switches [17].	O	O		O
		ZERO RETURN... This mode enables zero return. For an axis to be returned to the zero point, select with the push button switches [6]. For a feed rate, an override of 25% is applied. When zero return is completed, green lamp at [6] is lighted up, from the blinking state.	O	O		O
[5]	Program START / FEED HOLD switches	START... IN the AUTO or MDI mode, this push button switch start the program. While this push button switch is pressed and the program is running, a lamp located above lights up.	O	O	O	

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
[5]	Program START / FEED HOLD switches	FEED HOLD.. This push button switch suspends a program's progress. An axis being operated stops and a red lamp located above lights up. During operation by an auxiliary command (M, S or T), the program's progress stops after executing the remaining action of respective command.	O	O	O	
[6]	Axis move switches	These push button switches move the X, Y, Z and A (optional) axes respectively. Axis selection and its moving direction abide by the indication given on the respective push button switches. For a feed rate, select among the FEEDRATE switches [22]. When executing rapid traverse, push the rapid traverse button at the some time.	O			O
[7]	Initial position check	APC INITIAL POSITION... This lamp indicates that each machine section is located at its specified position, when executing an automatic. pallet change (APC) command.		O	O	
		ATC INITIAL POSITION... This lamp indicates that each machine section is located at its specified position, when executing an automatic tool change (ATC) command. During T cycle operates, this lamp lights up.		O	O	
[8]	Status display lamp	ALARM ... A red lamp lights up when an alarm occurred.		O	O	O
		PROGRAM STOP... When M00 or M01 is executed during operation in an automatic mode, a red lamp lights up. It is lit off by starting or continuing the program.		O	O	

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
[9]	Automatic power shut-off switch	This makes the automatic power shut-off function effective. After machining cycle is completed, the power is automatically shut-off by M30 when no pallet exists.	O	O	O	
[10]	Call right off switch	This switches off the lighting call light (yellow lamp melody horning.	O		O	O
[11]	Work setter	Makes the following functions effective, making use of a reference touch tool				
		TOOL SETTER... Measures a tool length and a tool diameter and automatically sets their offset amounts in the offset memory.	O	O		O
		REF. SURFACE, REF. HOLE ... Measures the coordinate system shift amounts of the reference surface and hole against the machine coordinate system and sets them in the coordinate system shift memory.	O	O		O
		COORD ALIGN... By measuring 3 points on the orthogonal or jig located on the X-Y plane, differences in rotation angle with the X and Y axes are calculated and stored, thus compensating X-axis and Y-axis program coordinate values.	O	O		O
[12]	Safety check	Measure a machining tool length by the 1st program start. Measure the workpiece at an actual machining position with the reference touch tool mounted to the spindle by the 2nd program start. A workpiece-tool interference check is made by Z-axis approach (G00) command, adding the tool offset amounts used for these two information.				

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
[12]	Safety check	TOOL LENGTH... Measures and stores the length of the tool used for an actual machining program.	O	O	O	
		COLLATION... Sets the Z axis to a machine lock state, operates the machine with an actual machining program and checks whether there is an interference between the workpiece and tool when Z axis approaches.	O	O	O	
[14]	Override	MEMORY... Memorizes optimum override values (feed rate, spindle speed) found in trial cutting, etc., making them corresponding to the tools used.	O	O	O	
		AUTO. ... The override values memorized are automatically input to a program.	O	O	O	
[15]	Feedrate override switches	These push button switches apply an override (ranging from 0 to 200%) to a feed rate in automatic (cycle) operation. During a canned cycle for tapping, the override is ignored, resulting in 100%.	(Rotary switch)		O	
[16]	Spindle override switches	These push button switches apply an override (ranging from 50 to 150%) to a spindle speed command.	(Rotary switch)		O	O

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
[17]	Rapid override switches	These push button switches apply an override (ranging from 0 to 100%) to a specified rapid traverse rate. When the power is turned ON, the override value becomes less than 25%. In the manual operation, the rapid traverse rate becomes 0~25%. ***Note****)The manual feed is not available in 0%*.	(Rotary switch)		O	O
[18]	Conveyor switch	This push button switch runs the chip conveyor.	O	O	O	O
[19]	Work light switch	This push button switch turns on a work light.	O	O	O	O
[20]	Coolant switches	These push button switches work various kinds of coolant indicated on them respectively. Each kind of coolant can be turned ON/OFF manually even during automatic operation.	O	O	O	O

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
[21]	Feedrate switches	These push button switches select a manual jog feed rate between 0 mm/min and 5,000 mm/min. (In case of the A-axis, it becomes between 10 deg/min and 1,400 deg/min.)	(Rotary switch)			O
[22]	Optional block skip	This switch makes the block skip function command by program effective. During programming, the slash "/" used.	O	O	O	
[22]	Restart of block	The RESTART OF BLOCK becomes effective by pressing this switch and the indication lamp lights up. This becomes ineffective by repressing this switch and the indication lamp goes out. When the RESET OF BLOCK is not provided as option, this switch becomes ineffective and the indication lamp puts out. Through designating a sequence No. that is not restarted by this functions, the machining can be restarted from that block.	O	O	O	
[24]	Power	ON ... turns on the main power to the NC unit.	O		–	–
		OFF ... Turns off the main power.	O		–	–
[25]	Standby	Drive preparation state as for the NC unit to make the machine ready for operation.	O	O	–	–
[26]	Emergency stop (Main/Sub-operation panel)	Stops the machine completely. All motors are stopped and the NC unit is reset.	O		O	O
[27]	Memory	LOCK... Protects the information stored in the NC unit. Normally, shift this switch to "LOCK".	(Key)		O	

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
[28]	Memory	WRITE ... Shift this switch to “WRITE” when writing or correcting memory information.	(Key)		O	
[29]	Speed meter	Indicates a spindle speed.	(Meter)		O	O
[30]	Load meter	Indicates a spindle load to a motor’s rated power in terms of percentage.	(Meter)		O	O
[31]	Tool clamp/ unclamp	CLAMP ... Clamp a tool to the spindle.	O			O
		UNCLAMP ... Unclamp the spindle tool. When the tool is unclamped, a yellow lamp lights up and you cannot run the spindle.	O	O		O
[32]	Optional stop	When M01 is commanded in the tape, tape operation stops after all the commands in the block are completed. The indication lamp lights up. This is released by repressing the START button and indication lamp puts out.	O	O	O	
[33]	Single block	Program commands are executed one block by one block. However, canned cycles are executed one cycle by oneb cycle.	(Toggle switch)	O	O	
[34]	Program restart	The program restarts. The indication lamp lights up. This is released by repressing the button and the indication lamp puts out.	O	O	O	
[35]	Manual pulse generator (Portable)	Get the handle mode by the push button and select the feed axis by the rotary switch. For a feed rate, you can select 1 pulse/graduation (1/1), 10 pulses/graduation (10/1) and 100 pulses/graduation (100/1). Since it is portable, you can draw it out to a convenient position for use. For inch specifications, you can select 1/1, 10/1 and 50/1.	O	O		O

Group No.	Name	Use		Operation method		Effective mode	
				Push button	Lamp	Auto.	Manual
[36]	State OK indication	Information such as mode, override, interlock, and spindle tool No. necessary for machining can be monitored.		O	O	O	O
[37]	Cutting monitor	The screen of the cutting monitor can be monitored.					
[38]	Cutting monitor	Monitoring data is automatically set by the “AUTOMATIC SETTING” teaching cutting data. Select switch INEFFECTIVE ... makes the cutting monitor function ineffective. EFFECTIVE ... makes the cutting monitor function effective.		(Select switch)		O	
[39]	Block skip	This makes the block skip 2~9 effective. “/(2~9)” is used in the program.		O	O	O	
[40]	AIR BLOW	Tool nose	Air is spouted out from the nozzle directed to the tool nose	O	O	O	O
		Through	Air is spouted out from the through coolant nozzle.	O	O	O	O
[41]	Magazine rotation (ATC)	The ATC magazine rotates to index tools. Forward rotation: CW Reverse rotation: CCW		O			O

Group No.	Name	Use	Operation method		Effective mode	
			Push button	Lamp	Auto.	Manual
[42]	Manual indexing	Replacement of tools can be done even when automatic operation is going on. ON ... “FORWARD” and “REVERSE” buttons are effective. OFF ... The above buttons are not effective. To switch off manual interrupt, keep pressing on the “OFF” switch for at least 3 seconds.		O		O
[43]	ATC initial position return	This is a push button for making the restoration easy after emergency stop or power failure etc. during ATC.	O			O
[44]	Auto door	OPEN ... The door is opened.	O			O
		CLOSED ... The door is closed.	O			O
[45]	DOOR INTERLOCK	‘‘Effective’’ : Interlock of door open becomes effective.	(Key switch)		O	O
		‘‘Ineffective’’ : Interlock of door open becomes ineffective.				

Main operation box (I)

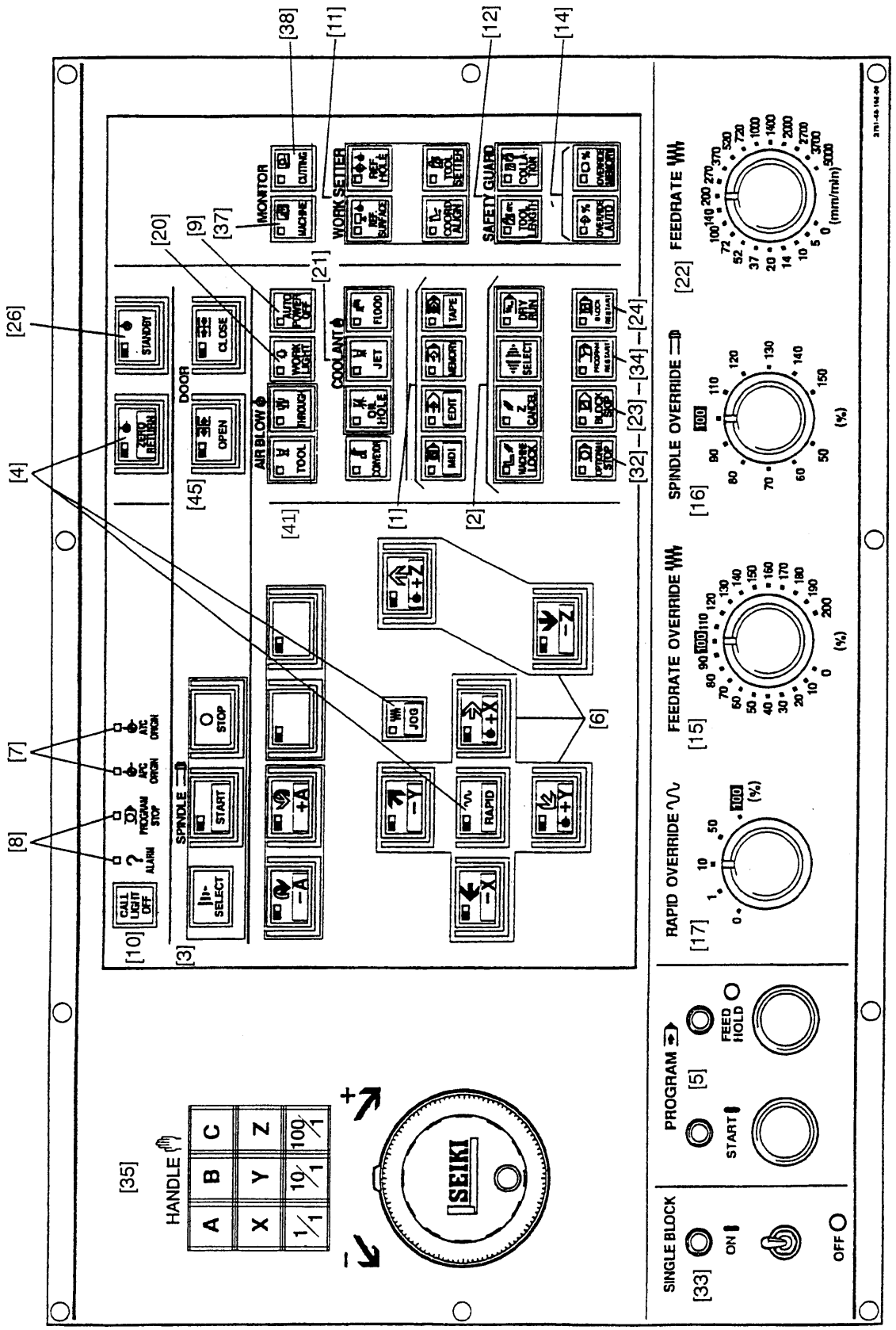
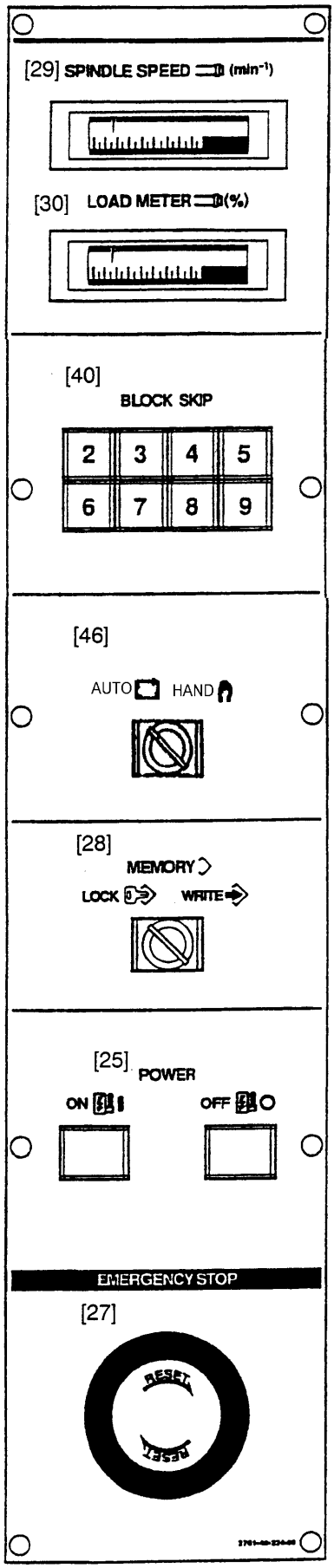
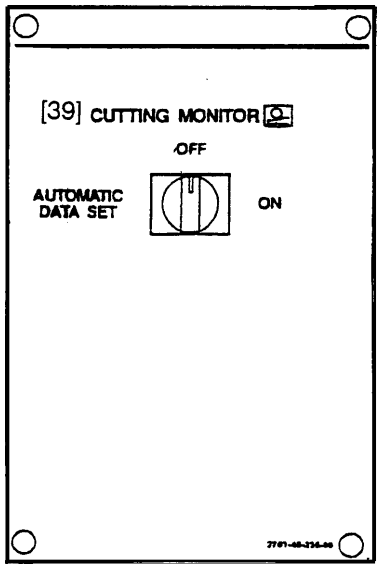


Fig.4-2 Main operation box (II)

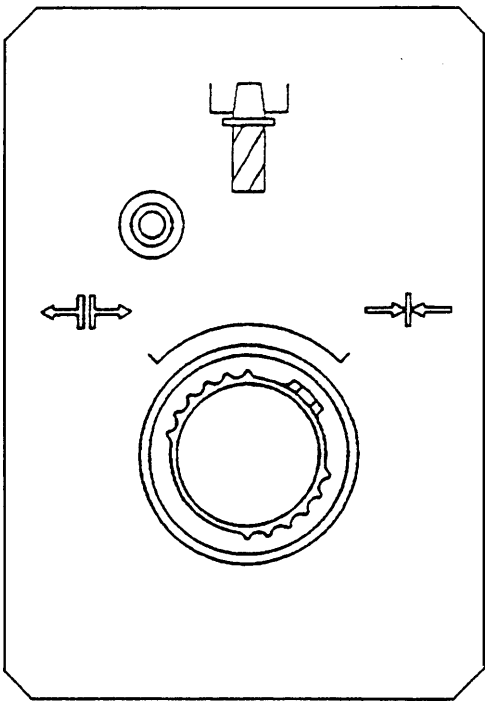




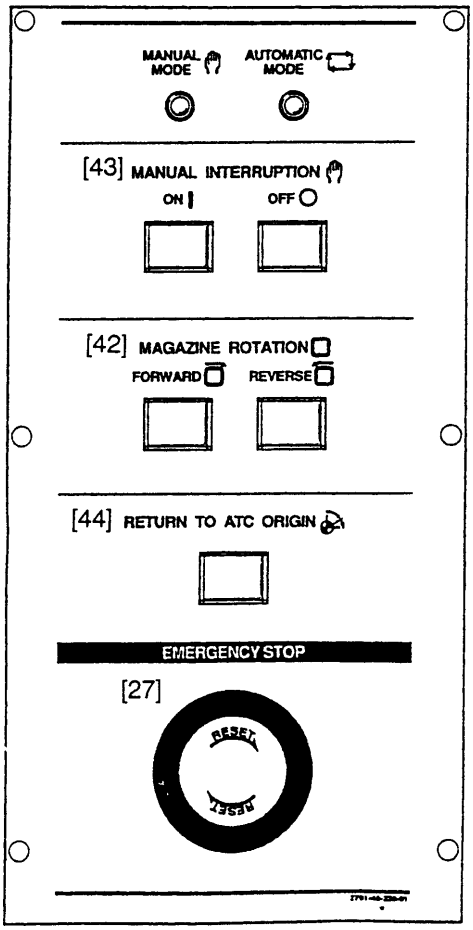
Auxiliary operation box (I)

Spindle tool clamp/unclamp

[31]

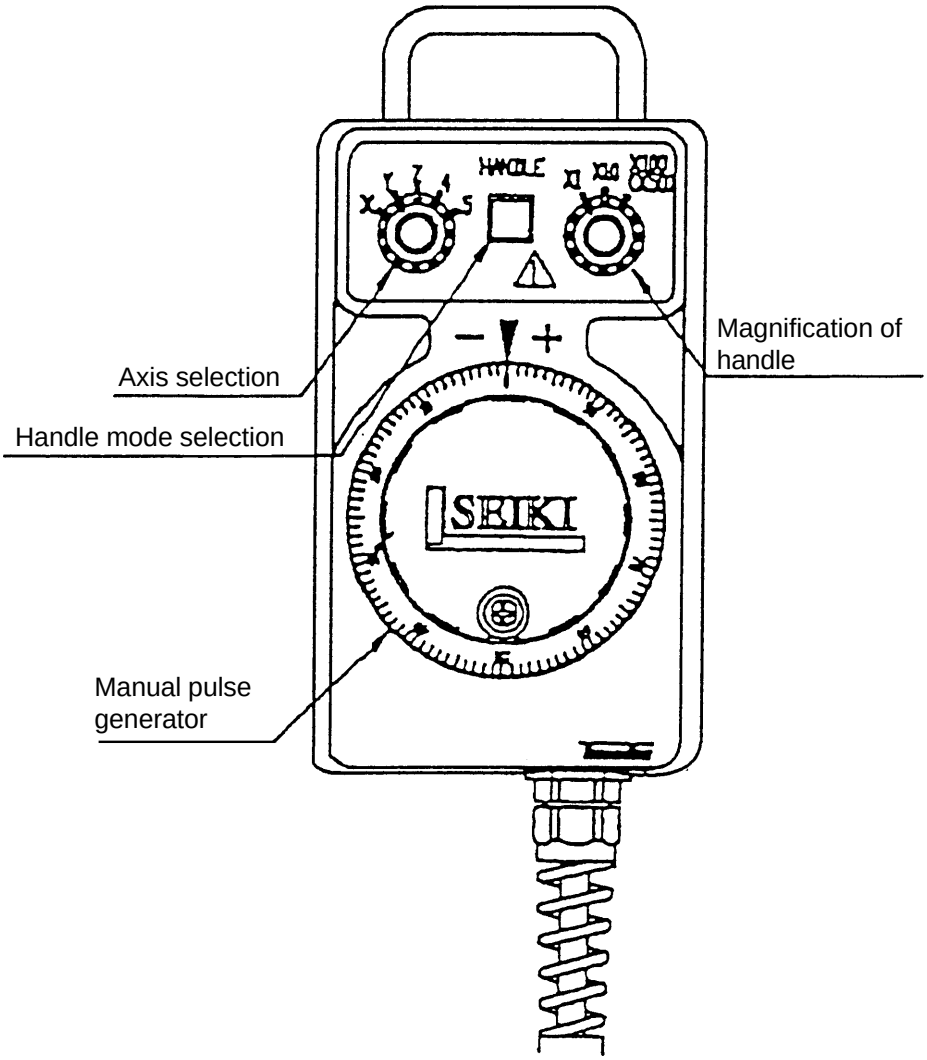


Auxiliary operation box (II) — ATC magazine

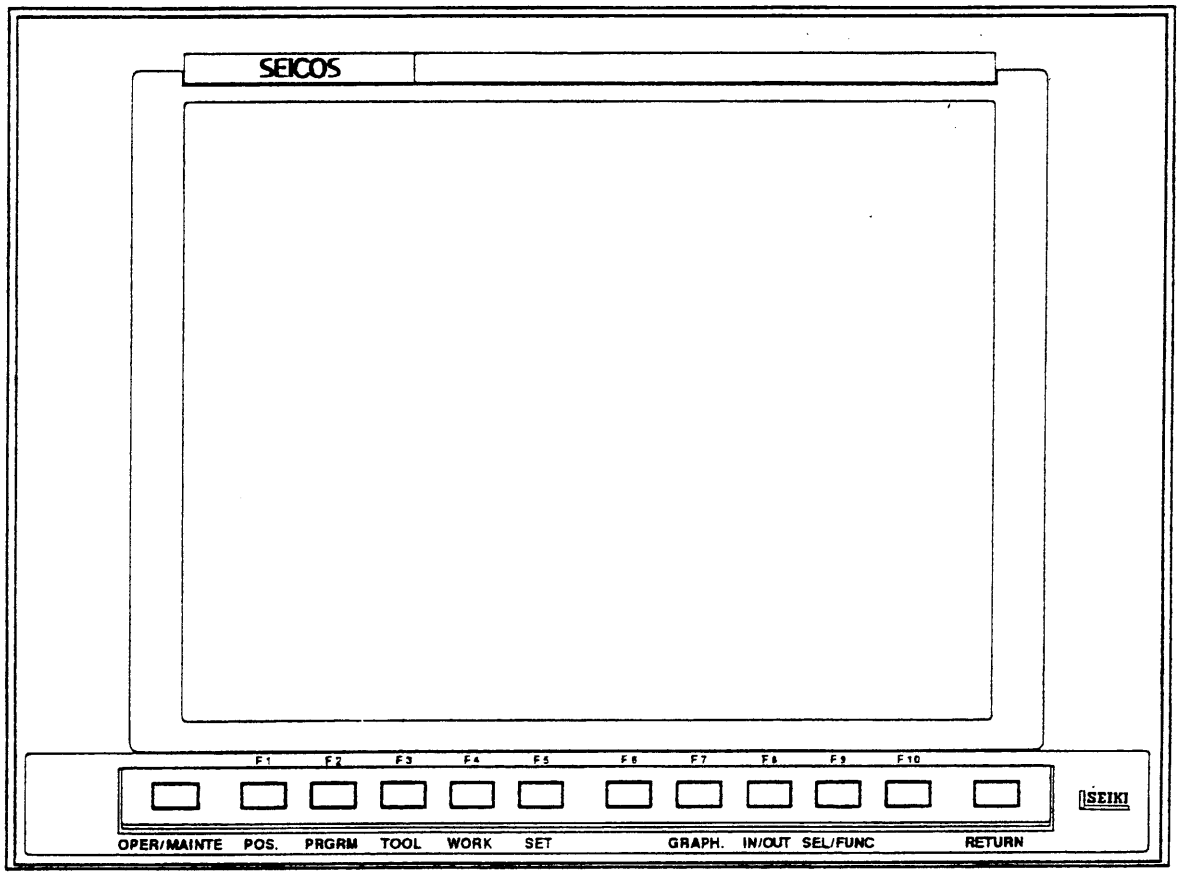


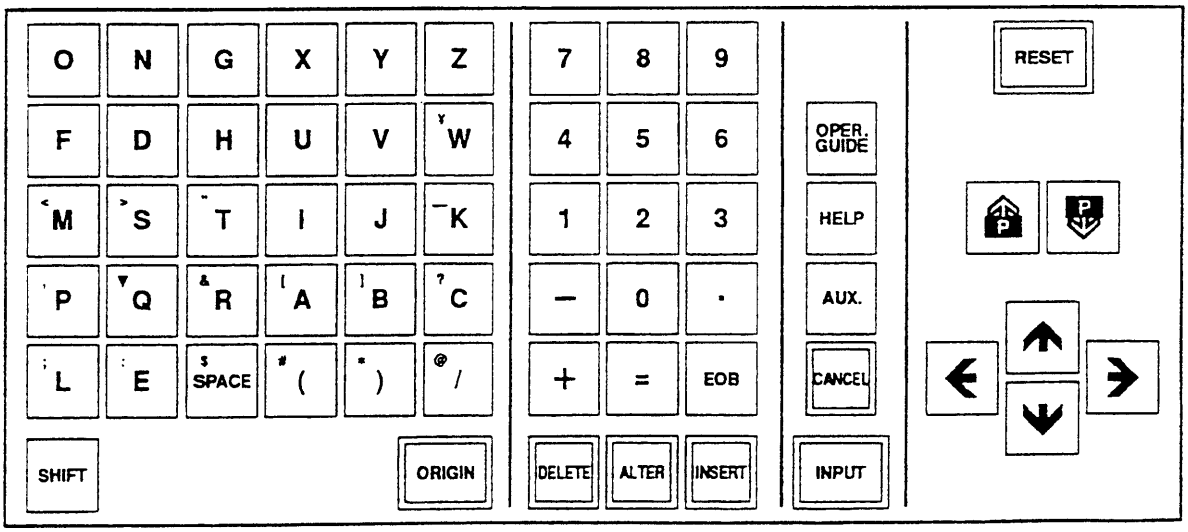
Handle operating panel

[35]

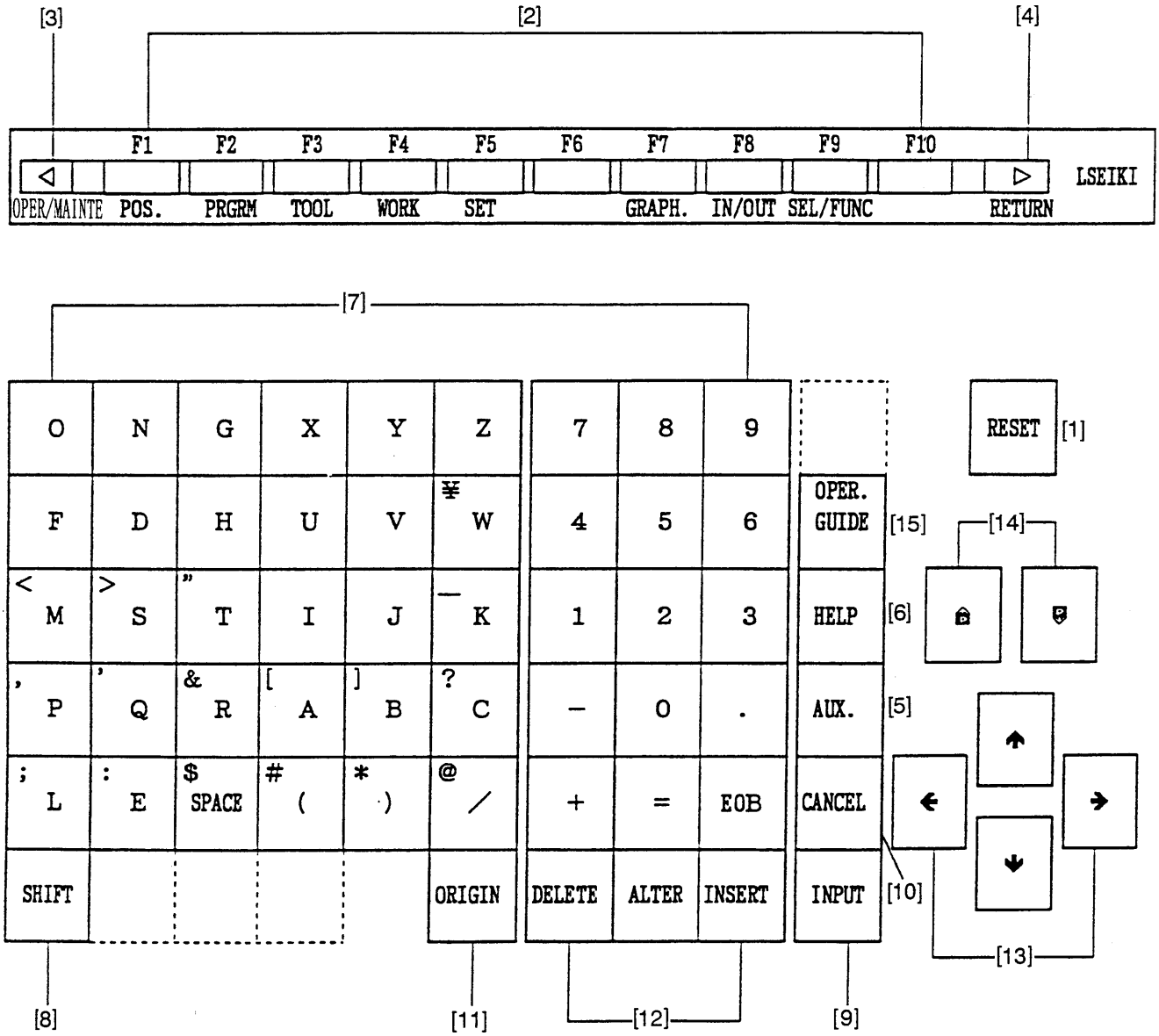


2-2 NC Operation Panel





NC Operation Panel



NC Operation Keys

No.	Name	Description
[1]	RESET key	Press this key when resetting the CNC unit in order to reset an alarm so on.
[2]	Function keys	When the function menu is displayed at the bottom of the CRT, there are the keys to select the menu. When the menu is not displayed, they serves as the keys to select the Position, Program, Tool, Work Coordinate, Setting, Plot and I/O screens. Press the F9/SEL/FUNC key when displayed the function menu at the bottom of the CRT.
[3]	OPER/MAINTE	Press this key when displaying the PC, Alarm or Maintenance screen. Pressing it once displays the function menu at the bottom of the CRT, and pressing it again erases the menu.
[4]	RETURN key	Press this key when you want to return to the Overall screen.
[5]	AUX. Key	
[6]	HELP key	
[7]	Address and Numerical keys	Used to input the alphabet, numbers, etc.
[8]	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.
[9]	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.
[10]	CANCEL key	Press this key when deleting the characters or symbols input into the key input buffer.
[11]	ORIGIN key	This key is used to clear the Plot screen.
[12]	DELET ALTER and INSRT keys	Used to perform deletion, alteration and insertion in editing the program. The alter key is also used for restoring F menu screen after having selected an item on the F menu.
[13]	Cursor move key	There are 4 keys which are used to move the cursor up/ down and right/left.
[14]	Page key	There are 2 keys which are used to page in the forward and backward directions.
[15]	OPER. GUIDE	

3. Operational Procedures

3-1 When Starting the Machine

(1) Turn on the supply power Switch.
(2) Turn on the power switch on the electric cabinets
(3) Press the POWER ON push button switch located on the upper part of the main operation panel.

Caution

The main panel and NC cabinet are enclosed lest the open air should enter directly. Therefore, do not leave each door open for a long time while the power is turned on. Check on the display and that cooling fan motors inside and outside the cabinet are started.

(4) Release EMERGENCY STOP button on the main operation panel.
(5) Press the STANDBY push button switch located of the main operation panel. (A green lamp lights up.)
(6) Before starting daily operation, reciprocate the X, Y and Z axes several times. (When this is done, be careful not to have the axes overtravelled.)
(7) Return each axis to the zero point (refer to the description for zero return). By performing zero return, a basic machine coordinate system is set and stored stroke limits are made effective.
(8) Turn on the conveyor switch.

Caution

Do not make operation under the state that massive chips are cast in the trough.

3-2 Warming-up Operation of Spindle

3-2-1 Warming-up at the Time of Starting Daily Operation

Caution

It is necessary to keep the bearing lubrication in a good condition for getting the normal spindle revolution. If the spindle is suddenly rotated at a high speed without warming-up, there is a danger of the oil film being cut and the bearing is burnt stuck. To have the spindle operated correctly in full capacity, warming up operation is necessary of which the following should always be practiced practice.

Warming-up before starting daily operation (30 minutes)

[1] 30% of maximum spindle revolution (10 minutes)

↓

[2] 50% of maximum spindle revolution (10 minutes)

↓

[3] 80% of maximum spindle revolution (10 minutes)

Practice warming in the order of [1] , [2] , [3] .

3-2-2 Warming-up When the Machine Was Not Operated for More Than One Week or Longer

Caution

In case of starting the spindle of oil air lubrication, after it has not been used for more than one week, warming-up must be practiced taking a longer time than usual, as oiling into bearing is necessary. In this case, please make sure up practice warming-up according to the following schedule.

1. #50 - Spindle of 10,000min^{-1} specification (oil air lubrication)

Warming-up after the spindle has not been used for more than one week (minimum warming-up time 4 hours)

[1] By 2,500min^{-1} for 1 hour or longer

↓

[2] By 5,000min^{-1} for 1 hour or longer

↓

[3] By 7,500min^{-1} for 1 hour or longer

↓

[4] By 10,000min^{-1} for 1 hour or longer

Practice warming in the order of [1] , [2] , [3] , [4] .

2. #40 - Spindle of 20,000min^{-1} specification (oil air lubrication)

Warming-up after the spindle has not been used for more than one week (minimum warming-up time 4 hours)

[1] By 5,000min^{-1} for 1 hour or longer

↓

[2] By 10,000min^{-1} for 1 hour or longer

↓

[3] By 15,000min^{-1} for 1 hour or longer

↓

[4] By 20,000min^{-1} for 1 hour or longer

Practice warming in the order of [1] , [2] , [3] , [4] .

3-3 Zero Return Procedures

Be sure to perform manual zero return so that you may set the basic machine coordinate system after turning on the power.

Display on the pushbutton for zero point direction of each axis flickers.

(1) Operate the mode switching push button for “Zero return”.

(2) Of the push button switches for the axis desiring “Zero return”, press the one with the origin mark ( ).

(3) By the specified speed of the rapid feed override, the axis returns to the zero point in the rapid feed speed, and stops at zero point. Then, the zero point position confirmation lamp of the axis is lit on.

(4) When the lamp lights up, release the switch.
Even when unhanded during decelerated speed movement, the axis stops after zero point is generated.
The above operations will complete zero return.

Caution

1. Make zero return operations for each axis one by one.
2. When the axis for which zero return is intended is already located close to the zero point, move axis once to the opposite side of the zero point by about 50mm then move it again to the direction of the designated zero return point.

3-4 When Completing the Daily Operation

(1) Confirm that START display lamp on the main operation panel is lit off.

(2) Press EMERGENCY STOP button on the main operation panel.

(3) Clean the machine.

(4) Release EMERGENCY STOP button on the main operation panel.

(5) Set operation preparation, and drive the conveyor.

(6) Stop the conveyor after all of cutting chips in the bed are discharged.

(7) Move each axis to the stop position.

X-axis ..... center of the stroke

Y-axis ..... center of the stroke

Z-axis ..... the head upward movement end

(8) Press EMERGENCY STOP button on the main operation panel.

(9) Press POWER OFF button of the main control panel to cut out the control device power.

(10) Cut out power of the high voltage control panel.

(11) Cut out supply power.

4. Manual Operation

4-1 Each Axial Feed

— In case of manual jog feed —

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

(2) Press the FEEDRATE push button switch to set an appropriate feed rate. Pressing the manual jog feed push button switch for a desired axis, move the machine in a desired direction.
When it comes to a specified position, release the switch.
(The machine moves only while the switch is pressed.)
Applications :
- Warming-up operation
- When bringing close to the zero point
- When cutting by manual operation
- Work rearranging operation

— In case of handle feed —

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

(2) Select an axis with the axis selector push button switch.

(3) With the manual handle, you can move the machine by 0.001 mm (0.0001 inch) per graduation (when 1/1 is selected) or by 0.01 mm (0.001 inch) per graduation (when 10/1 is selected).
Applications:
- When moving the machine by a very fine amount, such as when setting the zero point by centering a workpiece or fixture

4-2 Rapid Traverse

(1) While pushing the rapid traverse mode switch, press the push button switch for the axis you want to move at a rapid traverse rate (rapid traverse is applied only while the push button switch is pressed). At this time, an override of 0, 1, 10, 50 or 100% can be applied to a specified rapid traverse rate.

(2) Releasing rapid traverse push button switch changes the rapid traverse to normal feed.

4-3 Mounting and Dismounting the Tool

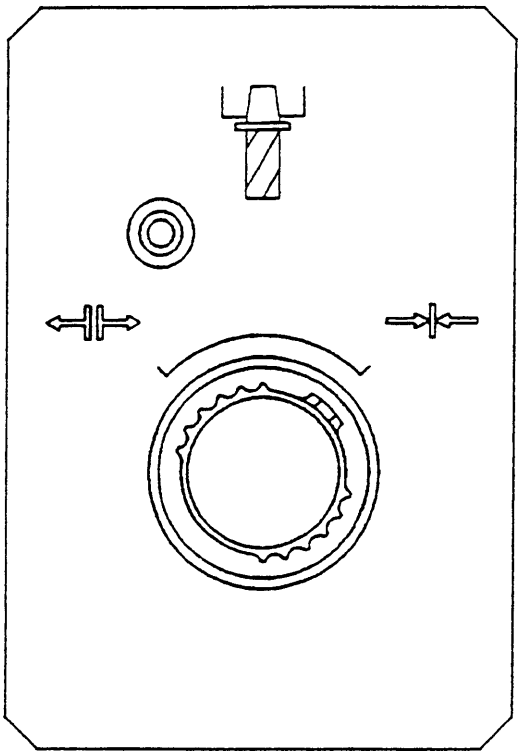
(1) Stop the spindle head at an arbitrary position.

(2) Select one of manual operation modes; JOG, RAPID, HANDLE or ZERO RETURN.

(3) Holding a tool firmly, shift to UNCLAMP the TOOL CLAMP/UNCLAMP selector push button switch, which is located on the front of the spindle head, and press it.
When the tool has been mounted, if the push button switch is released, the tip of the drawbar will push the pull-stud bolt, thus removing the tool from a spindle taper hole (hold the tool firmly lest it should drop). While the tool is unclamped, the yellow lamp located on the upper left part lights up.

(4) When mounting the tool, eliminate dust completely from the taper section of the tool shank. Clean the spindle taper hole properly with an accessory cleaner.

(5) Insert the tool shank into the spindle hole, and shift the TOOL CLAMP/UNCLAMP selector push button switch to CLAMP and press it. The tool is automatically mounted to the spindle and the yellow lamp lights off.



5. Manual Data Input (MDI) Operation

5-1 Each Axial Feed

(1) After pressing [RETURN] button to get the general screen, set the mode selection pushbutton switch to [MDI].
(2) General (MDI) is displayed on the display unit.
(3) Select the axis, which you want to mode, with the address keys, and enter the sign and the numerical value by the numerical keys then press the [RETURN] key.
(4) Press the [INSERT] button.
(5) Repeating the steps (2) and (3), input one block worth of data.
(6) Press the [PROGRAM START] button.

5-2 Coolant

(1) After pressing [RETURN] button to get the general screen, set the mode selection pushbutton switch to [MDI].
(2) General (MDI) is displayed on the display unit.
(3) Press the address key [M] .
(4) Enter [0] and [8] by operating the numerical keys then press the [RETURN] key.
(5) Press the [INSERT] button, and then, the [PROGRAM START] button.
Coolant is discharged from the nozzle at the spindle head.
(6) If you give M09 by similar operation, the coolant will stop.

5-3 Spindle Speed Change and Spindle Revolution

(1) After pressing [RETURN] button to get the general screen, set the mode selection pushbutton switch to [MDI].
(2) General (MDI) is displayed on the display unit.
(3) Press the address key [S] .
(4) Input the spindle revolution speed by the numerical keys then press the [RETURN] key.
(5) Press the [INSERT] button.
(6) Similarly, input an M code for spindle forward revolution or reverse revolution.
(7) Press the [PROGRAM START] button.
(8) To stop the spindle, give M05 by similar operation.

Example

[MDI] → [S] → [1] [5] [0] [0] → [EOB] → [M] → [0] [3] →
[INSERT] → [PROGRAM START]

5-4 Spindle Positioning

(1) After pressing RETURN button to get the general screen, set the mode selection pushbutton switch to [MDI].

(2) General (MDI) is displayed on the display unit.

(3) Press the address key M

(4) Enter 1 and 9 by operating the numerical keys then press the RETURN key.

(5) Press the INSERT button, and then, the PROGRAM START buttons, the spindle runs at a low speed and stops at its home position.

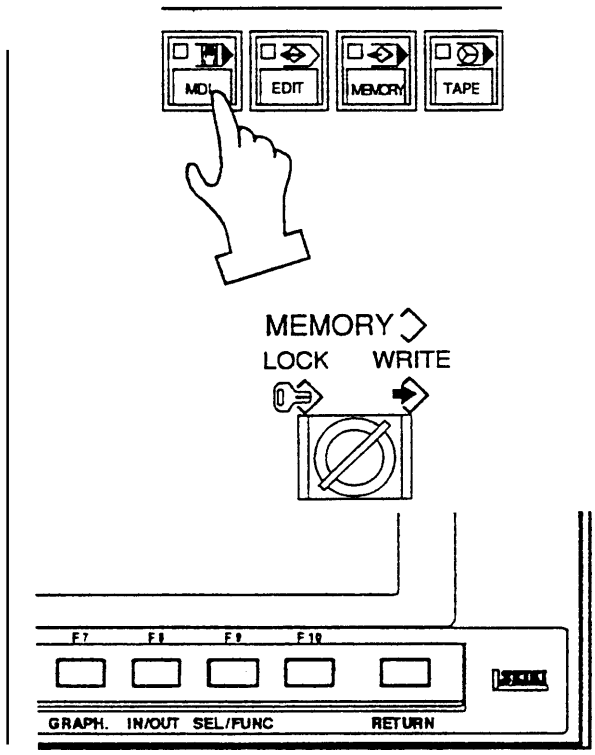
6. Registration of Program

There are following two methods to register a program into the NC unit.

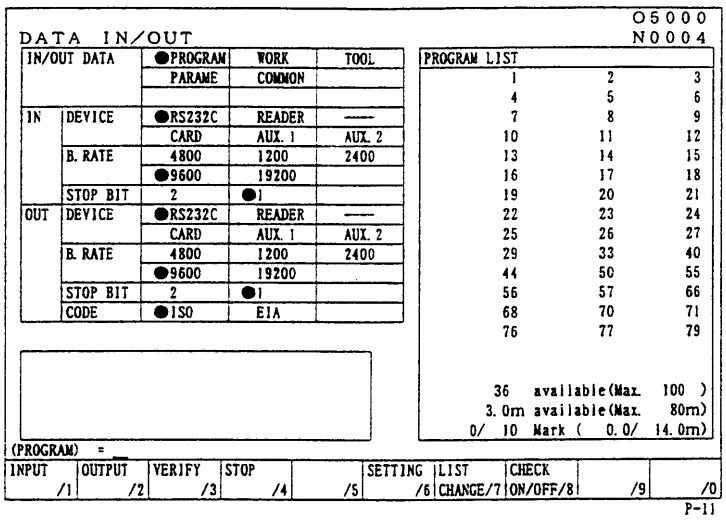
1. Registration from an external input device
2. manual registration by the address/numeral keys

6-1 Registration from an External Device

1) Connect an input device RS-232-C terminal and make a possible condition of transmission.
2) Set a mode to [EDIT].
3) Set the memory key to [WRITE].
4) Press the function key F8/IN-OUTPUT.



A right sketch is displayed.



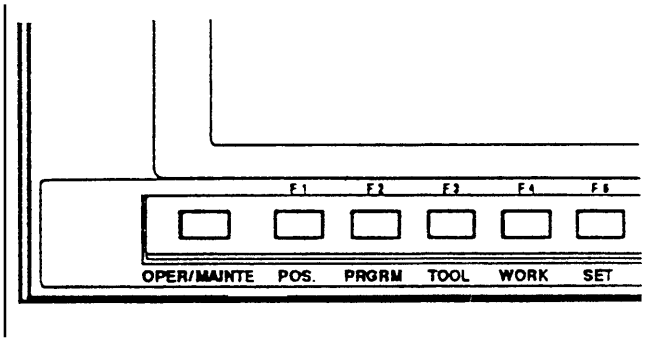
5) Press the F1/INPUT key.

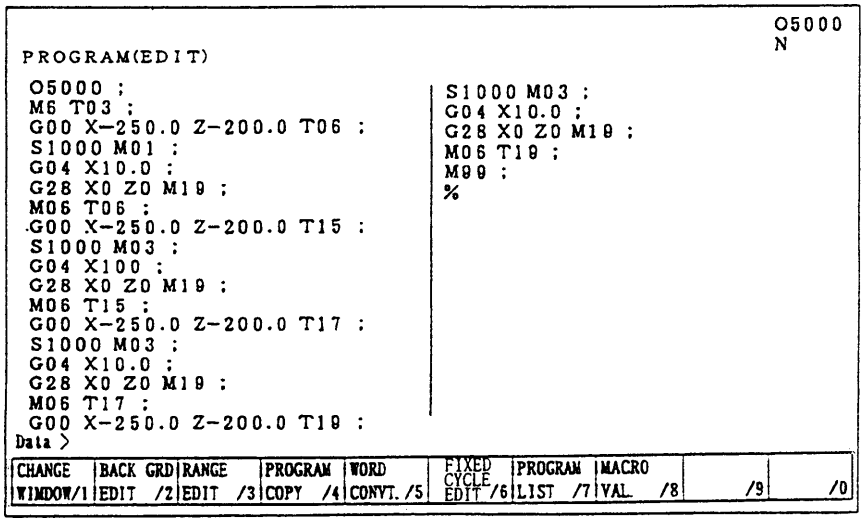
- Start reading from the first EOB of the NC program and continue until the % code.
- The program No. is registered the 0 No. registered in the input device.
- Display at the program No. list after completion of reading.
- At the time of input, the ISO/EIA information is recognized automatically.

Note) *If the program No. already registered is inputted, it becomes an alarm condition.*

6-2 Manual Registration by the Address/Numeral Keys

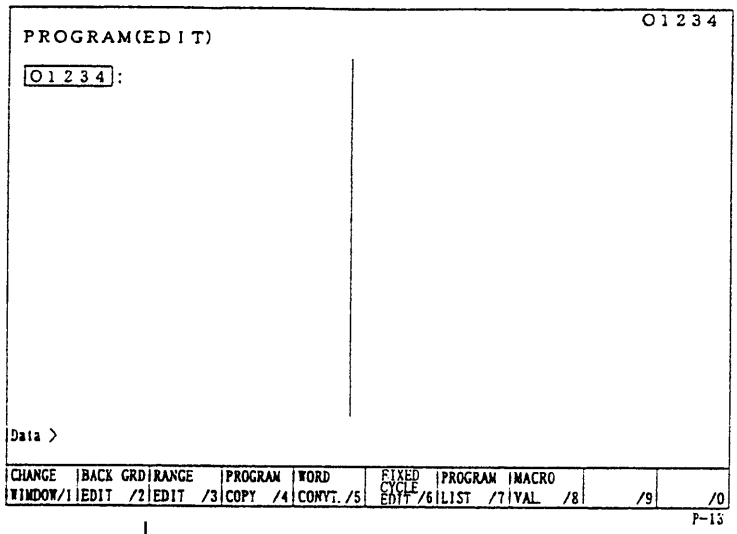
1) Set a mode [EDIT].
2) Set the memory key to [WRITE].
3) Press the F2/PRGRM key.





4) Key in a desired program No. and press the INSERT key.

Example: 0 1 2 3 4 INSRT



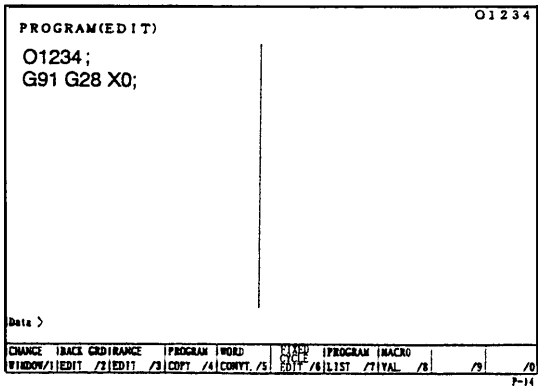
5) Set the cursor to; by the cursor key.

6) Input a program according to the order of the NC program.
Example:
Data>G91 G28 X0;

- The EOB key must be inputted at the end of one block.
- Press the CANCEL key when the data which has inputted want to be deleted. A word deletes one by one.

7) Press the RETURN key and return to the initial screen after input of all program is completed.

G 9 1 G 2 8 X 0
EOB INSRT



7. Program No. Search

There are following two methods to search a program.

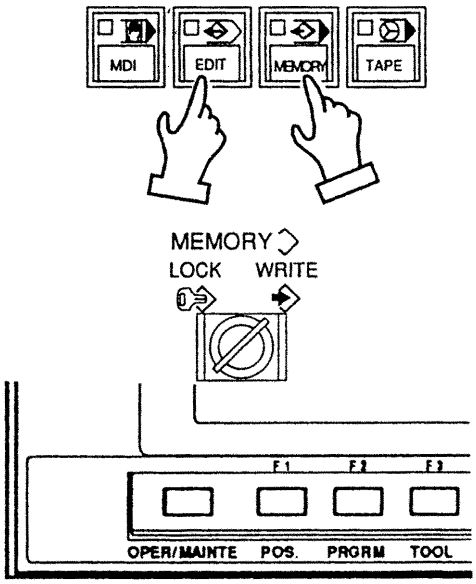
1. Search by key in a program No.
2. Search to utilize the program list.

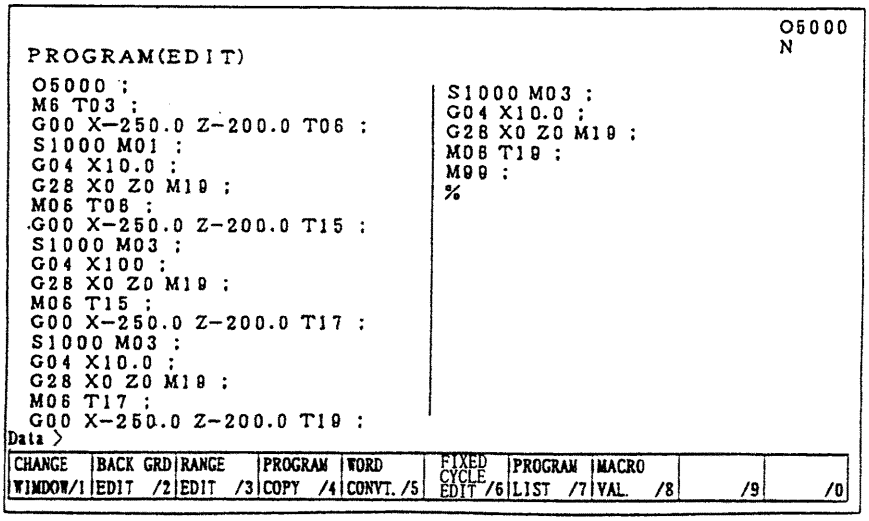
7-1 Search by Key in a Program No.

1) Set a mode to the [MEMORY] or [EDIT].

2) Set the memory key to [WRITE].

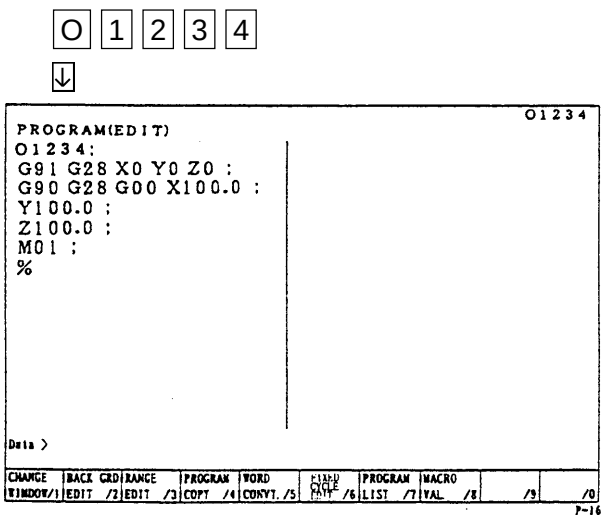
3) Press the function key F2/PRGRM.





4) Key in the program No. to be searched and press the ⇩ key.
Example: O 1 2 3 4

Calling up program is displayed.



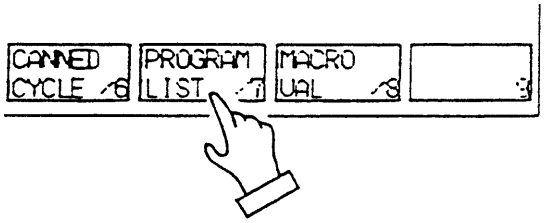
7-2 Search to Utilize the Program List.

1) Set a mode to the [MEMORY] or [EDIT].

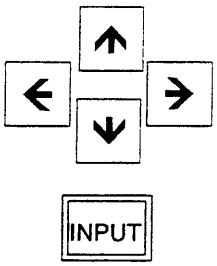
2) Set the memory key to [WRITE].

3) Press the function key F2/PRGRM.

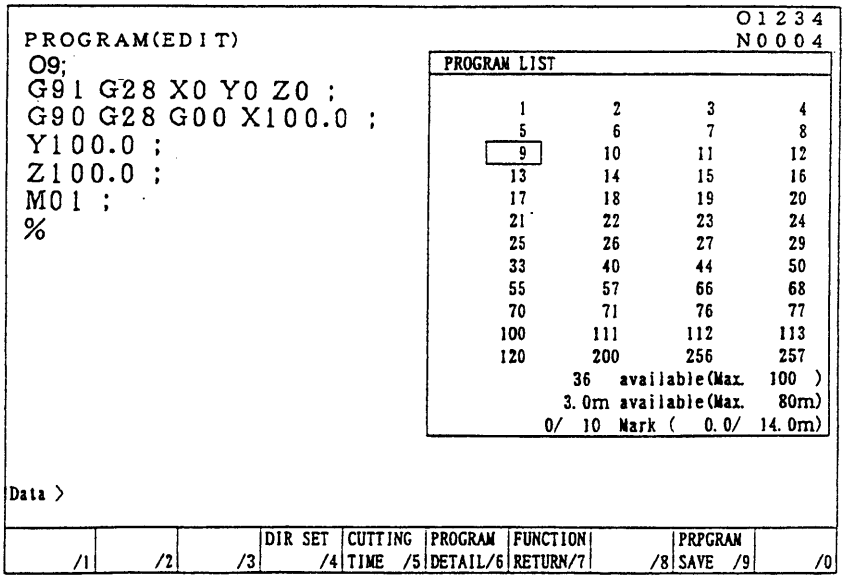
4) Press the F7/PROGRAM LIST key.



5) Set the cursor to the program No. to be searched by the cursor key and press the INPUT key.



Calling up program is displayed.



8. Edition of Program

The keys to edit a program are as follows;

INSERT : Insert a content of key input after the cursor.

ALTER : Alter a content of key input at a section of the cursor.

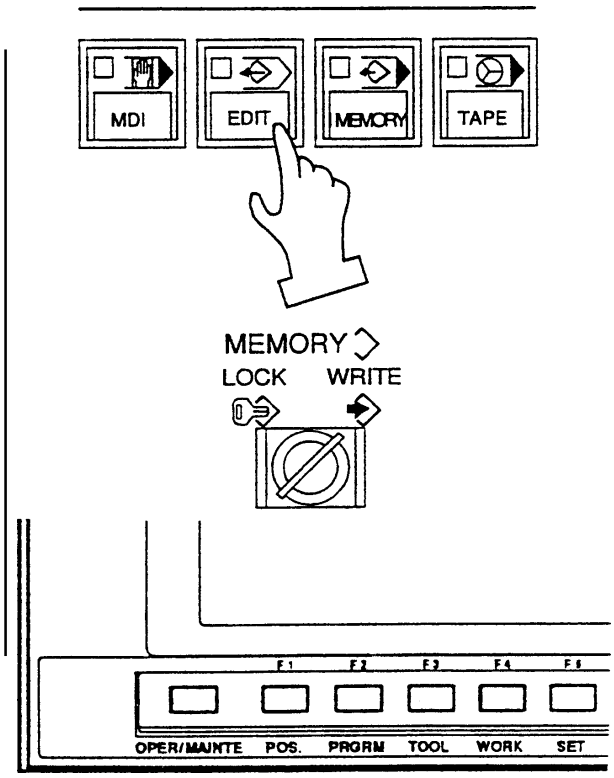
DELETE : Delete a section of the cursor.

Use it deletion of program as well.

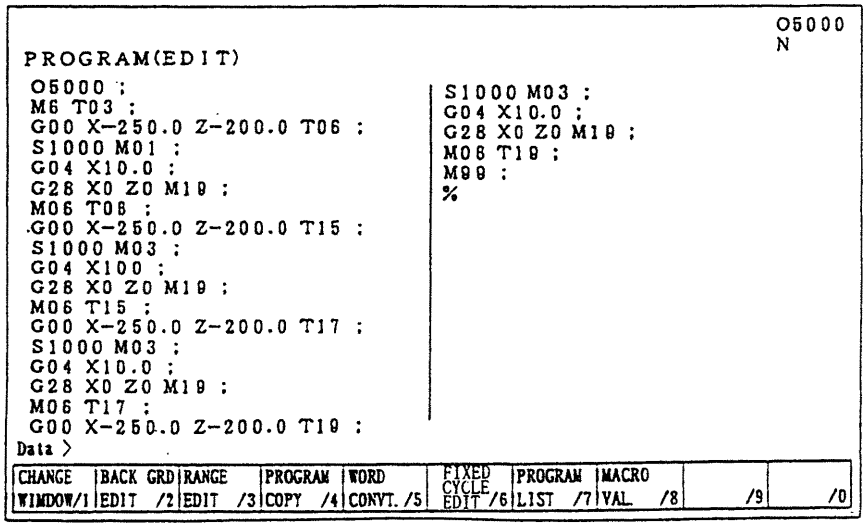
8-1 Preparation in Advance at the Time of the Edition of Program.

To edit a program, the following conditions should be made.

1) Set a mode [EDIT].
2) Set the memory key to [WRITE].



3) Press the function key F2/PRGRM.



8-2 Search of Word

A word can be searched by the following method.

1) A method by means of the page and cursor keys.

   [1] Press the page key and display the page to be edited.

   [2] Press the cursor key and move the cursor to the word to be edited.

   • The cursor moves at a block unit by the ↑ ↓ keys.

   • The cursor moves at a word unit by the ← → keys.

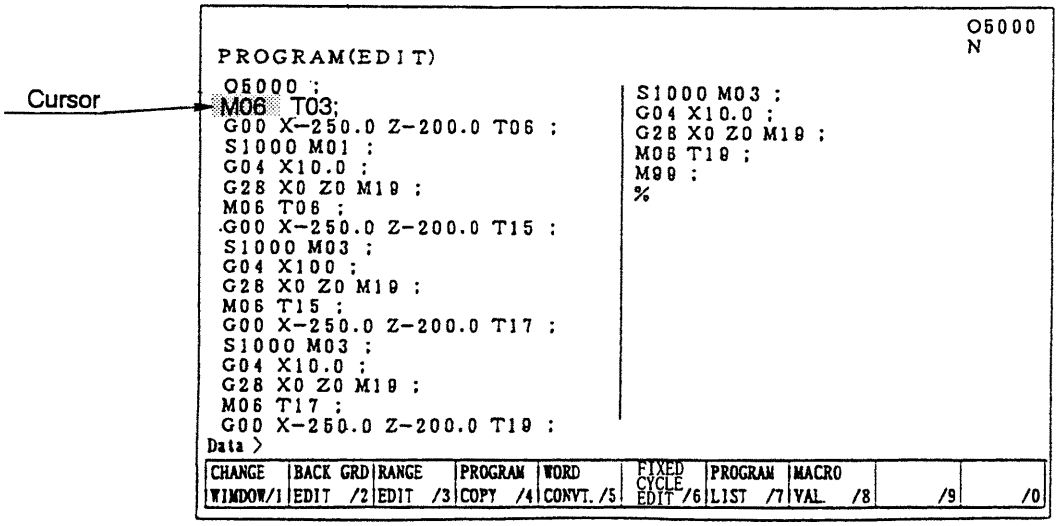
2) A method by means of word or address search.

   Since a message is displayed as “Not found” if it is not found, try it again.

   [1] Word search, No. search

   Key in the address and numerals to be searched and press ↓ key.

   Example: M06 M 0 6 ↓



When searching a section above the current position of cursor, press the ↑ key

[2] Block search

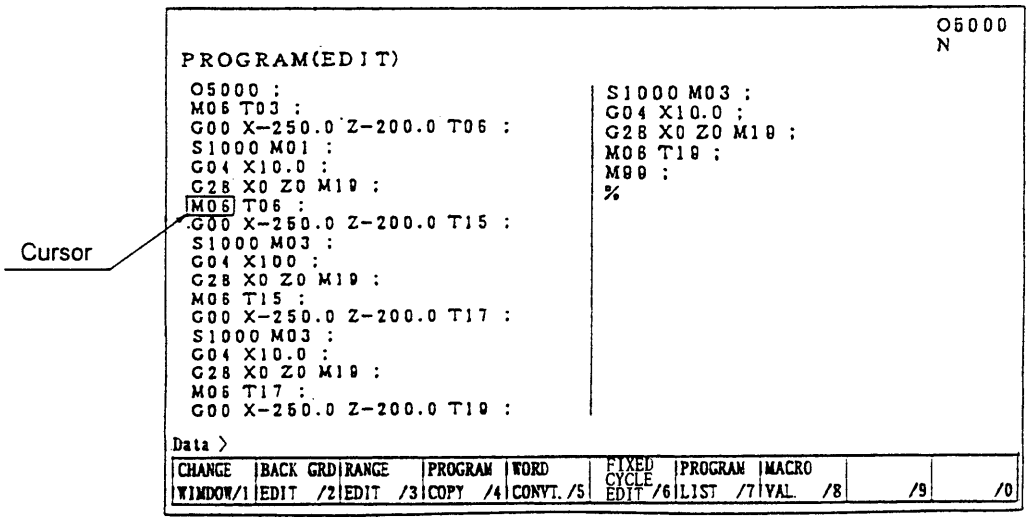
Check a word in a block and search a block which contains a relevant word only.

Key in all address and numerals of one block then press EOB and ↓ key.

Example: When searching a block of M06 T06;

M 0 6 T 0 6
EOB ↓

Note) *The EOB should be inputted at the end of a block.*



8-3 Edition of Program

(1) Insertion of word, block

New word is inserted just after the word currently located the cursor.

[1] Designate a word immediately before a section to be inserted.

[2] Key in a new data to be inserted then press the INSERT key.

Example: when inserting Y 100.0 after X-250.0

Y 1 0 0 .
0 INSRT

```
                                                                    O5000
                                                                    N
PROGRAM(EDIT)
 O5000 ;                                  S1000 M03 ;
 M6 T03 ;                                 G04 X10.0 ;
 G00 X-250.0 Z-200.0 T06 ;                G28 X0 Z0 M19 ;
 S1000 M01 ;                              M06 T19 ;
 G04 X10.0 ;                              M99 ;
 G28 X0 Z0 M19 ;                          %
 M06 T06 ;
 G00 X-250.0 Z-200.0 T15 ;
 S1000 M03 ;
 G04 X100 ;
 G28 X0 Z0 M19 ;
 M06 T15 ;
 G00 X-250.0 Z-200.0 T17 ;
 S1000 M03 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T17 ;
 G00 X-250.0 Z-200.0 T19 ;
Data >
CHANGE   BACK GRD RANGE   PROGRAM WORD     FIXED CYCLE PROGRAM MACRO
WINDOW/1 EDIT  /2 EDIT  /3 COPY  /4 CONVT. /5 EDIT /6 LIST  /7 VAL  /8  /9  /0
```

Cursor

```
                                                                    O5000
                                                                    N
PROGRAM(EDIT)
 O5000 ;                                  S1000 M03 ;
 M6 T03 ;                                 G04 X10.0 ;
 G00 X-250.0 Z-200.0 T06 ;                G28 X0 Z0 M19 ;
 S1000 M01 ;                              M06 T19 ;
 G04 X10.0 ;                              M99 ;
 G28 X0 Z0 M19 ;                          %
 M06 T06 ;
 G00 X-250.0 Z-200.0 T15 ;
 S1000 M03 ;
 G04 X100 ;
 G28 X0 Z0 M19 ;
 M06 T15 ;
 G00 X-250.0 Y100.0 Z-200.0 T17 ;
 S1000 M03 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T17 ;
 G00 X-250.0 Z-200.0 T19 ;
Data >
CHANGE   BACK GRD RANGE   PROGRAM WORD     FIXED CYCLE PROGRAM MACRO
WINDOW/1 EDIT  /2 EDIT  /3 COPY  /4 CONVT. /5 EDIT /6 LIST  /7 VAL  /8  /9  /0
```

After insertion

Cursor

[3] When inserting one block, key in data of one block and press EOB and INSERT key.

(2) Alteration of word

Alter a word, the cursor located currently, to the new word.

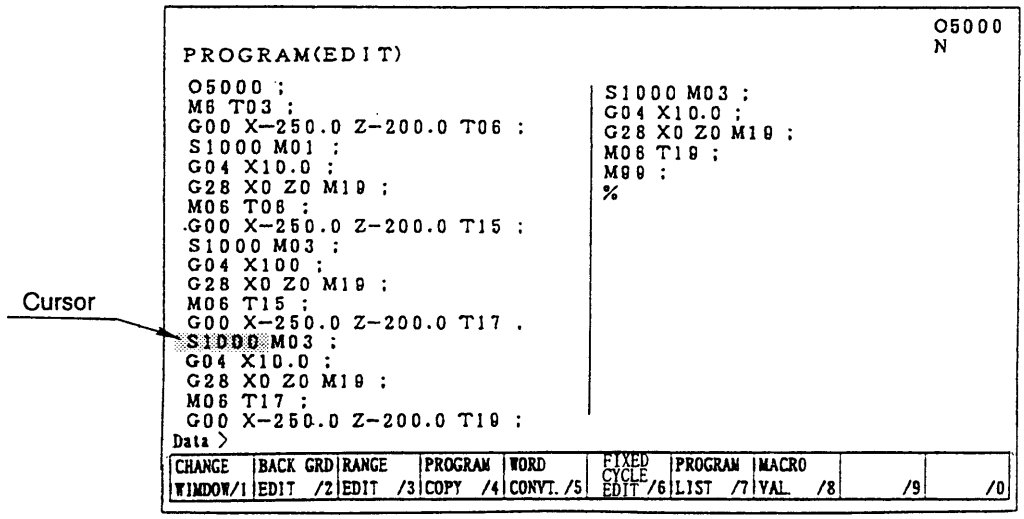
Alteration is done by a word unit.

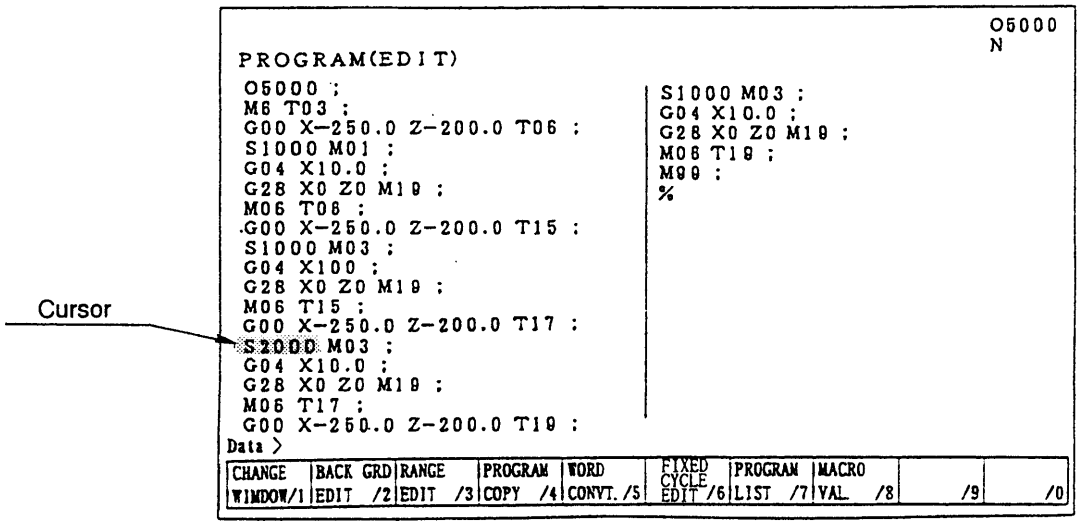
[1] Set the cursor to the word to be altered.

[2] Key in the new word then press the [ALTER] key.

Example: Alter S1000 to S2000.

[S] [2] [0] [0] [0]
[ALTER]





After alteration

S1000 replaces S2000.

(3) Deletion of word, block

A word currently located the cursor or a certain boundary of a program can be deleted.

(a) Deletion of word

[1] Set the cursor to a word to be deleted.

[2] Press the DELETE key.

Example: When deleting Z-200.0
Set the cursor to Z-200.0
then press the
DELETE key.

Cursor

```
                                                                    O5000
                                                                    N
PROGRAM(EDIT)
 O5000 ;                                  S1000 M03 ;
 M6 T03 ;                                 G04 X10.0 ;
 G00 X-250.0 Z-200.0 T06 ;                G28 X0 Z0 M19 ;
 S1000 M01 ;                              M06 T19 ;
 G04 X10.0 ;                              M99 ;
 G28 X0 Z0 M19 ;                          %
 M06 T06 ;
 G00 X-250.0 Z-200.0 T15 ;
 S1000 M03 ;
 G04 X100 ;
 G28 X0 Z0 M19 ;
 M06 T15 ;
 G00 X-250.0 Z-200.0 T17 ;
 S1000 M03 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T17 ;
 G00 X-250.0 Z-200.0 T19 ;
Data >
CHANGE WINDOW/1 | BACK GRD EDIT /2 | RANGE EDIT /3 | PROGRAM COPY /4 | WORD CONVT. /5 | FIXED CYCLE EDIT /6 | PROGRAM LIST /7 | MACRO VAL. /8 | /9 | /0
```

After deletion

Z-200.0 is deleted.

```
                                                                    O5000
                                                                    N
PROGRAM(EDIT)
 O5000 ;                                  S1000 M03 ;
 M6 T03 ;                                 G04 X10.0 ;
 G00 X-250.0 Z-200.0 T06 ;                G28 X0 Z0 M19 ;
 S1000 M01 ;                              M06 T19 ;
 G04 X10.0 ;                              M99 ;
 G28 X0 Z0 M19 ;                          %
 M06 T06 ;
 G00 X-250.0 Z-200.0 T15 ;
 S1000 M03 ;
 G04 X100 ;
 G28 X0 Z0 M19 ;
 M06 T15 ;
 G00 X-250.0 T17 ;
 S1000 M03 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T17 ;
 G00 X-250.0 Z-200.0 T19 ;
Data >
CHANGE WINDOW/1 | BACK GRD EDIT /2 | RANGE EDIT /3 | PROGRAM COPY /4 | WORD CONVT. /5 | FIXED CYCLE EDIT /6 | PROGRAM LIST /7 | MACRO VAL. /8 | /9 | /0
```

(b) Deletion of block

It can be deleted one block at a time.

[1] Set the cursor to the head of the block to be deleted.

[2] Press the EOB and DELETE.

Example: When deleting a block G04 X 10.0;

Set the cursor to G04 and press the EOB and DELETE key.

Cursor

```
                                                                    O5000
                                                                    N
PROGRAM(EDIT)
 O5000 ;                                  S1000 M03 ;
 M6 T03 ;                                 G04 X10.0 ;
 G00 X-250.0 Z-200.0 T06 ;                G28 X0 Z0 M19 ;
 S1000 M01 ;                              M06 T19 ;
 G04 X10.0 ;                              M99 ;
 G28 X0 Z0 M19 ;                          %
 M06 T06 ;
 G00 X-250.0 Z-200.0 T15 ;
 S1000 M03 ;
 G04 X100 ;
 G28 X0 Z0 M19 ;
 M06 T15 ;
 G00 X-250.0 Z-200.0 T17 ;
 S1000 M03 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T17 ;
 G00 X-250.0 Z-200.0 T19 ;
Data >
CHANGE WINDOW/1 | BACK GRD EDIT /2 | RANGE EDIT /3 | PROGRAM COPY /4 | WORD CONVT. /5 | FIXED CYCLE EDIT /6 | PROGRAM LIST /7 | MACRO VAL. /8 | /9 | /0
```

After deletion

The block G04... is deleted and program moves upward.

```
                                                                    O5000
                                                                    N
PROGRAM(EDIT)
 O5000 ;                                  S1000 M03 ;
 M6 T03 ;                                 G28 X0 Z0 M19 ;
 G00 X-250.0 Z-200.0 T06 ;                M06 T19 ;
 S1000 M01 ;                              M99 ;
 G04 X10.0 ;                              %
 G28 X0 Z0 M19 ;
 M06 T06 ;
 G00 X-250.0 Z-200.0 T15 ;
 S1000 M03 ;
 G04 X100 ;
 G28 X0 Z0 M19 ;
 M06 T15 ;
 G00 X-250.0 Z-200.0 T17 ;
 S1000 M03 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T17 ;
 G00 X-250.0 Z-200.0 T19 ;
Data >
CHANGE WINDOW/1 | BACK GRD EDIT /2 | RANGE EDIT /3 | PROGRAM COPY /4 | WORD CONVT. /5 | FIXED CYCLE EDIT /6 | PROGRAM LIST /7 | MACRO VAL. /8 | /9 | /0
```

(c) Boundary deletion

Delete blocks after the cursor to before the designated sequence No.

[1] Set the cursor to the head word to be deleted.

[2] Key in the sequence No. just after the last block to be deleted and press the DELETE key.

Note) *Search the sequence No. before deletion and check how far is it deleted.*

Example: When deleting

S1000 M03;

⋮

G28 X0 Z0 M19;

Set the cursor to S1000 and press N 3 DELETE key.

Cursor

```
                                                                    O5000
                                                                    N
PROGRAM(EDIT)
 O5000 ;                                  S1000 M03 ;
 M6 T03 ;                                 G04 X10.0 ;
 G00 X-250.0 Z-200.0 T06 ;                G28 X0 Z0 M19 ;
 S1000 M01 ;                              M06 T19 ;
 G04 X10.0 ;                              M99 ;
 G28 X0 Z0 M19 ;                          %
 M06 T06 ;
 G00 X-250.0 Z-200.0 T15 ;
 S1000 M03 ;
 G04 X100 ;
 G28 X0 Z0 M19 ;
 M06 T15 ;
 G00 X-250.0 Z-200.0 T17 ;
 S1000 M03 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T17 ;
 G00 X-250.0 Z-200.0 T19 ;
Data >
| CHANGE WINDOW/1 | BACK GRD EDIT /2 | RANGE EDIT /3 | PROGRAM COPY /4 | WORD CONVT. /5 | FIXED CYCLE EDIT /6 | PROGRAM LIST /7 | MACRO VAL. /8 | /9 | /0 |
```

After alteration

The program moves upward.

```
                                                                    O5000
                                                                    N
PROGRAM(EDIT)
 O5000 ;                                  M06 T19 ;
 M6 T03 ;                                 M99 ;
 G00 X-250.0 Z-200.0 T06 ;                %
 S1000 M01 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T06 ;
 G00 X-250.0 Z-200.0 T15 ;
 S1000 M03 ;
 G04 X100 ;
 G28 X0 Z0 M19 ;
 M06 T15 ;
 G00 X-250.0 Z-200.0 T17 ;
 S1000 M03 ;
 G04 X10.0 ;
 G28 X0 Z0 M19 ;
 M06 T17 ;
 G00 X-250.0 Z-200.0 T19 ;
Data >
| CHANGE WINDOW/1 | BACK GRD EDIT /2 | RANGE EDIT /3 | PROGRAM COPY /4 | WORD CONVT. /5 | FIXED CYCLE EDIT /6 | PROGRAM LIST /7 | MACRO VAL. /8 | /9 | /0 |
```

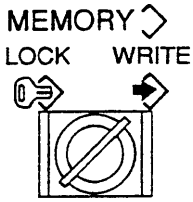
8-4 Back Ground Editing

Generally, “Editing” means front side editing, however this editing could not watch contents of program and also edition is not available while executing a program.

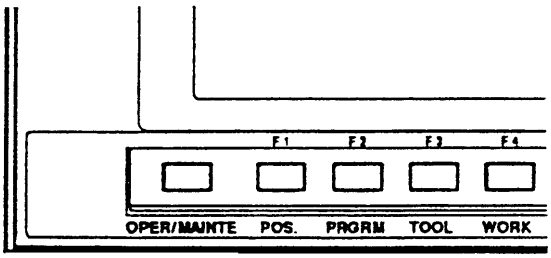
In fact, giving a possibility to edit a program while executing a program is a back ground editing.

- An editing is available to other than currently executing program.
- A program under back ground editing can not execute.
- Editing can be done both manual and automatic mode.

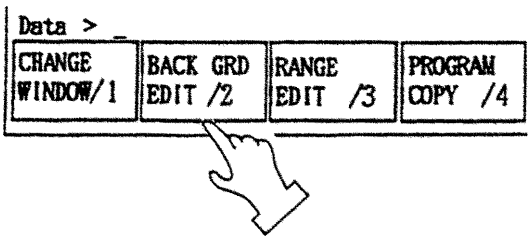
1) Turn the memory key to [WRITE].

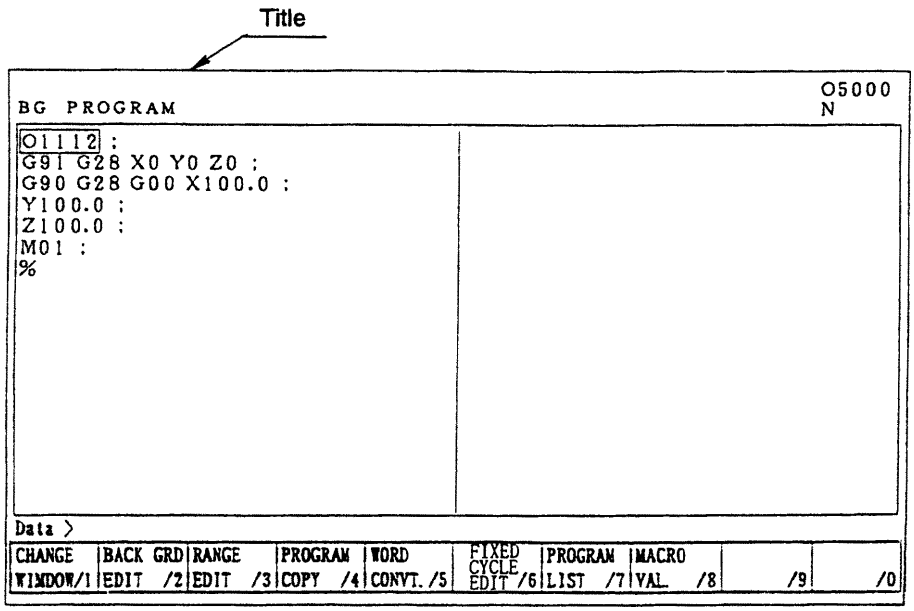


2) Press the F2/PROGRAM key.



3) Press the F2/BACK GRD EDIT key.
A title of the screen becomes a “Back ground editing”.





4) Search a program wanted to edit.
A procedure of search is exactly same as a (front) editing.

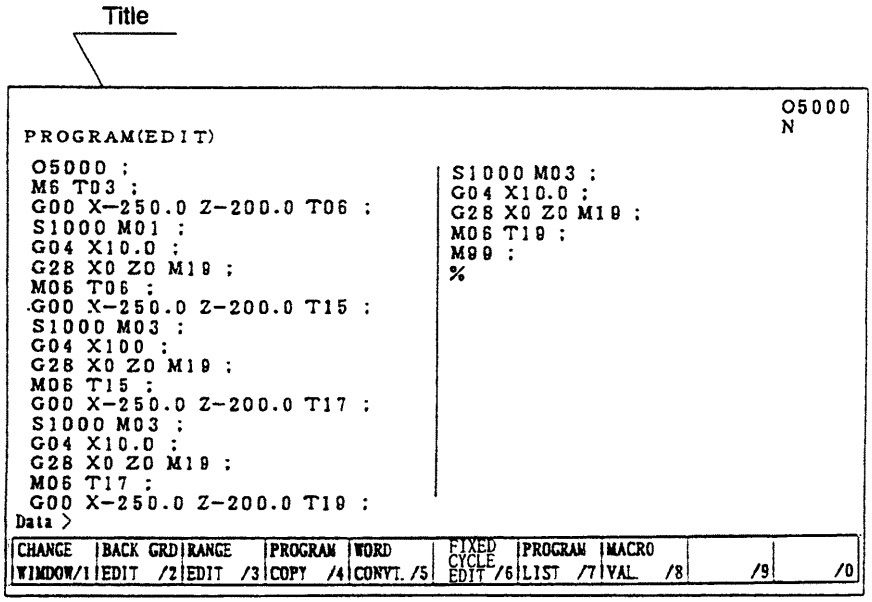
Caution

Never execute a reset operation, since the machine will stops if reset is done during machine operation at the time of back ground editing.

5) Execute edition of program.
A procedure of edition is exactly same as a (front) editing.

6) End of back ground editing

[1] Press the F2/BACK GRD EDIT key.
A title of screen becomes a “Program”.
It becomes normal editing screen.



8-5 Copy of Program

A program being displayed can be copied on the other number and displayed.

1) Display a program wanted to be copied.
2) Key in a new program No. and press INPUT key.

Example: When altering to O2001 — O 2 0 0 1 INPUT

8-6 Editing Procedure of Range Designation (Expanded Tape Editing/Option)

Designate a range of program and this “Insertion”, “Deletion” or “Storage” can be done at the program screen.

- Starting of range editing
  Operation..... Press the F3/RANGE EDIT key.
  The function changes into that for range editing.
- Designation of range
  Operation..... Press the F4/RANGE SET key.
  The cursor becomes a frame.
  A cursor range increases by moving this. That is a range.
- Release a designation of range
  Operation..... Press the F4/RANGE SET key and F3/EXIT key.
  It becomes a normal cursor and a range is released.
- Storage of range
  Operation..... Press the F5/RANGE STORE key.
  A part of range is stored.
  If a size of range exceeds 2,048 characters (approx. 5m), “Range exceeds a limit.” is displayed and not stored.
  It can be used by the range insertion at any time, since it is stored until a power turned off.
- Insertion of range
  Operation..... Press the F6/STORE→INSERT key.
  Insert a part of storage after the cursor by storage of range.
- Deletion of range
  Operation..... Press the F7/RANGE DELETE key.
  A part designated by range is deleted.
  If the size of range exceeds 2,048 characters (approx. 5m), a message “Range exceeds limit” is displayed and the storing is ineffective.
  While the source power is kept on, the stored data is maintained and the data is available by F8/DELETE→INSERT whenever desired.
- Inserting deletion
  Operation..... Press the F8/DELETE→INSERT key.
  The portion stored by range deletion is inserted at the cursor position.

8-7 Alteration of Word (Expanded Tape Editing)

At the program screen, search a designated word to be altered in the program and rewrite it to a word to be altered.

There are following two methods for alteration of word.

1) Search one word each and alter if after confirmation.
2) Alter words collectively (or after words continuously with display an altering condition).
   The words to be altered collectively are maximum 400,000 words.

A method to search a word is the same as the word search, words combined one character of alphabet (or "#" mark) and numerals becomes the number search as words showing the numeral value and recognize it with or without a decimal point.

For example, of changing "X.1" to "X.5", a row of characters such as "X0.1", "X0.100", "X00.10", "X.1" or "X.100" which coincide with "X.1" as numeral value, become the object of alteration and change all to "X.5".

Also, if adding a "?" mark on the word to be changed as "X.1?" such as changing "X.1?" to "X.5", a row of characters "X.1" becomes the object of alteration, then "X1" becomes "X.5" and "X.100" becomes "X.500", however, "X.01", "X0.100" and "X00.10" does not have the row of character "X.1" so it does not become an object of alteration.

Outline of operation

[1] Start alteration of word by pressing the F5/WORD CONVT. key.
INPUT in the order of a word to search and alter then a word to replace it.
If replacing word is blank, it becomes a deletion.

[2] When searching a word one by one, designate a searching direction by the cursor ↑, ↓ and press the Y key if found a word to be altered.

[3] For altering a word contained in many places of the file collectively, designate the searching direction by the function menu. Confirm that such overall alteration is unmistakable, then press Y key.
Once word alteration starts, it goes on to the beginning or end of the file, depending on the searching direction. For suspending the alteration,
Press the F5/EXIT key.
In case of the alteration of words collectively as mentioned item [3], alter it except a row of characters in the comment. To change inside of comment, alter it by the method [2].

OPERATION	MESSAGE
F5/WORD CONVT.	“INPUT a word to be altered.”
Input the word to be altered	“Input the word to be altered.”
Input the word to be replaced	“Alter word from....to....”
	The words are to be searched by cursor using ↑, ↓ keys. “Alter word from....to....”
... INPUT as it is blank	“Delete a word.....”

...Alteration of word will be completed even if pressing the CANCEL key from ● mark.

...Designate a searching direction of word by ↑, ↓ key.

...If it is not found ..... “Not found”

...“Search by cursor ↑, ↓ .”

...If it is found ..... “Alter a word....to....”

...Alter by Y key.

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.

...By operating ↑, ↓ keys collective alteration starts.

...Alteration is executed by pressing Y key.

When alteration has be completed ..... “Word replaced”

When the word is not found ..... “Not found”,

..... “0 word replaced”

Press F5/EXIT key. ..... “Alteration suspended”

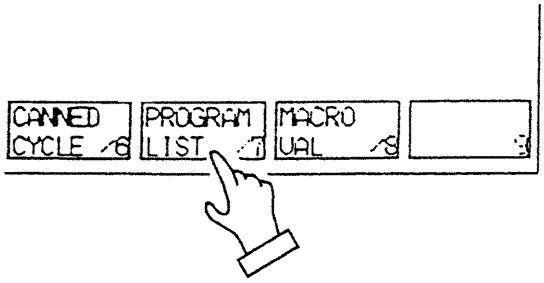
8-8 Deletion of Program

There are following two methods to delete a program.

1. Delete it by the program list screen.
2. Delete it by key input at the program screen.

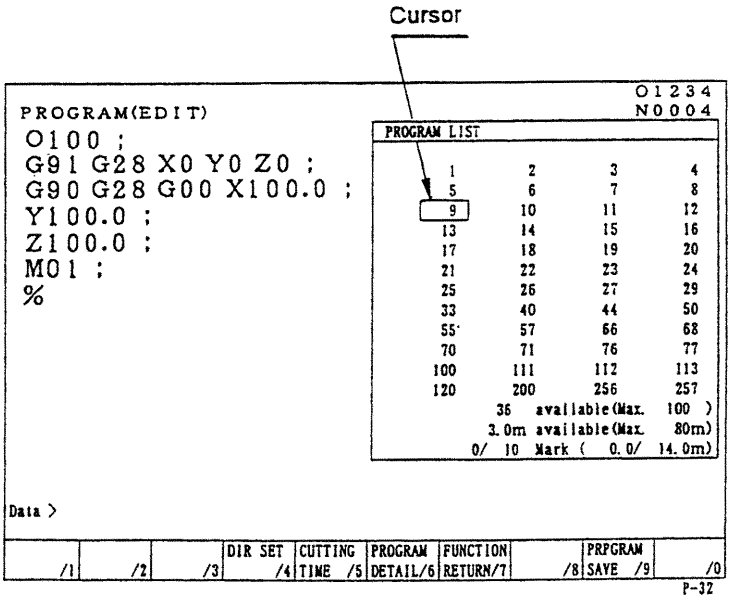
1) Deleting method by the program list screen.

[1] Press the F7/ PROGRAM LIST key at the program screen.



[2] Set the cursor at the program to be deleted by the program list.

Press the cursor ↑ ← → ↓ key.



[3] Press the DELETE key.

[4] Against a message “Is it all right to delete?”, press the Y key if you agree.
A program which is designated by the cursor is deleted.

2) Deleting method by key input

[1] Display the program screen.

[2] Key in the program No. to be deleted and press the DELETE key.
Example: In case of deleting O100

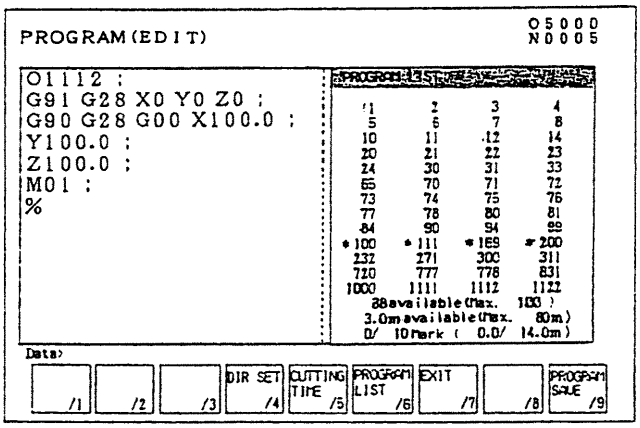
O 1 0 0 DELET

[3] Against a message “Is it all right to delete?”, press the Y key if you agree.
A program keyed in is deleted.

3) Continuous deletion by Program No.

[1] Press F7/PROGRAM LIST.

[2] Place the cursor at the Program No. to be deleted then press the SPACE bar.
An asterisk marked at the head of the Program Nos. selected.
Example: Screen display shown below is the case of deleting Program
Nos. O100, O111, O169, O200.



[3] Press the DELETE key.
For deleting the entire program, press ORIGIN and DELETE keys.

8-9 Arrangement of Program

When editing a program, a size of program becomes larger than actual size occasionally. In this case, available memory can be increased a little by arrangement of program. This operation is called “Condensation”.

Operation ... Confirm that it is in the editing mode, not background editing, and NC is in reset condition. (When there is secondary series or background drawings, it must also be in reset condition.)
Press the F1/CONDENSE key.
A comment message “Condensing program” appears on the display, then after a while, it changes to “Condensation complete”.
It takes several seconds to several minutes to complete program condensation, depending on the size of memory and the condition of memory usage. If any key is touched during the midst of this process, the comment message “Condensing program” disappears and the process is suspended.

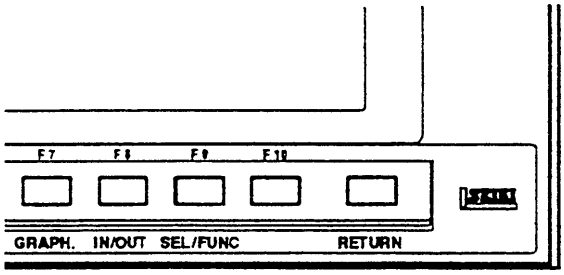
Caution

If source power is switched off during “Condensation”, the program is destroyed. When a program is found to be abnormal, initialize the program memory, and input the program anew.

8-10 Process After Edition

Press the RETURN key.

Return to the initial screen.

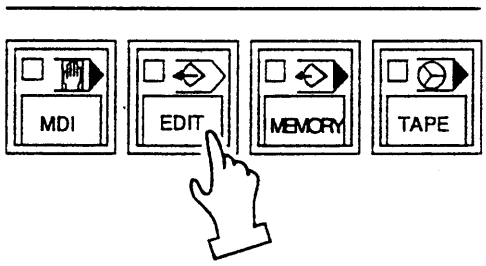


9. Output of Program

NC program can be outputted to the external in/output equipment.

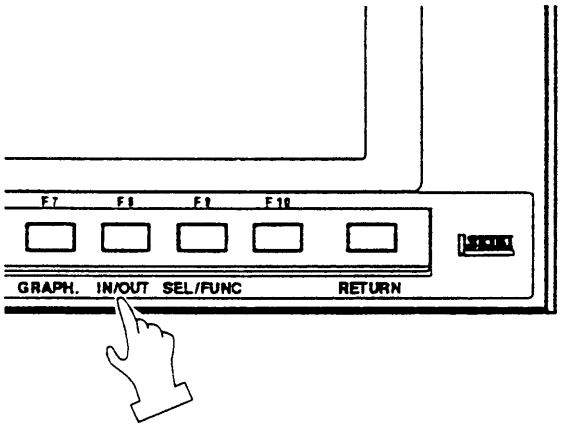
1) Connect an output device to the RS-232-C terminal and make it ready.
2) Make a mode selection to [EDIT].

Refer to the instruction manual of output device.

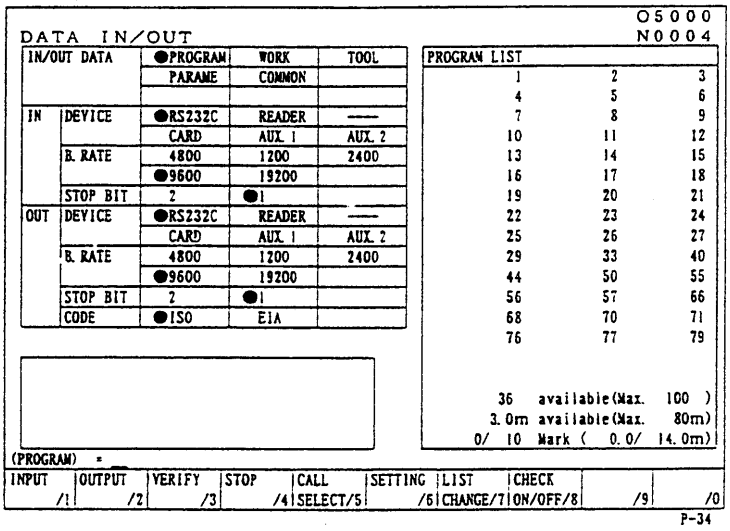


3) Press the function key F8/IN/OUT.

Note) *When "Dondon FD" is on the screen display, press the* F5/DONDON FD COMPLETE *key, and get the "Data Output" screen.*

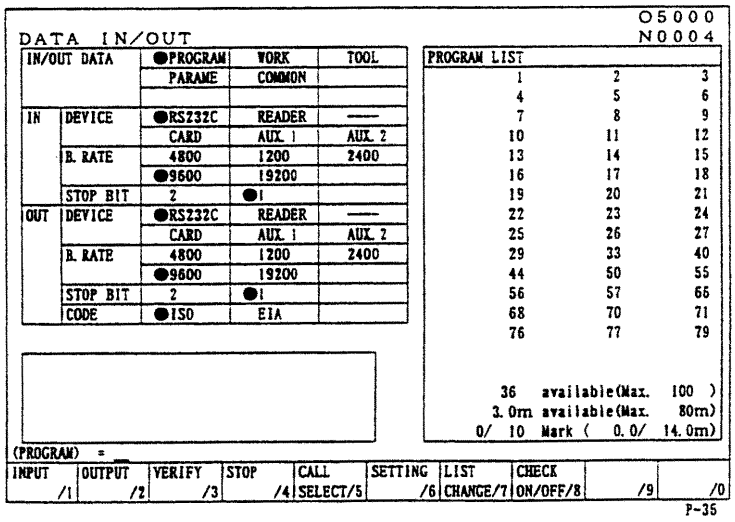


A right sketch is displayed.



4) By pressing F7/ LIST CHANGE, switch over the screen display from “Program No. List” to “Program No. Detail”.

- To select Program No.
  Place the cursor at the Program No. to be select then press the SPACE .
  An asterisk is marked at the head of the Program No. selected. When selecting all programs, repeat pressing “ORIGIN” several times until the mark “*” is displayed.
  Example: O5, O6, O7.



5) Press the RETURN key after completion of output and return to the original screen.

10. Automatic Operation

10-1 When Machining the First Workpiece, Checking the Newly Created Program

(1) Go through the steps (1) through (5) mentioned in 4-6-2 “Program Memory Operation”
(2) Turn on the [SINGLE BLOCK] switch. (When this is done, it is recommended to set a rapid override value to 10% for safety.)
(3) Press the PROGRAM START button.
(4) Check one block worth of functioning, press the PROGRAM START button again to sequentially advance a program.

10-2 Program Memory Operation

Caution

Close the front door and the ATC magazine door.

(1) Check that the ALARM lamp on the main operation panel is not lit up.
(2) Check and correct tool length and tool compensation data, if necessary.
(3) Set the mode selector switch to “MEMORY”. For normal operation, at this time, turn off the DRY RUN, OPTIONAL STOP, MACHINE LOCK, BLOCK SKIP and SINGLE BLOCK switches, and set the FEEDRATE OVERRIDE, SPINDLE OVERRIDE and RAPID OVERRIDE switches to 100%.
(4) Press the RESET button on the setting operation panel.
(5) Call the PROGRAM AUTOMATIC to the display screen by pressing the F2/PRGRM button to check the initial state.

⚠ WARNING

In order to suspend the machine durintg operation, press STOP button on the operation panel or turn [ON] SIGNAL BLOCK switch.

In case of emergency, press the EMERGENCY STOP button to immediately stop the machine on the operation panel.

10-3 When Inserting Manual Operation in Automatic Operation

(1) Press the PROGRAM STOP push button switch (while program operation is suspended, a red lamp lights up.) or suspend the machine operation by turnig [ON] SINGLE BLOCK switch.

(2) Select the operation mode selector push button switch either [HANDLE] or [JOG] and perform manual operation.

(3) On completion of manual operation, return the mode selection to [MEMORY].

(4) Press the PROGRAM START button to restart the program.

Caution

Even when the PROGRAM STOP button is pressed, the M, S and T functions continue until their actions are completed. Manual operation is available only after these actions are completed.

10-4 When Performing MDI Operation in Automatic Operation

(1) Turn on the [SINGLE BLOCK] switch on the operation panel.

(2) Shift the mode selection switch do [MDI] after the machine stops.

(3) Press RETURN button to get the general screen.

(4) Enter necessary actions with the address keys and numerical keys, and press the INSERT button.

(5) Press the PROGRAM START button after confirming the data of one block has been input.

(6) To restart the automatic operation, shift the mode selector switch to [MEMORY] or [TAPE] and turn off the SINGLE BLOCK switch.

(7) Press the PROGRAM START button.

Caution

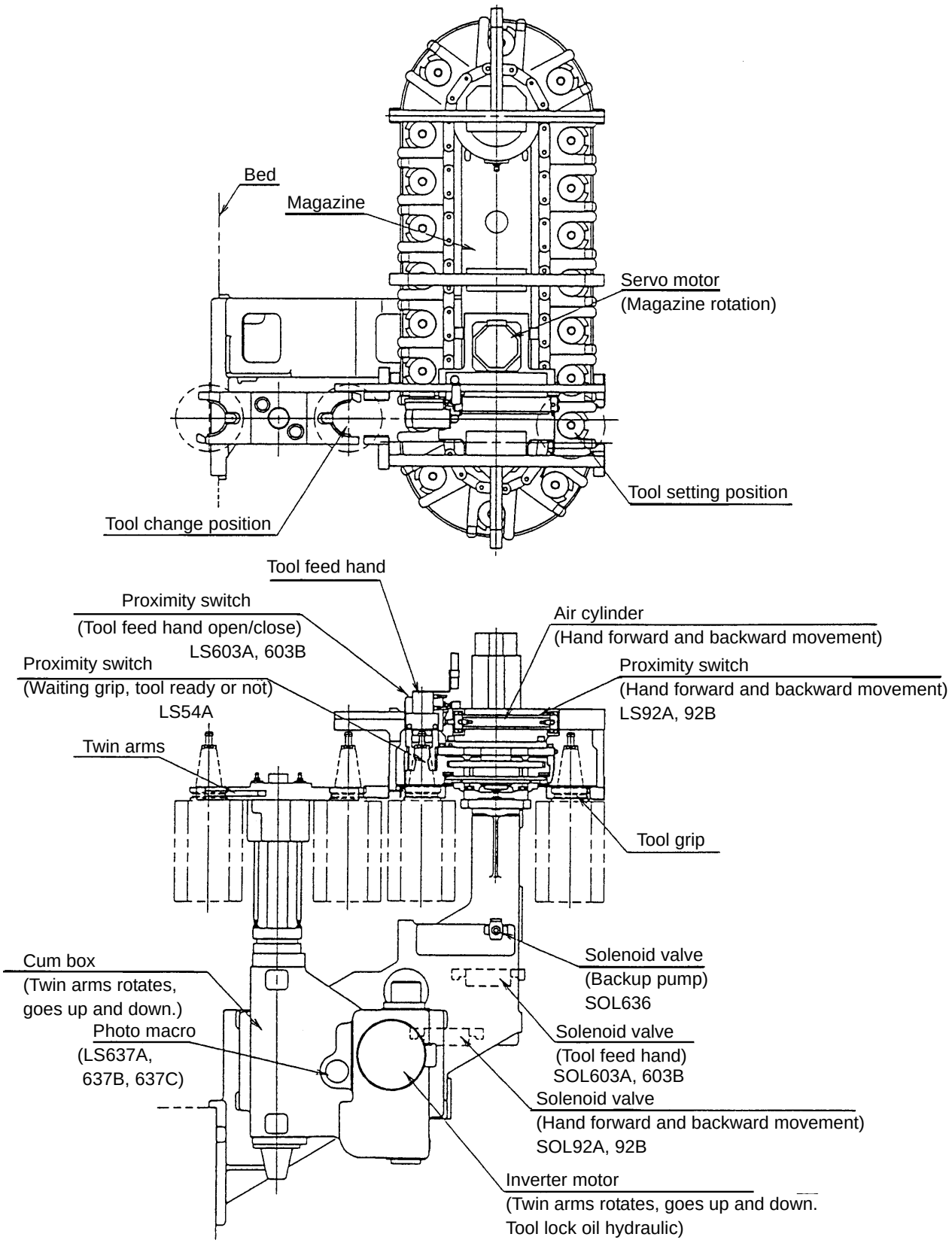
1. When an auto command immediately before is a canned cycle and you specify an action other than the canned cycle by MDI operation, be sure to specify a necessary G code.
2. When MDI operation is inserted in the state mentioned in Note 1, you must specify hole machining data preceding MDI insertion, if you want to execute the auto command again.

⚠ WARNING

After you enter actions by MDI operation, if you forget to execute them (due to a certain reason such as leaving a work site) and restart automatic operation, the machine functions in an unexpected way and endangers you, because buffer contents for automatic operation have been replaced with unexecuted MDI buffer contents. Be full aware of it.

11. Operation of Automatic Tool Changer (ATC)

11-1 Names of ATC Each Section



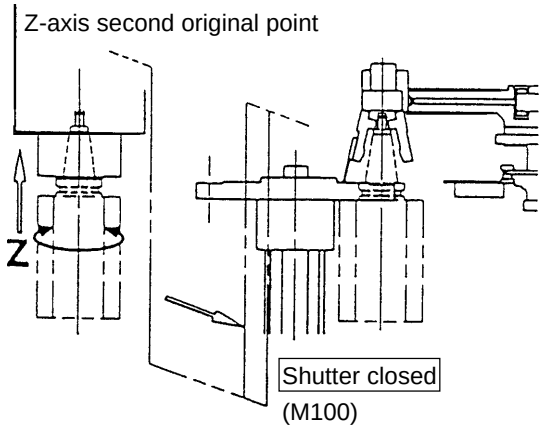
11-2 Explanation of Actions

ATC operates continuously according to command contents.

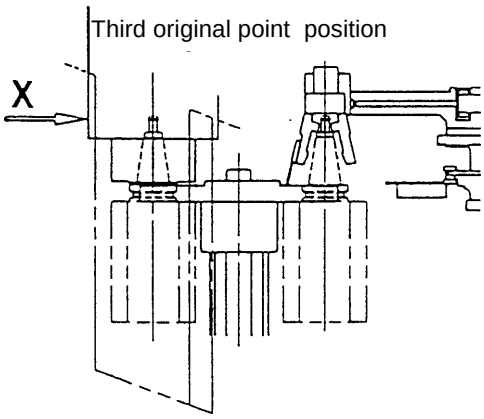
1. Change Spindle and Waiting Tool……M06 Operation

***Note**) (M1**) in the figures operates only at the maintenance mode.*

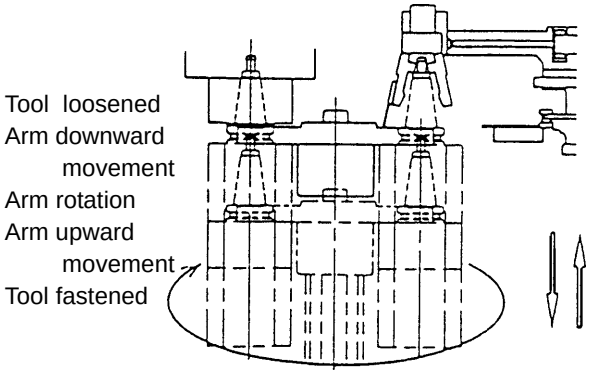
1) Open the shutter, and move to the spindle orientation Z-axis second original point.



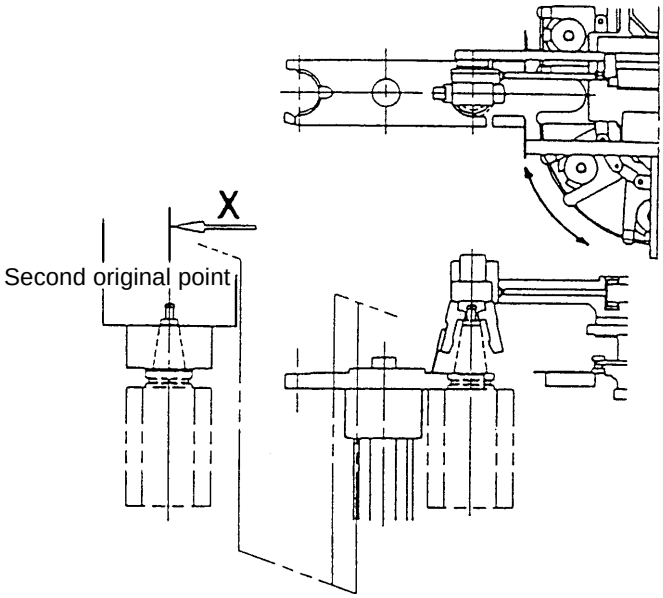
2) Move to X-axis third original point (tool change position).



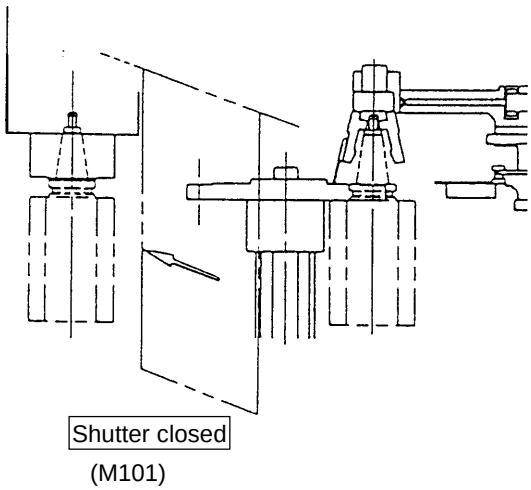
3) Tool change action (M117)



4) Move to X-axis second original point, and index magazine of waiting tool number.



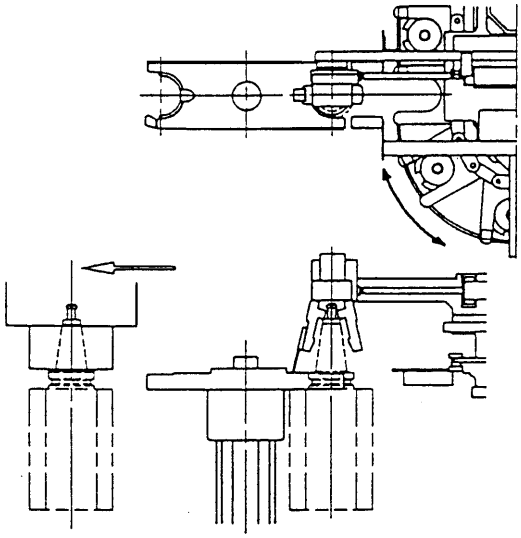
5) Shutter is closed.



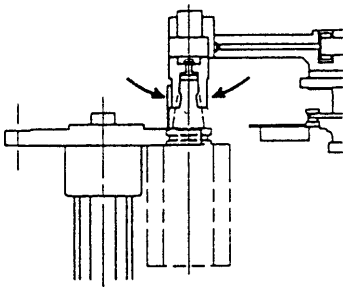
2. Call Tool to Waiting Position……T01~T20 (T30 Option)

***Note**) (M1**) in the figures operates only at the maintenance mode.*

1) Index magazine of waiting tool number.

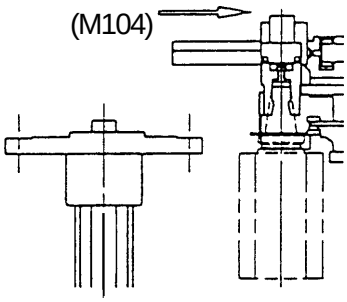


2) Feed hand is closed (M102).

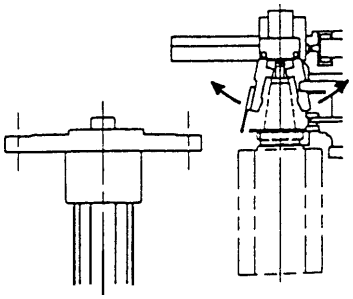


3) Move feed hand to magazine side.

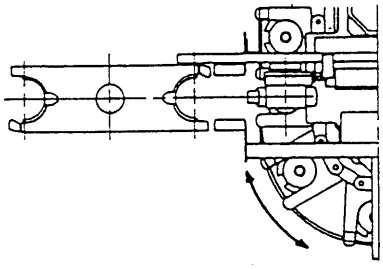
(M104)



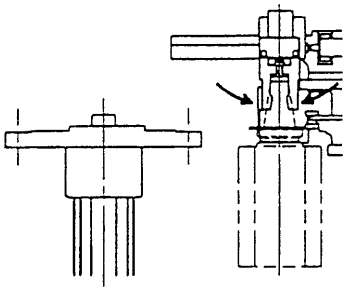
4) Feed hand is open (M103).



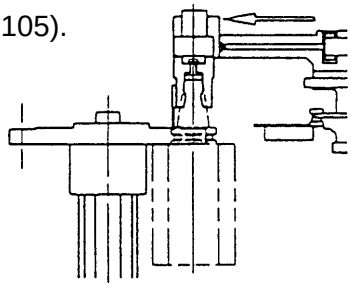
5) Index magazine of T code number.



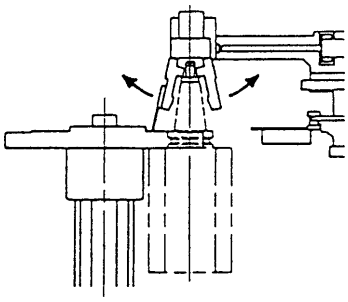
6) Feed hand is closed (M102).



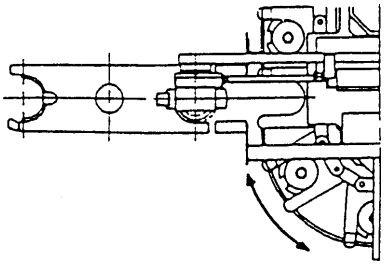
7) Move feed hand to twin arms side (M105).



8) Feed hand is open (M103).



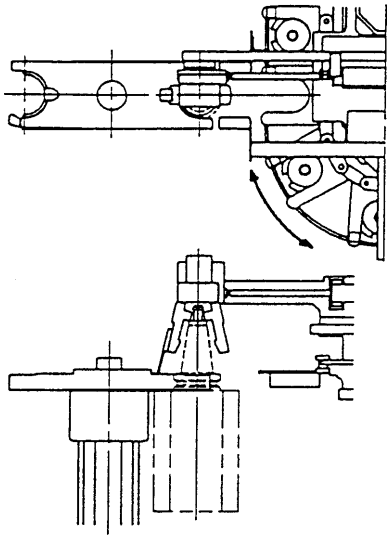
9) Index magazine of spindle tool number.



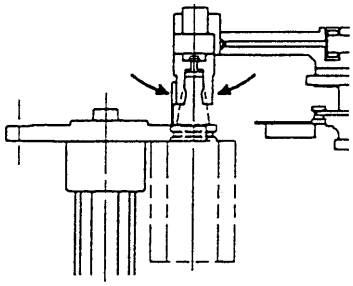
3. Return Waiting Tool to Magazine…….T00 Action

***Note**) (M1**) in the figures operates only at the maintenance mode.*

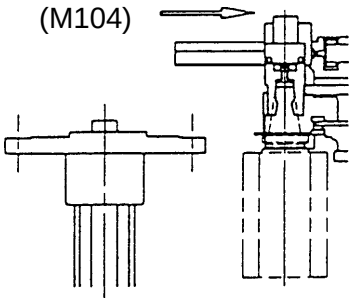
1) Index magazine of waiting tool number.



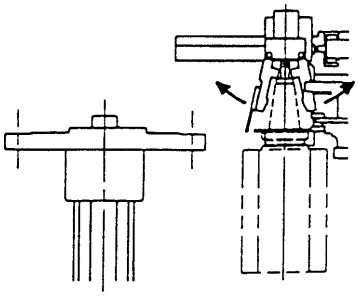
2) Feed hand is closed (M102).



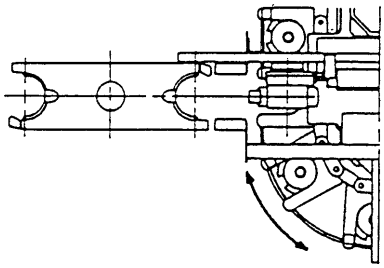
3) Move feed hand to magazine side. (M104)



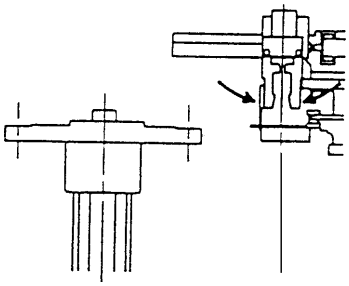
4) Feed hand is open (M103).



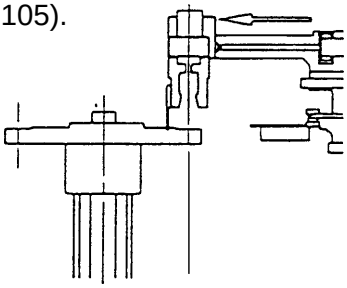
5) Index magazine of spindle tool number.



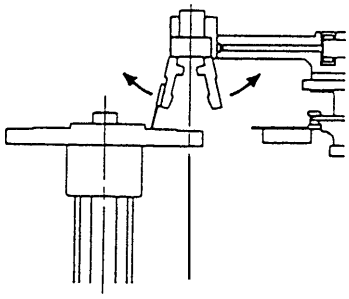
6) Feed hand is closed (M102).



7) Move feed hand to twin arms side (M105).



8) Feed hand is open (M103).



11-3 Rotation of Tool Storage Magazine (at Single Mode)

The magazine is rotated while the pushbutton switch on the ATC single operation panel is being pressed. When requested pot No. comes close to the specified position, release the pushbutton, which makes the magazine decelerate, position and stop.

11-4 Attachment and Detachment of Tools

Attachment and detachment of tools are accomplished through the tool attachment/ detachment window on the right side of the machine. The procedures are as follows.

[1] Keep pressing manual indexing button “ON” on ATC control panel until the button is lit on.

[2] Index the magazine to be attached or detached to the tool attachment/detachment position. (Operate until this stage, keep closing the tool attachment/detachment window.)

Tool attachment

[3] Align the tool holder V groove and the magazine grip. (Refer to the next section for tool key grooving method.)

[4] Push a tool horizontally into the magazine side. (Pay attention to the spring of the magazine clip section when pushing.)

Tool detachment

[5] Hold the taper section of the tool holder and the tool shank section firmly with both hands, and pull them out horizontally towards you.

[6] Close the tool attachment/detachment window

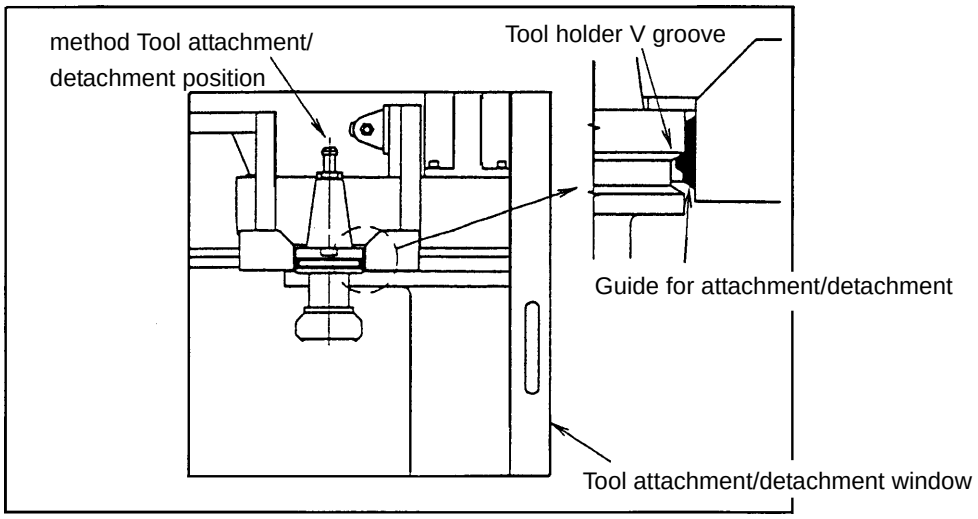
[7] Press the manual indexing button “OFF” on ATC control panel for three seconds or more.

Magazine manual indexing is executed by normal/reverse rotation selection button on ATC control panel. It rotates per pitch by inching operation, and rotates continuously by keeping pressing the button.

Caution

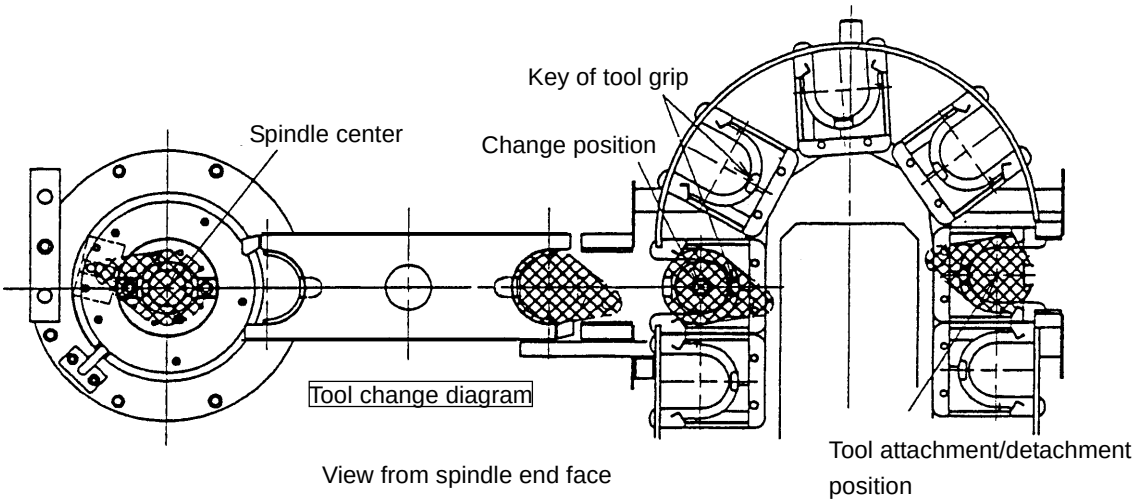
When attaching and detaching, give full attention to edges of tools and weight for safety operation.

Tool attachment/detachment



11-5 Relation Between Spindle Nose Key and ATC

When turning edges of tools which have directionality such a boring bar in the direction of key of the magazine grip, the edges, attached to the spindle by ATC action, turn to the left from the machine front.



11-6 Attention When Using Oil Hole Tool

- When attaching an oil hole tool, attach it so that the pilot plunger (positioning pin) may come in the direction of the key of the magazine clip.
- When using an oil hole tool, cutting chips attached to oil hole block side may cause troubles in insertion and positioning of pilot plunger of a tool into block. Regardless of manual or automatic operation, turn “ON” oil hole coolant, or blow down coolant tentatively by M50 command before attachment. After cleaning the block like this, change tools.

* Be sure to use an oil hole tool exclusively for HITACH SEIKI VS50/60.

11-7 ATC Program Operation

(1) Block operation by the program (automatic mode)

The ATC carries out the following block operations, depending on the program:

T∆∆	; Calls the tool No. ∆∆ from the tool magazine to an arm “standby position”.
M06	; Replace the tool in the spindle with that located at the arm standby position.
T∆∆M06	; Calls the tool No. ∆∆ from the magazine to the “spindle”. (ATC canned cycle)
T00	; Returns to the tool magazine the tool located at the arm standby position.

(2) ATC Single Block Stop

When single block is on, ATC action by M06 goes on by each step shown in M06 action (1) - (5) of ATC action explanation.

11-8 Condition Check During ATC Execution

1) • Tool No. on the spindle
   • Tool No. at the ATC arm standby position are displayed on the general screen.
2) • Tool No. commanded in a program is displayed on the commanded value screen.

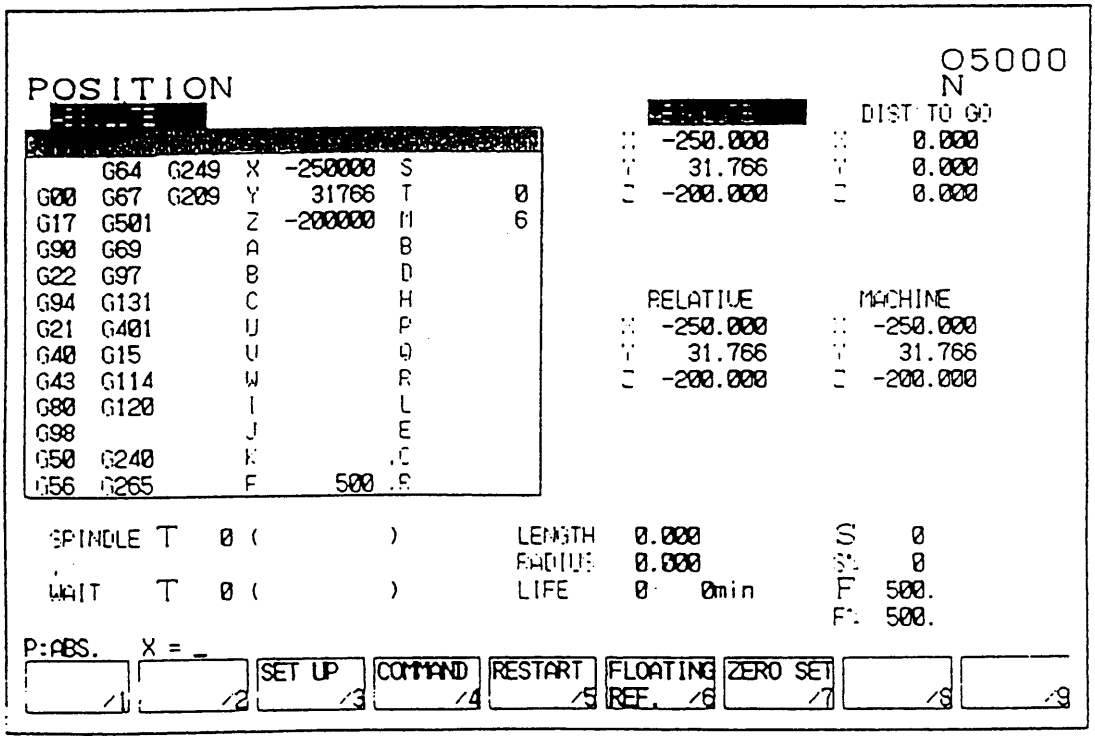
How to display each screen is as follows:

1) How to display the general screen
   When pressing the [RETURN] key, the general screen appears.

```
MAIN(AUTO)                                                    O5000
M06 T08 ;                                                     N
G00 X-250.0 Z-200.0 T15 ;
S1000 M03 ;
G04 X10.0 ;
G28 X0 Z0 M19 ;
M06 T15 ;
G00 X-250.0 Z-200.0 T17 ;
S1000 M03 ;
G04 X10.0 ;
G28 X0 Z0 M19 ;
M06 T17;

DATA>_

SPINDLE T     8 (FACE MILL )        ABSOLUTE            G01  G17  G90
LENGTH     0.000     S   1000       X     10.000        G40  G49  G80
RADIUS     0.000     S%  1200       Y     20.000        G98  G54
LIFE       0/   0min  F   3000       Z     30.000        M03  M08  M30
 (DWELL 6.070 sec)    F%  4920                           M55  M19  M
WAIT    T   15 (DRILL     )                              M    M    M

                                     DIST TO GO
      0%     50%     100% 120%       X      0.000   SET END   0:00:00
S  0                                 Y      0.000   CYCLE     4:25:07
X 13                                 Z      0.000   LAP T     5:36546
Y  8                                                 DATE   1996/01/18
Z 10                                                 TIME     18:48:15
```

2) How to display the commanded value screen

Display the general screen → Press the [F1/POSITION] key → Press the function key [F4/COMMAND VALUE] for commanded values → The commanded value screen appears.



12. Setting (Data)

12-1 Outline

Various setting data required to operate the machine can be set.

The setting (Data) screen is displayed by pressing the F5/SET key.

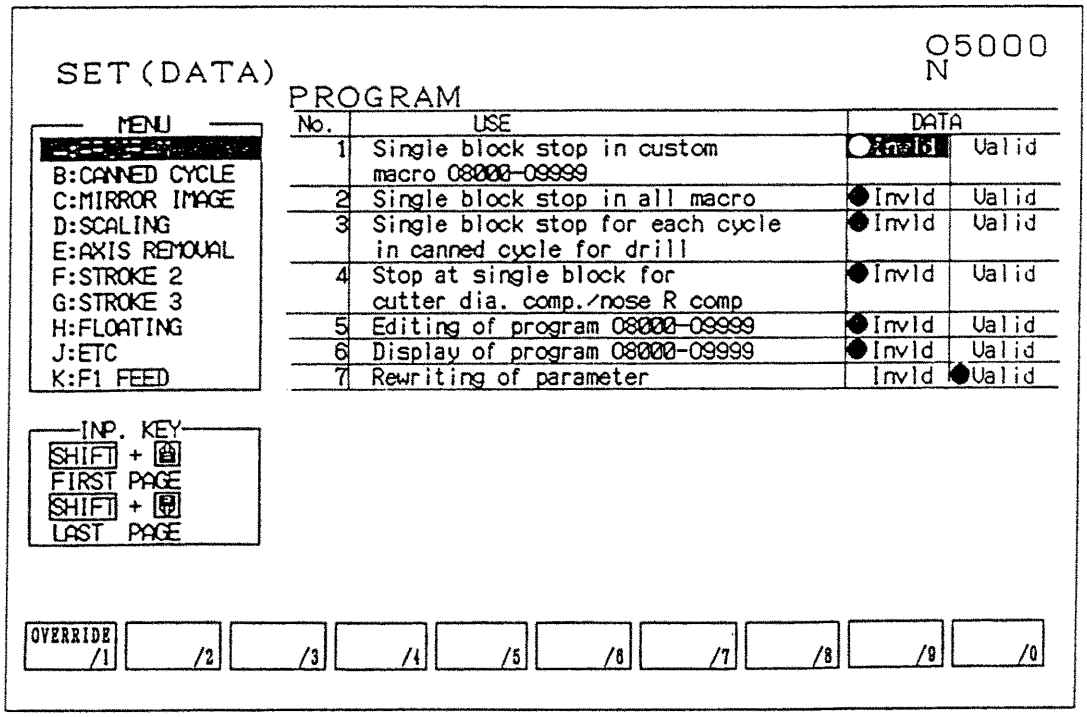
In this screen, parameters frequently used are classified together by item or purpose for easy setting.

An option is not displayed if it is not provided.

Since it consist of several pages, change it by pressing the page key or alphabetical key corresponding a menu at the left side of the screen.

The number search also available.

Setting (Data)



(1) Display and setting

In case of a selection such as “Do/Don’t”, ● marks are on at the side of setting has done if you want.

Set the cursor by the cursor key and press the INPUT key.

Further, when the cursor key is pressed, it moves at the side of ● mark. The HITACHI SEIKI’s standard is the left side.

Numeral data is set through the key input area. Needless to say, calculation and absolute/ increment value input are also available.

Caution

Input limit for writing key etc. is not applied on the setting screen is different from the parameter screen, change it with fully attention by an operator himself.

(2) Number search

Since all pages have serial number, a page or cursor can be moved at once by the number search. A search executes by pressing a cursor key ↑ ↓ → ← after inputting N and following numerals.

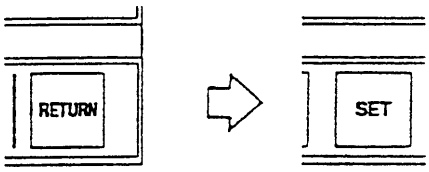
Example N: Number = 12 ↓ (Move to No. 12)

N: Number = 1 + 10 ↓ (Move to No. 11)

(3) Setting method

1. Set the key switch for writing of memory to “write side”.
2. Select the setting data screen.
   Press the RETURN key.

   Press the F5/SET key.

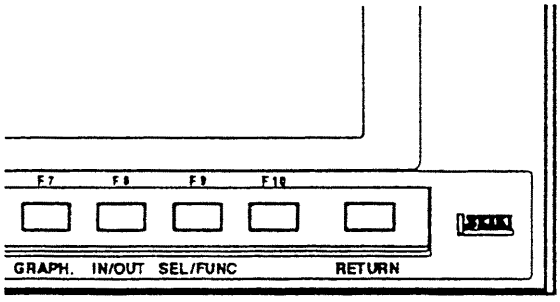


3. Set the cursor to the item to be changed by the page and cursor key or number search.

   Press the page key and display a section to be changed. ↑ ↓ or, N ▽▽ ↓ input the setting number.

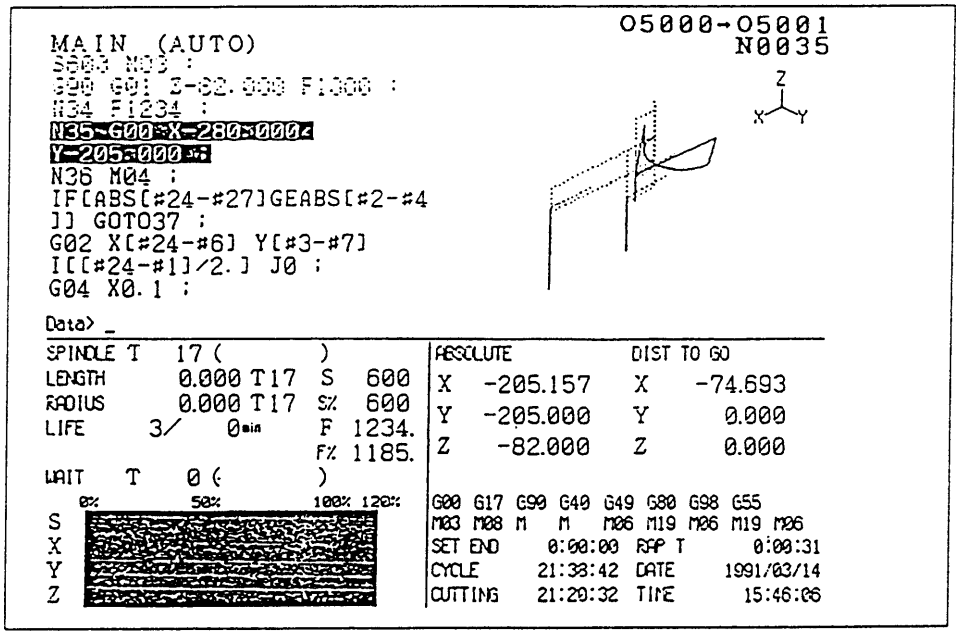
4. Key in a desired numeral and press the INPUT key.

5. press the RETURN key and display the initial screen after completion of setting.



13. Time Measuring, Data

Machining time and cutting time are displayed at the lower right of the general screen. It can be displayed by pressing the RETURN key.



13-1 Time Measuring

Four type of time is measured at the lower right of the screen. It can be measured in the limit of less than 10,000 hours.

Input in here execute by moving the cursor after pressing the F9/SEL/FUNC key

Example:	(Hour/Minute/Second) = 0	(Set to zero)
	(Hour/Minute/Second) = 1	(Input as 1:00:00)
	(Hour/Minute/Second) = 1/2	(Input as 1:02:00)
	(Hour/Minute/Second) = 1/2/3	(Input as 1:02:03)

(1) Prearrangement of completion

Advance notice of completion becomes effective when “SET END TIME” on the operation panel of the machine is “ON”.

A time of a prearrangement of completion is a time from start to the time of a advance notice of completion at the automatic operation.

If an operator inputs a time of a prearrangement of completion of a program, inform it by call light when a machining time reaches to the prearrangement of completion.

(2) Machining time

Accumulate a time during machining. (A time of the start lamp is being on.)

(3) Cutting time

Accumulate a time during cutting feed. (A time during cutting feed by G01, G02 or G03)

(4) Lap time

Measure a lap time between a tool and tool. At the time of tool change set it to zero automatically. Measurement does not execute while the NC is stand by condition.

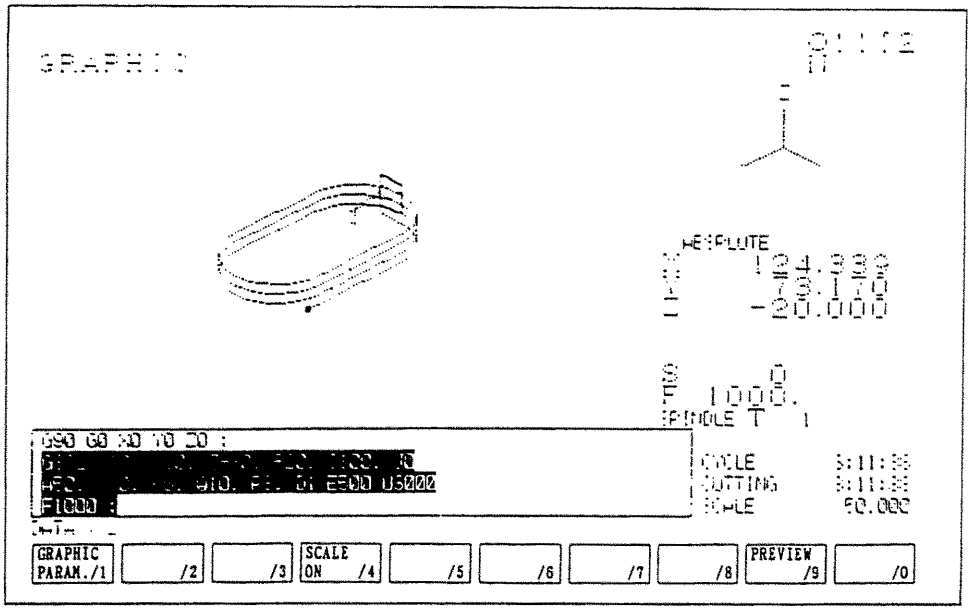
13-2 Date and Time

The date and time is displayed by the clock builted in.
The date shows by A.D. and time shows by 24 hours. (1 p.m. is 13 hours.)
Since it is backed up by a battery, it moves even if the power is turned off.
Do not change it unnecessarily, however, if discrepancy of the time has occurred from one cause or another, it should be set correctly because it uses at the time of periodical check.

14. Animated Drawing

14-1 Outline

When the F7/GRAPH key is pressed, the animated drawing screen is displayed.



Animated drawing of a locus of a tool is executed here. Also, the following data are displayed for the reference of animated drawing.

- Animated drawing plane of parameter
- Tool tip position
- Rotation S, Feed F, Tool command T
- Machining time, Cutting time
- List of programs currently under execution

A locus is drawn based on a value of the tool tip coordinate system. Since a moving point becomes unclear if drawing consist of plenty of lines, a tip is expressed by a small point.

Normally, synchronous drawing is displayed. For pre-processing drawing, press the F9/PREVIEW key and get the display of the list of function keys for pre-processing drawing. Then, press the F3/PREVIEW START key and pre-processing drawing will start.

Pressing the F2/AUTO RANGE key, pre-processing drawing will start and the drawing range is set in such a way that the locus of the cutting feed is properly shown within the frame of the screen when drawing finishes.

For canceling the display of a drawing, press the ORIGIN key.

Following setting can be done for drawing. Refer to the following item for details.

- Selection of drawing plane

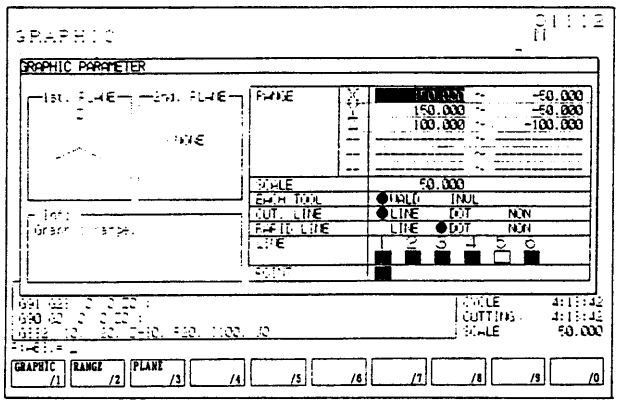
- Indication of rotating angle (Horizontal, Vertical)
- Drawing zone (Maximum, Minimum)
- Indication of drawing for each tool (Color designation)
- Selection of cutting feed line
- Selection of rapid traverse line
- Color designation for drawing dot lines.

14-2 Drawing Parameter

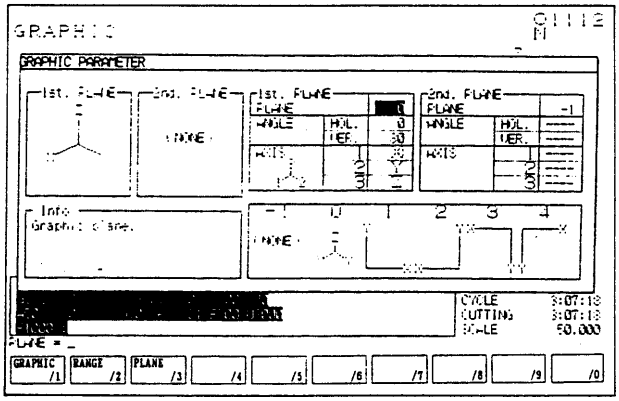
The drawing parameter screen is overlapped with the drawing screen when pushing the F1/GRAPHIC PARAM. of the drawing screen (F7/GRAPH.).

By pressing the F2/RANGE key, the display changes into the range setting screen.
By pressing the F3/PLANE key, the display of plane setting screen can be called up.

GRAPHIC parameter (Range setting)



GRAPHIC parameter (plane setting)



- Setting parameter for drawing: Move the cursor by cursor keys ↑ ↓ to the parameter column to be set. Brief explanation is displayed on the explanation column.

[Range setting]

(1) Drawing zone

Set the maximum or minimum value of each axis to draw.

Center coordinate (mean value of max. or min. value) and magnification (Max, and min. value are shown in the screen.) are decided.

Input a coordinate value (work coordinate) in the key input area and decide it by pressing the INPUT key.

In this case the maximum value should be set larger than the minimum value.

(2) Drawing for each tool

Change a drawing color at each time of tool change.

Applicable color are up to six and return to the beginning color beyond it.

Execute the setting by the color designation.

● marks are on at the side of setting has done. Change is decided by pressing the INPUT key after selection by cursor key ➔ 🡐.

(3) Cutting feed line

Designate the type of the cutting feed line.

● marks are on at the side of setting has done. Change is decided by pressing the INPUT key after selection by cursor key ➔ 🡐.

(4) Rapid traverse line

Designate the type of the rapid traverse line.

● marks are on at the side of setting has done. Change is decided by pressing the INPUT key after selection by cursor key ➔ 🡐.

(5) Color designation

Set a color to be changed when the drawing for each tool is effective.

Only this color designation change is available later.

Select a color designation by the cursor key 🡅 🡇 and designate a tool by the cursor key ➔ 🡐.

Decide it by pressing the INPUT key after inputting a numeral from 0 to 7 according to the instruction.

(6) Color of Drawing Point

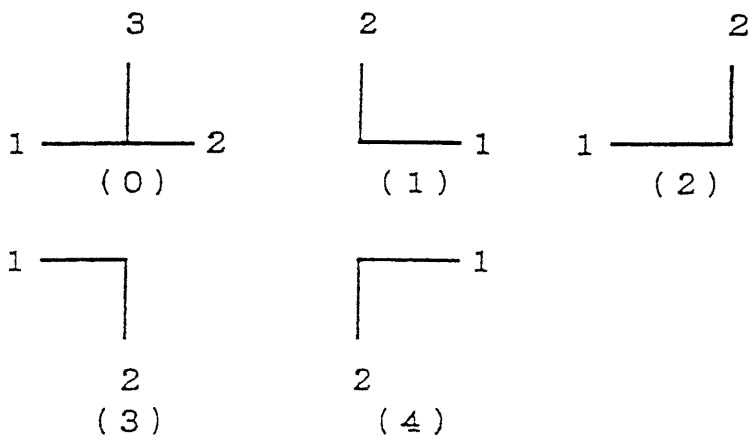
Designate color of the first point of drawing.

Select color of drawing point by cursor 🡅 🡇 for setting.

Select the figure between 0 and 7 according to the explanation, and press INPUT for deciding.

[Plane Setting]

(1) The drawing plane is designated. A plane is selectable from the following 5 options.



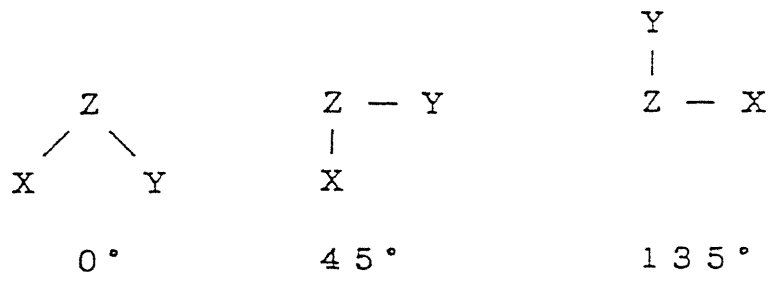
Enter numeral 0-4 (according to the above illustrations), then press the INPUT key.
The selected plane is displayed on the screen.
A numeral with minus sign is ineffective, which gives no drawing plane on the screen.

(2) Rotation Angle

This is effective for drawing plane “0” (3-axis expression) only.
Key in the value of angle in the appropriate area provided for entering, then press the INPUT key.

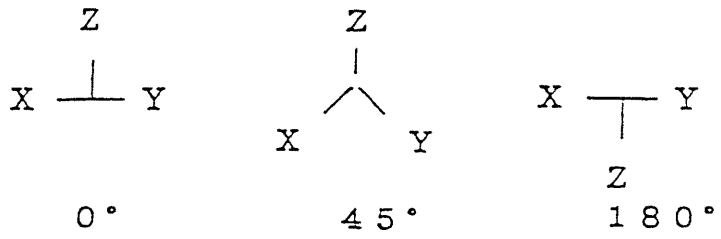
(a) Horizontal rotation angle

The rotation angle on the horizontal plane is designated within the range of ±180 degrees by step of 1 degree.
Example: The case of vertical rotation angle = 90° degrees in (XYZ).



(b) Vertical rotation angle

Angle is adjusted to the inclination of the vertical axis.
Example: The case of horizontal rotation angle = 0° degrees in (XYZ).



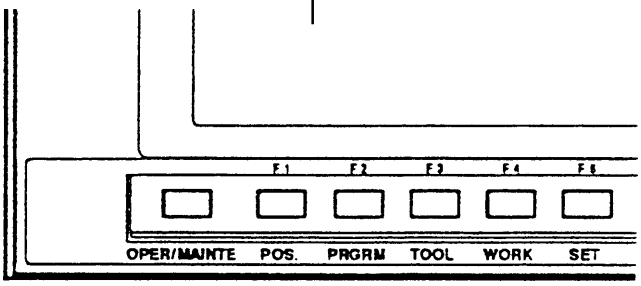
(3) Designation of Axis

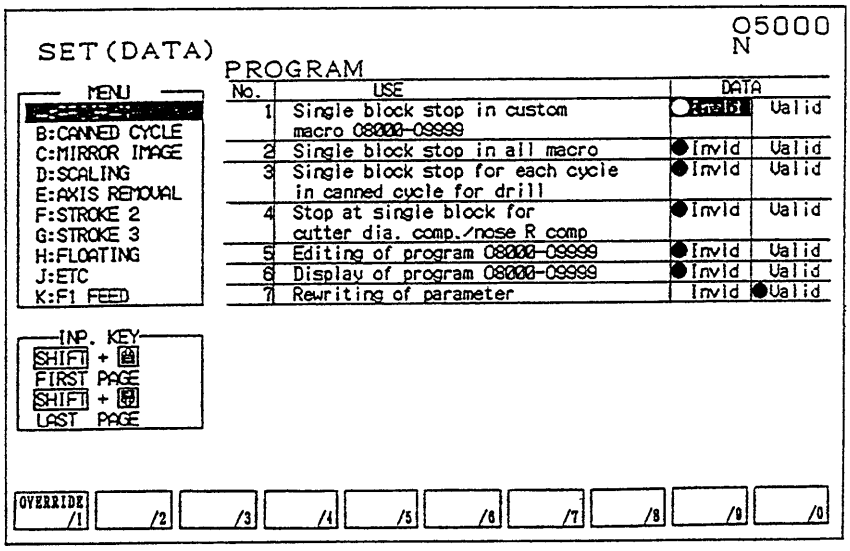
The axis used for the drawing is designated.
Enter the name of axis desired to be designated. Axis name with minus sign is allowed.

15. Parameter Setting

The parameter setting is executed by the following procedure.

1) Set the mode to the [MDI].
2) Set the memory key to the [Write].
3) Make effective the “Parameter rewriting” of the setting (data).
   [1] Press the F5/SET key.



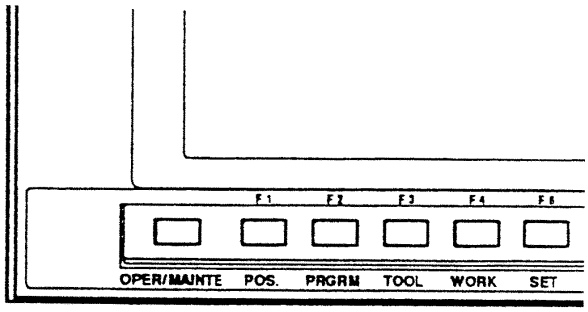


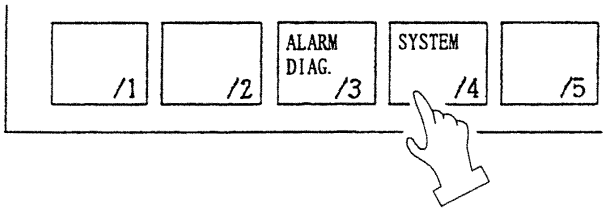
   [2] Set the cursor to “7. Rewriting of parameter” by the cursor key ↑ ↓.
   [3] Move the ● mark to the “Valid” by the cursor key ← → and press the INPUT key.

The parameter writing is ready.

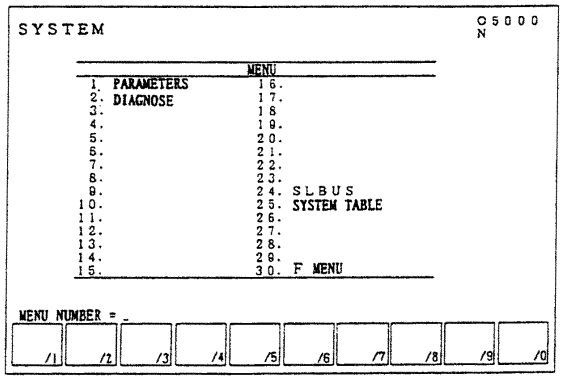
4) Press the RETURN key and make the initial screen.

5) Press the OPER/MAINTE key and then the F4/SYSTEM key.



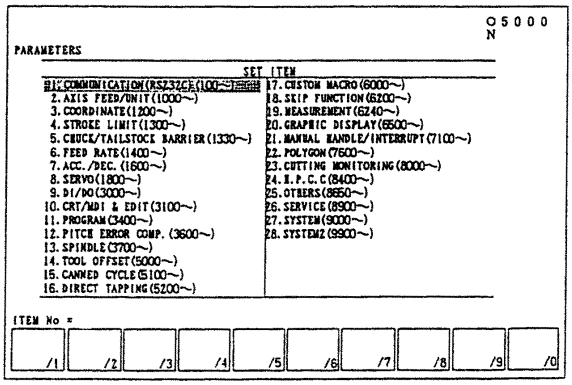


It becomes the menu screen.



6) Set the cursor to “1. PARAMETERS” by cursor key ↑ ↓ and press the INPUT key.

The screen changes to the parameter setting items.



7) Select an item to be changed by the cursor key ↑ ↓ and press the INPUT key.
The parameter list is displayed.
If the setting is “Selected”, it displays as the right side sketch for example.

PARAMETERS O5000 N0004

No.	7 6 5 4 3 2 1 0	HEX	No.	DATA
0100	0 0 1 0 0 0 0 0	20	0116	15
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

P-50

8) Set the cursor to the parameter No. to be changed by the page key ↑P ↓P or cursor key ↑ ↓ or direct call by N ΔΔ ↓ etc.
9) Change the parameter.
The parameter has two types as a data indicates by 0 or 1 and numerical data.
(a) A data indicated by 0 or 1 .
[1] Set the cursor to a bit to be changed by the cursor key →.
[2] Key in 0 or 1 and press the INPUT key at the condition of display as “P: ABS=” the lower left of the screen.
Move the cursor display for each bit to set.

[3] Return the cursor display to the "Number" column by the cursor key ← after completion of setting.
The parameter can be changed by direct input of hexadecimal number.

(b) A data indicated by numerals.

[1] Key in the new numerals and press the INPUT key at the condition of display as "P: ABS=" the lower left of the screen.

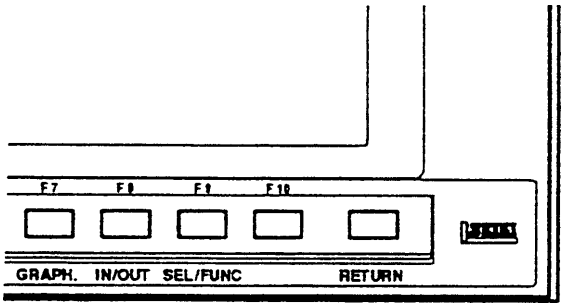
10) Procedure after setting

[1] Press the RETURN key and make the initial screen.

[2] Press the F5/SET and display the "Setting (Data)" screen.

[3] Make the "Rewriting of parameter", "Invid" by the cursor key ↓.

[4] Set the memory key to "Lock".



16. Operations Concerning W-Setter

16-1 Outline

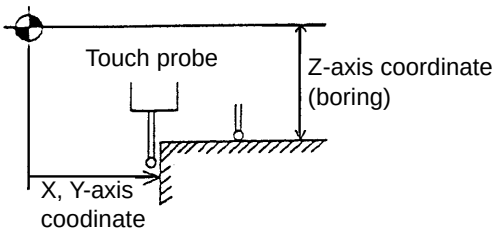
W-setter enables automatic setting of tool offset amount and work origin offset amount by a simple manual operation with a touch sensor mechanism.

- Work setter (Setting of work origin offset amount)
  - Reference surface
  - Reference hole
  - Coordinates modification
- Tool setter (Setting of tool offset amount)

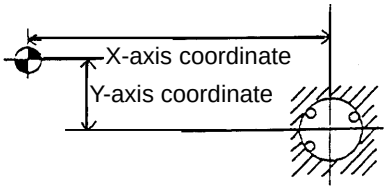
16-2 Input Signal and Various Operations

Work setter reference surface selection button ..... WSS
Work setter reference hole selection button ..... WSH
Work setter coordinates modification selection button ..... WSC
Tool setter selection button ..... OFH
Touch sensor signal ..... SKIP

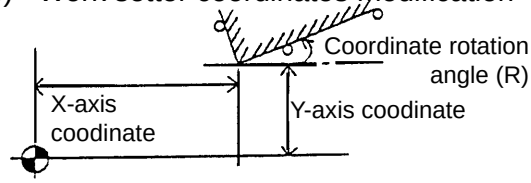
(1) Work setter reference surface



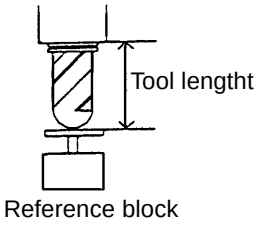
(2) Work setter reference hole



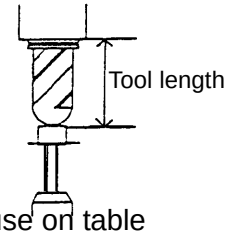
(3) Work setter coordinates modification



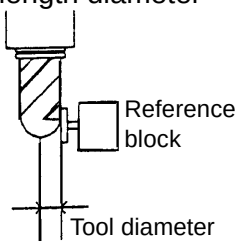
(4) Tool setter tool length



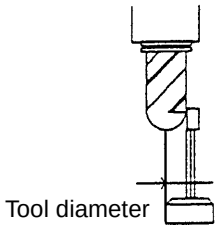
* Example of touch sensor use on table



(5) Tool setter tool length diameter



* Example of touch sensor use on table



16-3 Operation for Setting of Work Setter Reference Surface

(1) Execute reference point return of 3 standard axes manually. Attach the touch probe to the spindle.

(2) In the manual mode, have the work setter reference surface screen ON.
→ Work coordinate (compensation) screen (Fig. 1.8) is selected automatically.

(3) Select the work coordinate desired to be set by operating the page key and cursor keys.

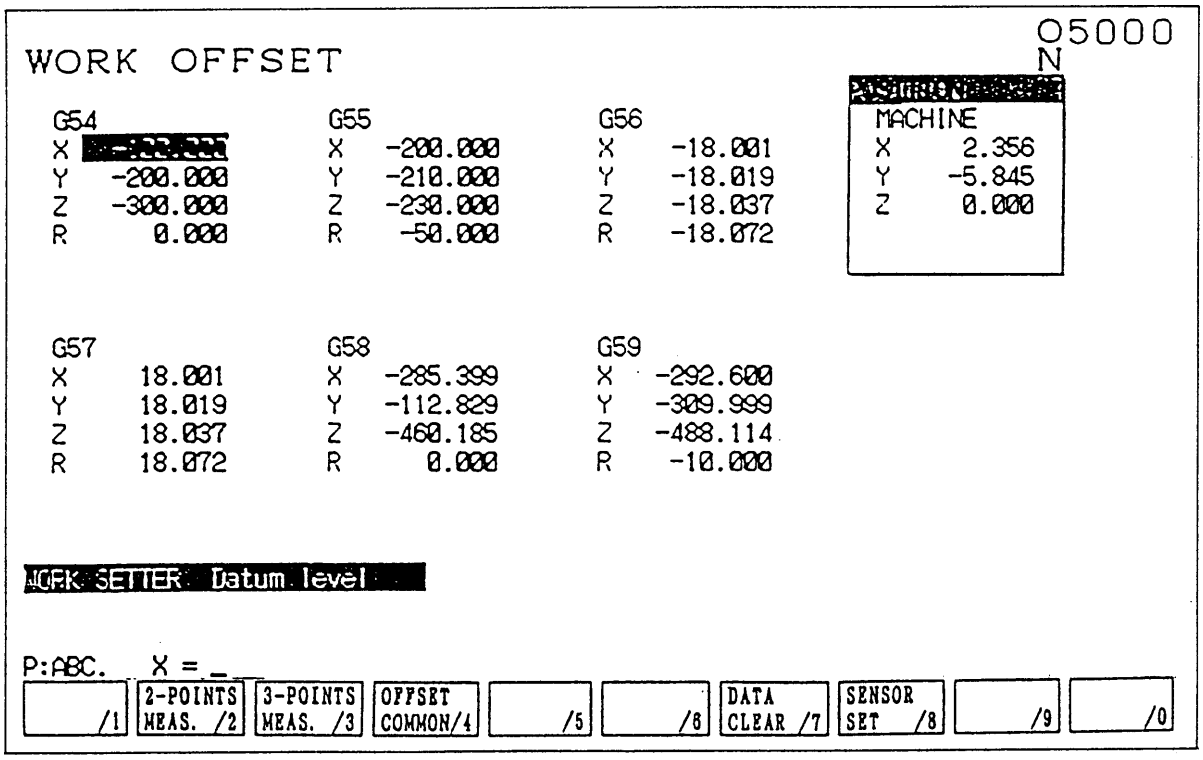
(4) Have the touch probe come into contact with the reference surface by the manual feed operation.

(5) The system automatically judges the axis of which the touch sensor signal comes on, calculates the work origin offset amount and the data is automatically written in.

(6) When the setting of all the necessary offset amounts for the work origin is completed, let the work setter reference surface screen OFF.

(Note 1) *The touch probe data used for calculating offset amount is to be set with parameter (No.6252~6255) in advance.*

Fig.1.8 Screen Display on Work Coordinate



16-4 Operation for Setting of Work Setter Reference Hole

(1) Execute reference point return of 3 standard axes manually. Attach the touch probe to the spindle.

(2) In the manual mode, have the work setter reference hole screen ON.
→Work coordinate (compensation) screen (Fig. 1.8) is selected automatically.

(3) Select the work coordinate desired to be set by operating the page key and cursor keys.

(4) Have the touch probe come into contact with an arbitrary point of the reference hole by the manual feed operation.
→The system memorizes the position of the main/auxiliary axis where the touch sensor signal became ON, and displays a commentary message to be ready for the measurement of the second point.

(5) Have the touch probe come into contact with the second measuring point of the reference hole.
→The system memorizes the position of the second point and displays a commentary message to be ready for the measurement of the third point.

(6) Have the touch probe come into contact with the third measuring point of the reference hole.
→The system memorizes the position of the third point and calculates the center of the three points and determines the work origin with which the work origin offset amount is written in automatically.
→Now ready for the measurement of the first point of another reference hole.

(7) When the setting of all the necessary offset amounts for the work origin is completed, let the work setter reference hole screen OFF.

* After the procedure (5) in the above, when the F2/2-POINTS MEAS. key is pressed, the system calculates the midpoint of the two points and the work origin offset amount thus calculated is written in automatically. In this case, the groove width is displayed on the right of the commentary message.
* When measurement in the Z-axis direction is made, the system calculates the third axis work origin and the offset amount, which is written into the system automatically.

(Note 1) *When the system is reset in the midst of the setting operation (during 2nd or 3rd point is being measured), the system returns to the initial status (the status to be ready for the 1st point measurement).*

(Note 2) *The mutual distances between the 3 points is short against the parameter setting (No.6257), the measurement cannot be made and a commentary message is displayed accordingly. By resetting, the status is released and the system is initialized.*

(Note 3) *When you only wish to know hole diameter or boss diameter without rewriting the work origin offset amount, press the F3/3-POINTS MEAS. key after the procedure (5) in the above then have the probe contacted to the third point.*

By this key operation, the hole diameter or boss diameter is displayed on the right of the commentary message.

(Note 4) *The touch probe data used for calculating work origin and offset amount is to be set with parameter (No.6252~6255) in advance.*

16-5 Operation for Setting of Work Setter Coordinate Modification

(1) Execute reference point return of 3 standard axes manually. Attach the touch probe to the spindle.

(2) In the manual mode, have the work setter coordinate modification screen ON.
→Work coordinate (compensate) screen (Fig. 3.1) is selected automatically.

(3) Select the work coordinate desired to be set by operating the page key and cursor keys.

(4) Have the touch probe come into contact with an arbitrary point of the reference surface by the manual feed operation.
→The system memorizes the position of the X, Y-axis positions where the touch sensor signal became ON, and displays a commentary message to be ready for the measurement of the second point.

(5) Have the touch probe come into contact with the second measuring point of the reference surface.
→The system memorizes the position of the second point and displays a commentary message to be ready for the measurement of the third point.

(6) Have the touch probe come into contact with the third measuring point on the surface intersecting rectangularity.
→The system memorizes the position of the third point, based on which it calculates the position of the corner and the coordinate rotation angle thereby the work origin offset amount is written in automatically.
→Now ready for the measurement of the first point of another reference surface.

(7) When the setting of all the necessary offset amounts for the work origin is completed, let the work setter coordinate modification screen OFF.

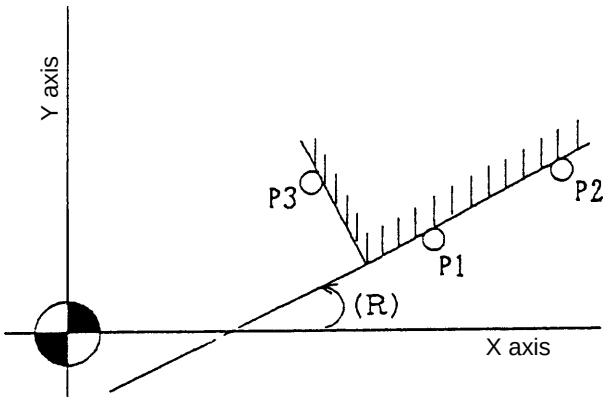
* After the procedure (5) in the above, when the F2/2-POINTS MEAS. key is pressed, the system calculates the coordinate rotation angle only and the work origin offset amount is written in automatically.

* When measurement in the Z-axis direction is made, the system calculates the Z-axis work origin offset amount, which is written into the system automatically.

(Note 1) *When the system is reset in the midst of the setting operation (during 2nd or 3rd point is being measured), the system returns to the initial status (the status to be ready for the 1st point measurement).*

(Note 2) *The mutual distances between the 3 points is short against the parameter setting (No.6257), the measurement cannot be made and a commentary message is displayed accordingly. By resetting, the status is released and the system is initialized.*

(Note 3) *Command G code of axis rotation in the program to operate by rotating the coordinate.*



Determine a corner position at the point where a line through the points P1 and P2, and a vertical line through P3 cross.
This axis rotation angle (R) is that between a line through P1 and P2, and X-axis.

16-6 Manual Overwriting of Work Zero Point Offset Amount.

(1) Press the function key F4/WORK COORDINATE for selecting the work coordinate (modify) screen (Fig. 1.8).

(2) Select the work zero point offset amount to be set by the page key and the cursor key.

(3) At first, press [P] (input the given amount as work zero point offset amount) or [**I**] (input the given amount added as work zero point offset amount). Next, select an axis to be inputted by the cursor key, or press axis name to be inputted, and press [Numerical value] INPUT which overwrites the specified work zero point offset amount.

16-7 Setting Operation of Tool Setter

(1) Return the basic 3 axes to manual reference point. Install the standard block on the table or the touch censor on the table.

(2) Turn on the tool setter by manual mode.

(3) Set the distance from the zero point to the standard block to the parameters (No. 6249~6251). When this value is fixed, the operation in this item is unnecessary. When position of the standard gauge is not fixed, following operation to set a correct value is necessary.
By pressing F2/REF. GAUGE parameters (No.6249~6259) are displayed. When the touch probe is touched to the standard block, by the same calculation as the work setter

(the standard surface), the distance between zero point and the standard block is set to the parameters (No.6249~6251). Also, by measuring from Z axis direction, safety guard parameters (No.6258~6259) are set at the same time. Be sure to return to the previous screen by pressing F2/REF. GAUGE .

(4) Select an offset number to be set by the page key and the cursor key.

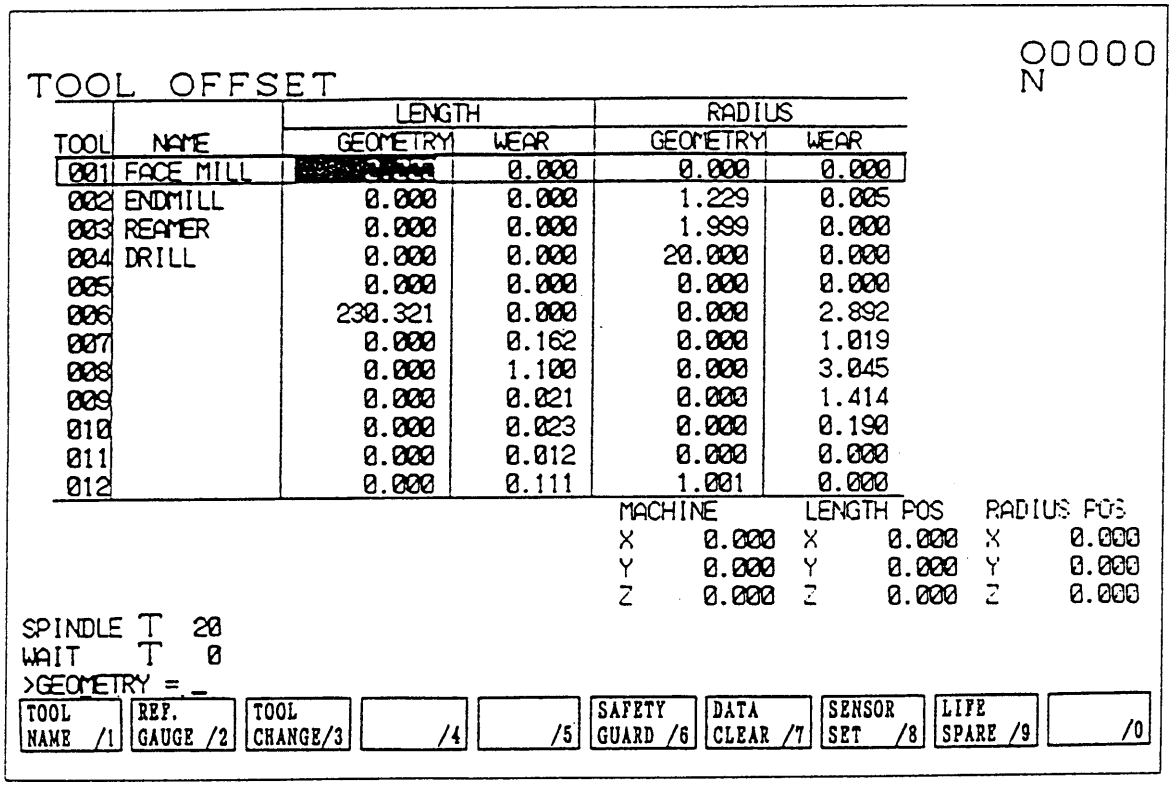
(5) Have the tool come into contact with the reference gauge.
→NC automatically judges the axis of which the touch sensor signal comes on, and the tool offset amount is automatically written in.

X, Y-axis ..... Tool diameter is calculated and the tool shape compensation amount for diameter compensation is rewritten.

Z-axis ..... Tool length is calculated and the tool shape compensation amount for length compensation is rewritten.

(6) When the setting of all the necessary offset amounts for the tool is completed, let the tool setter screen OFF.

Fig. 1.8 Tool (Compensation)

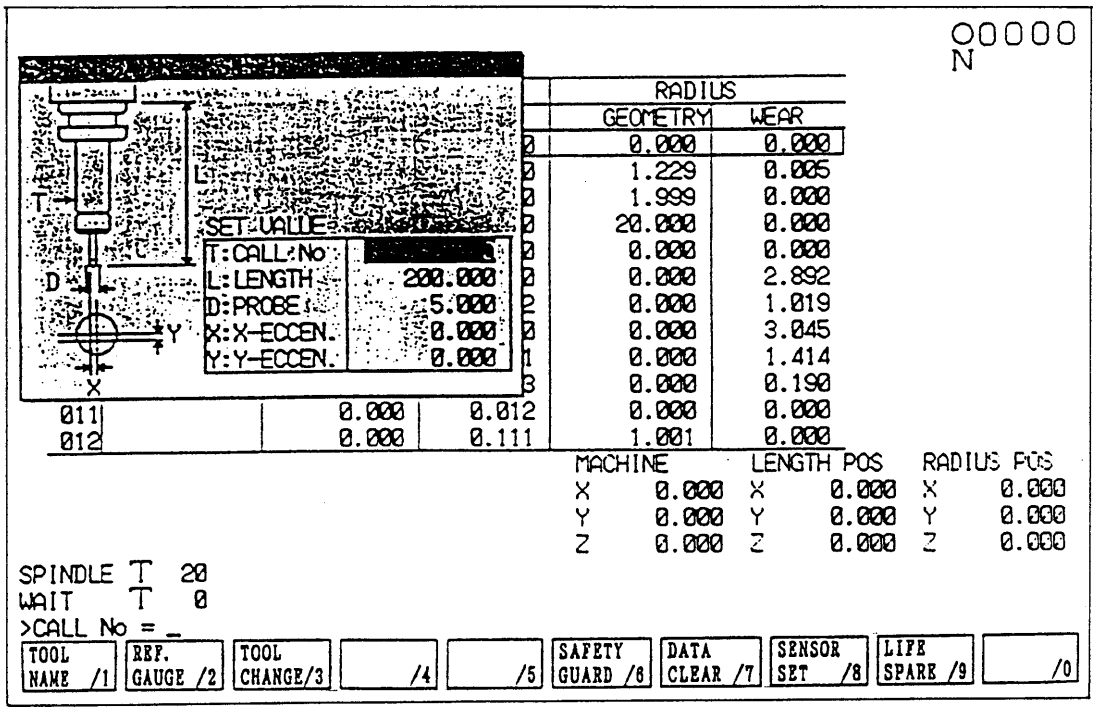


(Note 1) *When writing in a tool diameter, if measurements are repeated several times without changing the designated offset No., the offset amount is rewritten only when the result of a later measurement gives a larger value than that of previous one. If the designation of the offset No. is changed or tool setter button is pressed anew, however, the first measurement result is written in as the offset amount.*

(Note 2) *The reference gauge means the contact surface of a reference block. When using a touch sensor on a table other than a reference block, the reference gauge means its contact surface.*

(Note 3) *Data of the touch probe used for measurement is set in parameter (No.6252~6255) in advance. There are two ways of the setting, one is to write in as parameter and the other is to write in on the tool (compensation) screen. For the latter method, press* F8/SENSOR SET *and get the multi-window setting screen (Fig.1.9) on which data setting can be arranged.*

Fig. 1.9 Sensor Setting



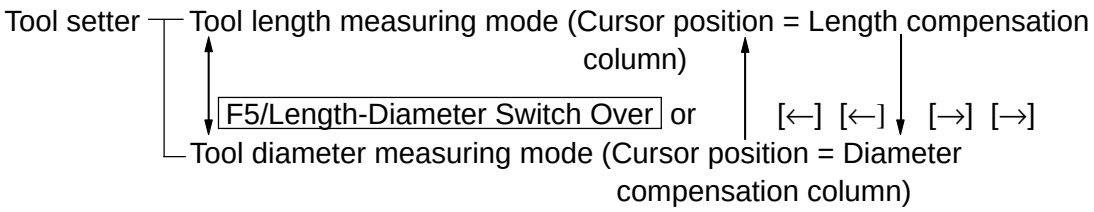
Instead of using a touch probe, it is also possible to measure the distance to the reference gauge by using a tool of known size (called as a reference tool). In this case, set the parameter (No.6242, 5-bit) as “1” and set the size of the reference tool in advance with parameter (No.6260~6263).

16-8 Tool Length Measuring Mode, Tool Diameter Measuring Mode

By setting “1” with parameter No.6242 4-bit, the tool length measuring mode and the tool diameter measuring mode.

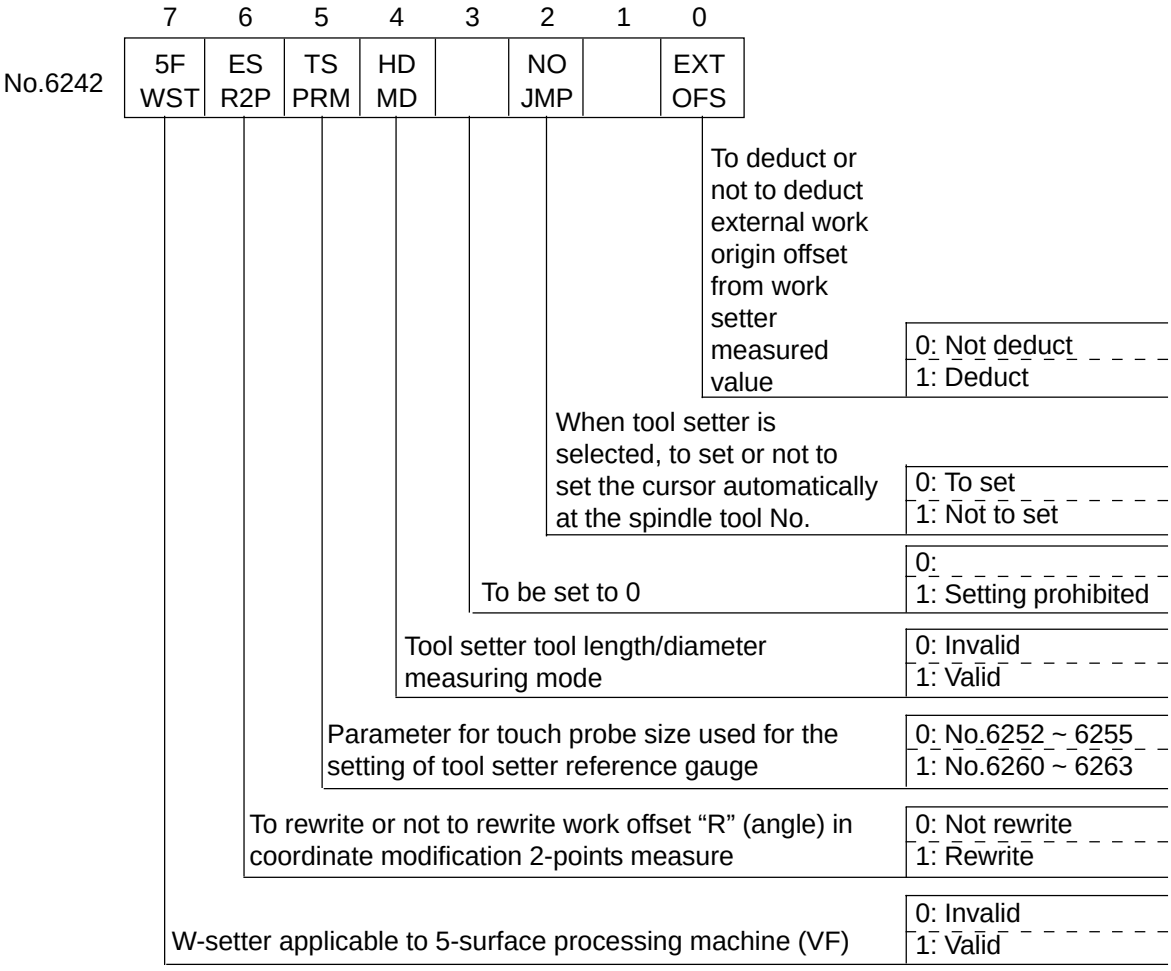
In these modes:
- If tool diameter is measured in the tool length measuring mode, alarm (No.716) is issued.
- If tool length is measured in the tool diameter measuring mode, alarm (No.716) is issued.
- If measurement is made on screen other than tool (compensation) scene, alarm (No.716) is issued.

This system enables to prevent offset setting error arising from any wrong probe contact.



The tool length/diameter measuring mode is effective during the reference gauge setting operation. In the tool length/diameter measuring mode, setting is made for X, Y positions/Z position respectively.

16-9 Parameter



6249 Distance from reference point to the X-axis reference gauge

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

6251 Distance from reference point to the Z-axis reference gauge

6252 Diameter of probe ball

6253 Eccentricity from the spindle axis center to the probe ball center in the main axis direction

6254 Eccentricity from the spindle axis center to the probe ball center in the auxiliary axis direction

6255 Sensor length measured from the gauge line
6256 Retracting amount at time of re-touching
6257 Minimum value of the distance for hole and corner 3-point measurement
6258 Relief guard measurement position (X)
6259 Relief guard measurement position (Y)
6260 Reference tool diameter
6261 Eccentricity from the spindle axis center to the reference tool center in the main axis direction
6262 Eccentricity from the spindle axis center to the reference tool center in the auxiliary axis direction
6263 Reference tool length measured from the gauge line

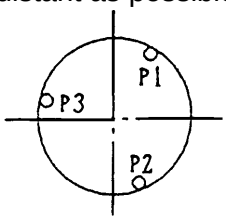
16-10 Others

Caution

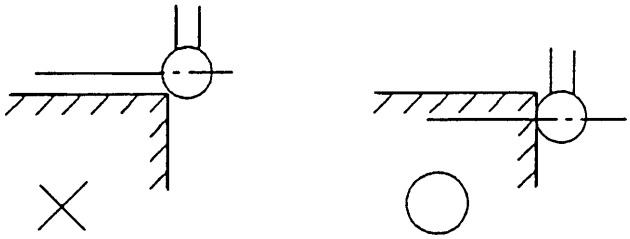
(1) Work setter and tool setter are effective in the manual mode.
(2) While the work setter or the tool setter is in operation, when a touch sensor signal comes on, further axis movement becomes possible only in the opposite direction, any other axis movement being prohibited.
(3) When a touch sensor signal comes on, buzzer begins to sound and it continues until the axis is moved out to the opposite direction by a distance equivalent to the value of the parameter setting (No.6265). While buzzer is sounding, rewriting of work origin offset amount and tool offset amount does not take place even when a touch sensor signal comes on.
(4) The automatic rewriting of the work origin offset amount or the tool offset amount in the work setter and tool setter operations functions on the offset amount on which the cursor is positioned at the time of the touch sensor signal coming on. Even when screen then displays other than work coordinate (compensation) or tool (compensation) scene, the system keeps the last location of the cursor in memory.
(5) When measuring with the handle, if the following methods are applied to make the censor touch, they may not keep the axis free (the axis doesn't move.) Pay attention.
    [1] Turn the handle intermittently by few scales for touching.
    [2] To bring the censor near or to keep it away (to turn the handle in + and - direction alternately) for touching.
    [3] To touch with extremely low speed.
    When these phenomena appear, press the reset button to free the axis, and measure again.

Caution

(6) When making a hole measurement, three measuring points for probe touch should be selected as mutually distant as possible.



(7) When making measurement, the center of the probe ball correctly positioned against the measuring face.



(8) When the thermal displacement compensation function is used together with the work setter function, attention should be paid to the following points.
Parameter No.6242 0-bit should be set as “1”.
Before making measurement, direct the command of ATC (M06Txx) or M code for effecting the thermal displacement compensation data.

17. Safety Guard

17-1 Safety Guard (Tool Length)

This function measures the tool length of the tool used for the machining program in the AUTO mode.

(1) Operation method

[1] Perform zero point return.

[2] Attach a touch probe to the spindle and a reference block to the table.

[3] Measure the reference block position. (Omissible if having been already measured.) For measuring method, see “Setting Operation of Tool Setter” of “Operation Concerning W Setter” in the operation version or (2) Reference Gauge.

[4] Call the machining program in the AUTO mode.

[5] Turn ON the TOOL LENGTH button.

[6] Press PROGRAM START.

[7] Execute the machining program’s T code, M06.

[8] Tool length measurement is carried out with H command.

(Note 1) *Tool length measurement is carried out when offset No. has been changed.*

(Note 2) *Measurement is carried out in M06 when tool offset by tool No. is effective.*

(Note 3) *As for measuring operation, see (4) Description of Measurement Operation.*

(Note 4) *Commands except for M00, M01, M02, M06, M30, M98, M99 and T are not available.*

(Note 5) *Neither M00 nor M01 operate when Parameter No. 6243, #3 = 1.*

[9] Data of the safety guard page are cleared with M02 or M30. For the safety guard page, push F3/TOOL → F6/SAFETY GUARD .

[10] Set TOOL LENGTH button to OFF.

(2) Reference gauge

[1] After setting [Handle] or [Feed] mode, push ON TOOL SETTER button.

[2] Press F3/TOOL (TOOL), F2/REF. GAUGE in that order. → The Reference Gauge Setting screen appears.

[3] Put the touch probe into contact with the Z directional contact surface of the reference block. → The buzzer sounds and the parameters No. 6251, 6258 and 6259 are rewritten.

[4] Relieve the Z axis until the buzzer sound goes off.

[5] Press F2/REF. GAUGE in that order.

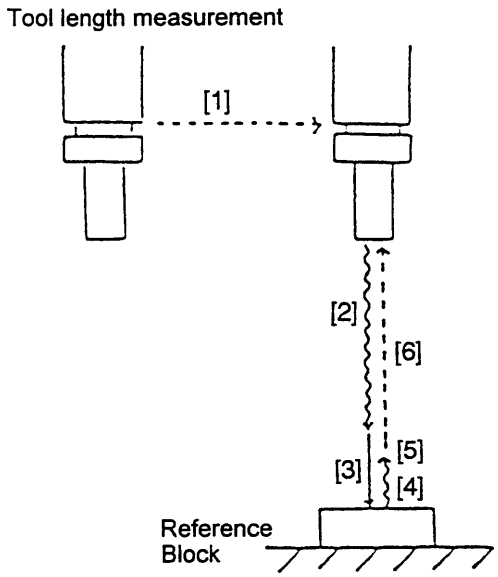
[6] Set TOOL SETTER button to OFF.

(3) Sample program

```
O0001;
N1  G54  G90  G00  X0  Y0;
N2  G30  G91  X0  Y0  Z0;
N3  T01  M06;                    -----Executes T01 and M06.
N4  G00  G90  X100.  Y100.
N5  G43  Z-100.  H1;             -----Measures the H1 tool length.
N6  M98  P2;
N7  T02;                         -----Executes T02.
N8  G30  G91  X0  Y0  Z0;
N9  M06;
N10  G00  G90  X200.  Y200.;
N11  G43  Z-100.  H2;            -----Measures the H2 tool length.
N12  M98  P2;
N13  M30;                        -----Clears the data on the screen.
%

O0002;
N100  G01  G91  X10.  F200;
N101  G02  J-10.;
N102  G01  Z50.;
N103  M99.
%
```

* The tool length has been measured is not measured until M30 or M02 is executed. For example, if the program is restarted from the beginning after it was executed as far as N5, tool length measurement is performed from the next block, invalidating H1 in the N5 block.

(4) Description of Measuring Operation



[1] The X and Y axis move to the reference block position at rapid traverse rate.

[2] Bring the Z axis close to the reference block by the [HANDLE] or [JOG] feed. (Operate just in the automatic mode.)

[3] Apply the Z axis to the reference block by the [HANDLE] or [JOG] feed.
→ The buzzer sounds and the offset amount of the H code is rewritten.

[4] Retract from the block by the [HANDLE] or [JOG] feed until the buzzer goes off.

[5] Press the [PROGRAM START] switch.

[6] The Z axis moves to the ATC position at rapid traverse rate and the program proceeds to the next block.

* If the [PROGRAM START] switch is pressed after completing the movement, [1], the program proceeds to the next block, cancelling the movement [2] on.

(5) ATC Operation

Ordinary ATC operations, as mentioned below, vary by machine type, which operate in the same way as ATC canned cycle.

ATC (M06)

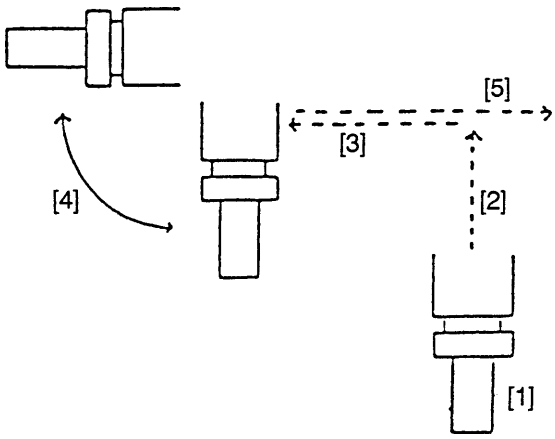
[1] The coolant stops and the spindle stops.

[2] The Z axis moves to the ATC position at rapid traverse rate.

[3] Spindle orientation is performed, and the X and Y axes move to the ATC position at rapid traverse rate.

[4] ATC

[5] The X and Y axes move to the reference block position at rapid traverse rate.



(Note) *As with the ATC canned cycle, set the parameters in advance.*

(6) Parameters

Refer to 17-2 Safety Guard (Comparison).

(7) Alarms

No. 213 Safety guard tool length operation error

[#001] TOOL LENGTH button has been pushed except in reset state.

[#002] After tool length measurement is started, prior to resetting (M02, M30, Reset key, %) being applied, TOOL LENGTH button is set OFF.

[#003] Measurement has taken place for 2 or more axes simultaneously.

[#004] On start of tool length measurement for 5-sided machine, a horizontal shaft head is fitted. Or, a horizontal shaft head is fitted while in measurement.

[#005] Either the manual or the handle mode switch has been pressed.

No. 715 Measurement error Used in machine-lock state.

(8) Cautions

[1] Operate tool length measurement with the SINGLE BLOCK switch turned OFF.

[2] When using tool offset or multiple corrections using a tool number, the tool length is measured per correction number.

[3] Perform operation with machine-lock OFF state.

[4] Perform tool length measurement as holding Auto mode.

[5] As ATC canned cycle is operated in M06, set the parameter for ATC canned cycle beforehand.

17-2 Safety Guard (Comparison)

This function executes the machining program in the AUTO mode with the X and Y axes moving and the Z axis machine-locked, measures the workpiece profile (Z-axis direction) in an optional Z-axis positioning block, and checks for an interference with workpiece.

(1) Operation method

[1] Perform zero point returning.

[2] Mount the spindle touch sensor.

[3] Set touch sensor scope. Setting, if having been already done, can be omitted.
As for setting procedures, see “Reference Gauge” of “Safety Guard (Tool Length).”

[4] Call a machining program in AUTO mode.

[5] Push ON COLLATION button.

[6] Push PROGRAM START.

* G00 Zxx;
  Measures the workpiece profile when executing the block containing a Z-axis positioning command, and at the same time, checks for an interference with the workpiece.
* Block other than G00 Zxx;
  Executed with Z axis machine locked, M06 invalidated and dry run.

(Note) *No verification is carried out with G00 Zxx; command which is given while in Special Canned Cycle or in Canned Cycle mode or together with canned cycle cancel command block.*
(EX) G80 G00 Zxx;... No verification carried out.

[7] With M02 or M30, data of the safety guard page are cleared.
For the safety guard page, push F3/TOOL → F6/SAFETY GUARD .

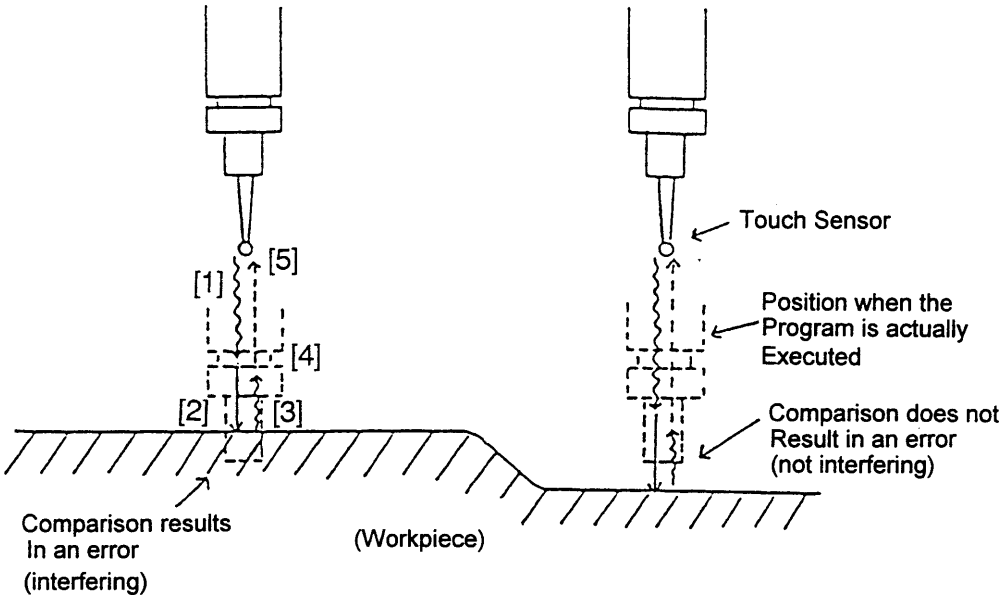
[8] Set COLLATION button to OFF.

[9] Perform zero point returning of Z axis.

(2) Sample program

```
O0001;
N1  G54  G90  G00  X0  Y0;
N2  G30  G91  X0  Y0  Z0;
N3  T01  M06;
N4  G00  G90  X100.  Y100.;
N5  G43  Z-100.  H1;              ----Comparison
N6  M98  P2;
N7  T02;
N8  G30  G91  X0  Y0  Z0;
N9  M06;
N10  G00  G90  X200.  Y200.;
N11  G43  Z-100.  H2;             -----Comparison
N12  M98  P2;
N13  M30;                         -----Clears the data on the screen.
%

O0002;
N100  G01  G91  X10.  F200;
N101  G02  J-10.;
N102  G01  Z50.;
N103  M99;
%
```

(3) Comparison movements



[1] Bring the Z axis close to workpiece by the [HANDLE] or [JOG] feed.

[2] Apply the Z axis to this workpiece by the [HANDLE] or [JOG] feed.
→The buzzer sounds and the work-piece profile is measured.

[3] Retreat from the block by the [HANDLE] or [JOG] feed until the buzzer goes off.

[4] Press the [PROGRAM START] switch. (execution of comparison)
→ A warning message if the position when the program is actually executed interferes with the workpiece.

[5] The Z axis moves to the ATC (comparison start) position at rapid traverse rate and the program proceeds to the next block.

* If the [PROGRAM START] switch is pressed after completing the movement [1], the program proceeds to the next block, canceling the movement [2] on.
* If the [PROGRAM START] switch is pressed when a warning No. 215 (the tool entered the disabled areas in safety guard comparison) occurred, operation can be continued, canceling the warning.

(4) Parameters

No. 6243,	#0 = 0	Compares only first G00 Z_ coming after a (T_) command.
	1	Compares all G00 Z_.
	#1 = 0	The measurement position Z of comparison is the difference between a command value and a measured value.
	1	The measurement position Z of comparison is the work coordinates of the touch position.
	#2 = 0	Does not position to the 2nd reference point at comparison start time.
		Positions to the 2nd reference point at comparison start time.

	#3 = 0	M00, M01 is executed on tool length execution.
	1	M00, M01 is ignored on tool length execution.
No. 6251		Distance from the reference point to the reference gauge Z end face.
No. 6255		Distance from the spindle nose to the -Z directional end face of the probe sphere.
No. 6256		Retouch return amount
No. 6258		Reference gauge position (X axis)
No. 6259		Reference gauge position (Z axis)

(5) Alarms

No. 214	Safety guard verification operation error
[#001]	COLLATION button has been pushed ON except in reset state.
[#002]	After verification is started, prior to resetting (M02, M30, Reset key, %) being applied, COLLATION button has been set OFF.
[#003]	Measurement has been performed for 2 or more axes simultaneously.
[#004]	On start of verification measurement for 5-sided machine, a horizontal shaft head is fitted. Or, in measurement, a horizontal shaft head is fitted.
[#005]	Manual or Handle mode switch has been pushed.
No. 215	Safety guard verification error ..... Tool has entered disabled area in the safety guard verification.
No. 715	Measurement error ..... Used in machine-locked state.

(6) Cautions

[1] Operate safety guard comparison with the SINGLE BLOCK switch turned off.

[2] Be sure to perform zero point return after operating safety guard comparison.

[3] Perform operation with machine-lock OFF.

[4] Perform verification as holding Auto mode.

18. Tool Life Management

18-1 Tool Life Management

1. Feature

The tool life management function counts the tool life use values of the tools used in the machining program according to the life units for each tool.

It compares a counted tool use value with a tool life set value to control a tool life. When you do not want tool life management in the measuring program, control this function by a G-code. When the power is turned on or the NC unit is reset, the G131 mode is selected.

G130 ----- Tool life management disabled

G131 ----- Tool life management enabled

2. Functioning

Tool life management carries out the following actions sequentially.

- Selecting a tool for the spindle (when a Txx command in the program is executed)
- Checking the spindle tool status (when M06 in the program is completed)
- Counting the tool life (during machining by the program)

[1] Selecting the tool for the spindle

When a Txx command is executed in the machining program, the tool life management function selects a tool to be called to the spindle. If the Txx command is a spare tool command, a defectless tool will be automatically selected out of a spare tool group.

* If the selected tool is an unused one, a new tool signal is output to the PC. (During TF)

[2] Checking the spindle tool status

The tool life management function checks the tool status of the tool called to the spindle by the M06 command in the machining program.

- When the tool is unused → Designates the tool status “USED” or “MONITORED.”
- When the tool is out of life → Displays a warning, “781 OUT-OF-LIFE TOOL WAS SELECTED.” In the M56 mode, however, the machine stops due to feedhold.
- When the tool is broken → Displays a warning, “780 BROKEN TOOL WAS SELECTED.” The machine stops due to feedhold.

[3] Counting the tool life

When the following requirements are met, the tool life management function counts the tool life use value of the spindle tool according to the tool life units preset for each tool.

- Under automatic operation in the MEMORY mode

- Dry run, Z-axis cancel, and machine lock are not turned on
- In the M53 mode (M54 monitoring function is not stopping)
- In the G131 mode

When a counted use value reaches a preset life value, the tool status for that tool is designated OUT OF LIFE, but machining continues as it is. When the life set value is zero, only counting of the life use value is performed and out-of-life judgment is not made.

* When the tool status is designated OUT OF LIFE and that tool or all the tools in the same tool group are defective, a tool change signals output to the PC.

3. Life Units

As tool life units, set time, use times, length, and number of holes for each tool. The tool life is counted as follows according to the life units.

- Time ..... Counts cutting feed time (minutes) during spindle forward rotation. When the parameter No. 8002 #0 is “1”, it is counted even during spindle reverse rotation.
- Use times ..... Counts how many times the tool has been called to the spindle (at the time of M06).
- Length..... Counts a cutting distance (meters) during spindle forward rotation.
- No. of holes ..... Counts the number of holes machined in a canned cycle for tapping or drilling. It is counted earlier than actual machining because it is counted by buffering.

(Note) *The data below a minute and meter (seconds and millimeters) are intern ally counted and memorized.*

```
TxxM06;                    ---------- If the life units are use times, use times is counted
                                      here.
S1000M03;
G54G90G00X0Y0;
G43Z50.0H12;
G01Z38.F500;   )
X50.F280;      )
Y10.;          )           ---------- If the lift unit are time and distance, time or cutting
X0.;           )                      distance is counted.
G00Z100.;
G91G28Z0M05;
```

4. Tool Status

The TOOL screen lists the status of each tool as follows. It is displayed when the machine specification requires it. Typical specification is mentioned in parentheses, but it is not always displayed.

USED ..... The tool is used.
MONITORED ..... The tool is used with load monitoring enabled.
OUT OF LIFE ..... The tool is out of life.
WORN OUT ..... The tool is worn out. (Cutting monitoring)
SKIP ..... The tool is skipped by a skip signal.
TOOL NOSE MEAS. N.G. ..... The tool was found out no good in tool nose measurement. (Macro measurement, etc.)
WORK MEAS. N.G. ..... The tool was found out no good in work measurement. (Macro measurement, etc.)
OVERLOAD ERROR ..... An overload error resulted. (Cutting monitoring)
NO-LOAD ERROR ..... A no-load error resulted. (Cutting monitoring)
EXTERNAL ERROR ..... An external error resulted. (Breakage detection, etc.)

18-2 Spare Tool Management

1. Feature

If the Txx command in the machining program is a spare tool command, the spare tool management function will automatically select a defectless tool out of a preregistered spare tool group. The tool is selected out of the spare tool group in order of used tool (MONITORED included), unused tool of higher order registration, and last selected tool if all the tools are out of life.

2. Functioning

Determine a group number for the tools of identical type which are specified as spare tools in the program, using an arbitrary number larger than the numerical value in the parameter no. 8078 (standard value: 1,000), and register it as a call number used to specify a spare tool in the SPARETOOL screen.
Register the spare tools in the tool group in ORDER 1 onward.
After setting the call number and spare tool number, specify a spare tool in the program.
The spare tool management function automatically selects a defectless tool out of the spare tool group.

3. Specifying the Offset

(1) Tool Offset by Tool Number

If a “tool offset by tool number” option is added to the NC unit for the SEICOS-Σ, tool offset can be applied by a spindle tool number without taking offset data “H” and “D” into account, even if the spare tools are used. (You only have to specify an offset instead of specifying H and D.)

(2) Tool Offset by Optional Block Skip

An offset for the spare tool is specified by informing the NC unit of a spare tool registration position as an optional block skip position. (For two or more tools, an NC option is required)

Caution

If the [OPTIONAL BLOCK SKIP] switch on the operation panel is used by an NC program, machine operation becomes unstable because it is duplicated by offset designation for the tool life (optional block skip). When using offset designation for the spare tool, separate the [OPTIONAL BLOCK SKIP] switch on the operation panel from the program used.

Make a machining program by combining an offset number with a block skip number.

4. Spare Tool Data

Since the tool life set value and life units for the tool group are set under CALL NUMBER, it is not necessary to set them for each spare tool. The tools registered as spare tools are displayed less clearly in the TOOLLIFE screen and you cannot set the data for them. Set the data in the SPARE TOOL screen.

If a tool number is registered in ORDER 1 in setting only the call number (the tool name and life set value of the call number are zero), the tool name and life data for the ORDER 1 tool will be set as the data for the call number.

Cutting monitoring is performed based on the data for the first spare tool.

18-3 Premachining Tool Check

1. Feature

At the beginning of the machining program, specify the tools to be used in the subsequent machining. The premachining tool check function will checks whether all the tools used, including spare tools, are defectless or not.

If there is even one defective tool, the machine stops, displaying a warning, “782 PREMACHINING TOOL CHECK ERROR.”

2. Functioning

If the tool used for machining is specified (spare tool) between M51 and M59 in the program, it will be checked if the specified tool is defectless or not. If there is even one defective tool, the machine stops upon completion of premachining tool check (M59), displaying “782 PREMACHINING TOOL CHECK ERROR.”

At this time, the tool numbers of defective tools are listed as defective tools in the TOOL STATUS screen. Interlocking this function with automatic return sets the next work and carries out next machining. (PPL system)
This function helps prevent production of incomplete products.

18-4 Tool Skip and Tool Reset

Input a tool skip signal from an external device. The tool status of the then spindle tool is designated SKIPPED and that tool will be skipped (not used) from the next time.

* When the tool status is designated SKIPPED and that tool or all the tools in the same tool group are defective, a tool change signal is output to the PC.

If a tool reset signal is input from an external device, the tool status and tool life use value of the then spindle tool will be cleared.
These signals are effective in any mode.

18-5 Tool Life Management Program

1. Special Purpose M-codes for Tool Life

(1) Disabling/Enabling the Tool Life and Cutting Monitoring Functions (M54/M53)
M54 is used when disabling the tool life and cutting monitoring functions. After using M54, cancel it by M53.
M54 ..... Stops the tool life/cutting monitoring function. Although it disables a tool status check, life count, premachining tool check, and tool life suspension (M55), spare tool call, tool offset, and life data setting remains enabled.
M53 ..... Cancels M54. When the power is turned on, M53 is selected.

(2) Machine Stop by Out-of-life Tool and its Reset (M56/M55)
M56 is used to stop the machine when a tool is out of life.
M56 ..... Stops the machine when an out-of-life tool comes to the spindle.
M55 ..... Cancels M56. When the power is turned on, M55 is selected.

(3) Assisting/Resetting Spare Tool Offset (M45/M46)
Block skip and optional block skip are used to select a spare tool offset. Be sure to enclose an offset selection by M45 and M46 to program it.
(Note) *Register M45 and M46 in the NC non-buffering M-codes.*

To select a tool offset, inform the NC unit of the registration position from "ORDER 1" registered as a spare tool by block skip.

Program Example (with Option)

```
T1100M06;
M45
G43Z_H1/2 H11/1 H10
M46
```

H12 -- Offset for the tool No. 12
H11 -- Offset for the tool No. 11
H10 -- Offset for the tool No. 10

CALL NO. T1100			BLOCK SKIP SIGNAL
ORDER	TOOL		
1	10		0
2	11		1
3	12		2

(Note 1) *When optional block skip is not effected, offsets for ORDER 3 onward are adjusted to the tool no. 10*.

(4) Premachining Tool Check (M51/M59)

This function checks whether or not the tools are defectless, including the spare tools. You specify the tool number you want to check between M51 and M59. A judgment result can be known when M59 is executed.

M51 ----- Starts a premachining tool check
M59 ----- Ends a premachining tool check

(Note) *A command other than Txx is specified between M51 and M59, an error message, "783 MONITORING FORMAT ERROR," appears. A T-code command for premachining tool check does not call a tool.*

(Example)

```
M51;
T01;
T20;
T1100;
M59;
```

Checks whether or not all the tools specified here are defectless.
(Checks the tools T01 and T20, and spare tool T1100)

(5) Life Data Automatic Setting (M57/M59)

This function downloads the tool life set value and use value from the NC program.

M57; ..... Starts downloading the life data.
Txx; ..... Specifies a tool number.
1st Sxxx; ..... Sets the life set value for Txx (Life set value for the group in case of call number)
2nd Sxxx; ..... Sets the life use value for Txx (Omissible, nothing is done in case of call number)
M59; ..... Ends downloading the life data

When the 1st S-code command is executed, it is determined by parameter setting to select the life units and clear the tool status and life use value.

- Parameter No. 8001 #6 = 1 ..... Times
- Parameter No. 8001 #7 = 1 ..... Minutes

- Parameter No. 8003 #7 = 0 ..... Clears the tool status and tool life use value when the tool life is set.
  8003 #7 = 1 ..... Does not clear the tool status and tool life use value when the tool life is set.

(Note) *When a command format is not correct, an error message, "783 MONITORING FORMAT ERROR," appears. After setting the data, be sure to specify a spindle rpm command, Sxxx.*

(Example)

M57; ..... Starts downloading the life data
T05; ..... Specifies the tool No. 5
S20; ..... Sets 20 as a life set value for the tool No. 5
;
T01; ..... Specifies the tool No. 1
S230; ..... Sets 230 as a life set value for the tool No. 1
S80; ..... Sets 80 as a use value for the tool No. 1
;
T5321; ..... Specifies the tool No. 5231
S150; ..... Sets 150 as a life set value for the call No. 5231
;
T30000; .... Specifies the call No.30000
S250; ..... Sets 250 as a life set value for the call No. 30000
S80; ..... Does not set a use value for the spare tool of the call No. 30000
;
M59; ..... Ends downloading the life data

2. Sample Program

Without Spare Tool		With Spare Tool
M51		M51
M51		M51
T10	Premachining tool check	T1100
T15		T1200
T30		T30
:		:
M59	Stops due to an out-of-life toll	M59
M56		M56
T10M06	Spindle tool command	T1100M06
T15	Standby tool command	T1200
G54G90S1000		G54G90S1000
G00X0Y0		G00X0Y0
		M45
G43Z50.0H10	Tool offset command	G43Z50.0H12/2H11/1H10
		M46
M03		
G98G81R3.0Z-10.0F500		
X-10.0		
Y-10.0		
G00Z100.0M15		
M01		
:		
M55	Does not stop due to an out-of-life tool	
:		

CALL NO. T1100
T10
T11
T12

CALL NO. T1200
T15
T16

18-6 Tool Life Management Operation

1. Preparations for Operation

After changing a setup and setting in the tool magazine the tools you want to use for next machining, input their tool data (tool name, life units, life set value, life use value) in the TOOL LIFE screen.

To operate with a spare tool, set in the SPARE TOOL screen the call number used to specify the spare tools in the machining program, and input the life units and life set values for that tool group.

Set the spare tools in ORDER 1, ORDER 2, and so on with tool numbers and set the life use value for each tool.

2. Machining Operation

Tool life management counts the tool lives of the tools used for machining by an NC program.

When the tool runs out of its life, this function assumes that tool OUT OF LIFE and continues machining. When the out-of-life tool is called to the spindle by a tool command in next machining, an error message, “781 OUT-OF-LIFE TOOL WAS SELECTED,” appears and a call light is turned on. In the M56 mode, the machine stops feeding the axes.

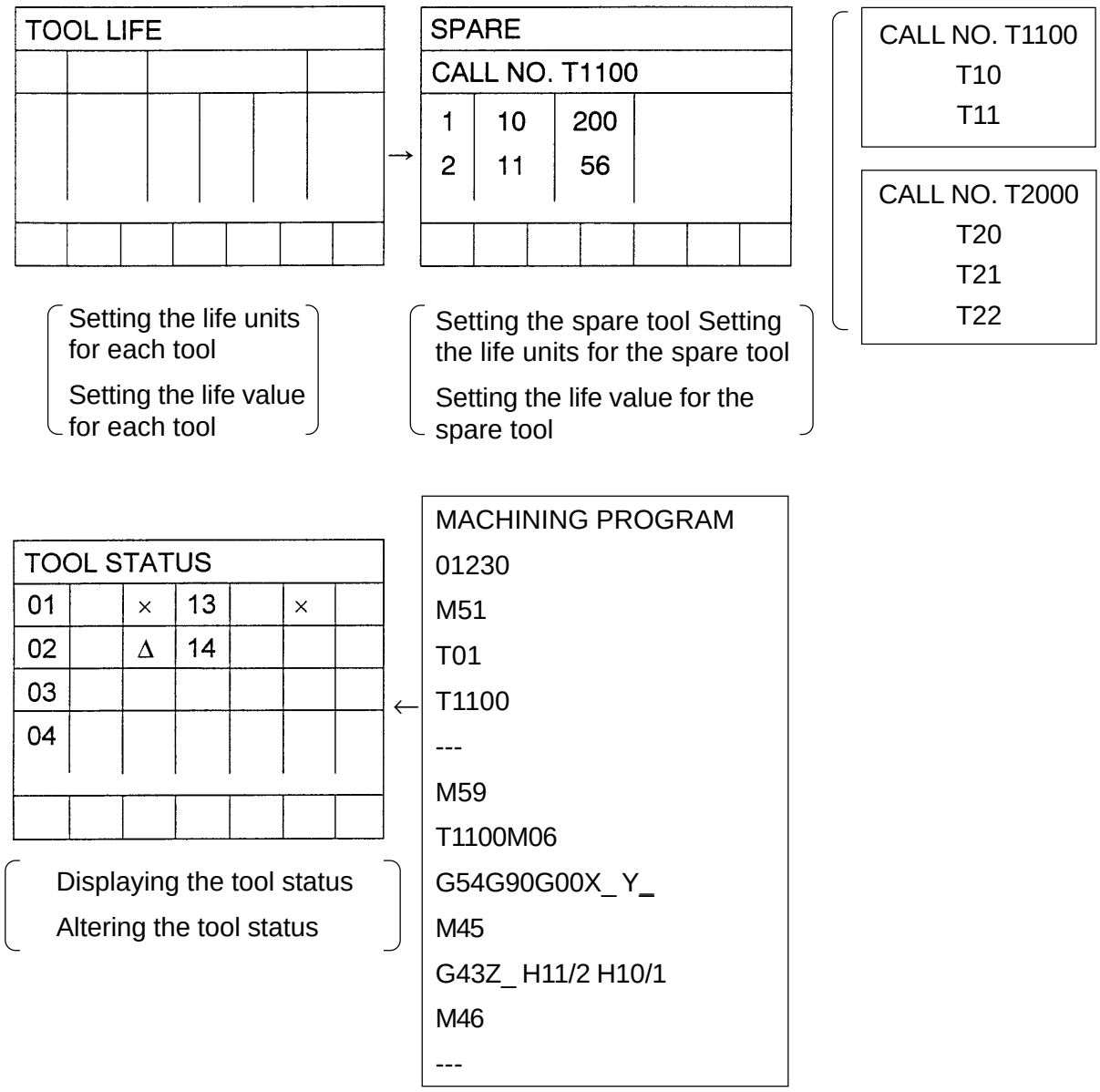
When the tool used is running out of its life, the tool status in the TOOL STATUS screen becomes “△”. Change the tool if early change is required.

3. Tool Change

When there is an out-of-life tool, stop the tool and change it to a new one.

After changing to the new tool, delete the tool life use value and tool status, and then, restart machining.

4. Tool Life Management Operational Procedure



18-7 Tool Life Screen

Press F3/TOOL in the OVERALL screen. The following function screen is displayed. Although the function keys, F1/TOOL NAME , F2/REF. GAUGE , F3/TOOL CHANGE , F4/ , F5/ , F6/SAFETY GUARD ,F7/DATA CLEAR , F8/SENSOR SET , F9/LIFE SPARE , and F0/ , are also displayed on the right end of the screen, they will be omitted hereinafter.

Press F9/LIFE SPARE .

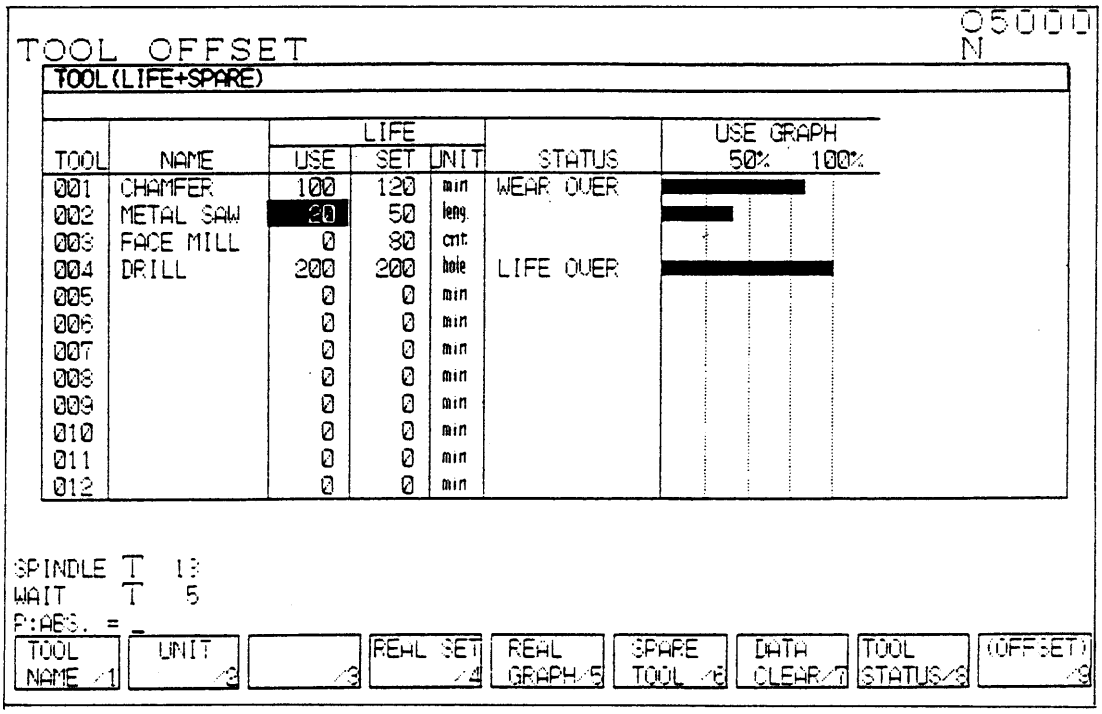


Fig. 1

(Note) *The data for the tools registered as spare tools are displayed dimly. No entry is allowed through the TOOL LIFE screen.*

1. Setting the Tool Name (A tool name can be set for each tool)

The tool life management function is not affected even if a tool name is not set.

[1] Using the Page keys [↑P] and [↓P] and cursor keys [↑] and [↓], move the cursor to the tool number whose tool number you want to set. Or, enter [N] followed by a tool number (numerical value) and press [↓].

[2] Press [F1/TOOL NAME] → A Tool Name Setting window is displayed.

[3] Input a tool diameter, etc. in the alphanumerical entry →The input data is displayed in the key input area.

[4] Move the cursor with the [↓] key and select a tool name.

[5] Press [INPUT] → The Tool Setting window disappears. Set the tool name.

○ When interrupting operation (turning off the Tool Setting window), press [F1/TOOL NAME] .

[Example] Setting [100 milling cutter] for T003

[1] Using the ↓ key, move the cursor to “003”.

[2] Press F1/TOOL NAME → The cursor is located at an alphanumerical entry.

[3] Press the keys, 1 , 0 , 0 , and SPACE in that order.

[4] Using the ↓ key, move the cursor to “MILL”.

[5] Press INPUT .

	TOOL NAME
	ALPHANUMERAL INPUT
	DELETE TOOL NAME
	MILL
	SENSOR
	DRILL
	BURNISH
	TAP
	REAMER
	END MILL
	CHAMFEWR

	TOOL NAME
	CHAMFER
	BORE
	BACK BORE
	SPOT FACE
	BACK SPOT FACE
	CORNER MILL
	SHELL
	ANGULAR
	SIDE MILL
	METAL SAW

2. Setting the Life Units (Setting the life units for each tool)

Life units ..... Minutes: Time, Times: ATC times,
Length: Cutting length,
Hole: Number of holes (number of holes machined by a canned cycle for drilling/tapping)

	LIFE UNITS
	MINUTES TIME
	TIMES: ATC TIMES
	LENGTH: CUTTING LENGTH (m)
	HOLE: NUMBER OF HOLES

Initially, “Minutes” is selected. When you want to alter it;

[1] Using the ⇧P, ⇩P, ↑, and ↓ keys, move the cursor to the tool number for which you want to set a life unit. Or, enter ↓ followed by a tool number (numerical value), and press N .

[2] Press F2/UNIT → A Life Unit Setting window is displayed.

[3] Using the ↓ key, move the cursor and select a life unit.

[4] Press INPUT →The Life Unit Setting window disappears and the life unit is set.

○ When interrupting operation (turning off the window), press F2/UNIT .

3. Setting the Life Value

[1] Using the ⇧P, ⇩P, ↑, and ↓ keys, move the cursor to the tool number for which you want to set a life value. Or, enter ↓ followed by a tool number (numerical value), and press N .

[2] Using the → key, move the cursor to the [SETTING] field.

[3] Input a life value (numerical value) and press INPUT .

For a tool life value, set the life value preset for each tool, when the tool with unknown life value is used, machining is performed assuming a set value zero. When the tool becomes

unsharp, take note of the life use data. That data is used as a life set value.
The tool is judge out of life in the following cases:

- Rough finishing surface
- Occurrence of burr
- High cutting sound
- Much chattering
- Adhesion of a built-up edge

4. Deleting the Use and Setting Data

- Deleting the Use, Setting, and Status Data for All the Tools
  When there are old data remaining, delete all the data first.
  [1] Press F7/DATA CLEAR → A Delete Item window is displayed.
  [2] Using the ↓ key, move the cursor to “DELETE (USE, SETTING, & STATUS) FOR ALL TOOLS”.
  [3] Press INPUT → A message, “OK.? Y-YES N-NO,” is displayed.
  [4] Press Y → The Delete Item window disappears and the use and setting data for all the tools are deleted.
  ○ When interrupting operation (turning off the Delete Item window), press F7/DATA CLEAR again or press N in [4].

	DELETE ITEM
	DELETE USE & STATUS
	DELETE (USE & STATUS) FOR ALL TOOLS
	DELETE (USE, SETTING, & STATUS) FOR ALL TOOLS

- Clearing the use value when the out-of-life tool is changed
  [1] Using the ⇧, ⇩, ↑, and ↓ keys, move the cursor to the tool number whose tool was changed. Or, enter N followed by the tool number (numerical value), and press ↓.
  [2] Move the cursor to [DELETE USE & STATUS].
  [3] Press INPUT → A message, “OK.? Y-YES N-NO,” is displayed.
  [4] Press Y →The Delete Item window disappears and the tool use data and status are deleted.

Or,

  [1] Using the ⇧, ⇩, ↑, and ↓ keys, move the cursor to the tool number whose tool was changed. Or, input the N tool number (numerical value), and press ↓.
  [2] Using the → key, move the cursor to “USE”.
  [3] Press 0 following by INPUT → The use value and status are deleted.

18-8 Spare Tool Screen

In the screen shown in Fig. 1, press F6/SPARE TOOL. The SPARE TOOL screen in Fig. 2 appears.

```
TOOL OFFSET                                                    O5000
                                                               N
TOOL(LIFE+SPARE)

2.GROUP   T1100          (FACE MILL ) LIFE SET   200 min
                                    LENGTH              RADIUS
No   TOOL   USE   STATUS   GEOMETRY   WEAR   GEOMETRY   WEAR
1    10     200   LIFE OVER  0.000   0.000   0.000      0.000
2    11     56               0.000   0.000   0.000      0.000
3    12     0                0.000   0.000   0.000      0.000
4

3.GROUP   T12000         (DRILL     ) LIFE SET   500 cnt
                                    LENGTH              RADIUS
No   TOOL   USE   STATUS   GEOMETRY   WEAR   GEOMETRY   WEAR
1    15     0                0.000   0.000   0.000      0.000
2

SPINDLE T  13
WAIT    T   5
GROUP No (ALT) = _
TOOL NAME /1 | UNIT /2 | TOOL INSERT/3 | REAL SET /4 | REAL GRAPH/5 | TOOL LIFE /6 | DATA CLEAR/7 | TOOL STATUS/8 | (OFFSET) /9
```

Fig. 2

1. Setting the Call Number

[1] Move the cursor to a position of “CALL NO.”.

[2] Input a call number (1,000 or greater) and press INPUT.

○ The call number must be greater than the value set in the monitor parameter No. 8078. (Standard value: 1,000)

** What Is the Call Number ? **

This is a number used to change tools in the machining program. It is a tool number for each tool group.

```
Machining Program      Call No. T1100
                                     T10
T1100M06                             T11
G54G90G00X__Y__                      T12
G43Z__H__
                                T1200
T1200M06                             T20
```

2. Setting the Tool Name

Set a tool name for the tool group. The spare tool function is not affected even if the tool name is not set.

3. Setting the Life Units

Set the life units for the tool group.

4. Setting the Life Value

Set a life value for the tool group

[1] Move the cursor to “LIFE SETTING”.

[2] Input a life value (numerical value) for the tool group and press INPUT.

5. Registering the Call Number (Spare Group)

[1] Move the cursor to a call number (any number will do).

[2] Press F3/TOOL INSERT.

[3] Input the call number (numerical value) you want to register additionally and press INPUT.

→ With the cursor located at a position of the added call number, the call numbers are redisplayed in the numerical order.

6. Deleting the Call Number (Spare Group)

- Deleting the arbitrary call number (spare group)

[1] Move the cursor to the call number you want to delete.

[2] Press 0 followed by INPUT → The call number and spare tool are deleted.

	DELETE ITEM
	DELETE LIFE & STATUS
	DELETE SPARE GROUP
	DELETE ALL SPARE GROUPS
	DELETE ALL GROUPS

- Deleting all the call numbers (spare group)

[1] Press F7/DATA CLEAR → The Delete Item window is displayed.

[2] Using the ↓ key, move the cursor to “DELETE ALL GROUPS”.

[3] Press INPUT → A message, “OK.? Y-YES N-NO,” appears.

[4] Press Y →The Delete Item window disappears and all the call numbers and spare tools are deleted.

○ When interrupting operation, press F7/DATA CLEAR again.

7. Registering the Spare Tool

- Registering the first spare tool

[1] Move the cursor to “TOOL” for ORDER 1.

[2] Input a tool number which you want to register as a spare tool, and press INPUT .

(Note) *When a spare tool is set in ORDER 1 with the tool name of the call number and its life setting data left unset, the tool name and life set value of the tool registered in ORDER 1 are also set for the call number.*

- Additionally registering to the end of the spare tools

[1] Move the cursor to “TOOL” for last ORDER X.

[2] Input the number you want to register as a spare tool, and press INPUT.

- Registering halfway the spare tools

[1] Move the cursor to the position where you want to add a spare tool.

[2] Press F3/TOOL INSERT . (The spare tool is inserted into the line above the cursor)

[3] Input the number you want to add, and press INPUT.

○ When interrupting operation, press F3/TOOL INSERT again.

8. Deleting the Spare Tool

- Deleting an arbitrary spare tool

[1] Move the cursor to the tool you want to delete.

[2] Press 0 (zero) followed by INPUT .

- Deleting the spare tool group of the call number

[1] Move the cursor to any position of the call number group you want to delete.

[2] Press F7/DATA CLEAR → The Delete Item window is displayed.

[3] Using the ↓ cursor, move the cursor to “DELETE SPARE GROUP”.

[4] Press INPUT → A message, “OK.? Y-YES N-NO,” appears.

[5] Press Y →The Delete Item window disappears and the spare tools in the group are deleted.

○ When interrupting operation, press F7/DATA CLEAR again.

18-9 Tool Status List Screen

In the screen shown in Fig. 1, press F8/TOOOL STATUS.

○ When interrupting operation, press F8/TOOOL CHECK to display the DEFECTIVE TOOL LIST screen, and then, press F8/TOOL STATUS .

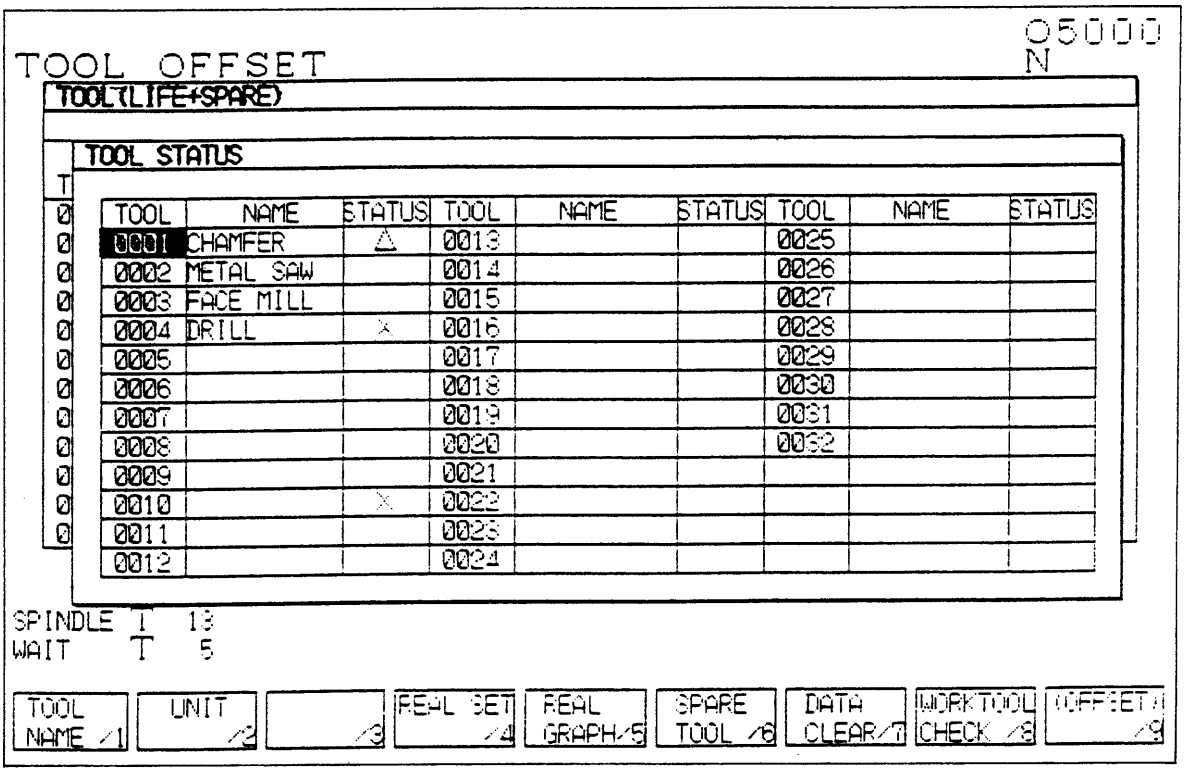


Fig. 3

<Tool Status> × ..... Defective tool (broken or out of life)

△ ..... Running out of life (○○ % or more of the parameter No. 8060)

Blank ..... Usable (Less than ○○ % of the parameter No. 8060)

1. Deleting and Skipping the Tool Status

[1] Move the cursor to a tool number.

[2] Press F7/DATA CLEAR →The Delete Item window appears.

[3] Move the cursor to your desired delete item to delete.

[4] Press INPUT → A message, “OK.? Y-YES N-NO,” appears.

[5] Press Y →The Delete Item window disappears and the selected item is deleted.

	DELETE ITEM
	DELETE THE USE AND STATUS DATA. SKIP STATUS.

2. Listing the Defective Tools

Press [F8/WORK TOOL CHECK] in the TOOL STATUS LIST screen. The DEFECTIVE TOOL LIST screen appears.

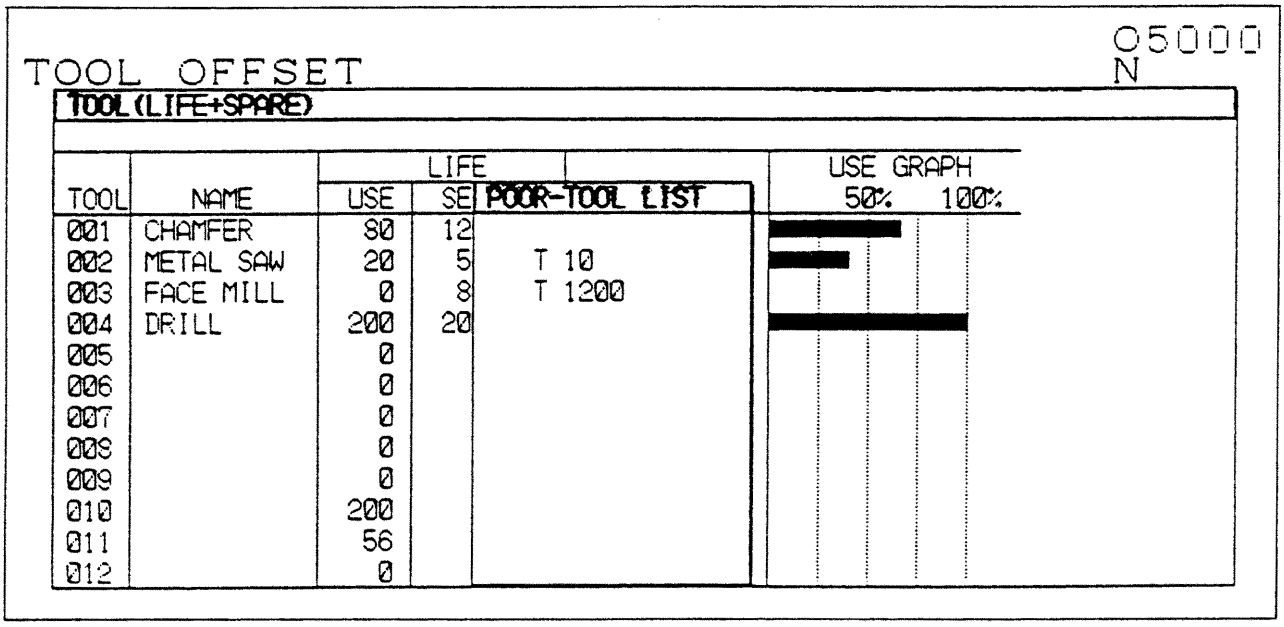


Fig. 4

This screen lists up to 10 tools found defective in one premachining tool check (M51 to M59).

Since the defective tools are not displayed cumulatively, the screen displays results of the last premachining tool check.

18-10 Tool Life Management Parameters and Their Setting

1. Displaying the Parameters

Press [F4/SYSTEM] and enter [1], [INPUT], followed by [N] [8] [0] [0] [0]. With the [↓] arrow key, display a cutting monitoring parameter, No. 8000.

2. Setting the Parameter

[1] [MDI] mode
[2] Reset state
[3] Turn on the [WRITE] switch.
[4] Rewriting of parameters is enabled (SETTING screen)
[5] Move the cursor to the number you want to set
[6] Input the setting data with the keys
[7] Press the INPUT key

3. Parameters Related to Tool Life

- No. 8001 #6 ..... Sets the tool life in terms of times when deleting all the data.
- No. 8001 #7 ..... Sets the tool life in terms of minutes when deleting all the modes.
- No. 8003 #0 ..... Spare tool position output 1 to 9
- No. 8003 #7 ..... Clears the tool status and life use value when downloading the life set value by M57.
- No. 8060 ..... % data marked by "Δ" in the TOOL STATUS LIST screen (Standard value: 80)
- No. 8078 ..... In call number offset, tool numbers greater than this number will be call numbers for the spare tools. (Standard value: 1000)

18-11 Alarms

780 BROKEN TOOL WAS SELECTED

This alarm is issued when a broken tool is called to the spindle position. When manually resetting this alarm, check and change the tool and reset the alarm.
After replacing the tool by a new one, clear the life use value and tool status for the old tool.

781 OUT-OF-LIFE TOOL WAS SELECTED

This alarm is issued when an out-of-life tool is called to the spindle position. When manually resetting this alarm, check and change the tool and reset the alarm.
After replacing the tool by a new one, clear the life use value and tool status for the old tool.

782 PREMACHINING TOOL CHECK ERROR

One of the tools checked in a premachining tool check was found defective (out of life, broken, and so on)
Replace the defective tool by a new one and clear the life use value and tool status for the defective tool.

783 CUTTING MONITORING FORMAT ERROR

There is a programming error in premachining tool check or tool life data automatic setting.
Correct a program command and rerun.

19. Cutting Monitoring Unit

19-1 Outline

The cutting monitoring unit monitors the cutting loads of the spindle and feed axes of the NC machine tool to prevent abnormal cutting and manufacturing of defective products.
You can set monitoring data and monitoring functions for each tool to ensure detailed monitoring.
The tool life management function is required because this unit is interlocked with it.

19-2 Monitoring Method

1. Monitoring Axes

The cutting monitoring unit monitors the loads of the spindle motor and NC feed axes (X-, Y-, and Z-axis) at real time. Monitoring is performed by a combination of a main monitoring axis, where all the monitoring functions work, and a sub-monitoring axes, where overload detection and wear detection work. You set for each tool which axis should monitor in what way.
The main monitoring axis is one of the spindle, X-axis, Y-axis, and Z-axis, and the sub-monitoring axes are the other axes.

- Main monitoring axis ..... Mainstay of cutting monitoring and provides overload detection, no-load detection, wear detection, adaptive control, and idle cutting time reduction.
- Sub-monitoring axes ..... Detects an overload and wear of the sub-monitoring axes in combination with the main monitoring axis.

Normally, monitoring by the main monitoring axis is enough, where the spindle motor has been set as the main monitoring axis. When you want to monitor by the feed axes simultaneously, however, turn on monitoring by the sub-monitoring axes. For a large-diameter drill, and so on whose cutting load is clearly detected in the thrust direction, the Z-axis can be used as the main monitoring axis.
When monitoring the load of the feed axes, monitor them without changing the cutting conditions. They may malfunction if you change a feed rate override, and so on halfway.
When the load of a cutting axis is not known or a cutting load is small, do no monitor by the feed axes because monitoring by them will result in erroneous detection.

2. Monitoring Data Unit

Monitoring data is displayed in terms of load factor, down to the first decimal place.
A load factor of “100.0%” has the following meaning.

- Spindle ..... 30-minute rated output (120% at maximum)
- Feed axes ..... Maximum continuous rated output

3. Selecting the Monitoring Function

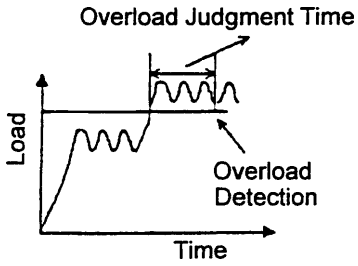
The monitoring functions for the main monitoring axis include overload detection, no-load detection, wear detection, adaptive control, and idle cutting time reduction, and those for the sub-monitoring axes include overload detection and wear detection.
Select a desired function for each tool in the SET LOAD screen.

19-3 Monitoring Functions

1. Overload Detection

When a cutting load keeps exceeding an overload detection value for an overload judgment time, this function detects an overload, judging it a cutting error (tool-workpiece interference, improper machining condition, or defective tool nose).
When an overload is detected, an error message, “O AXIS OVERLOAD ALARM,” is displayed and the machine stops feeding the axes and stops the spindle.

Spindle overload judgment time ..... Parameter 8031 × 0.1 second
X-axis overload judgment time ..... Parameter 8035 × 0.1 second
Y-axis overload judgment time ..... Parameter 8039 × 0.1 second
Z-axis overload judgment time ..... Parameter 8043 × 0.1 second

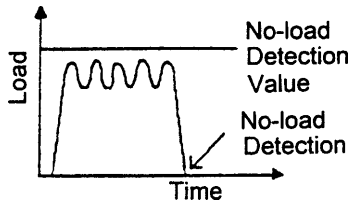


2. No-load Detection

When a cutting load does not exceed a no-load detection value in each cutting, this function detects a state of no load, judging it a cutting error (tool breakage, improper workpiece mounting, no tool setting, or program error), because there was no cutting state found.
When the state of no load is detected, an error message, “NO-LOAD ALARM,” is displayed and the machine stops feeding the axes and stops the spindle.
(Note) *No-load detection is activated when overload detection is enabled.*

Each cutting is;

- Spindle ..... Between M03 and M04/M05 (M13 or M14 also allowed)
- Feed axes .... Between start and end of cutting feed.



3. Wear Detection

When a cutting load reaches a state of wear load, that tool's status is regarded "worn out." Machining continues even if wear is detected, but that tool will be treated in the same manner as an out-of-life tool in a tool command in next machining.

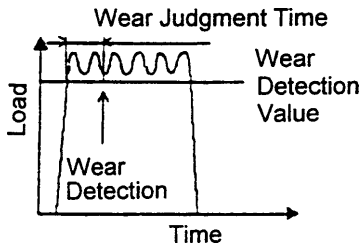
Since wear detection and adaptive control cannot be enabled simultaneously, disable adaptive control when you enable wear detection. There are two wear detecting methods; load time detection or load calculation detection.

(Note) *Wear detection is activated when overload detection is enabled.*

(a) Load time detection mode (Wear judgment time parameter ≠ 0)

When a cutting load keeps exceeding a wear detection value for wear judgment time, wear is detected and that tool is regarded worn out.

Spindle wear judgment time .... Parameter 8032 × 0.1 second
X-axis wear judgment time ..... Parameter 8036 × 0.1 second
Y-axis wear judgment time ..... Parameter 8040 × 0.1 second
Z-axis wear judgment time ..... Parameter 8044 × 0.1 second



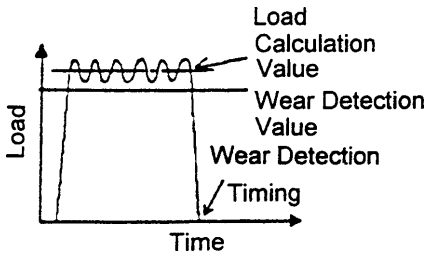
(b) Load calculation detection mode (Wear judgment time parameter = 0)

A cutting load value is calculated out of each cutting load, and when it exceeds a wear detection value, wear is detected and that tool is regarded worn out.

A cutting load value is calculated out of all the load data collected from one cutting by a data sampling and averaging method (sampling and averaging the load values in a certain range). A specific calculation range is specified by a parameter.

Spindle ..... Parameter 8010/8011
X-axis ..... Parameter 8016/8017
Y-axis ..... Parameter 8022/8023
Z-axis ..... Parameter 8028/8029

Unless the parameters are correct, a maximum load value is regarded a cutting load value.



4. Adaptive Control (Feed Rate Override Control)

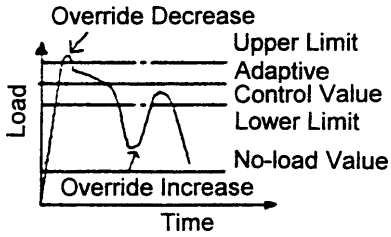
When a cutting load exceeds a no-load value into a cutting state, a feed rate override is controlled so that the cutting load will stay between an upper-limit adaptive value and lower-limit adaptive value.

When the cutting load is still exceeding the upper-limit adaptive value for 2 seconds even if the override is minimized, an overload is detected, judging it a cutting error.

Since adaptive control and wear detection cannot be enabled simultaneously, disable wear detection when you enable adaptive control.

(Note) *Overload detection must be working.*

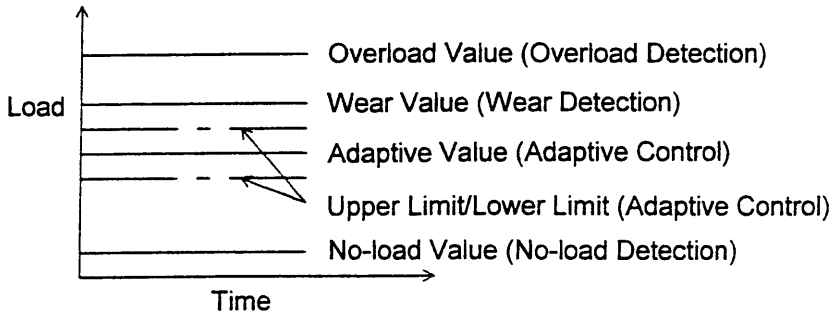
Upper-limit adaptive value ..... Adaptive control value × Parameter 8050
Lower-limit adaptive value ..... Adaptive control value × Parameter 8051
Decrease judgment time ..... Parameter 8055 × 0.1 second
Increase judgment time ..... Parameter 8056 × 0.1 second
Override interval ..... Interval between override increase and decrease
Maximum override ..... Maximum override value
Minimum override ..... Minimum override value



5. Idle Cutting Time Reduction (Feed Rate Override Control)

In an idle cutting state where a cutting load is lower than a no-load value, a feed rate override is set to 200% to reduce an idle cutting time.

* The following figure lists each set value in terms of load versus time.



19-4 Cutting Monitoring Program

1. Cutting Monitoring Special M-code

(1) Stopping/Resetting the Tool Life Management Function and Cutting Monitoring Function (M54/M53)

Since a cutting load does not take place up to an actual machining start point in remachining from halfway, M54 is executed to disable the tool life management function and cutting monitoring function when cutting monitoring makes erroneous detection. Operation is restarted after executing M53 at the actual machining start point.

M54 ..... Stop the tool life management function and cutting monitoring function. Each detecting function related to cutting monitoring does not work. Note that the tool life management related functions do not work.

M53 ..... Cancels M54. When the power is turned on, M53 is selected.

(2) Monitoring Data Automatic Setting (M58/M59)

This function downloads the monitoring data from an NC program. Settable data include an overload value, adaptive value, and wear value for the main monitoring axis, and an overload value for the sub-monitoring axis.

M58 ; ..... Starts downloading the monitoring data
Txx ; ..... Specifies a tool number
1st Sxxx ; ..... Sets overload value data for the main monitoring axis
2nd Sxxx ; ..... Sets adaptive value data for the main monitoring axis
3rd Sxxx ; ..... Sets wear value data for the main monitoring axis
4th Sxxx ; ..... Sets overload data for the X-axis position of the sub-monitoring axis
5th Sxxx ; ..... Sets overload data for the Y-axis position of the sub-monitoring axis
6th Sxxx ; ..... Sets overload data for the Z-axis position of the sub-monitoring axis
M59 ; ..... Ends downloading the monitoring data

<Notes>

- *When a tool number is a call number, the data is set in the TOOL column of ORDER 1 registered in CALL NO.*

CALL NO. 1100			
ORDER	TOOL	USE	STATUS
1	25	0	
2	26	0	
3			

M58;
T1100;... Indicates a call No. 1100. A data setting area is TOOL 25 of ORDER 1.
S200;... Sets a main monitoring axis overload value for TOOL 25 to 20.0.
S150;... Sets a main monitoring axis adaptive value for TOOL 25 to 15.0.
M59;

- *For Sxxxx;, set the data in the format of xxx.x.*
- *When the valid data is set, the monitoring functions are enabled.*
- *When an overload value for the main monitoring axis is set, other monitoring data are cleared.*
- *When an adaptive value is set, a no-load value is automatically calculated and set.*

  No-load value = Adaptive value × Parameter 8009 ÷ Parameter 8008
- *The 2nd S-code command onward are omitted unless there is setting data.*
- *When a data format is not correct, an error message, "783 MONITORING PROGRAM FORMAT ERROR," appears.*
- *Specify a spindle rotation command again after downloading the data.*

```
M58;
T01;
S125;
T05;
S450;
S370;
S0;
S176;
T1100;
S200;
S150;
M59;
```

Sets a main monitoring axis overload value for the tool No. 01 to 12.5.

Sets a main monitoring axis overload value for the tool No. 05 to 45.0.
Sets a main monitoring axis adaptive value for the tool No. 05 to 37.0.
Sets a main monitoring axis adaptive value for the tool No. 05 to 0.0.
Sets a sub-monitoring X-axis overload value for the tool No. 05 to 17.6.

Sets a main monitoring axis overload value for the tool No. 25 to 20.0.
Sets a main monitoring axis adaptive value for the tool No. 25 to 15.0.

2. Sample Program

```
O77;
M51;
T01;
T1000;
M59;
```

Checks the tool status of T01 and T1000 in a premachining tool check → An alarm is issued to stop the machine if M59; there is any defective tool.

```
M56;
```

..... Selects a feed hold stop because of an out-of-life tool.

```
G91G28X0Y0Z0;
G91G30X0Y0Z0M19;
T01M06;
```

..... Tool command (T01 to the spindle).
The machine stops feeding the axes if the tool is out of life.

```
G54G90G00X0Y100.;
M03S1000T1000;
G43Z50.H01;
G01Y0.F620;
G00X10.;
G01Y100.;
G00Z100.M15;
```

Spindle motor monitoring section ..... Feed monitoring section

..... Feed monitoring section

```
G91G30Z0;
G30X0Y0M19;
;
T1100M06;
```

..... Spare tool command (T25/T26/T27 to the spindle)

CALL NO. 1100			
ORDER	TOOL	USE	STATUS
1	25	0	
2	26	0	
3	27	0	

```
G54G90G00X0Y100.;
M03S1000;
M45;
G43Z30. H27/2 H26/1 H25;
M46;
G01Y0.F620;
G00X10.;
G01Y100.;
H00Z100.M15;
G91G30Z0;
G30X0Y0M19;
M30;
```

Selects a spare tool offset
(See Tool Life Management)
.....Feed monitoring section

Spindle motor monitoring section .....Feed monitoring section

19-5 Data Setting

1. Selecting the Main Monitoring Axis

In the SET LOAD screen, select the spindle, X-axis, Y-axis, or Z-axis as the main monitoring axis for each tool. When machining is done with this tool, determine on which one of the spindle motor, X-axis, Y-axis, and Z-axis a change of the cutting load is reflected, and set that axis as the main monitoring axis.

Normally, select the spindle motor as the main monitoring axis to monitor, but it is possible to select one of the X-, Y-, and Z-axis as the main monitoring axis to monitor cutting.

The main monitoring axis monitors the loaded state of a selected axis to activate overload detection, no-load detection, wear detection, adaptive control, and idle cutting time reduction.

2. Selecting the Monitoring Functions

In the SET LOAD screen, you can turn on/off each function switch for each tool to select how you want to activate the cutting monitoring functions on which axis, such as overload detection, no-load detection, wear detection, adaptive control, and idle cutting time reduction.

3. Setting the Monitoring Data

(1) Manual Setting

When the cutting monitoring data for the tool is known, set in the SET LOAD screen the monitoring data corresponding to a tool number (pot) at setup time.

[1] Setting the individual data

To set the individual data such as OVERLOAD, WEAR, ADAPTIVE, and NO LOAD, move the cursor to a data setting position, input the setting data, and press INPUT.

[2] Setting the data with a reference value

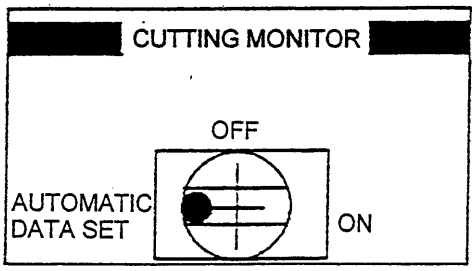
This is a setting method with a load reference value. When a cutting load reference value for that tool is input with the cursor located at REF. VALUE, the monitoring data for the function set to "● ON" is automatically set according to the parameter setting. The data for the function set to "● OFF" is cleared.

<Example> Calculated load values when the spindle motor's reference value is input

- OVERLOAD ..... (Reference value × Parameter No. 8006)/100
  (X: 8012, Y: 8018, Z: 8024)
- WEAR ..... (Reference value × Parameter No. 8007)/100
  (X: 8013, Y: 8019, Z: 8025)
- ADAPTIVE ..... (Reference value × Parameter No. 8008)/100
  (X: 8014, Y: 8020, Z: 8026)
- NO LOAD ..... (Reference value × Parameter No. 8009)/100
  (X: 8015, Y: 8021, Z: 8027)

[3] Setting the data by downloading
Download the monitoring data from the NC program.

(2) Automatic Setting



Set the CUTTING MONITOR mode selector switch on the operation panel to AUTOMATIC DATA SET and perform teach cutting. The load reference value for the tool used for machining is calculated from the cutting data for the monitoring section and set. If the reference value is set, the monitoring data for the selected function is automatically set as with 2 Setting the data with the reference value in (1) Manual Setting.
Monitoring is performed with the automatically set data. When the monitoring data is low, however, erroneous detection may occur because the monitoring conditions become strict. If this is the case, alter the monitoring data to change the monitoring conditions, and monitor cutting. The data is altered as described in 1 Setting the individual data in (1) Manual Setting.
When setting the data automatically, the standard data have been set in the parameters as reference data as shown in the table below. When you want to change them depending on the machining conditions or monitoring conditions, set the data according to the working conditions.

	Spindle		X-axis		Y-axis		Z-axis	
	Parameter NO.	Data	Parameter NO.	Data	Parameter NO.	Data	Parameter NO.	Data
OVERLOAD	8006	150	8012	200	8018	200	8024	200
WEAR	8007	120	8013	120	8019	120	8025	120
ADAPTIVE	8008	100	8014	100	8020	100	8026	100
NO LOAD	8009	60	8015	60	8021	60	8027	60
AV HIG	8010	95	8016	95	8022	95	8028	95
AV LOW	8011	90	8017	85	8023	85	8029	85

<Calculation Example> When the load reference value for teach cutting is 40.0% in monitoring the spindle motor load = (40 × 150 (parameter 8006))/100 = 60.0%

4. Monitoring Section

The spindle motor is monitored while the spindle is rotating in the forward direction (spindle reverse rotation also allowed), and the NC feed axes are monitored in the cutting feed section. During data unstable time at the time of start, the data for each axis is cancelled for the time specified by the respective parameters.

Spindle ..... Parameter 8030 × 0.1 second
X-axis ..... Parameter 8034 × 0.1 second
Y-axis ..... Parameter 8038 × 0.1 second
Z-axis ..... Parameter 8042 × 0.1 second

Caution

Every time the S-data is altered while the spindle is rotating , a cancellation time is activated. For the feed axes, the cancellation time is activated when cutting starts, but it is not between continuous cutting feed block. When the machine is operated in the SINGLE BLOCK mode, an axis is started at machine start time, and therefore, the cancellation time is activated every time. Note that the monitoring conditions are changed by turning on/off the [SINGLE BLOCK] switch. Monitor cutting at a constant feed rate override.

19-6 Cutting Monitoring Operation

Prior to starting monitoring operation, set the monitoring functions and monitoring data for the tool whose cutting you want to monitor. The monitoring data becomes valid when the tool is brought from a standby position to the spindle. If the spindle has the same tool since a start of machining after the power is turned on, cutting is not monitored until next tool change. The same applies when the monitoring data is changed halfway monitoring.

To enable the monitoring function, turn on a relevant function switch and set the monitoring data for the function. Overload detection must be working.

The following table lists the conditions to enable each of the monitoring function.

○----- Condition required to activate the function

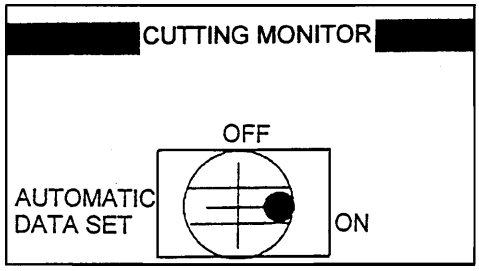
× ----- Condition you should not select to activate the function

	Overload Detection	Wear Detection		Adaptive Control	No-load Detection	Idle Cutting Time Reduction
		By Time	By Calculation			
Overload Detection ON	○	○	○	○	○	○
Overload	○	○	○	○	○	○
Wear Detection ON		○	○			
Wear Value		○	○			
Adaptive Control ON		×	×	○		
Adaptive Value				○		
No-load Detection ON				○	○	○
No-load Value					○	○
Idle Cutting Reduction ALL/BEFORE/AFTER						○

<Example> To enable this function in wear detection, set the overload detecting function to ON, an overload value to non-zero, wear detecting function to ON, and a wear value to non-zero in the SET LOAD screen in case of either wear detection by time or calculation. However, it will not work if the adaptive control function is set to ON.

1. Monitoring Conditions

Basic monitoring conditions are as follows:



[1] Completion of tool change operation (From the standby position to the spindle)

[2] In memory operation

[3] The [CUTTING MONITOR] switch is set to [ON]

[4] M53 mode (M54 is not specified in the program)

[5] Overload detection is working (Unless overload detection is working, the other monitoring functions will not)

When the above-mentioned conditions are met, the following monitoring sections are monitored according to the selected function and set monitoring data.

Spindle motor ..... During spindle forward rotation (reverse rotation also allowed by parameter setting)

Feed axis motor ..... During cutting feed

2. Overload Detection

Turn on the overload detecting function for a desired tool and set the overload detection data. Overload monitoring operation is activated in the monitoring section. When the monitoring conditions are met, an overload is monitored simultaneously with four axes.

To activate each of the functions such as no-load detection, wear detection, adaptive control, and idle cutting time reduction, be sure to activate overload detection.

When an overload is detected, an error message, "○ AXIS OVERLOAD ALARM," is displayed and the machine stops the spindle after stopping axis feed.

(1) Manual return

Reset the NC unit to move each axis to a safe position, and replace a broken tool. After replacing it, clear the life use value and maximum load value for the broken tool.

(2) Automatic return (depending on the machine specification)

In case of automatic return, store the broken tool status. When an overload is detected, the NC unit is reset by a sequence and a return program is started by an NC unit external sequence search to perform next machining. After a broken tool is replaced, clear the tool status and life use value for the replaced tool, and the maximum load value.

3. No-load Detection

Turn on the no-load detecting function for a desired tool and set the no-load detection data. No-load detection is made every time cutting is completed.

When no-load detection is made, an error message, "NO-LOAD ALARM," is displayed and the machine stops the spindle after stopping axis feed. A resetting method is the same as in 2. Overload Detection.

Each cutting is;

- Spindle ..... Between M03 and M04/M05 (M13 or M14 also allowed)
- Feed axes .... Between start and end of cutting feed

4. Wear Detection

Turn on the wear detecting function for a desired tool and set the wear detection data. Wear detecting operation is activated. When wear is detected, that tool is regarded worn out, but cutting continues as it is.

If there is a spare tool in next machining, it will be selected automatically. If a worn-out tool is brought to the spindle, an error message, “781 OUT-OF-LIFE TOOL WAS SELECTED,” is displayed. When M56 has been executed in advance, the machine stops after stopping axis feed.

Replace a worn-out tool and clear the tool status and life use value for the replaced tool, and the maximum load value.

5. Adaptive Control

no-load data. Adaptive control is activated.

When a cutting load exceeds a no-load value into a cutting state, a feed rate override is controlled so that the cutting load will stay between an upper-limit adaptive value and lower-limit adaptive value.

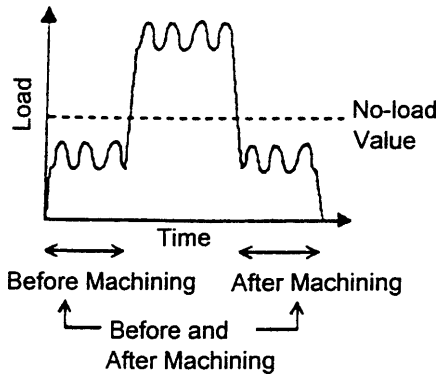
When the cutting load keeps on exceeding the upper-limit adaptive value for a descent judgment time, the override is decreased, and when it keeps staying lower than the lower-limit adaptive value for an increase judgment time, it is increased.

Even if the override is minimized, an overload is detected if the cutting load keeps exceeding the upper-limit adaptive value.

6. Idle Cutting Time Reduction

Select the idle cutting function to either “ALL,” “BEFORE,” or “AFTER” for the tool whose idle cutting time you want to reduce, and set the no-load data. A feed rate override is set to 200% in the idle cutting state in which the cutting load does not exceed a no-load value.

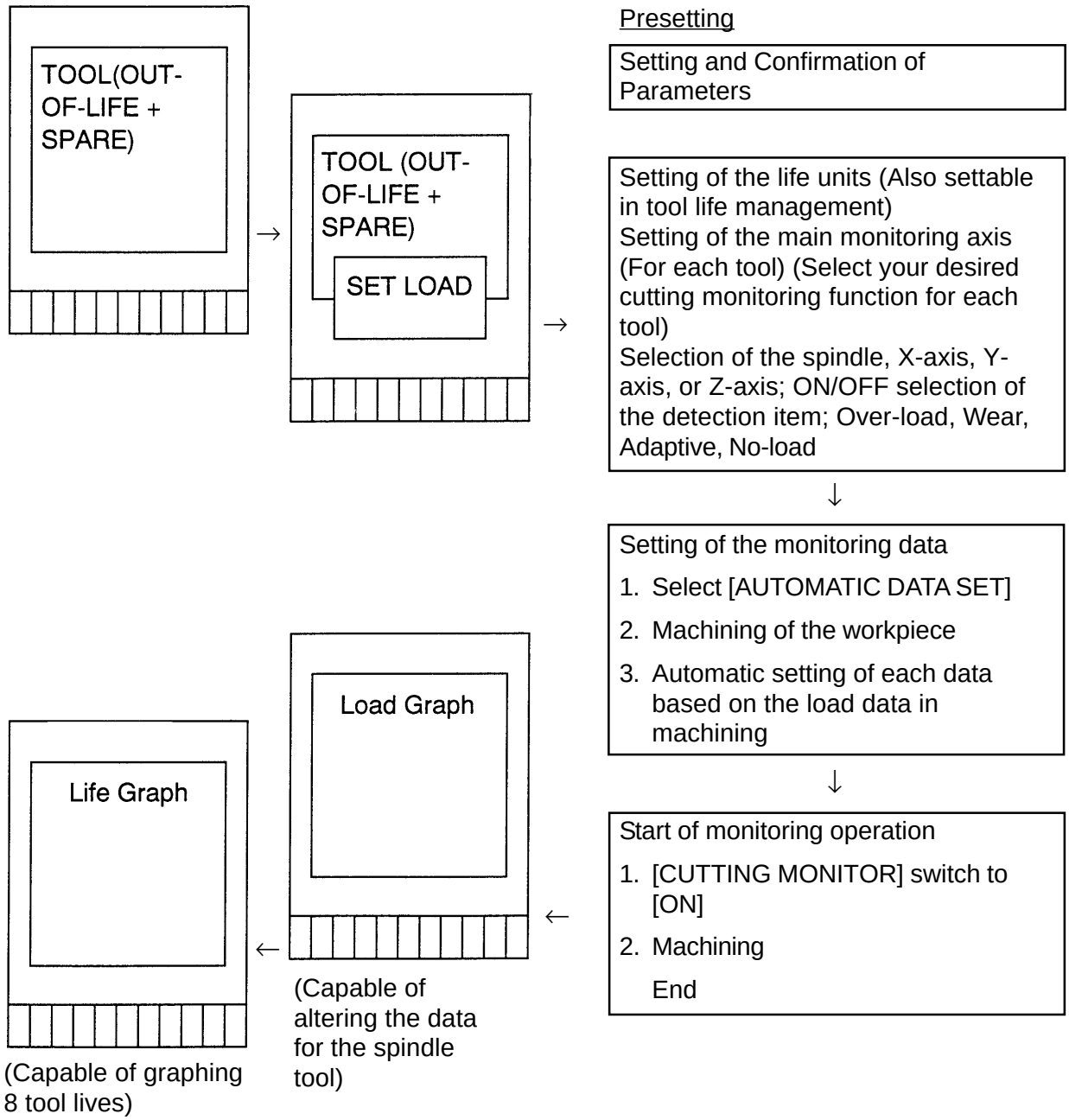
The idle cutting function can be effected before, after, or before and after machining. Select either one of them.



Caution

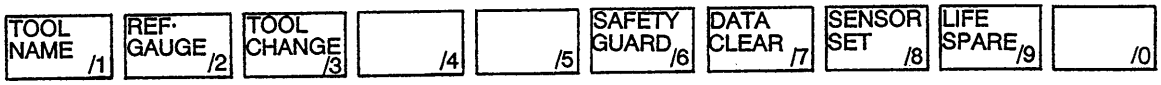
Do not use a tool whose loaded state is difficult to determine.
As an undercut load is expected, select a proper tool.

7. Cutting Monitoring Operational Procedure

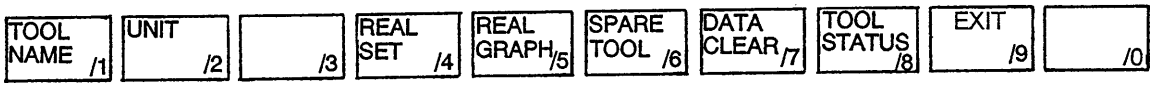


19-7 Set Load Screen

[1] Press F3/TOOL in the OVERALL screen → The TOOL screen appears and displays the following functions.



[2] Press F9/LIFE SPARE → The TOOL LIFE screen appears and displays the following functions.



[3] Press F4/REAL SET → The SET LOAD screen appears.

○ When interrupting operation (turning off the SET LOAD screen), press F4/REAL SET again.

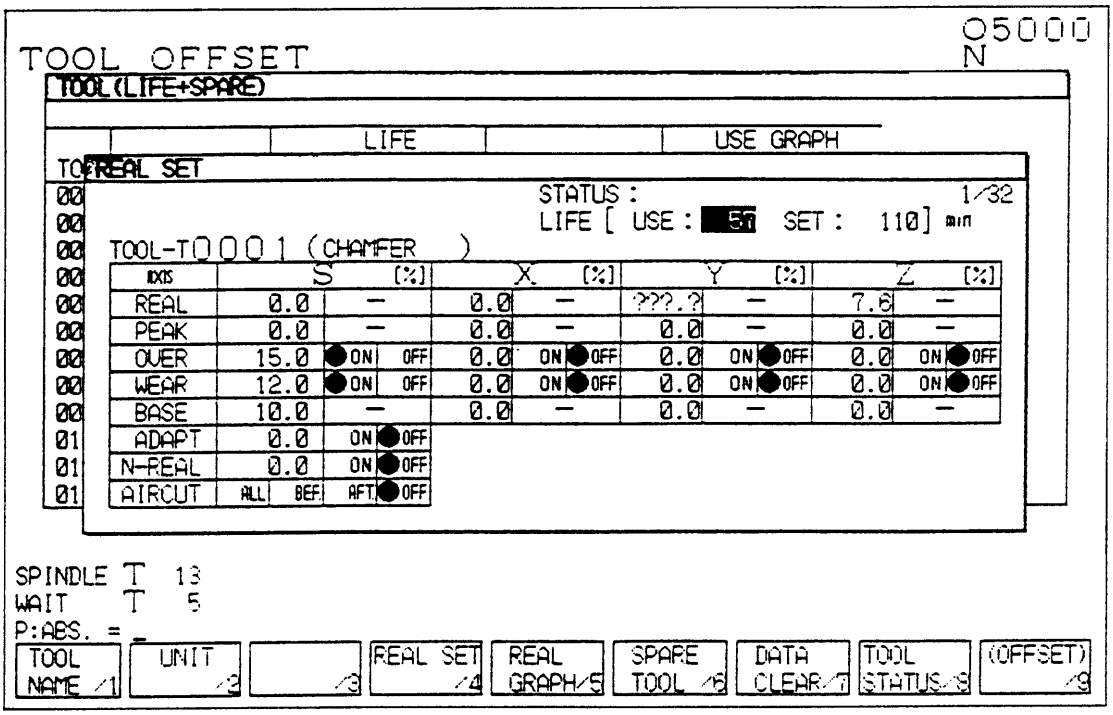


Fig. 5

- Screen selecting method

[1] Press the ⇧ (previous page) key to display the previous TOOL NUMBER screen.

[2] Press the ⇩ (next page) key to display the next TOOL NUMBER screen.

[3] Enter ↓ following by a tool number (numerical value) and press the N key.
Pressing N replaces “P:ABS. =” on the lower left of the screen by “N: TOOL NO. =.”

```
P: ABSOLUTE VALUE=
TOOL NAME /1    UNIT /2
```

```
N: TOOL NO.=
TOOL NAME /1    UNIT /2
```

- Input position selecting method

  Using the cursor move keys, ↑, ↓ , →, and ←, move the cursor to the position where you want to alter or input the data.

1. Selecting the Main Monitoring Axis

Initially, the "S (SPINDLE)" has been set as the main monitoring axis. When you want to change it;

[1] Using the ↓ key, move the cursor to "AXIS."

[2] Select and enter an axis name you want to change to; S , X , Y , or Z .

[3] Press INPUT .

(Note) *Changing the main monitoring axis means to replace it and one of the sub-monitoring axes with each other. The set data are also replaced, but ADAPTIVE, NO-LOAD, and IDLE CUT data are cleared.*

2. Selecting the Monitoring Function

Select for each tool which monitoring functions should be activated for the main monitoring axis and sub-monitoring axis. Set to "● ON" a relevant selector switch for the function you want to select.

- Turning on the function

[1] Using the ↑, ↓ , →, and ← keys, move the cursor to "ON" of your desired monitoring function.

[2] Press INPUT → A dot mark, "●", appears on the "ON" side.

- Turning off the function

[1] Using the ↑, ↓ , →, and ← keys, move the cursor to "OFF" of the monitoring function you want to deselect.

[2] Press INPUT → A dot mark, "●", appears on the "OFF" side.

3. Setting the Data

Set the monitoring data for each tool. There are two kinds of methods available; setting for individual items and setting with a load reference value.

- Setting the individual data

  To set the individual data such as OVERLOAD, WEAR, ADAPTIVE, and NO LOAD, move the cursor to a data setting position, input the setting data, and press INPUT .

- Setting the data with the reference value

  This is a setting method with the load reference value. When a cutting load reference value for that tool is input with the cursor located at REF. VALUE, the monitoring data for the function set to "● ON" is automatically calculated and set according to the parameter setting. The data for the function set to "● OFF" is cleared.

<Example> Calculated load values when the spindle motor's reference value is input

- OVERLOAD ..... (Reference value × Parameter No. 8006)/100
  (X: 8012, Y: 8018, Z: 8024)
- WEAR ..... (Reference value × Parameter No. 8007)/100
  (X: 8013, Y: 8019, Z: 8025)
- ADAPTIVE ..... (Reference value × Parameter No. 8008)/100
  (X: 8014, Y: 8020, Z: 8026)
- NO LOAD ..... (Reference value × Parameter No. 8009)/100
  (X: 8015, Y: 8021, Z: 8027)

4. Deleting the Data

[1] Press F7/DATA CLEAR → The Delete Screen window appears.

[2] Using the ↓ key, move the cursor to a selected item.

[3] Press INPUT → A message, "OK.? Y-YES N-NO," appears.

	DELETE ITEM
	DELETE MAXIMUM VALUE
	DELETE ALL DATA
	DELETE MAXIMUM VALUES OF ALL TOOLS
	DELETE ALL DATA OF ALL TOOLS

[4] Press Y → The Delete Item window disappears and the data are deleted.

○ When interrupting operation (turning off the window), press F7/DATA CLEAR again or press N in [4].

- DELETE MAXIMUM VALUE ..... Deletes the maximum value data of the tool displayed in the screen.
- DELETE ALL DATA ..... Deletes all the monitoring data of the tool displayed in the screen.
- DELETE MAXIMUM VALUES OF ALL TOOLS ..... Deletes the maximum value data of all the tools.
- DELETE ALL DATA OF ALL TOOLS ... Deletes all the monitoring tools of all the tools. When this is done, the main monitoring axis is the spindle and overload detection is set to "ON."

(Note) *When you wan to turn off overload detection with DELETE ALL DATA OF ALL TOOLS selected, change setting of the parameter No. 8001.*

19-8 Load Graph Screen

Press the [CUTTING MONITOR] key on the operation panel or press F5/REAL GRAPH .
The LOAD GRAPH screen appears.

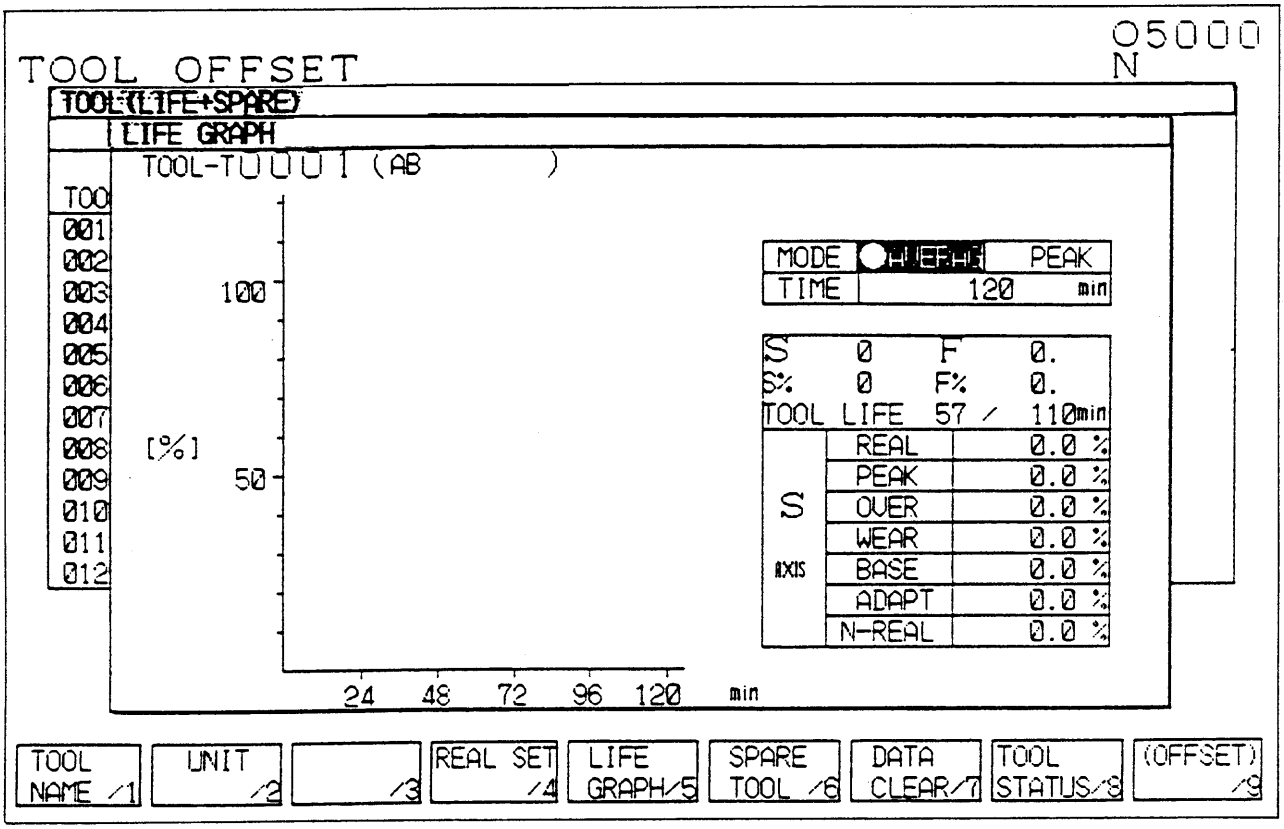


Fig. 6

The LOAD GRAPH screen is a display with respect to the spindle tool and graphs a loaded state after one of the spindle, X-axis, Y-axis, or Z-axis is placed under monitoring.
This screen is displayed when the MONITOR mode is set to ON .

- Display time ..... Set arbitrary data whose display time width is 1 minute or longer.
  Move the cursor to “TIME”, input display time data, and press INPUT.
- Display mode
  AVERAGE ..... Displays an average value of load data per unit time of display.
  PEAK ..... Displays the maximum and minimum values of load data per unit time of display in the form of bar graph.
- Altering the life data and monitoring data
  The life data and monitoring data being displayed in the load graph can be altered through this screen.
  [1] Using the ↑ and ↓ keys, move the cursor to a setting position.
  [2] Input the numerical data and press INPUT.

- Use the Page keys ⇧ and ⇩ to select the display for the spindle, X-axis, Y-axis, or Z-axis.

19-9 Life Graph Screen

Press F5/LIFE GRAPH in the REAL GRAPH screen. The LIFT GRAPH screen appears.

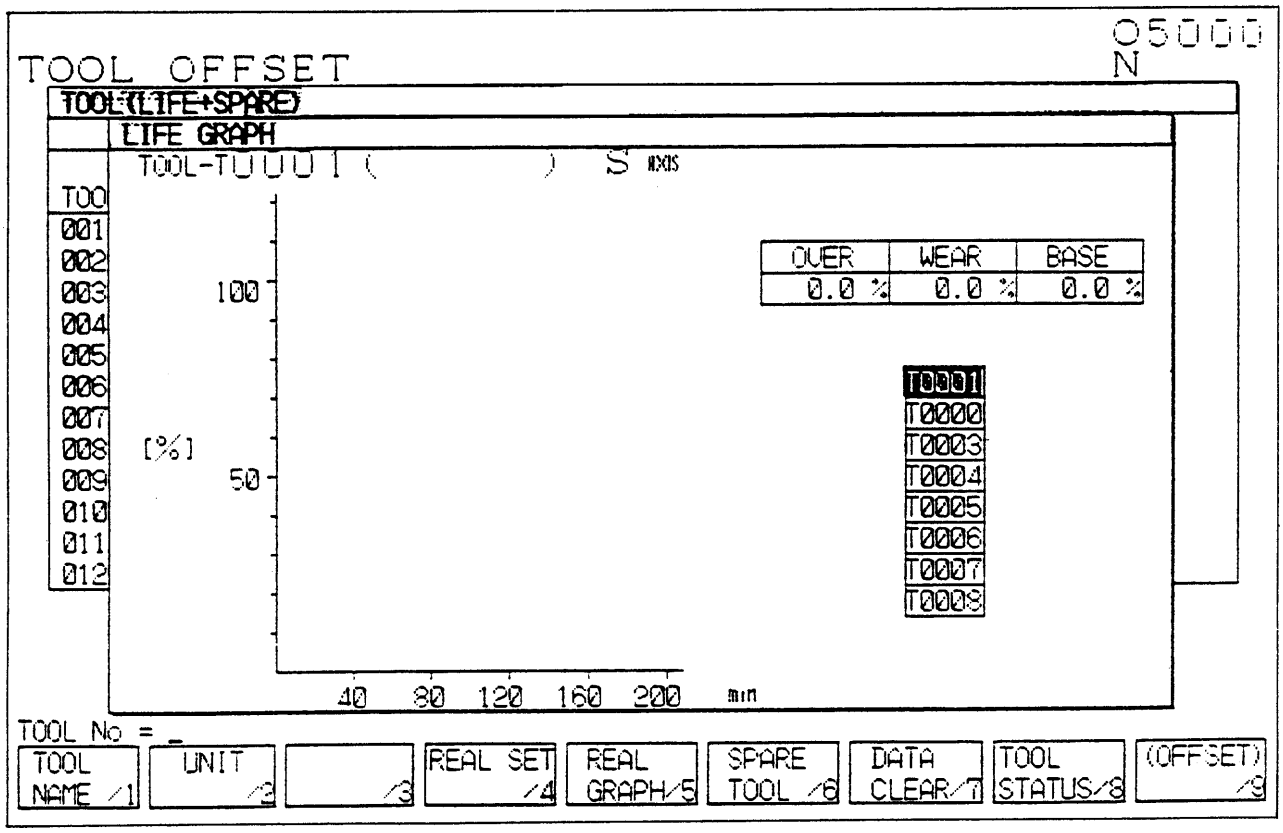


Fig. 7

The LIFE GRAPH screen graphs the cutting load data ranging from a start of using a life managed tool to its point of running out of life. You can see at a glance the loaded state by lapse of time until the tool runs out of life.

This screen can display the life graphs for preregistered 8 tools; it displays the post-registration cutting data. No graph is displayed unless monitoring operation is enabled.

- Selecting the display

  Select the life graph for the tool number you want to display.

  [1] Using the ↑ and ↓ keys, move the cursor to the tool number whose life graph you want to display.

  [2] Press INPUT →The life graph for the selected tool is displayed.

- Setting the tool number

  Set the tool number you want to display in the life graph.

[1] Using the ↑ and ↓ keys, move the cursor to a setting position.
[2] Input the tool number you want to set.
[3] Press INPUT → The registered tool number is set.

(Note) *The life graph collects the cutting data on the eight registered tools.*
If a tool which is halfway its life is registered, the cutting data after that point will be displayed.

19-10 Display and Setting of Parameters

1. Displaying the Parameter

Press F4/SYSTEM and enter 1 , INPUT , and N 8 0 0 0 . Then, use the ↓ key to display the parameter No. 8000 for cutting monitoring.

2. Setting the Parameter

[1] Select the [MDI] mode
[2] Press the [RESET] switch
[3] Turn on the [WRITE] switch
[4] Enable rewriting of parameters (SETTING screen)
[5] Move the cursor to the number you want to set
[6] Input the setting data
[7] Press the INPUT key

3. Selecting the Monitoring Data System (Parameter 8000)

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

- Parameter 8000 (0: Absolute value system 1: Reference value system)

Bit 0 ..... Spindle monitoring reference value system
Bit 1 ..... X-axis monitoring reference value system
Bit 2 ..... Y-axis monitoring reference value system
Bit 3 ..... Z-axis monitoring reference value system

4. Selecting Initial Setting (Parameter 8001)

When you select and execute “DELETE ALL DATA OF ALL TOOLS” with the DATA DELETE function in the SET LOAD screen, the monitoring functions selected with this parameter are set to “ON.”

- Parameter 8001 (0: Not selected, 1: Selected)

Bit 0 ..... Overload of the main monitoring axis
Bit 1 ..... Wear of the main monitoring axis

Bit 2 ..... Adaptive control of the main monitoring axis
Bit 3 .....
Bit 4 ..... Overload of the sub-monitoring axis
Bit 5 ..... Wear of the sub-monitoring axis
Bit 6 ..... Initialization in the tool life times mode
Bit 7 ..... Initialization in the tool life time mode

5. Selecting the Function (Parameter 8002)

- Parameter 8002 (0: Not selected, 1: Selected)

Bit 0 ..... Spindle reverse rotation

6. Displaying the Monitoring Data (Parameter 8004)

- Parameter 8004 (0: Displays the data in percentage (%), 1: Displays the data in ampere (A))

Bit 6 = 1 ..... Displays the monitoring data in ampere (A)

A displayed data amount complies with the parameters No. 8070 to No. 8073.

7. Automatic Calculation Parameters (8006 to 8029)

You set the parameters which automatically calculate the monitoring data (detection values) for the spindle, X-axis, Y-axis, and Z-axis, and those which calculate cutting load reference value.

Unit of Setting Values: %

	Spindle	X-axis	Y-axis	Z-axis
OVERLOAD	8006	8012	8018	8024
WEAR	8007	8013	8019	8025
ADAPTIVE	8008	8014	8020	8026
NO LOAD	8009	8015	8021	8027
AV HIG	8010	8016	8022	8028
AV LOW	8011	8017	8023	8029

<Calculation Example> When the spindle is monitored and a machining load reference value is 40%
An overload is set as follows: 40 × 150 (Parameter 8006)/100 = 60%

(1) Detection value calculation parameters

- Spindle overload value = Key-input reference load (or Calculated load value) × Parameter 8006/100
- X-axis overload value = Key-input reference load (or Calculated load value) × Parameter 8012/100
- Y-axis overload value = Key-input reference load (or Calculated load value) × Parameter 8018/100

- Z-axis overload value = Key-input reference load (or Calculated load value) × Parameter 8024/100
- Wear value = Key-input reference load (or Calculated load value) × Parameter 8007 (8013, 8019, or 8025)/100
- Adaptive value = Key-input reference load (or Calculated load value) × Parameter 8008 (8014, 8020, or 8026)/100
- No-load value = Key-input reference load (or Calculated load value) × Parameter 8009 (8015, 8021, or 8027)/100

(2) Reference load calculation parameters

- Spindle load calculation value = $\frac{\Sigma \text{ Load data}}{\text{Parameter 8010}} / \frac{\Sigma \text{ No. of load data}}{\text{Parameter 8010}}$ / $\frac{\text{Parameter 8011}}{\text{Parameter 8011}}$
- X-axis load calculation value = $\frac{\Sigma \text{ Load data}}{\text{Parameter 8016}} / \frac{\Sigma \text{ No. of load data}}{\text{Parameter 8016}}$ / $\frac{\text{Parameter 8017}}{\text{Parameter 8017}}$
- Y-axis load calculation value = $\frac{\Sigma \text{ Load data}}{\text{Parameter 8022}} / \frac{\Sigma \text{ No. of load data}}{\text{Parameter 8022}}$ / $\frac{\text{Parameter 8023}}{\text{Parameter 8023}}$
- Z-axis load calculation value = $\frac{\Sigma \text{ Load data}}{\text{Parameter 8028}} / \frac{\Sigma \text{ No. of load data}}{\text{Parameter 8028}}$ / $\frac{\text{Parameter 8029}}{\text{Parameter 8029}}$

8. Timer Parameters (8030 to 8044)

You set the timer data used for monitoring. A timer setting increment is 0.1 second.

	Spindle	X-axis	Y-axis	Z-axis
Cancellation time	8030	8034	8038	8042
Overload judgment time	8031	8035	8039	8043
Wear judgment time	8032	8036	8040	8044

- The cancellation time refers to a data invalid time upon starting the spindle, changing a spindle speed, or starting cutting feed.
- The overload judgment time refers to a time required to determine an overload when a cutting load exceeds an overload value.
- The wear judgment time refers to a time required to determine wear when a cutting load exceeds a wear value.

When you need more than one spindle cancellation time, use the following parameters other than 8030.

Low-speed Gear (1st)		High-Speed Gear (2nd)	
Switched rpm for spindle cancellation time	Spindle cancellation time 1-4	Switched rpm for spindle cancellation time	Spindle cancellation time 5-8
	Cancellation time 1 8030		Cancellation time 5 8066
1-2 switching point 8080	2 8063	5-6 switching point 8083	6 8067
2-3 8081	3 8064	6-7 8084	7 8068
3-4 8082	4 8065	7-8 8085	8 8069

(Notes)

- *When none of the 1st through 4th gears has been selected (when the machine has no gear, see the parameter for the low-speed gear.*
- *When the 3rd or 4th gear is selected, see the parameter for the high-speed gear.*
- *Even if the high-speed gear has been selected, see 8030 when 8083 equals 0 and 8066 equals 0.*
- *See 8030 when 8080 equals 0.*

9. Adaptive Parameters

(1) Upper-limit adaptive value (Parameter 8050)
A parameter to set a target load upper-limit value when activating adaptive control.
Upper-limit adaptive value = Adaptive value × Parameter 8050/100

(2) Lower-limit adaptive value (Parameter 8051)
A parameter to set a target load lower-limit value when activating adaptive control.
Lower-limit adaptive value = Adaptive value × Parameter 8051/100

(3) Adaptive maximum override (Parameter 8052)
Set a maximum override for adaptive control to “110 to 200.”

(4) Adaptive minimum override (Parameter 8053)
Set a minimum override for adaptive control to “10 to 90.”

(5) Adaptive override step (Parameter 8054)
Set to “10 to 50” feed rate override change intervals which are controlled by adaptive control.

(6) Adaptive decrease judgment time (Parameter 8055)
When a cutting load exceeds an upper-limit adaptive value in adaptive control, set a time to determine to decrease an override. A setting increment is 0.1 second.

(7) Adaptive decrease judgment time (Parameter 8056)
When a cutting load becomes lower than a lower-limit adaptive value in adaptive control, set a time to determine to increase an override. A setting increment is 0.1 second.

10. Data Parameters

(1) No. of spindle data averaging times (Parameter 8058)
Set to “0 to 40” the number of times to average the spindle load data form the A/D.

(2) No. of feed axis data averaging times (Parameter 8059)
Set to “0 to 40” the number of times to average the feed axis load data from the servo.

(3) Spindle load 100% data (Parameter 8070)

(4) X-axis load 100% data (Parameter 8071)

(5) Y-axis load 100% data (Parameter 8072)

(6) Z-axis load 100% data (Parameter 8073)

19-11 Alarms

770 SPINDLE OVERLOAD ALARM
A spindle load exceeded an overload value during cutting.

771 NO-LOAD ALARM
Since machining is completed without a no-load value being exceeded by a cutting load for the main monitoring axis, a no-load alarm results, judging it erroneous mounting or tool breakage.

772 X-AXIS OVERLOAD ALARM
An X-axis load exceeded an overload value during cutting.

773 Y-AXIS OVERLOAD ALARM
A Y-axis load exceeded an overload value during cutting.

774 Z-AXIS OVERLOAD ALARM
A Z-axis load exceeded an overload value during cutting.

When the above-mentioned alarms take place, the machine stops feeding the axes and stops the spindle in 3 seconds. When manually returning the machine, move each axis to a safe position, check a tool, and replace it if necessary.

VERTICAL MACHINING CENTER

VS50/60

INSTRUCTION MANUAL

OPERATION

SEIKI-SEICOS Σ16M/18M

Version 1.01

9-2000

8-1997 First Edition