CNC SYSTEMS

OSP5020L
OSP500L-G

OPERATION MANUAL (11th Edition)

OKUMA

Publication No. 3272-E-R4
(January 1993)

CNC SYSTEMS

OSP5020L
OSP500L-G

OPERATION MANUAL (11th Edition)

LE32-003-R11

SAFETY PRECAUTIONS

The machine is equipped with a variety of safety devices designed to prevent accidents and injuries from occurring. Still, operators must use the machine carefully and learn not to rely on these safety devices.

This section describes the general safety precautions required for machine operation. Every operator must read this section carefully and observe the guidelines it describes.

1. Before Turning on the Power

(1) Make sure that the doors to the operation panel and the electric control cabinet are closed.

(2) Make sure that there are no obstacles around the machine.

(3) Turn on the main power disconnect switch before turning on the **CONTROL ON** switch on the operation panel.

2. Chuck Precautions

(1) Always close the front shield before starting the spindle or cutting operations.

(2) Always observe the spindle speed maximums for the installed chuck.

Never run the spindle exceeding the maximum allowable spindle speed.

(3) If a chuck or fixture is unique to your application, check the maximum allowable spindle speed and stay within the limit. Also, take note of the workpiece gripping force and balance.

(4) The maximum spindle speed can be limited by inputting a G50 command with the spindle speed (S****). The G50 command helps to ensure safety in operation.

(5) If the spindle must be rotated close to the maximum allowable spindle speed, observe the following points:

- Make sure that the workpiece clamped in the chuck is balanced.
- Apply the allowable maximum amount of pressure to grip the workpiece because centrifugal force reduces the chuck gripping force.

The maximum allowable spindle speed and application pressure are indicated on the name plate on the front shield and on the chuck body. The allowable maximum speed and the applicable pressure ensure a chucking force that is more than one-third over the original chuck gripping force with the standard soft-top jaw set in line with the periphery of the chuck body.

(6) If special jaws (larger than standard soft-top jaws) are used, observe the following points:

- Lower the spindle speed because centrifugal force and lower efficiency reduce the chuck gripping force.
- If the jaw tightening nut (jaw nut) is outside of the periphery of the chuck, only one tightening bolt is holding the jaws in place. This is a very dangerous condition. Jaw nuts must always be within the periphery of the chuck.
- Machine the jaws to the workpiece shape.

(7) Securely tighten the bolts on the chuck body, the jaws, and the block to the specified torque. Use lubricating oil. Make sure that the torque is at least 390 - 490 N [40 - 50 kg (88 - 110 lb)].

3. General Checks

(1) Check the amount of lubricating oil every day before starting operation.

(2) Always use the specified brand of lubricating oil.

(3) Use the recommended type of cutting fluid (coolant) when possible.

(4) Change and replenish the lubricating oil and coolant in each reservoir according to the schedules in the manual.

(5) Clean the filters according to the schedules in the manual.

(6) Make sure that each pressure gauge on the air and hydraulic lines display the correct value as described in this manual.

(7) Always turn off the power before beginning any work inside the front shield. In addition, turn off the power before beginning work at the back of the machine that requires an operator to enter the machine operating zone.

4. Before Starting Operation

(1) Always follow the instructions in the operation manual.

(2) Always operate the machine with all of the protective covers and shields in place.

(3) Always close the front shield before starting operation.

(4) Never attempt to run a new program without checking its operation. Run the program without a workpiece set in the chuck and make sure that there is no interference. After making sure that the program has no bugs, cut a workpiece in the single block mode. If no problems are discovered, automatic operation may be started.

(5) Before attempting the following operations, make sure that they can be accomplished safety.

- Spindle rotation
- Turret indexing
- Axis movement

(6) Never touch chips or the workpiece while the spindle is rotating.

(7) Never attempt to stop a moving object by hand or with a tool.

(8) Check the jaw installation conditions, the hydraulic pressure, and the maximum allowable spindle speed for the power chuck.

(9) Check the installation and arrangement of the tools.

(10) Check the tool offset settings.

(11) Check the zero offset settings.

(12) Make sure that the spindle speed and feedrate override settings are at 100%.

(13) Before feeding the turret, check the software limit setting and the emergency limit LS (limit switch) dog positions for the X-, Y-, and Z-axes.

(14) Check the turret index/rotation position.

(15) Check the tailstock position.

(16) Make sure the cutting operation is within the allowable transmission power and torque ranges.

(17) Make sure that the workpiece securely fitted in the chuck or fixture.

(18) Check the cutting fluid nozzle positions. They must be set to properly supply cutting fluid to the appropriate points.

5. Setup

(1) Make sure that setup is complete.

(2) If the setup is changed, operate the machine step-by-step to make sure that cutting can be performed without any problems.

(3) Before changing the chuck and/or chuck jaws, make sure that the chuck fits the intended jog.

(4) If two or more workers must work together, establish signals so that they can communicate (for example, when lifting or setting heavy objects). Each worker should be aware when a new process is about to begin.

(5) Use the crane or equivalent tool to handle heavy objects.

(6) When attempting an unfamiliar setup, recheck the setup before beginning operation.

6. Workpiece Loading and Unloading

(1) Make sure that workpieces are loaded and unloaded securely.

(2) Before loading or unloading a workpiece, retract the turret so that the cutting tools in the turret cannot injure the operator.

(3) Before loading and unloading a workpiece, make sure that the spindle has come to a complete stop.

(4) Before running a new program, rotate the spindle to make sure that the workpiece is securely clamped in the chuck.

(5) Before machining an irregularly-shaped workpiece, make sure that it is balanced properly.

(6) When handling heavy workpieces, use the a crane, hoist, or other similar tool.

(7) Before loading a workpiece, make sure that the workpiece has a portion that can be used for proper chucking.

7. At the End of the Day

(1) Clean the machine.

(2) Move the turret to the predetermined retraction position.

(3) Make sure all power switches are turned off.

(4) Turn off the **CONTROL ON** switch on the operation panel, before turning off the main power disconnect switch.

8. When a Problem Occurs

(1) Stop the machine immediately by pressing the **EMERGENCY STOP** switch on the operation panel.

(2) Consult with the person in charge of maintenance to determine what corrective measures need to be taken.

(3) If two or more workers must work together, establish signals so that they can communicate (for example, when lifting or setting heavy objects). Each worker should be aware when a new process is about to begin.

(4) Only use specified replacement parts and fuses.

9. General Precautions

(1) Wear appropriate clothing.

(2) Keep the machine and the area around it clean and organized.

(3) Never touch controls or switches with wet hands.

10. Symbols

This manual uses the following symbols to emphasize the items which must be strictly observed:



REMARKS : Precaution in machine operations.

Default of this item affects the machining accuracy and smooth operation.



CAUTION : Default of this item may cause faulty machine operation, machine damage, or operator injury.

More care must be exercised to this item than the items indicated under "REMARKS".



WARNING :

Default of this item may cause not only machine damage but also serious accidents that affect human life.

Sufficient care must be taken to this item.

TABLE OF CONTENTS

PAGE

SAFETY PRECAUTIONS

SECTION 1 OUTLINE 1

SECTION 2 SPECIFICATIONS 2

1. OSP5020L/OSP500L-G Specifications 2

SECTION 3 OPERATION 11

1. Basic Construction of Operation Panels 17

2. Outline of Controls on Operation Panels 20
   2-1. Controls on NC Operation Panel 20
   2-2. Machine Operation Panel 24
   2-3. Setting Keys on Machine Operation Panel 29

3. Fundamental Machine Operation Procedure 32
   3-1. Manual Operations 32

   3-2. Preparation of Machine for Operation 43
      3-2-1. Tool Setting 43
      3-2-2. Setting Zero Offsets 47
      3-2-3. Setting Tool Offsets 59
      3-2-4. Setting Turret Position Error Compensation Value 74
      3-2-5. Automatic Operations 76
      3-2-6. Manual Data Input (MDI) Operation 87
      3-2-7. Interlock Functions 92

SECTION 4 APPLICATIONS 95

1. Application of Operations - Automatic Mode Operation 105
   1-1. Program Selection and Operation 105
   1-2. Sequence Restart 110
   1-3. Sequence Number Search Operation 118
   1-4. Operation Resumption after Manual Operation Intervention 121
   1-5. Other Operations while in Automatic Mode Operation 124
   1-6. Scheduled Operation 127
   1-7. CRT Display while in Operation 133
   1-8. Tool Layout Function 155

PAGE

2. AVAILABLE OPERATIONS IN EDIT AUX MODE 158
2-1. Transfer of Main Program Data 159

2-2. Edition of Main Program Data 175
2-2-1. Fundamental Editing Operation 176
2-2-2. Application of Fundamental Editing Operation 189
2-2-3. Extended Function of Editing 196

2-3. Other Program Operation Functions 215

3. Operation in Parameter Mode 234
3-1. User Parameter 240
3-2. Common Variables 259
3-3. System Parameter 260
3-4. Optional Parameter (Long Word) 266
3-5. Optional Parameter (Word) 277
3-6. Optional Parameter (Bit) 295
3-7. Turret Indexing Angle 326
3-8. Spindle Control Parameter 327

SECTION 1 OUTLINE

The OSP5020L/OSP500L-G is a high-performance numerical control system in which the basic and effective functions available with OSP5000L/5000L-G series controllers of world renown, have been further up-graded. In addition, it features many new functions for easier operation and maintenance.

(1) Improved Operation

The control has a 12-inch CRT (9-inch CRT for OSP500L-G) as a standard feature. Function keys are assigned with respective functions available in the selected operation mode, and the CRT shows the available functions for each function key, thereby eliminating complicated operation procedure. Actual position data, part program data, error messages and other various information are shown on the CRT for correct easy operation.

(2) Improved System Extensibility and Versatility - Never Obsolete

Functions available with OSP5000L/5000L-G series controllers, such as automatic programming functions (LAP/MAP), tool nose radius compensation function,simultaneous 4-axis control and, user task are all extended. The system configuration also permits easy future functional extension through adding software and hardware such as graphic data processing function, and DNC communication. The control is, therefore, never obsolete.

(3) Adoption of Multitask System

With the multitask system, tape editing can be performed during machine operation; down time is minimized.

(4) Highly Reliable and Easy for Maintenance

Both electrical circuit boards and the control circuitry are housed in a single enclosure provided at the back of the machine.

The computer section uses LSIs to minimize the number of component parts; along with the fully enclosed construction, this feature assures high reliability.

Extended self-diagnostic function, with detailed CRT display of diagnosed results. The operator can readily locate the cause of trouble for easy maintenance work.

SECTION 2 SPECIFICATIONS

1. OSP5020L/OSP500L-G Specifications

(1) Basic Functions

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Control	X, Z simultaneous 2-axis control (Simultaneous 4-axis control for 2S*), Linear and circular interpolation * 2S: Not applicable for OSP500L-G.	○	○
Position detection	OSP absolute position encoder (no reference zero return required)	○	○
Tape format	Metric system: N4, G2, X ± 5.3, Z ± 5.3, I ± 5.3, K ± 5.3, F5.3, P4, S4, T4, M2	○	○
	Inch system: N4, G2, X + 4.4, Z + 4.4, I + 4.4, K + 4.4, F4.4, P4, S4, T4, M2		
Tape reader	Photoelectric bi-directional tape reader, 270 characters/sec, Reverse possible	○	○
Programming	Combined use of absolute/incremental programming; ISO (R840) or EIA (RS-244A) codes	○	○
Minimum input increment	Metric system: 1 μm for both X- and Z-axis Inch system: 0.0001 in. for both X- and Z-axis	○	○
Maximum input increment	± 99999.999 mm (8-digit decimal number)	○	○
Input unit setting	"mm", "10 μm" and "1 μm" ("in." and "0.0001 in." for inch system) units can be set as desired by parameter. The decimal point indicates the following units:	○	○
mm unit (in. unit)	mm (in.)	○	○
10 μm unit	10 μm	○	○
1 μm unit (0.0001 in.unit)	1 μm (0.0001 in.)	○	○

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Inch/Metric switchable	Inch or metric system can be set by parameter.	△	△
Decimal point data input	Data with a decimal point can be entered for any input unit system; mm, 10 μm, 1 μm.	○	○
Feed function			
Rapid feedrate	X-axis: Max. 56 m/min Z-axis: Max. 56 m/min with automatic acceleration/deceleration Differs depending on the machine model. See individual machine operation manuals.	○	○
Cutting feedrate	X-axis: Max. 56 m/min Z-axis: Max. 56 m/min F-code mm/rev direct feedrate command Differs depending on the machine model. See individual machine operation manuals.	○	○
Override	Feedrate override from 0 to 200% Used also for jog feedrate change	○	○
Dwell	0.01 to 99999.99 sec	○	○
Tool function			
Tool selection (Depends on machine specifications)	"A" turret: Max. 12 stations, 01 to 12, 1st and 2nd digits following address T "B" turret: Max. 8 stations, 01 to 08, 1st and 2nd digits following address T	○	○
Tool offset selection	32 pairs for "A" and "B" turrets, each Max. offset value: 99999.999 mm	○	○
Automatic tool offset calculation	Tool offset automatically calculated through direct entry of measured values or tool wear amount	○	○

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Spindle DC motor drive Spindle VAC motor drive			
Direct spindle rpm command	4-digit S command	○	○
Constant cutting speed control	Maintaining at specified cutting speeds	○	○
Spindle speed override	In 50 to 200% range	○	○
Max. spindle speed limitation	To set the maximum rpm limit	○	○
Manual functions	Spindle jog/CW/CCW, Tool index, Coolant ON/OFF, Manual X- and Z-axis jog feed, Spindle speed selection, Manual pulse feed handwheel in magnifications (×1, ×10, ×50)	○	○
Miscellaneous functions	Single block, Machine lock, Block delete, Optional stop, Dry run, Independent A/B turret operation (for 2S), Overtravel release, etc.	○	○
Display function			
CRT display	Actual position, Program, Block data, Check, Alarm, Operation guide displayed on screen	○	○
Status indication	Operation conditions monitored by six lamps	○	○
Graphic display	Display of tool path and animated tool movements	△	△
Memory mode operation	Part program data are stored in the memory and machine operation is controlled by the stored program; 30 meter tape length (12,000 characters) [Storage capacity extensible up to 10,240 meter tape length (optional). Up to 3,840 m for OSP500L-G]	○	○
Multitask processing	Tape store, edit, and punchout while machining	○	○
Self-diagnostic function	Program, operation, machine and control system are constantly self-checked.	○	○

(2) Operation Functions

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Program selection	Selects one of the stored programs. Selection by cursor from directory page possible	○	○
Sequence number search	Cursor advances to a specified sequence in the selected program.	○	○
Sequence restart	Restarts from the beginning of an interrupted sequence.	○	○
Mid-auto manual mode & auto restart	Interrupts automatic operation for manual operation. Return to interrupted position automatically.	○	○
Schedule operation	Scheduled sequential running of stored multiple programs	○	○
Data setting	Zero offset, Tool offset, Tool interference barrier (for 2S), Travel limit, Chuck barrier, Droop control, and other data	○	○
Program operation			
Edit function	Screen editor simplifies program editing on the CRT.	○	○
Edit auxiliary function	Tape read/verification, output of part program, sequence No. arrangement, file protection, dating, display of lists, deletion of specified file	○	○
Tape storage capacity	60 m, 160 m, 320 m, 640 m, 1,280 m, 2,560 m, 3,840 m, 5,120 m, 6,400 m, 7,680 m, 8,960 m and 10,240 m (up to 3,840 m for OSP500L-G)	△	△
One part program capacity	60 m, 160 m, 320 m and 640 m	△ *1	△
Interactive color graphic manual data input function (IGF)	Easy to input programs directly from drawings in front of the controller	△	△

*1: 60 m for standard

(3) Programming Functions

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Mirror image	Functions to simplify programming for 2-turret models	○	○
Programmed feedrate command	mm/min (IPM) or mm/rev (IPR) selection by G codes	○	○
Programmed zero offset command	Zero offset by G codes	○	○
Arc radius direct program function	Circular interpolation by commanding radius (L) and arc end point (X/Z)	○	○
Taper angle direct program function	Linear interpolation by commanding the angle from the Z-axis and the endpoint (X or Z)	○	○
Automatic chamfering	Chamfering (straight or arc) by simple commands	○	○
Thread cutting function			
Thread lead	0.001 to 1000.000 mm (Thread lead of less than 1 μm is programmable.)	○	○
Command of number of threads	Specify number of threads by J codes. Actual thread lead is F/J. J word other than integer is programmable.	○	○
Fixed thread cutting cycle	G33: Longitudinal fixed thread cutting	○	○
	G32: Transverse fixed thread cutting		
	Straight, taper and variable lead thread cutting		
	M23: Chamfering ON M22: Chamfering OFF (Chamfering amount programmable)		
	Shift of thread cutting starting point possible		
	Feed hold during thread cutting		
Non-fixed thread cutting cycle	G34, G35	○	○
Special fixed cycle			
Thread cutting cycle	Creates several G32 or G33 paths in one block.	○	○
Grooving cycle	Grooving cycle program in one sequence	○	○
Drilling cycle	Drilling cycle program in one sequence	○	○

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Tool nose radius compensation 2B	Automatic correction of tool nose radius errors on any straight/curved cuts	○	△
Lathe auto-programming function (LAP4)	Allows roughing and finishing cycles in bar and copy turning from final workpiece dimensions.	△	△
	Enables time-saving cutting by designating the blank material shape.		
	Allows both longitudinal and face cutting.		
	Allows cutting condition changes during roughing.		
	Allows various pattern of thread cutting cycles: cutting on one side of the tool, zigzag infeeding, constant stock removal.		
User task 1	For GOTO and IF statements, arithmetic operations, extensive address characters, common variables, local variables and system variables	○	○
User task 2			
Subprograms	CALL, RTS, MODIN and MODOUT statements	△	△
Math-function	Trigonometric functions and logical operations	△	△
I/O variables	Variables related with input/output usable	△	△
READ/WRITE statement	Communication possible from part program to external device using RS232C interface	△	△
Automatic any-angle chamfering	Chamfering on corners or edges having an arbitrary angle possible	○	△
Circular thread cutting	Thread cutting conducted along an arc	△	△

(4) Automation, Peripheral Functions

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Data input/output functions			
Tape punch/print interface (cable length ... 3 m)	FACIT punch (FACIT interface)	△	△
	FACIT punch (RS232C)	△	△
	Okuma punch/printer PP-5001, PP-5002 (RS232C)	△	△
	Citizen protyper 7652 (RS232C)	△	△
	Casio typuter 650NC (RS232C)	△	△
	Tanaka business PT-30RS, PT-30RP (RS232C) FANUC-PPR (RS232C) Kyoritsu KTP8050, KTP8250 (RS232C) JBM PR-30 (RS232C)	△	△
Tape punch/printer	FACIT punch (FACIT interface)	△	△
	FACIT punch (RS232C)	△	△
	Citizen protyper 7652 (RS232C)	△	△
	Casio typuter 650NC (RS232C)	△	△
Portable floppy disk drive	Floppy disk drive interface	△	△
	Cable connector (RS232C)		
	Floppy disk drive (8" disk)		
3.5" floppy disk		△	--
Automation, peripheral functions (1)			
Automatic chuck open/close	Automatic chuck open/close control by M codes (with chuck gripping confirmation)	△	△
Chuck high/low clamp pressure selection	Chucking pressure selection, high/low, by M codes	△	△
Automatic tailstock spindle advance/ retract	Automatic tailstock spindle advance/retract control by M codes (with confirmation)	△	△
Tailstock spindle high/low thrust selection	Tailstock spindle high/low thrust selection control by M codes	△	△
Automatic front door open/close	Automatic front door open/close control by M codes	△	△
Air blower function	Air blow control by M codes To chuck (and tailstock quill)	△	△

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Automation, peripheral functions (1)			
Operation end lamp	Turns on when M02, M00, or M01 is executed.	△	△
Alarm lamp	Turns on when alarm takes place.	△	△
Hour meter	Accumulates time in which the NC has been operated in the M03/M04 mode (spindle rotation).	△	△
Work counter	Number of M02 code execution is counted. (Cycle stop possible at a predetermined number)	△	△
Tool life management	Automatic tool indexing by counted workpieces or cutting time	△	△
Overload detection	Overload condition is detected using the X- and Z-axis drive motor current. At the detection of overload, the control is alarm-stopped.	△	△
Spindle orientation	Spindle orientation by M code (Pin-type, brake-type, electric-type)	△	△
NC operation monitor	Accumulation of cutting hour, running hour, spindle running; NC work count	△	△
Automatic index of index chuck	By M codes	△	△
Load monitor	Monitors spindle, feed axis, and rotary tool axis	△	△
Cycle time over check	Checks if machining is completed in pre-determined time.	△	△

○ : Standard
△ : Optional

Item	Description	OSP 5020L	OSP 500L-G
Automation, peripheral functions (2)			
Functions to meet users' needs and machine spec.	Cycle time reduction function: Simultaneous initiation of axis motion with spindle start/stop Simultaneous initiation of axis motion with turret index	△	△
	Programmable tailstock	△	△
	Lubrication monitor	△	△
	Bar feeder interface	△	△
	Loader interface	△	△
	Robot interface	△	△
	Work gauging function	△	△
	Tool gauging function	△	△
	DNC hook-up (DNC-A, DNC-B, DNC-C)	△	△

SECTION 3 OPERATION

Prior to the detailed explanation on respective operations, the section contents is outlined as follows:

Machine operation procedures are classified into three categories: "Manual Operation", "MDI Operation", and "Automatic Operation".

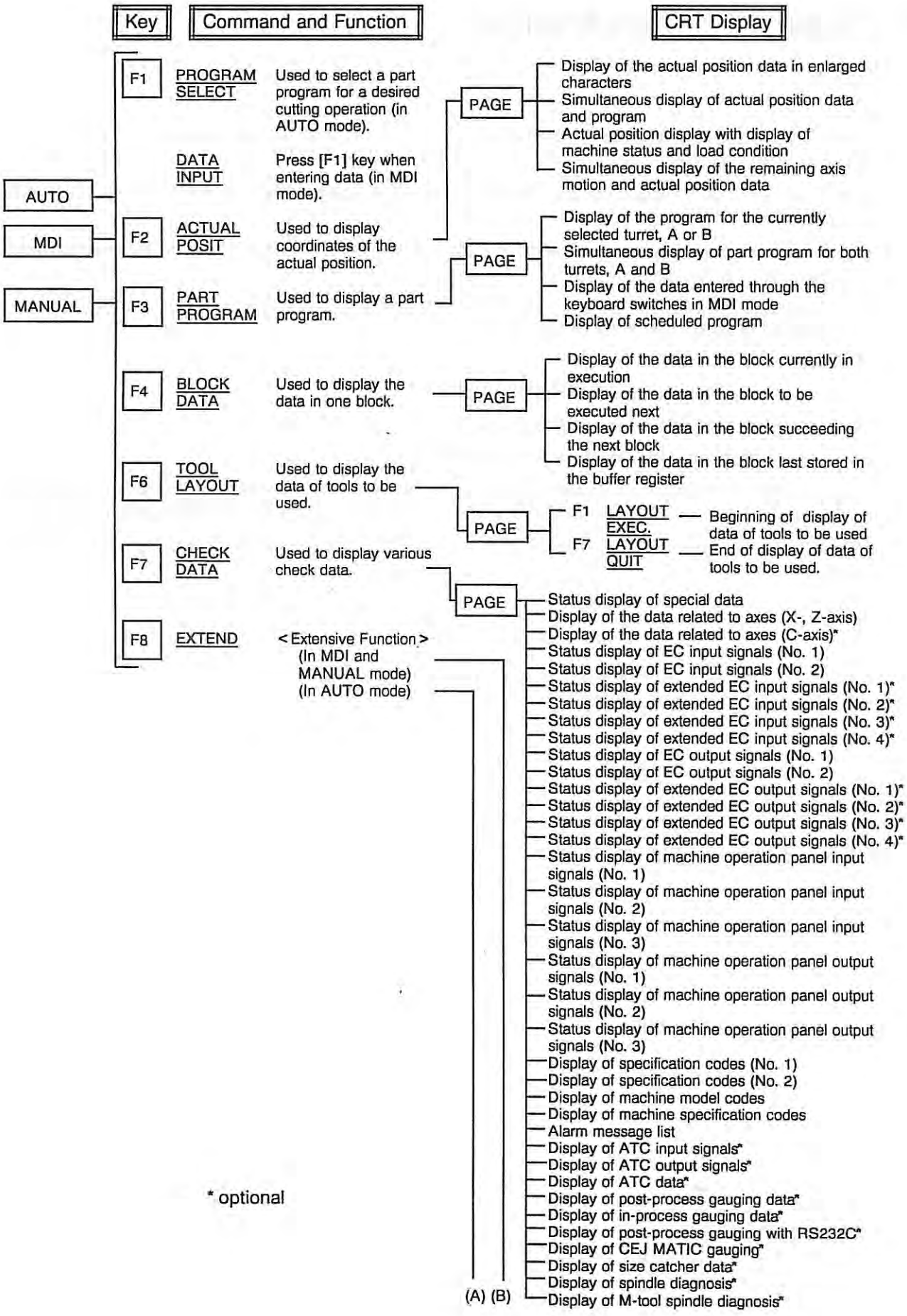
As preliminary steps for machine operations, reference zero setting, tool data setting, programming and parameter setting are necessary.

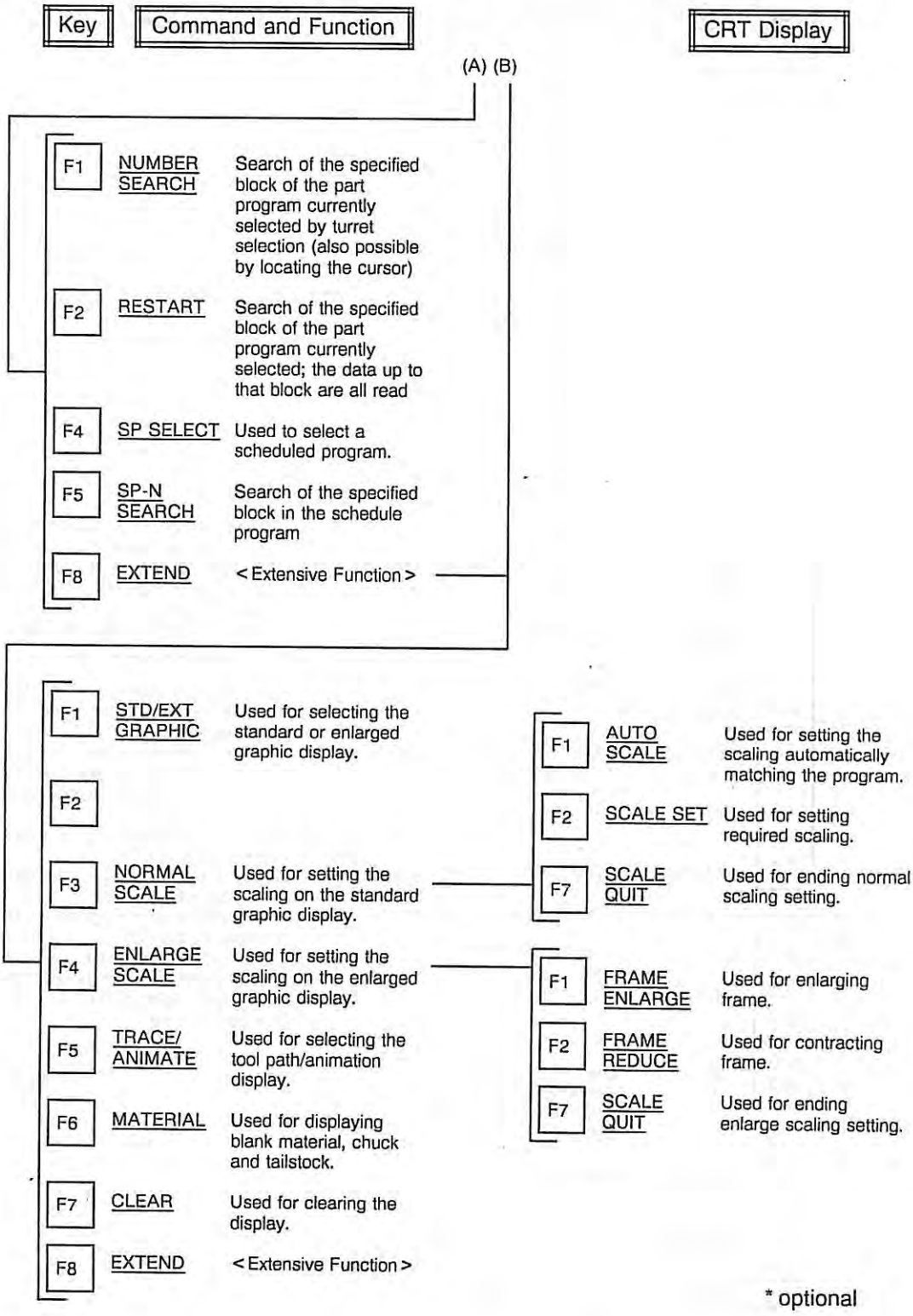
Each operation includes various procedures. A description of these might complicate the explanation. The following order has been adopted to provide you with a clear concept of the operation.

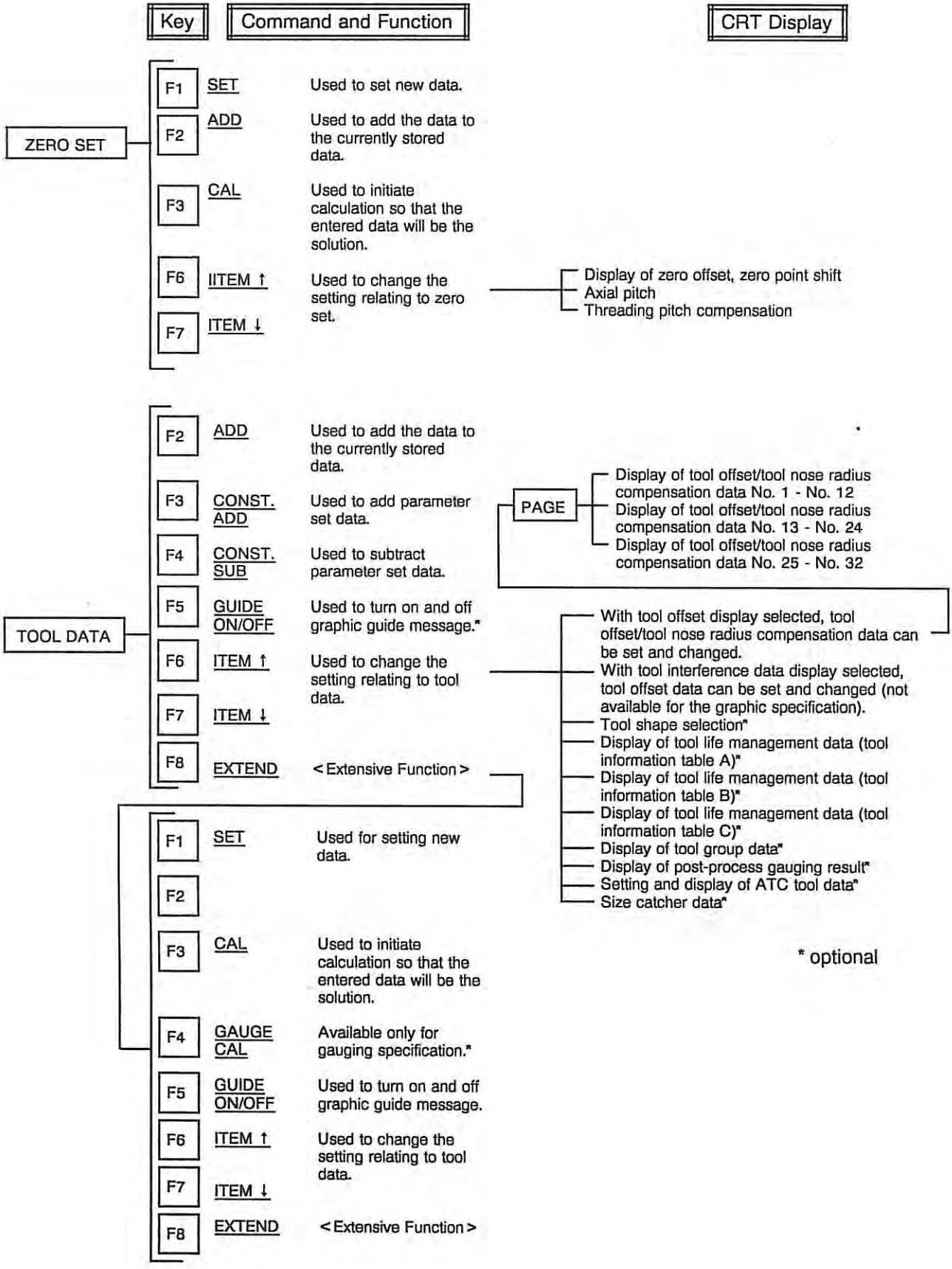
- Basic Construction of Operation Panels
- Outline of Controls on Operation Panels
- Fundamental Machine Operation Procedure

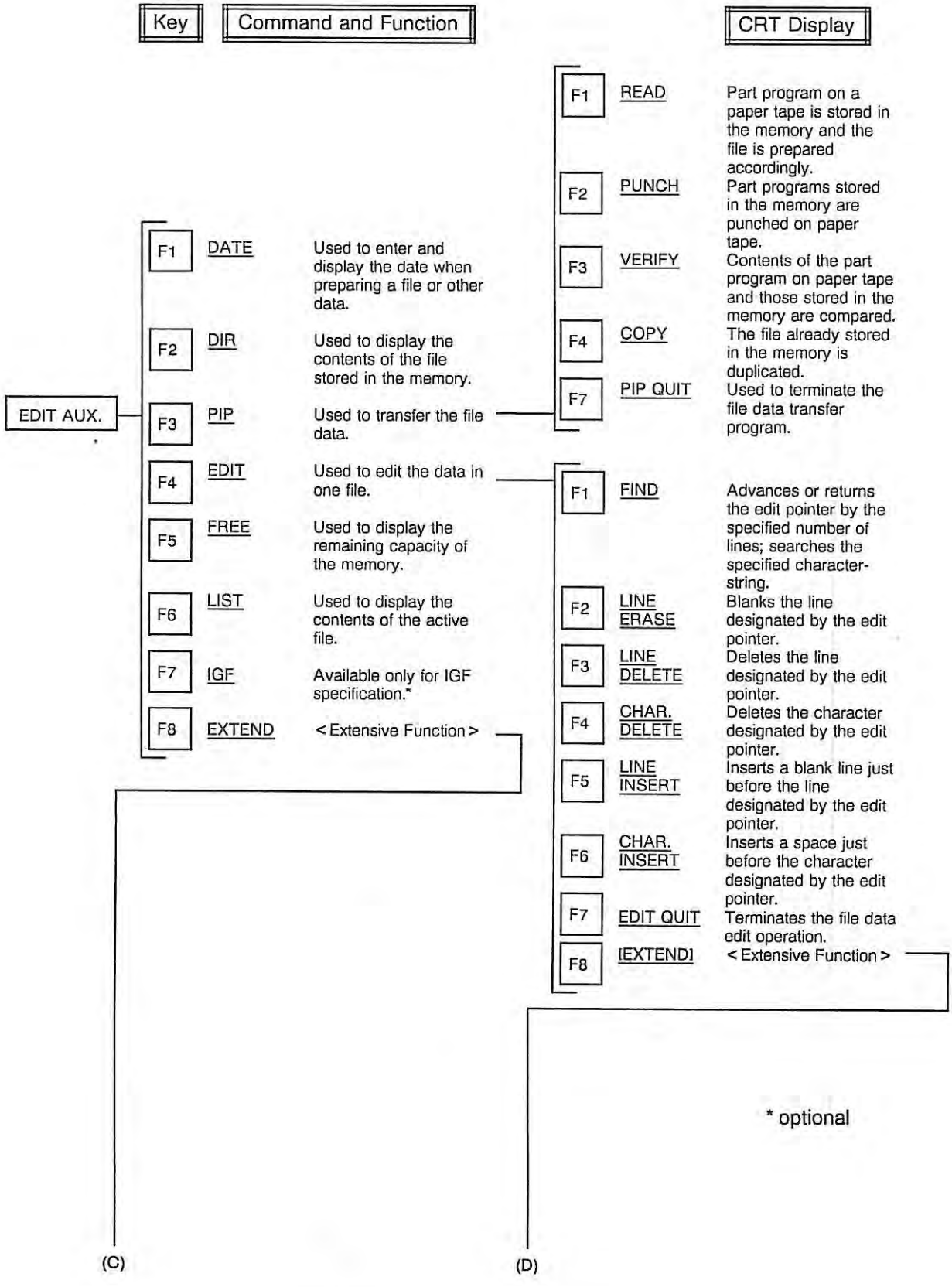
This section is intended to provide familiarization with the fundamental functions and operation procedures of the control. Section 4 is reserved for advanced operations and detailed information.

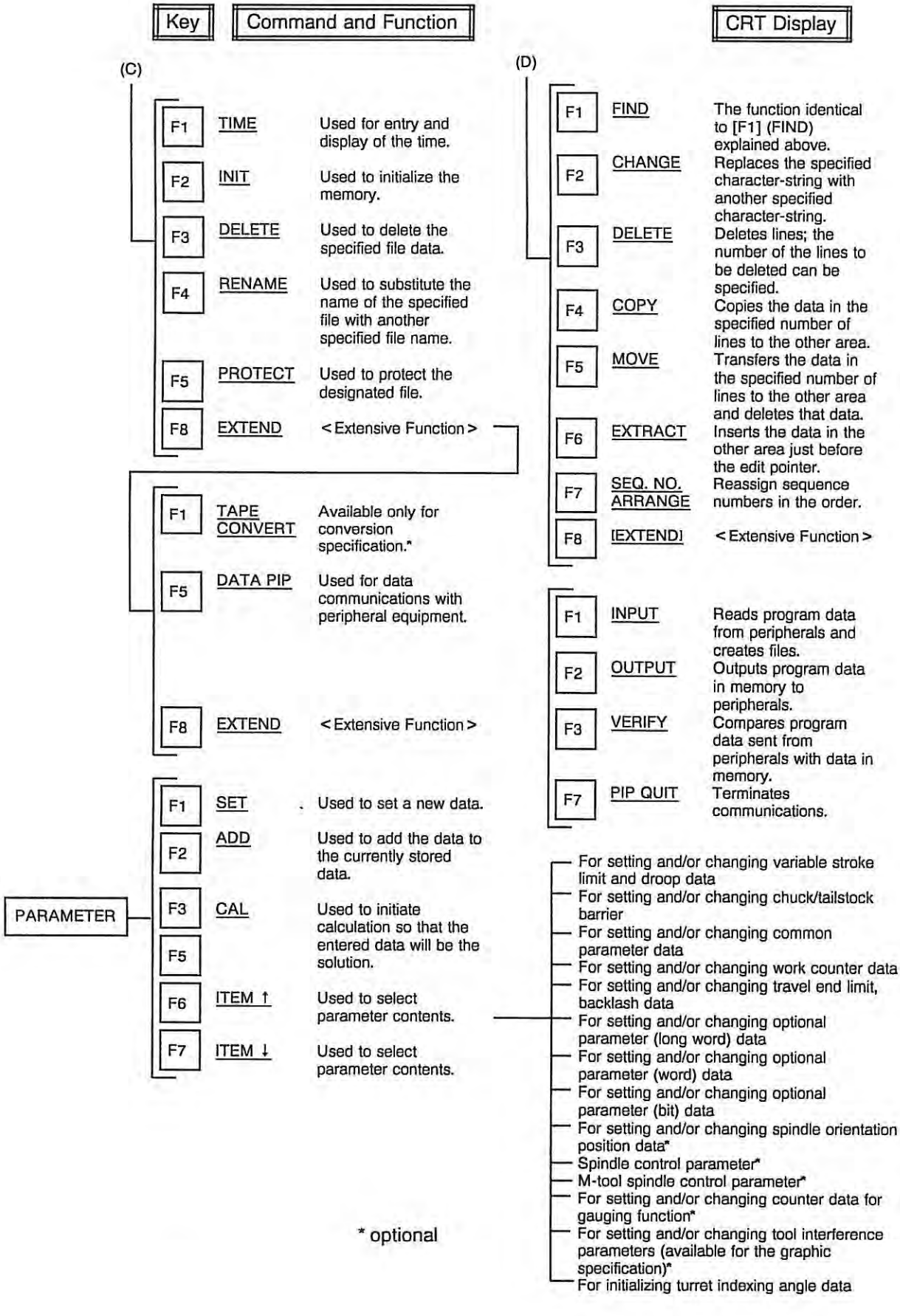
An outline of key operations to access to the desired operation mode is provided on the following pages.


Key
Command and Function
CRT Display
AUTO
MDI
MANUAL
F1
PROGRAM SELECT
Used to select a part program for a desired cutting operation (in AUTO mode).
DATA INPUT
Press [F1] key when entering data (in MDI mode).
F2
ACTUAL POSIT
Used to display coordinates of the actual position.
PAGE
Display of the actual position data in enlarged characters
Simultaneous display of actual position data and program
Actual position display with display of machine status and load condition
Simultaneous display of the remaining axis motion and actual position data
F3
PART PROGRAM
Used to display a part program.
PAGE
Display of the program for the currently selected turret, A or B
Simultaneous display of part program for both turrets, A and B
Display of the data entered through the keyboard switches in MDI mode
Display of scheduled program
F4
BLOCK DATA
Used to display the data in one block.
PAGE
Display of the data in the block currently in execution
Display of the data in the block to be executed next
Display of the data in the block succeeding the next block
Display of the data in the block last stored in the buffer register
F6
TOOL LAYOUT
Used to display the data of tools to be used.
PAGE
F1 LAYOUT EXEC. — Beginning of display of data of tools to be used
F7 LAYOUT QUIT — End of display of data of tools to be used.
F7
CHECK DATA
Used to display various check data.
PAGE
Status display of special data
Display of the data related to axes (X-, Z-axis)
Display of the data related to axes (C-axis)*
Status display of EC input signals (No. 1)
Status display of EC input signals (No. 2)
Status display of extended EC input signals (No. 1)*
Status display of extended EC input signals (No. 2)*
Status display of extended EC input signals (No. 3)*
Status display of extended EC input signals (No. 4)*
Status display of EC output signals (No. 1)
Status display of EC output signals (No. 2)
Status display of extended EC output signals (No. 1)*
Status display of extended EC output signals (No. 2)*
Status display of extended EC output signals (No. 3)*
Status display of extended EC output signals (No. 4)*
Status display of machine operation panel input signals (No. 1)
Status display of machine operation panel input signals (No. 2)
Status display of machine operation panel input signals (No. 3)
Status display of machine operation panel output signals (No. 1)
Status display of machine operation panel output signals (No. 2)
Status display of machine operation panel output signals (No. 3)
Display of specification codes (No. 1)
Display of specification codes (No. 2)
Display of machine model codes
Display of machine specification codes
Alarm message list
Display of ATC input signals*
Display of ATC output signals*
Display of ATC data*
Display of post-process gauging data*
Display of in-process gauging data*
Display of post-process gauging with RS232C*
Display of CEJ MATIC gauging*
Display of size catcher data*
Display of spindle diagnosis*
Display of M-tool spindle diagnosis*
F8
EXTEND
< Extensive Function >
(In MDI and MANUAL mode)
(In AUTO mode)
(A) (B)
* optional


Key
Command and Function
CRT Display
(A) (B)
F1 NUMBER SEARCH
Search of the specified block of the part program currently selected by turret selection (also possible by locating the cursor)
F2 RESTART
Search of the specified block of the part program currently selected; the data up to that block are all read
F4 SP SELECT
Used to select a scheduled program.
F5 SP-N SEARCH
Search of the specified block in the schedule program
F8 EXTEND
< Extensive Function >
F1 STD/EXT GRAPHIC
Used for selecting the standard or enlarged graphic display.
F2
F3 NORMAL SCALE
Used for setting the scaling on the standard graphic display.
F1 AUTO SCALE
Used for setting the scaling automatically matching the program.
F2 SCALE SET
Used for setting required scaling.
F7 SCALE QUIT
Used for ending normal scaling setting.
F4 ENLARGE SCALE
Used for setting the scaling on the enlarged graphic display.
F1 FRAME ENLARGE
Used for enlarging frame.
F2 FRAME REDUCE
Used for contracting frame.
F7 SCALE QUIT
Used for ending enlarge scaling setting.
F5 TRACE/ ANIMATE
Used for selecting the tool path/animation display.
F6 MATERIAL
Used for displaying blank material, chuck and tailstock.
F7 CLEAR
Used for clearing the display.
F8 EXTEND
< Extensive Function >
* optional


Key
Command and Function
CRT Display
ZERO SET
F1 SET
Used to set new data.
F2 ADD
Used to add the data to the currently stored data.
F3 CAL
Used to initiate calculation so that the entered data will be the solution.
F6 IITEM ↑
F7 ITEM ↓
Used to change the setting relating to zero set.
Display of zero offset, zero point shift
Axial pitch
Threading pitch compensation
TOOL DATA
F2 ADD
Used to add the data to the currently stored data.
F3 CONST. ADD
Used to add parameter set data.
F4 CONST. SUB
Used to subtract parameter set data.
F5 GUIDE ON/OFF
Used to turn on and off graphic guide message.*
F6 ITEM ↑
F7 ITEM ↓
Used to change the setting relating to tool data.
F8 EXTEND
< Extensive Function >
PAGE
Display of tool offset/tool nose radius compensation data No. 1 - No. 12
Display of tool offset/tool nose radius compensation data No. 13 - No. 24
Display of tool offset/tool nose radius compensation data No. 25 - No. 32
With tool offset display selected, tool offset/tool nose radius compensation data can be set and changed.
With tool interference data display selected, tool offset data can be set and changed (not available for the graphic specification).
Tool shape selection*
Display of tool life management data (tool information table A)*
Display of tool life management data (tool information table B)*
Display of tool life management data (tool information table C)*
Display of tool group data*
Display of post-process gauging result*
Setting and display of ATC tool data*
Size catcher data*
F1 SET
Used for setting new data.
F2
F3 CAL
Used to initiate calculation so that the entered data will be the solution.
F4 GAUGE CAL
Available only for gauging specification.*
F5 GUIDE ON/OFF
Used to turn on and off graphic guide message.
F6 ITEM ↑
F7 ITEM ↓
Used to change the setting relating to tool data.
F8 EXTEND
< Extensive Function >
* optional


Key
Command and Function
CRT Display
EDIT AUX.
F1 DATE
Used to enter and display the date when preparing a file or other data.
F2 DIR
Used to display the contents of the file stored in the memory.
F3 PIP
Used to transfer the file data.
F4 EDIT
Used to edit the data in one file.
F5 FREE
Used to display the remaining capacity of the memory.
F6 LIST
Used to display the contents of the active file.
F7 IGF
Available only for IGF specification.*
F8 EXTEND
< Extensive Function >
F1 READ
Part program on a paper tape is stored in the memory and the file is prepared accordingly.
F2 PUNCH
Part programs stored in the memory are punched on paper tape.
F3 VERIFY
Contents of the part program on paper tape and those stored in the memory are compared.
F4 COPY
The file already stored in the memory is duplicated.
F7 PIP QUIT
Used to terminate the file data transfer program.
F1 FIND
Advances or returns the edit pointer by the specified number of lines; searches the specified character-string.
F2 LINE ERASE
Blanks the line designated by the edit pointer.
F3 LINE DELETE
Deletes the line designated by the edit pointer.
F4 CHAR. DELETE
Deletes the character designated by the edit pointer.
F5 LINE INSERT
Inserts a blank line just before the line designated by the edit pointer.
F6 CHAR. INSERT
Inserts a space just before the character designated by the edit pointer.
F7 EDIT QUIT
Terminates the file data edit operation.
F8 [EXTEND]
< Extensive Function >
(C)
(D)
* optional


Key
Command and Function
CRT Display
(C)
F1 TIME Used for entry and display of the time.
F2 INIT Used to initialize the memory.
F3 DELETE Used to delete the specified file data.
F4 RENAME Used to substitute the name of the specified file with another specified file name.
F5 PROTECT Used to protect the designated file.
F8 EXTEND <Extensive Function>
F1 TAPE CONVERT Available only for conversion specification.*
F5 DATA PIP Used for data communications with peripheral equipment.
F8 EXTEND <Extensive Function>
PARAMETER
F1 SET Used to set a new data.
F2 ADD Used to add the data to the currently stored data.
F3 CAL Used to initiate calculation so that the entered data will be the solution.
F5
F6 ITEM ↑ Used to select parameter contents.
F7 ITEM ↓ Used to select parameter contents.
(D)
F1 FIND The function identical to [F1] (FIND) explained above.
F2 CHANGE Replaces the specified character-string with another specified character-string.
F3 DELETE Deletes lines; the number of the lines to be deleted can be specified.
F4 COPY Copies the data in the specified number of lines to the other area.
F5 MOVE Transfers the data in the specified number of lines to the other area and deletes that data.
F6 EXTRACT Inserts the data in the other area just before the edit pointer.
F7 SEQ. NO. ARRANGE Reassign sequence numbers in the order.
F8 [EXTEND] <Extensive Function>
F1 INPUT Reads program data from peripherals and creates files.
F2 OUTPUT Outputs program data in memory to peripherals.
F3 VERIFY Compares program data sent from peripherals with data in memory.
F7 PIP QUIT Terminates communications.
For setting and/or changing variable stroke limit and droop data
For setting and/or changing chuck/tailstock barrier
For setting and/or changing common parameter data
For setting and/or changing work counter data
For setting and/or changing travel end limit, backlash data
For setting and/or changing optional parameter (long word) data
For setting and/or changing optional parameter (word) data
For setting and/or changing optional parameter (bit) data
For setting and/or changing spindle orientation position data*
Spindle control parameter*
M-tool spindle control parameter*
For setting and/or changing counter data for gauging function*
For setting and/or changing tool interference parameters (available for the graphic specification)*
For initializing turret indexing angle data
* optional

1. Basic Construction of Operation Panels

The operation panel mainly consists of three panels:

(1) NC Operation Panel (Common Panel)

The NC operation panel is used on all models.

Equipped with a 12-inch CRT (9-inch CRT for OSP500L-G) it is used for all NC operations other than manual.

(2) Machine Operation Panel

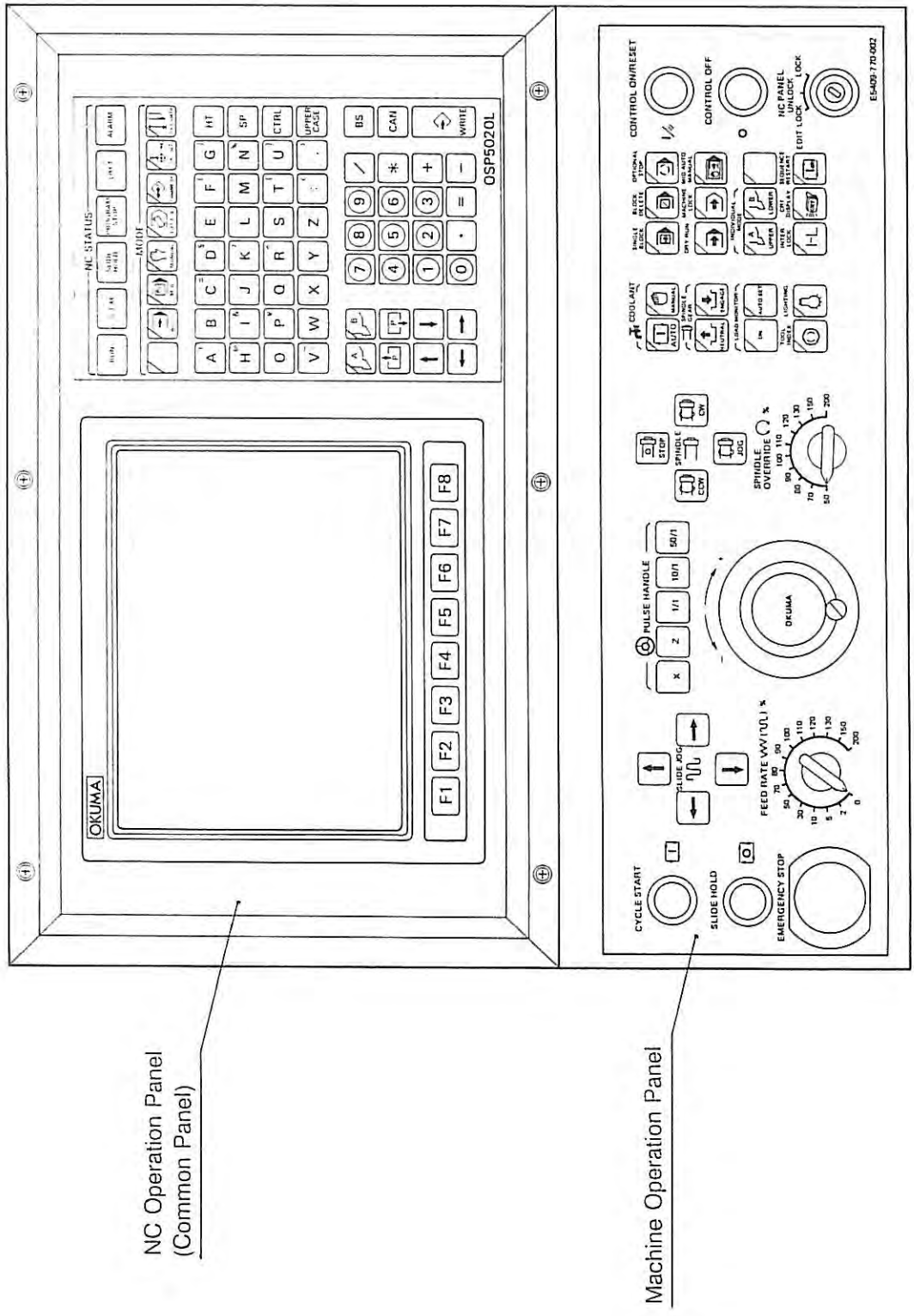
The machine operation panel, made for each respective machine model, mainly holds the controls for manual operation. This panel varies from model to model depending on the available functions.

(3) Option Panel

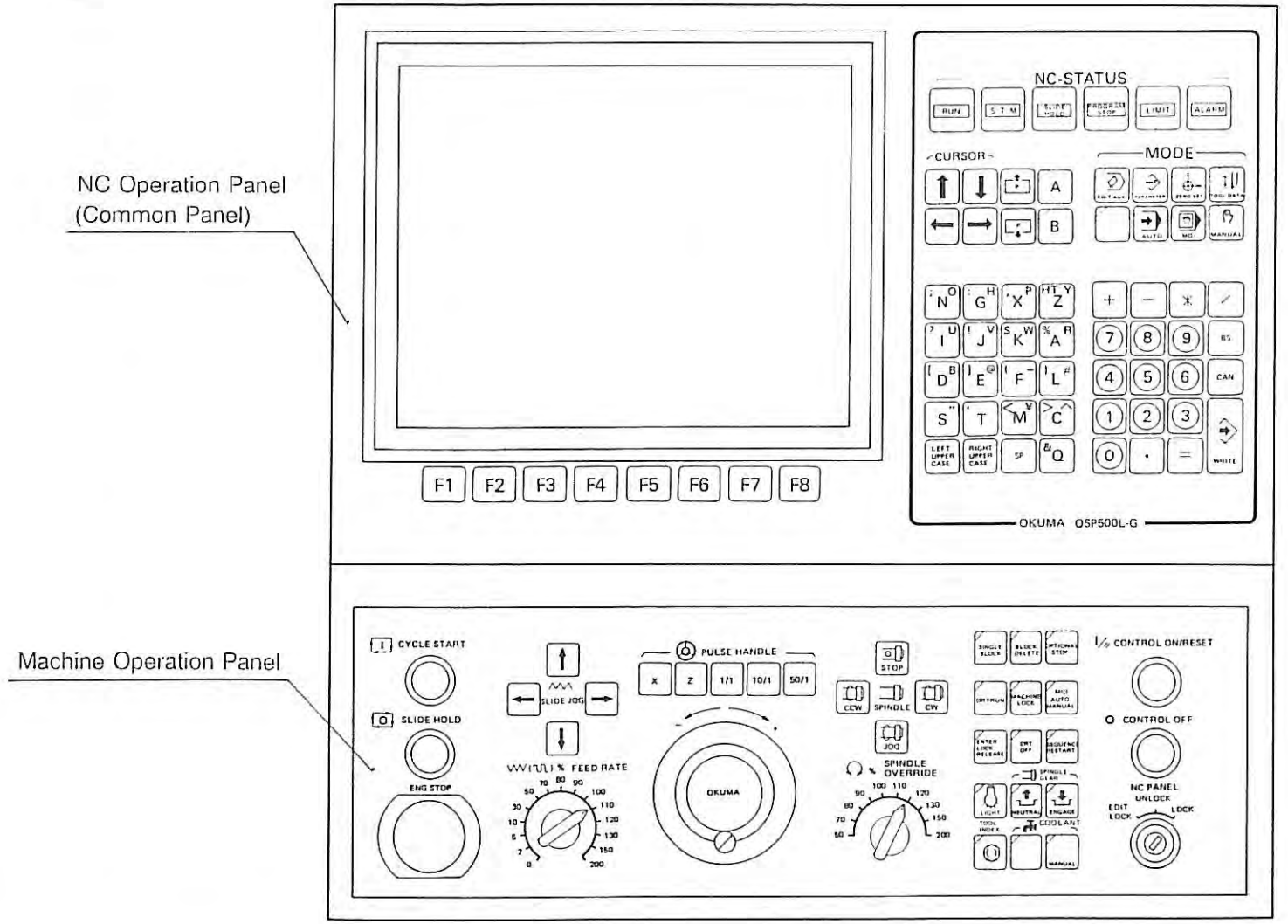
Provided on the option panel are switches and lamps related with the optional functions selected.

External view of these operation panels, arranged on one unit, are shown on the following page.

Note: *Further explanations are included with the drawing of the OSP5020L NC operation panel. The key layout on the OSP500L-G is different, but operators should press the corresponding keys.*



Operation Panel - OSP5020L



Operation Panel - OSP500L-G

2. Outline of Controls on Operation Panels

In this section, an outline of the controls on the NC operation panel and the machine operation panel is provided.

2-1. Controls on NC Operation Panel

(1) **MANUAL**

Press this **MANUAL** key when operating the machine manually, i.e., when the machine operation through the controls on the machine operation panel is required.

(2) **MDI** (Manual Data Input)

Press this **MDI** key when operating the machine with the commands entered through the keyboard (MDI mode operation).

(3) **AUTO**

Press this **AUTO** key when operating the machine in the automatic mode by the stored part program in automatic mode.

(4) **TOOL DATA**

Press this **TOOL DATA** key when setting, modifying, or checking tool offset data and tool nose radius data. (Up to 32 pairs of data can be stored in the memory.)

(5) **ZERO SET**

Press this **ZERO SET** key when setting or modifying the reference point of the machining (reference zero), or checking the stored coordinate values of such position.

(6) **PARAMETER**

Press this **PARAMETER** key when setting, modifying, or checking parameters:

Travel end, variable soft-limit, backlash, chuck barrier, droop, spindle jog speed, etc.

(7) **EDIT AUX**

Press this **EDIT AUX** key when editing or inputting/outputting the stored part program, or when checking contents of the program.

(8) NC STATUS indicating lamps

a) RUN

The RUN indicating lamp goes on when the machine is normally running in the AUTO or MDI mode.

b) S.T.M

The S.T.M indicating lamp goes on when a command, other than that calling for axis motion, such as spindle speed range change, tool change, spindle rotation/stop, is executed.

When an axis motion command is designated in a block containing S, T and/or M command, the axis motion command is executed after the execution of S, T, and M commands. If the optional cycle time reduction function is active, it is executed simultaneously with them.

When a spindle speed range selection command, spindle speed command, or tool number command is changed during automatic mode operation with the manual operation intervention function activated, the S.T.M indicating lamp flickers.

c) SLIDE HOLD

The SLIDE HOLD indicating lamp goes on when the **SLIDE HOLD** button on the machine operation panel is pressed in the AUTO or MDI mode.

This lamp will also come on when the designated commands have been completed on one of the two saddles while the other saddle is placed in the slide hold mode with the single block function activated in the AUTO mode operation.

d) PROGRAM STOP

The PROGRAM STOP indicating lamp goes on during the execution of the program stop (M00) or optional stop (M01) function in the AUTO or MDI mode. And the indicating lamp flickers during the execution of the dwell (G04) function.

e) LIMIT

The LIMIT indicating lamp goes on when either of X- and Z-axis reaches the set variable soft-limit position.

It starts flickering when the spindle speed command calls for the spindle speed lower or higher than the available spindle speeds of the presently selected range, or when the active spindle speed reaches the higher limit specified by the maximum spindle speed limit function.

f) ALARM

The ALARM indicating lamp goes on when an erroneous machine operation is intended, the wrong part program data is read and decoded, or the computer fails to function normally.

The contents of the alarm are displayed on the CRT. Details concerning the alarm message display are provided in the Alarm & Error List Manual.

g) Status indicating lamps ON/blinking conditions table

Status Indicating Lamp	Condition for ON	Condition for Blinking
RUN	- The machine is normally running in the AUTO or MDI mode (except for during the SLIDE HOLD and PROGRAM STOP mode).	- The program selection command in a schedule program is being executed.
S.T.M	- The machine is waiting for the operation completion of an M code command. (Spindle rotation command, gear command, etc.). - A spindle speed command is given (S command). - A turret rotation command is given (T command). Only for the multi-machining specification - The machine is waiting for the operation completion of an multi-machining M code command. (C-axis joint command, M-tool spindle rotation command, etc.). - An M-tool spindle speed command is given (SB command). - The C-axis brake pressure is switched between high and low. Only for the ATC specification - The machine is waiting for the operation completion of the MG, MT or TN command.	- The following items have been changed during manual intervention. Tool number Spindle rotation/gear command Only for the multi-machining specification C-axis joint state C-axis clamp M-tool spindle rotation/gear command

Status Indicating Lamp	Condition for ON	Condition for Blinking
SLIDE HOLD	- The SLIDE HOLD button has been pressed in the AUTO or MDI mode. - A block of program commands has been executed on one saddle while the other saddle is placed in the slide hold mode with the single block function activated in the AUTO mode.	Does not blink
PROGRAM STOP	- M00 or M01 is designated in the AUTO or MDI mode.	- The dwell command (G04) is executed.
LIMIT	- Either X- or Z-axis has reached the variable soft-limit position.	- The spindle speed has reached the limit speed in the selected gear range. - The spindle speed has reached the limit speed specified by the maximum spindle speed designation function. Only for the electrically controlled spindle orientation specification - The spindle position is controlled after the completion of spindle orientation (DIFF control). - The M-tool spindle position is controlled after the completion of spindle orientation.
ALARM	- Any erroneous operation is attempted (An alarm of level A, B, C, or D).	Does not blink

(9) FUNCTION KEYS: F1 to F8

There are eight function keys on the NC operation panel.

When the operator selects the desired operation mode, the CRT displays the necessary operation functions at the bottom line. Each function corresponds to a function key (F1 through F8). Select the function to execute and press the corresponding function key.

The CRT provides guide of operative steps corresponding to the sequence of operation. The operator can proceed with the intended operation simply by pressing the function keys according to the guide messages given on the CRT.

For those functions, refer to pages 12 through 16.

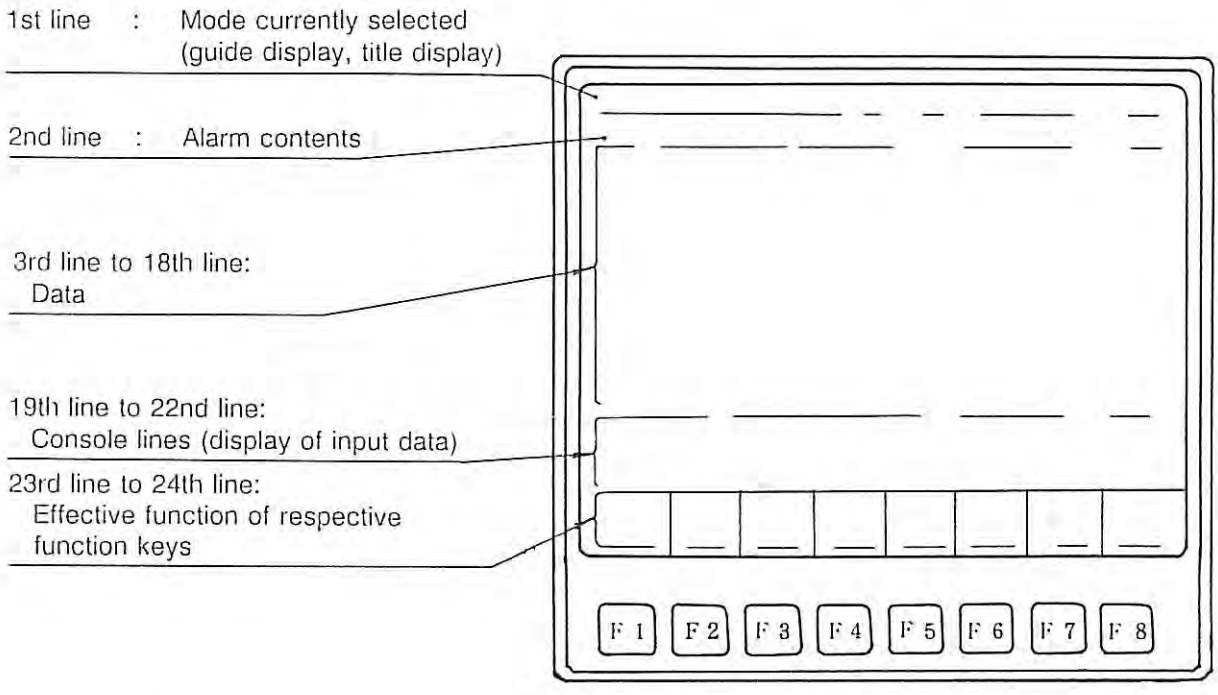
(10) CRT

The CRT can display 64 characters × 24 lines.

It shows actual position, part program, part program block, reference zero coordinates, tool offset values, parameter data, alarm, and others.

The contents of display are determined by the selected operation mode, active operation step, and pressed function key (F1 through F8).

The basic format of CRT display is as shown below:



(11) **WRITE**

Press the **WRITE** key to select the operation mode or to set the data in the MDI mode.

(12) **BS**

Press the **BS** key when erroneous data has been entered. Each time this key is pressed, the last character entered is erased. This key is also used to display the next page of the directory.

(13) **CANCEL**

Press the **CANCEL** key when erroneous data has been entered. Each time this key is pressed, a line of input data entered is erased.

(14) **TURRET**

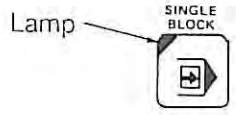
These keys are used to select A- or B-turret for the 2S and 2-turret models. These keys are also effective in the manual mode.

2-2. Machine Operation Panel

Flat keys:

- Flat keys with indicating lamps

  For flat keys with indicating LED at the upper left corner, input status is maintained by NC software.



  LED ON . . . Input ON

  LED OFF . . Input OFF

  *Note: Input status is not maintained for the **INTERLOCK RELEASE** key.*

- Flat keys without indicating lamps

  With flat keys not provided with the indicating lamp, input status is not maintained.

  Key being held down . . . . . Input ON

  Key not being held down . . . Input OFF

(1) CONTROL ON/RESET

Press the **CONTROL ON/RESET** button to turn on the power supply to the control and the servo system after the main switch at the side of the control enclosure has been turned on. When the control power is turned on, the pilot lamp in this button illuminates.

This button is also used to reset the control.

(2) CONTROL OFF

Press the **CONTROL OFF** button to shut off the power supply to the control and the servo system.

To turn off power supply to the machine after the completion of daily operation or for the mid-day recess, be sure to press this button first before turning off the main switch.

(3) NC PANEL

a) NC PANEL ON



All controls on both the NC and machine operation panel are enabled.

b) NC PANEL OPT



EDIT AUX and **PARAMETER** keys on the NC operation panel are disabled.

c) NC PANEL OFF



All controls on the NC operation panel are disabled.

(4) **CYCLE START**

Press the **CYCLE START** button to initiate machine operations according to the commands.

(5) **SLIDE HOLD**

Press the **SLIDE HOLD** button to stop axis movements of X- and/or Z-axis, immediately. To resume axis movements, press **CYCLE START** button.

Note: *If the **SLIDE HOLD** pushbutton is pressed while axis movement is not active, the machine cycle stops when the commands in the present block are completed or when the axis movement command is read and decoded.*

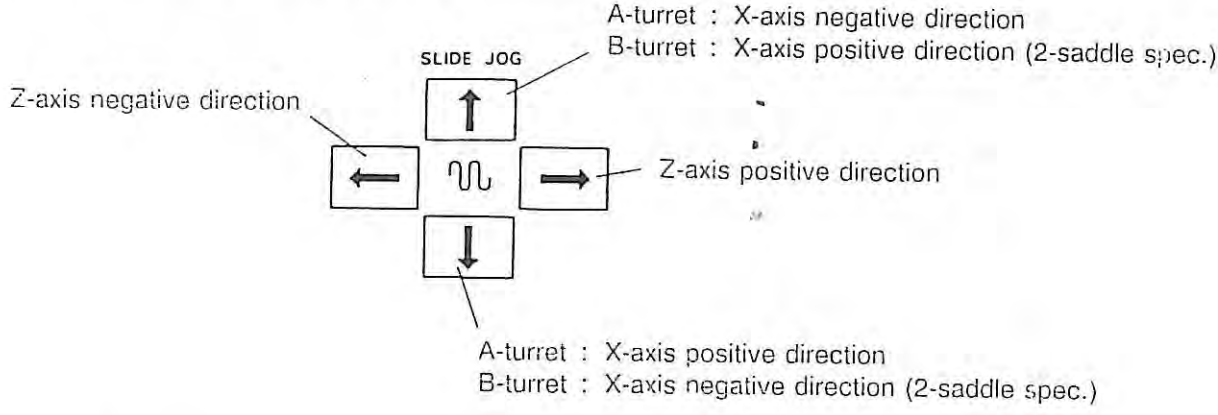
(6) **EMG. STOP**

Press the **EMG. STOP** pushbutton when an emergency state takes place.

When it is pressed, power supply to the control is shut off.

(7) **SLIDE JOG**

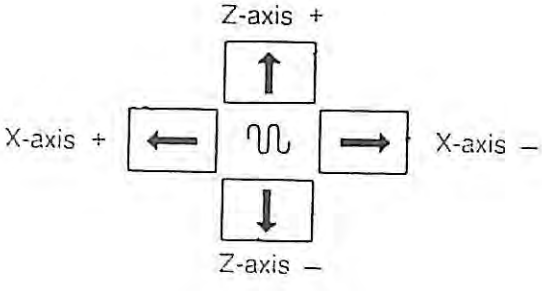
The four **SLIDE JOG** buttons are used to manually jog the axes.



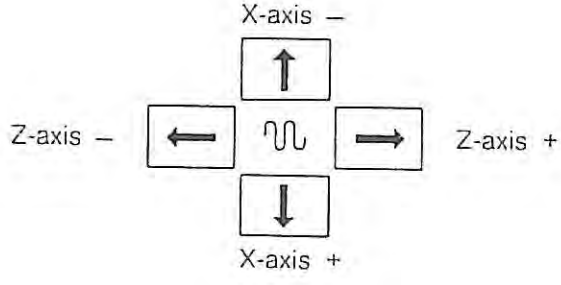
Axis movement directions differ on the models indicated below:

LPC4, LB6, LH35, LH55, LS30

1) LPC4



2) LH35, LH55, LS30N



Note: *Special patterns may be applied on some machine models.*

The axis moves while any of the **SLIDE JOG** buttons is pressed.

Setting of the **FEEDRATE** override dial is effective:

X-axis . . . . 1,200 mm/min at 100% setting (in radius)
Z-axis . . . . 2,400 mm/min at 100% setting

(8) **FEEDRATE** (%) override dial

The **FEEDRATE** override dial is used to modify the specified or commanded feedrate for optimizing the cutting.

Override range is 0 to 200% in 15 steps.

The **FEEDRATE** override dial setting is ignored when the thread cutting mode is effective.

The **FEEDRATE** override dial setting is effective in the G00 mode. However, the following requirements must be met:

- The Single block mode ON in the automatic operation mode
- Regardless of the on/off status of the single block mode in the MDI operation mode.
- Effective override range is 0 through 100% in 10 steps.

The **FEEDRATE** override dial setting is effective for manual operations. In this case, effective override range is 0 to 200% in 13 steps.

(9) **PULSE HANDLE**

By rotating the pulse handle, the axes can be fed in the same manner as with conventional lathes.

X/Z toggle switch:

Used to select the axis to be fed.

Magnification switch:

Used to select axis feed amount per pulse. Three positions are available:

10/1 . . . . . 10 μm/pulse (1/1000 inch/pulse)
1/1 . . . . . 1 μm/pulse (1/10000 inch/pulse)
50/1 . . . . . 50 μm/pulse (5/1000 inch/pulse)

(10) **TOOL INDEX**

Press the **TOOL INDEX** key to rotate (index) the turret and select the desired tool position in manual mode operation.

For two-saddle or two-turret models, it is necessary to select the turret to be indexed before pressing the **TOOL INDEX** key.

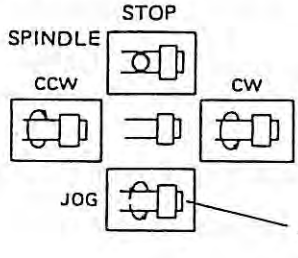
To enable turret indexing, the following conditions must be satisfied.

LB series LC series LP series LR series LH series LS30-N (V turret specification)	The X- or Z-axis must be at the variable soft-limit position in the positive direction. This is checked with the LIMIT indicating lamp under NC status on the NC operation panel; when either X- or Z-axis is at the variable soft-limit position in the positive direction, the indicating lamp is illuminated.
LH series LS30-N (V turret specification)	The X-axis must be at the variable soft-limit position in the positive direction.
LH series LS30-N (H6 and H8 specification)	The Z-axis must be at the variable soft-limit position in the positive direction.
LH series LS30-N (2-turret specification)	The saddle may be at any position.
FTL series	The Z-axis must be at the variable soft-limit position in the positive direction. Or the X-axis must be at the variable soft-limit in the positive direction and at the same time the Z-axis must be at outside the interference area.

One push of the **TOOL INDEX** key rotates the turret by one station. To rotate the turret continuously, hold down on this key.

(11) **SPINDLE - STOP/CCW/CW/JOG**

Four **SPINDLE** buttons are used to rotate/stop/jog the spindle manually.



The spindle jogs (FWD) while this button is held down.

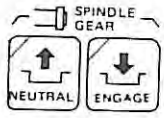
To manually rotate the spindle, the following requirements must be met:

- The chuck fulfills the spindle rotation conditions;
  Chuck close during OD gripping
  Chuck open during ID gripping

  For spindle jog operations, this requirement does not apply.
- Spindle speed is entered beforehand through the MDI keyboard.
- Spindle speed gear range selection (M41 - M44) is made beforehand. (only for models with spindle speed change gear mechanism)
- For center work, the tailstock spindle is at the specified position.

(12) **SPINDLE SPEED**

Spindle speed gear range is selected by pressing the keys indicated below in the manual operation mode.
(only for models with spindle speed change gear mechanism)



When the **NEUTRAL** key is pressed, the spindle is placed in the neutral state and the chuck may be rotated by hand lightly. The gear position data is saved and can be called out even after the power supply is turned off.

When the **ENGAGE** key is pressed, the spindle speed gear range selected previously is automatically selected. The selected range is displayed on the CRT screen in corresponding M code. (At MACHINE STATUS display on the ACTUAL POSITION display screen)

The lamp at the upper left corner lights up when the output of gears and confirmation limit switch match. If they do not match, it flickers in intervals of 0.4 sec.

This is also true when the spindle speed range is selected in the automatic or MDI mode. The spindle speed does not change even when the gear is changed.

(13) **SPINDLE OVERRIDE** (%)

The **SPINDLE OVERRIDE** dial is used to modify the commanded spindle speed in the range of 50 to 200% in 10 steps.

The spindle speed is not overridden if the operator selects a high percentage position that causes the spindle speed to exceed either the preset permissible spindle speed or that set by G50. In this case, the spindle rotates at the permissible highest spindle speed.

(14) **OT RELEASE - OFF/ON** (provided inside the control box)

Outside the variable soft-limit position of X- and Z-axis, there are emergency limit switches (hard-limit) that inhibit any axis travel exceeding them. If any of these emergency limit switches is tripped, power supply to the servo system and electrical control circuit is shut off and the two axes cannot be moved anymore.

In such a case, follow the steps below after turning the **OT RELEASE** selector switch to the ON position. Proceed very carefully.

a) Select the MANUAL OPERATION mode by pressing the **MANUAL** key.

b) Press the **POWER ON** pushbutton while turning the **OT RELEASE** selector switch to the ON position. Keep the **OT RELEASE** selector switch in the ON position until the axis releases from the overtravel state.

c) Turn the pulse handle to move the axis in the direction opposite the overtraveled direction.

d) Move the axis until the tripped limit switch is disengaged from the dog.

After that, turn the **OT RELEASE** selector switch to the OFF position.

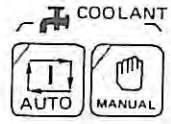
Note 1: Be sure to move the correct axis.

Note 2: Be sure to move the axis in the correct direction.

Note 3: If the axis fails to move smoothly while releasing it from the hard-limit position, or the axis hunts for instance, stop the releasing operation immediately by placing the ***OT RELEASE*** *selector switch OFF.*

(15) **COOLANT**

The **COOLANT** keys are used to control the coolant supply condition:



AUTO and **MANUAL** OFF . . Coolant is not supplied irrespective of the operation mode selected.

MANUAL ON . . . . . Coolant is supplied in any operation mode.

AUTO ON . . . . . Coolant ON/OFF is controlled by M codes in the automatic and MDI mode:

M08 . . . Coolant ON
M09 . . . Coolant OFF

Note: ***AUTO*** *and* ***MANUAL*** *keys cannot be on at the same time.*

(16) **LIGHT**



The **LIGHT** key is used to turn on and off the work lamp of the machine.

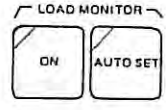
(17) **CRT DISPLAY**



The **CRT DISPLAY** key is used to turn on and off the CRT display.

(18) **LOAD MONITOR**

The **LOAD MONITOR** keys are used to monitor the cutting load of the feed axes or the spindle.



2-3. Setting Keys on Machine Operation Panel

(1) **SINGLE BLOCK**

Press the **SINGLE BLOCK** key ON when executing part program blocks one by one in the AUTO and MDI mode. When it is set to the OFF position, part program blocks are continuously executed.

This SINGLE BLOCK function is effectively used when cutting the first part.

(2) **BLOCK DELETE**

When the **BLOCK DELETE** key is turned to the ON position, commands preceded by a slash (/) code are ignored up to the CR code in that block.

Program the slash code either at the beginning of the block or right after the sequence number (name) of the block.

(3) **OPTIONAL STOP**

When the **OPTIONAL STOP** key is pressed, the machine cycle, including spindle rotation and coolant supply, stops after the commands in the block containing M01 are completed. The operation can be resumed by pressing the **CYCLE START** pushbutton.

(4) **DRYRUN**

Press the **DRYRUN** key when checking a newly prepared part program in the AUTO mode. With the **DRYRUN** function activated, cutting feedrate commands, with the exception of manual feed and G00 mode feed, are all executed at the milling feedrate (mm/min) set by the parameter.

The milling feedrate (parameter (long word) No. 9) is factory-set at 2,400 mm/min.

(5) **MACHINE LOCK**

When a part program is executed with this **MACHINE LOCK** key pressed, all commands in the part program are executed without actual machine operation. The simulated operation can be checked on the CRT. In this mode, cutting feedrate is determined based on the programmed spindle speed.

(6) **INDIVIDUAL MODE**

INDIVIDUAL MODE keys (A, B) are available only with two-saddle models.

With two-saddle models, normal operation is made in the simultaneous 4-axis control mode. These keys are used to select the required operation mode:

Individual A turret operation:

A ..... ON
B ..... OFF

With the setting above, only turret A is activated in the AUTO and MDI mode.

Individual B turret operation:

A ..... OFF
B ..... ON

With the setting above, only B turret is activated in the AUTO and MDI mode.

Individual A/B turret operation:

A ..... ON
B ..... ON

With the setting above, both A and B turrets are activated individually according to the designated synchronizing commands.

Normal operation:

A ..... OFF
B ..... OFF

Normal machine cycle is performed according to the programmed commands.

Note: *To activate the following keys, the* **INTERLOCK** *key must be pressed at the same time:*

- *DRY RUN*
- *MACHINE LOCK*
- *INDIVIDUAL MODE keys (A/B)*

(7) **SEQUENCE RESTART**

Press the **SEQUENCE RESTART** key to restart the AUTO mode operation which has been interrupted due to tool breakage or because of too long machining cycle time, after resetting the control. For details of sequence restart operation, refer to the instructions in Section 4.

(8) **MID AUTO MANUAL**

Press the **MID AUTO MANUAL** key to interrupt AUTO mode operation; carry out the required manual mode operation, and then resume the AUTO mode operation.

(9) **CRT OFF**

The CRT display may be turned off by holding down this key.

3. Fundamental Machine Operation Procedure

3-1. Manual Operations

NC lathes operate in three different modes: MANUAL, AUTO (stored part programs), and MDI. This section deals with manual machine operation procedure, which will provide you with the most fundamental knowledge to operate the machine.

It is advisable to actually operate the machine while reading this manual.

(1) Turning ON/OFF Power

When turning power supply to the machine ON for the first time after machine installation, make sure that your power supply is compatible with the machine and the control. Consult our service engineer before doing this. (See Fig. 3-1.)

a) Turning power ON:

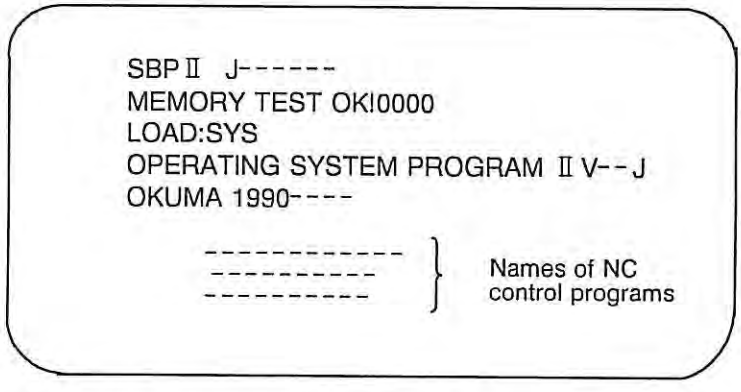
1) Turn the main switch (no-fuse circuit breaker) on the left side of the electrical control enclosure ON.

This turns 200 volt AC power to the electrical control system. The hydraulic power unit pump motor starts at the same time.

2) Press the **CONTROL ON/RESET** pushbutton on the machine operation panel.

This turns on power supply to the CPU and relay circuit.

The CRT will display the contents shown below in five to six seconds after the **CONTROL/RESET** pushbutton has been pressed, and the NC software is loaded.



After the NC software has been loaded, the NC starts running with the following messages displayed in succession. These messages allow the operator to know the faulty loading process if start was impossible:

- PBU FILE ON LOADING
- PITCH COMPLEMENT DATA ON LOADING
- EC BUS ON INITIAL PROCESSOR
- GRAPHIC DATA ON INITIAL PROCESSOR
- SERVO PROCESSOR ON INITIAL PROCESSOR
- AXIS PROCESSOR ON INITIAL PROCESSOR
- EC PROCESSOR ON INITIAL PROCESSOR

Note: *Messages corresponding only to the specifications selected are displayed.*

30 minutes after that, the control becomes ready to operate and the CRT display changes to NC operation data (actual position, program, etc.). The servo drive circuit and other control circuits are energized at the same time.

The **CONTROL ON/RESET** pushbutton also serves as the **RESET** pushbutton; press it to reset the control during operation or when the ALARM indicator is illuminated.

b) Turning power OFF

1) Press the **CONTROL OFF** pushbutton on the machine operation panel.

This turns off power supply to the CPU, servo drive system, relay and magnetic circuits.

2) Turn the main switch (no-fuse circuit breaker) on the left side of the control enclosure OFF.

This shuts off power supply to the machine.

REMARKS

To turn on the power to the machine after shop power supply has been turned off or after the main switch has been turned off without pressing the **CONTROL OFF** button, allow at least one minute to elapse.

c) Emergency stop

To stop machine operation when some abnormal state takes place, press the **EMG. STOP** pushbutton on the machine operation panel. This cuts off power supply to the hydraulic power unit and servo drive unit with **"Alarm A EMERGENCY STOP"** displayed. Even after this, the CPU is still active. Releasing the **EMG STOP** pushbutton and pressing the power **ON** button restores the control to the normal state.

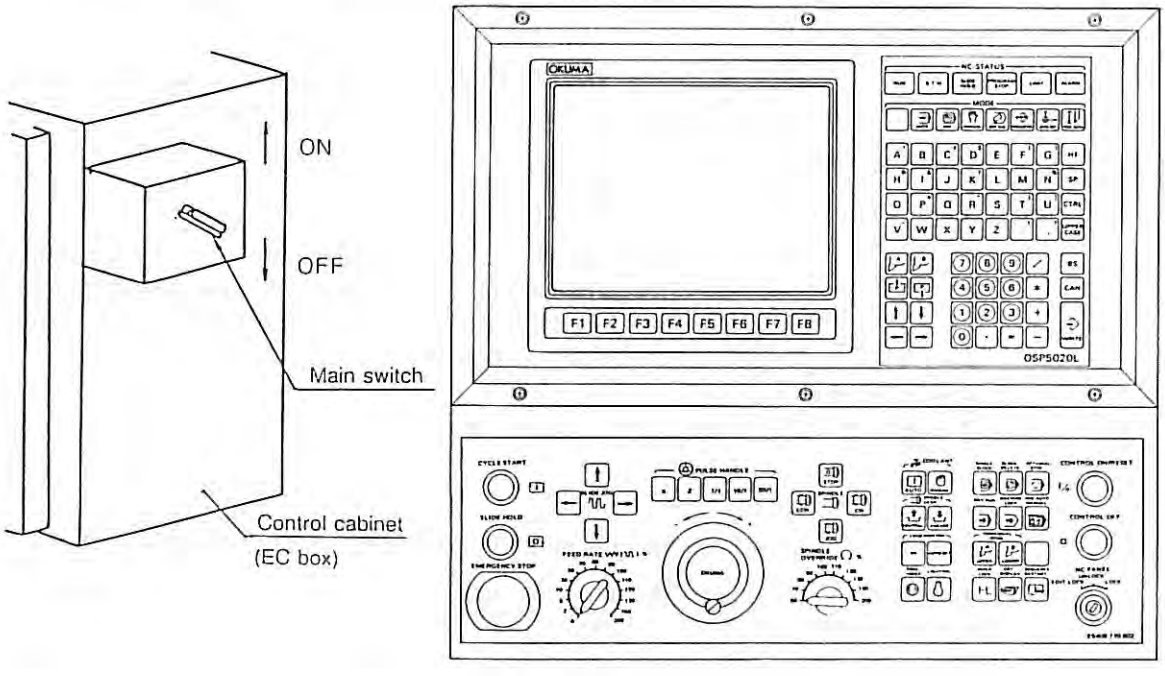


Fig. 3-1

(2) Starting Spindle Rotation

To prevent hazards to operator and machine, carry out the following steps without workpiece on the machine.

CAUTION

a) Check whether the allowable chuck speed is set at the parameter.

The allowable chuck speed is indicated on each individual chuck. If the spindle speed exceeds the allowable chuck speed, it is really dangerous to the operators and to the machine as well.

To provide the safety, the allowable chuck speed is set at the parameter and if the actual spindle speed exceeds 120 percent of the set value, an alarm occurs to stop the machine.

Whenever a chuck is changed, always set the allowable chuck speed at the parameter.

CAUTION

The allowable chuck speed is set at the following parameter.

Parameter	. . . . .	Optional parameter (word) No. 73
Unit	. . . . .	rpm
Setting range	. . .	0 to max. rpm of individual models

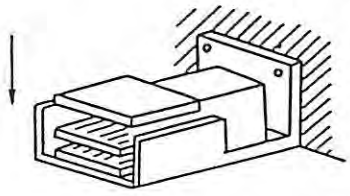
For the procedure to set the parameter, refer to Section 4, 3.

Check the parameter set value whether it matches the allowable speed of the chuck currently in use.

Note: *When the **DOOR INTERLOCK** switch is placed in the ON position, an alarm occurs if the door is open when the spindle is started.*

For details, refer to 3-2-6.

b) Bring the chuck in the work chucking position.



Chuck Opening/Closing Foot Pedal

Close the chuck for OD gripping and open it for ID gripping. Unless the chuck is set in this state, the spindle cannot rotate. Press the foot pedal, and the chuck opens and closes alternately each time it is pressed.

Selection of ID and OD gripping is made by parameter. (for details, refer to Section 4, 3).

For the machine equipped with the tailstock, advance or retract the tailstock sleeve corresponding to the setting of the CHUCK WORK or CENTER WORK selection.

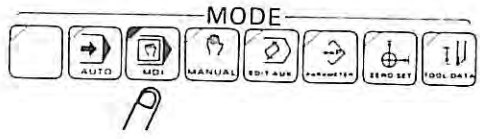
Selection of the CHUCK WORK and CENTER WORK is made by the parameter setting as with the selection of ID/OD gripping.

c) Set the desired spindle speed.

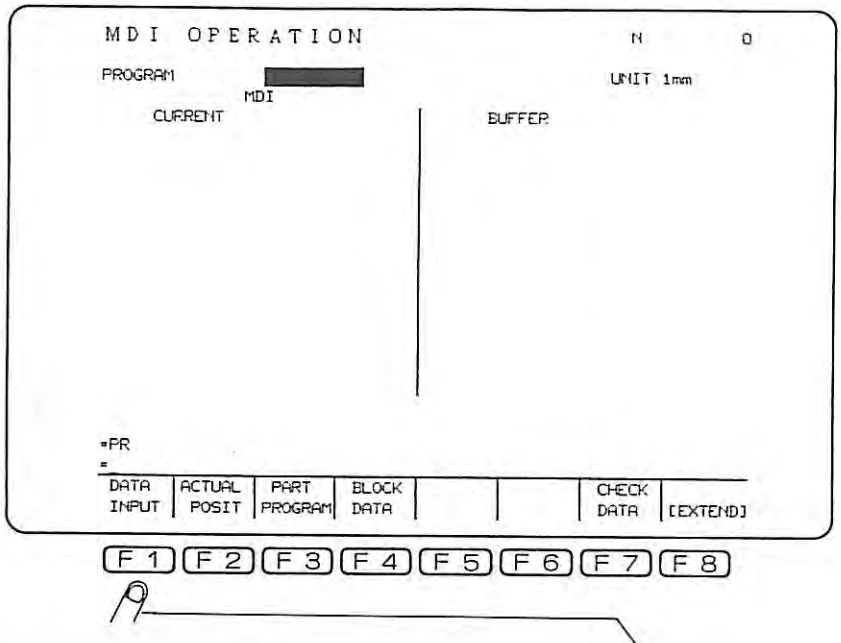
Example: To select 1,000 rpm.

Procedure:

1) Select the MDI operation mode by pressing the **MDI** key.



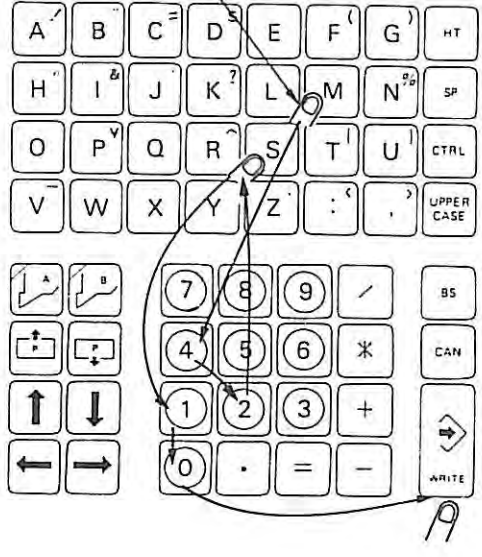
2) Press the function key [F3] (PART PROGRAM) and press the **PAGE** key repeatedly until the page shown below is displayed.



3) Press the function key [F1] (DATA INPUT).

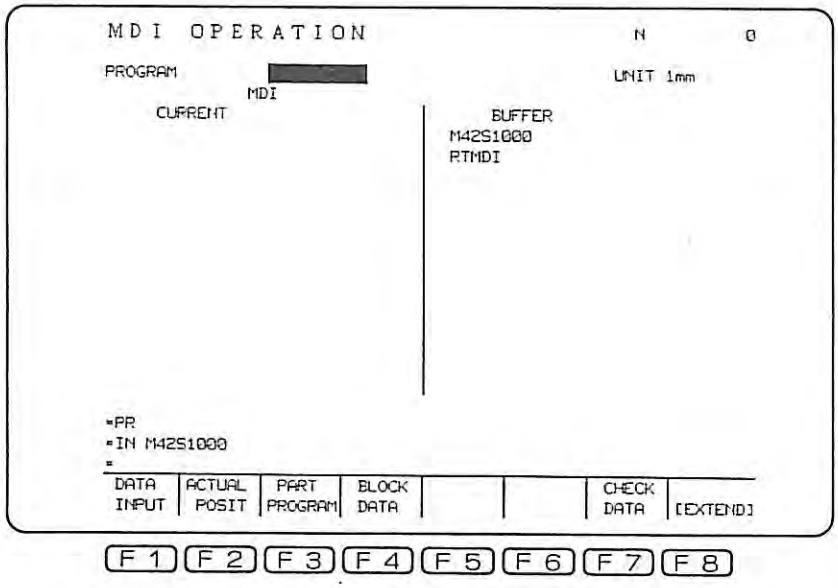
Key in [M][4][2][S][1][0][0][0] and press the **WRITE** key.

(This enters the spindle speed range and the required spindle speed.)

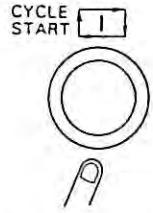


4) The CRT displays the data just keyed-in.

(When the **CYCLE START** pushbutton is pressed, the data in the BUFFER column is transferred to the CURRENT column.)



5) Press the **CYCLE START** pushbutton.



This only enters the keyed-in data, S1000, and spindle does not start.

To start spindle rotation, follow the steps provided in c).

Note 1: *For spindle speed range selection, the relationship between the M codes and the available spindle speeds differ from model to model. For spindle speed range selection M codes, refer to the Operation Manual for the respective model.*

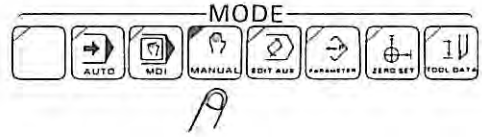
Before entering spindle speed commands, M and S words, check if they match.

Note 2: *For details of the MDI mode operation procedure, refer to instructions in 3-2-5.*

d) Start spindle rotation.

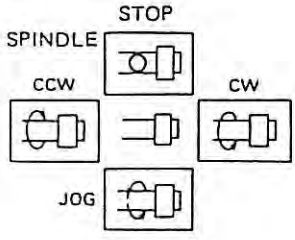
Procedure:

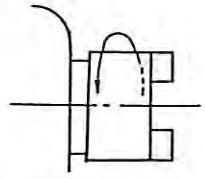
1) Select the MANUAL OPERATION mode by pressing the **MANUAL** key.



2) Press the **SPINDLE CW/CCW/JOG/STOP** key on the machine operation panel to rotate the spindle.

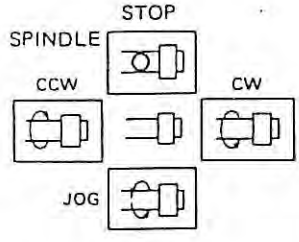
SPINDLE - CW key

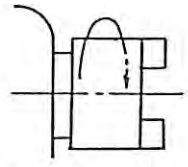




Press the **SPINDLE - CW** key, and the spindle starts CW rotation at 1,000 rpm. The spindle keeps rotating until the **SPINDLE - STOP** key is pressed.

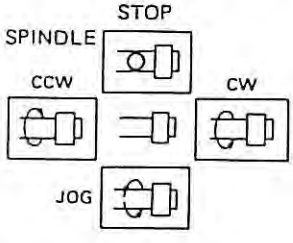
SPINDLE - CCW key

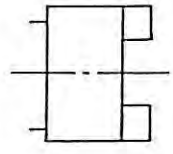




Press the **SPINDLE - CCW** key, and the spindle starts CCW rotation at 1,000 rpm. The spindle keeps rotating until the **SPINDLE - STOP** key is pressed.

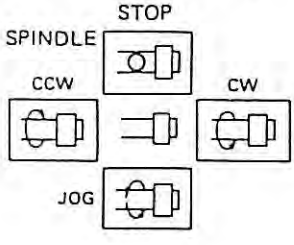
SPINDLE - STOP key

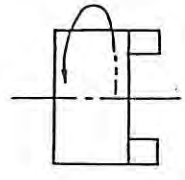




Press the **SPINDLE - STOP** key, and the spindle rotation stops.

SPINDLE - JOG key





Press the **SPINDLE - JOG** key, and the spindle rotates in the forward direction while the key is pressed.

Note 1: *For spindle jog speed setting, refer to Section 4, 3. "Operation in Parameter Mode".*

Note 2: *While the chuck is open, spindle CW/CCW operation is not possible though spindle jog can be performed.*

Note 3: *The commanded spindle speed can be overridden in the range from 50 to 200% using the* ***SPINDLE OVERRIDE*** *dial.*

Note 4: *On machine models with DC spindle drive motors (LC30, LC40, LC50, and LH55-N) it is not possible to directly change from CW to CCW or from CCW to CW.*

(3) Moving Turret

CAUTION

In order to prevent accidental collision of the turret, observe the following points carefully:

Possibility of collision between the turret and the chuck

Possibility of collision between the upper (A) and lower (B) turrets (for machine with two saddles)

Tailstock position. Be sure to position the tailstock at its rightmost position (for machine with tailstock).

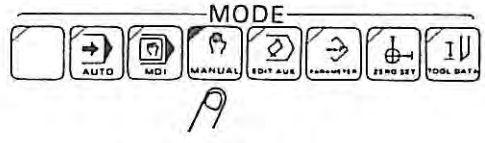
When the turret is moved with the door open, an alarm occurs if the door interlock function is active (**DOOR INTERLOCK** switch ON). For details, refer to 3-2-6.

Note: Soft-limit positions of X- and Z-axis are factory-set.

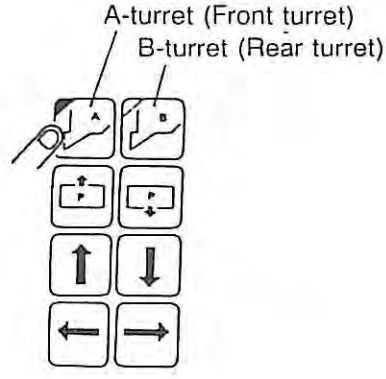
a) Moving turret at rapid feedrate

Procedure:

1) Select the MANUAL OPERATION mode by pressing the **MANUAL** key.



For the 2-turret/2-saddle models, specify the turret.



2) The turret moves while any of the **SLIDE JOG** keys is pressed in the direction indicated by an arrow mark on it.

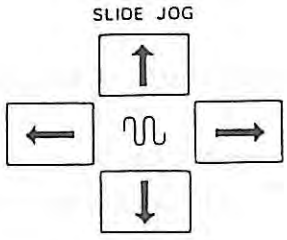
The rapid feedrate of each axis is (**FEEDRATE** override dial setting: 100%):

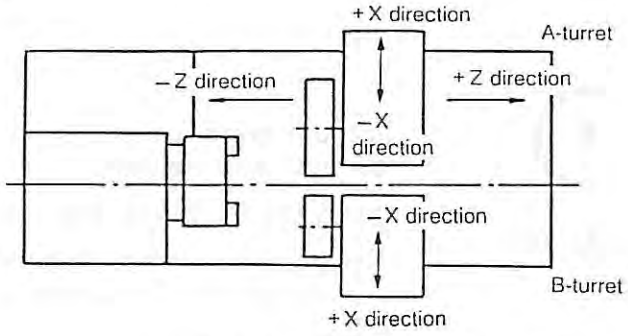
X-axis . . . 1,200 mm/min (in radius)
Z-axis . . . 2,400 mm/min

Rapid feedrate can be modified using the **FEEDRATE** override dial.

Axis motion directions are shown below.

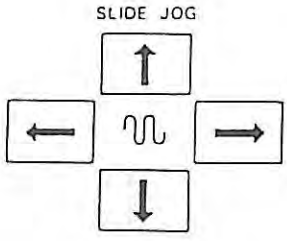
- 2-saddle model (1-saddle model)

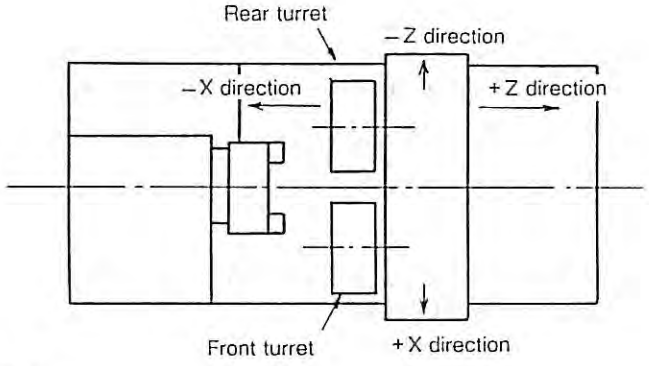




A single-saddle model has only A-turret.

- 2-turret model

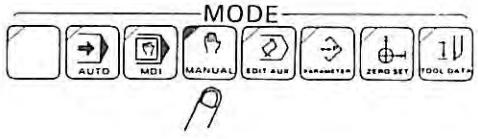




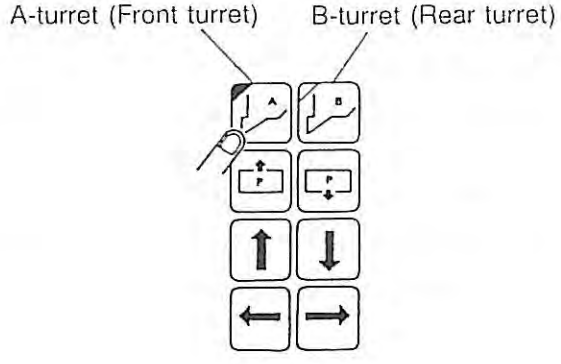
b) Moving turret by pulse handle

Procedure:

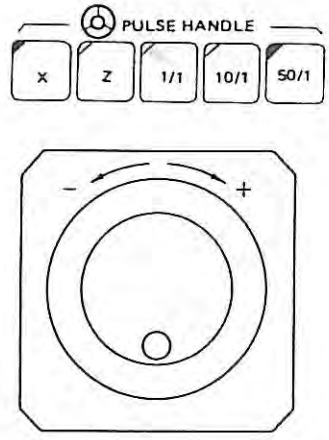
1) Press the MANUAL OPERATION mode by pressing the **MANUAL** key.



For the 2-turret/2-saddle models, specify the turret.



2) Select the axis to move and place the magnification switch in the desired position.



3) Turret feed speed varies in accordance with the pulse handle rotation speed.

Magnification	Metric System			Inch System		
	1/1	10/1	50/1	1/1	10/1	50/1
Feed per division	1/1000 mm (1 μm) mm/div.	1/100 mm (10 μm) mm/div.	5/100 mm (50 μm) mm/div.	1/10000 inch/div.	1/1000 inch/div.	5/1000 inch/div.
Feed per turn	0.1 mm/turn	1 mm/turn	5 mm/turn	0.01 inch/turn	0.1 inch/turn	0.5 inch/turn

The turret is fed rightward when the pulse handle is rotated clockwise and leftward when it is rotated counterclockwise.

As explained in (3), the turret can be manually fed in a simple operation; manual feed operation is effectively used to:

- produce one-of-a-kind parts with relatively simple configurations,
- cut off soft blanks for the power chuck jaws,
- minimize work deflections by turning the chucking diameters of parts,
- set up the correct relationship of a cutting tool point with respect to the part to be cut,
- move the turret to change cutting tools, and so on.

(4) Indexing Turret

When indexing the turret, observe the following point to assure safety in turret index operation.

Make sure that no turret-mounted tool interferes with the power chuck and tailstock (if installed).

CAUTION

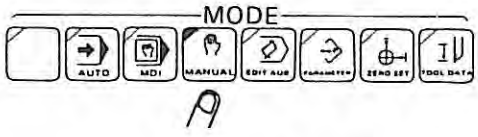
When the turret is moved with the door open, an alarm occurs if the door interlock function is active (**DOOR INTERLOCK** switch ON). For details, refer to 3-2-6.

Procedure:

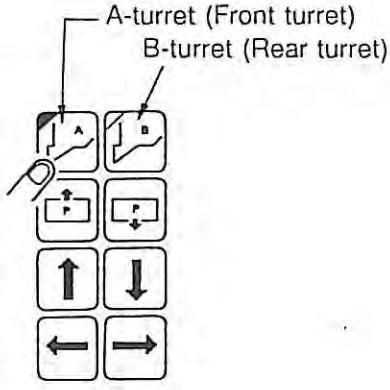
1) Move the turret to the turret indexing position using the **SLIDE JOG** keys on the machine operation panel.

   Turret Indexing Position . . . . Positive soft-limit position of X-axis or Z-axis.

2) Select the MANUAL OPERATION mode by pressing the **MANUAL** key.

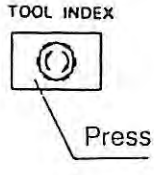


3) Select the turret to index.



4) Press the **TOOL INDEX** key.

A push on the key indexes the turret by one station. When it is held down, the turret rotates continuously.



Rotation direction can be commanded by M codes. However, this is not possible with the following models:

LH series and LS30-N

M codes are as follows:

M86 . . . Turret right rotation command
M87 . . . Turret left rotation command

This completes a brief explanation of the operating procedure for the essential movements of the machine. Frequent repetition of this procedure will insure speedy familiarization with your NC lathe.

3-2. Preparation of Machine for Operation

The following is a detailed explanation of the necessary steps and precautions for setting up the machine.

3-2-1. Tool Setting

The optional special tool setting caliper is used to set the individual cutting tools and holders on the machine.

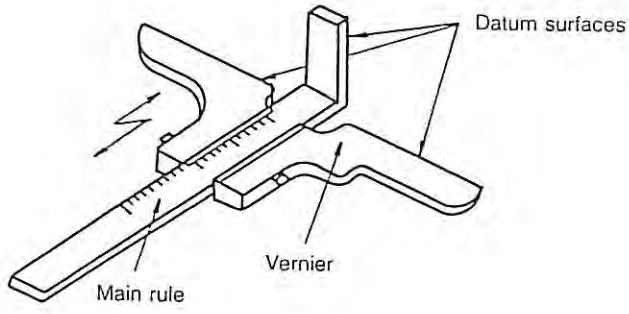


Fig. 3-2 Setting Caliper (optional)

Design:

The caliper consists of a gauge head with a slot to accommodate a narrow main rule. Featuring a total of three finished datum surfaces, the gauge is designed on the same principle as a caliper unit.

The main ruler is graduated up to 150 mm and may be used to measure the tool projection amount from 0 to 150 mm.

How to Use the Caliper:

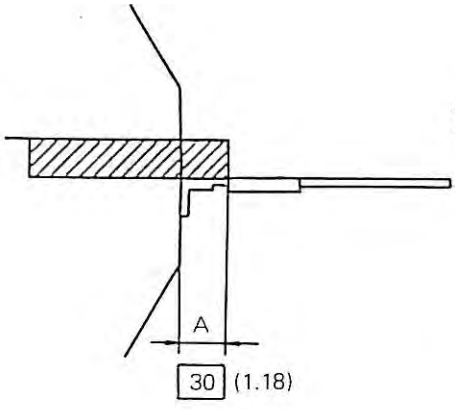
Take the reading with the datum surface held flush against the datum surface of the toolholder and the cutting tool, usually the tool point.

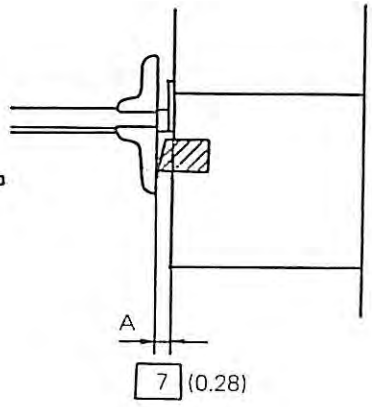
Carry out measurement in both X- and Z-axis directions. The read value indicates the tool projection amount from the datum surface of the toolholder.

(1) OD Tool Turret

a) Standard Tool

X, Z : Tool offset values
A : Measured value



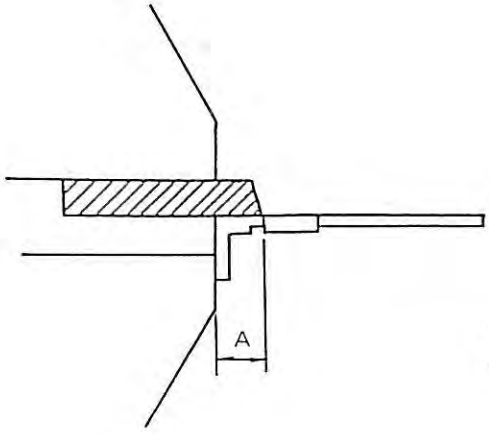


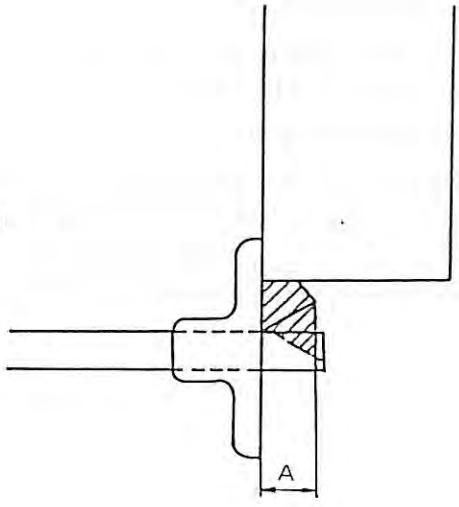
$X = +2 (A-30)$

$Z = A-7$

b) Inverted Tool

X : Same as for standard tool above



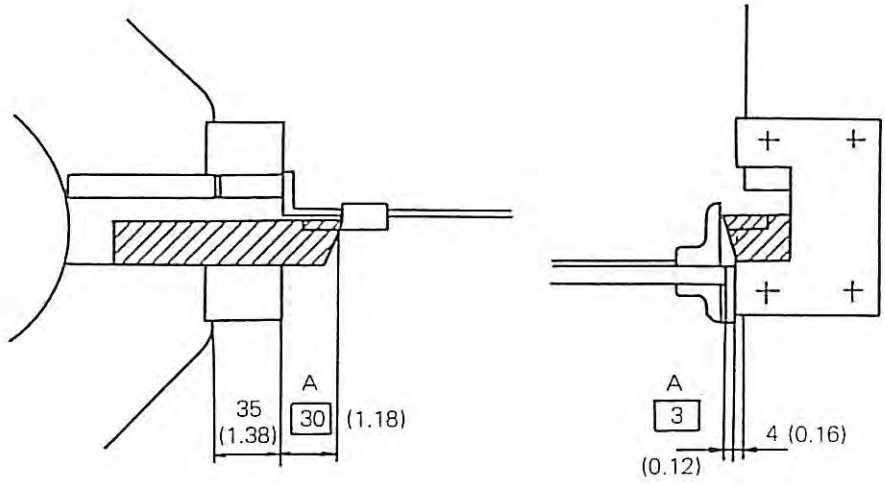


$X = +2(A-30)$

$X = -(A+7)$

Unit: mm (in.)

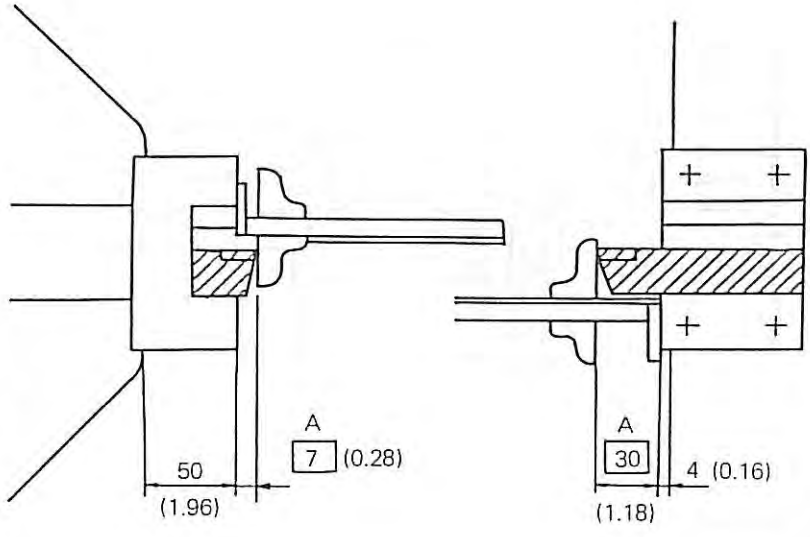
(2) OD Type Ⅰ Toolholder



X = 2(A+5)

Z = A−3

(3) OD Type Ⅱ Toolholder



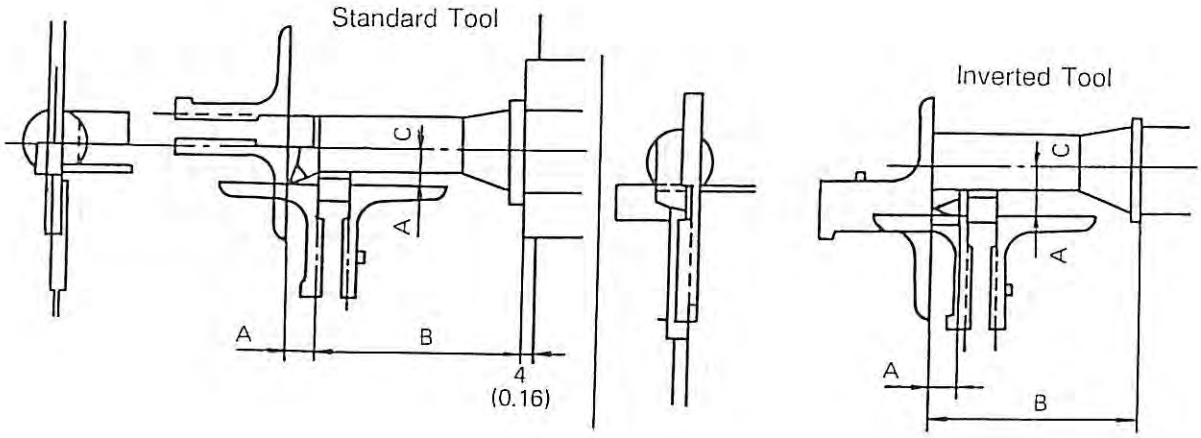
X = 2(A+20)

Z = A−3

Unit: mm (in.)

(4) ID Toolholder

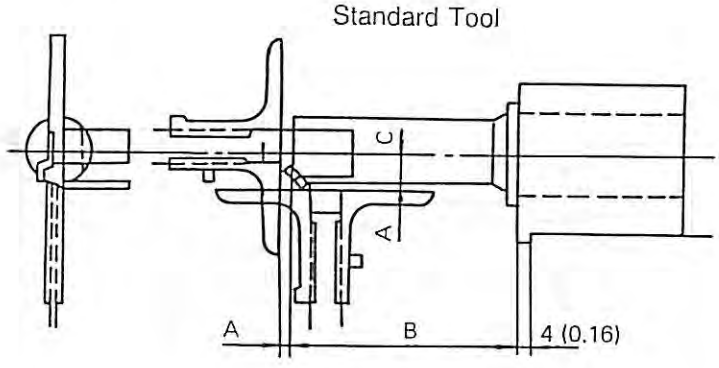
X, Z : Tool offset values
A : Measured value



X = −2 (A+C)
Z = A+B−3

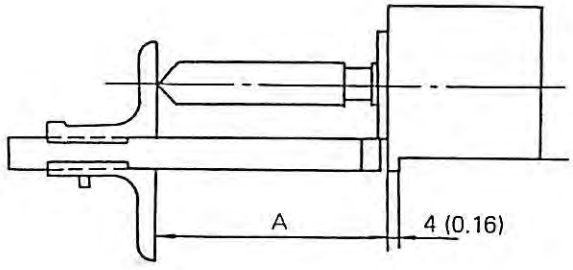
X = −2 (A+C)
Z = B− A−3

(5) Boring Bar



X = −2 (A+C)
Z = A+B−3

(6) Drill Sleeve



X = 0
Z = A−3

Unit: mm (in.)

3-2-2. Setting Zero Offsets

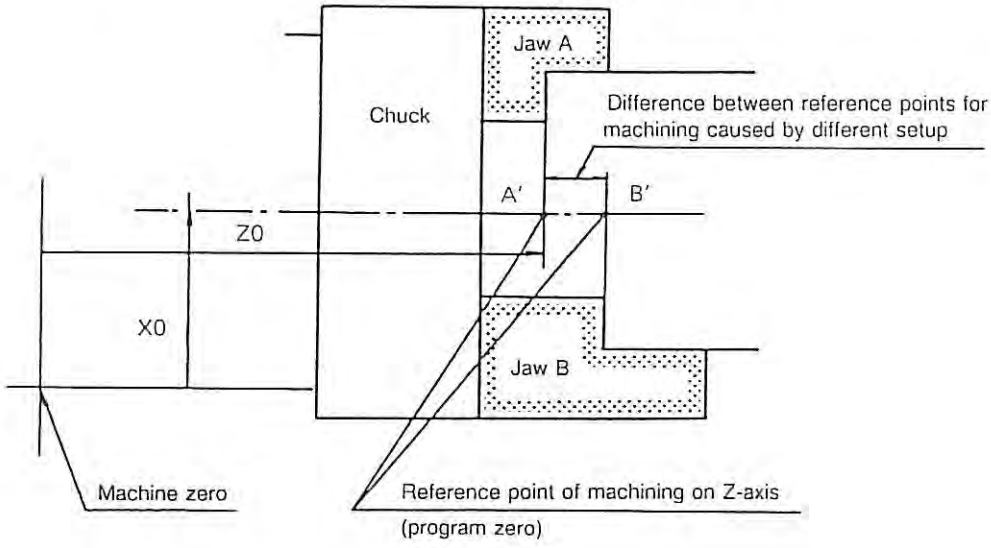
(1) What is Zero Offset?

The common coordinate position from which a complete program is made for a particular component is termed "zero point or program zero".

The programming zero is located at the fixed position (center of the spindle) on the X-axis. However, the program zero on the Z-axis will vary depending on the setup (incl. chuck, jaws, etc.).

With the NC lathe, the program origin (program starting point) is fixed anywhere on the Z-axis, that is, on the longitudinal axis of the spindle. It may vary with respect to the direction of Z-axis, according to the chucking requirements. As shown below, there is a difference in the coordinate position of zero points between one program using jaws A and another program using jaws B. This is caused by the difference in jaw sizes used in respective programs.

The zero offset feature provides for shifting the zero point of the program with respect to the zero point of the machine to match differences in individual workpieces or setups.

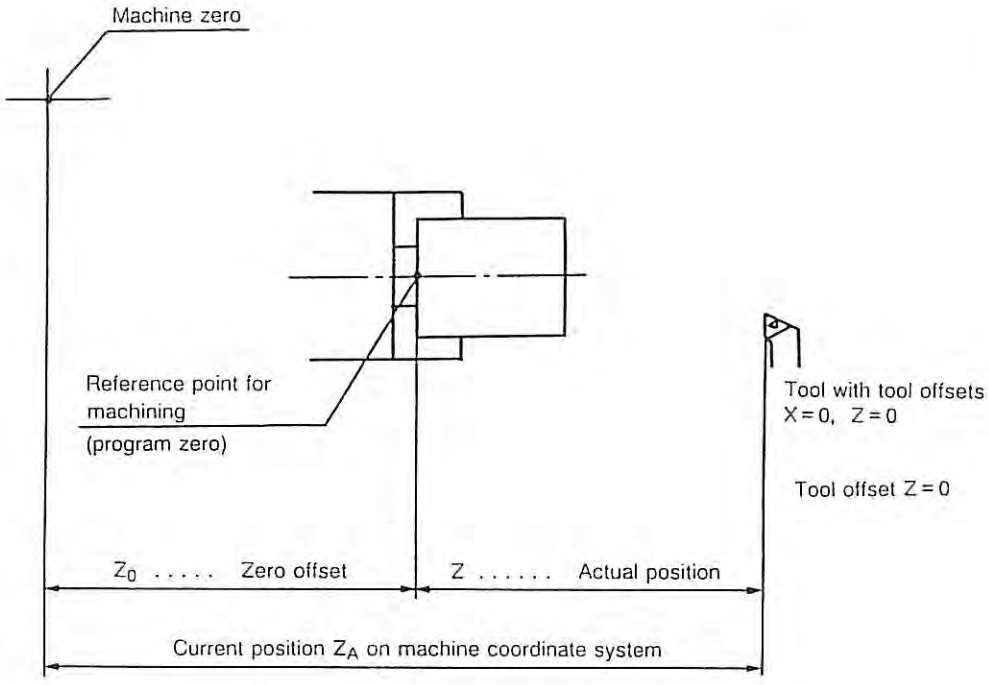


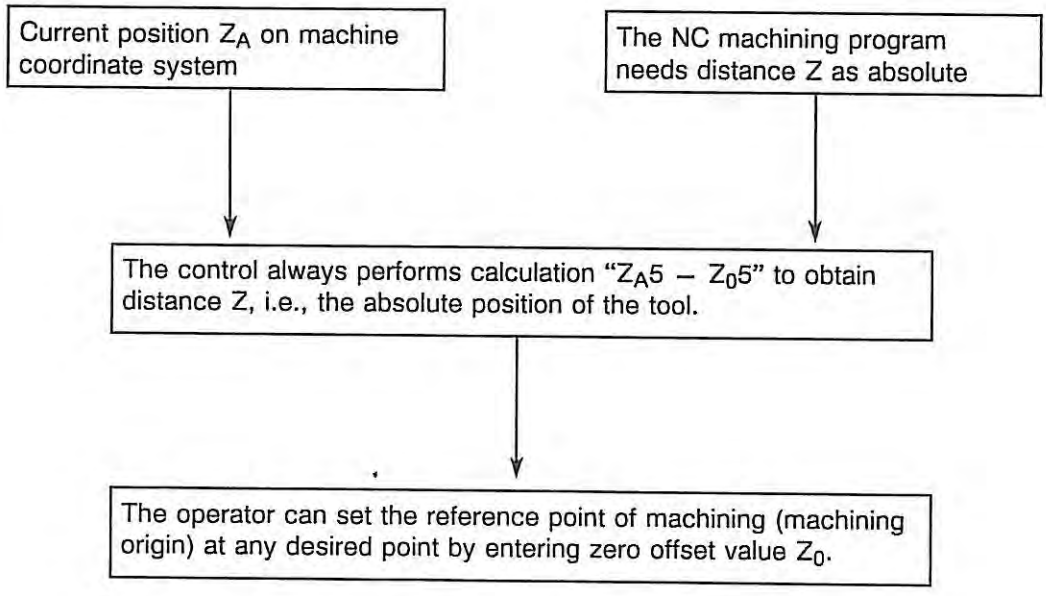
The operator can establish the reference point for machining (zero point of a program) by entering X_0 and Z_0 through the keyboard dimensioned from the fixed zero point of the machine.

"X_0, Z_0" is called Zero Offset Values.

(2) Relation between Machine Zero, Program Origin, Zero Offset Value and Actual Position

Shown below is the positional relationship between the factors involved in the Zero Offset function:





(3) Setting Zero Offset Values

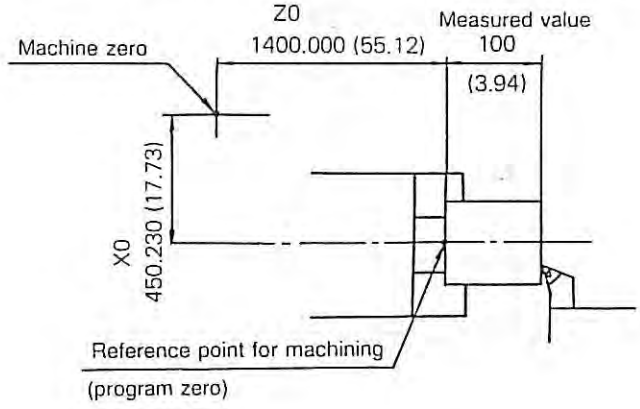
There are three possible cases for entering zero offset values:

a) Where zero offset values are unknown, as in cutting the first workpiece for instance.

b) Where zero offset values are known, as in cutting workpieces of repetitive lots.

c) Where the stored offset values are modified.

Explanation for each case is provided in this paragraph with the following example.



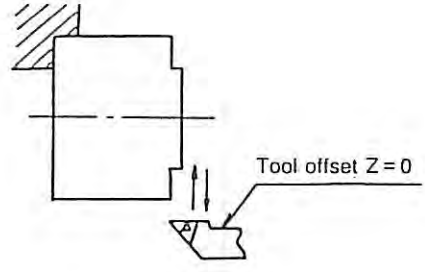
a) Case where zero offset value is unknown:

The explanation below is provided with 1 mm unit system.

To set the zero offset value of Z-axis, proceed as follows:

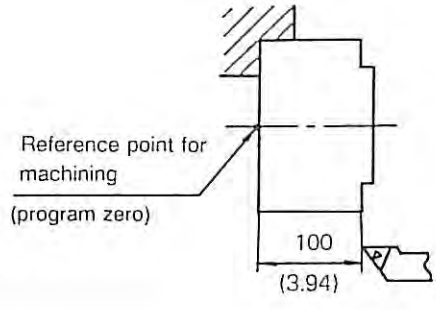
Procedure:

1) Turn the end face of the part with a proper depth of cut in the manual mode.



2) Measure the workpiece length to obtain the actual position of the tool dimensioned from the program zero.

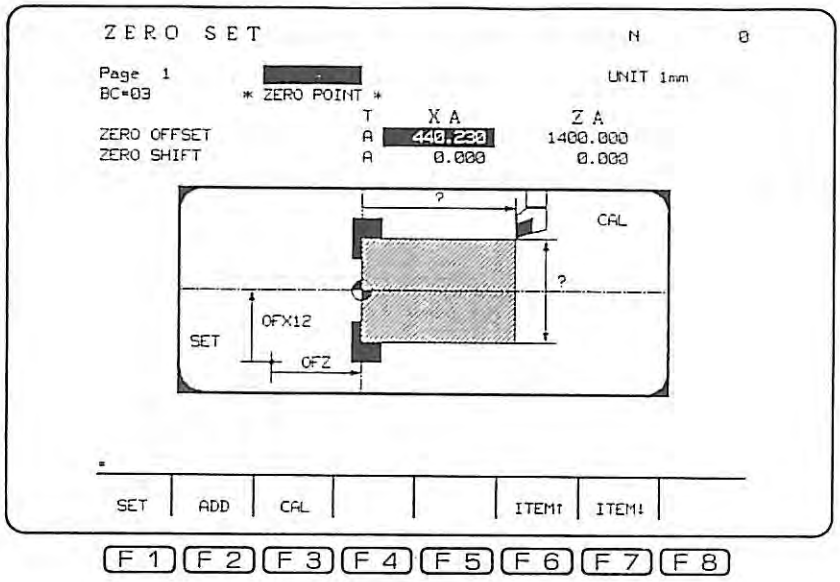
Assume this dimension is measured as 100 mm (100.000).



3) Select the ZERO SET mode by pressing the **ZERO SET** key.



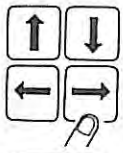
4) The CRT display is as shown below.

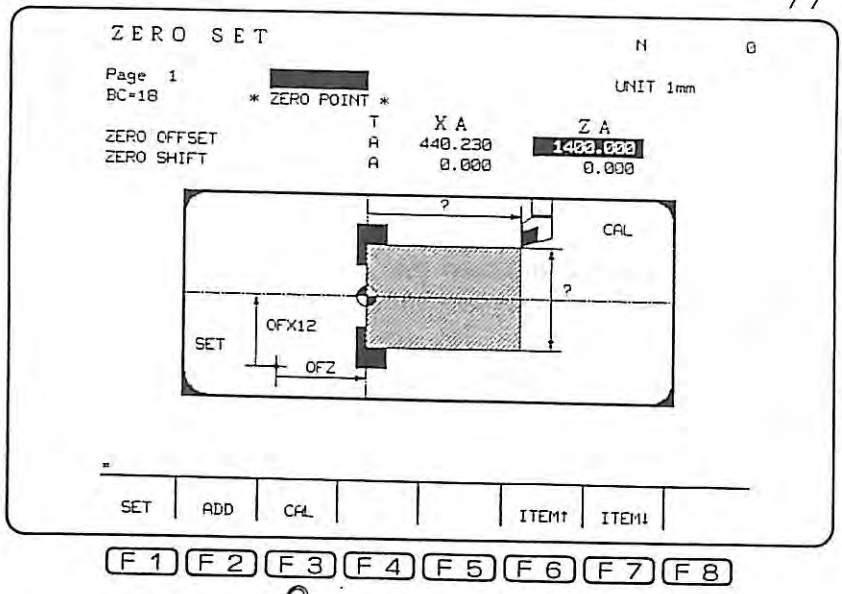


5) Select the turret, either A- or B-turret (for two-saddle and two-turret models).

Each time  or  is pressed, turret A and B is selected alternately.

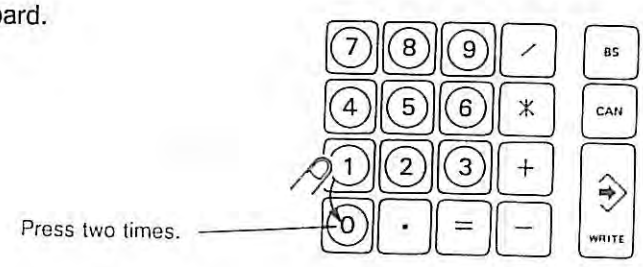
6) With the cursor control keys, move the cursor to the data column of ZERO OFFSET - ZA.

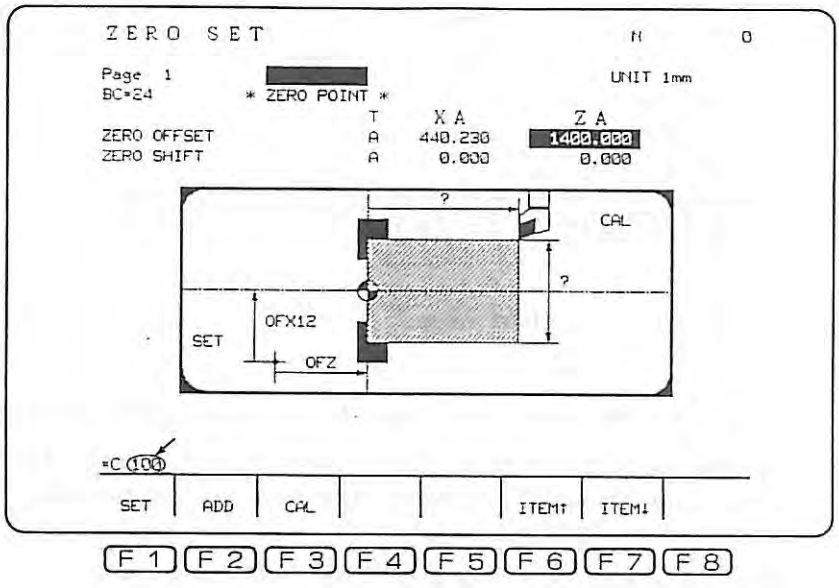




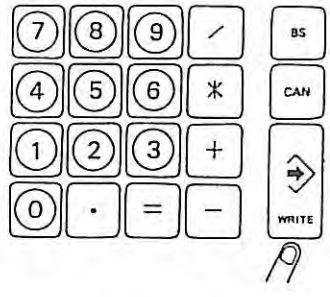
7) Press the function key [F3] (CAL).

8) Key-in [1][0][0] through the keyboard.



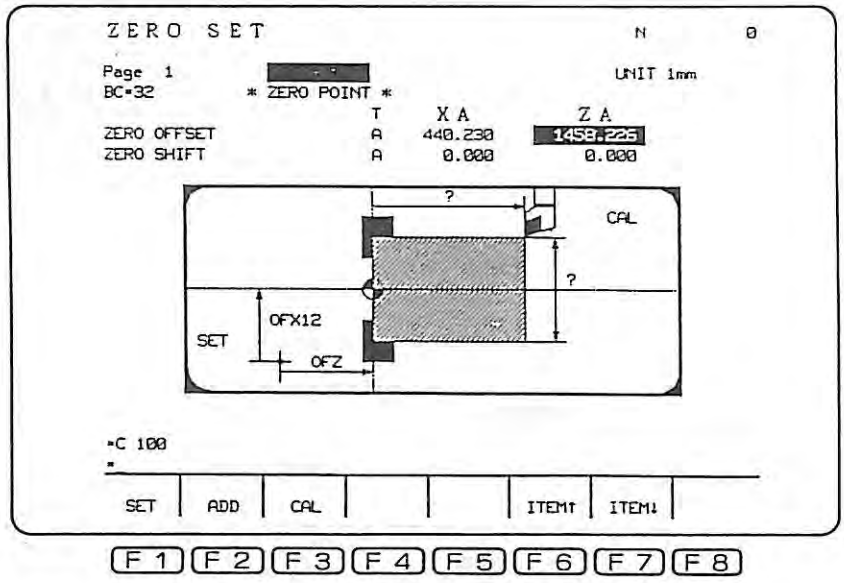


9) Press the **WRITE** key.



With this, the coordinate system is established so that the present tool position takes coordinate value Z100 mm.

10) The CRT displays the results of calculation or set value.



11) This completes setting the zero offset value.

Note 1: *Never move the turret in the Z-axis direction until zero offset setting is complete.*

Note 2: *For the X-axis, the reference point does not change even when the chucking method or setup changes. Therefore, there is no need to carry out zero offset each time the set up changes.*

Note 3: *Use a tool with offset values of X=0, Z=0, where practicable, for zero offset setting. If the tool offset values are not zero, the zero offset setting procedure will differ from the procedure indicated above. Refer to the following page.*

When a tool with tool offset values is used to set the zero offset value:

The numeral data to be entered through the keyboard is the sum of "measured value" and "tool offset value".

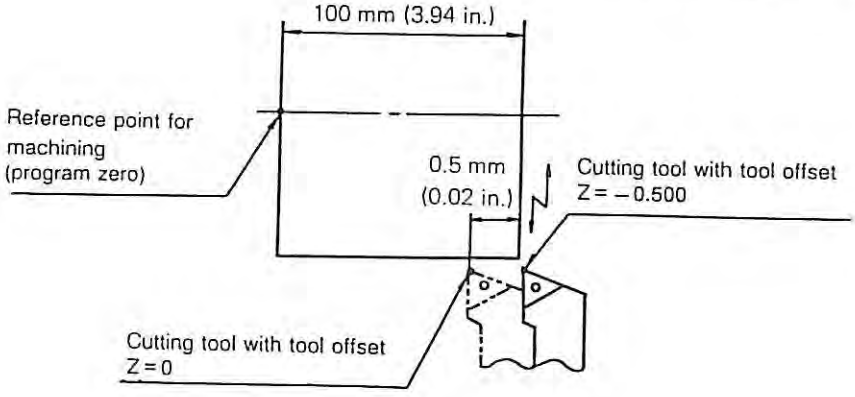
Setting Value = Measured Value + Tool Offset Value

Example: A tool with a tool offset value of Z = −0.500 is used.

The zero offset value is calculated as

100.000 + (−0.500) = 99.500

Setting value
Tool offset value
Measured value



If the workpiece length is 100 mm when it has been cut using a tool with a tool offset of Z = − 0.500 mm, the position of the tool with a tool offset of Z = 0 is 99.5 mm from the reference point (program zero).

A procedure that does not require a modification of the set zero offset value is described below.

1) Carry out steps 1) and 2) as explained before.
2) Enter tool offset data to tool offset #1 register.

   To enter tool offset, refer to 3-2-3.
3) Cut the end face of the part by moving only the X-axis in the MDI mode with T01 01 active. (Refer to 3-2-5.)

T01 01
↗ ↖
Tool No. Tool offset No.

4) Carry out steps 3) through 9) as explained before.

With the procedure above, it is not necessary to modify the set zero offset value by taking the tool offset value into consideration.



Never reset the control after cutting the part in the MDI mode.

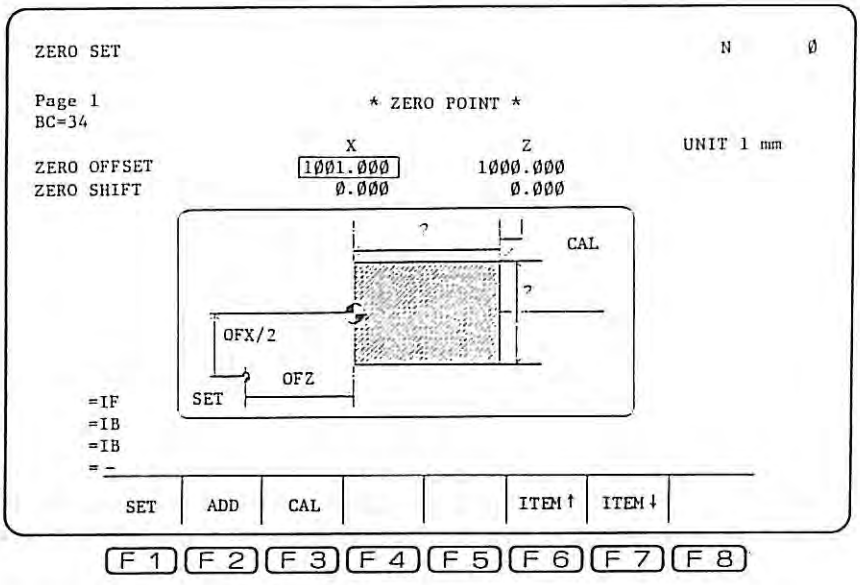
REMARKS

The OSP5020L/OSP500L-G has four different types of display. They are;

- Color graphic display
- Monochrome graphic display
- Semi-graphic display
- Monochrome character display

Display screens used in the explanation on the following pages assume the graphic display screens. If the color graphic screen is used, display is provided in color. With the monochrome character display, guide display is not available. The semi-graphic display screen provides virtually the same level of display capability as the graphic display screen except that the chuck jaws are not displayed and cutting tool shape is different from the actual shape.

See the example display below.



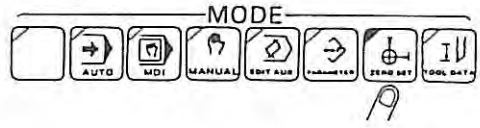
b) Where the zero offset value is known:

To set zero offset value of X-axis, proceed as follows:

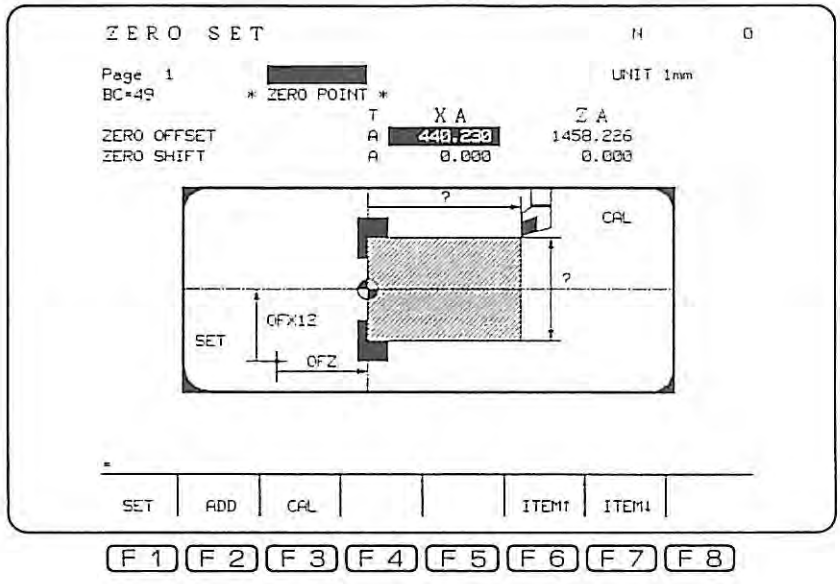
Example: X_0 = 450.230
Z_0 = 1400.000

Procedure:

1) Select the ZERO SET mode by pressing the **ZERO SET** key.



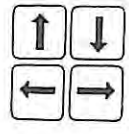
2) The CRT display is as shown below.



3) Select the turret, either A- or B-turret (for two-saddle and two-turret models).

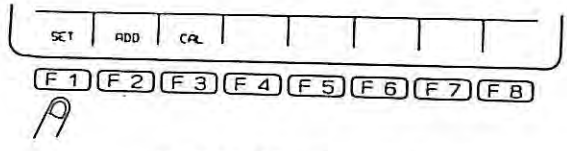
Each time  or  is pressed, turret A and B is selected alternately.

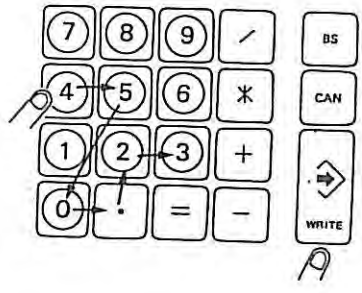
4) With the cursor controls keys, move the cursor to the data column of ZERO OFFSET - XA.

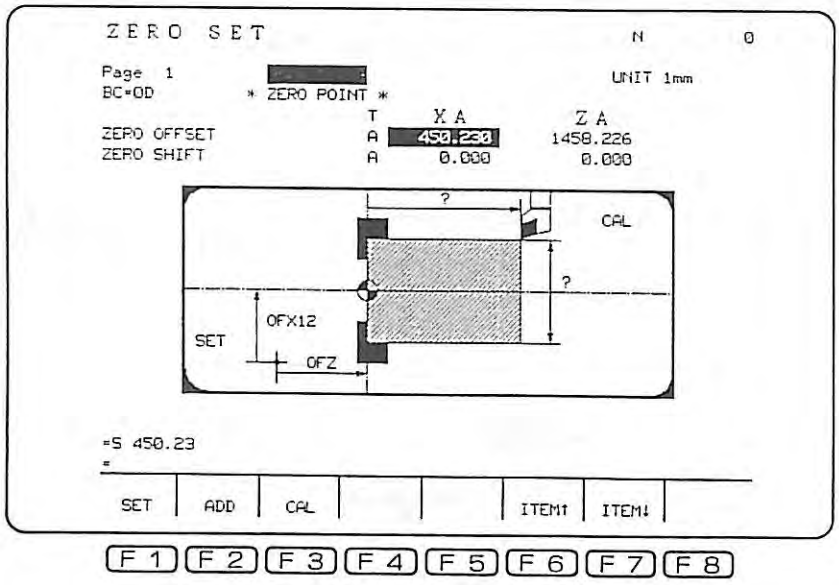


5) After pressing function key [F1] (SET), key in [4][5][0][.][2][3] through the keyboard.

6) Press the **WRITE** key.







With the steps indicated above, keyed-in zero offset value is stored in the zero offset area of the memory.

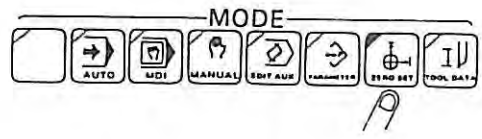
For Z-axis zero offset entry, the same procedure applies.

c) Where the stored zero offset value is to be modified:

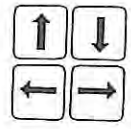
Example: X_0 = 450.230 . . . . . to subtract 10.000
Z_0 = 1400.000 . . . . . to add 10.000

Procedure:

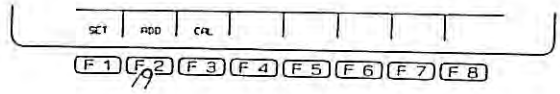
1) Select the ZERO SET mode by pressing the **ZERO SET** key.



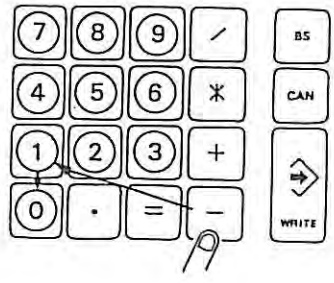
2) With the cursor control keys, move the cursor to the data column ZERO OFFSET - XA .



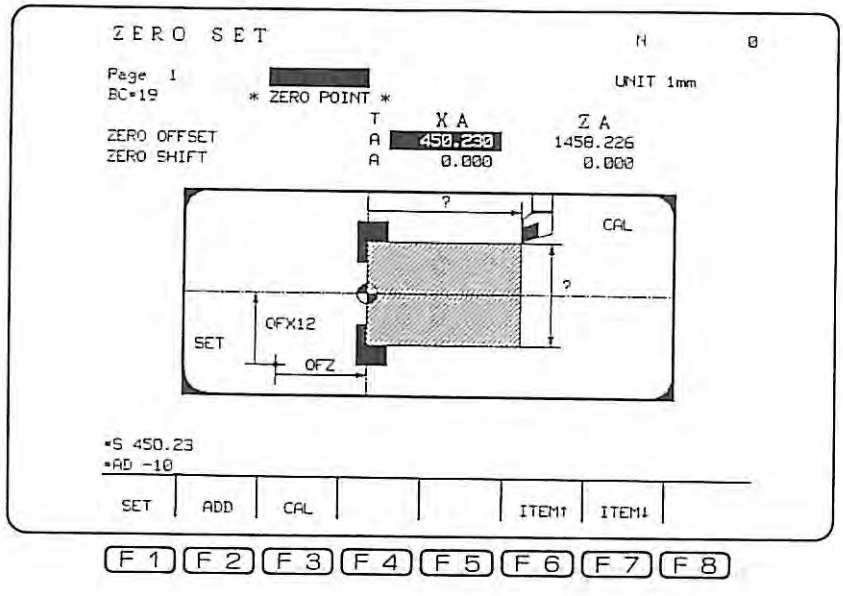
3) Press the function key [F2] (ADD).



4) Key in [−][1][0] through the keyboard.



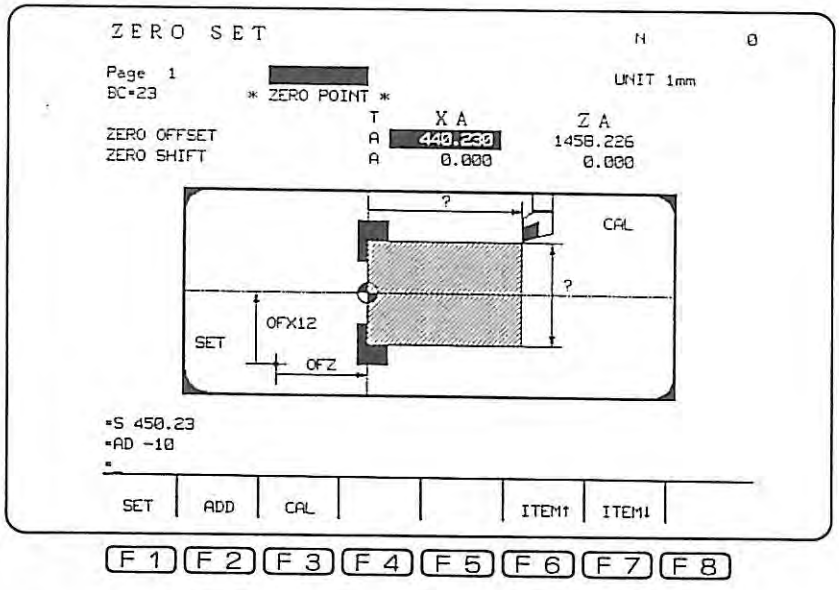
5) The corresponding CRT display is:



6) Press the **WRITE** key, and the CRT display changes as shown below.

With the **WRITE** key pressed, the following calculation is performed in the control and the result is stored as the X-axis zero offset value.

450.230 + (– 10.000) = 440.230



For Z-axis zero offset entry, the same procedure applies.

3-2-3. Setting Tool Offsets

(1) What is tool offset?

As illustrated in the foregoing diagrams, the reference dimensions or datum positions for individual cutting tools and holders have been predetermined for each type of turret mounted on the machine.

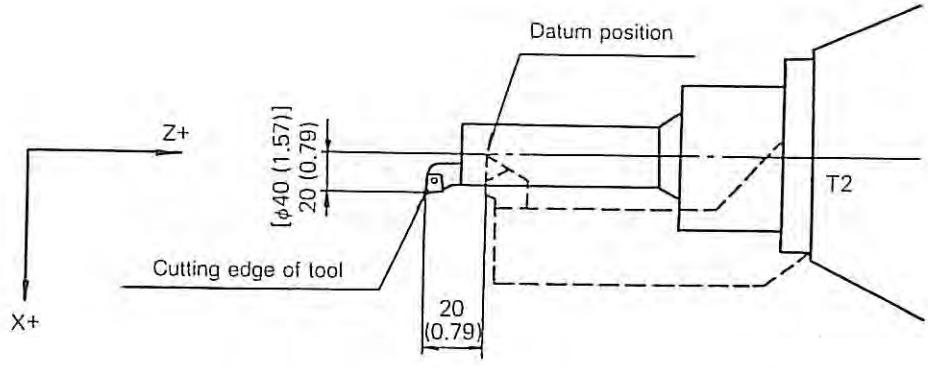
Tools and holders are often mounted in positions different from these datum positions. When installing the tools in the turret, it is time consuming to set the cutting point of each tool to the exact datum position. During operation, the tool point will also wear, resulting in over-sized parts.

On the other hand, the NC program has been prepared on the assumption that all cutting tools to be used are set exactly to the datum positions. This is because the programming procedure becomes extremely difficult if variable factors such as setting errors and tool wear are taken into consideration. It is thus necessary to correct the differences with respect to the required setting.

These differences or the amount of correction to be made are called "tool offset values".

Adjustments of tool positions are necessary before running the first part, after the first cut, and during operation, so that compensation can be made for the amount of wear developed even if the tool has been initially set to the required setting. The control has a "Tool Offset" control which automatically corrects tool setting errors as each tool is indexed into position for cutting. Manual switches can be used to make up for these differences with each tool in the turret.

Example:



The end point of the tool is set in the following coordinate position with respect to the required setting or datum point:

Along X-axis . . . . . +40 mm (in diameter)
Along Z-axis . . . . . −20 mm

These differences are usually measured by the tool setting calipers.

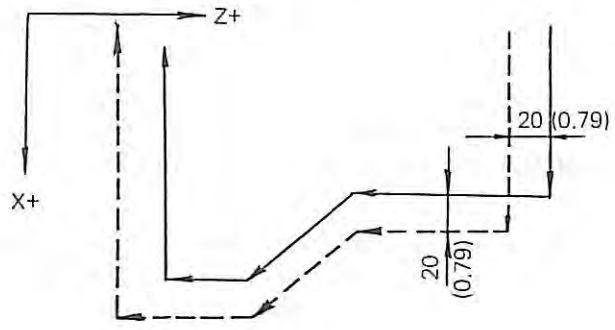
The tool offset values of this tool are:

X = −40.00
Z = +20.00

The tool offset values of the tool set at the datum point are:

X = 0
Z = 0

The tool paths of the tool are shown below:



← Tool path of the tool with tool offset X = 0, Z = 0

<--- Tool path of the tool with tool offset X = −40.00, Z = 20.00

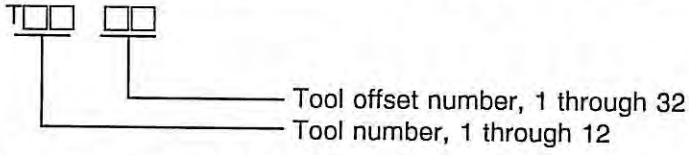
With the tool offset values stored in the control, compensation of the tool path is automatically made generating the path as programmed.

(2) Setting Tool Offset Values

When entering tool offset values, it is advisable to use a tool offset number identical to the number of the tool on which the offset function is activated.

Example: T0101 T0202

T word consists of "tool number" and "tool offset number":



There are three possible cases for entering tool offset values:

a) Where tool offset values are unknown, as when setting a tool on the turret for the first time.

b) When modifying the stored offset values.

This tool offset value setting procedure is effective when a workpiece is finished in incorrect dimensions due to tool wear.

c) Where tool offset values are known.

Explanation for each case is provided in this paragraph assuming that:

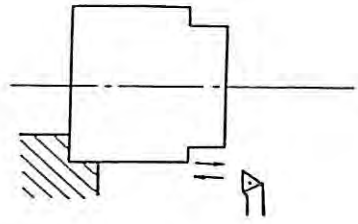
- numeral values are all expressed in units of 1 μm,
- setting of zero offset values of X- and Z-axis has been completed.

a) Where the tool offset value is unknown:

To set the zero offset value of X-axis (tool offset No.: 2):

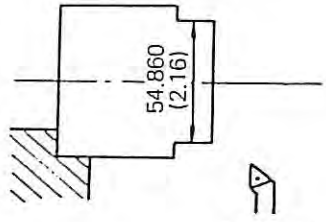
Procedure:

1) Turn the outside diameter of the workpiece with a proper depth of cut in the MANUAL OPERATION mode.



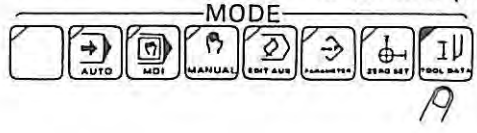
2) Measure the workpiece OD with a micrometer.

Assume that this dimension is measured as 54.860 mm.

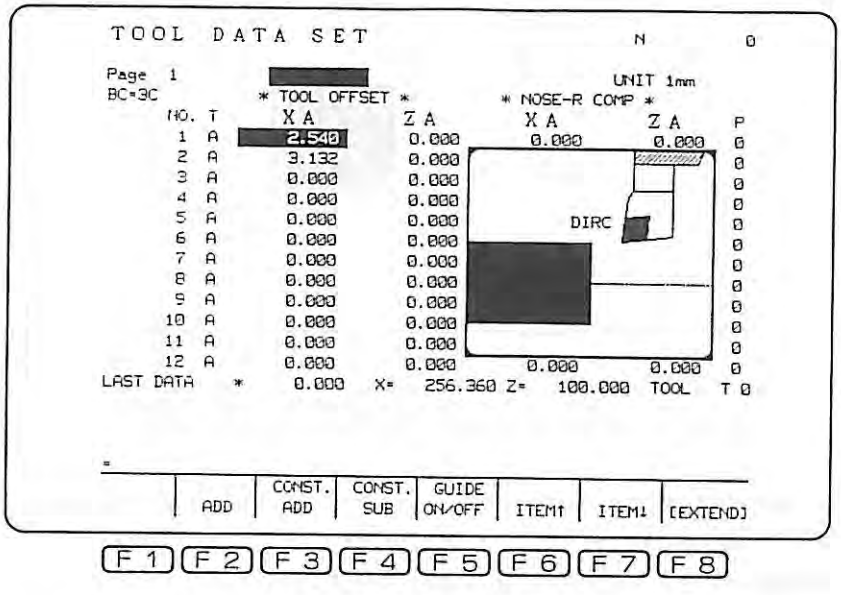


Unit: mm (in.)

3) Select the TOOL DATA SET mode by pressing the **TOOL DATA** key.



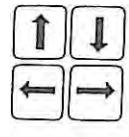
4) The CRT display is as shown below.



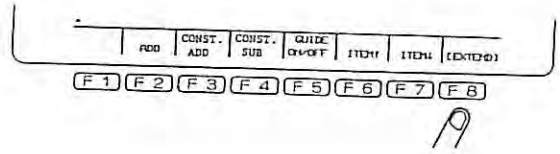
5) Select the turret, either A- or B-turret (for 2-turret/2-saddle models).

Each time  or  is pressed, turret A and B is selected alternately.

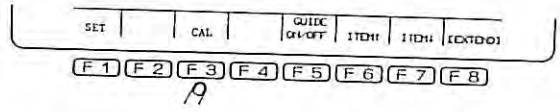
6) With the cursor control keys, move the cursor to the data column of No. 2 TOOL OFFSET - XA.



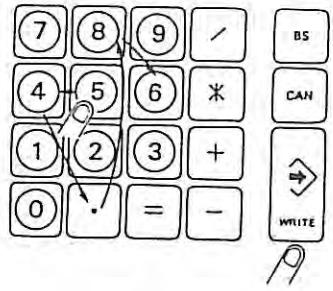
7) Press the function key [F8] (EXTEND) and make sure that the function name CAL appears at function key [F3].



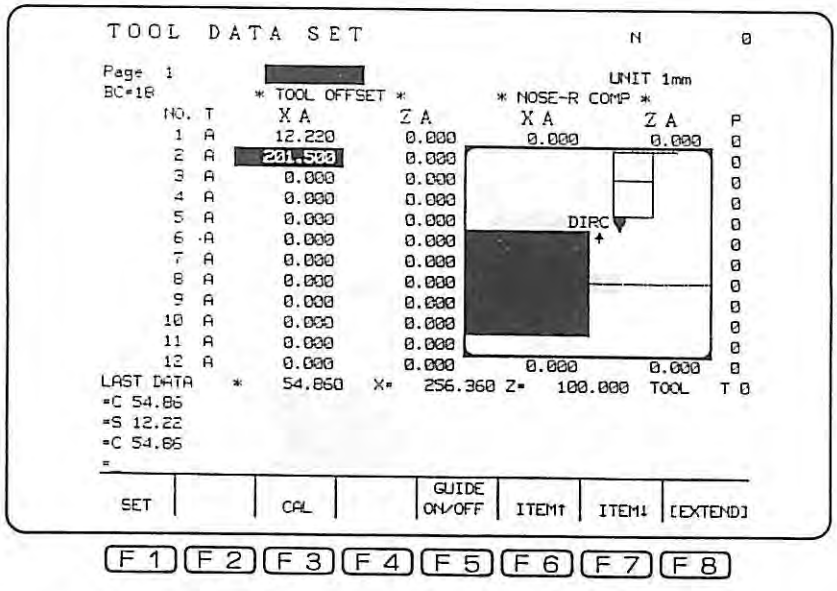
8) After pressing function key [F3] (CAL), key in [5][4][.][8][6] through the keyboard.



9) Press **WRITE** key.



The CRT display is as shown below.



With the steps indicated above, the tool offset value is automatically calculated and stored in offset No. 2 area of memory.

Note 1: *Never move the turret in the X-axis direction until tool offset setting of the active tool is completed.*

Note 2: *Perform the same procedure for the Z-axis and other tools.*

Note 3: *Calculation of the tool offset value is carried out in the equation below:*

$$\text{Tool Offset Value} = \text{Calculated Value} - \text{Zero Offset Value} - \text{Input Value}$$

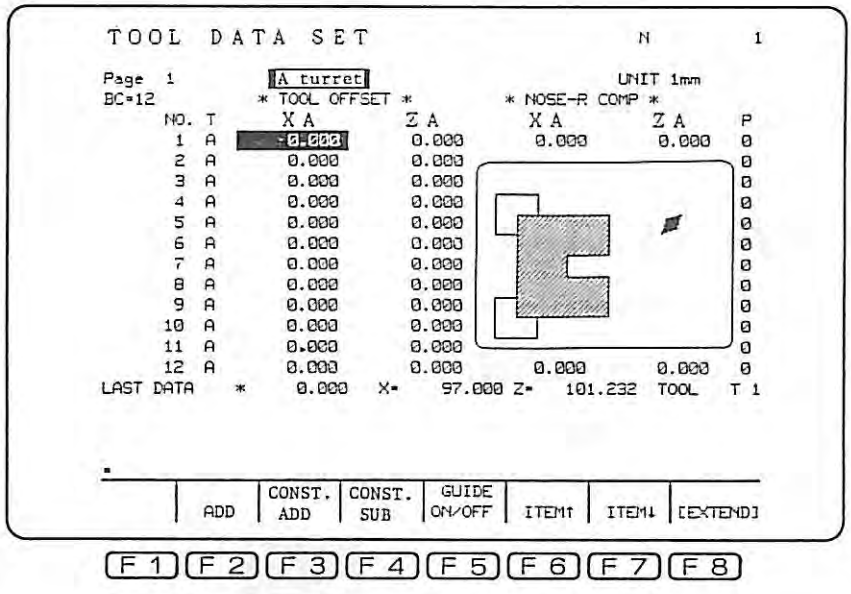
(Actual tool position) *(Measured value with micrometer)*

The OSP5020L/OSP500L-G has four different types of display. They are

- Color graphic display
- Monochrome graphic display
- Semi-graphic display
- Monochrome character display

Display screens used in the explanation on the following pages assume the graphic display screens. If the color graphic screen is used, display is provided in color. With the monochrome display is not available. The semi-graphic display screen is as shown below and for tool shape, only tip is displayed.

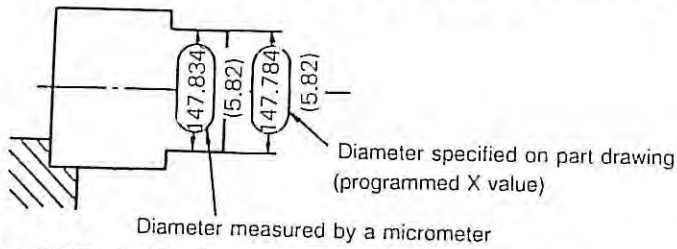
Guide display can be turned off for the graphic and the semi-graphic CRT by pressing the function key [F5] (GUIDE ON/OFF).



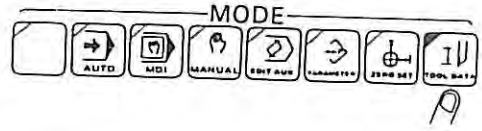
b) To modify the stored tool offset value (ADD):

OD cutting with the offset value in No. 2 memory area finished the workpiece 0.05 mm larger than the programmed dimension. To modify the tool offset value, proceed as follows:

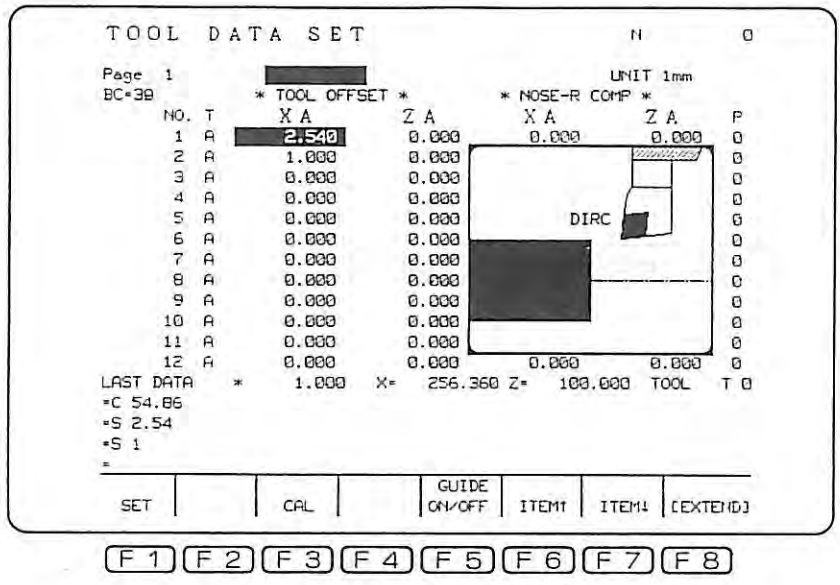
Note: *The limit value for the ADD operation can be set within a range of 0 - 1 mm (or 0 - 0.1 in.) with the parameter (long word) No. 33. This limit, however, can be canceled by setting a proper parameter data (details should be referred to Section 4, 3-6. "Optional Parameter (Bit)").*



1) Select the TOOL DATA SET mode by pressing the **TOOL DATA** key.



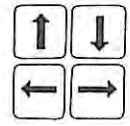
2) The CRT display is as shown below.



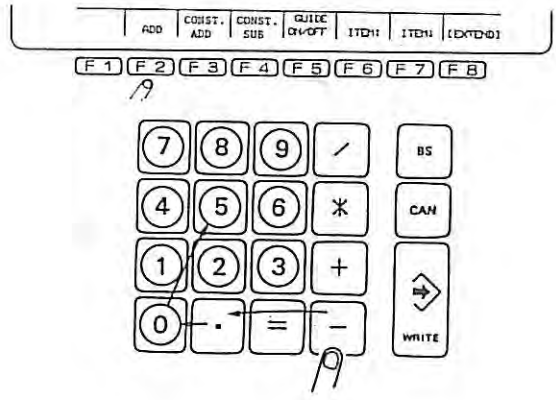
3) Select the turret, either A- or B-turret (for two-saddle and two-turret models).

Each time  or  is pressed, turret A and B is selected alternately.

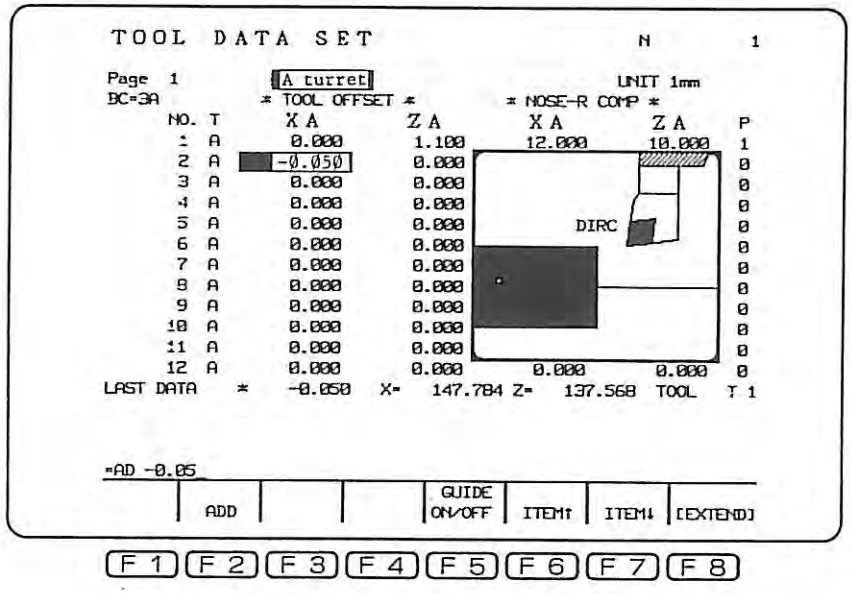
4) With the cursor control keys, move the cursor to the data column of No. 2 TOOL OFFSET - XA.



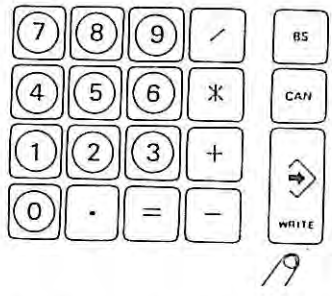
5) After pressing the function key [F2] (ADD), key in [−][0][.][0][5] through the keyboard.



The CRT display is as shown below.



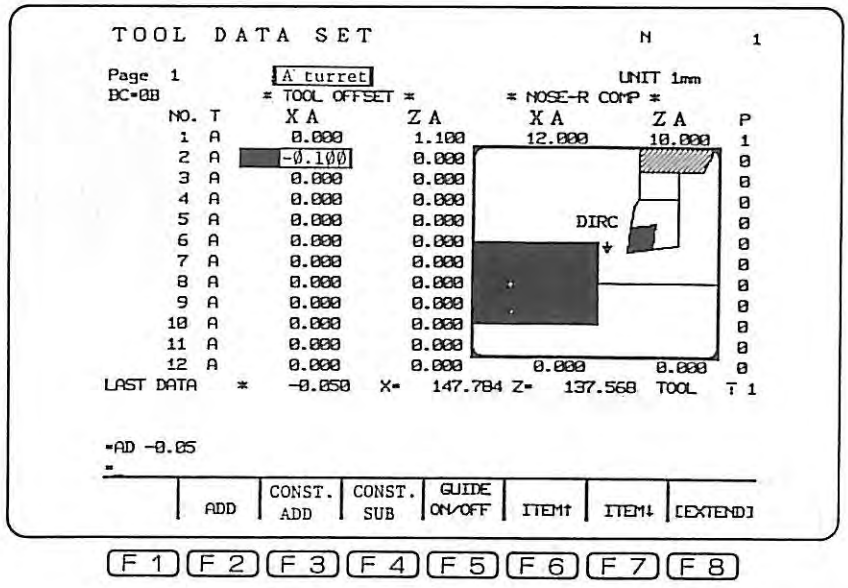
6) Press the **WRITE** key.



7) With the steps indicated above, calculation for the new tool offset data is made according to the equation below, and the result of the calculation is stored in No. 2 memory area, thus renewing the data.

New Offset Value = Old Offset Value + Keyed-in Data
−0.100 = −0.050 + (−0.050)

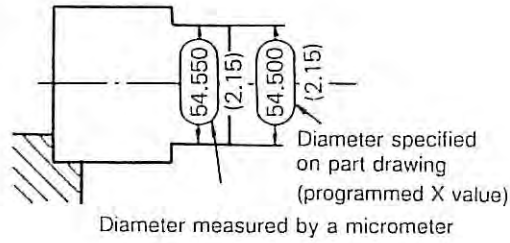
8) The CRT display is as shown below.



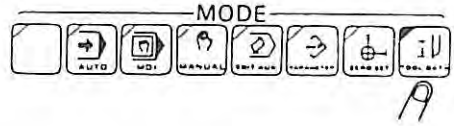
c) To modify the stored tool offset value (CONST. ADD), (CONST. SUB):

To alter the tool offset values using the functions CONST. ADD and CONST. SUB, preset a fixed amount by the parameter and add or subtract the preset value to and from the original tool offset value by simply pressing the function key [F3](CONST. ADD) and [F4] (CONST. SUB). This fixed amount is set to the optional parameter (long word) No. 32 in units of 1 μm within a range of 0 - 1000 μm.

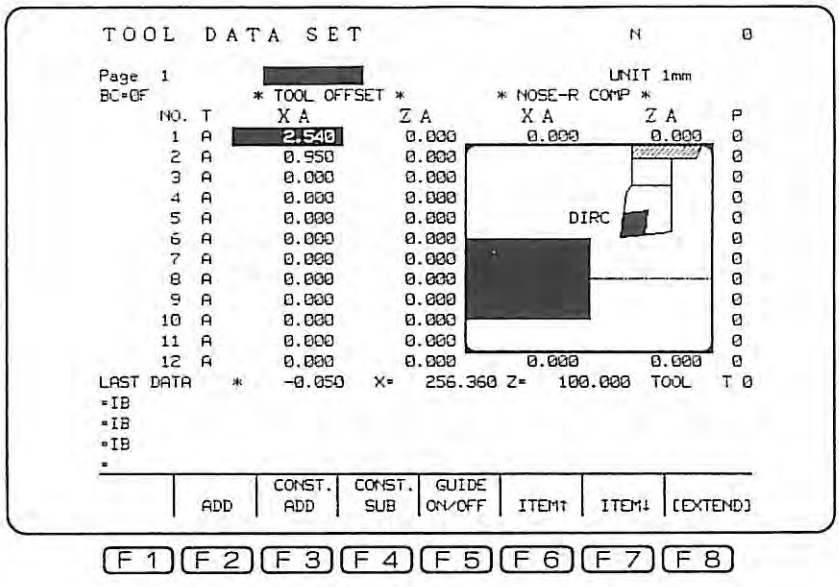
In turning operation using tool offset number 2, the finished outside diameter is 0.05 mm larger than the commanded value. Thus, the offset value is required to be compensated.



1) Select the TOOL DATA SET mode by pressing the **TOOL DATA** key.



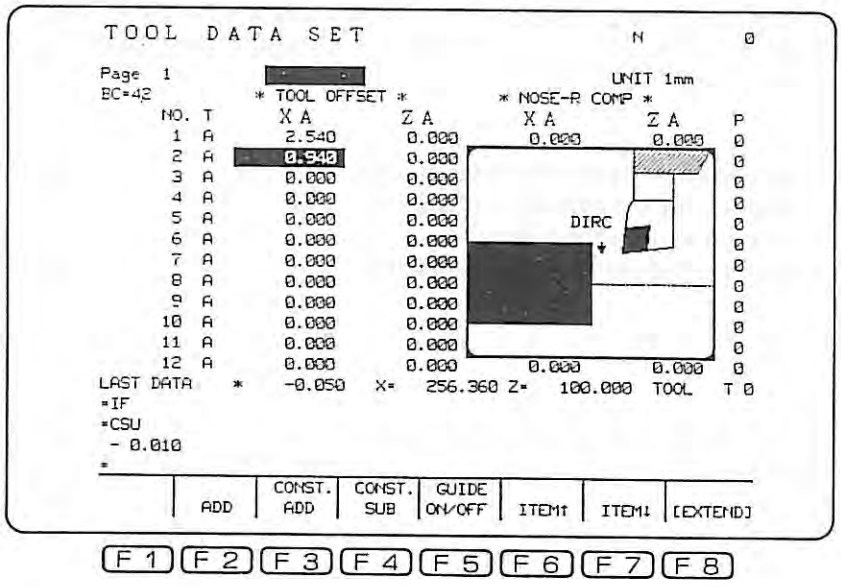
2) The CRT display is as shown below.



3) Select the turret, either A- or B-turret (for two-saddle and two-turret models).

Each time or is pressed, turret A and B is selected alternately.

4) With the cursor control keys, move the cursor to the data column of No. 2 TOOL OFFSET - XA.



5) Press the function key [F4] (CONST. SUB).

The following calculation is made in the control and the result of the calculation is stored in X-axis tool offset data No. 2, updating the previous data.

Assume the tool offset data in No. 2 be "– 0.100" and the data set at parameter (long word) No. 32 be 10 (μm).

New Offset Data = Old Offset Data + Setting of Optional Parameter (long word) No. 32

– 0.110 = – 0.100 – 0.010

Therefore, if the function key [F4] (CONST. SUB) is pressed five times, the tool offset data is updated to "– 0.150". This means that the tool offset data has been compensated by "– 0.050 mm".

> In turning operation using the tool offset No. 2, the outside diameter was finished 0.03 mm smaller than the commanded value. Therefore, this error must be compensated for by changing the offset data.

In this case, use the function key [F3] (CONST. ADD) instead of [F4](CONST. SUB). Pressing it three times can compensate for the tool offset data.

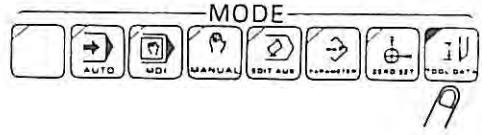
Note: *When the CONST. ADD and CONST. SUB functions are used, addition and subtraction are possible disregarding of the limit value set with parameter (long word) No. 33.*

d) Case where tool offset value is known:

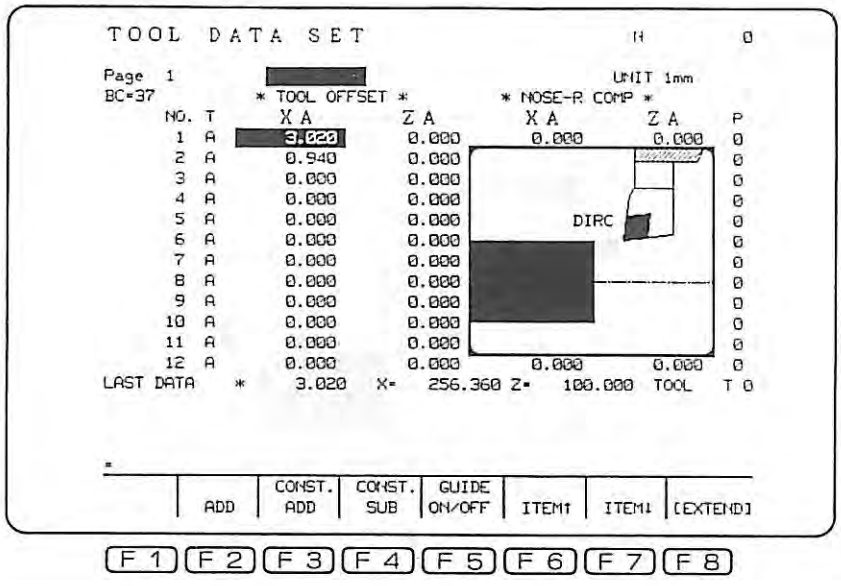
To enter the known tool offset value as X = 2.540, proceed as follows:

Procedure:

1) Select the TOOL DATA SET mode by pressing the **TOOL DATA** key.



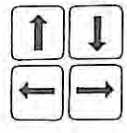
2) The CRT display is as shown below.



3) Select the turret, either A- or B-turret (only for two-saddle model)

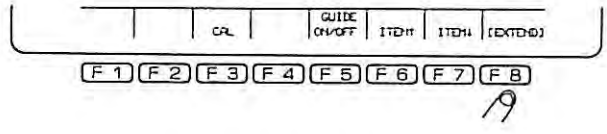
Each time [A] or [B] is pressed, turret A and B is selected alternately.

4) With the cursor control keys, move the cursor to the data column of No. 1 TOOL OFFSET - XA .

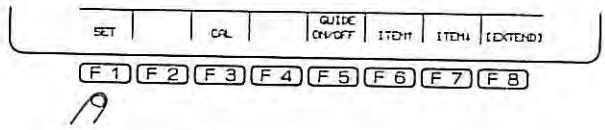


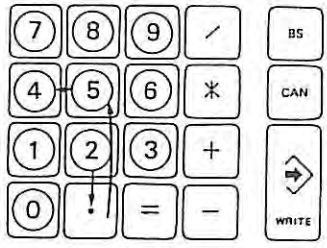
5) Press the function key [F8] (EXTEND).

Make sure that the function SET is displayed for the function key [F1].

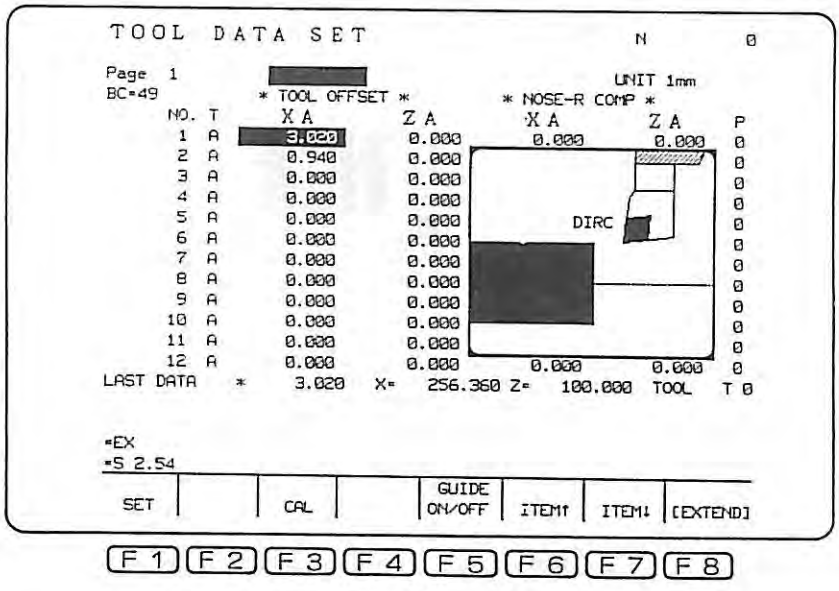


6) After pressing the function key [F1] (SET), key in [2][.] [5][4] through the keyboard.

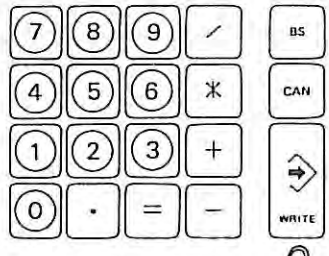




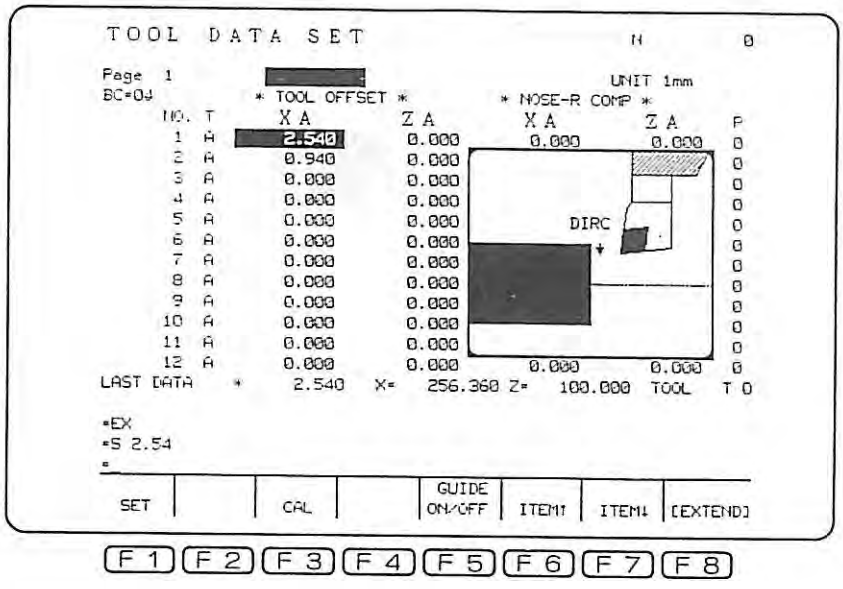
The CRT display is as shown below.



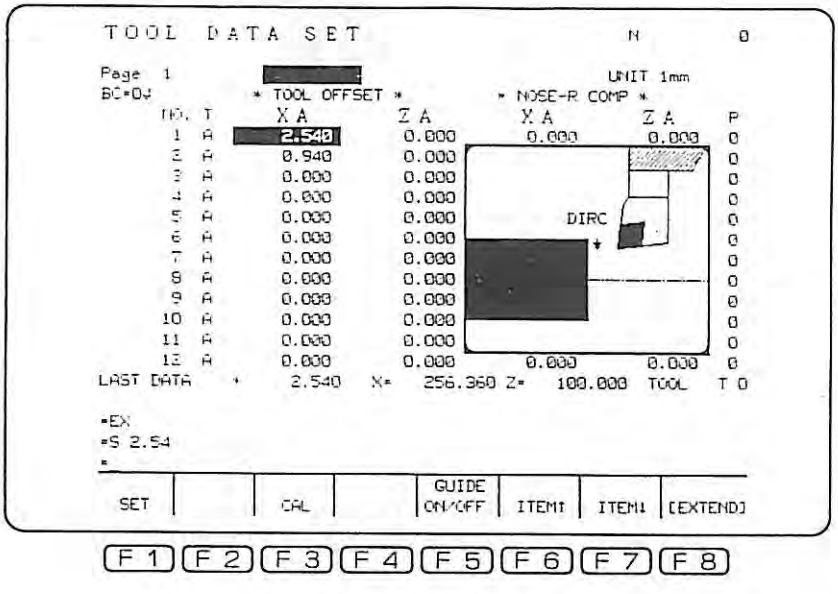
7) Press the **WRITE** key.



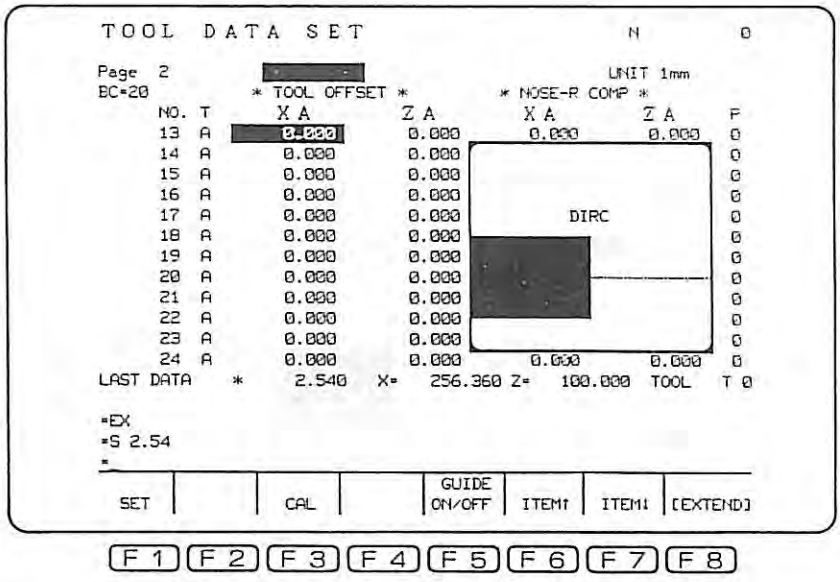
8) With the steps indicated above, keyed-in data is stored at NO. 1 TOOL OFFSET - XA data column.

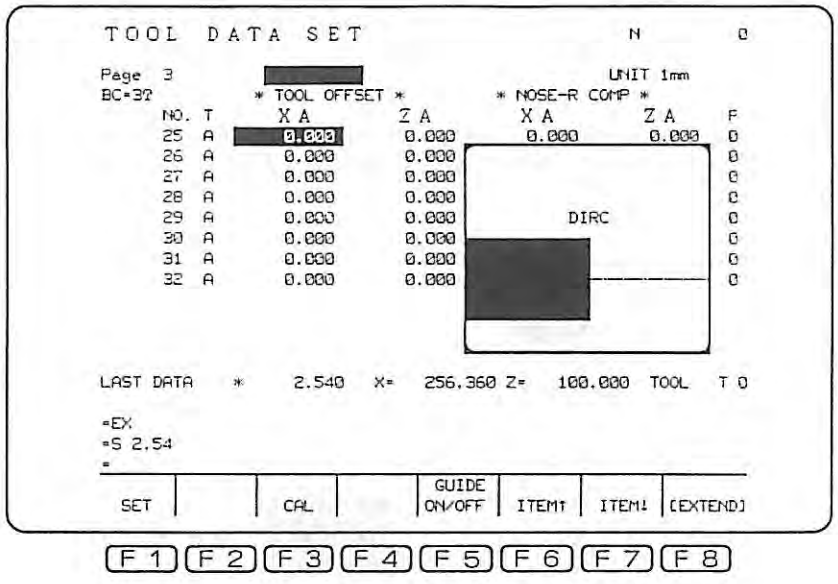


Note 1: *There are 32 pairs of tool offset data for each of A- and B-turret. One page of the CRT screen can show 12 pairs of offset data.*



To obtain the data of other offset numbers, press ***PAGE*** *key,*  *or*  *, and the display changes as below each time the key is pressed.*

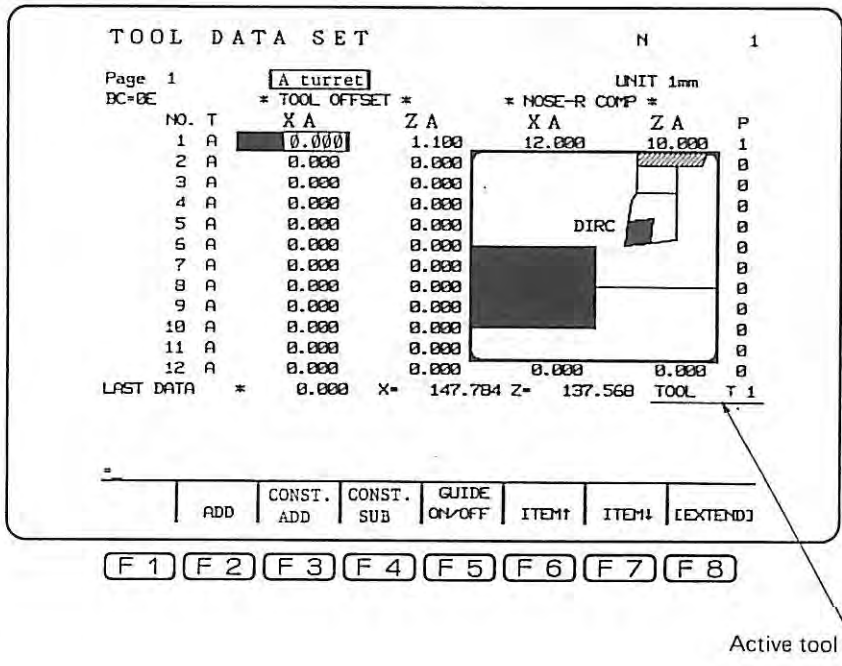




Note 2: *Never turn off power supply to the control right after entering or changing the tool offset and/or zero offset data. Be sure to keep the control ON for at least two to three minutes after that. Should power supply be turned off while entering or changing the data or right after that, check the newly data since the data might not be renewed.*

(3) Display of TOOL NO. on the CRT Screen

Tool numbers are displayed on the CRT screen (TOOL OFFSET screen and ANIMATED DISPLAY screen) in the manner as indicated below. This allows an operator to check the tool number of the tool being used without switching the display screen.



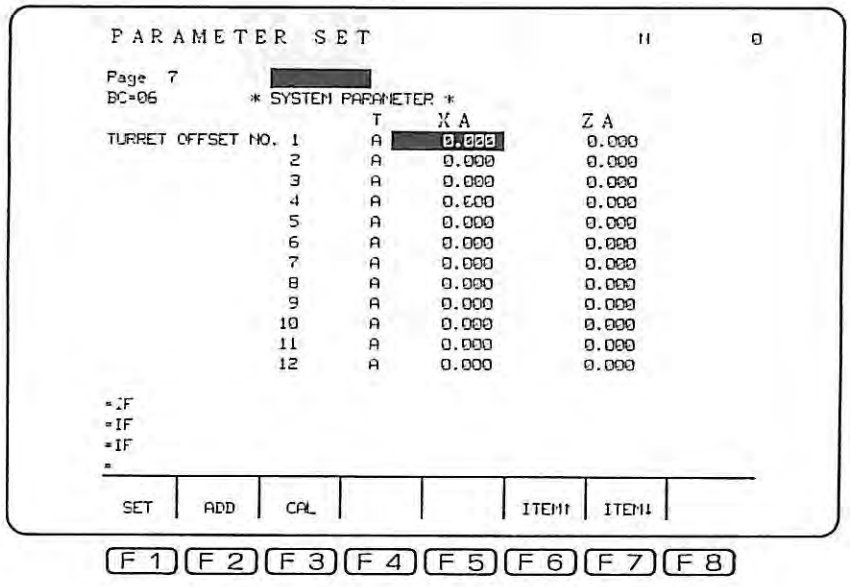
3-2-4. Setting Turret Position Error Compensation Value

(1) What is the Turret Position Error Compensation Value?

With the quick change turret, the tool can be set at any turret position easily. However, because the turret has an indexing error, the tool offset value measured by setting a tool in one of the turret position cannot be used if the same tool is set in a different turret position. The error between turret positions is called the turret position error and the function to compensate the error is called the turret position error compensation function.

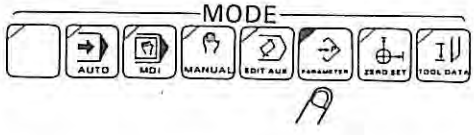
(2) Turret Position Error Compensation Value Setting Screen

The turret position error compensation value is set at the system parameter screen in the parameter mode.



The procedure to display the system parameter screen is indicated below:

1) Select the PARAMETER SET mode by pressing the **PARAMETER** key.



2) Press the function key [F7] (ITEM ↓) until the SYSTEM PARAMETER screen is displayed.

3) Press the **PAGE** key until the TURRET OFFSET screen is displayed.

(3) Setting Turret Position Error Compensation Values

Procedure:

1) Mount a reference tool in the turret position which is taken as the reference turret position.

   The turret position error compensation value for the reference position is taken as "0".

2) After turning the workpiece OD properly, measure the OD with a micrometer. Record the measured value and the actual position data.

   Example: Measured OD 54.860
   Actual position data 60.000

3) Index the turret to the position for which the turret position error compensation value is to be set. Mount the reference tool to the indexed turret position.

4) Turn the workpiece with the reference tool until the actual position data on the screen reaches the value as read in step 2). Measure the diameter of the turned workpiece.

   Example: Measured OD 56.960

5) Calculate the difference between the measured outside diameters and set the calculated result as the compensation value.

   Measured value (turret position for which compensation value is to be set) – Measured value (reference turret position)

   Example: 56.960 – 54.860 = 2.100

   Set "2.100".

Carry out the same steps for setting the compensation value in the Z-axis direction and also for other turret positions.

(4) Precautions

a) The turret position error compensation value becomes effective when the corresponding turret position is indexed.

b) For the two-saddle specification, the compensation values can be set for turrets A and B independently.

c) The turret position error compensation function is effective only for LC, LB, LR and LP series lathes. This function is not available with LS and LH series.

3-2-5. Automatic Operations

NC lathes are mostly operated in the automatic mode in which machine operations are controlled by taped commands. This section deals with the procedure for automatic mode operations assuming part program tapes have been prepared.

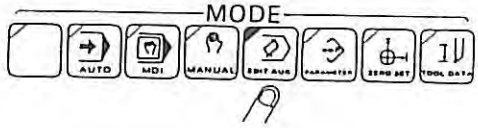
Note that automatic operation is possible with stored part programs and the automatic operation directly from a command tape is not possible.

Before starting the operation, refer to 3-2-6. to ensure safe operations.

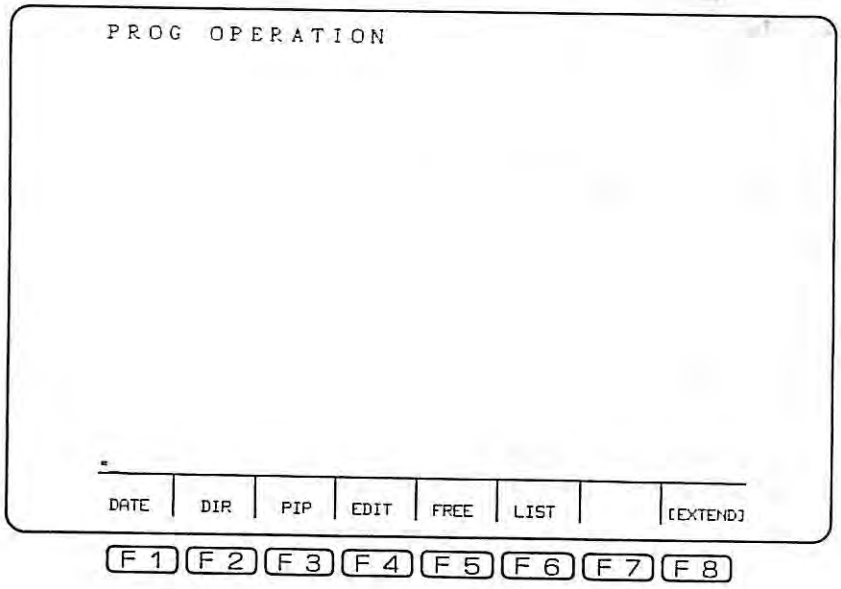
(1) Procedure to Store Part Programs into Memory

Procedure:

1) Select the PROG OPERATION by pressing the **EDIT AUX** key.



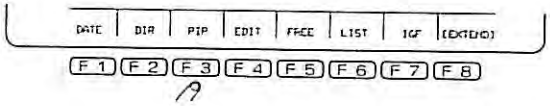
2) The CRT display is as shown below.



3) Set the part program tape in the tape reader following the steps indicated on page 77.

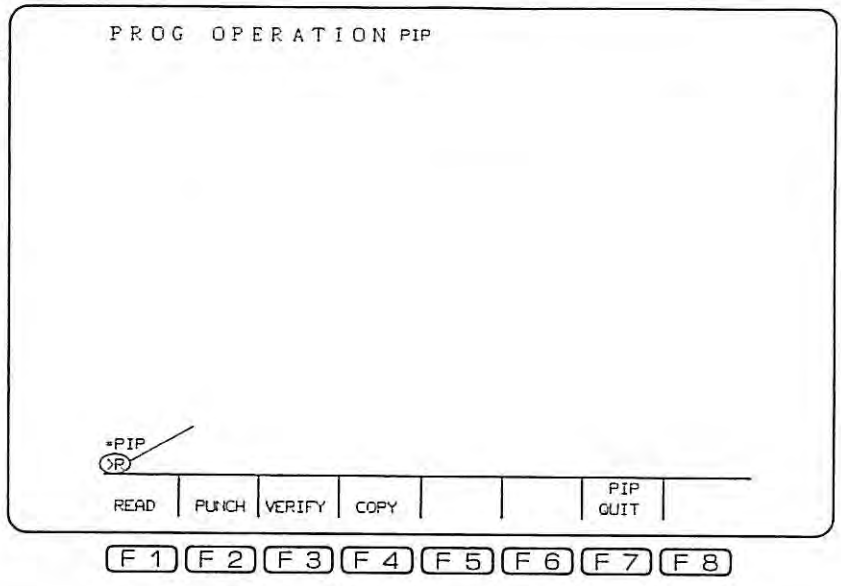
4) Press the function key [F3] (PIP).

The CRT shows PIP message.



5) Press the function key [F1] (READ).

With this the control is ready to read a part program tape.

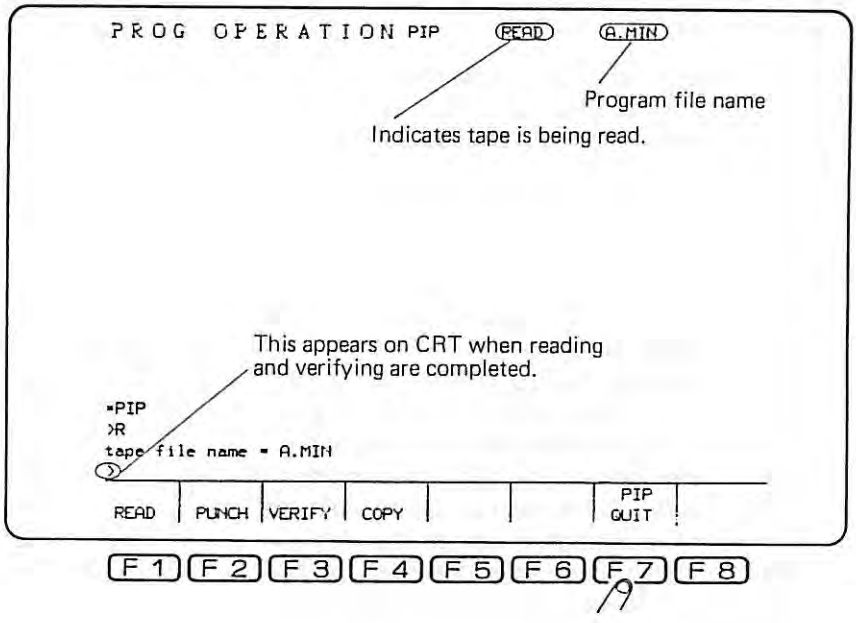


6) Press the **WRITE** key.

This initiates tape reading-in and storing operations. While the part program tape is being read, the CRT shows READ message and A.MIN (file name).

After the completion of tape reading-in and storage, the tape is fed backward to compare the stored part program data and the data on the tape. This comparison feature is called "tape verifying function".

When the tape reading-in and verifying is completed, the prompt ">_" appears right above the function key [F1] (READ).

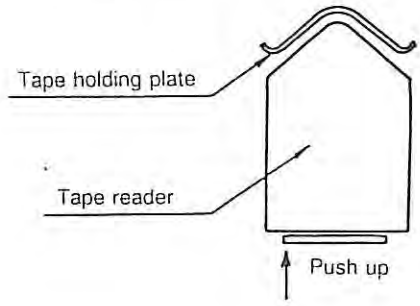


7) After making sure that tape storage is complete, press the function key [F7](PIP QUIT).

The tape storage operation is complete; the CRT display restores the state right after the **EDIT AUX** key is pressed.

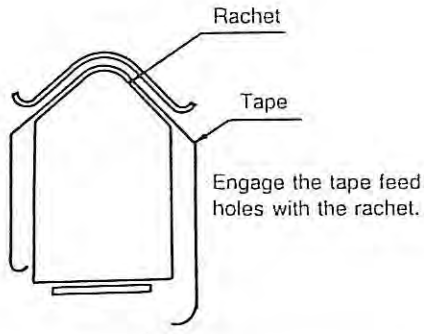
Setting Tape in Tape reader

1) Push up the plate which is under the tape reader to move up the plate holding the tape.



2) Place the tape to shown to the right.

Engage feed holes on the tape to match the sprockets.

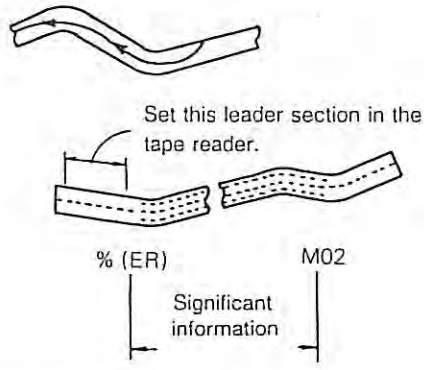


IMPORTANT

Use only "black" tapes. Never use tapes of other colors. That could cause malfunctions of the machine due to misreading.

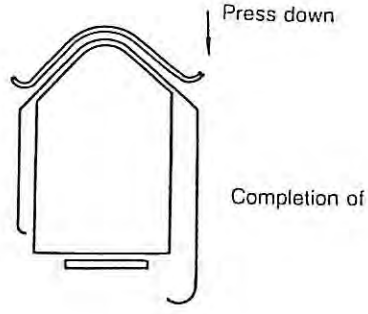
Set the tape with the white arrow pointing to the left.

Set the leader section containing only feed holes preceding the first % or ER code on the tape, in the tape reader.



3) Press down the tape holding plate.

Note: *The tape reader is equipped with the out-of-step detection mechanism. When the tape is pulled by a force greater than 2.9 N 300 g, the mechanism is activated. In this case, the PTR ERROR lamp in front of the tape reader comes on and the data can not be read. This error can be eliminated using the pushbutton under the lamp. When restarting tape read operation, make sure that no excessive force is applied to the tape.*



(2) Preparations for Automatic Operation

Before starting automatic operation, confirm the following points without fail.

a) After changing the chuck, always set the allowable chuck speed with the parameter.

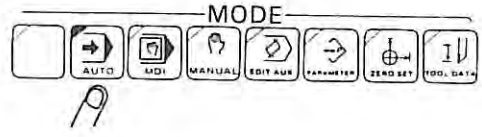
b) In a machining program, designate the maximum spindle speed following G50. This speed must be lower than the allowable chuck speed.

(3) Block-to-Block Operation (Single Block Operation)

In single block mode operation, part program blocks are executed block by block. This operation is useful for checking newly prepared part programs.

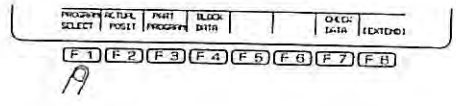
Procedure:

1) Select the AUTO OPERATION mode by pressing the **AUTO** key.



2) Press the function key [F1] (PROGRAM SELECT).

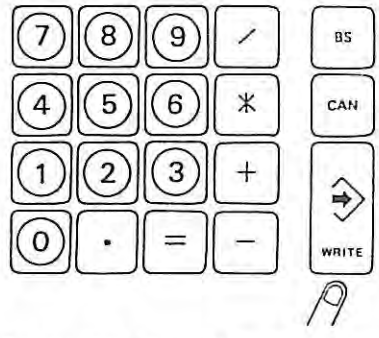
If the function PROGRAM SELECT is not assigned to [F1], press [F8] (EXTEND).



3) Press the **WRITE** key.

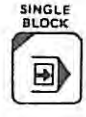
This selects the program to be executed. After the program selection has been completed, the display above [F1] on the CRT changes as shown below:

=PS → **=PS**
=



In case more than one program is stored in memory, refer to the procedure to select the desired program explained in Section 4.

4) Press the **SINGLE BLOCK** key.

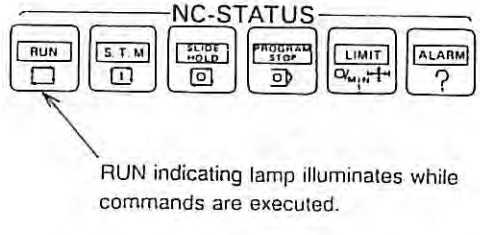


5) Press the **CYCLE START** button.

With this, part program blocks are executed one by one.

While the commands in the block are executed, the RUN indicating lamp illuminates. It goes off when the execution has been completed.

Press the **CYCLE START** button each time the execution of one block commands has been completed.



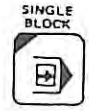
(4) Automatic Continuous Operation

After making sure that the operation with the new tape is correct, you can run the machine in the automatic continuous mode.

Follow the steps indicated below:

1) Perform steps 1) in (3) "Single Block Operation".

2) Press the **SINGLE BLOCK** key (SINGLE BLOCK function OFF).



3) Press the **CYCLE START** button.

The part program is read out from memory and executed continuously.

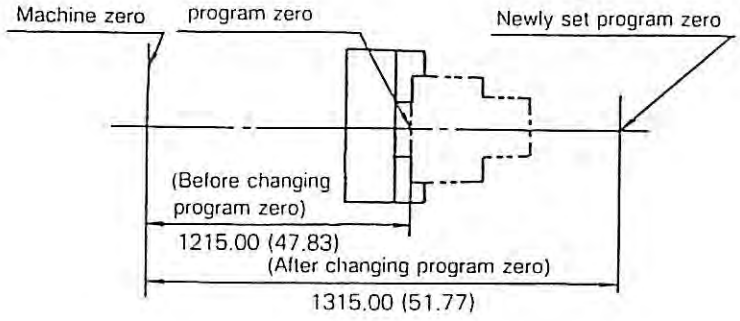
Note 1: *Before starting automatic continuous operation, be sure to check the chuck:*

The chuck must be closed for OD GRIPPING, and open for ID GRIPPING. Otherwise, the spindle will not start even when the ***CYCLE START*** *button is pressed.*

Note 2: *Before operating the machine with a new program, be sure to check the safety of axis movements by executing the program in the single block mode.*

Note 3: *It is also recommended to run the new program without setting a workpiece on the machine or shifting the zero point of Z-axis toward the tailstock side.*

Example:



By changing the zero offset value to "Z=1315.00" dimension words are executed offset by 100 mm in the positive direction of Z-axis from the programmed points.



CAUTION

When the tailstock is used, watch out for interference between the turret and the tailstock; retract the tailstock to avoid accidental collision between the turret and the tailstock.

(5) Display of Active Commands in Single Block Mode Operation

While checking programmed commands by executing them in the single block mode, the contents of active commands can be checked on the CRT by following the steps below:

Procedure:

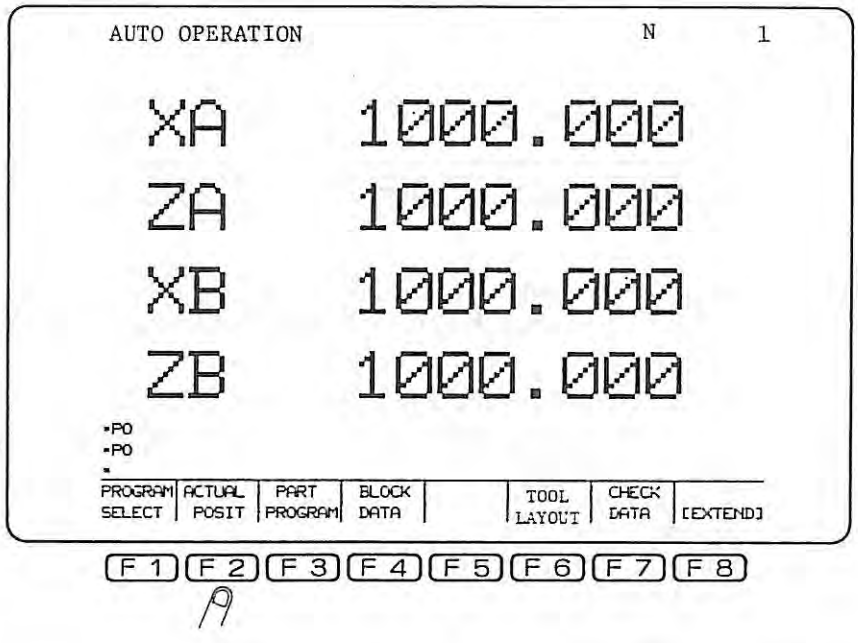
1) Press the function key [F2] (ACTUAL POSIT).

The actual position of X- and Z-axis can be displayed on the CRT in enlarged characters.

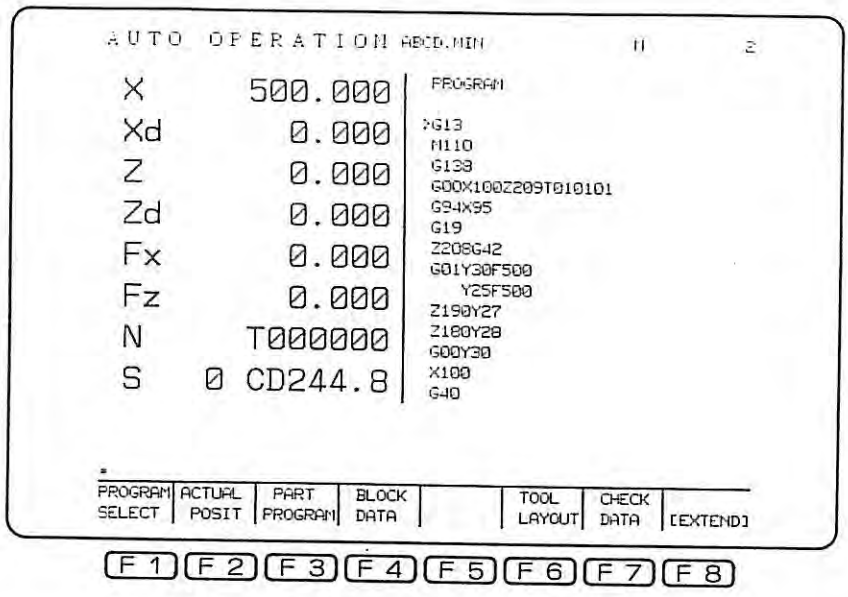
The actual position data means the value expressed as "ZA" in the figure below. It indicates the distance between the program zero and the tool tip point.

When the rear turret is selected on the two-turret model, the display is given by XB and ZB.

For two-saddle models, XA, XB, ZA and ZB are displayed at the same time (see below).



2) Press the [ ] key. This changes the display as shown below.



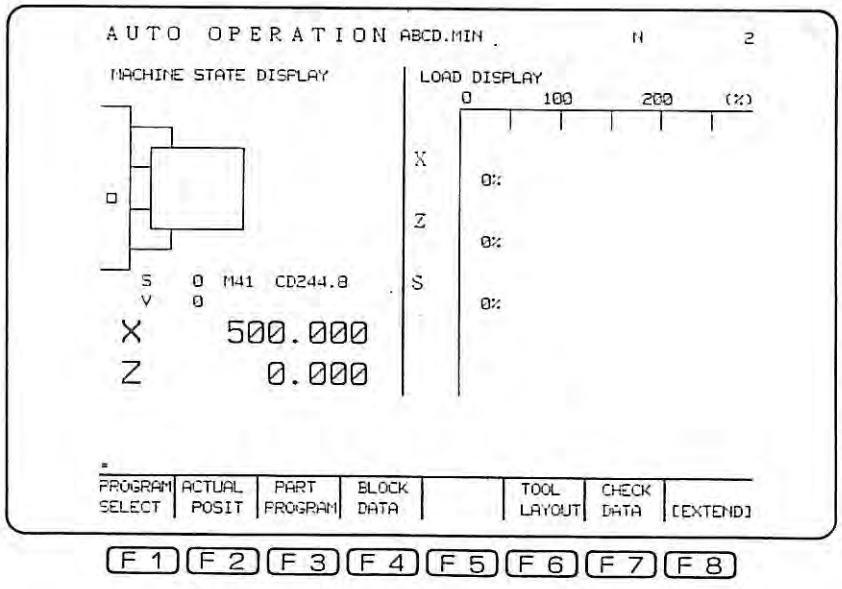
The CRT display is divided into two sections. Actual position and other related data is displayed on the left section, and the program presently selected is displayed on the right section.

X, Z	:	Distance between the tool tip point and the program zero
Xd, Zd	:	Remaining axis movement distance (= command value – actual position)
N	:	Sequence number
T	:	T command value
S	:	Spindle speed
CD	:	Actual position of spindle
FX, FZ	:	Axis feedrates

Display of actual spindle position can be omitted by setting "1" at bit 6 of parameter (bit) No. 19. (Initial setting: 0 (OFF))

3) Press the [P] key. This changes the display as shown below.

The CRT display is divided into two sections. Machine status, actual position of X- and Z-axis and other related data are displayed at the left section, and the loaded condition of the spindle, X- and Z-axis is displayed at the right section.



X, Z	:	Distance between the tool tip point and the program zero
T	:	T command value (Displayed only during cutting)
S	:	Spindle speed (min^{-1} {rpm})
V	:	Cutting speed (m/min)
CD	:	Actual position of spindle The angular position of the spindle calculated from the output of the pulse generator; with the single-phase pulse generator, display is " ***.* ".

Display of actual spindle position can be omitted by setting "1" at bit 6 of parameter (bit) No. 19. (Initial setting: 0 (OFF))

Semi-graphic guide display (machine status):

a) Spindle

While the spindle is rotating, symbol " ↻ " is displayed.

b) Chuck

Display shape varies depending on the selection of ID/OD gripping .

c) Work

Displayed only when the chuck is in the clamped state. The shape displayed varies depending on the selection of ID/OD gripping.

d) Tool

Displayed only while cutting is in progress.

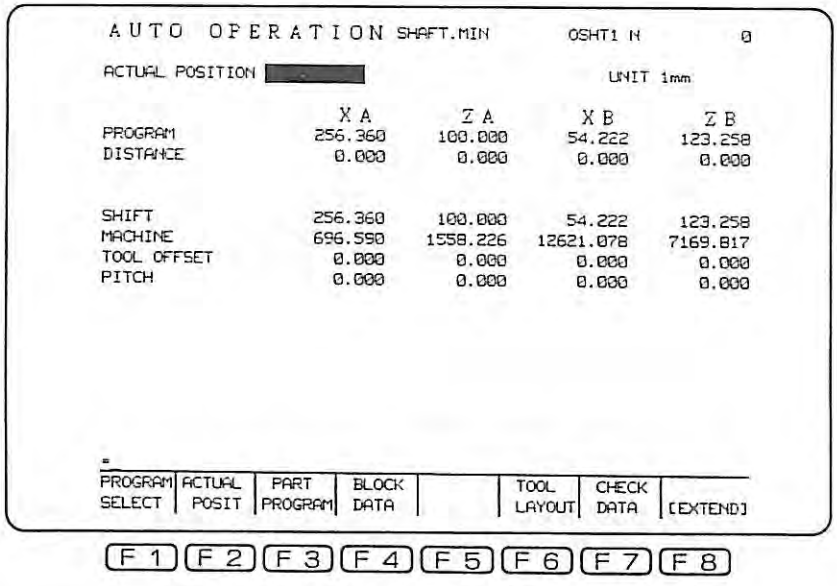
e) Tailstock spindle

Displayed only for the center work operation on the center work specification model. The tailstock spindle shape varies depending on the position of it -- advance end or retract end.

Load condition display:

Loaded condition of the spindle is displayed by bars. The bar is displayed in units of 10 percent and the numerical value is displayed below the bar. The value displayed is the average value of "n" samples collected in intervals of 12.8 msec. Value "n" can be set by parameter.

4) Press the [ ] key. This changes the display as shown below.



a) Active sequence number

b) PROGRAM (Z_A in the figure below)

Dimension between the program zero and the tool tip point.

c) DISTANCE (Commanded value − Actual position value)

This indicates the remaining axis motion distance.

d) SHIFT ($Z_A + Z_b + Z_t$ in the figure below)

Actual position data + Zero offset amount + Tool offset amount

e) MACHINE (Z_a in the figure below)

Output from OSP position encoder

f) TOOL OFFSET (Z_t in the figure below.)

This indicates the tool offset amount of the tool presently used.

g) PITCH

This indicates the active pitch error compensation value.

5) Pressing the [⇄] key again restores the CRT display as obtained in step 1). After that, each pressing of the key changes the CRT display alternately between 1) and 4).

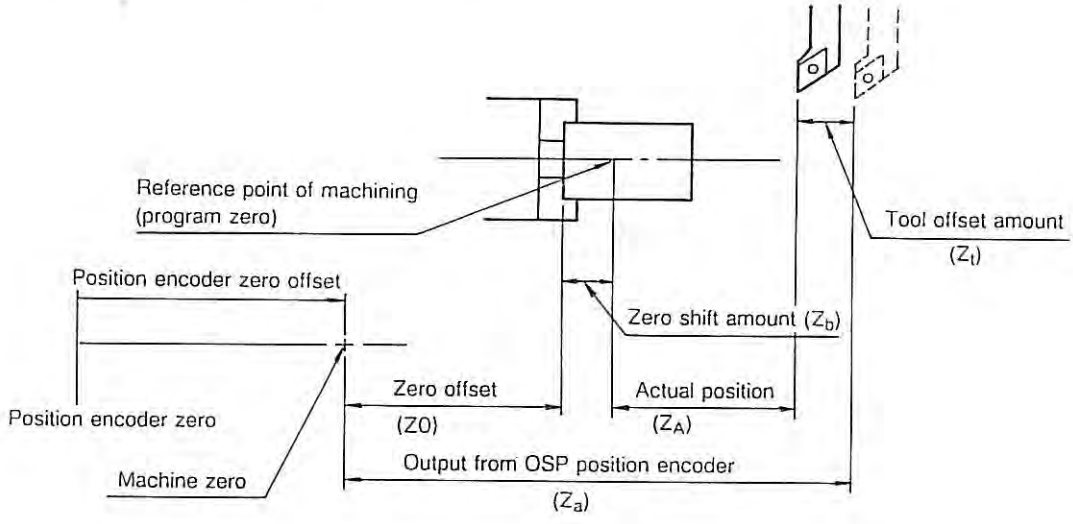


Fig. 3-3

Machine zero:

The position where the output from the position encoder becomes zero is usually called the machine zero.

Position encoder zero offset:

The error between the previously and currently mounted position encoder output zero positions is called the position encoder zero offset; this setting is necessary when the position encoder is replaced and thus it is usually "0".

Zero offset amount:

The distance between the machine zero and the program zero is called the zero offset.

Zero shift amount:

The zero shift amount is set to change the program zero.

Tool offset amount:

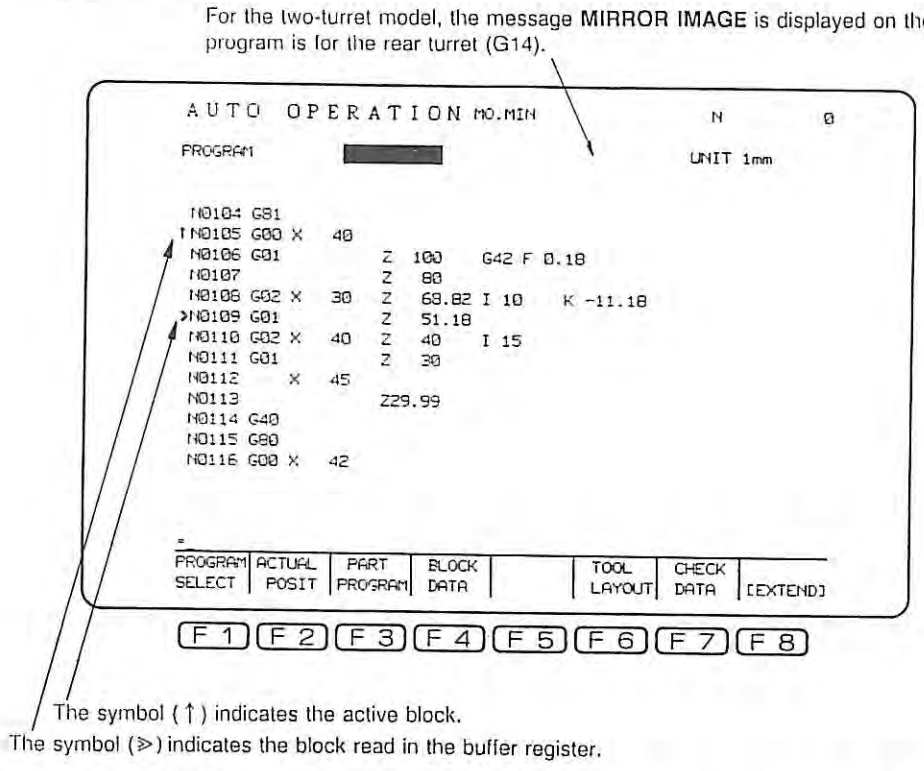
The tool offset for a reference tool is "0". For other tools, difference from the reference tool position is set as the tool offset amount.

(6) Display of Stored Program Data

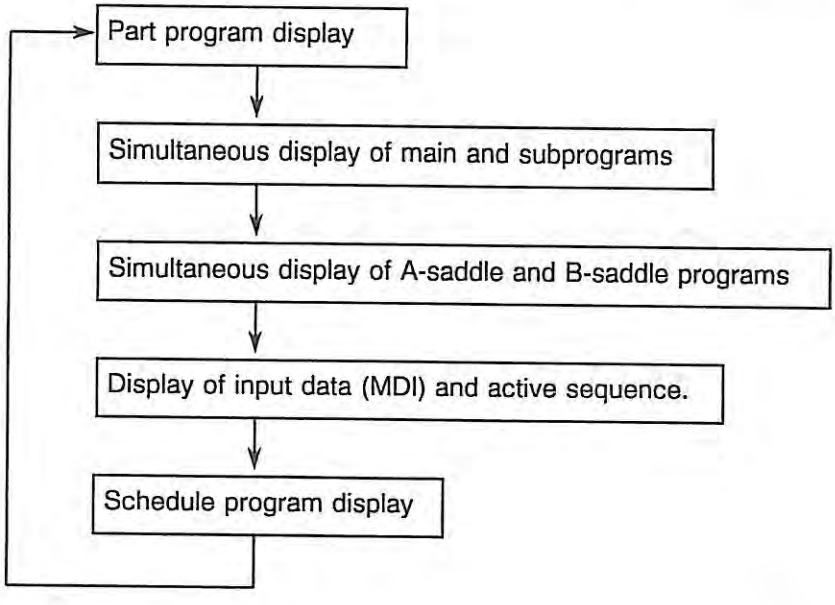
Stored program data can be displayed on the CRT:

Procedure:

1) Press the function key [F3] (PART PROGRAM). The programmed commands are displayed on the CRT.



2) Pressing the  key after that changes the CRT display in the order indicated below.



Note: *Details of part program display are provided in Section 4, "Application"*

3-2-6. Manual Data Input (MDI) Operation

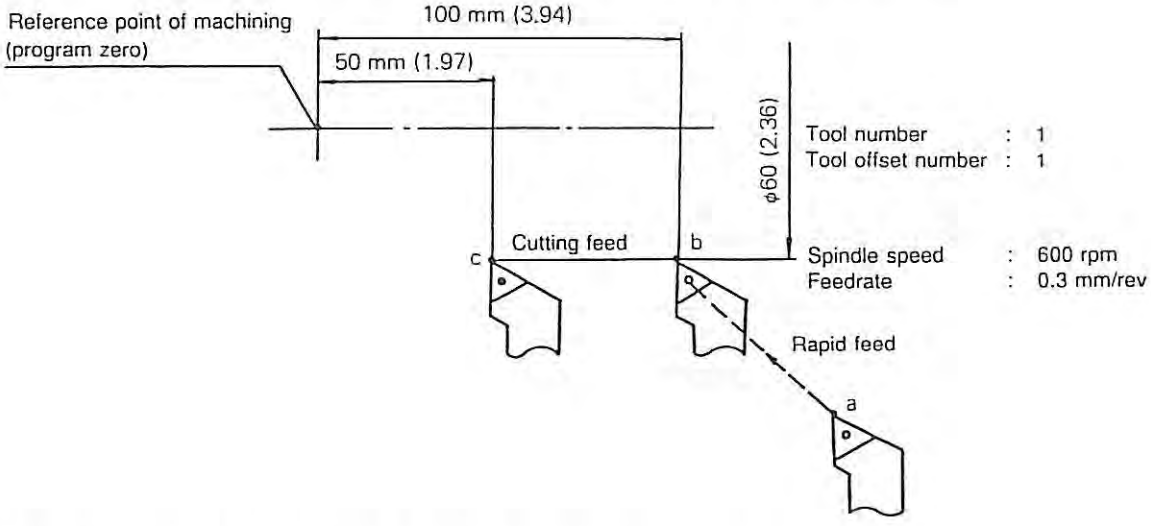
The operator can operate the machine with the commands entered through the keyboard instead of tape commands. This operation mode is called the Manual Data Input (MDI) mode.

The MDI mode is effectively used for simple cutting operations such as shaping soft jaws, cutting workpieces with simple contour and turning workpiece part to be chucked.

Note: *Before pressing the **CYCLE START** button, check if all the commands entered are correct.*

Set the FEEDRATE OVERRIDE dial to the lowest percentage setting (10%). It may be set at the 100% position only after the operator has assured axis movements.

The operation procedure is explained taking the following case as an example:



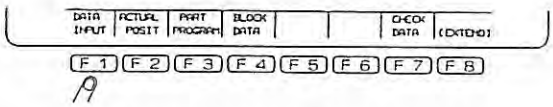
Commands to feed the cutting tool from "a" to "b" and then to "c":

a → b	G00 X60 Z100 S600 M41 M03 T0101
b → c	G01 Z50 F0.3

(1) Axis Movement from "a" to "b"

Procedure:

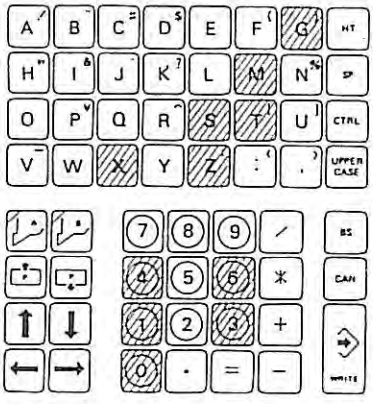
1) Press the function key [F1] (DATA INPUT).



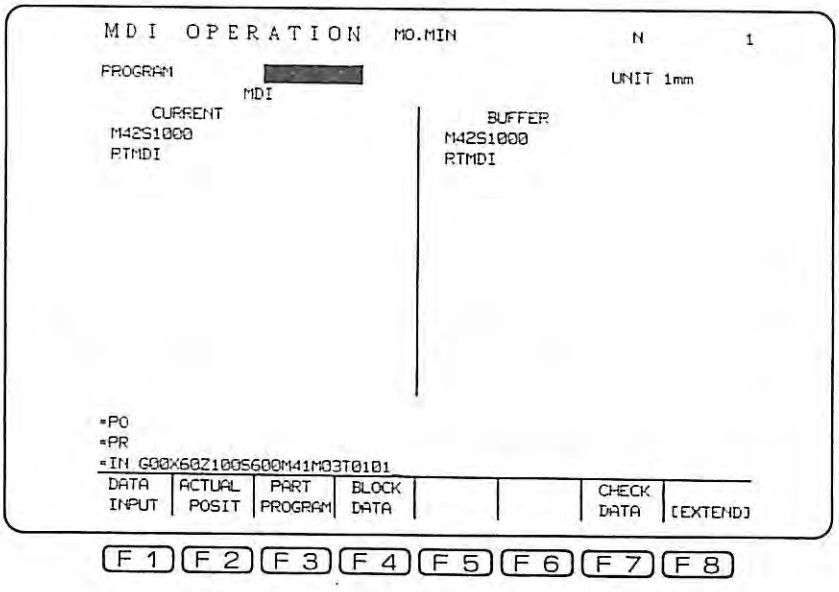
The prompt "IN" appears above "F1" display, telling the operator that the control is ready for data entry.

2) Enter a block of data through the keyboard:

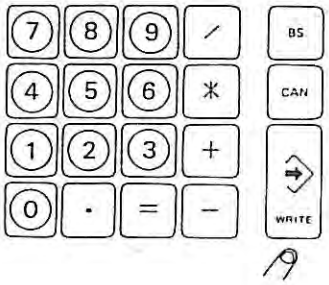
G00 X60 Z100
S600 M41 M03 T0101



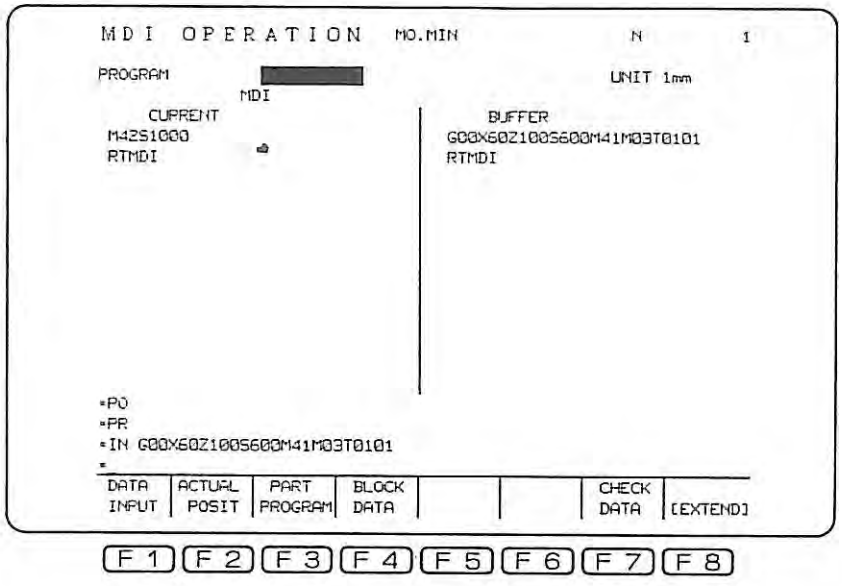
3) The keyed-in data is displayed on the CRT.



4) Press the **WRITE** key.

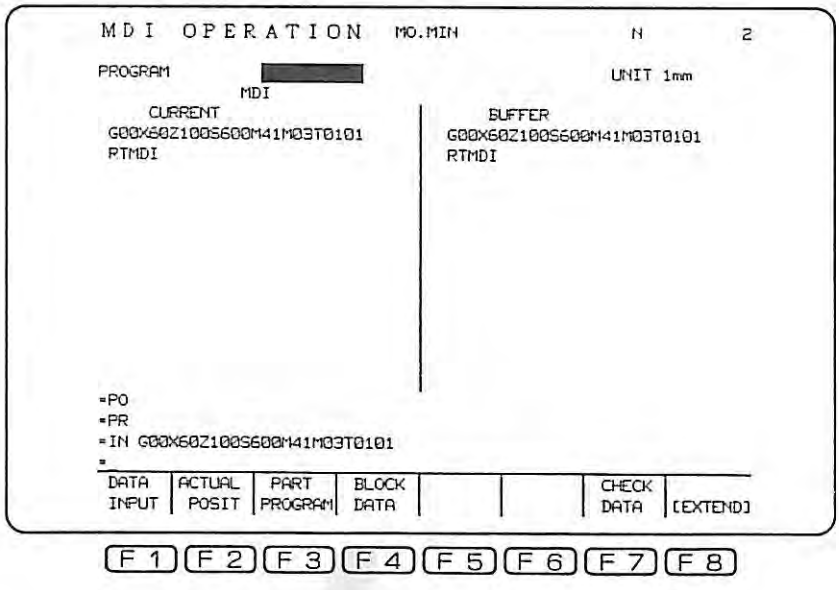


This transfers the keyed-in data to the BUFFER area.



5) Press the **CYCLE START** button.

This transfers the data in the BUFFER area to the CURRENT area.

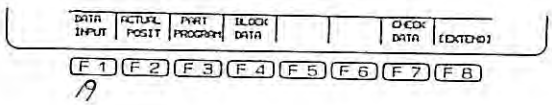


At the same time, the tool selection command (T0101) and spindle function commands (S600 M41 M03) are executed. After the completion of these commands, axis movement from "a" to "b" is initiated in the G00 rapid feed mode.

(2) Axis movement from "b" to "c"

Procedure :

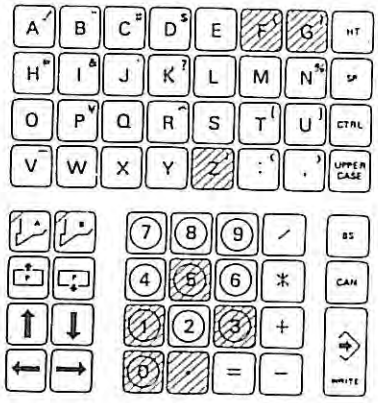
1) Press the function key [F1] (DATA INPUT).



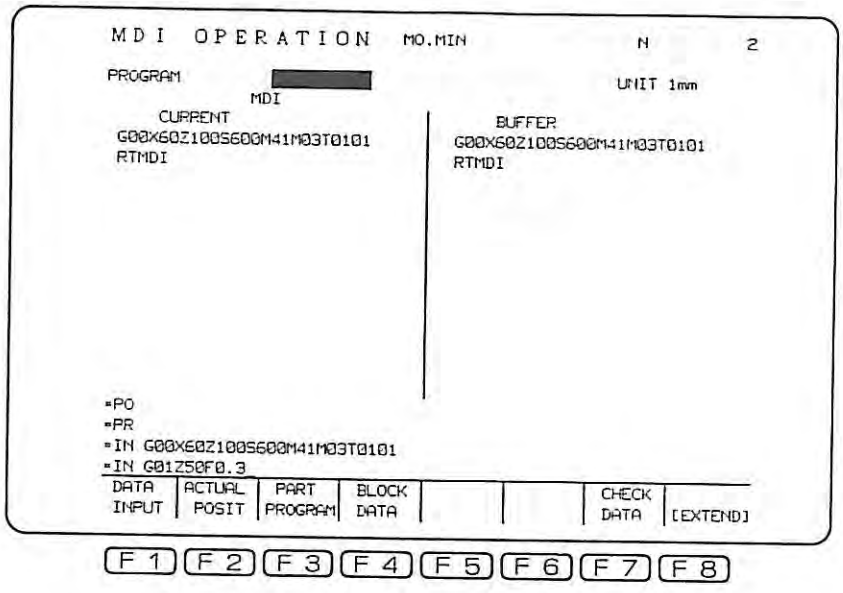
The prompt "IN" appears above "F1" display, telling the operator that the control is ready for data entry.

2) Enter a block of data through the keyboard:

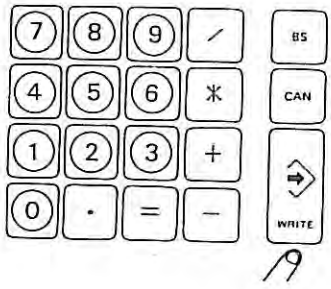
G01 Z50 F0.3



3) The keyed-in data is displayed on the CRT.

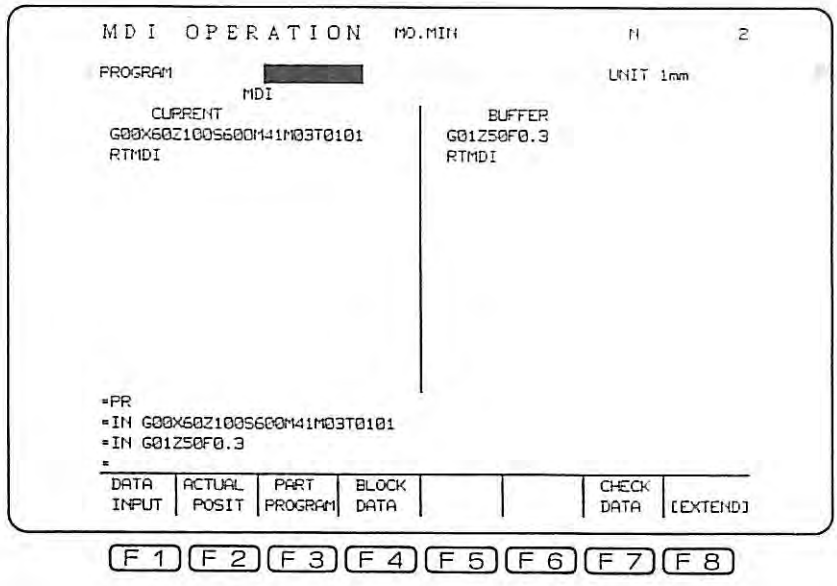


4) Press the **WRITE** key.



This transfers the keyed-in data to the BUFFER area.

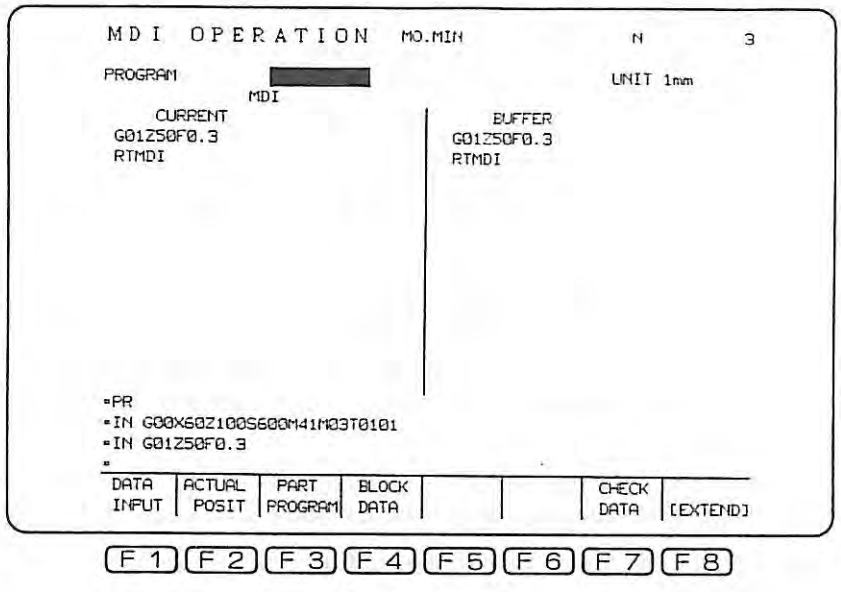
The data of the preceding block is cleared.



5) Press the **CYCLE START** button.

This transfers the data in the BUFFER area to the CURRENT area.

The data of the preceding block is cleared.



At the same time, axis movement from "b" to "c" is initiated at the commanded feedrate (G01 F0.3).

If operation of the rear turret is required, key-in "G14" and press the **WRITE** key first. Then press the **CYCLE START** button to select the rear turret. After this, the operation procedure explained above may be followed.

3-2-7. Interlock Functions

As the standard OSP5020L/OSP500L-G interlock function, the maximum spindle speed interlock function and the door interlock function are provided. These two interlock functions are provided to ensure safety in machine operations. Therefore, they must be correctly used following the instructions given below.

(1) Spindle Speed Interlock Function

a) Overview

In addition to the conventional maximum spindle speed designation function using the G50 command, the allowable chuck speed can be designated. The spindle speed is limited using these two speeds.

The interlock is taken so that the spindle cannot be started unless the maximum spindle speed is designated in a program.

b) Function

1) Level A alarm occurs if the M03/M04 command is executed unless the maximum spindle speed is designated with the G50 command in a block preceding the M03/M04 command.

This check is not conducted when a program is executed after the cursor movement or sequence number search.

2) Set the allowable chuck speed at the optional parameter (word). Each time the chuck is replaced, set the allowable speed which is indicated on the chuck.

Parameter	:	Optional parameter (word) No. 73
Initial value	:	0
Unit	:	Revolution per minute (rpm)
Setting range	:	0 - allowable speed for each machine

If the setting of this parameter is "0", level D alarm occurs, which cannot be reset until a value is set at this parameter.

3) The following interlock becomes effective according to the setting of the allowable chuck speed.

i) If the designated S value preceded by G50 exceeds the allowable chuck speed which is set at the optional parameter (word), level B alarm occurs.

ii) The spindle speed is limited by any of the following settings value, whichever lower; the maximum spindle speed designated by G50 or the allowable chuck speed.

iii) If the actual spindle speed exceeds 120% of the maximum spindle speed designated following G50 or the allowable chuck speed, level A alarm occurs.

iv) Spindle speed is always checked in any operation mode (automatic, MDI, and manual).

Note: *For controls with the optional IGF, pay attention to the following point.*

In the program output using the IGF, the value set by the IGF integer parameter No. 11 MAXIMUM SPINDLE RPM is output for an S command following G50. Because an alarm occurs if this S command value is greater than the allowable chuck speed, changing of this value becomes necessary.

(Refer to the Operation Manual for Interactive Graphic MDI Function (IGF-L3) for OSP5020L/OSP5000L-G/OSP500L-G (Publication No. 3280-E).)

(2) Door Interlock C Function

a) Overview

There are cases which cause hazard to the operator if the spindle is rotated or the turret is moved while the front door is open. The door interlock C function inhibits spindle rotation and turret movement while the door is not closed.

b) Interlock function

The following interlock becomes effective in any operation mode (automatic, MDI, and manual) while the door is open when the **DOOR INTERLOCK** switch at the side panel of the machine operation panel is ON.

1) Level A alarm occurs when an attempt is made to start the spindle.

For spindle jog, oscillation, and orientation, spindle rotation below the parameter set speed does not cause an alarm even if the door is open.

Parameter	:	Optional parameter (word) No. 74
Unit	:	rpm
Setting range	:	1 - 500
Initial setting	:	50 (for all models)

To prevent an occurrence of an alarm even when the spindle jog or oscillation speed exceeds 50 rpm, set proper value at the parameter.

2) Level A alarm occurs if an attempt is made to start the M-spindle.

This interlock is effective only for the multi-machining model.

3) Level A alarm occurs if an attempt is made to feed the turret.

4) Level A alarm occurs if an attempt is made to rotate the turret.

5) Level A alarm occurs if the signal which indicates that the coupled external device (bar feeder, for example) is in operation is ON.

If the door is opened in any of the following conditions, level A alarm occurs.

- Spindle is revolving.
- Turret is moving.
- Turret is rotating.
- M-spindle is revolving.
- External signal from coupled device is ON.

Note 1: *The door interlock is effective even during manual intervention operation while the **DOOR INTERLOCK** switch is ON. Therefore, to open the door during the manual intervention mode, stop the spindle first. If turret rotation/motion is required, it is necessary to turn off the **DOOR INTERLOCK** switch.*

Note 2: *The door interlock function is ineffective when the machine lock mode is on.*

c) Canceling interlock

The interlock is canceled in the following two methods:

- To turn OFF the **DOOR INTERLOCK** switch on the side panel of the operation panel.
- To input proper M code command.

1) Cancellation using **DOOR INTERLOCK** switch

Use this method for setup change, etc.

2) Cancellation using an M code

Use this method for automatic loading/unloading using a robot, etc. so that the alarm does not occur when the door is opened for robot service.

Once the door interlock cancel M code is designated in a program, the alarm check based on door opening is not conducted until the M code to cancel the door interlock cancel mode is executed or the control is reset (M02 included).

Door interlock cancel M209
Cancel of door interlock cancel M208

SECTION 4 APPLICATIONS

With the instructions provided in the previous section, the operator can handle the machine to cut workpieces with relatively simple contour.

This section deals with various types of operation procedures for more efficient working, such as automatic mode operation, program handling, and parameter setting. The explanation provided hereafter contains some technical terms related with computerized NC; those terms are detailed below as an introduction to advanced operation technique.

(1) Bubble Memory

All the information necessary as NC data, such as part program, tool offset values, zero offset values, and soft-limit data are all stored in the bubble memory. Since the bubble memory employs bubble element, which is a type of magnetic element, stored data can be retained even after the power is turned off.

On the OSP5020L/OSP500L-G, tapeless operation which makes the most of the bubble memory characteristics, assures a reliability of the highest terms.

To this effect, it is necessary to store machining programs in the bubble memory prior to starting machine operation, according to the instructions on tape storage procedures. Then that selection of the desired machining program is made from a number of machining programs stored in the bubble memory. Machine operation is controlled by that selected machining program.

Once a machining program is stored in the bubble memory, that program is retained even after power supply to the control is cut off. However, to resume automatic mode operation with the selected program, it is necessary to call out the desired program again after turning power supply on.

Selection of the desired program is necessary since several machining programs are stored in the bubble memory. The advantage lies in that the bubble memory can retain the stored data when power supply is cut off, thus eliminating repeated tape reading-in operations otherwise necessary to cut workpieces with the same part program.

(2) File Management

File management refers to the unit that stores machining program data in the bubble memory; OSP5020L/OSP500L-G employs a Filing System similar to the filing methods commonly used for documents in an office.

Fig. 4-1 exposes the system structure of the file management. It is useful as a guide to the concept of file management and its relevant methods.

The system structure is widely classified into three elements.

a) Bubble memory (data bank)

It functions as racks or cabinets used for keeping documents.

b) Files

The function is equivalent to that of files consisting of documents or ledgers.

c) Program name or number (process sheet)

The function is similar to that of documents on file.

Description of the structure elements:

a) Bubble memory (data bank)

The bubble memory serves as a data bank. Program data is stored in file units. They act as racks or cabinets where documents are kept.

Data exceeding the capacity of the bubble memory cannot be stored. To store such data, either clear unnecessary data, or increase the capacity (optional).

b) Files

There are four types of files as described later. They correspond to filed documents, or ledgers; to manage NC data, each file is stored with the name of a part such as gear, shaft or flange assigned to it. File names consist of "main file names" and "extensions".

Main file name . . . . . Consists of a character-string of 16 characters beginning with an alphabetic character.

Extension . . . . . Consists of a character-string of 3 characters beginning with an alphabetic character.

Note that in assigning usable characters a file name are: alphabetic characters, numeral characters and "-" (hyphen).

A file name is assigned in the order of "main file name" first, then "extension".

A period must be entered as a delimiter between the main file name and the extension.

Type of files:

1) Main program file . . . . . Character-string (16 characters) beginning with an alphabetic character .MIN

2) Subprogram file . . . . . Character-string (16 characters) beginning with an alphabetic character .SUB

3) System subprogram file . . . . . Character-string (16 characters) beginning with an alphabetic character .SSB

4) Schedule program file . . . . . Character-string (16 characters) beginning with an alphabetic character .SDF

Main file name

Extension

Description of respective files:

1) Main program file

A file comprising one or more main programs, with the main file name followed by the extension "MIN".

To make a main program file, there are two methods:

i) Classified depending on types of workpieces such as gear, shaft, and flange.

ii) Program(s) for each workpiece to be assigned with one file name.

2) Subprogram file

Patterns often repeated in cutting parts such as Vee-groove, and parting off cycle are filed in the subprogram file when preparing part programs. The subprogram file comprises one or more subprograms; the main file name is followed by the extension "SUB".

When the called main program contains the command that calls the subprograms, it is necessary to call the subprogram file of these subprograms also when selecting the program(s).

3) System subprogram file

Basically, the system subprogram file is the same as the subprogram file explained above; in the system subprogram file, repetitive patterns often used by respective users are filed.

The system subprogram file comprises one or more system subprograms, and the name consists of the main file name and extension "SSB".

Programs, once stored in the bubble memory, can be accessed from the main program as needed. In other words, it is not necessary to call out the system subprogram file.

4) Schedule program file

This file is used when controlling several part programs prepared to meet the production schedule. A schedule program file consists of one schedule program, and its file name is suffixed by "SDF".

c) Program name or number

A program name or number corresponds to the program name or number assigned to the tape or process sheet of each part program. For instance, a part name or drawing number can be used.

A program name or number is made up from:

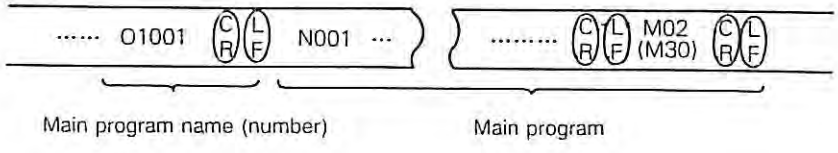
Program name . . . . . Address character "O" and four alphanumerics beginning with an alphabetic character.

Program number . . . . . Address character "O" and four numeral characters.

There are four types of program names and program numbers that correspond to the type of files explained in b).

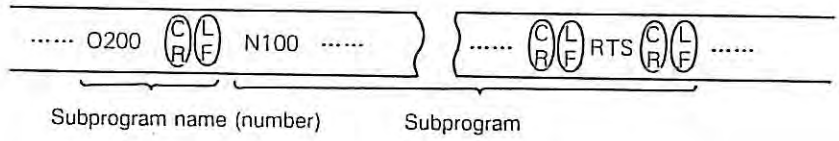
1) Main program

A main program begins with "program name (number) CR LF" and ends with "M02 CRLF" or "M30 CR LF".



2) Subprogram

A subprogram begins with "program name (number) CR LF" and ends with "RTS CR LF".



3) System subprogram

A system subprogram begins with "program name (number) CR LF" and ends with "RTS CR LF" as a subprogram indicated above.

4) Schedule program

All the data in the schedule program file becomes schedule programs.

The program format is as follows:

- Program selection block
- IF block
- GOTO block
- Variable setting block
- Schedule end block

For details, refer to 1-6. "Scheduled Operation".

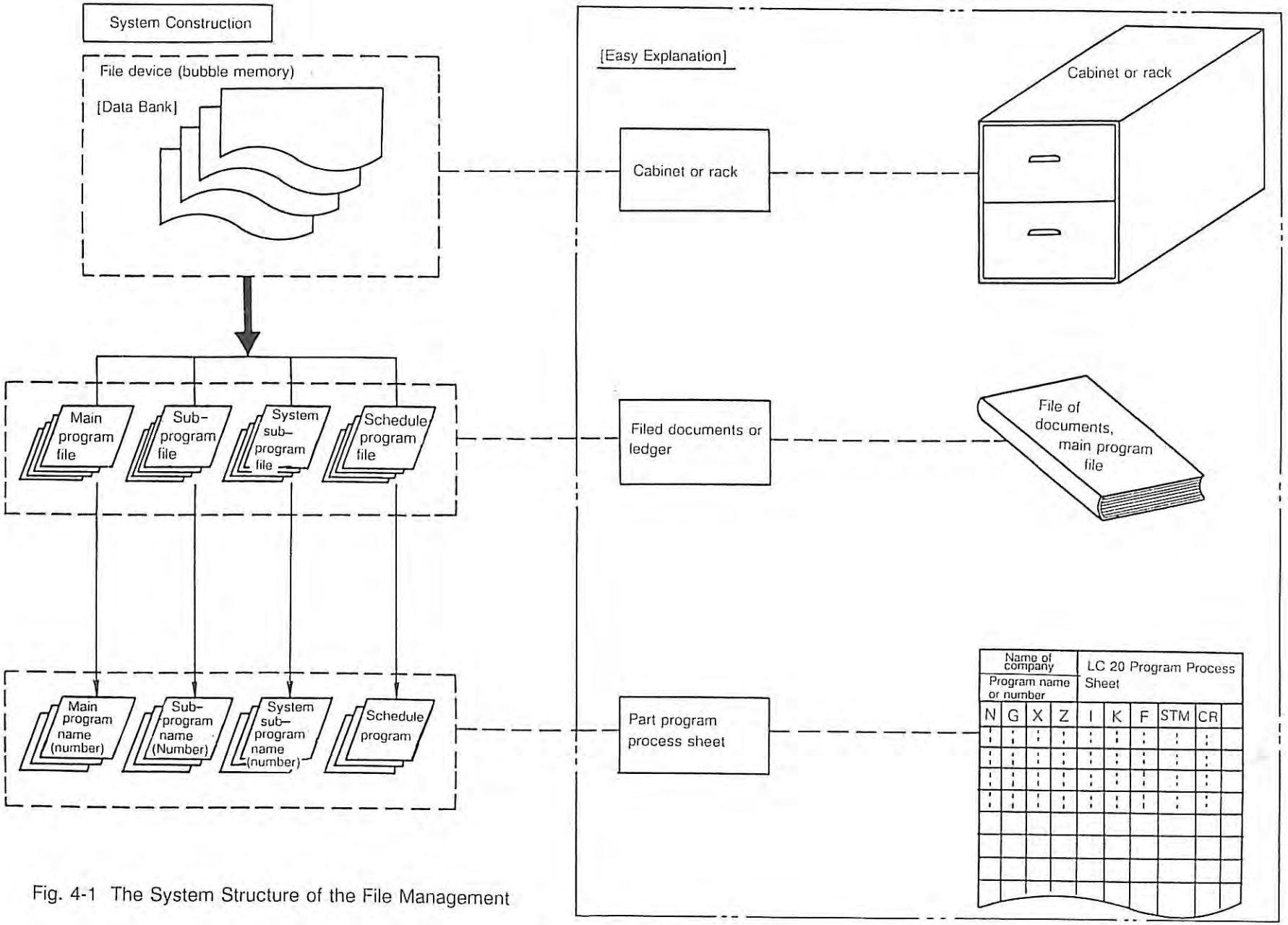


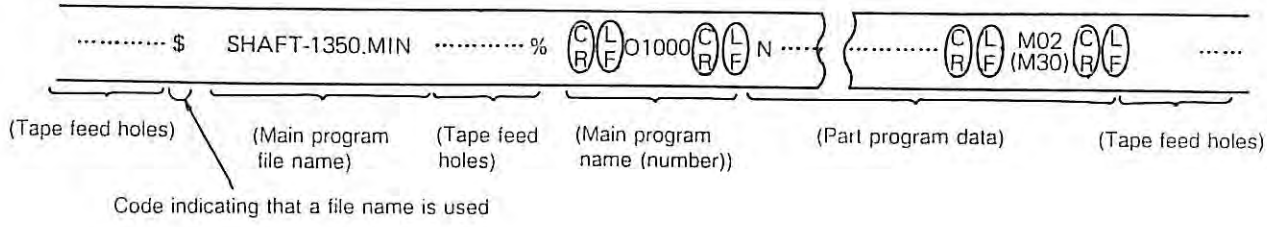
Fig. 4-1 The System Structure of the File Management

(3) Construction of Part Program (Format)

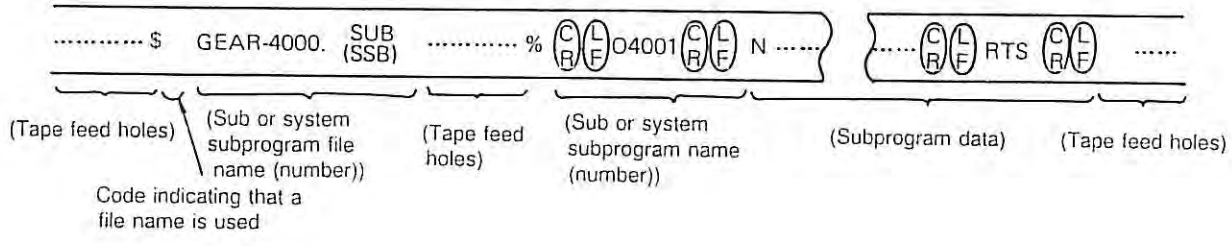
The general explanation concerning programs and how they are handled was provided in (1), Bubble Memory and (2), Managing User File.

This paragraph deals with the formats used to create a part program.

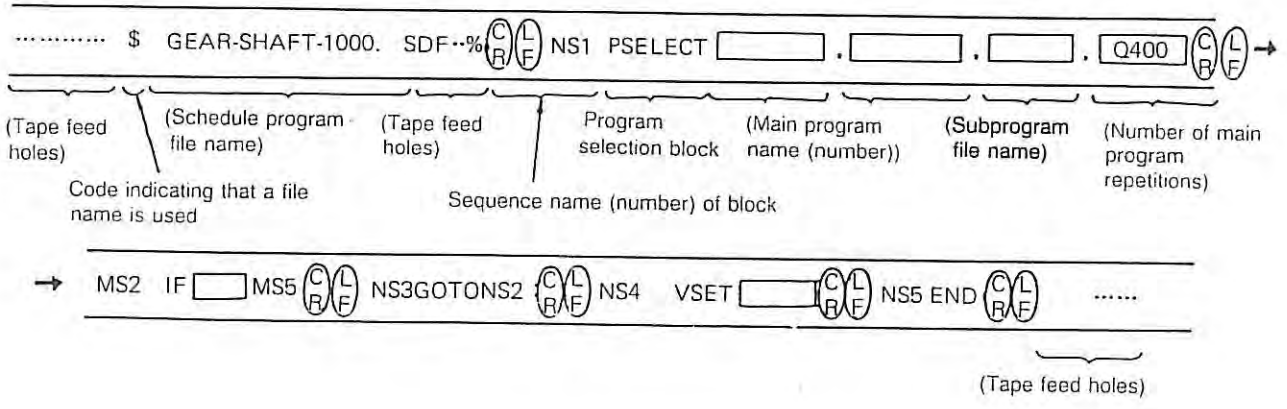
1) Main program
[Main program tape]



2) Subprogram and system subprogram
[Subprogram tape]



3) Schedule program
[Schedule program tape]



[Precautions on Programming]

Note 1: *When the first character in a program is "$", it indicates that a file name follows it.*

Note 2: *If no file name is provided at the beginning of a machining program, there are two ways for assigning a file name to that program:*

*1) Press the **WRITE** key right after pressing the function key [F1] (READ). A file name A.MIN is automatically provided after the read-in data, and the data is stored in the bubble memory with that file name.*

*2) After pressing the function key [F1] (READ), key in "[File Name]" and then press the **WRITE** key. The read-in data is stored in the bubble memory with the keyed-in file name.*

Note 3: *To store several part programs in the bubble memory (data bank), it is necessary to name each program with a different file name. Or each program should be given a specific file name in accordance with step 2) in Note 2 when stored in the bubble memory.*

Note 4: *There are two possible ways to designate the coding system of a program, either EIA or ISO:*

- *Setting of parameter (bit) data,*
- *Designation of option when carrying out Read, Punch or Verify operation.*

1) Parameter (bit) data setting

When "tape code parity check" bit is "1":

- *ISO and EIA coding systems are automatically recognized when reading or verifying is carried out.*
- *When the program is punched out, data is punched out in the ISO code when "tape code ISO code" bit is "1", and in EIA code when it is "0".*

When "tape code parity check" bit is "0":

- *When "tape code ISO code" bit is "1", data is handled as coded according to the ISO coding system for tape reading or verifying operation. When it is "0", the EIA coding system is assumed. Therefore, if the bit data setting and actual coding system do not match, an alarm occurs.*
- *When the program is punched out, data is punched out in the ISO code when "tape code ISO code" bit is "1", and in the EIA code when it is "0".*

2) Option designation

For individual operations, tape read-in, verifying, and punching-out, data is handled as coded in the coding system designated as "option" disregarding the setting of parameter bit data.

For details, refer to instructions in 2-1. "Transfer of Main Program Data".

Note 5: *In a part program, data following the first LF (or CR) code is effective.*

Note 6: *End of data in a part program is judged by the tape feed holes. However, it can also be judged using the "% (ER)" code by setting a proper bit data.*

Note 7: *The symbols $, *, = and [,] are not included in the EIA code, but it is possible to register EIA code patterns to correspond to these symbols at the optional parameter (bit) which enables the use of the symbols $, *, = and [,] in the EIA code as well.*

(4) Math Operation Function for Parameter Data Input

To set data for tool offset, for example, in the parameter set, zero set, and tool data set modes, arithmetic and function operation expressions can be used in addition to the conventional direct numerical data input method.

a) Arithmetic and function operation symbols

Arithmetic and function operation expressions are designated using the symbols below.

Operation Symbol List

Symbol	Contents	Example	Remark
[+] [−]	Plus sign Minus sign	[+] 12.34 [[−] SIN 90]	Usable only at the beginning of an expression or right after the "[" symbol.
[+] [−]	Addition Subtraction	12.3 [+] 456.7 [12.3 [−] 4] [−] [5 [−] 6]	
[*]	Multiplication	12.34 [*] 56.7	[*] indicates the multiplication symbol (×).
[/]	Division	[12.3 [/] 4] [/] 5.6	[/] indicates the division symbol (÷).
[R] [S][Q][R][T]	Square root	[R] (SP) 30 [S][Q][R][T] [30 − 20]	If symbols [ ] are not used to indicate figures (operand) following the function symbol, place at least one space between the function symbol and the operand.
[S] [S][I][N]	Sine	[S] [45*2] [S][I][N] (SP) 60	The figure written following the function symbol is an angle and expressed in units of degrees. If symbols [ ] are not used to indicate figures (operand) following the function symbol, place at least one space between the function symbol and the operand.
[C] [C][O][S]	Cosine C O S [15 + 45]	[C] (SP) 30 [C] [O] [S] [15 + 45]	
[T] [T][A][N]	Tangent	[T] [45 − 15] [T] [A] [N] SP [15*3/2]	
[A][S] [A][S][I][N]	Arc sine	[A][S] (SP) 0.5 [A][S][I][N] [15.5/22.2]	The result of operation is an angle in units of degrees.
[A][C] [A][C][O][S]	Arc cosine	[A][C] [0.8*0.6] [A][C][O][S] SP 0.45	If symbols [ ] are not used to indicate figures (operand) following the function symbol, place at least one space between the function symbol and the operand.

Symbol	Contents	Example	Remark
[A][T] [A][T][A][N]	Arc tangent (−90° to 90°)	[A][T] (SP) 45 [A][T][A][N] [45*2]	The result of operation is an angle in units of degrees. If symbols [ ] are not used to indicate figures (operand) following the function symbol, place at least one space between the function symbol and the operand.
[A][T][2] [A][T][A][N][2]	Arc tangent (−180° to 180°)	[A][T][2] [0.5, 1.5] [A][T][A][N] 2 [0.45, 1]	Designation should always be AT2 [numerator, denominator]. Symbols [ ] cannot be omitted.

Remarks: 1) If the expression does not follow the format indicated above, or an overflow occurs in the course of operation, a calculation error occurs.

Example: S30 → Calculation error (No space between "S" and "30")

2) As indicated in the list above, symbols [ ] can be used for operation. Although the nesting level for the usage of them is not specially limited, fourth nesting level is the maximum depth to guarantee the results of operations.

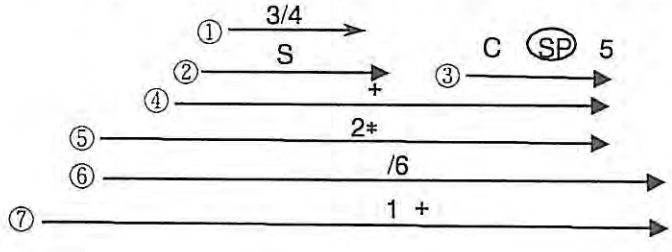
Example: R (SP) [180 + [S*[30 − 5] − 20*[6 − 2]]]

3) Calculation order follows usual arithmetic operation rules:

In parentheses → Functions → Multiplication/division → Addition/subtraction

If operations of the same calculation priority are used, calculation is made from the leftmost operations in order.

Example: 1 + 2* [S [3 / 4] + C SP 5] / 6



b) Input Example using Expressions

With the machine having the rapid feedrate of 10 m/min., manual feedrate is usually 2.4 m/min. However, this feedrate can be changed by changing the manual feed unit amount set at the system parameter. However, since the setting is made in units of μm/12.8 msec., complicated calculation is required to obtain the setting value.

To change the manual feedrate to X = 2 m/min and Z = 1.5 m/min, calculation required would be:

$$\text{X-axis manual feed unit amount} = \frac{2 \times 10000000}{60 \times 1000} \times 12.8$$

$$\text{Z-axis manual feed unit amount} = \frac{1.5 \times 10000000}{60 \times 1000} \times 12.8$$

To input the data, calculation function can be used in the following manner.

Procedure:

1) Locate the cursor at the manual feed unit amount column of X-axis.
2) After pressing the function key [F1] (SET), input as indicated below.

   =S—[2][*][1][0][0][0][0][0][0][0][0][/][[][6][0][*][1][0][0][0][]][*][1][2][.][8] **WRITE**

   Then, the following is displayed on the console line of the CRT screen;

   =S— 2*10000000/[60*1000]*12.8

   and the result of the calculation is displayed at the cursor position. In this example, the calculation result is "426".
3) Move the cursor to the manual feed unit amount column of Z-axis.
4) After pressing the function key [F1] (SET), input as indicated below.

   =S—[1][.][5][*][1][0][0][0][0][0][0][0][/][[][6][0][*][1][0][0][0][]][*][1][2][.][8] **WRITE**

   The following is displayed on the console line of the CRT screen;

   =S—1.5*10000000/[60*1000]*12.8

   and the result of the calculation is displayed at the cursor position.

Note: *Input of symbol "[":*

OSP5020L ..... *Press* [T[] *while holding down the* ***UPPER CASE*** *key.*

OSP500L-G ..... *Press* [[D B] *while holding down the* ***LEFT UPPER CASE*** *key.*

Input of symbol "] ":

OSP5020L ..... *Press* [U]] *while holding down the* ***UPPER CASE*** *key.*

OSP500L-G ..... *Press* [ ]E @] *while holding down the* ***LEFT UPPER CASE*** *key.*

1. Application of Operations - Automatic Mode Operation

1-1. Program Selection and Operation

To execute a part program in the automatic mode, the desired program must be selected. This is because the bubble memory stores several part programs, and it is necessary to call the specific program for execution.

There are two methods which can be used for selecting programs:

(1) Selection from the directory display page

(2) Direct designation

(1) Selection from the Directory Display Page

Program selection including a schedule program is possible from the program directory page by placing the cursor (reverse display) on the desired program name.

Procedure:

1) Select the AUTO OPERATION mode by pressing the **AUTO** key.

2) Press the function key [F1] (PROGRAM SELECT).

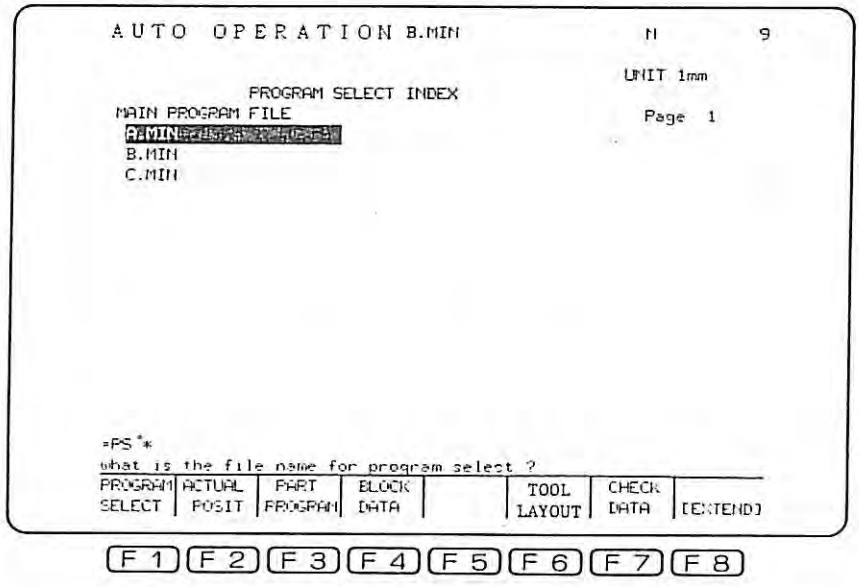
3) Key in an asterisk (*) through the keyboard.

The CRT will display:

=PS *

4) Press the **WRITE** key.

The program directory page will be displayed by the operations above. One page of this display shows a total of 12 file names, and if more than 12 file names are registered, press the ↵ key to display the second page.



5) Move the cursor to the file name desired.

6) Press the **WRITE** key.

This selects the program and the CRT display is restored to the program display page with the program designated on it.

7) Press the **CYCLE START** button.

This starts the execution of a program elected.

For the selection of a schedule program, press the function key [F8] (EXTEND) after the completion of step 1) above and press the function key [F4] (SP SELECT) instead of the operation of step 2). Then, follow the steps above.

Note 1: *During the operations in steps 4) through 6), keys other than*  ,  ,  *and*  *are not operative.*

Note 2: *The only files which can be selected by this operation are programs with the extension "*.MIN" and schedule programs with the extension "*.SDF".*

Note 3: *Since designation of a program name is not possible, the first main program in the designated main program file is automatically selected, if the designated main program file has more than one main program.*

Note 4: *Since subprogram names cannot be designated, subprograms usable in a main program are limited to those indicated below:*

a) Those in the subprogram file of ".SSB"*

b) Those in the subprogram file of ".MSB"*

c) Those in the main program to be selected

Subprogram file ".SSB":*

Subprogram file which a user can create as needed. It is edited and created under the file name ".SSB" (* represents a character-string of up to 16alphanumerics.)*

Subprogram file ".MSB":*

Subprogram file created by Okuma and called out by a user as needed. Mainly called out for execution the gauging cycle.

Note 5: *If there is no main programs or schedule programs, the message* ***"not found main program file"*** *or* ***"not found schedule program file"*** *will be displayed on the CRT.*

(2) Direct Designation of Programs

Assume the bubble memory stores several part programs including the one to be executed now.

Example of registered programs:

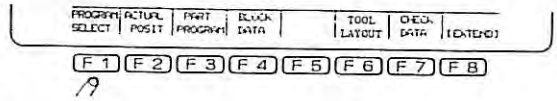
File Name	Program Name	Contents
A.MIN	OMIN1 OMIN2	Part program 1 Part program 2
SHAFT.MIN	OSHT1 OSHT2	Part program for shaft 1 Part program for shaft 2

Explanation and notes below are provided according to the example above.

1) Select the AUTO OPERATION mode by pressing the **AUTO** key.



2) Press the function key [F1] (PROGRAM SELECT).

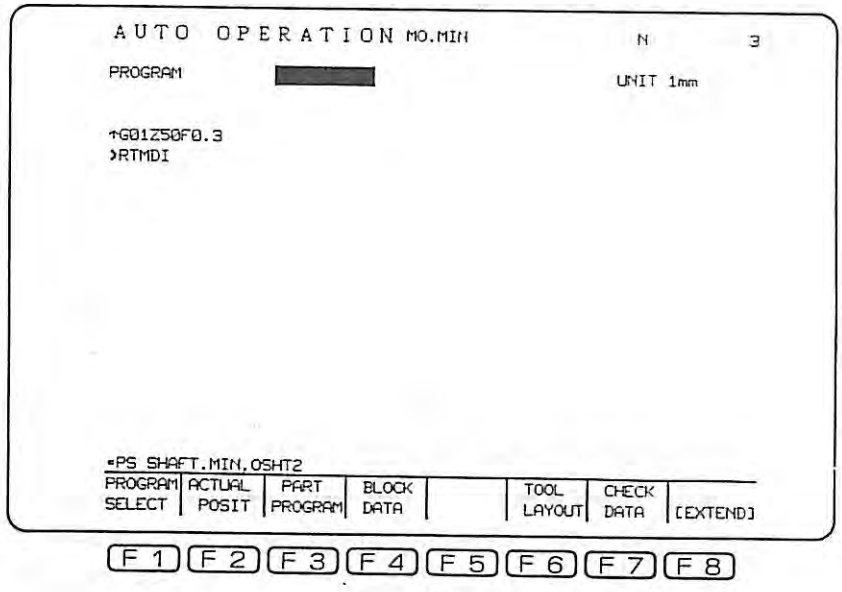


The CRT displays prompt "**PS**" on its console line and the control is ready for data input.

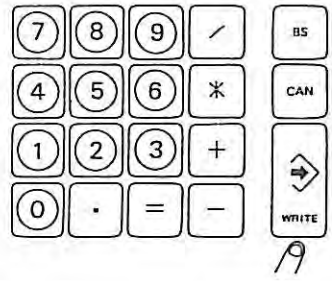
3) Key in the file name and program name through the keyboard. Use a comma "," as a delimiter.

=PS SHAFT.MIN,OSHT2

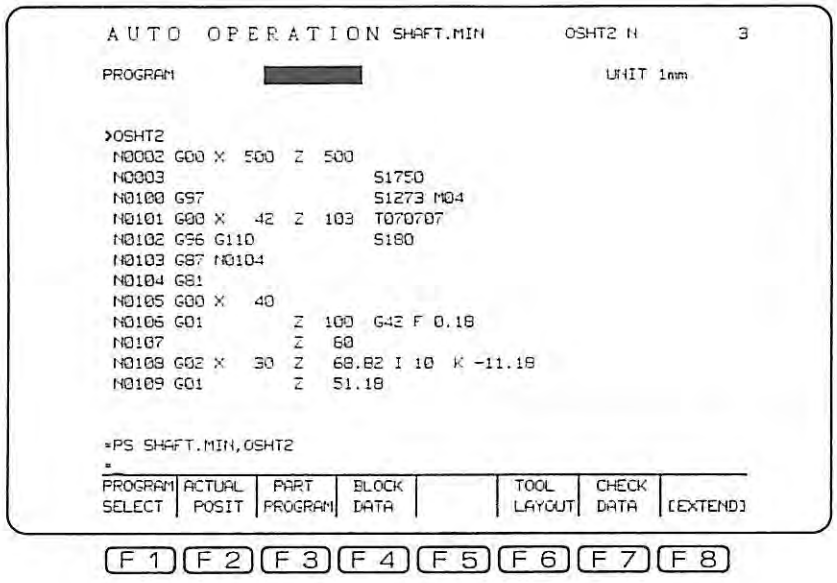
Keyed-in data



4) Press the **WRITE** key.



This selects the program to be executed, and the contents of the selected program are displayed on the CRT. Also the designated file name and the program are displayed at the top of the CRT.



5) Press the **CYCLE START** button.

With steps 1) through 5), the part program OSHT2 in the file named SHAFT is executed.

Note 1: *Omission of the file name and/or program name during designation:*

- *When the extension of a file is "MIN", it can be omitted.*

PS SHAFT.MIN,OSHT2 → PS SHAFT,OSHT2

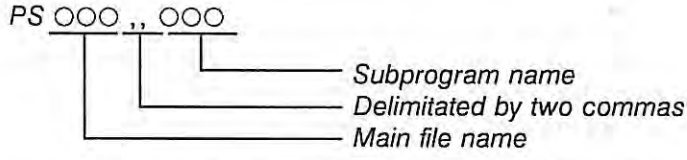
- *When both or either the file name or program name is omitted.*

	File Name/Program Name	Effective Command
a	= PS Omitted Omitted	= PS A.MIN , OMIN1
b	= PS SHAFT.MIN Omitted	= PS SHAFT.MIN , OSHT1
c	= PS SHAFT.MIN , OSHT2	= PS SHAFT.MIN , OSHT2
d	= PS Omitted , OMIN2	= PS A.MIN , OMIN2

When a file name is omitted, "A.MIN" is automatically selected as the file name . . . . . a) and d)

When a program name is omitted, the first program in the specified file is selected . . . . . a) and b)

In case a subprogram is called in the main program:



Note 2: *If the bubble memory does not have the designated file or program name, an alarm results. In this case, the program previously selected becomes ineffective. The effective file and the program name appear on the first line of the CRT; be sure to check those when selecting the program.*

Note 3: *The program once selected remains effective until another program is selected. Turning power supply on/off has no influence on the selected program. If this operation is attempted, symbol "□" will appear after the program name and file name. However, selection of a schedule program makes the selected program ineffective.*

Note 4: *If, when the power has been switched on and the machine has been operating in the MDI mode, the automatic mode is selected without first pressing the* ***RESET*** *button, an error occurs during the execution of the main program, thus the program selection is not made.*

Note 5: *A maximum length of the program (including subprograms) usable for program selection operation is 30 meters in tape length. This value may be expanded to up to 640 meters optionally.*

1-2. Sequence Restart

When the execution of a program is interrupted with the control reset or the **EMERGENCY STOP** button pressed due to tool breakage or entangled cutting chips, the use of "sequence restart" feature permits the resumption of the program from the block right before the one where the operation was interrupted; it is not necessary to start the machine operation from the beginning.

There are two ways for sequence restarting:

1) Sequence restart with the designation of a sequence number.
2) Sequence restart with the designation of a block counter.

Example: SHAFT-1.MIN

```
OSHT1
N100 G00 X300 Z300
N110 G50                S3500
N120                    S 424 M41 M03 M08
N130 G00 X105 Z92.048   T0202
N140          Z90
N150                    S 424 M09
N160     X300 Z300      T0200
N170                    S 792 M08
N180 G00 X106 Z90       T0303
N190     X102
N200     X 66
N210 G01      Z85       F0.1
N220     X 70 Z83
N230          Z60
N240     X 80 Z45
N250          Z29
N260 G02 X 88 Z25       I 4
N270 G01 X 90           F0.45
N280          Z20       F0.3
N290     X 98           F0.45
N300     X100 Z19       F0.3
N              :
               :
               :
```

To restart the sequence from N180, follow the procedures below.

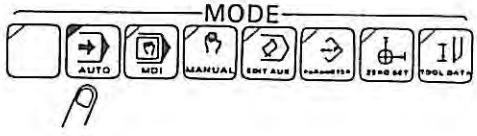
(1) To Restart the Sequence by Designating the Sequence Number (Name)

1) After selecting the MANUAL OPERATION mode, retract the turret to the desired position.

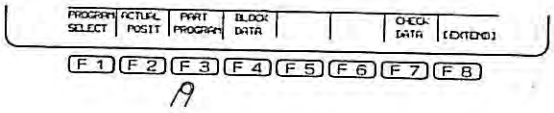
   If sequence restart is to be made from the block calling for ID turning, retract the turret carefully so that the cutting tool does not interfere with the workpiece.

2) Change the broken tool or remove entangled chips.

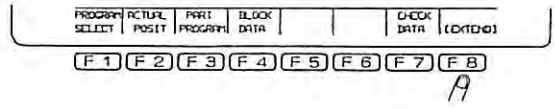
3) Select the AUTO OPERATION mode by pressing the **AUTO** key.

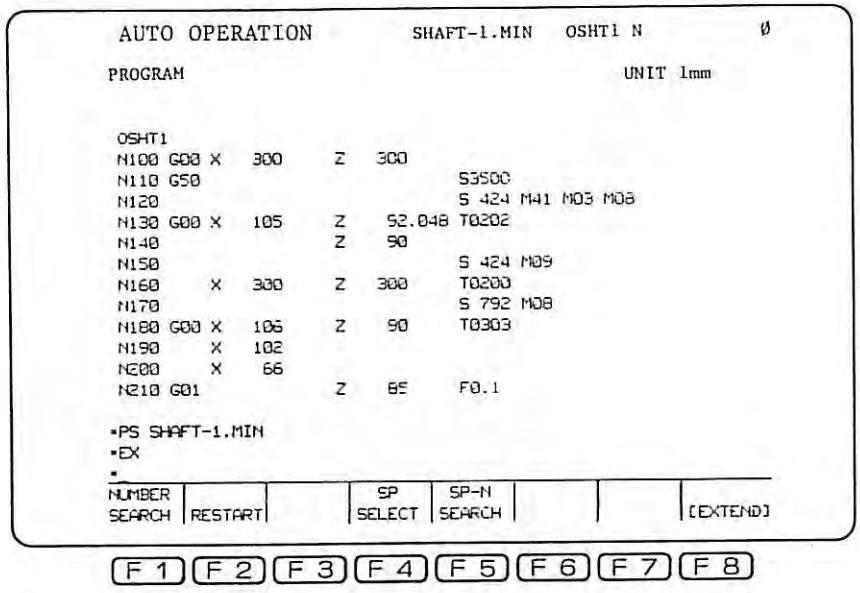


4) Press the function key [F3] (PART PROGRAM) to display the part program on the CRT.



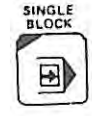
5) Press the function key [F8] (EXTEND) to select the page that allows the designation of a sequence number.



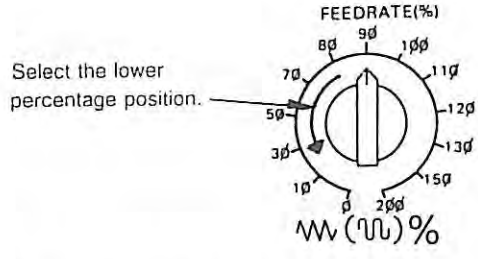


Make sure the guide display of functions has changed.

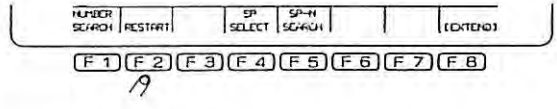
6) Press the **SINGLE BLOCK** key and select the low percentage position on the **FEEDRATE** override dial.



This is to prevent unexpected hazards (mainly collision between the cutting tool and the workpiece) when sequence restart is performed erroneously. Therefore, make sure this step is performed.

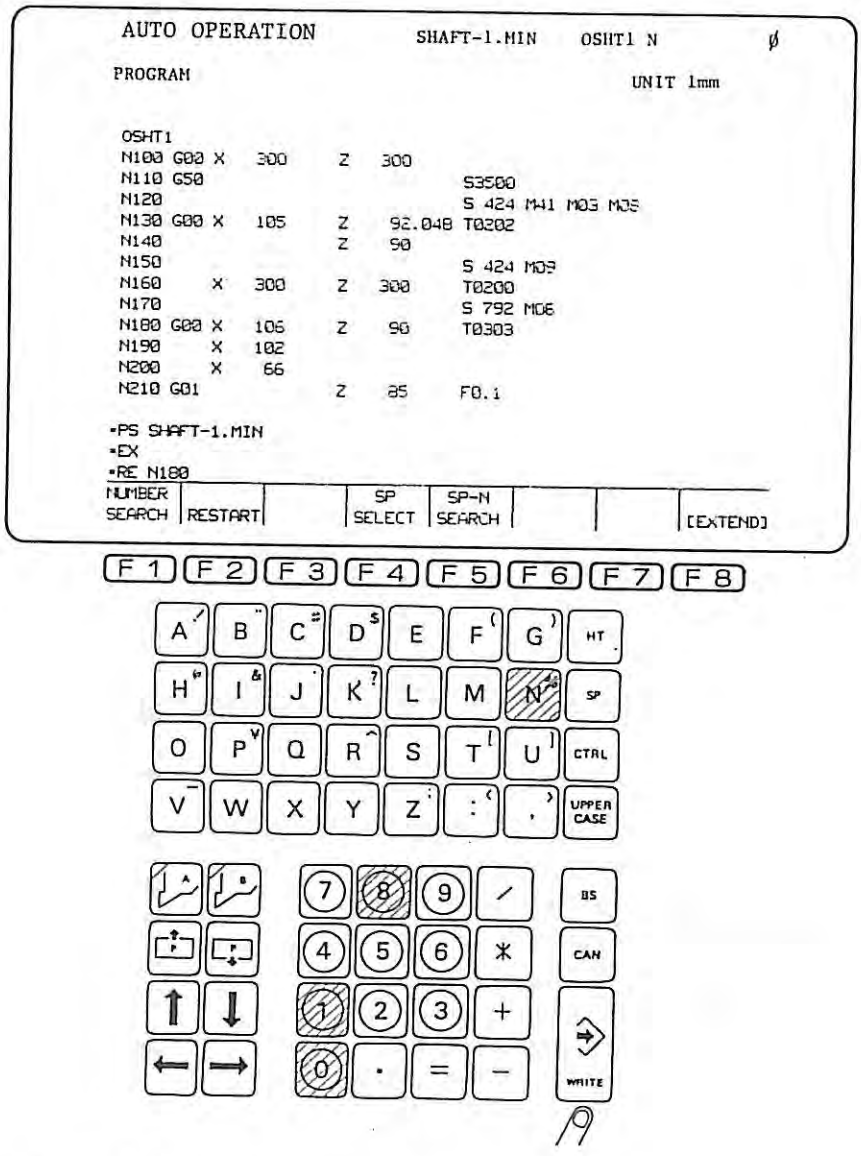


7) Read out the sequence number N180 where the sequence is to be restarted.



Press the function key [F2] (RESTART).

After keying in N180 through the keyboard, press the **WRITE** key.



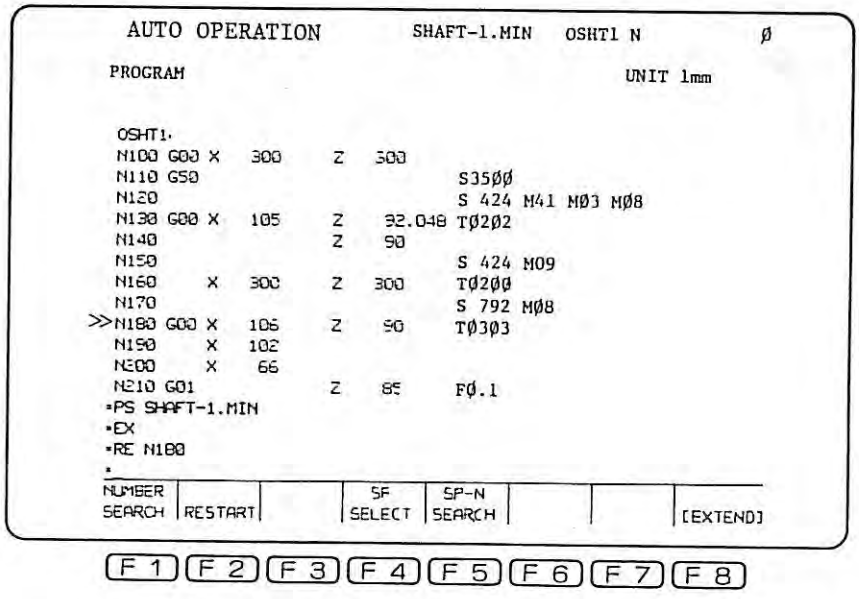
Note 1: *Leading zeros of sequence numbers cannot be omitted.*

Example : *N0100*

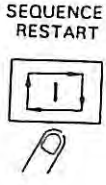
This "0" cannot be omitted

Note 2: *For two-saddle models, the sequence number should be read by designating the restart numbers for both the A- and B-turret.*

This places the address pointer "≫" at the sequence number N180.



8) Press the **SEQUENCE RESTART** button.



With the steps above, the turret is fed to X300.00, Z300.00 programmed in N160 at the rapid feedrate after miscellaneous functions (S, T and M commands) have been completed.

That is, the coordinate point where the axes (turret) are positioned is the point commanded in the block preceding the read out block.

The axes move to the sequence restart point at the rapid feedrate (G00) irrelevant to the programmed feed mode.

【 Supplement 】

When one program contains two or more blocks assigned with the same sequence number, and to restart the program from that sequence number, it is necessary to indicate where the required sequence appears, at the 1st, 2nd, etc.

Example:

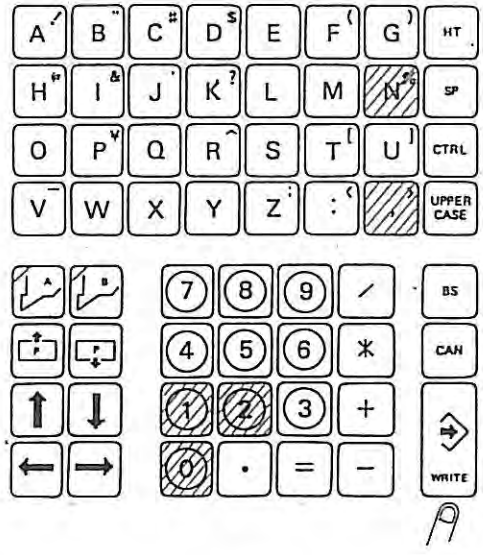
To restart the sequence from this N120 (2nd N120)

```
N100 G00 X300 Z300
N110 G50                S3500
N120                    S 424 M41 M03 M08
N130 G00 X105 Z92.048   T0202
N140          Z90
N150                    S 424 M09
N160      X300 Z300     T0200
N170                    S 792 M08
N120 G00 X106 Z90       T0303
N130      X102
N160      X 66
N210 G01        Z85     F0.1
N220      X 70 Z83
N230           Z60
N240      X 80 Z45
```

1) Key in as indicated below for step 7) explained before.

[F2] (RESTART) N120,2 **WRITE**

This indicates the 2nd N120 in the selected part program. The numerical data following delimiter "," is used to indicate where the desired sequence is, 1st, 2nd, etc.

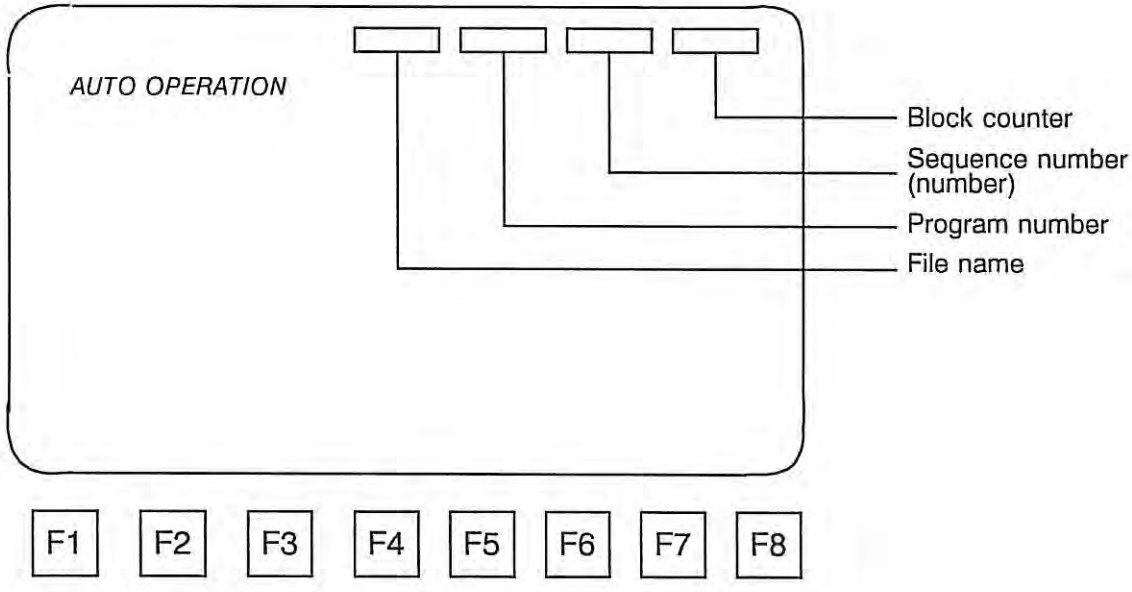


(2) To Restart the Sequence by Designating The Block Counter

Block counter:

In automatic mode operation, the control counts the number of executed blocks taking the block executed first as "1". This number is, therefore, not the sequence number but the number of actually executed blocks. Therefore, in LAP or subprogram which repeatedly executes the same block many times, the block counter counts the number of executed blocks. This greatly helps sequence restart operation when it is attempted from a block in LAP or thread cutting subprogram.

Block counter data is displayed at the upper right part of the CRT.



Note 1: *The block counter is not cleared when the control is reset or the operation mode is switched.*

Note 2: *The clock counter is cleared when power supply to the control is shut off. Therefore, if power supply to the control is shut off by emergency stop or similar operation during machine operation, sequence restart using the block counter cannot be made.*

When it is necessary to use the block counter feature after turning off power supply once to restart the sequence in LAP, thread cutting subprogram, or parameter program, find the block counter data at which emergency stop was activated by actually running the program in the Machine lock or Dry run mode.

Procedure to restart sequence by designating the block counter number:

Basically, the same procedure to restart the sequence by designating the sequence number applies. In this case, however, the block counter number is designated instead of the sequence number in step 7) in (1), "To Restart the Sequence by Designating the Sequence Number (Name)".

Follow the steps below:

Assume the block counter is 8 for sequence number N180

1) Do as follows instead of step 7) and following steps.

Read out the block counter number "8" where the sequence is to be restarted.

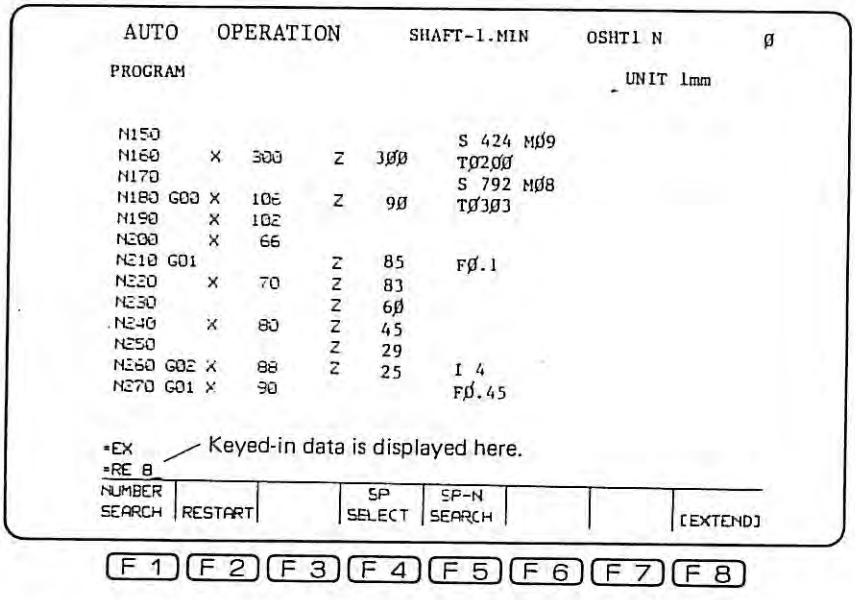
Press the function key [F2] (RESTART).

↓

Key-in "8" through the keyboard.

↓

Press the **WRITE** key.



This reads out the block counter number.

【 Remarks on Carrying Out Sequence Restart Operation 】

(1) Interference between Cutting Tool and Workpiece

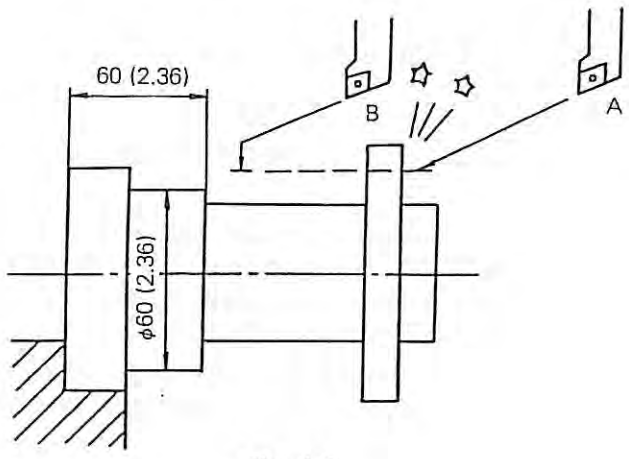


Fig. 4-2

If sequence restart is performed with the cutting tool positioned at point A, the cutting tool will collide against the workpiece. Be sure to move the cutting tool to point B with manual controls on the NC operation panel before carrying out sequence restart operation.

Index the turret manually to select the necessary tool for sequence restart before pressing

(2) Sequence Restart from The First Block in LAP

To restart the designated commands from the very beginning of LAP, sequence restart must be made from the block containing the G code used to call the LAP cycle to be executed.

G codes used to call LAP cycle . . . G85, G86, G87 and G88

Example:

```
NAP1 G81
N001 G00 X 50 Z100
N002 G01 X 54 Z 98 F0.25 S1000
 :  :
 :  :
N020 G80
N101 G00 X800 Z300
N102 X100 Z200 S750 T0101 M03 M42
N103 G85 NAP1
 :  :
 :  :
```

Sequence restart must be done from the N103 block containing G85 NAP1.

The CRT displays the commands in the first sequence N001 of the contour defining commands of LAP.

(3) Sequence Restart from a Block within LAP

To restart the program from a block within LAP blocks, be sure to use the block counter number. If sequence restart is intended by designating the sequence number, it is difficult to find the desired infeeding point since the number of infeeding and the reading-in time of contour definition blocks do not match.

(4) If the operator intends sequence restarting while a program is being executed, or after MDI operation, it results in "MAIN PROGRAM EXECUTING ERROR". In such cases, reset the control first.

1-3. Sequence Number Search Operation

Sequence number search is the function that serves to search for the block assigned with the commanded sequence number. The data in the previous blocks are all ignored, and those in that block become effective.

This feature provides effective means to restart the program from the desired block. If the same effect as sequence restart operation is desired in sequence number search operation, be sure to search for the block which contains all the conditions necessary to restart the operation intended, i.e., the block containing G, X, Z, S, T and M commands.

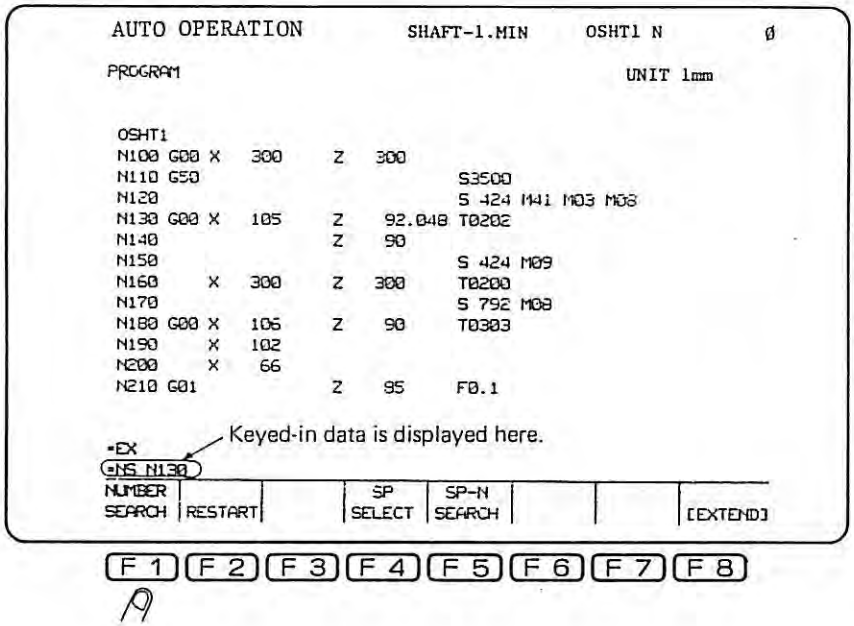
In order to remind the operator of the fact that the cycle is going to start in the midst of a program, both hands are used to start operation. (Press **CYCLE START** button while holding down the **INTERLOCK** key.)

Sequence number search procedure is detailed below taking the same example as used in 1-2. "Sequence Restart" of this section.

The sequence number to be searched for is: N130

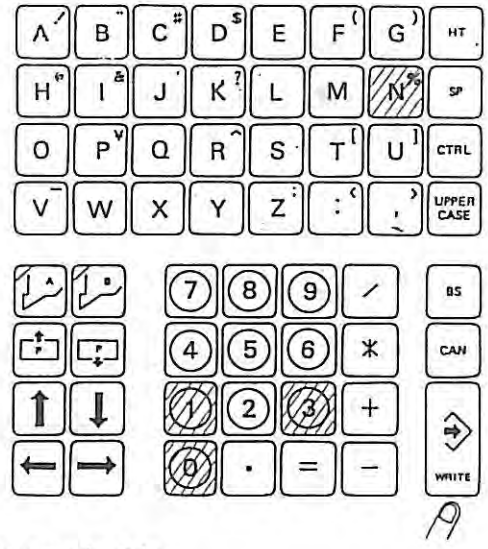
Follow the steps below:

1) Follow the steps 1) through 6) in 1-2. of this section.
2) Press the function key [F1] (NUMBER SEARCH) to read out the sequence number N130.
3) Key in N130 through the keyboard.

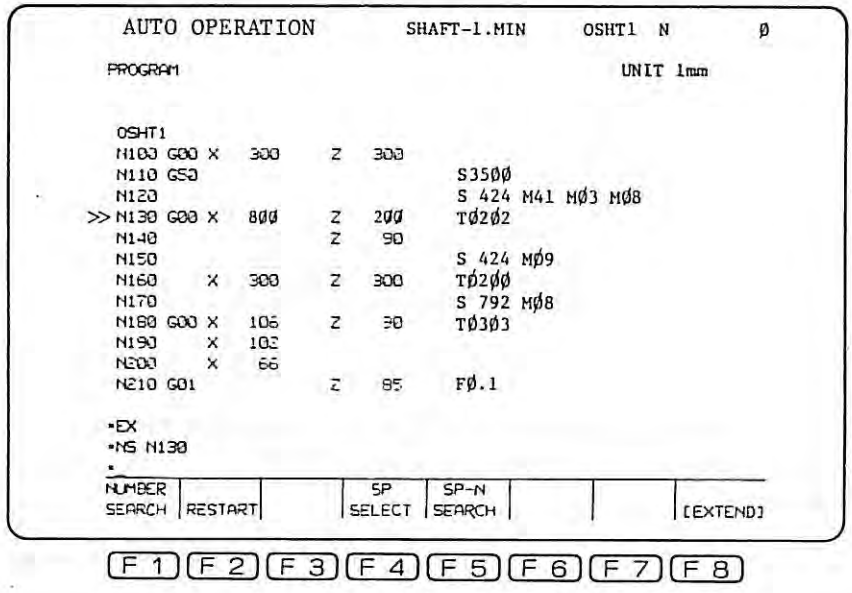


9

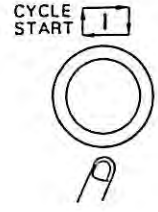
4) Press the **WRITE** key.



With this, the desired sequence number N130 is searched for.



The address pointer (≫) is moved to the designated sequence number (N130).



5) While holding down the **INTERLOCK RELEASE** key, press the **CYCLE START** button to initiate the execution of the commands in read out blocks.

With the **CYCLE START** button pressed, S, T and M commands are executed first. After the completion of those commands, the turret is positioned to the designated coordinate point, in the mode specified by the commanded G codes. In this example, the turret moves to the point (X800, Z200) designated in N130 at the rapid feedrate (G00).

PLEASE NOTICE THE DIFFERENCE BETWEEN SEQUENCE RESTART AND SEQUENCE NUMBER SEARCH.

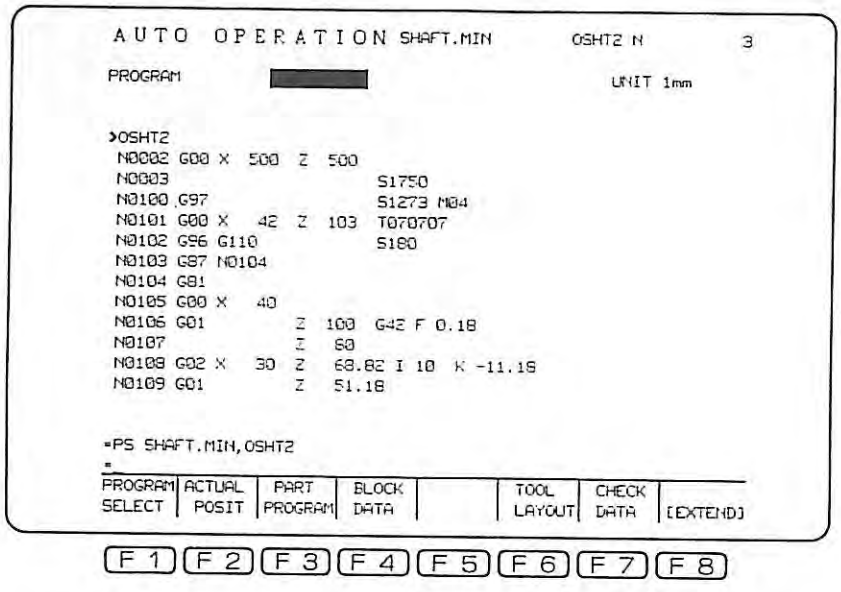
【 Supplement 】

The cursor control keys [↑] and [↓] may also be used for sequence number search operation:

Procedure:

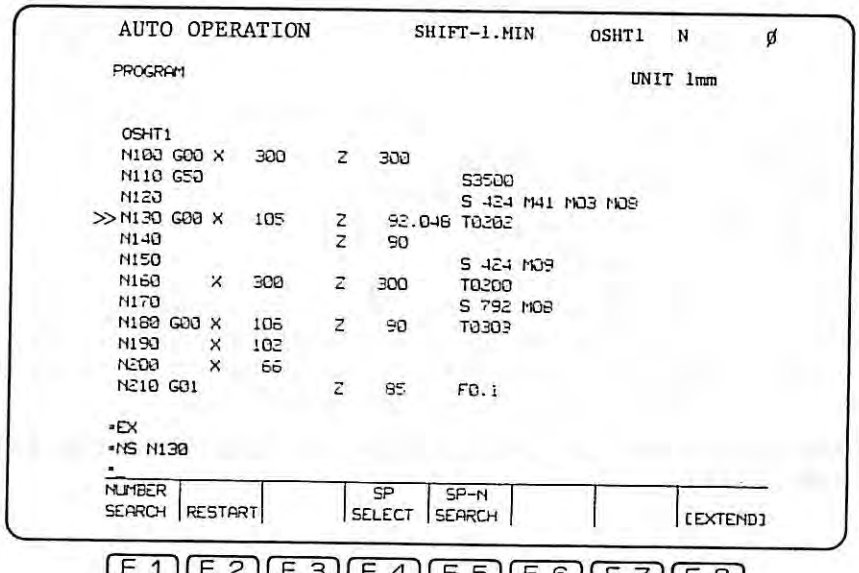
1) Follow steps 1) through 6) in 1-2. of this section.

2) Press [↓] key until the address pointer "≫" is placed at N130.



Note: *When the cursor control key is kept pressed, the address pointer "≫" can advance successively. If it passes the required block, press [↑] key to return it.*

3) When the address pointer "≫" reaches the N130 block, sequence number search operation is complete.



4) While holding down the **INTERLOCK RELEASE** key, press **CYCLE START** button to initiate the execution of the commands.

This feature allows the operator to check the contents of the selected program.

1-4. Operation Resumption after Manual Operation Intervention

This feature allows manual operation intervention in automatic mode operation, by interrupting automatic mode operation through the activation of slide hold or single block (including the M00 and M01 command); after the completion of manual operation, automatic mode operation can be resumed.

This feature can be effectively used when an abnormal condition which does not require emergency stop or control resetting occurs, such as chip entanglement, replacement of cutting tip and checking of finished dimensions.

This function can be activated for each spindle independently.

(1) Operation Procedure

Example:

Program

```
N101  G00  X800  Z200  S600  T0101  M42
N102       X132  Z60   M08   M03
N103  G01  X78         F0.3
N104  G00  X129  Z63
N105  G01        Z29
  :                :
  :                :
```

Manual operation intervention is performed while the program above is being executed by pressing the **SLIDE HOLD** button.

The sequence number when the slide hold function is activated is:

N105

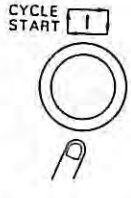
Coordinate point of the turret when the slide hold function is activated is:

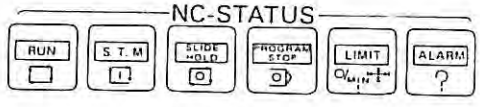
X = 129.000
Z = 43.256

Follow the steps below:

1) Press the **SLIDE HOLD** button.

This stops axis motion of both X- and Z-axis and turns on the SLIDE HOLD indicating lamp under NC STATUS, telling the operator that the system is in the slide hold state.





2) Press the **MID AUTO MANUAL** key on the machine operation panel.



3) Manual operation intervention:

Operative manual operation is as follows:

a) Manual axis feed

b) Axis feed by manual pulse handle

c) Spindle control (CW/CCW/STOP/JOG)

d) Change of spindle drive gear range

e) Turret indexing

When operation c), d) and/or e) is executed, the S.T.M indicating lamp starts flickering, telling the operator that S.T.M condition is different from the programmed condition.

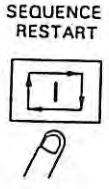
Other operation:

a) Change/setting of tool offset values:

When the tool offset data of the active tool offset number is changed, the new tool offset value is effective from the block after the read-in block displayed on the CRT when the program display mode is selected.

Read-in block: If the address pointer "≫" is on the CRT, the block identified is the read-in block. When such symbol is not on the CRT, the block identified by "↑" is the read-in block.

4) Returning to the point where manual operation intervention has been activated.



To return the axes to the point where manual operation intervention has been activated, simply press the **SEQUENCE RESTART** key.

- Pressing the **SEQUENCE RESTART** key returns the turret to the operation resumption point at a jog feedrate:

  X = 129.000
  Z = 43.256

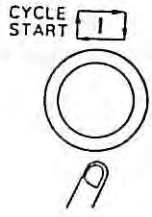
- If the active spindle speed, tool number and/or spindle drive gear range is different from the programmed one, the S.T.M indicating lamp keeps flickering, inhibiting the axes to return to the operation resumption point.

- When axis return to the operation resumption point is complete, the **MID AUTO MANUAL** indicating lamp turns off, indicating that return to the operation resumption point is complete.

- The **FEEDRATE** override switch setting is effective while the axes are returning to the operation resumption point.

- The **SLIDE HOLD** button is inoperative while the axes are returning to the operation resumption point. To stop axis feed, in this case, turn the **FEEDRATE** override dial to "0%" position.

5) Press the **CYCLE START** button.



With this, the interrupted program is continuously executed.

CAUTION

When the SEQUENCE RESTART key is pressed, the X- and Z-axis return to the operation resumption point simultaneously. It is advisable to bring the axes (turret) near the operation resumption point manually before pressing SEQUENCE RESTART key to avoid unexpected collision between the turret-mounted tools and the workpiece.

(2) Example

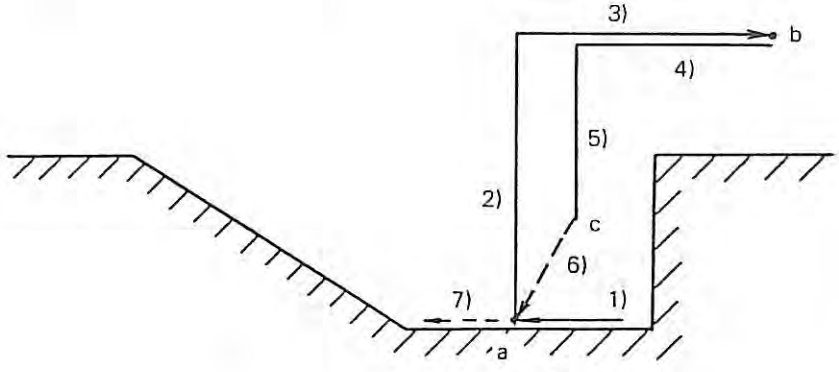


Fig. 4-3

1) Press the **SLIDE HOLD** button at point "a" during cutting in the AUTO mode. (The SLIDE HOLD indicating lamp goes on.)

2), 3) Press the **MID AUTO MANUAL** key (indicating lamp illuminates) to retract the cutting tool to point "b".

At point "b", necessary manual operations such as spindle stop, turret indexing, and change of tool offset value. (If S, T and/or M mode becomes different from what was active at point "a", the S.T.M. indicating lamp starts flickering.)

4), 5) Restore the S, T, and/or M mode as active at point "a", move the cutting tool to point "c"near point "a" (operation resumption point) in the MANUAL mode.

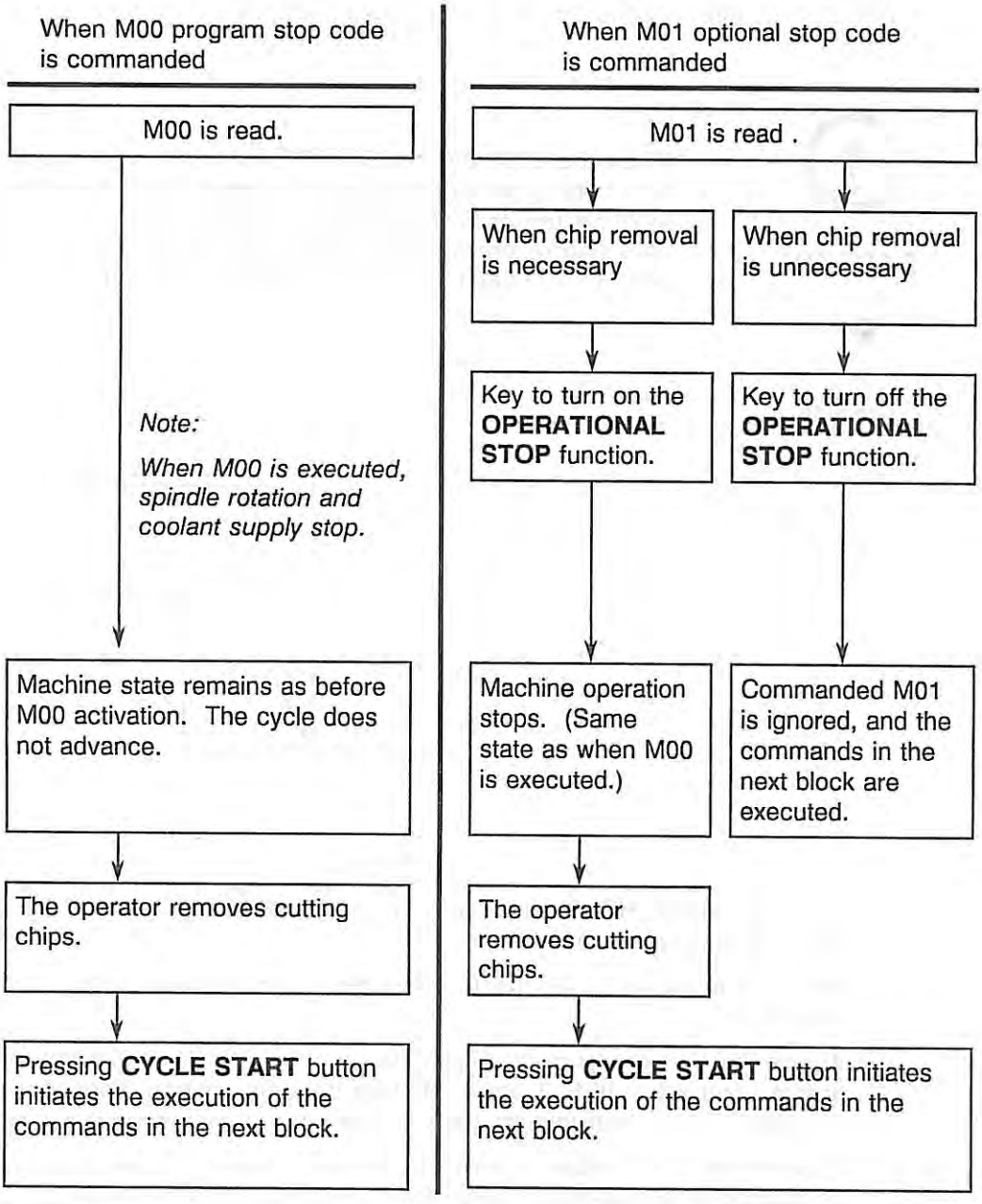
6) Press the **SEQUENCE RESTART** key, and the cutting tool is positioned to point "a" at a jog feedrate. As soon as positioning is complete, the LED in the **MID AUTO MANUAL** key turns off.

7) Press the **CYCLE START** button. The SLIDE HOLD indicating lamp turns off and the interrupted operation is resumed.

1-5. Other Operations while in Automatic Mode Operation

(1) To carry out manual operation to remove chips or to measure dimensions (Program Stop/Optional Stop):

Program stop (M00) or optional stop (M01) code.



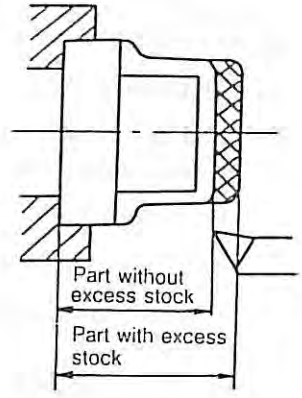
Spindle and coolant supply state is restored as before M00 activation.

As explained above, programmed M00 always interrupts machine operation while M01 leaves option to the operator whether cycle is to be stopped or continued according to the switch setting.

(2) Block Delete

This feature provides a means for skipping certain blocks in the program by programming a slash(/) code ahead of the block. This feature is useful when the operator desires to leave off certain cuts on a particular part configuration.

In case variations in stock removal are desired, the program is prepared for the workpiece with a maximum amount of stock removal. This can also be used for parts with a smaller amount of stock removal by setting the **BLOCK DELETE** key ON for these parts and eliminate the possibility of air-cutting. The direct advantage of this feature is a reduction in cycle time.



Example: Program a slash (/) code in the block containing the commands to cut excess stock as indicated by hatching lines in the figure above.

N101
/ N102
/ N103
/ N104
/ N105
N106

Program for cutting excess stock off the above part.

During operation, the **BLOCK DELETE** key gives the operator the programmed option to activate the programmed slash codes.

Parts with excess stock	Parts with excess stock
Machine Operation Panel	Machine Operation Panel
Press the **BLOCK DELETE** key to turn off the block delete function.	Press the **BLOCK DELETE** key to turn on the block delete function.
Programmed commands are executed in the order as programmed: N101 through N106. The excess stock is removed.	Blocks of commands N102 through N105, all preceded by a slash command (/), are ignored. And execution of the program jumps from N101 to N106.

(3) Checking New Program (Machine Lock, Dry Run)

It is very dangerous to cut parts with a new program without checking its contents. The Machine Lock and Dry Run features stated herein are effective to check the newly prepared program before using it to actually cut parts.

a) Machine Lock

With the Machine Lock feature, the operator can check the contents of the program without actually operating the machine.

Procedure:

1) Press the **MACHINE LOCK** key.

This locks output to the axis drive circuitry, the EC relay circuitry, and the spindle drive circuitry.

2) Press the **CYCLE START** button.

Execution of the program is initiated. The operator can check the contents of the programmed commands on the CRT.

Note: When feedrate commands (G01, G02, G03, G33, G34, etc.) are designated in the mm/rev (ipr) mode, the control executes them assuming spindle speed of designated in the program.

The spindle speed override, however, can be set as desired using a parameter. For details of parameter setting, refer to 3., "Operation in Parameter Mode" of this section.

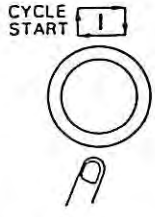
b) Dry Run

This feature permits the operator to check the actual tool path and other function executing points by actually operating the machine in a short time.

1) Press the **DRY RUN** key .



2) Press the **CYCLE START** button.



This starts the execution of the program. Take the following instructions into account.

Note 1: Feed commands in the following feed modes, such as G01, G02, G03, G33 and G34, are executed at the milling speed set by the parameter, usually set to 2400 mm/min.

Note 2: Therefore, feed commands F and vari-pitch commands E are all ignored.

Note 3: When both the Machine Lock and the Dry Run features are active, the Machine Lock feature precedes.

1-6. Scheduled Operation

To cut several different workpieces continuously, this scheduled operation feature is very effective. The scheduled operation feature permits programs to be automatically executed in the order specified in the schedule program, which is prepared according to the instructions provided below.

Assume 100 workpieces are to be cut, and the machine is equipped with a robot or auto-loader for automatic loading/unloading of workpieces.

The schedule program can be designated so that the part program is repeated 100 times in combination with the programs controlling loading/unloading cycles.

Or assume 20 pieces of workpiece A, 10 pieces of workpiece B and 50 pieces of workpiece C are to be machined on the machine equipped with an automatic loading/unloading unit.

With the machining programs for respective workpieces, and control programs for loading and unloading cycles prepared, the schedule program can specify the order of execution of those programs.

The schedule program features the following functions to perform the above indicated control:

(1) Function to select (or specify) a main program.

(2) Function to designate the execution order of the blocks of the schedule program.

(3) Function to count the number of cut workpieces, to receive signals from external switches and external control circuits, and to output signals to external control circuits. Also referred to as "variable setting function".

The preparation of a schedule program permits the effective use of the various functions. Continuous cutting of different workpieces can be performed in the automatic mode through the selection of the schedule program.

Scheduled operation is explained taking the following case as an example:

Continuous cutting of three different workpieces, A, B and C.

Workpiece A . . . . . 20 pieces
Workpiece B . . . . . 10 pieces
Workpiece C . . . . . 50 pieces

LIST BB1:SGEAR.SDF

```
N100   VSET V1=1
N101   PSELECT   LOADER.MIN
N102   PSELECT    GEAR.MIN,O001
N103   PSELECT    UNLOADER.MIN
N104   VSET      V1=V1+21
N105   IF [V1 LE 20]  N101
N200   VSET      V2=1
N201   PSELECT    LOADER.MIN
N202   PSELECT    GEAR.MIN,O002
N203   PSELECT    UNLOADER.MIN
N204   VSET  V2=V2+2
N205   IF [V2 LE 10]  N201
N300   VSET  V3=1
N301   PSELECT    LOADER.MIN
N302   PSELECT    GEAR.MIN,O003
N303   PSELECT    UNLOADER.MIN
N304   VSET  V3=V3+1
N305   IF [V3 LE 50] N301
N999   END
```

Note 1: *Main program file named GEAR.MIN contains the following machining programs:*

Machining program for workpiece A . . . . . Program No. "O001"
Machining program for workpiece B . . . . . Program No. "O002"
Machining program for workpiece C . . . . . Program No. "O003"

Note 2: *Programs controlling loading and unloading cycles are filed in the main program file, LOADER.MIN and UNLOADER.MIN.*

(1) Selection and Execution of Schedule Program

To perform the scheduled operation, first prepare a schedule program and then store it in the memory following the steps used in storing part programs. For details, refer to the instructions on Tape Reading-in Operation.

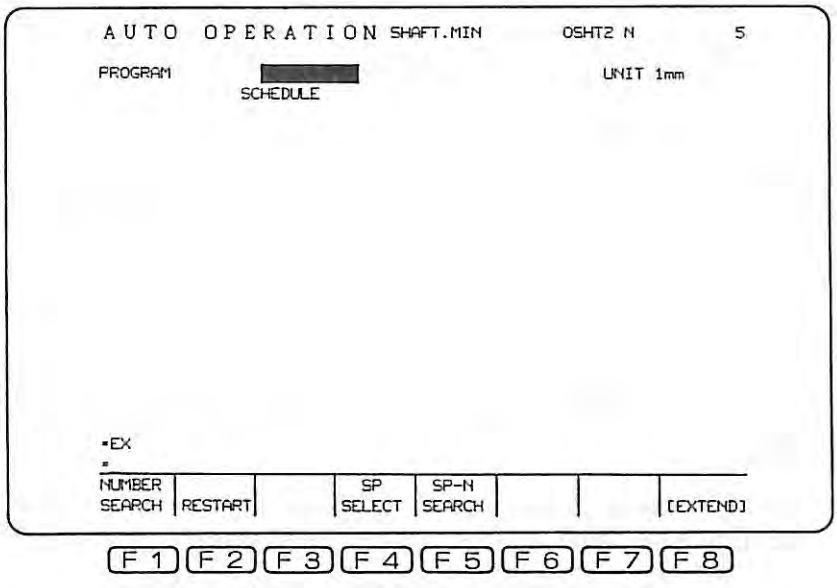
After that carry out the automatic operation steps as indicated below.

Procedure:

1) Select the AUTO OPERATION mode by pressing the **AUTO** key.

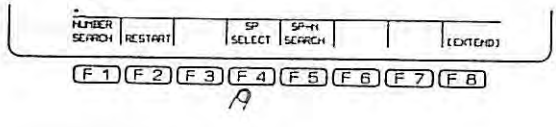


2) Display the SCHEDULE page by pressing [key] key.



3) Press the function key [F8] (EXTEND) to display the function by which schedule program selection is possible.

4) Press the function key [F4] (SP SELECT).

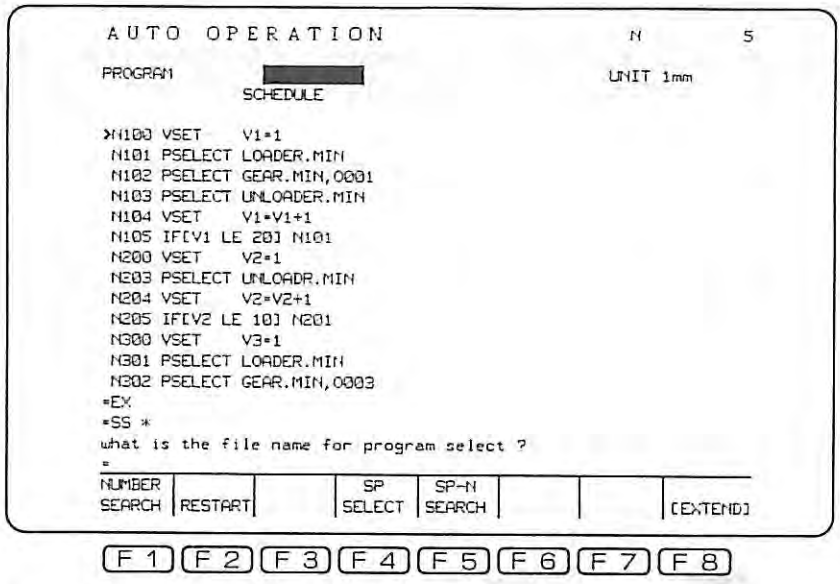


5) Key in an asterisk (*) through the keyboard.

=SS *

6) Press the **WRITE** key.

The program directory page will be displayed by the steps above. One page of this display shows a total of 12 file names and if more than 12 file names are registered, press the [key] key to display the second page.



7) Move the cursor to the file name desired.

8) Press the **WRITE** key.

   This selects the program and the CRT display is restored to the program display page with the program designated on it.

9) Press the **CYCLE START** button.

   This starts the execution of the selected program.

With the steps above, continuous operation begins according to the schedule program. The example program is used to machine 20 pieces of workpiece A, 10 pieces of workpiece B, and 50 pieces of workpiece C.

Note 1: *The schedule program selection operation (steps 5) - 8)) is also possible by directly inputting a file name. See 1-1., "Direct Designation of Programs".*

Note 2: *Select the schedule program after resetting the control.*

If it is selected while in operation, an error results.

Note 3: *When normal operation (automatic operation by selecting a program) is desired after selecting a schedule program, select the desired program again.*

Note 4: *When scheduled operation is initiated with the single block function activated, a main program is selected by the schedule program first and the control waits for cycle start operation.*

*Pressing the **CYCLE START** button after that initiates the execution of the commands in the first block of the main program, and normal single block mode operation is performed after that.*

Note 5: *When the optional **CYCLE STOP** switch is set ON in the scheduled operation mode, machine cycle stops each time one main program is completed. Pressing the **CYCLE START** button resumes the operation, i.e., executes another cycle.*

Note 6: *To execute the schedule program on the machine equipped with the robot, bar feeder or other automatic loading/unloading system, it is possible to place the control in the cycle start ready state after the completion of a program (M02, M30 executed) by using an external cycle stop signal.*

For instance, the schedule program below executes the main program A.MIN infinitely. When an external cycle stop signal is input after the completion of a lot or when the bar material is used up from an external device to the NC, it is placed in the cycle start signal waiting state after the completion of the main program.

```
A.SDF
PSELECT A.MIN,,,Q9999
END
```

Repetition of the program infinitely

Note 7: *When the **RESET** button is pressed while in the scheduled operation mode, and when the interrupted program is then resumed, the program selected at that time is executed from the beginning.*

*When the program selection block in the schedule program contains the specified number of main program repetitions, the cycle interrupted is not counted. For instance, if the number of repetitions is specified as "10", and when the **RESET** button is pressed while the 8th cycle is being executed, the control does not count the interrupted cycle. Pressing the **CYCLE START** button resumes the cycle from the 8th cycle.*

That is, counting-up of the number of cycle repetitions is activated when M02 (or M30) is executed.

(2) Schedule Program Number Search

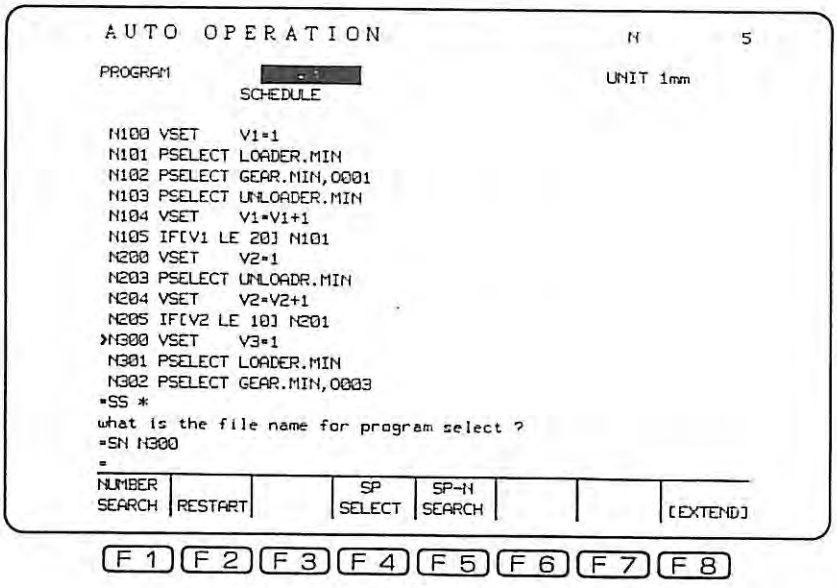
When machining only 50 pieces of workpiece C using the example schedule program, without machining workpieces A and B, the schedule program number search function is convenient.

After the selection of the schedule program following the steps 1) - 6) in (1), conduct the schedule program number search operation for block N300.

Procedure:

1) Press the function key [F5] (SP-N SEARCH).
2) Key in "N300".
3) Press the **WRITE** key.

Sequence number search operation is executed for block N300 and the symbol "≫" is placed at the left of N300 displayed on the screen.



4) Press the **CYCLE START** button.

With the steps above, 50 pieces of workpiece C are machined.

【 SUPPLEMENTS 】

The cursor control keys ↑ and ↓ can be used for schedule program search operation.

The keys are operative when the control is reset, the automatic operation mode is selected, and the CRT displays the schedule program. Follow the steps below, instead of carrying out steps 1) through 3).

1) Press cursor control key ↓ .

This advances "≫" by one block.

Hold down ↓ key until the address pointer "≫" reaches the N300 block.

Note: While the cursor control key is held down, the address pointer "≫" advances successively. If it passes the required block, press ↑ key to return it.

When the address pointer "≫" reaches the N300 block, schedule program search is complete.

2) Press the **CYCLE START** button.

Note: When conducting schedule program search, always perform schedule program selection operation and cancel the previous schedule.

1-7. CRT Display while in Operation

CRT displays which are often used in machine operation are explained in 3-2-4. and 3-2-5. of Section 3. This paragraph deals with all CRT displays available during machine operation.

As seen on the following pages, there is no difference in CRT displays between the selected operation modes, AUTO, MDI and MANUAL with the exception of the tool layout screen in the AUTO mode.

The tool layout screen which is available only in the AUTO mode and the functions accessible from this screen are explained in detail in 1-8. "Tool Layout Function" of this section. Thus, this section provides explanations of other screens which are common to all operation modes.

CRT displays are largely divided into four modes, depending on the pressed functions keys, [F2](ACTUAL POSIT), [F3] (PART PROGRAM), [F4] (BLOCK DATA) and [F7] (CHECK DATA).

(1) Actual position data . . . . . [F2]
(2) Main program . . . . . [F3]
(3) Programmed data in one block . . . . . [F4]
(4) Check data . . . . . [F7]

Since it is not necessary to read check data during normal operation, the function key [F7] is separated from other function keys.

By setting proper parameter bit data, display of corresponding pages can be disabled. See below.

Parameter bit No. 18

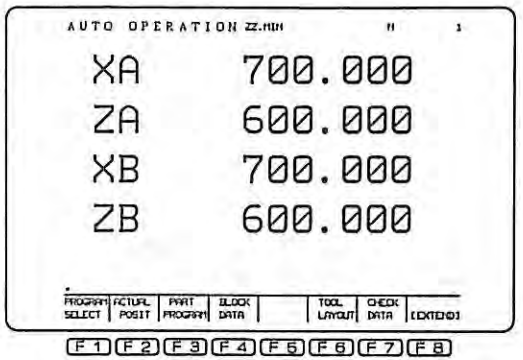
bit 0: Axis data
bit 1: EC input 1, 2
bit 2: EC input (extended) 1, 2
bit 3: EC output 1, 2
bit 4: EC output (extended) 1, 2
bit 5: Machine operation panel input 1, 2, 3
bit 6: Machine operation panel output 1, 2, 3
bit 7: Specification code

Parameter bit No. 35

bit 0: EC input (extended) 3, 4
bit 1: EC output (extended) 3, 4

Provided on the following pages are the explanations for respective CRT displays.

Operation Mode	Command and Function Mode			CRT Display
AUTO MDI MANUAL	F2	ACTUAL POSIT	Used to display coordinates of the actual position.	Display of actual position data in enlarged characters (1-1) Simultaneous display of actual position data and program (1-2) Actual position display with display of machine status and load condition (1-3) Simultaneous display of remaining axis motion amount and actual position data (1-4)
	F3	PART PROGRAM	Used to display a part program.	Display of the program (2-1) Display of the data entered through the keyboard in MDI mode (2-2) Display of schedule program (2-3)
	F4	BLOCK DATA	Used to display the data in one block.	Display of the data in the currently executed block (3-1) Display of the data in the block to be executed next (3-2) Display of the data in the block after the next block (3-3) Display of the data in the block stored in the buffer register last (3-4)
	F6	TOOL LAYOUT	Used to display the data of tools to be used.	
	F7	CHECK DATA	Used to display various check data.	Status display of diagnosis data (4-1) Display of the data related to axes (4-2) Status display of EC input signal (No. 1) (4-3) Status display of EC input signal (No. 2) Status display of extended EC input signal (No. 1) Status display of extended EC input signal (No. 2) Status display of extended EC input signal (No. 3) Status display of extended EC input signal (No. 4) Status display of EC output signal (No. 1) Status display of EC output signal (No. 2) Status display of extended EC output signal (No. 1) Status display of extended EC output signal (No. 2) Status display of extended EC output signal (No. 3) Status display of extended EC output signal (No. 4) Status display of machine operation panel input signal (No. 1) Status display of machine operation panel input signal (No. 2) Status display of machine operation panel input signal (No. 3) Status display of machine operation panel output signal (No. 1) Status display of machine operation panel output signal (No. 2) Status display of machine operation panel output signal (No. 3) Display of specification codes (No. 1) (4-4) Display of specification codes (No. 2) (4-5) Display of machine model codes (4-6) Display of machine specification codes (4-7) Alarm message list (4-8)

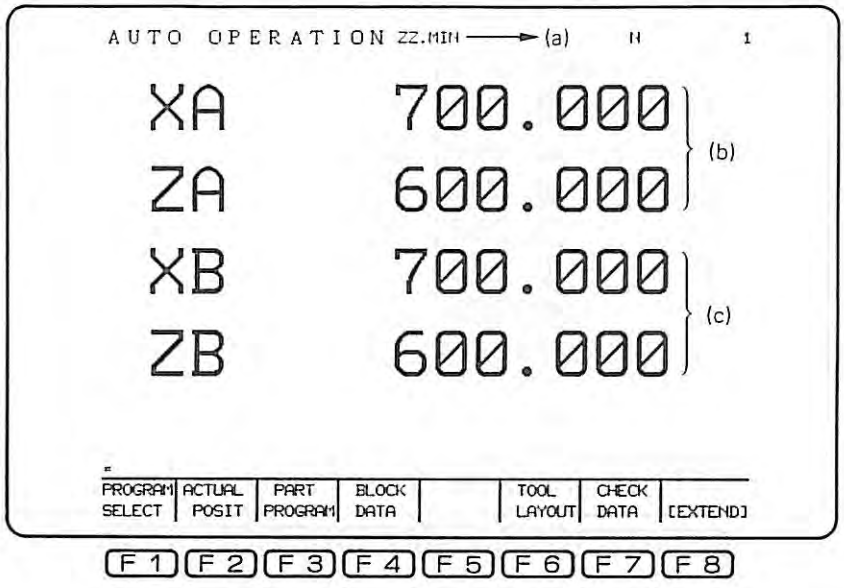


These are explained in the Maintenance Manual.

Details of respective CRT displays are provided in the following pages.
The CRT display changes in the order indicated by an arrow mark when [P↓] key is pressed.

(1) Display of Actual Position Data

(1-1) Display of Actual Position Data in Enlarged Characters

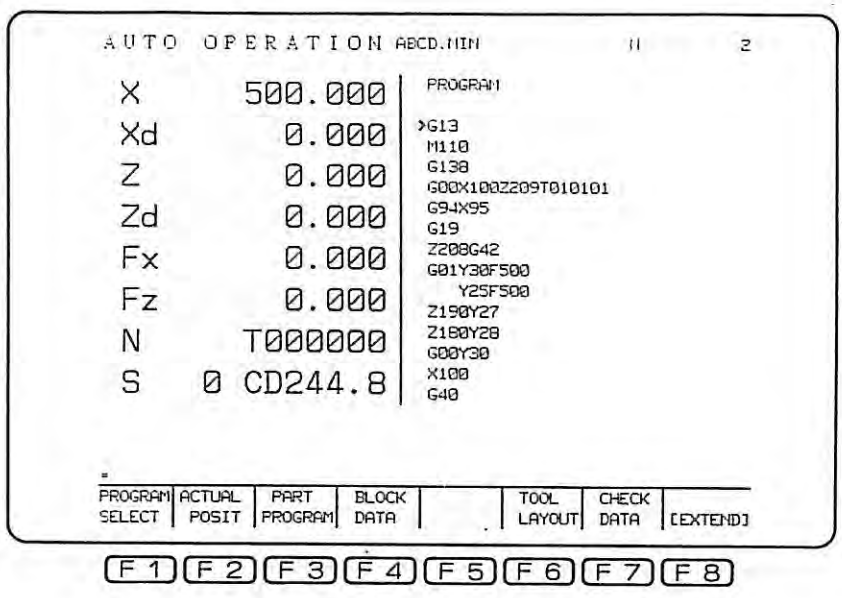


(a) Indicates the operation mode currently selected.

(b) Actual position data of A-turret

(c) Actual position data of B-turret (two-saddle, or two-turret models)

(1-2) Simultaneous Display of Actual Position Data and Program



The program number (name) of the main program selected by program selection operation is displayed.

The file name of the main program selected by program selection operation is displayed.

Left section:

- X : Actual position of the X-axis (program)
- Xd : Remaining X-axis movement distance to the target point
- Z : Actual position of the Z-axis (program)
- Zd : Remaining Z-axis movement distance to the target point
- N : Sequence number
- S : Spindle speed (min^{-1} {rpm})
- FX : X-axis feedrate (mm/min)
- FZ : Z-axis feedrate (mm/min)
- T : T command
- CD : Actual position of the spindle (deg.)

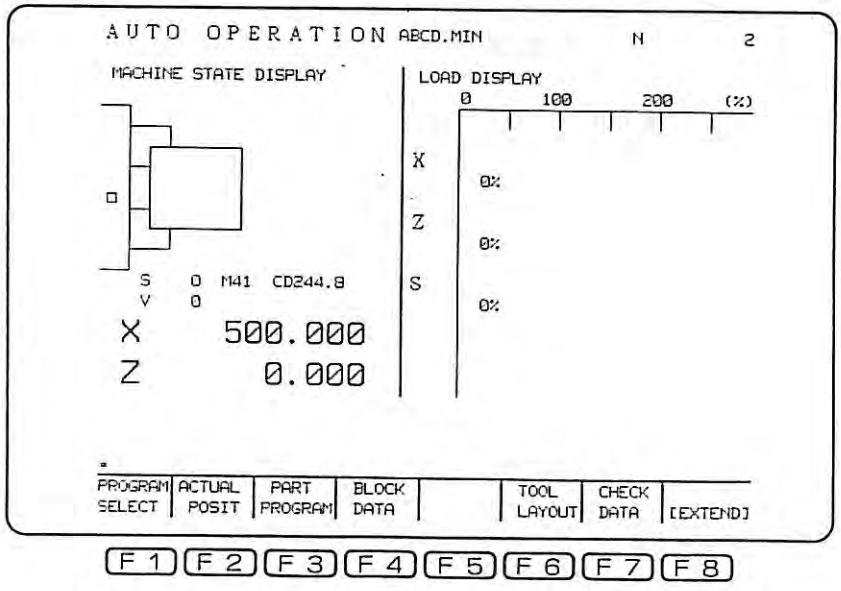
Display of actual spindle position can be omitted by setting "1" at bit 6 of parameter (bit) No.19. (Initial setting: 0 (OFF))

Right section:

Program presently selected.

The line identified by the up-arrow (↑) mark is the block being executed. Blocks up to the one identified by the symbol (≫) preceding the sequence number N are read and stored in buffer.

(1-3) Actual Position Display with Display of Machine Status and Load Condition



Left section:

Actual position display

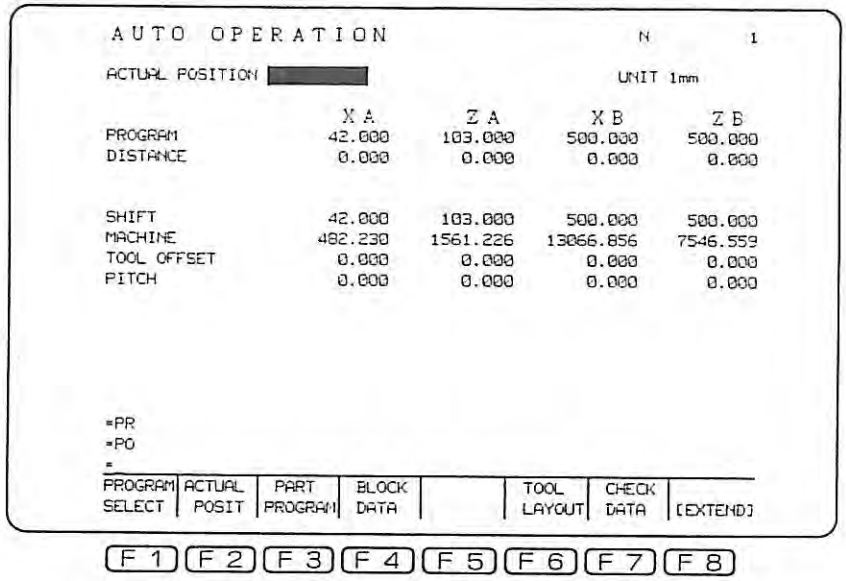
X : Actual position of X-axis (program)
Z : Actual position of Z-axis (program)
M : Spindle gear range
S : Spindle speed
V : Cutting speed
CD : Actual position of spindle

Machine status

Right section:

Load status of spindle, X- and Z-axis

(1-4) Simultaneous Display of Remaining Axis Motion Amount and Actual Position Data



(a) Indicates the program number (name) of the main program selected by program selection operation.

(b) Indicates the file name of the main program selected by program selection operation.

(c) Programmed position data of A- and B-turret

(d) Remaining axis motion amount of A- and B-turret

(e) Actual position of A- and B-turret on the offset coordinate system

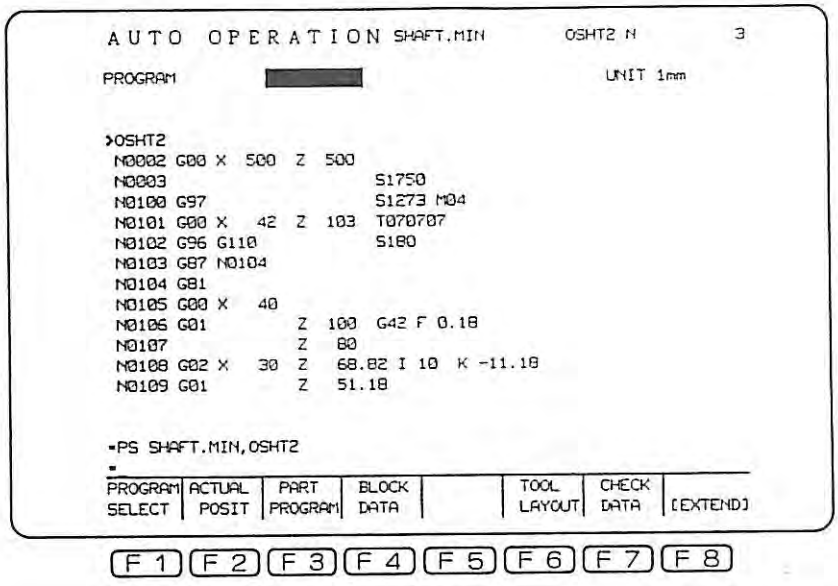
(f) Position of A- and B-turret referenced to machine zero

(g) Active tool offset amounts of A- and B-turret

(h) Ball screw pitch error compensation

(2) Display of Part Program

(2-1-1) Display of Program



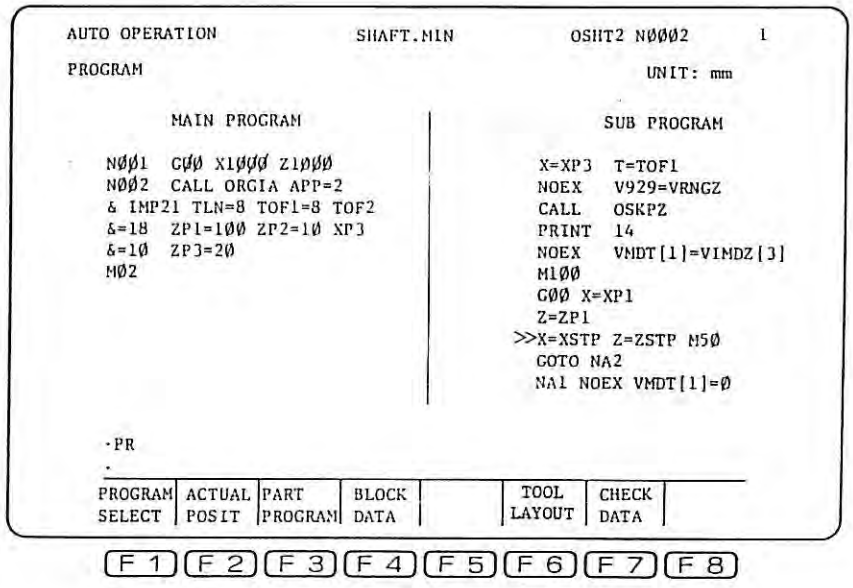
(a) Indicates the block counter number.

(b) Indicates the number (name) of the sequence currently executed.

(c) Cutting is carried out according to the commands in the block identified by "↑" preceding sequence number.

(d) The commands in the blocks up to the one identified by "≫" are read into the buffer registers.

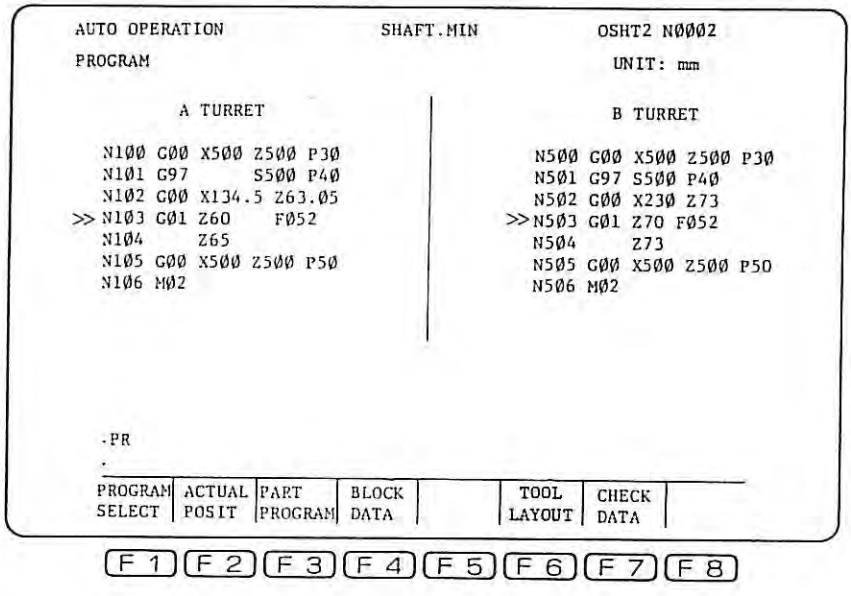
(2-1-2) Simultaneous Display of Main Program and Subprogram



On the PROGRAM page*, the subprogram and the main program from which the subprogram is called are displayed on one screen.

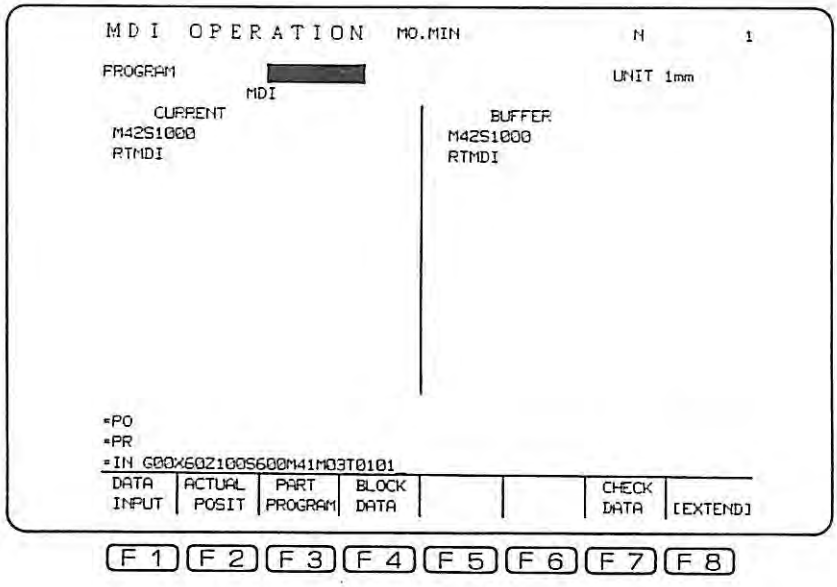
* Except display on which programs of both saddles A and B are displayed.

(2-1-3) Simultaneous Display of Program for Saddles A and B (only for two-saddle model)



For two-saddle models, programs for saddles A and B are displayed simultaneously.

(2-2) Display of Data Entered through Keyboard in MDI Mode



(a) Turret selection is possible by pressing  or  key.

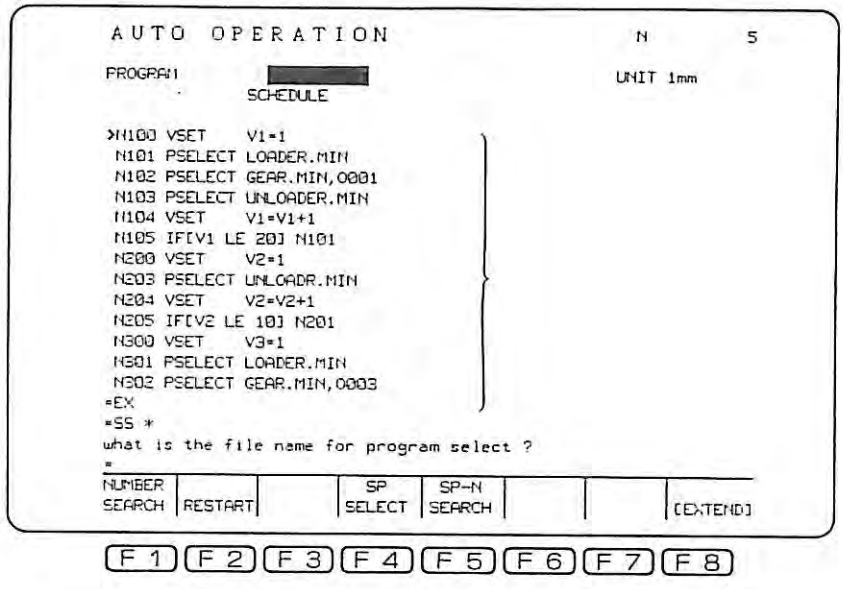
(b) Indicates the data stored in the buffer area.

The data are keyed in into this area by pressing the function key [F1] (DATA INPUT).

(c) Indicates the data stored in the active area.

Pressing the **CYCLE START** button transfers the data in the buffer area to the active area; the transferred data are then executed.

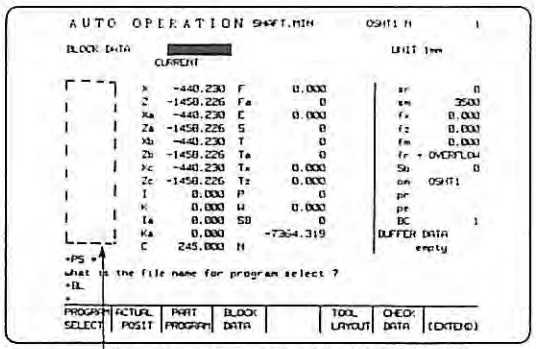
(2-3) Display of Schedule Program



(a) Displays schedule program.

(3) Display of Data in One Block

(3-1) Display of Data in Currently Executed Block



Effective G and M codes

Displays one block of the data.

The data below are always the active data, whichever page the operator selects.

Description of respective data:

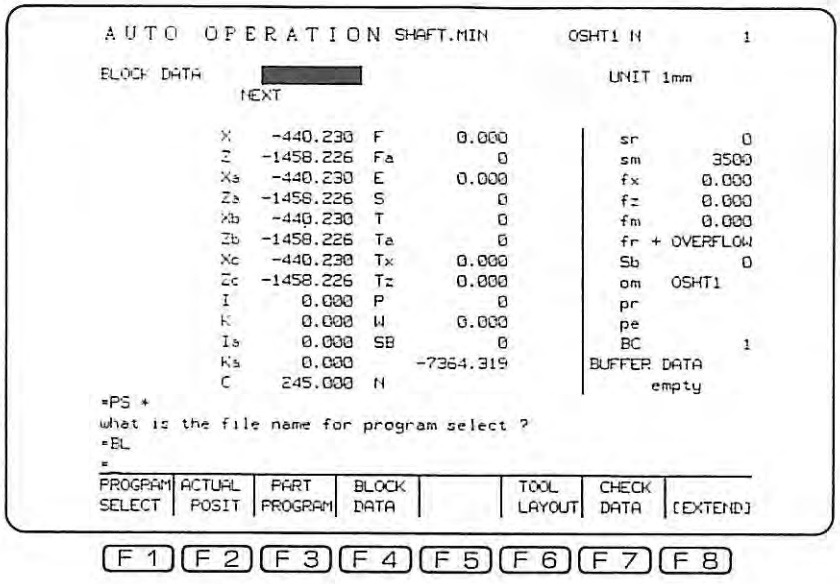
X	Target value : X-axis
Z	Target value : Z-axis
Xa	Target value : X-axis, in G50 thread cutting mode
Za	Target value : Z-axis, in G50 thread cutting mode
Xb	Starting point : X-axis, thread cutting
Zb	Starting point : Z-axis, thread cutting
Xc	Target point: X-axis, thread cutting
Zc	Target point: Z-axis, thread cutting
I	Programmed I value
K	Programmed K value
Ia	End point : I
Ka	End point : K
C	Target value : C-axis
F	Programmed F value
Fa	Programmed dwell length
E	Programmed E value
S	Programmed S value
T	Tool number
Ta	Tool offset number
Tx	Tool offset value: X-axis
Tz	Tool offset value: Z-axis
P	Programmed P value
W	Target value; W-axis
SB	Rotary tool command
N	Sequence name

Sr	Spindle speed (rpm)
Sm	Maximum spindle speed (rpm)
fx	Feedrate of X-axis (mm/min.)
fz	Feedrate of Z-axis (mm/min.)
fm	Axis traverse speed (mm/min.)
fr	Axis traverse speed (mm/rev.)
Sb	Rotary tool spindle speed (rpm)
Om	Main program name
Pr	Programmed number of main program repetitions
Pe	Actual program executed number
BC	Block counter
full	Data in buffer register
empty	Data not in buffer register

(These data are available only while in scheduled operation mode.)

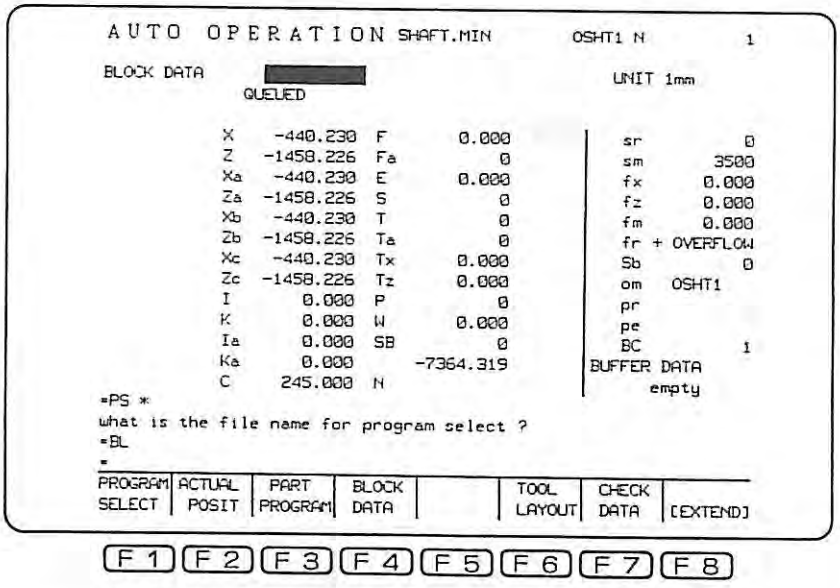
In this display S, T or M character blinks while S, T or M function is being executed. This blink display is canceled when an answer signal is returned or the NC is reset.

(3-2) Display of Data in The Block to Be Executed Next



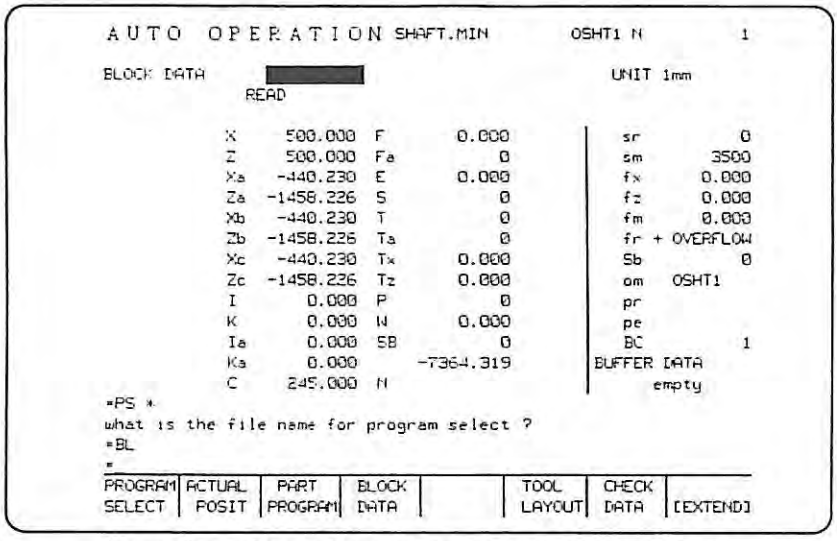
(a) Displays the data in the block to be executed next.

(3-3) Display of Data in The Block to be Executed after The "Next" Block Shown in (3-2)



(a) Displays the data in the block after the "Next" block.

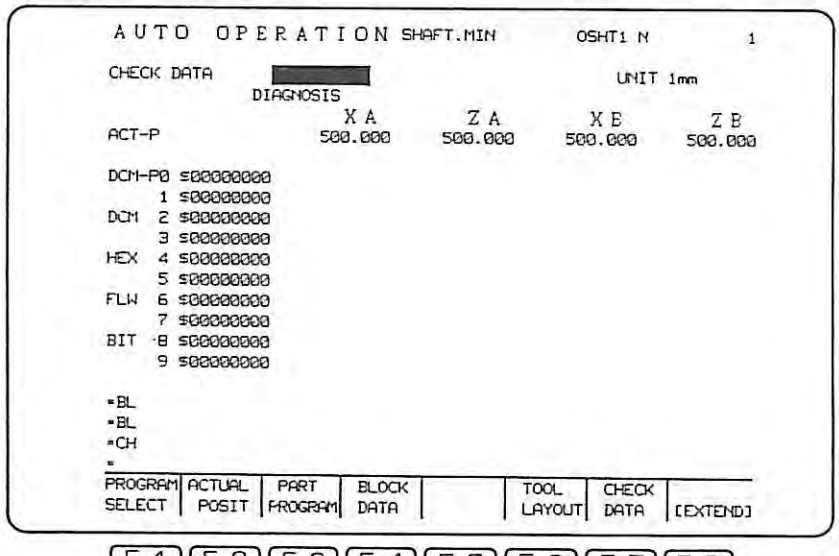
(3-4) Display of Data in The Block Stored in The Buffer Register Last



(a) Displays the data in the block stored in the buffer register last.

(4) Display of Various Check Data

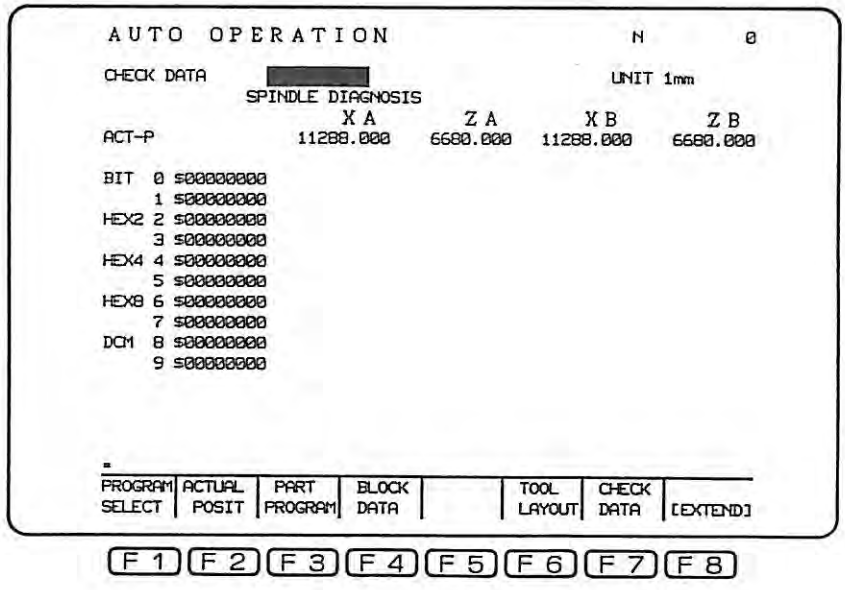
(4-1) Status Display of Diagnosis Data



This screen is provided for checking the machine and control status. The information is necessary only for maintenance and thus displaying of this page is not required unless requested by Okuma.

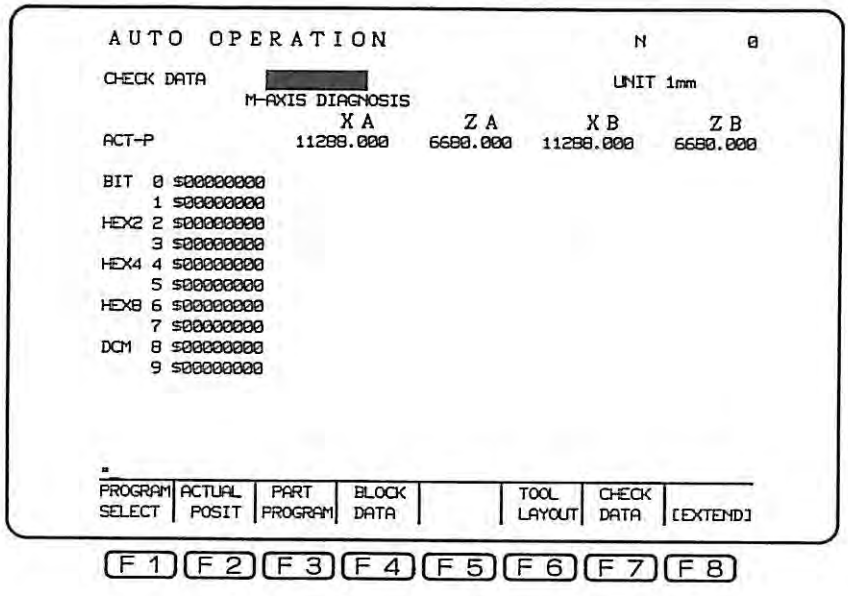
The following displays are added when the high-precision VAC specification is selected for the spindle and M-tool spindle.

(4-1-a) Status Display of Spindle Diagnosis



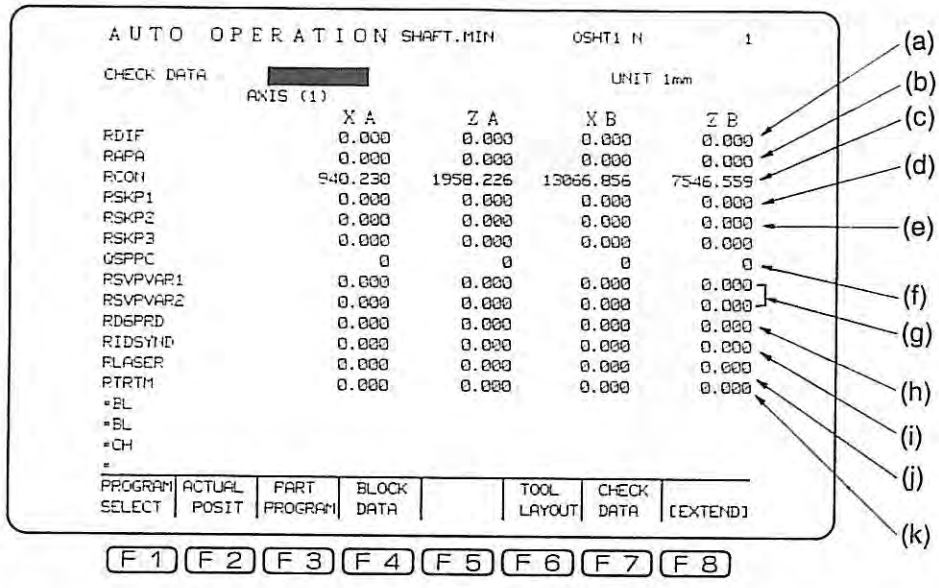
This screen is provided for checking the VAC status. The information is necessary only for maintenance and thus displaying of this page is not required unless requested by Okuma.

(4-1-b) Status Display of M-tool Spindle Diagnosis



This screen is provided for checking the VAC status. The information is necessary only for maintenance and thus displaying of this page is not required unless requested by Okuma.

(4-2) Display of Data Related to Axes



(a) Follow error of axis drive servomotors

Difference between the calculated value (RCON) and output from the position encoder (RAPA)

(b) Actual position data of the OSP position encoder read

(c) Calculated current (actual) position data

(d) Coordinates of touch point (sensor 1)

(e) Coordinates of touch point (sensor 2)

(f) Spindle position (4096/turn)

(g) Servo data

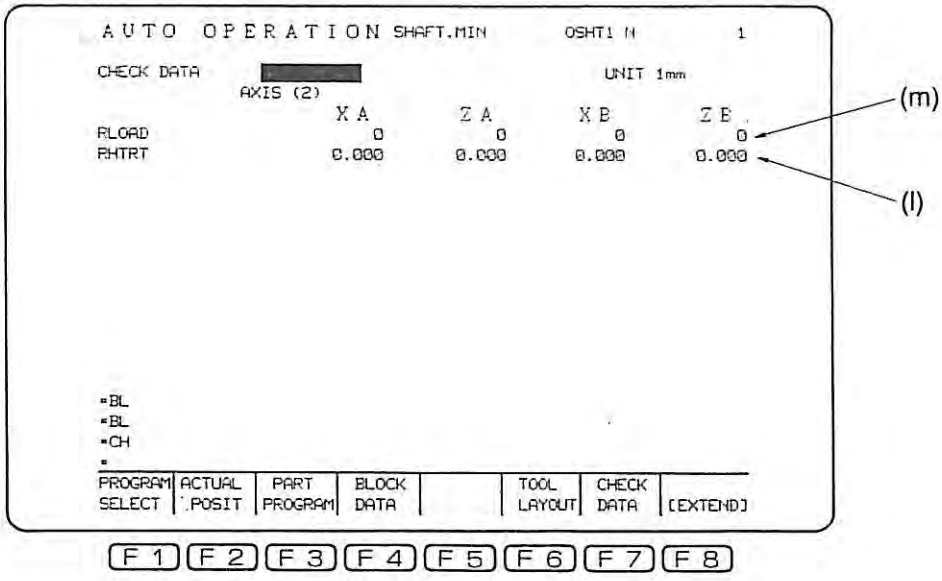
(h) Output from type F position encoder

(i) Output from Inductosyn scale

(j) Output from laser beam measuring device

(k) Tool rotation time (Time duration between the start of unclamp and the completion of unclamp)

The same data is displayed for the X and Z axes for each turret. With the mirror image specification, the data for the turret to be displayed can be selected by pressing either A or B key on the operation panel.



(l) Rotation speed of high-speed turret (rpm)

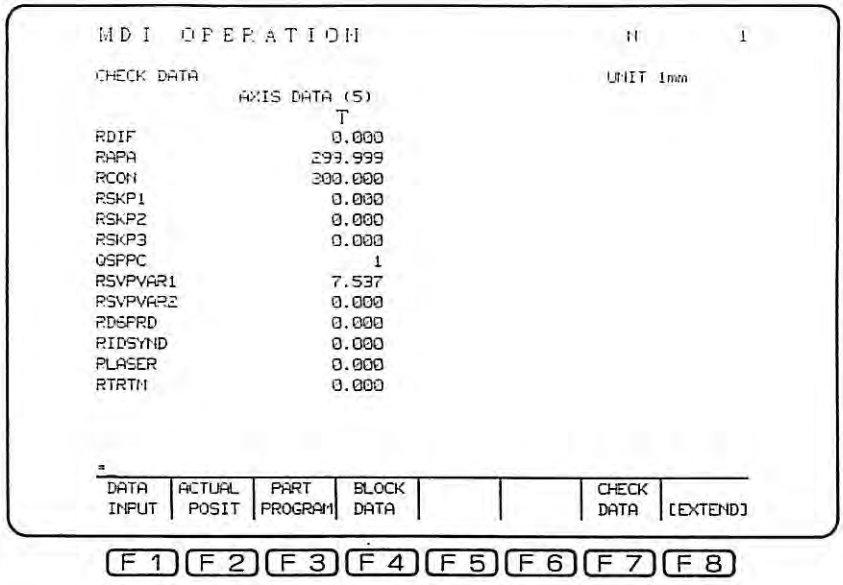
(m) Torque

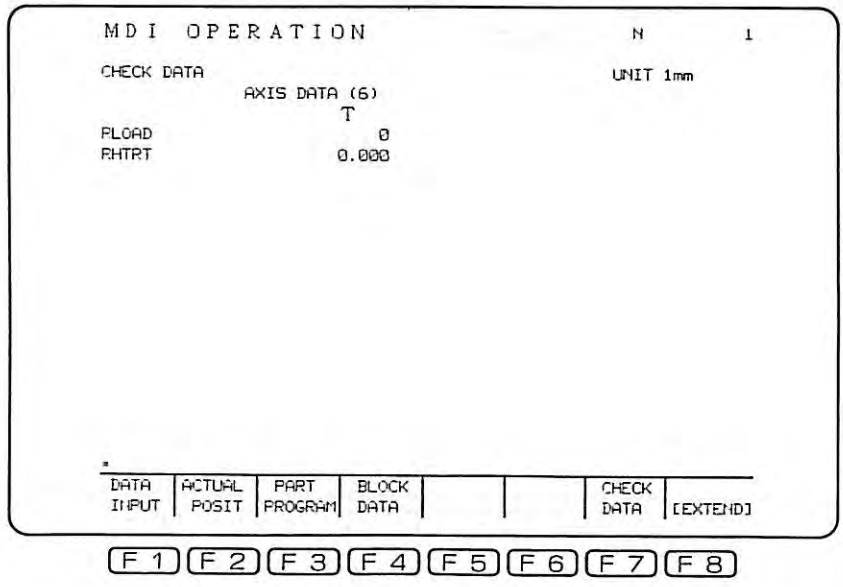
Torque* of individual axis drive servomotors is displayed.

100% at rated torque

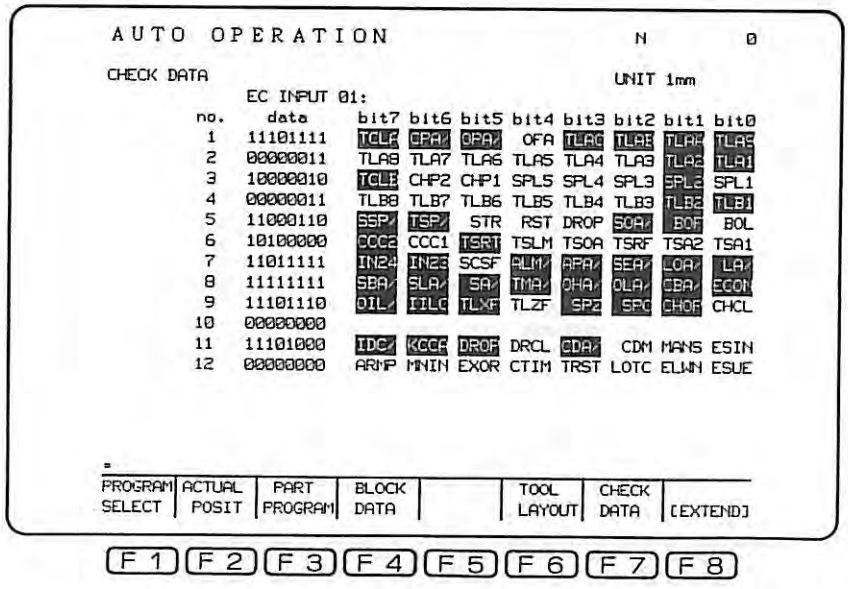
* Value calculated by servo processor

* Two axis data-related screen papers are with the LB 15Ⅱ.



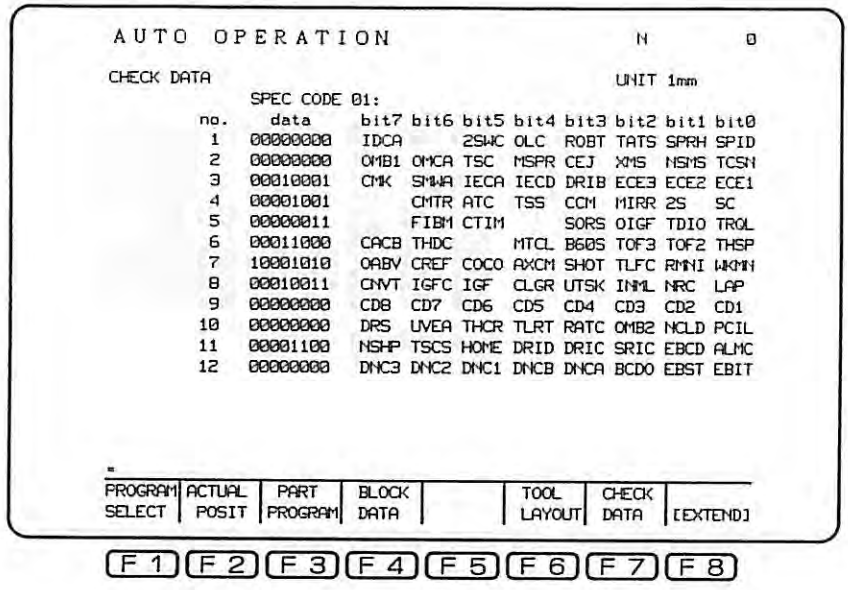

MDI OPERATION
N 1
CHECK DATA
UNIT 1mm
AXIS DATA (6)
T
PLOAD 0
RHTRT 0.000
DATA INPUT
ACTUAL POSIT
PART PROGRAM
BLOCK DATA
CHECK DATA
[EXTEND]
F1 F2 F3 F4 F5 F6 F7 F8

(4-3) Status Display of EC Input Signal (No. 1)



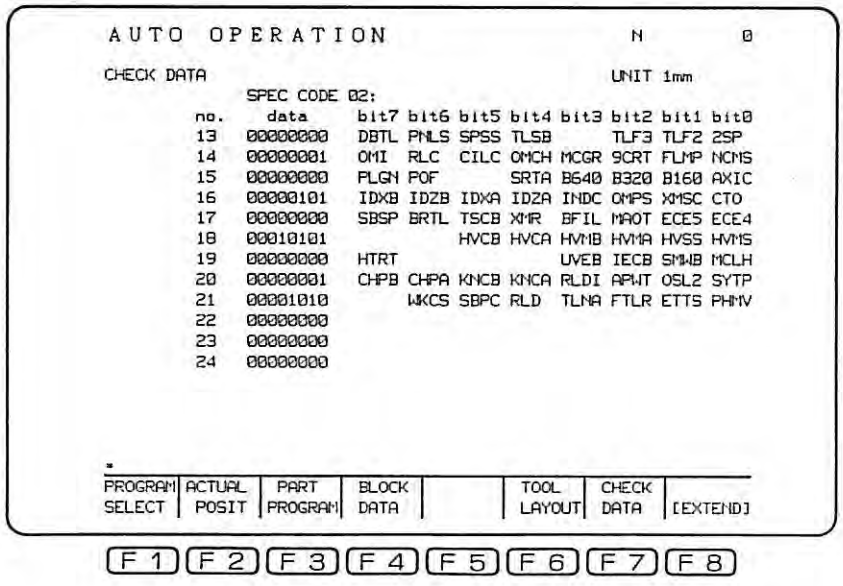
No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
1	Turret A clamp	Chuck pressure low/	Hydraulic source pressure low/	Alarm - oil filter clogged	← Rotary switch - turret A, #9 through #12 →			
2	← Rotary switch - turret A, #1 through #8 →							
3	Turret B clamp	Chuck pedal #2	Chuck pedal #1	← Spindle gear confirmation limit switch →				
4	← Rotary switch - turret B, #1 through #8 →							
5	External cycle stop/	External slide hold/	External cycle start	External reset	Door open confirmation switch	Spindle lubrication pressure low/	Slideway lubricating oil flow	Slideway lubricating oil level low
6	Chip cover close confirmation switch 2	Chip cover close confirmation switch 1	Tailstock spindle retraction confirmation switch	Tailstock spindle in-position confirmation switch	Tailstock spindle over-advance confirmation switch	Tailstock spindle retraction foot switch	Tailstock spindle advance foot switch 2	Tailstock spindle advance foot switch 1
7	External input 24	Emergency limit switch release	Spindle orientation completion	External alarm/	SMW chuck pressure low/	Alarm - travel end/	Alarm - LDU overload/	Alarm-static pressure unit cooler
8	Spindle preparation completed	Alarm - spindle overload/	Alarm - SDU/	Alarm - CPU temperature/	Alarm - transformer overheat/	Alarm - EC overload/	Alarm - EC circuit breaker/	Emergency stop
9	External interlock/	Internal interlock released	Turret X-axis free	Turret Z-axis free	Spindle zero speed	Spindle at constant speed	Chuck open confirmation switch	Chuck close confirmation switch
10								
11	Index chuck completion limit/	Sensor protect limit/	Door open	Door close	Coupling device alarm/	Automatic mode	Answer for spare M code	External spindle jog
12		Operation monitor external input	External spindle orientation	Calendar timer input	CEJ transfer start	Lot complete		External start disabled

(4-4) Display of Specification Codes (No. 1)



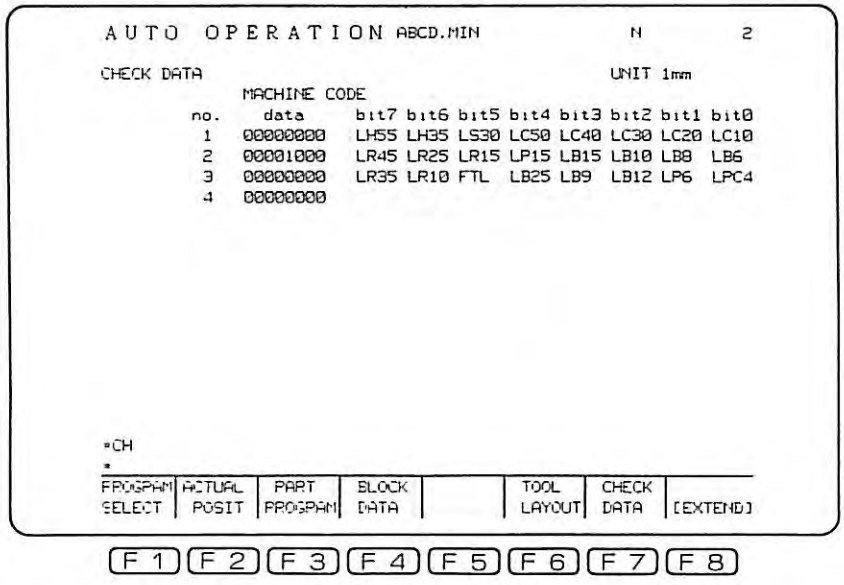
No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
1	Index chuck		2-saddle work catcher	Overload detection	Robot (Okuma)	Program-mable tailstock	Orientation (electric)	Orientation (pin/brake)
2	Load monitor - B	Load monitor - A	Touch setter	Gauging data print	CEJ matic gauging	Post-process gauging	Tool gauging system	Touch sensor measuring
3	Chucking miss detect	Air chuck	IEC chuck	IEC door	Door interlock B	EC board (card) addition 3	EC board (card) addition 2	EC board (card) addition 1
4		Gang tooling	LC40M-ATC	Tailstock swing	Multiple machine	Mirror image	4-axis 2-saddle machine	Center work
5		Floppy IN/OUT (IBM)	Calendar timer		Edit interlock	One-touch IGF	Tape data input/output	NC torque limiter
6	C-axis joint B (high-speed)	Thread start paint offset		Cycle time calculation	Buffer 60 m	Tool offset 96 sets	Tool offset 64 sets	G834/35 slide hold
7	Expand auto chamfering	Profile generation	Coordinate system conversion	Pitch error compensation	Cycle time reduction	Tool life management	NC work monitor	NC work counter
8	Tape convert	IGF convert	IGF	Color graphic	User task 2	Inch/Metric switchable	Nose R 2B	LAP
9	Coupling spec. 8	Coupling spec. 7	Coupling spec. 6	Coupling spec. 5	Coupling spec. 4	Coupling spec. 3	Coupling spec. 2	Coupling spec. 1
10	Door open/ close special	Front drive air chuck	Arc thread	Tool retract cycle	LR15M-ATC	Load monitor B2	NC loader	Parts catcher interlock
11	NC torque limiter (with share pin)	Touch setter (slide)	Home position	Door interlock D	Door interlock C	Spindle rotation interlock	Program select C2	Alarm C output
12	DNC-C3	DNC-C2	DNC-C1	DNC-B	DNC-A	← External program selection → C	B	A

(4-5) Display of Specification Codes (No. 2) (OSP5020L/OSP500L-G)



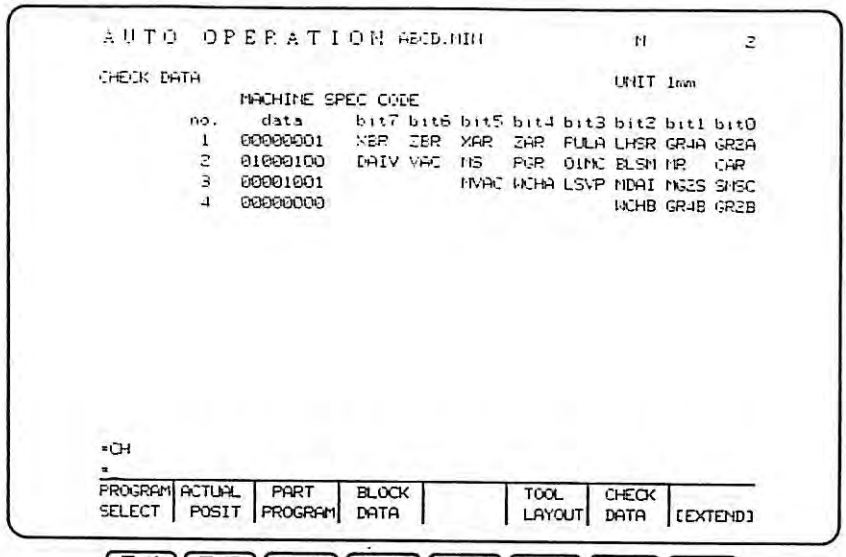
No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
13	Double tooling	Panel position special	Program stop	Tool rotation type B		Tool life management 96 sets	Tool life management 64 sets	2-spindle machine
14	OMI message display	Robot/loader interface C	Chuck interlock release	Old mono. character	Mono. graphic	OSP500L-G panel	Flat panel	NC master
15	Flat turning	Pick off		Tailstock motion in spindle rotation	Buffer 640 m	Buffer 320 m	Buffer 160 m	Inductosyn pitch error compensation
16	← X_B	Z_B	Inductosyn X_A	Z_A →	C	Load monitor switch	Size catcher	Cycle time over
17	Sub spindle	M-axis at B side	Touch setter at B side	RS232C gauging	Bar-feeder/chuck interlock	M-tool spindle orientation	EC board addition 5	EC board addition 4
18			C-axis 2 VAC	C-axis 1 VAC	MB-axis VAC	MA-axis VAC	2nd spindle VACII	Main spindle VACII
19	High-speed turret				← 2nd spindle chuck → UVE	IEC	SMW	M-axis 1 point clutch
20	Chuck high/low pressure special 2nd spindle	Chuck high/low pressure special main spindle	KNCS chuck 2nd spindle	KNCS chuck main spindle	Robot/loader door auto open/close I.L	System air pressure monitor	Okuma simple loader	Synchronized tapping
21	Sequencer abnormal	Work count special	2S sub spindle parts catcher	Robot/loader interface D	NC turret	FTL R side	Simple tow along	Pulse handle portable type
22								
23								
24								

(4-6) Display of Machine Codes



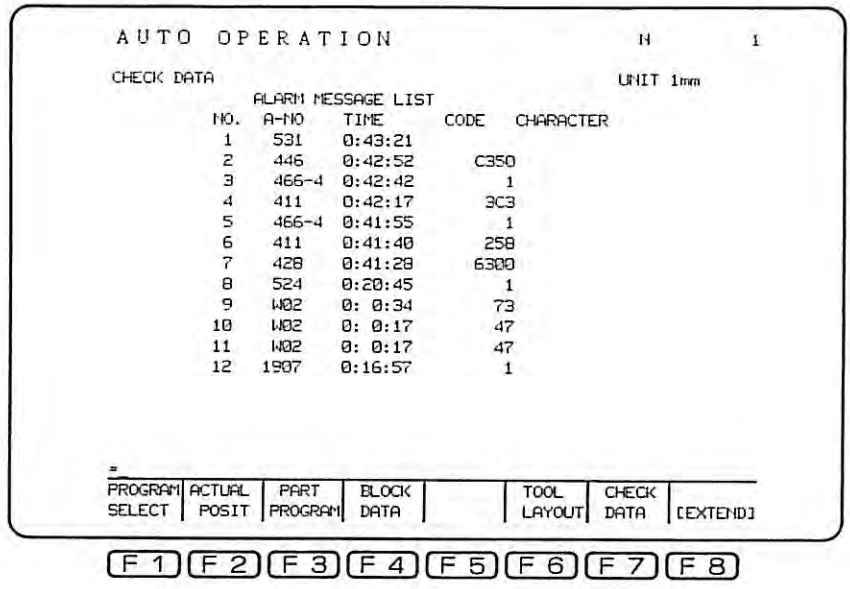
No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
1	LH55	LH35	LS30	LC50	LC40	LC30	LC20	LC10
2	LR40	LR30	LR20	LP15	LB15	LB10	LB8	LB6
3	LR35	LR10	FTL	LB25	LB9	LB12	LP6	LPC4
4								
5								
6								
7								
8								

(4-7) Display of Machine Specification Codes



No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
1	Encoder reversed XB	Encoder reversed ZB	Encoder reversed XA	Encoder reversed ZA	Pulley change	LH50 standard spindle	Gear 4 steps	Gear 2 steps
2	DA output signal reversed	VAC motor	Slight modification spec.	Pulse generator reverse rotation	0.1μm control	Brushless servomotor	ATC magazine position encoder reversed	Encoder reversed C
3			M-tool spindle Okuma VAC motor	VAC winding changeover motor for 1st spindle	High-precision servomotor	M-tool spindle DA signal reversed	M-tool spindle gear 2 steps	Slideway lubrication motor soft
4						VAC winding changeover motor for 2nd spindle	2nd spindle gear 4 steps	2nd spindle gear 2 steps
5								
6								
7								
8								
9								
10								
11								
12								

(4-8) Display of Alarm Message List



Up to 12 alarms which have taken place previously are displayed. However, the Alarm P and CPU alarm are not displayed. The larger the number is, the more previously the alarm took place. This list is not cleared even when the power is turned off. In order to clear this display, designate the following command.

= ALMC←

(Key in [A][L][M][C] and press **WRITE** key.)

Reverse display of signal names at the I/O CHECK DATA display screen:

Signal names corresponding to those for which bit data is "1" are displayed in reverse to allow easy checkup of the signal status. Note that the reverse display is available only for OSP5020L. With OSP500L-G, display illuminance is changed between bit 1 and bit 0. Full illuminance for bit 1 and half illuminance for bit 0.

1-8. Tool Layout Function

When changing the setup, it is often required to know which tools are used in the program. After changing the cutting tools, it is also necessary to check if the tools set in the turret(s) match the tools actually designated in the program.

The tool layout function displays all tools designated in a program.

(1) Display Data

The tool layout function displays the following data:

a) Sequence number which contains the tool command.

b) Designated tool number, tool offset number, and the nose R compensation number.

c) The tool offset data called by the tool offset number displayed in b).

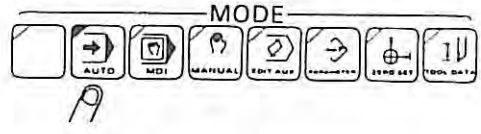
d) The tool nose R compensation number called by the tool nose R compensation number displayed in b).

e) Nose R compensation pattern number.

(2) Operating Procedure

The tool layout display is possible in the following steps.

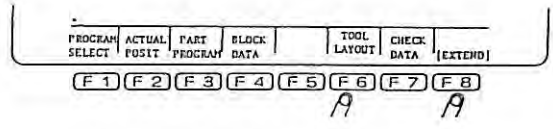
1) Select the AUTO OPERATION mode by pressing the **AUTO** key.



2) Select the program for which the tool layout is to be displayed.

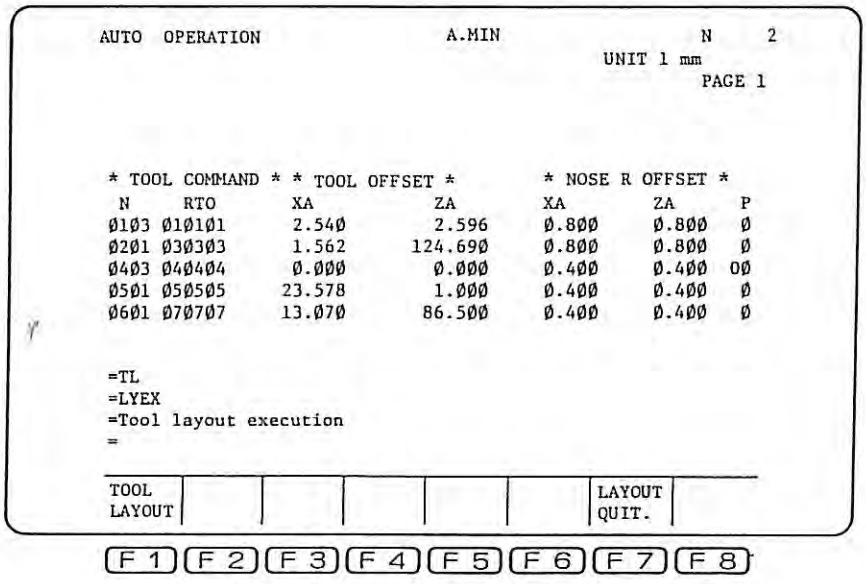
Program selection operation is explained in 1-1. "Program Selection and Operation" of this section. When the required program has already been selected, program selection is not necessary.

3) Press the function key [F6] (TOOL LAYOUT).



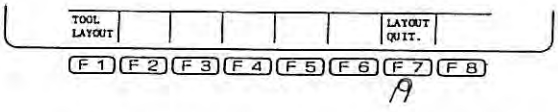
If the function name TOOL LAYOUT is not assigned to the function key [F6], press the function key[F8] (EXTEND). By pressing the function key [F6] (TOOL LAYOUT), the screen changes to the tool layout screen. However, the tool layout data is not displayed even when the function key [F6] (TOOL LAYOUT) is pressed unless the layout has been executed after turning on the power to the control.

4) Press the function key [F1] (LAYOUT EXEC.).



The selected program is read and the tool layout data is displayed on the screen. While the program is being read, the message "tool layout execution" is displayed on the screen.

5) Press the function key [F7] (LAYOUT QUIT). This restores the screen having been displayed before starting the tool layout.



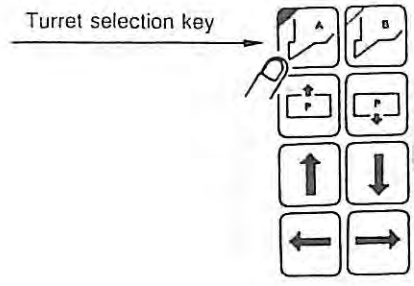
Once the tool layout display is given, the same data is displayed by simply pressing the function key [F6] (TOOL LAYOUT).

The tool layout display data remains active until tool layout is attempted for another program. The tool layout data is cleared when the power is turned off.

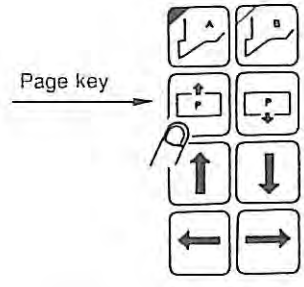
(3) Supplements

a) For two-saddle models, the tool layout display is possible for A and B saddles independently.

Selection of the saddle for which the tool layout data is to be displayed is made using the turret selection keys on the operation panel.



b) One page of the tool layout screen display provides the tool data for 12 tools. Six pages of tool layout screen are provided (for two-saddle models, six pages for each saddle) and pages are advanced and returned by pressing the page keys on the operation panel.



c) For the machine equipped with the tool life management specification (optional), tool commands are given by the tool groups (TG) and the tool offset groups (OG). However, the display of the tool layout data is given using the tool number of the tool actually used in the designated tool group and the tool offset number actually called.

2. Available Operations in Edit Aux Mode

There are three types of program operations:

(1) Transfer . . . . Transfer of main program data is performed.

(2) Editing . . . . . Editing of main program data is performed.

(3) Others . . . . . Editing and other operations of files are performed.

(1) Transfer

"Transfer" refers to storing the main program data into the NC memory using the tape reader, and punching out the stored main program data on a paper tape using the optional tape puncher.

A function to verify the tape data and the stored data is also available.

The standard tape storage capacity is about 12,000 characters (30 meter (98.4 feet) tape length) for OSP5000L-G/OSP500L-G and 24,000 characters (60 meter (196.9 feet) tape length) for OSP5020L; if necessary a larger memory capacity is available as an optional feature.

(2) Edit

"Edit" means altering, inserting, or deleting data stored in the main program. Editing is performed using the keyboard switches while observing the main program data displayed on the CRT.

After editing the main program data stored in the memory, it can be punched out on tape using the transfer function. The use of this function permits the operator to directly store the main program data into the memory using the keyboard switches without preparing a tape.

Note: The bubble memory is used to store the main program.

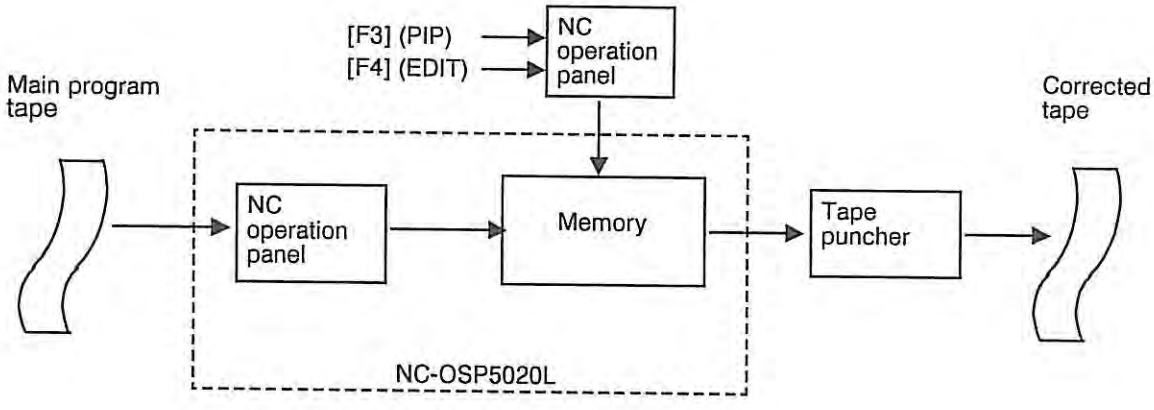


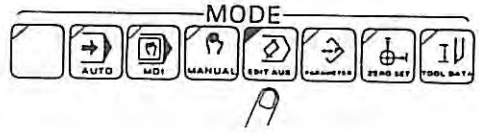
Fig. 4-4 Transfer and Edit of Main Program Date

2-1. Transfer of Main Program Data

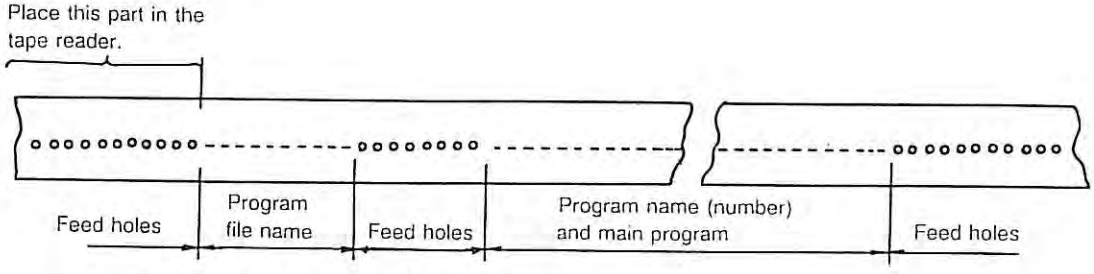
(1) Read-in of Main Program Data

The main program data on the tape can be read into the bubble memory using the tape reader as follows:

1) Select the PROG OPERATION by pressing the **EDIT AUX** key.



2) Place the tape in the tape reader.



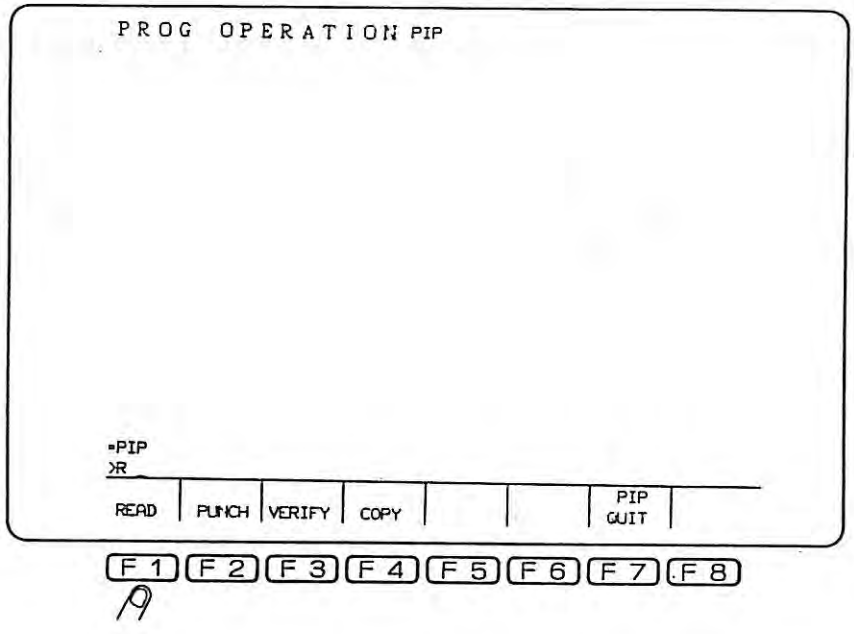
Note: *Place the leading feed holes in the tape reader.*

3) Press the function key [F3] (PIP).

Make sure that the display changes.

4) Press the function key [F1] (READ).

Prompt "R" appears on the console line of the CRT. The control is ready to read the tape.



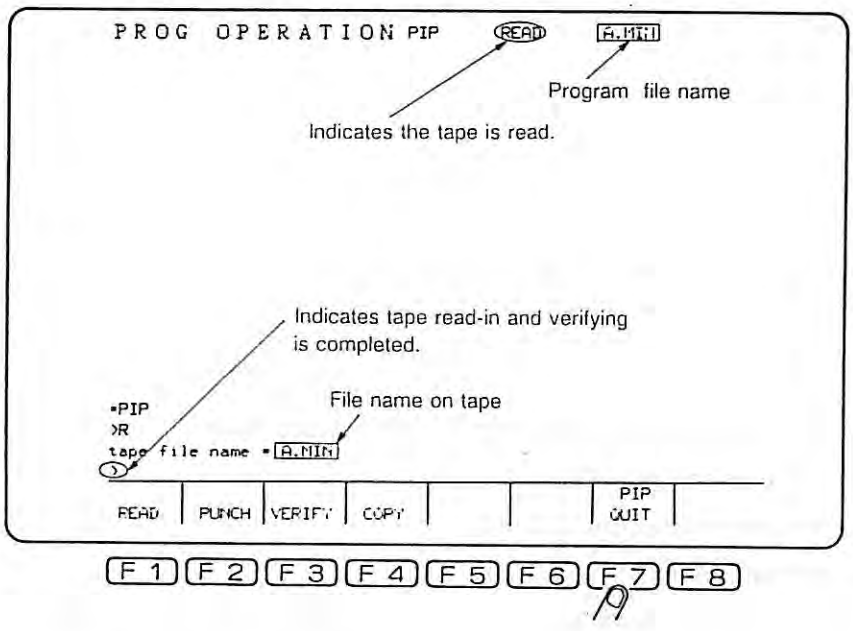
5) Press **WRITE** key.

Tape reader operation will start, and the commands on the tape are read and the data on the tape are stored into the memory.

While the tape is being read, the CRT displays "READ" message along with the "file name" on the first line.

After the completion of the operation above, the tape is fed backward to compare the tape data and the data stored in the memory; this is called "verifying".

When the verification is complete, the prompt " > " appears on the console line on the CRT.



6) Press the function key [F7] (PIP QUIT).

This completes the read-in of the tape and the CRT display returns to the display of step 2).

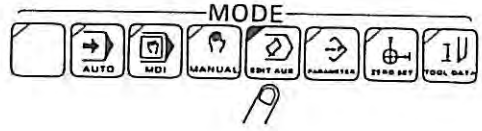
【 SUPPLEMENT 】

It is not necessary to punch feed holes after the program file name. When characters other than A through Z, 0 through 9 or " . " (period) of ISO code are read after "$", the file name is considered to be over and the reading of the program is initiated.

(2) Punching Out the Stored Program (optional)

To punch out the program data after correcting cutting conditions, feedrates, and other commands found improper in trial cut, follow the steps below:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Press the function key [F3] (PIP).

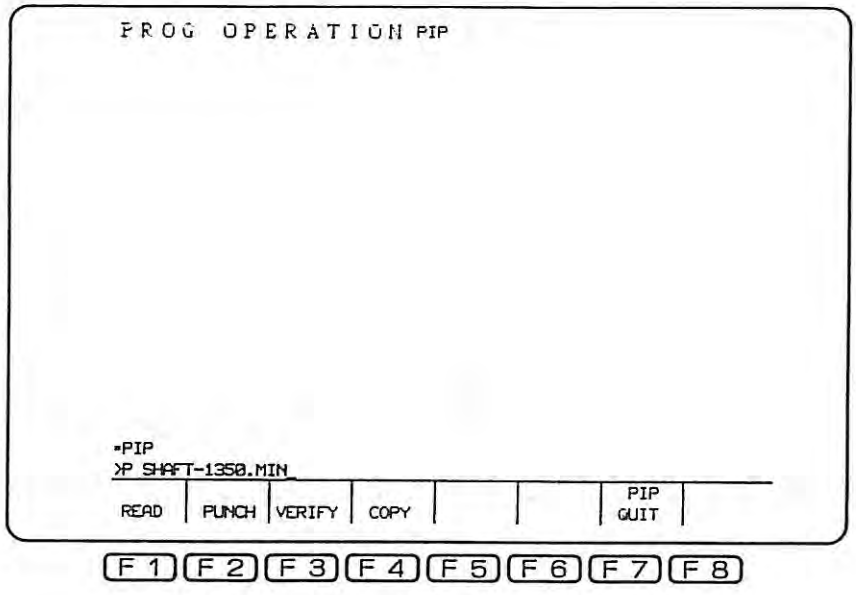
Make sure that the display has changed.

3) Press the function key [F2] (PUNCH).

The prompt "P" appears on the console line of the CRT. The control is ready to punch out the program data.

4) Enter the program file name of the program to be punched out through the keyboard, SHAFT-1350.MIN, for instance.

For the program for which a file name is not designated, i.e., when the file name is "A.MIN", this operation is unnecessary.

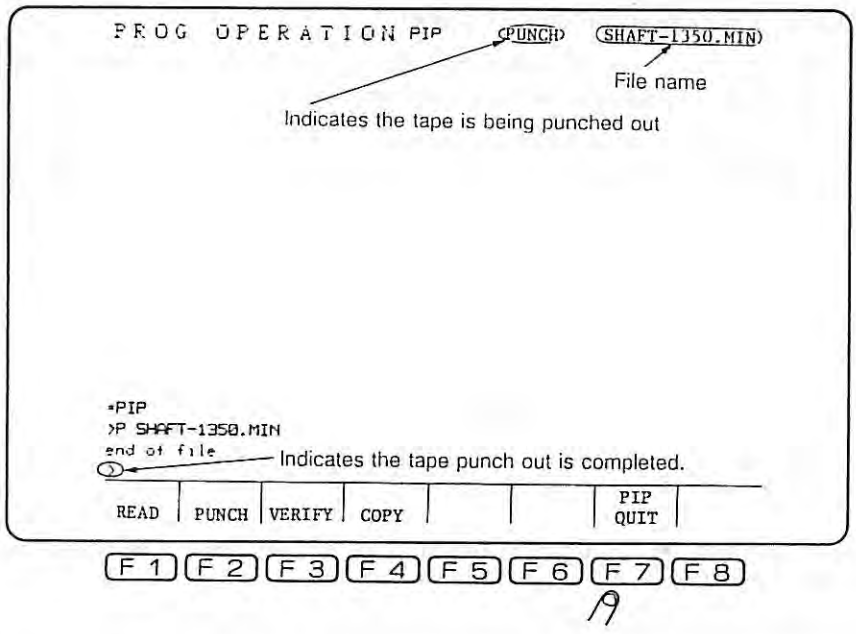


5) Press the **WRITE** key.

This starts tape punch out.

While in tape punch out operation, the CRT displays "PUNCH" message along with the "file name" on the first line.

When tape punch out is complete, "end of file" message appears on the console line and ">" on the next line of the CRT.



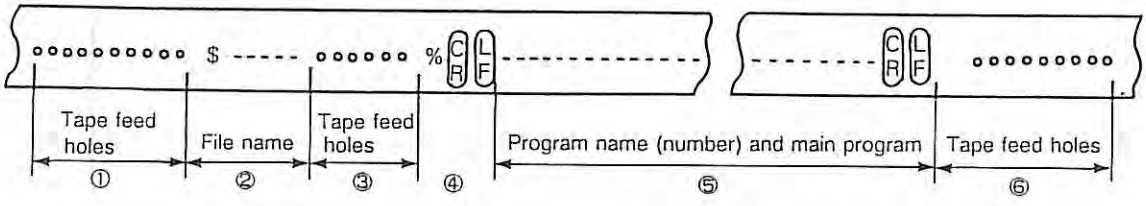
6) Press the function key [F7] (PIP QUIT).

This completes punch out of the stored program and the CRT display returns to the display of step 2).

When the tape punch out is performed during machine operation, the tape punch out speed might be lower than normal.

【 SUPPLEMENT 】

The format of tape punch out is as follows:



① 250 tape feed holes are punched in the leader section of the tape.

The number of punched-out feed holes can be set as desired in the range from 100 to 2000 with a parameter.

② For details, refer to 3., "Operation in Parameter Mode" of this section.

The file name is punched-out following the "$" code. (Program data is punched out in ISO coding system.)

③ 50 tape feed holes are punched.

The number of feed holes cannot be changed.

④ "%", "CR" and "LF" code are punched in succession.

A selection can be made at the parameters to designate "CR and LF" punching, or "LF" punching.

⑤ The main program data is punched out following the program name (number).

⑥ The same number of feed holes as in ① are punched out in the tape trailing section.

⑦ Using a parameter setting, it is possible to set whether the file name is to be punched out or not. When bit 3 of parameter (bit) No. 2 is:

0 . . File name is punched out
1 . . File name is not punched out

Initial setting is 0.

Note 1: *When the EIA coding system is selected for tape punch out, "CR" code is punched out instead of "CR" and "LF" codes.*

Note 2: *When punching out the program data in the EIA code, the presence of a code not available in EIA coding system causes an error; tape punch out stops and an error message is indicated on the CRT.*

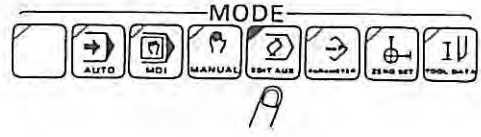
Note 3: *When the tape delimiting codes is " % (ER)"*, the " %" code is punched out preceding the tape feed holes ⑥ .*

* *Bit 3 of optional parameter (bit) No. 1 is "1".*

(3) Verifying the Data Punched Out

To check whether or not the data punched out is all correct, follow the steps below:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Place the tape to be verified in the tape reader.

3) Press the function key [F3] (PIP).

Make sure that the display has changed.

4) Press the function key [F3] (VERIFY).

The prompt "V" appears on the console line of the CRT. The control is ready to verify the program data punched out.

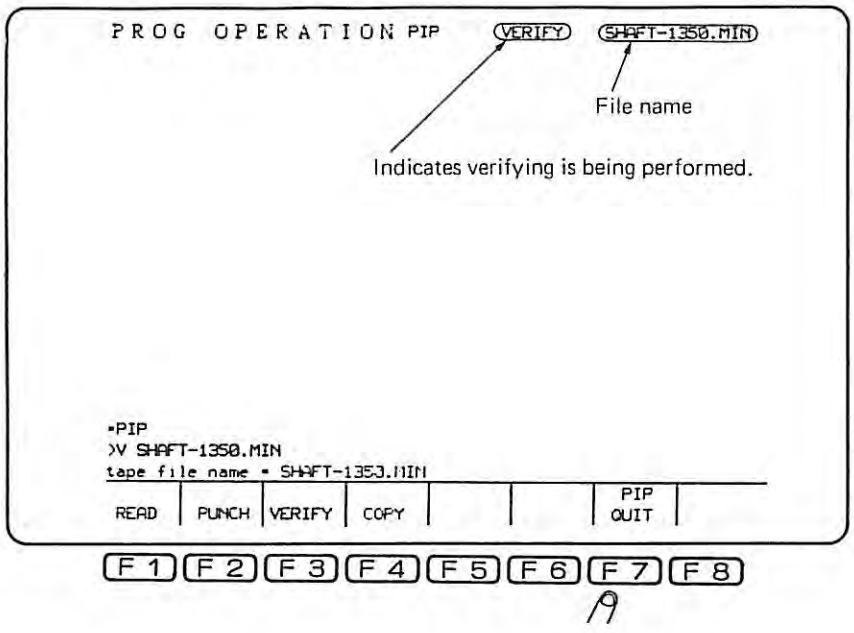
5) Enter the program file name of the program to be verified through the keyboard, SHAFT-1350.MIN, for instance.

For the program for which a file name is designated, i.e., when the file name is "A.MIN", this operation is unnecessary.

6) Press the **WRITE** key.

This starts the tape reader, and program data on the tape is read and compared with the stored program data.

While in program data verifying operation, the CRT displays "VERIFY" message along with the "filename" on the first line.



7) Press the function key [F7] (PIP QUIT).

This completes verification of the punched-out program data and the CRT display returns to the display of step 2).

Note: *When data mismatch is found in tape verifying operation, the block (line) containing inconsistent data is shown in the console line of the CRT, and the address of that data blinks.*

*"**verify continue (Y/N)!**" message appears, asking the operator if he wants to continue verifying or stop.*

When all the data match in verifying operation, the CRT shows the messages as follows:

end of tape
end of file
all same data

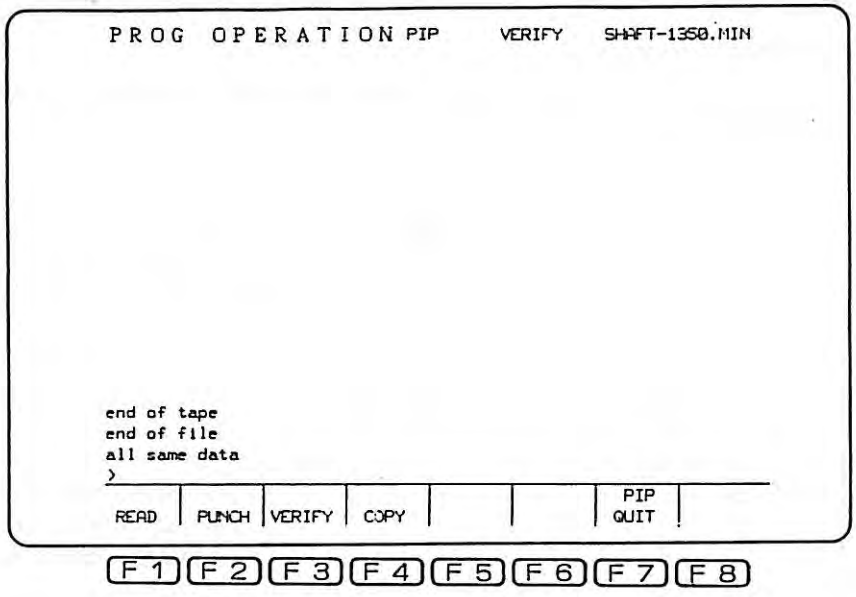
If any file data remains even after all the program data on tape is read, messages are:

end of tape
all same data

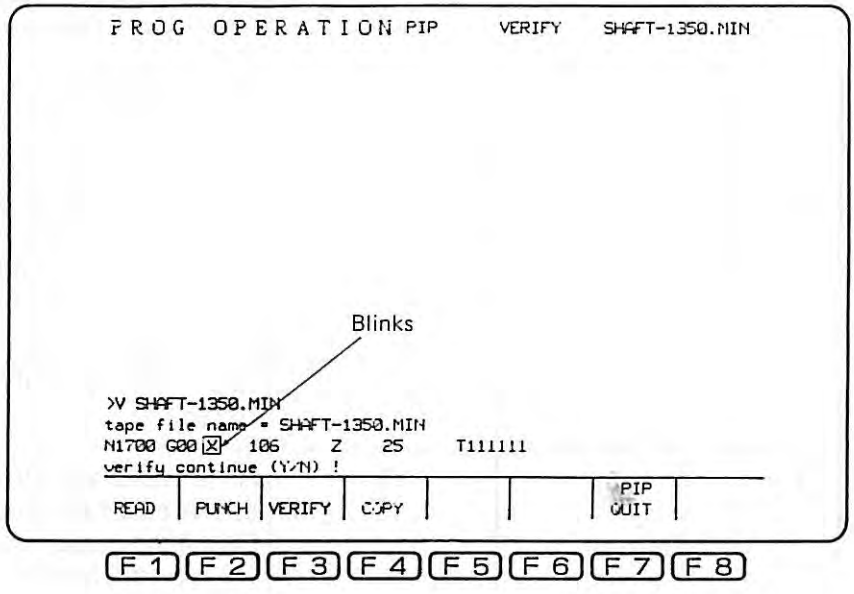
If program data on tape remains even after all the stored file data is read, messages are:

end of file
all same data

When all the data match in verifying operation:



When data mismatch is found:



(4) Precautions on Tape Read-in, Punch-out and Verifying Operations

There are two coding systems: EIA and ISO; selection of the coding system can be made by

a) Parameter setting, or

b) Designating the desired coding system each time, read-in, punch-out or verifying operation is made.

a) Parameter Setting

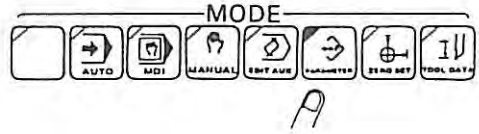
Two parameters are used to determine the coding system: "Tape Code Parity Discrimination" and "Tape Code ISO Code". The coding system in each program operation mode is determined by the combination of these two parameters.

	Tape Code Parity Recognition	Tape Code ISO Code	Operation Condition
(A)	1	1	In the READ and VERIFY modes, the coding system is automatically recognized - ISO or EIA. In the PUNCH mode, program data is punched in the ISO coding system.
(B)	1	0	In the READ and VERIFY modes, the coding system is automatically recognized - ISO or EIA. In the PUNCH mode, the program data is punched in the EIA coding system.
(C)	0	1	In the READ and VERIFY modes, the control assumes the coding system is ISO. (If the actual coding system is not ISO, an error results.) In the PUNCH mode, the program data is punched in the ISO coding system.
(D)	0	0	In the READ and VERIFY modes, the control assumes the coding system is EIA. (If the actual coding system is not EIA, an error results.) In the PUNCH mode, the program data is punched in EIA coding system.

The standard parameter setting is (A), in which the control can read the data coded according to EIA and ISO coding systems. Punch out data is coded in ISO.

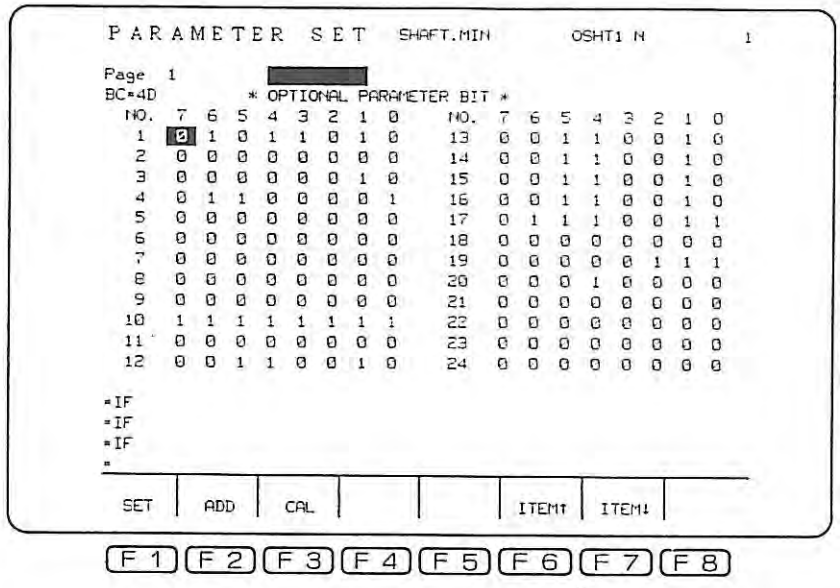
To change the parameter setting from (A) to (B), i.e., to punch out stored program data in the EIA code, follow the steps below:

1) Select the PARAMETER SET mode by pressing the **PARAMETER** key.

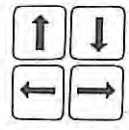


2) Press the function key [F7] (ITEM ↓).

Press this function key repeatedly until the display shown below is obtained.



3) Move the cursor to No. 1 position, using the cursor control keys.



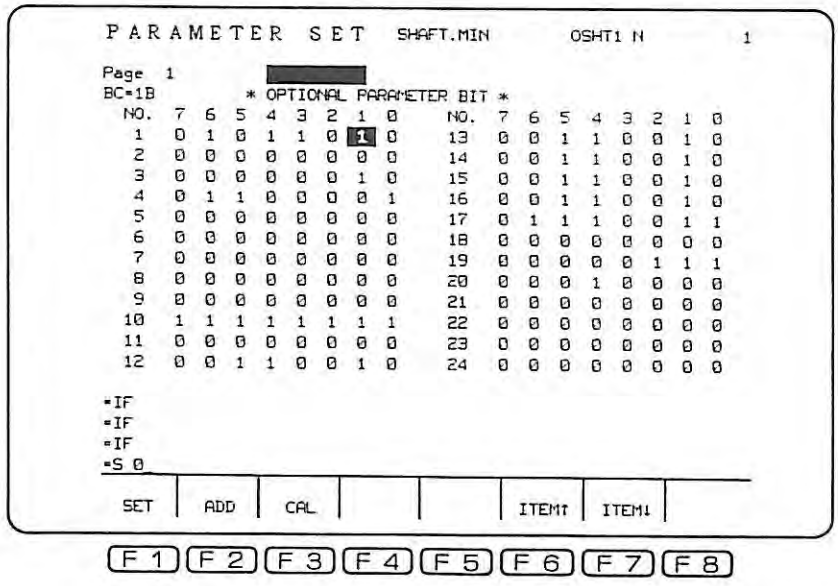
(The cursor is usually placed at that position.)

The two lowest bits of No. 1 parameter correspond to "tape code parity recognition" and "tape code ISO code".

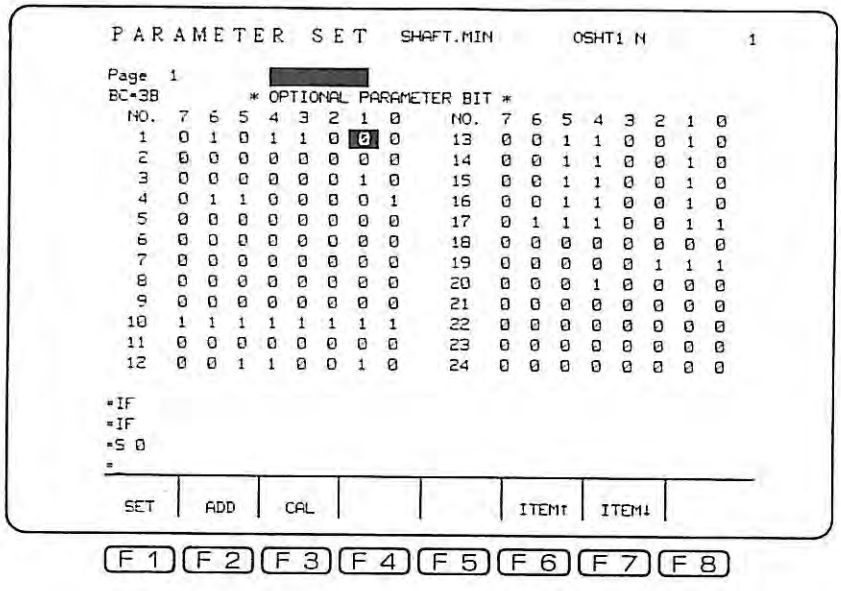
No.	7	6	5	4	3	2	1	0
1	0	1	0	1	0	0	1	1

Tape code ISO code
Tape code parity recognition

4) Press the function key [F1] (SET), and key in the data "0" through the keyboard.



5) Press the **WRITE** key.



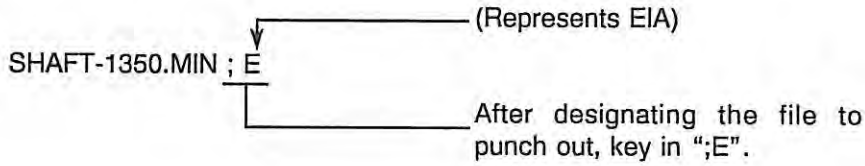
This transfers the data set in step 4) to No. 1 parameter and old parameter data is substituted by the newly entered one.

b) ISO or EIA Designation for Respective Program Operation - READ, VERIFY, and PUNCH

By performing the following steps when executing the required tape operation, READ, VERIFY and/or PUNCH, the operator can directly select the coding system, ISO or EIA, disregarding the coding system selected by parameter setting.

Example: To punch out the stored program data in the EIA code.

Key in the following command in step 4) for punching out stored program data.



Note 1: *Designation of EIA, ISO and Verifying.*

;E . . . . . EIA code
;I . . . . . ISO code
;V . . . . . Verifying

Note 2: *To operate the machine by storing a main program in the memory, there are two cases:*

1) One main program is stored in the memory and machine operation is controlled by it.

2) More than two main programs are stored in the memory, and the necessary program is read out and executed as needed.

These two cases are explained below.

1) Where one main program is stored in the memory:

In this case it is not necessary to assign the main program with a file name. In the memory, however, it is assigned with the file name "A.MIN".

2) Where two or more programs are stored in the memory:

In this case, there are two methods to execute a program:

a) One file for one program

Only one program is registered in one file.

b) One file for several programs

More than two main programs are registered in one file.

In either case, it is advisable to make a program by assigning a tape with a file name. When storing a main program not assigned with a file name on the tape, follow the steps below to assign it with a file name.

i) Press the function keys for READ operation.

ii) Key in, " file name " through the keyboard.

*iii) Press the **WRITE** key.*

Through the steps above, the program data on the tape is stored in the memory and assigned with the keyed-in file name.

For easy control of main program tapes, it is advisable to employ filing system a), "one file for one program".

Note 3: *To store the main program data following the program data in the file already registered in the memory, follow the steps below:*

1) Press the function keys for READ operation.

2) Key in, " file name ;A " through the keyboard.

3) Press the ***WRITE*** *key.*

Note 4: *A file name can be assigned or changed as required following the format below:*

[F1] (READ) "input-file-name", "output-file-name"

When no input file name is designated, the file name on the tape is used as the "input-file-name". If no file name is punched on the tape, the program is assigned with the file name "A.MIN".

When the "input-file-name" is designated, the control checks whether the designated input file name matches the file name on the tape. If they do not match, and error results.

When an "output-file-name" is designated, that file name is created in the memory.

When no "output-file-name" is designated, the "input-file-name" serves as an "output-file-name".

When designation of an output file name is omitted, the delimiter " , " can be omitted, too.

When only an output file name is provided without an input file name, it is necessary to enter the delimiter " , ".

Example 1:

[F1] (READ) SHAFT-1.MIN, SHAFT-2.MIN ***WRITE***

A program assigned with the file name "SHAFT-1.MIN" is stored in the memory, with its file name changed to "SHAFT-2.MIN".

Example 2:

[F1] (READ), SHAFT-2.MIN ***WRITE***

A program is stored in the memory assigned with the file name "SHAFT-2.MAIN" disregarding its current file name.

Example 3:

[F1] (READ) SHAFT-2.MIN ***WRITE***

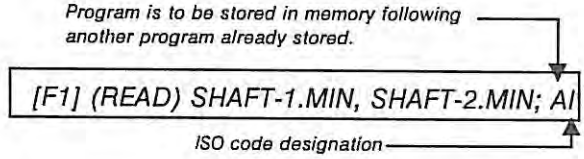
The control first checks whether the file name of the program tape is "SHAFT-2.MIN". After that the program is stored in the memory assigned with the file name "SHAFT-2.MIN".

Example 4:

[F1] (READ) ***WRITE***

The program is stored in the memory assigned with the file name on the tape. If no file name is on the tape, the program is assigned with the file name "A.MIN".

Example 5:



The program data coded in the ISO coding system and assigned with the file name "SHAFT-1.MIN" is stored in the memory and filed with "SHAFT-2.MIN", which is already registered in the memory.

Note 5:

*a) For reading and punching operation of programs in the EIA code, codes not supported by the EIA code cannot be handled. These codes are $, *, =, and [ ]. However, if the EIA code patterns corresponding to them at the optional parameter (bit) data No. 26 through No. 30, they can be used in EIA code mode operation.*

For the read operation, if the first code is the EIA code corresponding to the code "$" (code data matches with the pattern set at optional parameter (bit) No. 30), the file name can be read in the EIA code.

*When the EIA codes corresponding to the codes =, *, [,] and $ are present in the program while the program is being read in the EIA code setting, they can be read as they are.*

For tape punch operation, the file name is punched out in the EIA code if the EIA code(optional parameter (bit) No. 30) corresponding to the code "$" is set.

*When codes =, *, [,] and # are present in a program, they are punched in the corresponding EIA code as set by the parameters.*

b) Both in the EIA and ISO codes, certain codes may be used assuming different codes for program read and punch operations.

Set the code (irregular code) to be regarded as a different code at the optional parameter (bit) No. 31 and the code (correct code) corresponding to the irregular code at the optional parameter (bit) No. 32. Note that the correct code must be set in the ISO code.

If an irregular code appears in the program during reading operation, it is read as the correct code.

If a correct code appears during tape punch operation, the irregular code corresponding to the correct code is punched.

c) Conversion explained in b) is given priority to conversion a).

Note 6: *When designating the read or verify operation, option C code may be added. When the option C code is designated, read or verify operation can be continued even when an error (parity error, EIA code error, special code error, TV check error) is detected during the operation. The detected error code is converted into the " ! " code.*

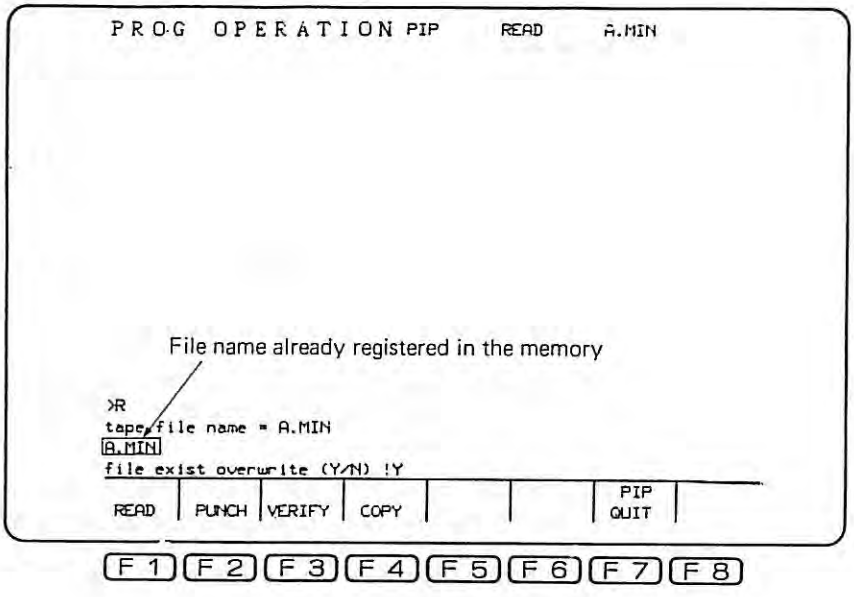
Option C designation format:

[F1] (READ) "input-file-name", "output-file-name" ;C

[F3] (VERIFY) "input-file-name", "output-file-name" ;C

Note 7: *To store a new program data in the file already registered in the memory after deleting the existing data in that file, follow the steps below.*

1) [F1] (READ) file-name **WRITE**
2) The message "**file exist overwrite (Y/N)!**" appears on the console line of the CRT display.



3) After keying in "Y" through the keyboard, press the **WRITE** key.

   This deletes the data in the designated file, and stores the new data through the tape reader.

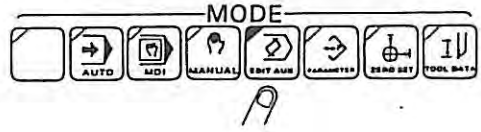
   Remarks: 1. When the operator does not require the new data to be stored, key in "N" through the keyboard and then press the **WRITE** key.

   2. When new data is to be stored under the file name "A.MIN" which is used to file a program not assigned with a file name, and when there is no file name on the tape of the new program data, it is not necessary to designate the file name as in step 1).

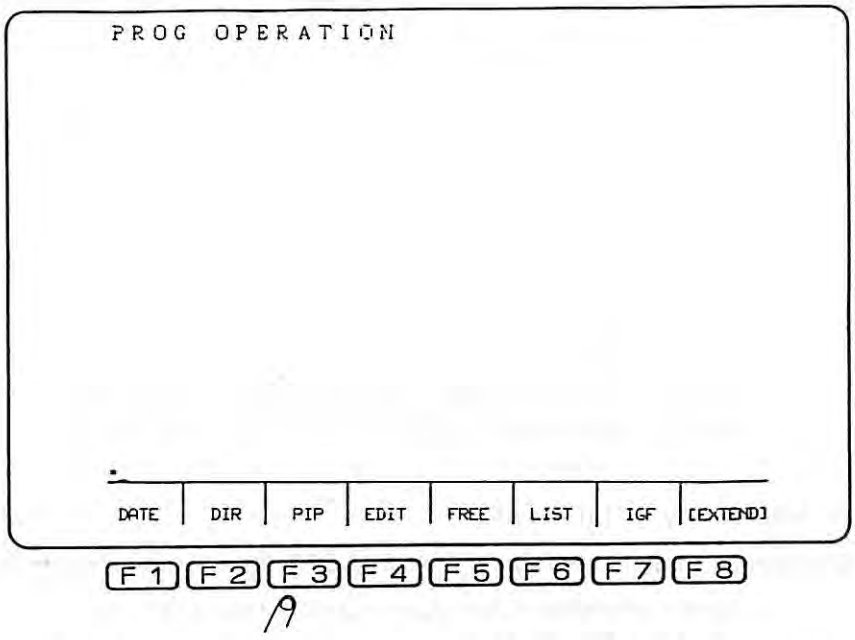
(5) Backup of the Main Program Data

Backup of the file data in the bubble memory is possible. Follow the steps below:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.

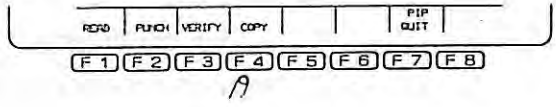


2) The CRT display is as shown below.



3) Press the function key [F3] (PIP).

Make sure that function key display changes.



4) Press the function key [F4] (COPY).

The prompt "CO" appears on the console line of the CRT, telling the operator that the control is ready to back up the program data.

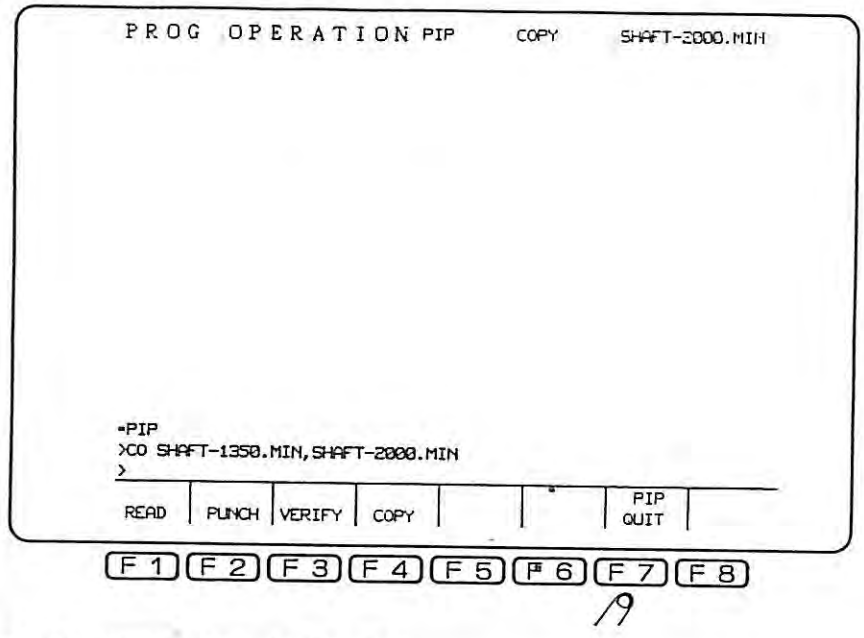
5) Enter the file name of the program to be backed up through the keyboard.

Example: SHAFT-1350.MIN, SHAFT-2000.MIN

↑ Input file name ↑ Output file name

6) Press the **WRITE** key.

With this, the file data "SHAFT-1350.MIN" in the bubble memory is backed up under the file name "SHAFT-2000.MIN".



7) Press the function key [F7] (PIP QUIT).

This completes backup of the program data. The CRT display is restored to 2).

Note 1: *When the designated file name "SHAFT-1350.MIN" does not reside in the bubble memory, the CRT displays:*

file not found

Note 2: *When the newly designated file name "SHAFT-2000.MIN" already resides in the bubble memory, the CRT displays:*

SHAFT-2000.MIN
file exist overwrite (Y/N)!

Enter "Y" by pressing [Y] and ***WRITE*** *keys, to delete the old file data and to duplicate the designated file data.*

Enter "N" by pressing [N] and ***WRITE*** *keys to keep the old file data.*

Note 3: *When the output file name is the same as the input file name, the output file name can be omitted.*

Note 4: *When the output file name is omitted, " * " and "?" can be used in the input file name. In this case, all corresponding files are duplicated.*

For details, refer to the instructions related to Index.

Note 5: *The control has optional features:*

[F4] "input-file-name", "output-file-name";A

File data designated by the "input-file-name" is duplicated following the existent file data of the file designated by the output file name.

[F4] "input-file-name", "output-file-name";V

Before the backup of the file data, message ***"copy OK (Y/N)!"*** *appears on the console line of the CRT.*

Press [Y] and ***WRITE*** *keys, and the intended the backup is performed.*

Press [N] and ***WRITE*** *keys, and backup is not performed.*

Note 6: *The backup function is effective when an optional external memory device such as the bubble memory cassette is selected.*

2-2. Edition of Main Program Data

The operator can edit the program data stored in the memory by observing the display on the CRT. This is generally called "Screen Editor".

When editing the program data, it is read out from the memory in units of files.

Provided below are the explanations of technical terms used in editing operation:

Edit Line:

This line permits the edition of the data. The symbol "≫" is placed at the left of the edit line. One line can contain up to 63 characters.

Edit Pointer:

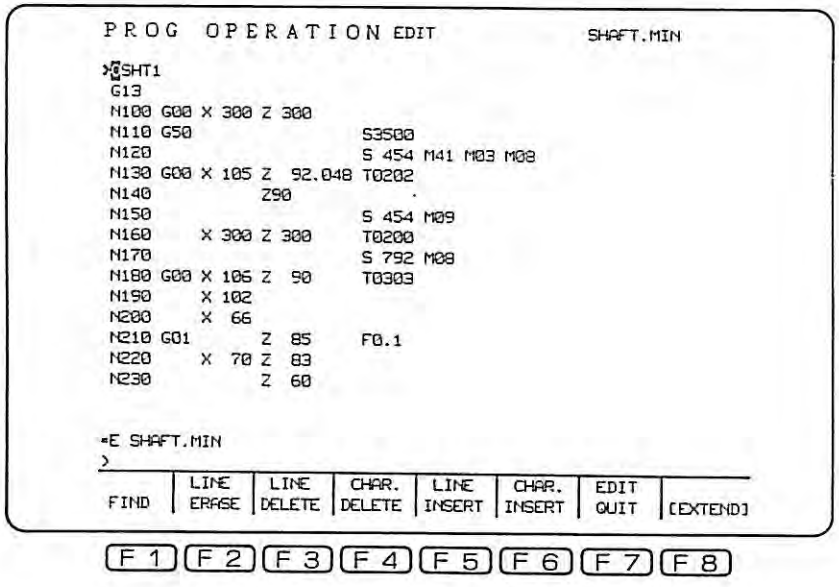
This position indicating the intended start of data edition.

The address character or numeral data is identified by a bright square.

Record:

Called block in a part program.

A unit of data beginning with the character preceded by "LF" and ending with the next "LF". Since the number of characters shown in one line of the CRT is 63, a record is displayed in more than one line if it contains more than 63 characters. In this case, the symbol "&" is displayed at the beginning of the second line.



The detailed procedure for program editing is provided below. **DO NOT FORGET TO PRESS THE FUNCTION KEY [F7] (EDIT QUIT) WHEN EDITING IS COMPLETED.**

If the operator forgets to press that key, the data edited is not stored in the memory.

2-2-1. Fundamental Editing Operation

(1) Read Out Program to Be Edited

There are two methods which can be used to read out a program file from memory for editing:

a) Selection from program edit directory display
b) Direct selection of program to be edited

a) Selection from Program Edit Directory Display

Programs may be selected for editing by accessing the Program Edit Directory on the CRT and using the cursor (reverse display) to indicate the program to be edited.

Procedure:

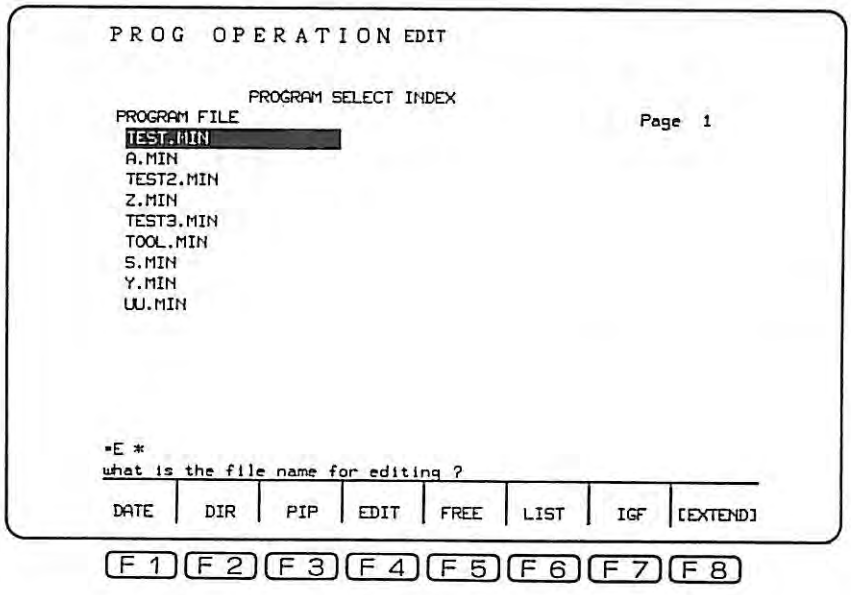
1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.
2) Press the function key [F4] (EDIT).
3) Key in an asterisk (*) through the keyboard.

   The CRT will display:

   =E *

4) Press the **WRITE** key.

   The program directory page will be displayed by the steps above. One page of the display shows a total of 12 file names and if more than 12 file names are registered, press the **PAGE** key to display the second page.



5) Move the cursor to the file name desired.

6) Press the **WRITE** key.

   The cursor-placed program will be searched and read out in the edit area. It will also be displayed on the display screen

Note: *During the operations in steps 4) through 6), keys other than*  ,  ,  *and*  *are not operative.*

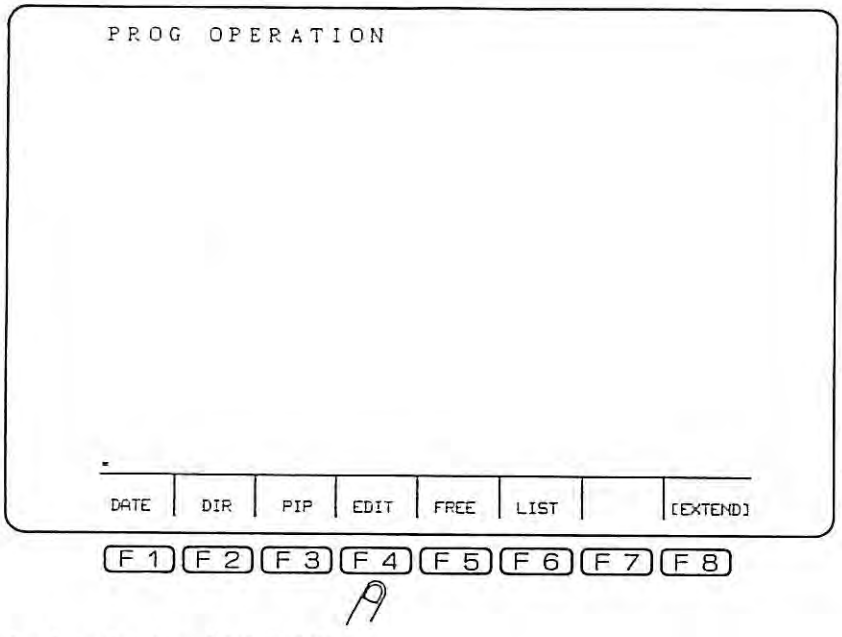
b) Direct Selection of The Program to be Edited

Procedure:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



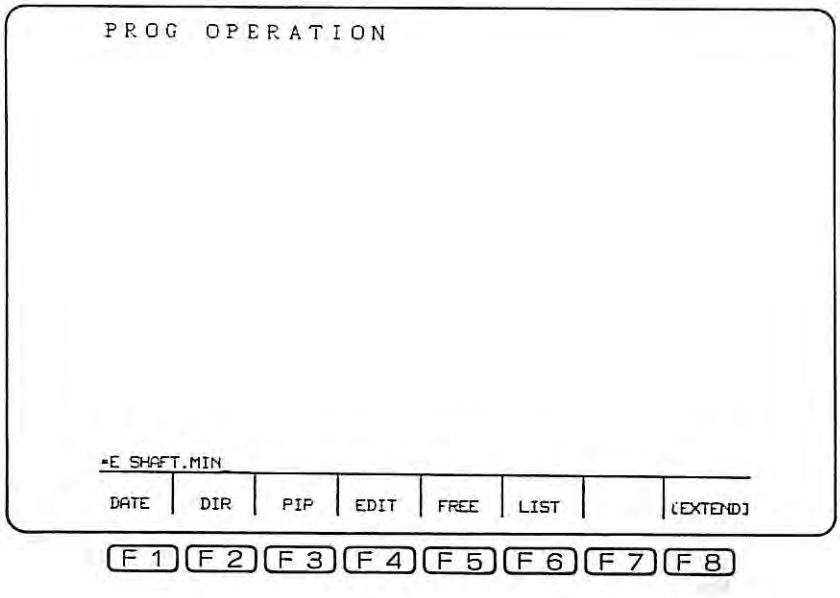
2) The CRT display is as shown below.



3) Press the function key [F4] (EDIT).

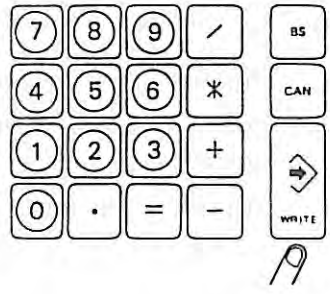
4) Key in the file name of the main program through the keyboard.

Example: SHAFT.MIN

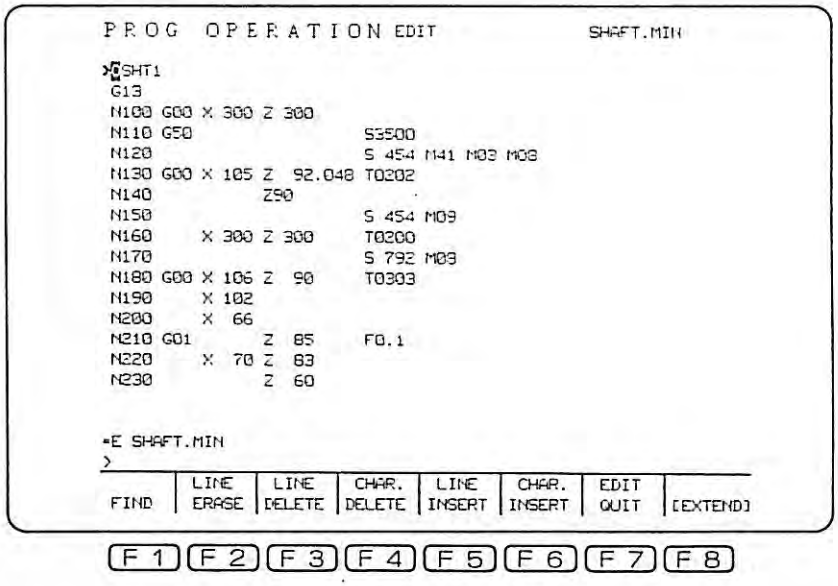


When the file name is not registered, i.e., when the file name is "A.MIN", it is not necessary to key-in the file name.

5) Press the **WRITE** key.



With this, the program data of the designated file is searched for, read and transferred to the editing area. The read out program data is displayed on the CRT at the same time.



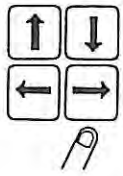
(2) Moving Edit Pointer with Cursor Control Keys

Edit pointer and edit line can be shifted by pressing the cursor control keys:

a) Cursor [→] key

Pressing [→] key shifts the edit pointer rightward. Each time it is pressed, the edit pointer shifts rightward by one character.

When it is kept pressed, the edit pointer shifts continuously.



```
PROG  OPERATION EDIT                    SHAFT.MIN

>OSHT1
 G13
 N100 G00 X 300 Z 300
 N110 G50                    S3500
 N120                        S 454 M41 M03 M08
 N130 G00 X 105 Z  92.048 T0202
 N140           Z90
 N150                        S 454 M09
 N160     X 300 Z 300        T0200
 N170                        S 792 M08
 N180 G00 X 106 Z  90        T0303
 N190     X 102
 N200     X  66
 N210 G01       Z  85        F0.1
 N220     X  70 Z  83
 N230           Z  60

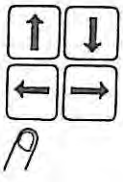

*E SHAFT.MIN
>
| FIND | LINE ERASE | LINE DELETE | CHAR. DELETE | LINE INSERT | CHAR. INSERT | EDIT QUIT | [EXTEND] |
```

[F1] [F2] [F3] [F4] [F5] [F6] [F7] [F8]

b) Cursor [←] key

Pressing [←] key shifts the edit pointer leftward. Each time it is pressed, the edit pointer shifts leftward by one character.

When it is kept pressed, the edit pointer shifts continuously.

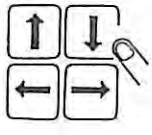


c) Cursor [↓] key

Each time [↓] is pressed, both the edit pointer and the edit line shift downward by one line.

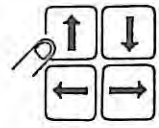
When it is kept pressed, the edit pointer and the "≫" shift continuously.

When it is pressed while the edit line marked by "≫" is at the lowermost line, the edit line returns to the top line.



d) Cursor [↑] key

Each time [↓] is pressed, both the edit pointer and the edit line shift upward by one line.



When it is kept pressed, the edit pointer and the "≫" shift continuously.

When it is pressed while the edit line marked by "≫" is at the top line (at OSHT2 on the CRT page), the edit line returns to the bottom line.

```
PROG  OPERATION EDIT                    SHAFT.MIN

 OSHT1
 G13
 N100 G00 X 300 Z 300
 N110 G50                 S3500
 N120                     S 454 M41 M03 M08
 N130 G00 X 105 Z  92.048 T0202
 N140           Z90
>N150                     S 454 M09
 N160     X 300 Z 300     T0200
 N170                     S 792 M08
 N180 G00 X 106 Z  90     T0303
 N190     X 102
 N200     X  66
 N210 G01       Z  85     F0.1
 N220     X  70 Z  83
 N230           Z  60


=E SHAFT.MIN
>
 FIND | LINE ERASE | LINE DELETE | CHAR. DELETE | LINE INSERT | CHAR. INSERT | EDIT QUIT | [EXTEND]
```

[F1] [F2] [F3] [F4] [F5] [F6] [F7] [F8]

【 SUPPLEMENTS 】

While shifting the edit pointer by pressing [→] or [←] key, the edit line shifts downward or upward when the key is still pressed and the edit pointer is placed at the right or left end of the line.

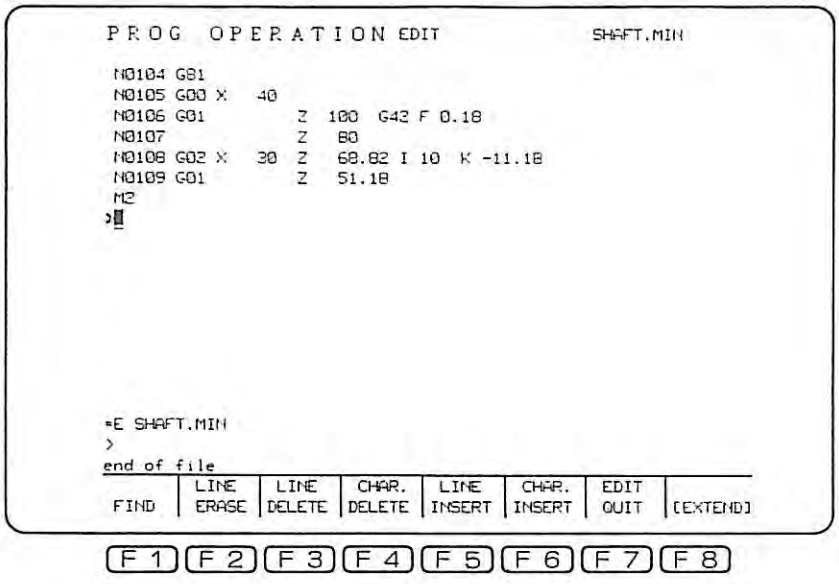
The edit pointer is always on the edit line. In other words, the edit pointer and the edit line indicating symbol "≫" move in pair.

Therefore, when the edit line moves up or down, the edit pointer also shifts up or down along with "≫" symbol.

(3) Page Feeding

Pressing the [↓] key on the NC operation panel advances the CRT display to the next page.

The CRT can display program data of only 16 lines at a time. Therefore, a long program of many lines cannot be displayed in one CRT display. The page key is used to turn the CRT display page.

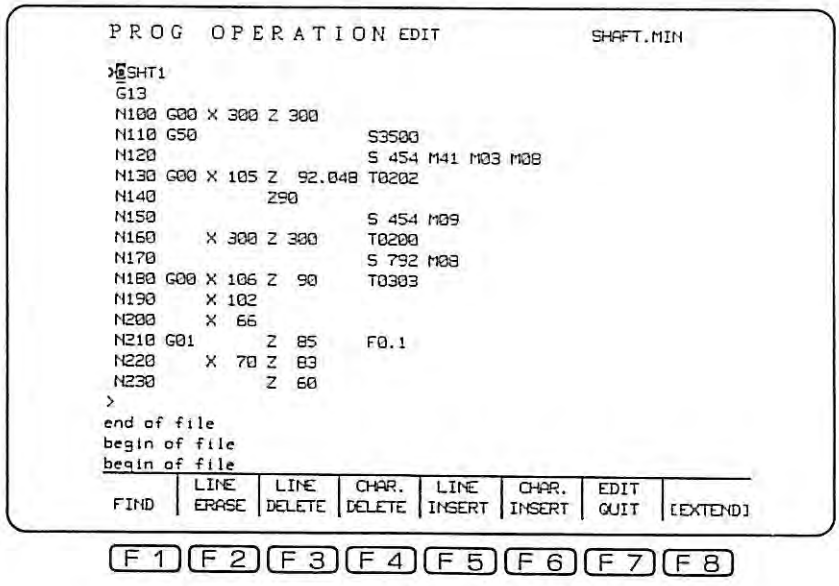


The position of the edit pointer and the edit line does not change.

When the final page of the data is displayed on the CRT, message **"end of file"** appears on the CRT at the console line indicating there is no further data.

(4) Page Reversing

Pressing the [↑] key on the NC operation panel returns the CRT display by one page.



The position of the edit pointer and the edit line do not change.

When the first page is displayed on the CRT with [↑] kept pressed, the message **"begin of file"** appears on the CRT at the console line and pressing [↑] key cannot renew the page any more.

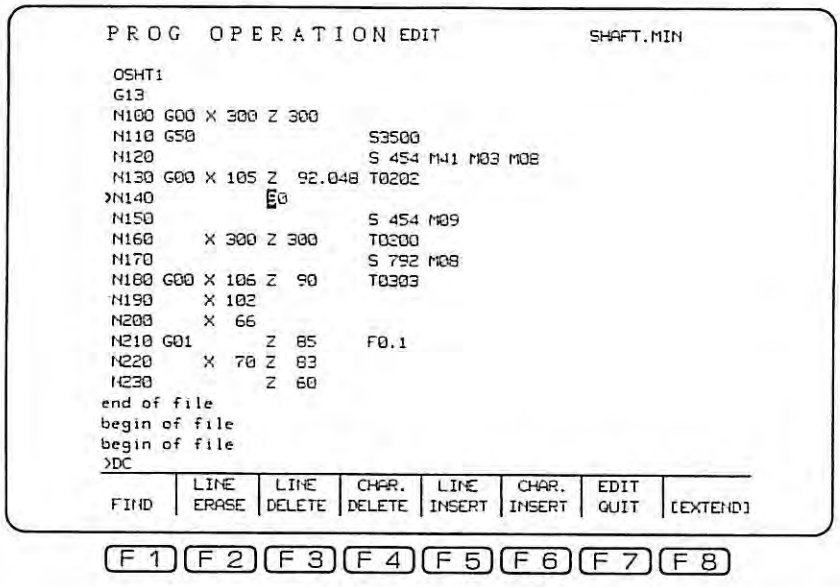
(5) Deleting Character [F4] (CHAR. DELETE)

Pressing the function key [F4] (CHAR. DELETE) deletes the character indicated by the edit pointer.

The character is deleted from the CRT and all the characters on the edit line after the edit pointer move left by one space.

The position of the edit pointer does not change.

The prompt "DC" appear on the console line of the CRT.



(6) Inserting Character [F6] (CHAR. INSERT)

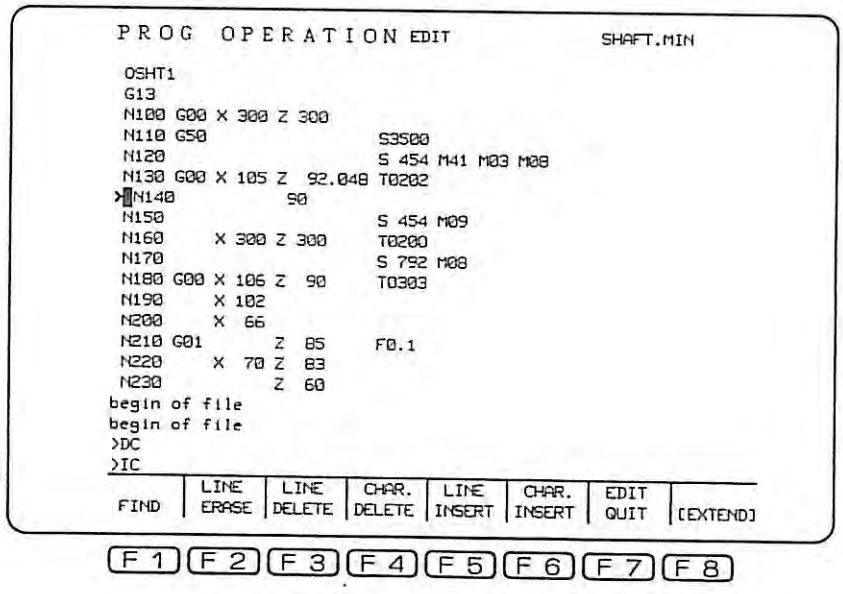
Pressing the function key [F6] (CHAR. INSERT) inserts a space right before the edit pointer.

All the characters on the edit line after the edit pointer move right by one space.

If there is a character requiring other than a space at the end of the edit line, the message **"insert character impossible"** appears on the CRT and the insertion operation cannot be performed.

This operation is effective when inserting characters (numerical values).

When the insertion is made, the prompt "IC" appear on the console line of the CRT.



(7) Deleting Line [F3] (LINE DELETE)

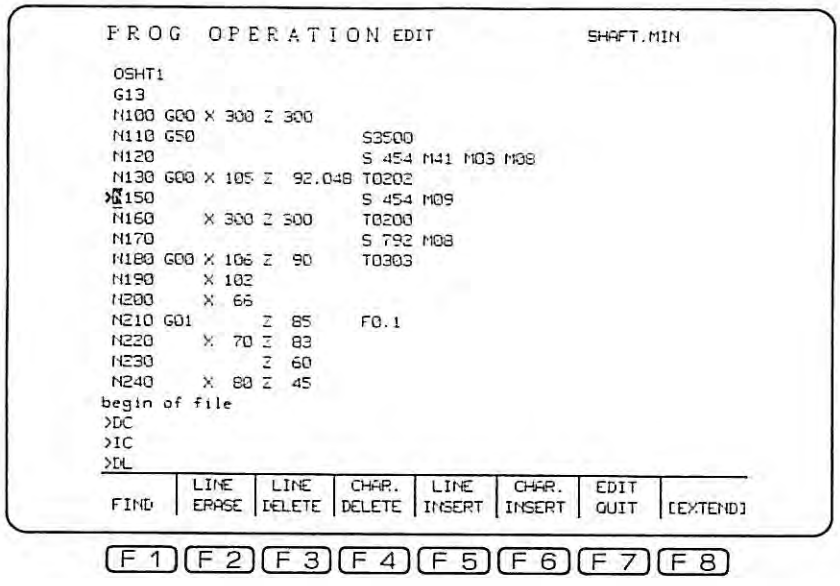
Pressing the function key [F3] (LINE DELETE) deletes one line of the data identified by "≫".

The lines after that shift up to fill the deleted line, and the first line of the next page is displayed at the bottom line.

The edit pointer (reverse display) moves to the first character of the next line.

The prompt "DL" appear on the console line of the CRT.

This operation is effective when deleting all the data in one line.



(8) Inserting Line [F5] (LINE INSERT)

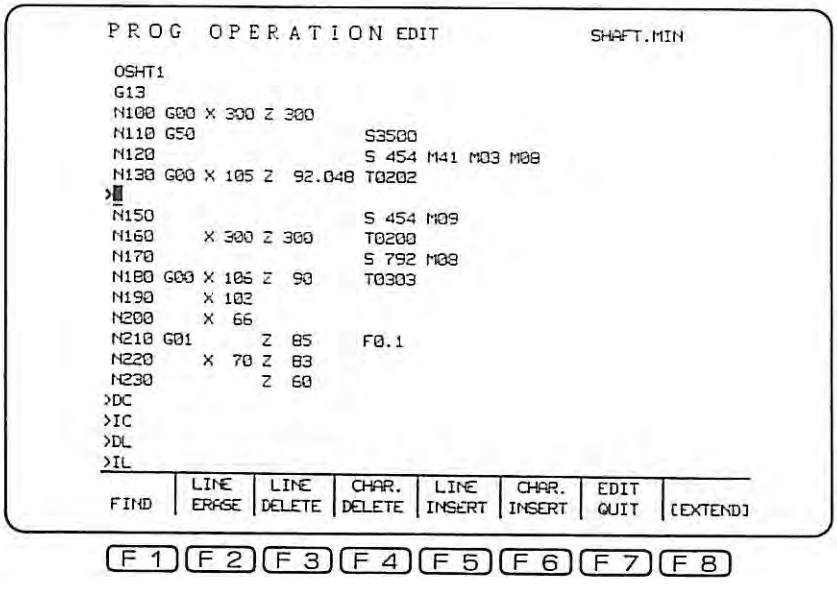
Pressing the function key [F5] (LINE INSERT) inserts a blank line right before the edit line identified by "≫":.

Lines after that all shift downward and the line at the bottom of the CRT is transferred to the next page, thus disappearing from the display.

The edit pointer moves to the beginning of the blank line.

The prompt "IL" appear on the CRT.

This operation is effective when inserting a line.



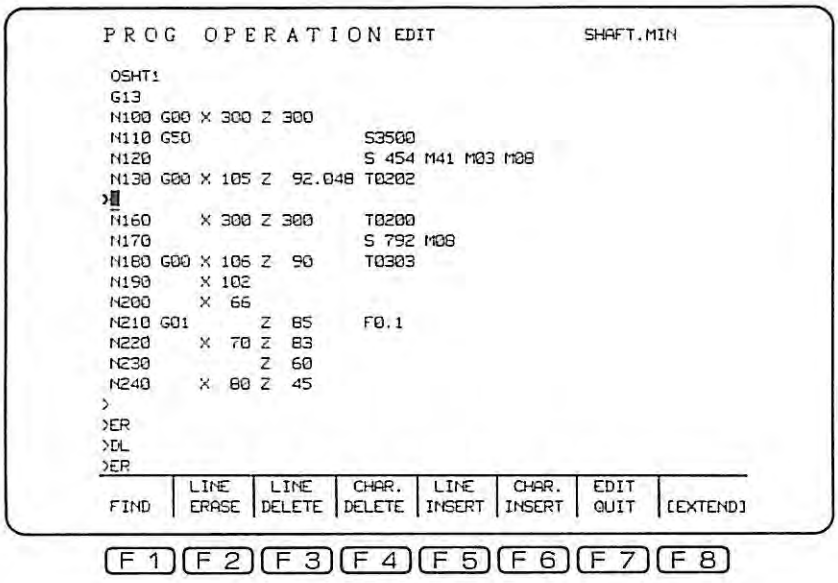
(9) Erasing Line [F2] (LINE ERASE)

Pressing the function key [F2] (LINE ERASE) erases one line of the data, leaving a blank line.

The edit pointer is placed at the beginning of that line.

The prompt "ER" appear on the console line of the CRT.

This operation is effective when erasing all the data in one line and adding another data in the same line.



2-2-2. Application of Fundamental Editing Operation

Provided in this paragraph are the explanations for actual editing operation, in which program data in Program A is edited in to Program B.

Program A

```
OGR1
NG13   G13
N001   G00   X800   Z800    F.1     M04   M42   S250   T010101
N002   G42   X200   Z100
N003   G01          Z75
N004         X225   Z50
N005   G03   X250   Z20     L20
N006   G02   X255   Z17.5   L2.5
N007   G01          Z10
N008         X275
N009   G00   X800   Z800
N010                                M02
```

Program B

```
OGR1
NG13   G13
N001   G00   X800   Z800    F.15(a) M04   M42   S250   T010101
N002   G42   X200   Z100
N003   G01          Z75
N004         X225   Z50
N005   G03   X250   Z20     L20
N006   G01   X260   Z15 (b)
N007   ___(c)       Z10
N008   G40(d) X275
N009   G00   X800   Z800
N010                                M02
```

Correction is made on underlined commands in Program B; (a) through (d).

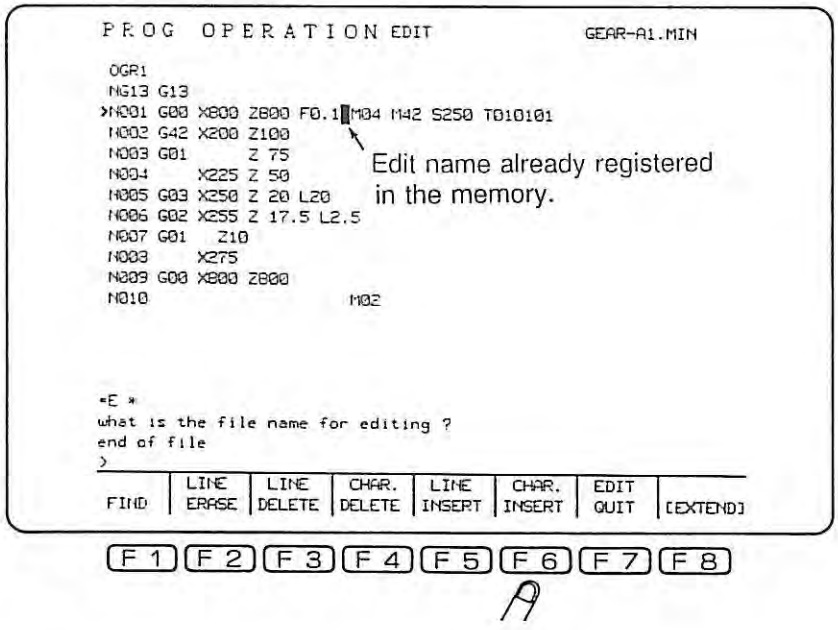
Follow the steps below:

Correction of command (a):

"F.1" on N001 line is to be corrected to "F.15".

Procedure:

1) Move the edit pointer to the position after numeral "1" of "F.1".

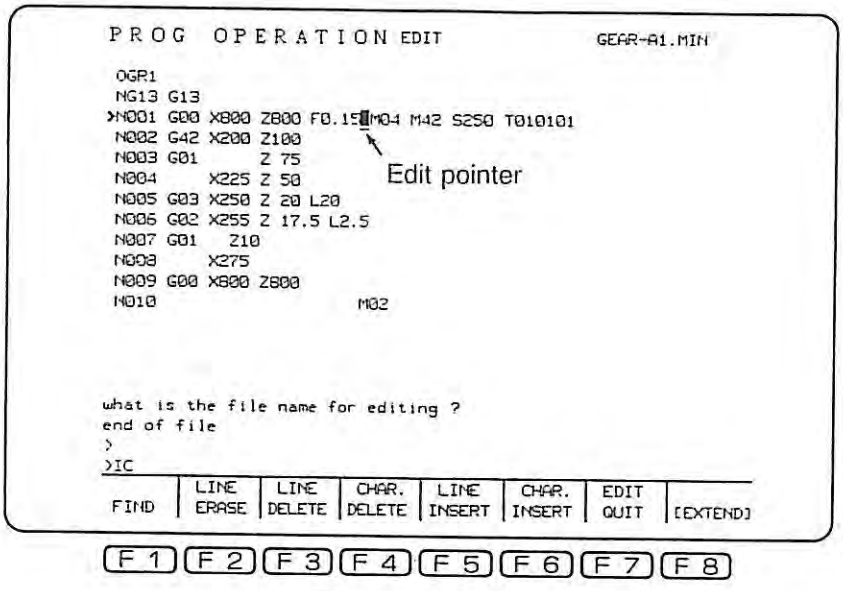


2) Press the function key [F6] (CHAR. INSERT).

This shifts all the commands after the edit pointer rightward by one character, leaving a space for the entry of numeral "5".

3) Press numeral key [5] on the keyboard.

Numeral "5" appears at the position edit pointer is placed.



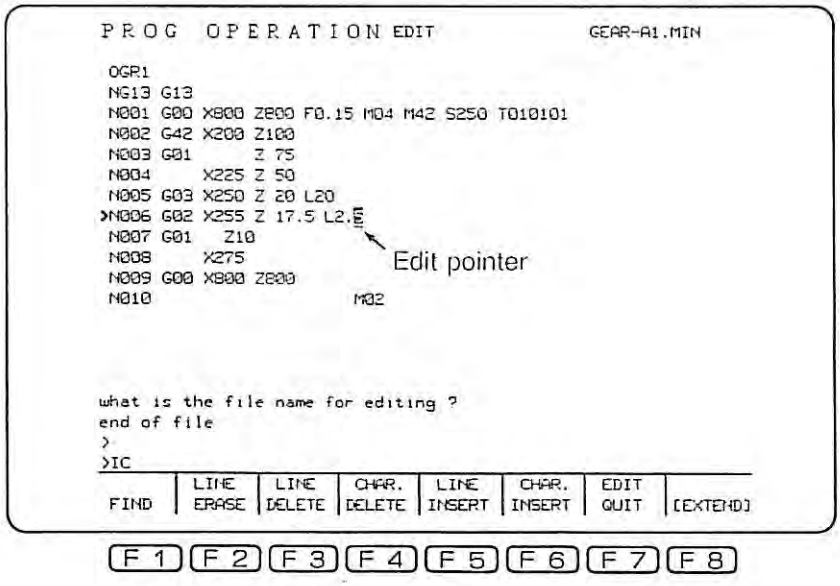
Correction of commands (b):

Commands on the N006 line are to be corrected.

Procedure:

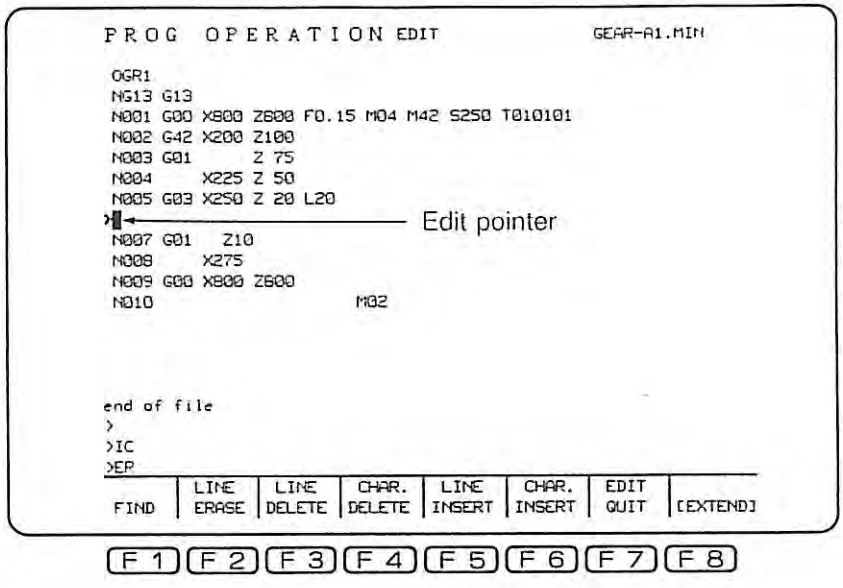
1) Shift the edit pointer to the N006 line.

The edit pointer may be placed at any character on the N006 line.



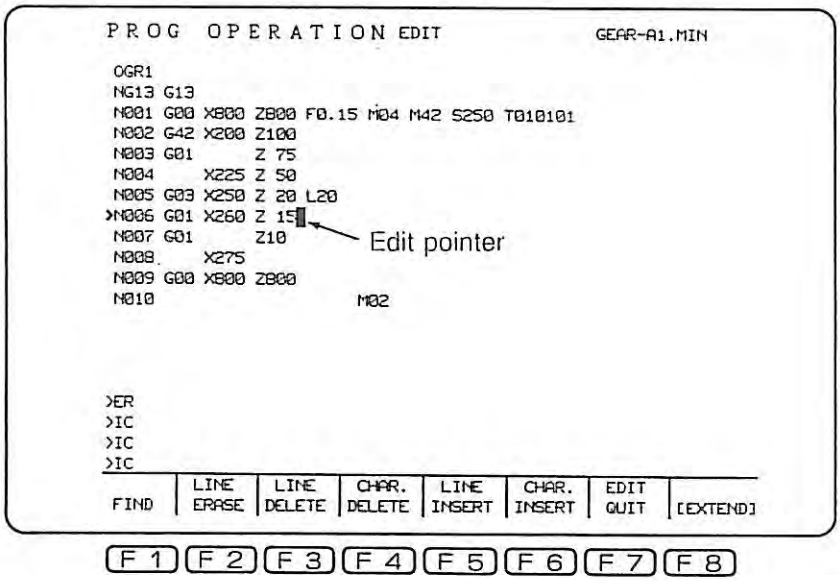
2) Press the function key [F2] (LINE ERASE).

The commands on the N006 line all disappear from the CRT and the edit pointer comes to the first character position of erased N006 line.



3) Key in new data through the keyboard.

The new data is entered in the blank line.

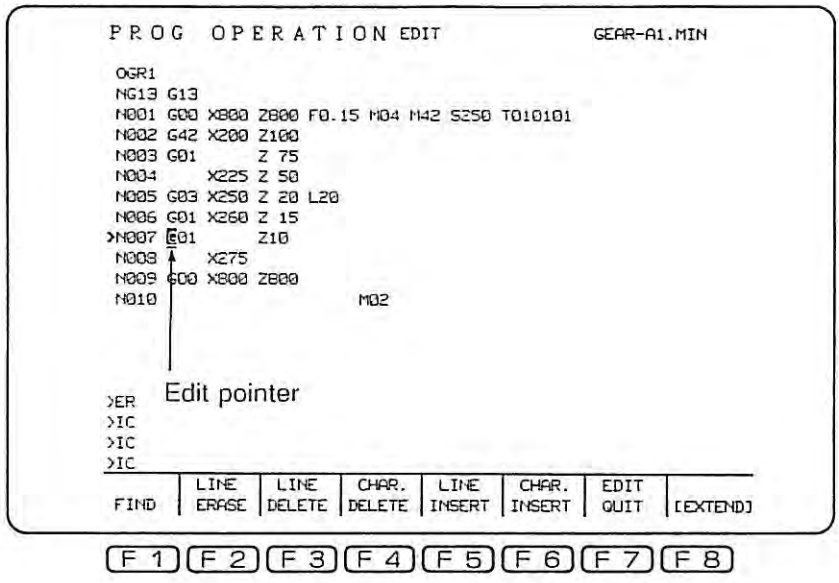


Deletion of command (c):

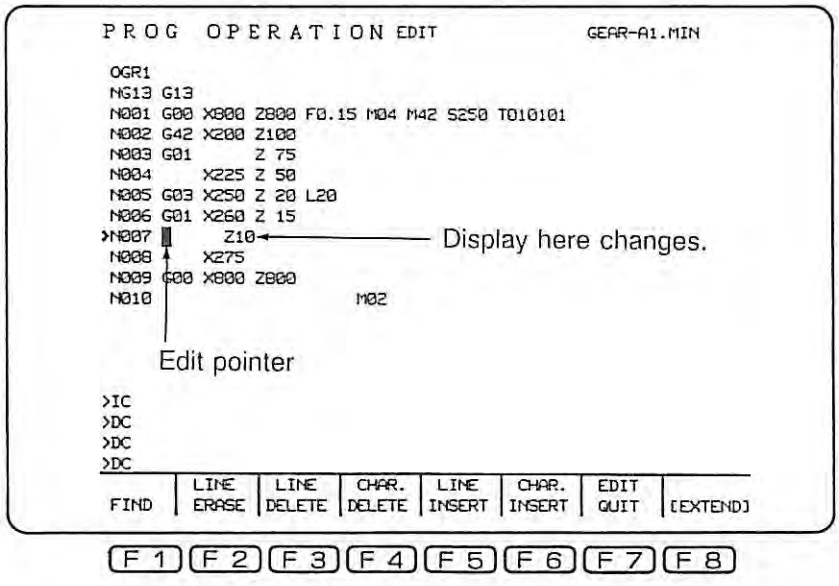
G01 in N007 is to be deleted.

Procedure:

1) Move the edit pointer to G01 in N007 line.



2) Press the function key [F4] (CHAR. DELETE) three times to delete three characters. This deletes "G01".



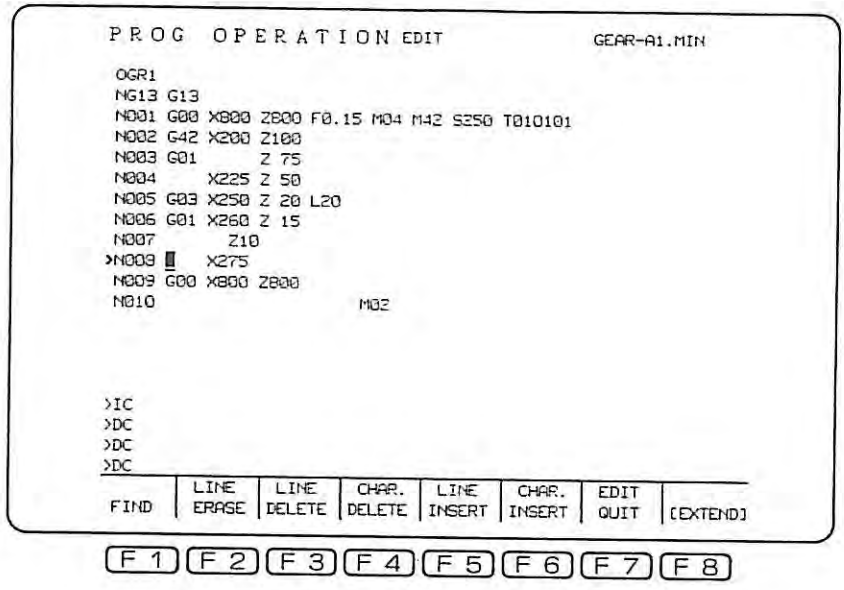
Pressing the **SP** key three times has the same effect; however, when a command is deleted using the **SP** key, commands following the deleted one are not shifted leftward. (Program B is this case.)

Addition of command (d):

G40 is to be added in N008.

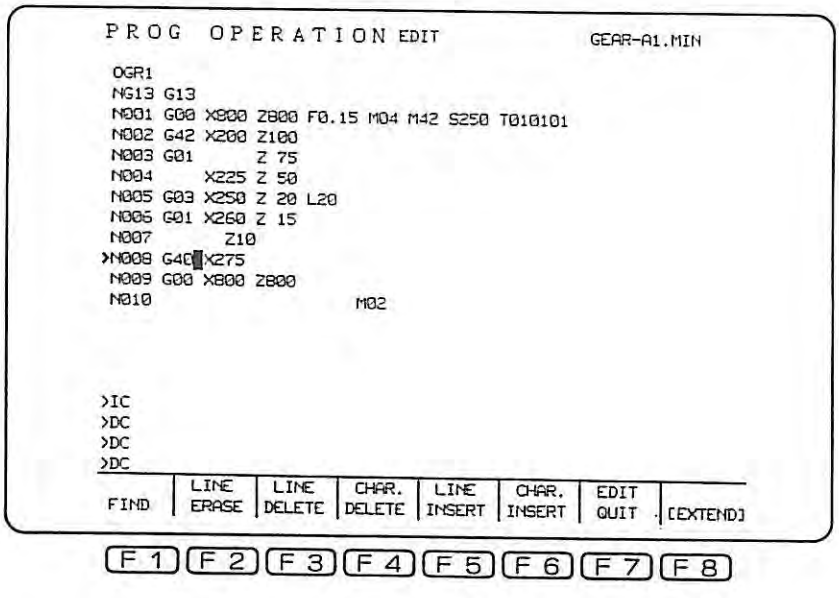
Procedure:

1) Move the edit pointer to the space right after "N008".

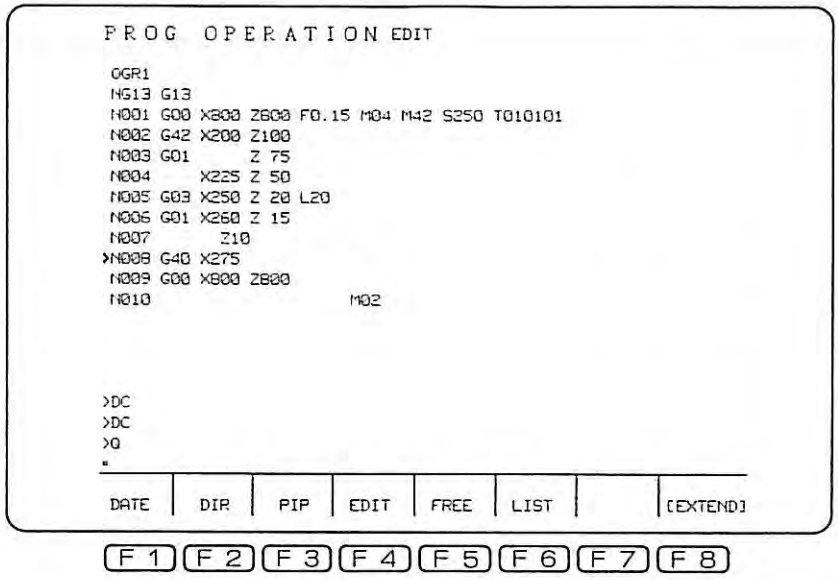


2) Key in "G40" through the keyboard.

"G40" is entered at the position where the edit pointer is placed.



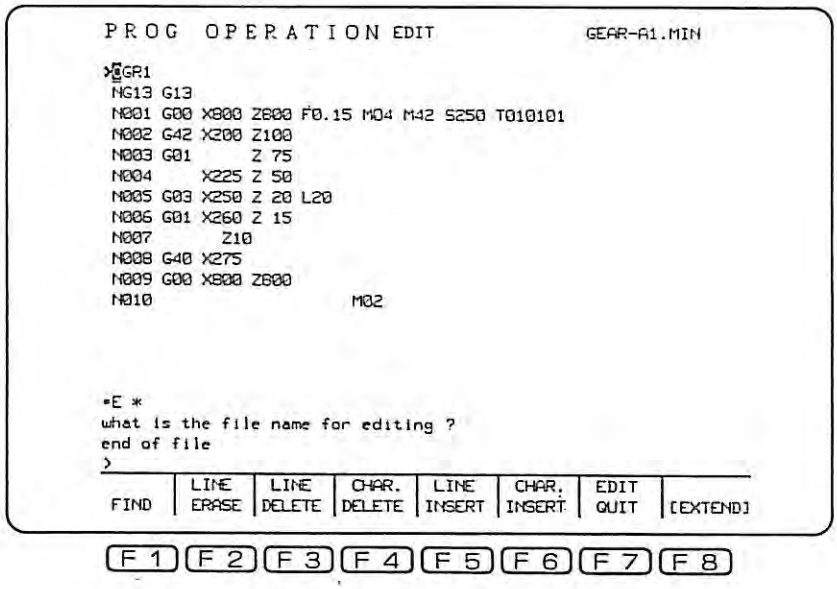
This completes the edition of Program A to Program B. Press the function key [F7] (EDIT QUIT).



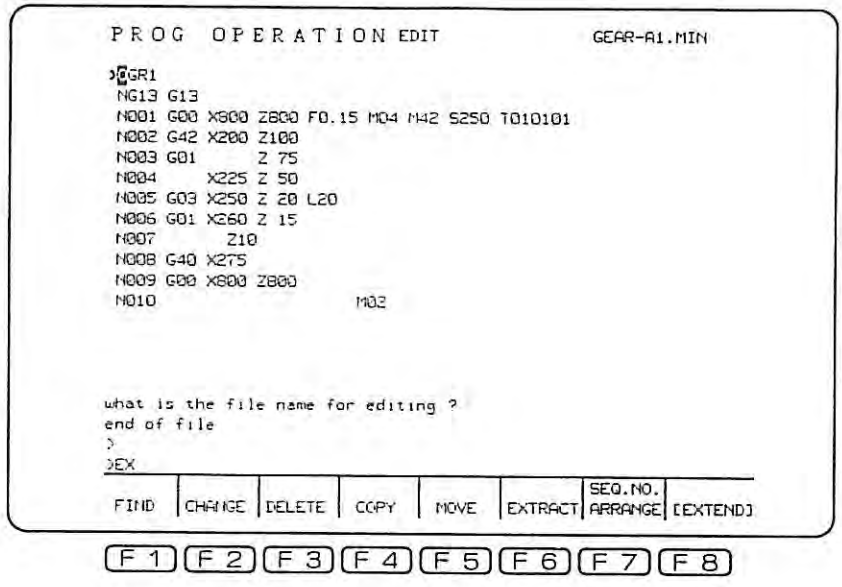
2-2-3. Extended Function of Editing

This section deals with extended functions of editing. Read the instructions in this section only after fundamental editing procedures are well understood.

The explanation below is provided on the assumption that the function key [F8] (EXTEND) has been pressed.



Display of function line changes.



(1) Find [F1]

This function can search for the character-string or the block by specifying the desired character-string or by specifying the desired blocks in terms of the number of lines from the present edit line.

a) Searching for character-string

The character-string specified through the keyboard is searched for from the character after the one identified by the edit pointer.

When the specified character-string is searched for, the edit pointer is placed at its first character.

When specifying a character-string, it is necessary to enclose it by the same character other than the characters appearing in the specified character-string and those specified below:

+, −, ', ;, :, 0 through 9, space

Example: /G02/

"?" in the character-string to be designated is used to indicate one arbitrary character.

Example: /N?01/

With this, the control searches for three-digit N words consisting of "01" at its lower two digits.

Once the required character-string is specified, the control performs the searching operation each time [F1] (FIND) and **WRITE** keys are pressed.

"@" in the character-string to be designated is used to indicate one character except numerical characters and decimal point.

Example: F −/X10@/

This allows the search of "X10" in such as "X10Z10".

Compare with the following commands:

F− /X10/ . . . . . This searches X100 and X10.5 also.
F− /X10−/ . . . . Search of "X10" in "X10Z10" is impossible.

Once the character-string is designated, its search is activated each time [F1] (FIND) and **WRITE** keys are pressed.

When spaces are placed before character-string, these spaces are handled as characters.

Example: To search X−−−30,

/X−−−30/

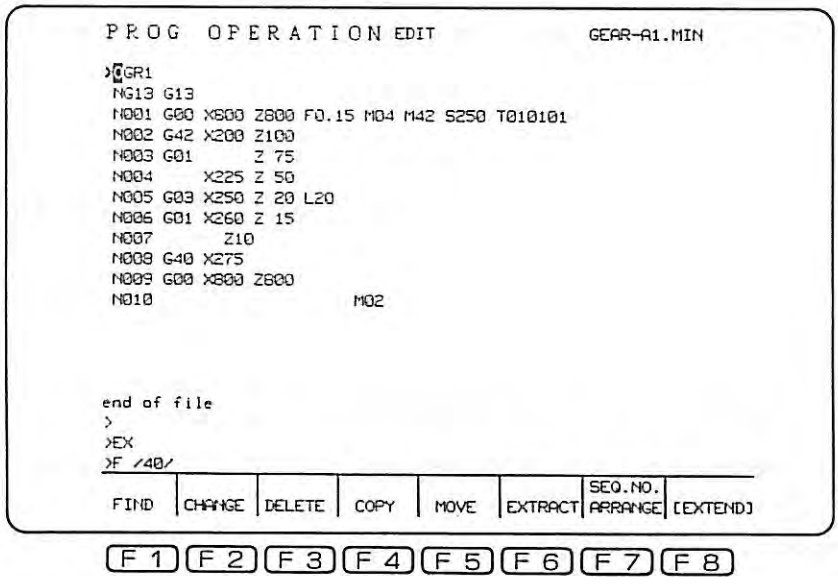
When searching for "40" from the program shown on the previous page:

Key-in "/40/" through the keyboard.

↓

Press the **WRITE** key.

↓



b) Searching for the required block by specifying the number of lines.

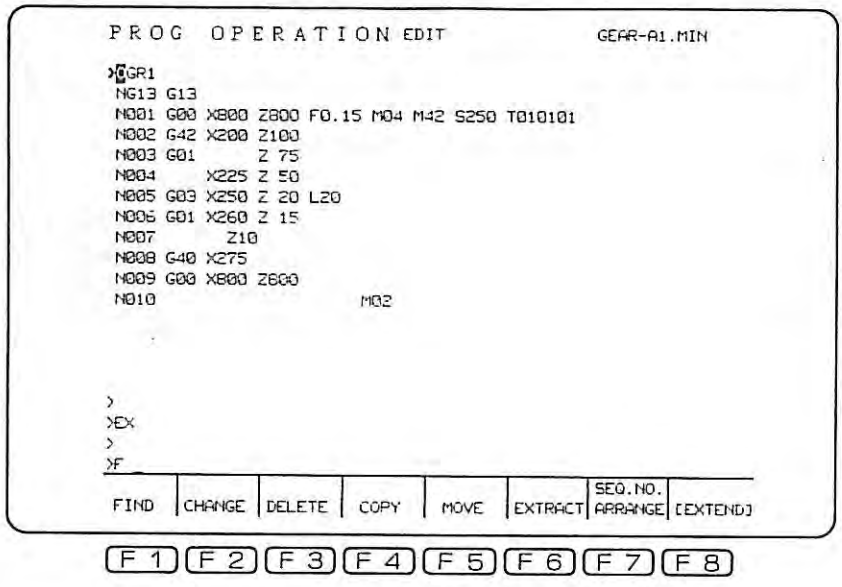
"≫" moves from the present edit line as much as the specified number of lines.

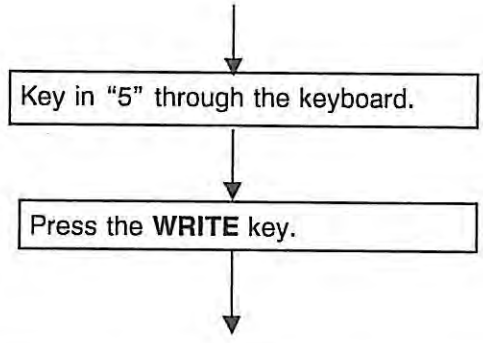
The CRT display changes so that the line searched for is displayed at the first line of the display.

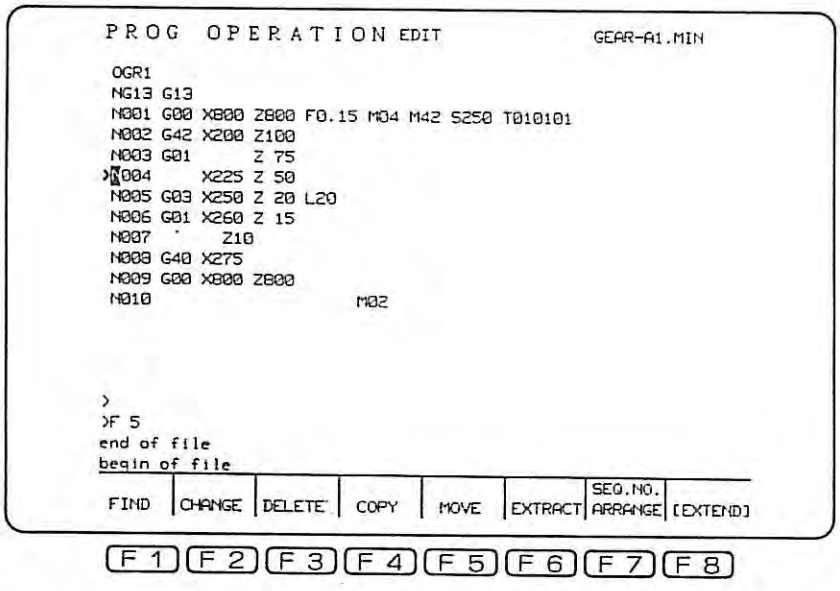
When the specified number is so large as to exceed the final line of the file, the edit pointer is placed at the line right after the final line of the file. In this case, the program of 15 lines from the last line (block) is displayed on the CRT with the last line at the bottom, and the message "**end of file**" appears on the console line of the CRT.

When a negative number is specified, the edit line shifts backward by the specified number of lines, and the edit pointer is placed at the first character of the new edit line. When a negative number to pass the first line of the file is specified, the edit pointer is placed at the first character of the file. The CRT then displays the message "**begin of file**" on the console line.

When specifying the fifth block from the edit line:







This function permits the operator to replace a command (character-string) with another command (character-string) by keying in those through the keyboard.

The edit pointer is placed at the first character of the new character-string.

When the specified character-string cannot be found within the file, the message "**string not found**" appears on the console line of the CRT and the edit pointer does not move.

Enclose the character-string in the same manner as explained in (1) Find.

Character "?" can also be used in the same manner as in "Find" operation.

If several character-strings identical to the specified character-string to be changed are in a program, the change of the character-string may be performed simply by pressing [F2] (CHANGE) and **WRITE** keys.

Note: *;A option specification*

If ";A" option is designated, global search and replace of the character-string is possible.

After keying in the character-strings, key in "A"(ALL) preceded by delimiter ";".

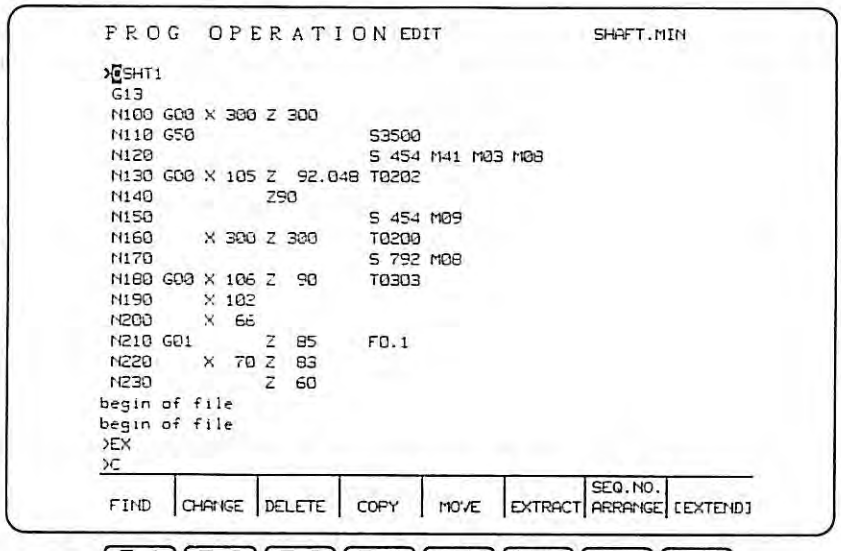
Example: *C /S_454/S_554/;A* ***WRITE***

With the input as indicated above, all the designated character-strings are replaced with the designated one in the area from the word on which the edit pointer is located up to the end of the file.

After the completion of replacement, the total number of character-strings which have been replaced is displayed at the CRT as indicated below.

****** *change***

When changing "S_454" in N120 to "S_554":

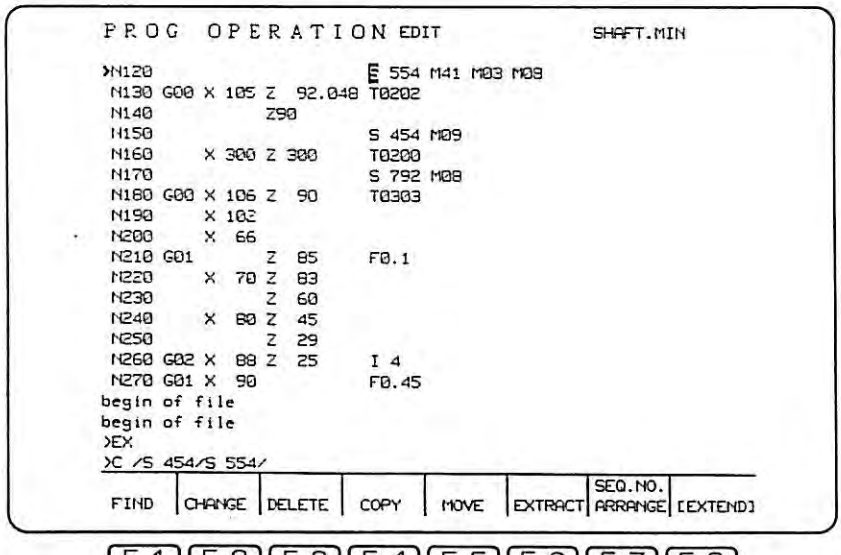


F1 F2 F3 F4 F5 F6 F7 F8

/9

Key in "/S_454/S_554/" through the keyboard.

Press the **WRITE** key.



F1 F2 F3 F4 F5 F6 F7 F8

The edit pointer is placed at the first character of the line right after the deleted lines.

If the specified number of lines to be deleted is so large as to exceed the final line of that file, all the data up to the final line is deleted. The symbol "≫" indicates the edit line is located in the line right after the final line of the file. The message "**end of file**" appears on the console line of the CRT.

After the deletion, the number of deleted lines is indicated on the console line of the CRT as "** **record deleted**".

When the lines are deleted, those following them are shifted upward to fill the deleted blank lines.

[F3] (DELETE) – 8 **WRITE**

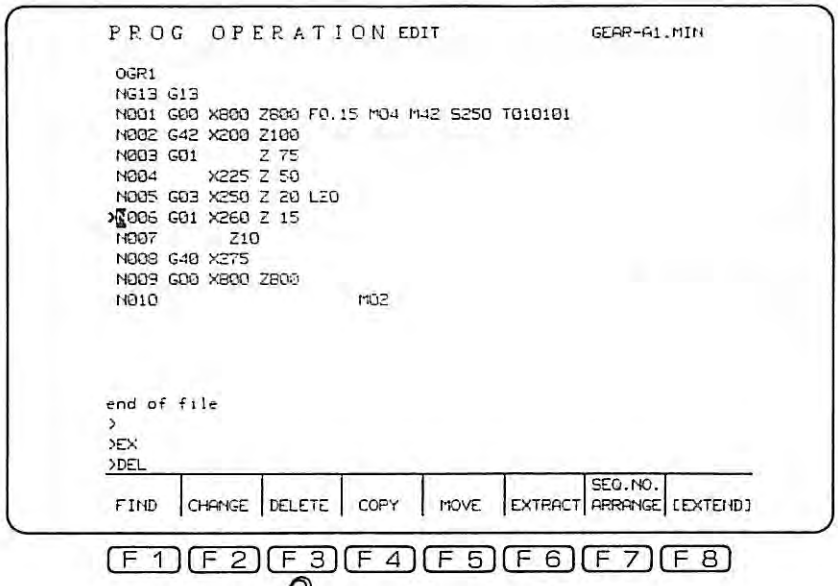
This deletes eight lines before the edit line (edit line is not deleted.).

[F3] (DELETE) **WRITE**

This deletes only the edit line.

To delete two lines (blocks) from N006 to N007:

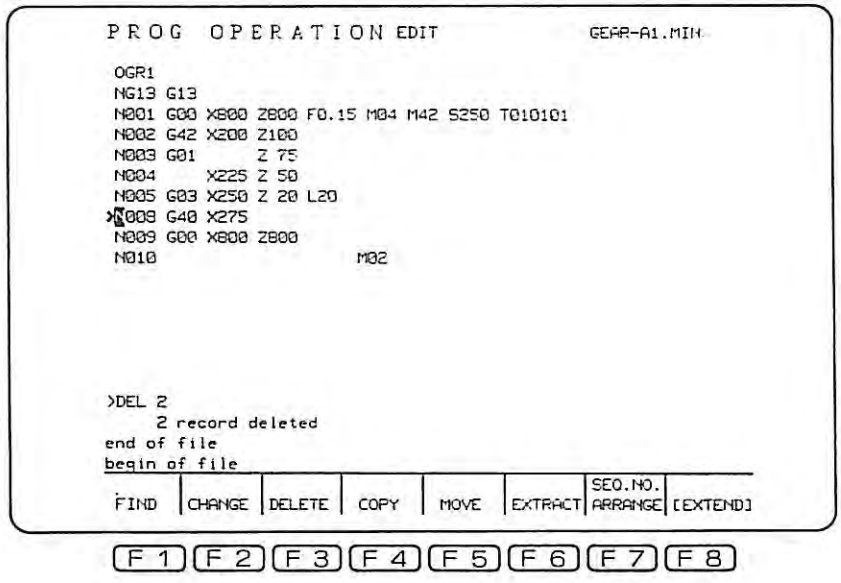
Move the cursor to the sequence number N006.



Key in "2" through the keyboard.

Press the **WRITE** key.

The console line of the CRT displays the following:



(4) Copy [F4]

The commands in the specified number of lines (blocks) from the edit line identified by "≫" (edit line included) are duplicated in another area of the memory.

The edit pointer moves to the first character of the line following the duplicated blocks.

The data already stored in the area where the data is duplicated is deleted.

When the number of lines specified is so large as to exceed the final line of the file, commands up to the final line of the file are duplicated.

When a negative number is specified, blocks before the edit line (excluding the edit line) are duplicated. If the specified number exceeds the first block of the file, the data up to the first block of the file is duplicated.

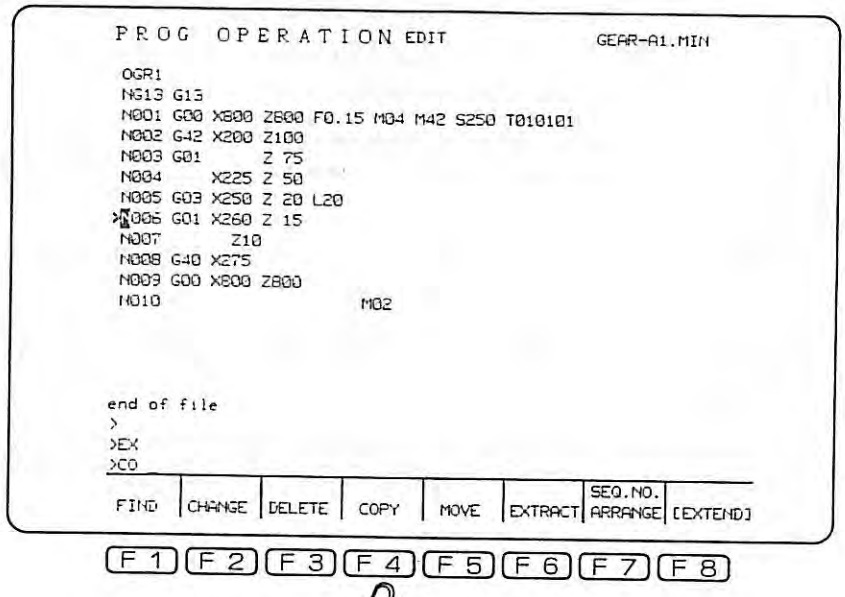
When the **WRITE** key is pressed without keying in the number of lines (blocks), only the present edit line is duplicated.

If the volume of the data to be duplicated is larger than the available capacity of the area for data duplication, the message **"extract buffer overflow"** appears on the CRT, and duplication of the data is not performed.

The data saved in the extract buffer by copy operation can be inserted at any required position in a program by pressing the function key [F6] (EXTRACT). See (6), "Extract [F6]".

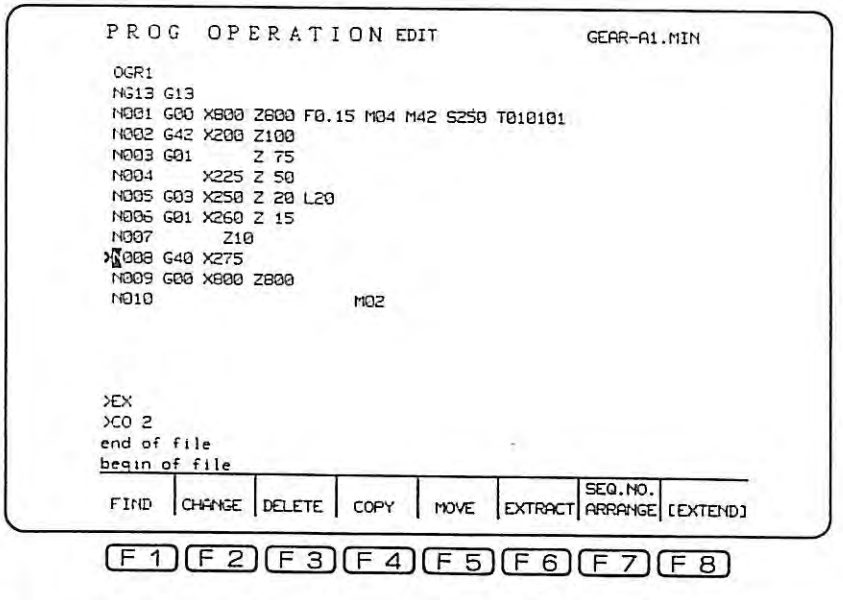
When duplicating blocks from N006 through N007:

Move the cursor to the sequence number N006.



Key in "2" through the keyboard.

Press the **WRITE** key.



(5) Move [F5]

This function permits the operator to transfer the data in the specified number of blocks from the editing area.

The commands in the specified number of lines (blocks) from the edit line identified by "≫" (edit line included) are transferred from the editing area.

The lines of those transferred blocks are deleted.

The edit pointer moves to the first character of the line following the transferred blocks.

The data already stored in the area where the transferred data is stored is deleted.

When the number of lines specified is so large as to exceed the final line of the file, commands up to the final line of the file are transferred.

After the completion of transfer, the message "** **record deleted**" appears on the console line of the CRT.

When a negative number is specified, blocks before the edit line (excluding the edit line) are transferred. If the specified number exceeds the first block of the file, the data up to the first block of the file is transferred.

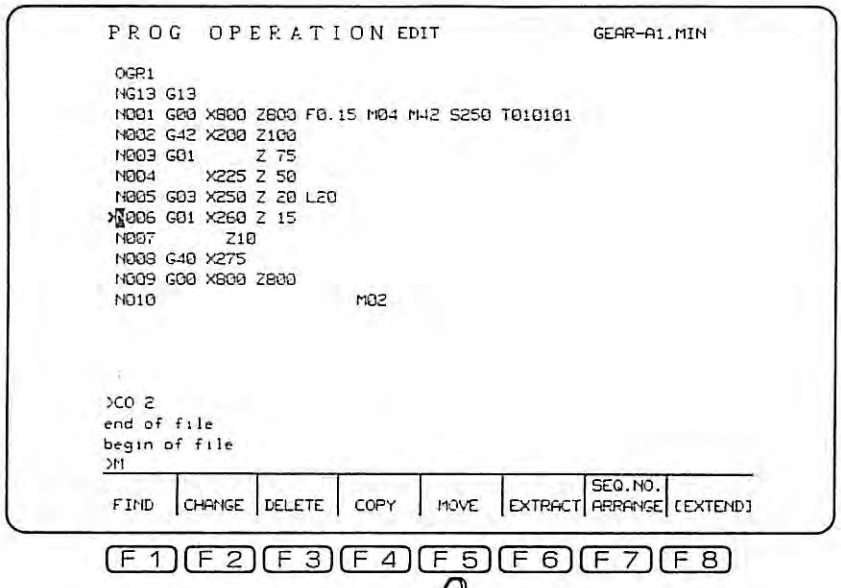
If the volume of the data to be transferred is larger than the available capacity of the area for data transfer, the message "**extract buffer overflow**" appears on the CRT, and the transfer of the data is not performed.

When the **WRITE** key is pressed without keying in the number of lines (blocks), only the present edit line is tranferred.

The data saved in the extract buffer by move operation can be inserted at any required position in a program by pressing the function key [F6] (EXTRACT). See (6), "Extract [F6]."

When transferring blocks N006 through N007:

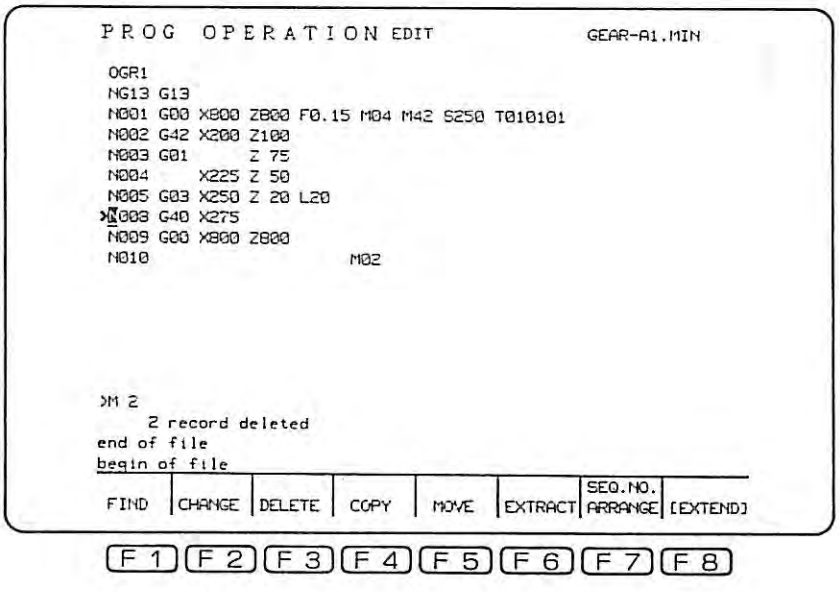
Move the cursor to the sequence number N006.



Key in "2" through the keyboard.

Press the **WRITE** key.

The console line of the CRT displays the following:



(6) Extract [F6]

This function permits the operator to insert the data stored in another memory area to the line before the one preceded by the edit line indicator "≫" each time the **WRITE** key is pressed.

Data stored in another memory area is inserted in to the line(s) before the present edit line.

When the insertion of the data stored in another memory area is performed, that data is not cleared.

The position of the edit pointer does not change.

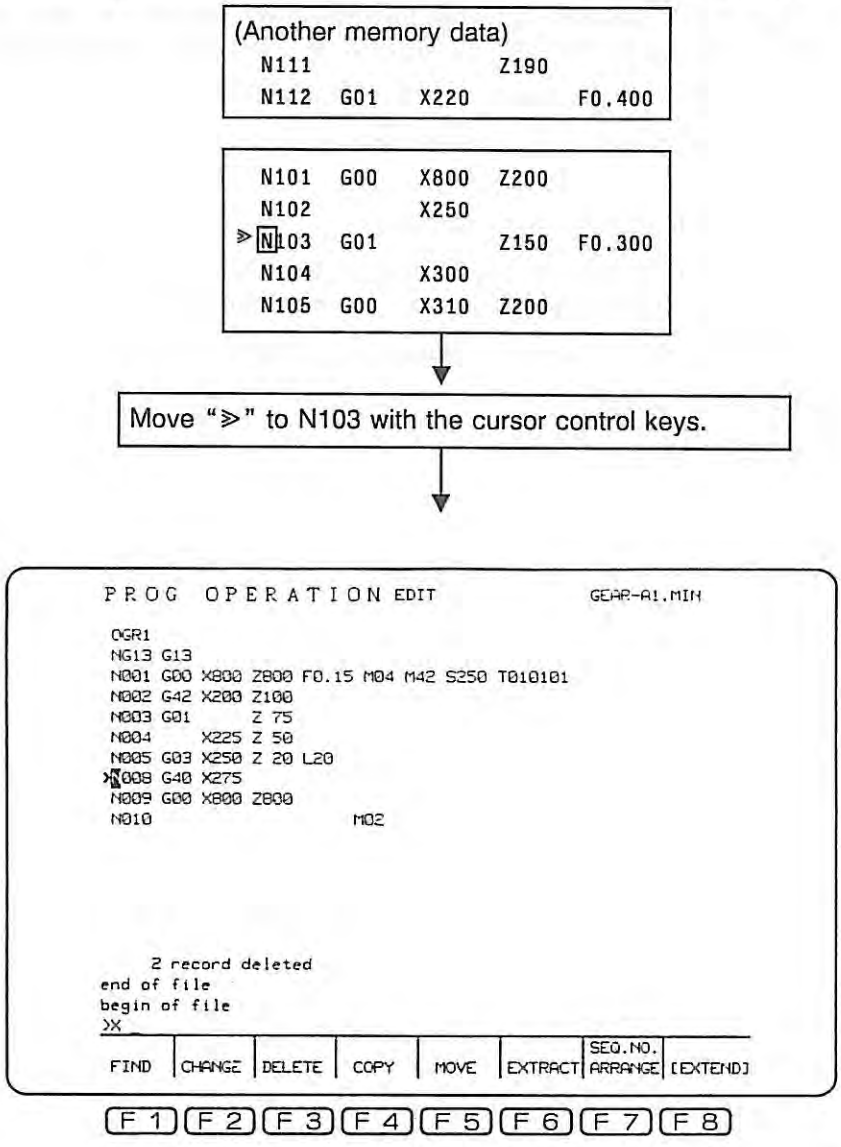
If no data is stored in another memory area when insertion is intended, the CRT displays the message "**extract buffer empty**" and insertion is not performed.

To delete the data stored in another memory area, perform as follows:

[F6] (EXTRACT); C **WRITE**

File data is not changed.

When inserting two blocks of data in another memory area before N103:



(7) Sequence Number Arrange [F7]

This function rearranges the sequence numbers of the program after the completion of editing. Pressing the function key [F7] (SEQ. NO. ARRANGE) begins this function.

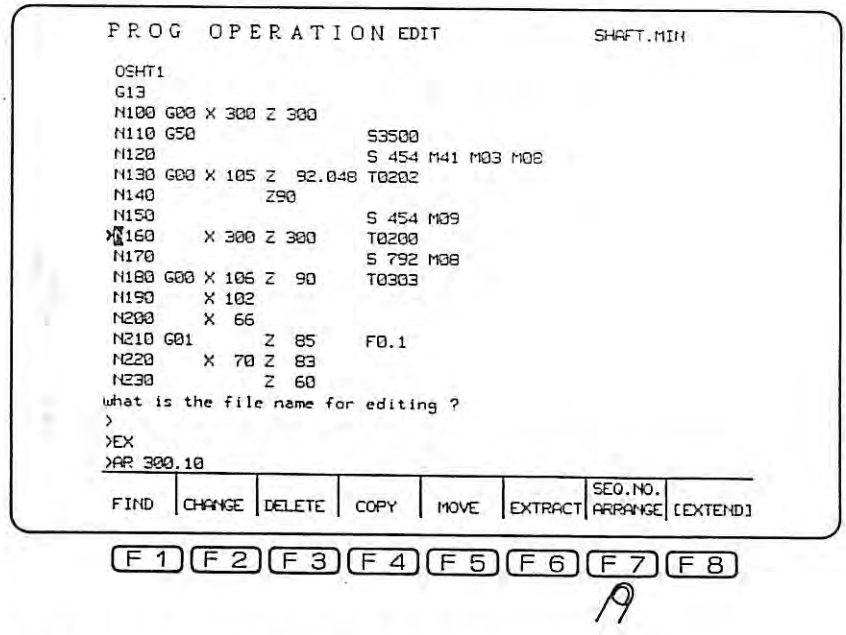
Procedure:

1) Designate the first line of sequence number rearrangement operation by the edit pointer (≫).

In this example, the sequence numbers from N160 is rearranged.

2) Press the function key [F7] (SEQ. NO. ARRANGE).

The prompt "AR" will appear on the console line.



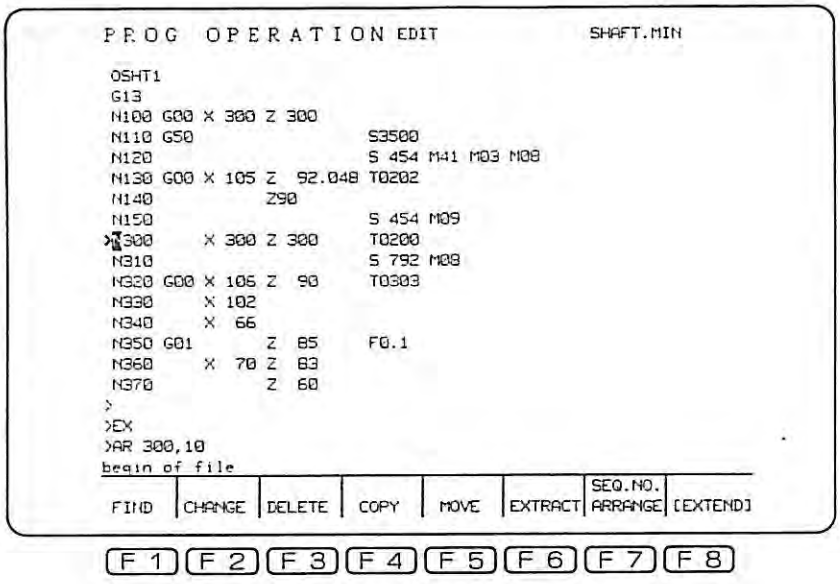
3) Key in the starting sequence number and increments.

In this example, the new sequence number N160 starts from "N300" in increments of "10".

If the sequence number increment value is omitted, it indicates "1".

4) Press the **WRITE** key.

This assigns the new sequence numbers to the program from N300 up to the program end sequence (M02, M30, RTS).



Note 1: *Destination sequences of GOTO and IF statements and LAP shape definition sequences should be assigned with sequence name. This arrangement operation does not change sequence names.*

Note 2: *Spaces, tabs, slashes placed in front of and after the previous sequence number are all left as they were specified.*

Note 3: *Previous sequence names must not be preceded by any characters other than space, tab and slash.*

Note 4: *Sequences assigned with neither sequence number nor sequence name are newly assigned with sequence numbers.*

Sequences in the program name and within control OUT area are not assigned with sequence numbers.

Note 5: *If the sequence on which the edit pointer is placed is assigned with the sequence name, sequence numbers are rearranged from the sequence assigned with a sequence number and appearing first after the sequence located by the edit pointer.*

Note 6: *If the number of digits of the sequence number exceeds four, an alarm occurs and sequence number rearrange is not effective for those which are to be assigned with the sequence numbers of more than four digits.*

Note 7: *Leading zeros in sequence numbers should be suppressed.*

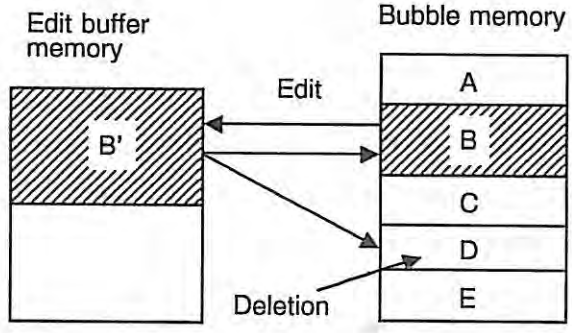
Note 8: *Rearrangement of sequence numbers is carried out in units of programs. In other words, sequence numbers are rearranged only up to the sequence containing M02, M2, M30 or RTS.*

Note 9: *For the line beginning with the symbol "&", sequence number is not assigned because such a line is interpreted as the line continuing from the previous line.*

(8) Restoration and Preserving of Edit Data

Outline:

This function is useful when there is no area to store the edited program after finishing program edit. First, save the edited program in memory and then secure blank area in the bubble memory. Thus the temporarily saved program can be stored in the bubble memory.



Procedure:

1) If the function key [F7] (EDIT QUIT) is pressed, when the bubble memory is full, **"2154 SAT full"** is displayed on the console line and editing cannot be completed.
2) Press the function key [F7] (EDIT QUIT) to save it in memory. Then **"edit buffer restore"** is displayed and editing can be completed temporarily.
3) Secure blank area by deleting unnecessary data in the bubble memory.
4) Press [F4] (EDIT) again and input the file name. Then input the option code R and press **WRITE** key.

   =E↵ (file-name);R **WRITE**

   The message **"edit buffer preserve"** is displayed and the editing operation of 1) can be continued.
5) Press the function key [F7] (EDIT QUIT) and store the data saved in memory in the bubble memory.

Note 1: When the floppy input/output function (optional) is selected, take the same procedure for the files on the floppy disk.

*Note 2: After finishing editing with **"edit data store"**, turning off the power or execution of tape convert, IGF or IGF convert erases the saved data. When proceeding with the next operation after editing, be sure to delete unnecessary file and store the edited data in the bubble memory.*

(9) Other Extended Functions

By modifying one portion of a program already stored in the bubble memory, such as outside diameter, cutting conditions or other, a new main program can be created on that base without affecting the previous program.

Follow the steps below:

Assume the program file name is "SHAFT-1350.MIN".

1) Since the program above must not be changed, assign a new file name for the newly created program. (Let the new file name be "SHAFT-1350-1.MIN".)

2) After pressing the function key [F4] (EDIT), key in as follows:

SHAFT-1350.MIN, SHAFT-1350-1.MIN

3) Press the **WRITE** key.

This creates a new file name, SHAFT-1350-1.MIN. If the bubble already contains a file named SHAFT-1350-1.MIN, the message **"file exist overwrite (Y/N)!"** appears on the console line of the CRT.

Pressing [Y] and **WRITE** keys delete the previous file data.

4) After editing the necessary data, press the function key [F7] (EDIT QUIT) to indicate the completion of editing.

With the steps above, a new program is created with the name SHAFT-1350-1. MIN: select this program in AUTO mode and carry out machining with those commands.

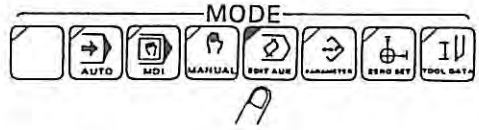
2-3. Other Program Operation Functions

(1) Entry of File Generating Date

When a new file is registered, or when the program tape with a new file name is read, it is stored in the bubble memory with the assigned date. Therefore, it is necessary to enter the date of program, if no date is entered, the date determined by the system is entered.

Procedure:

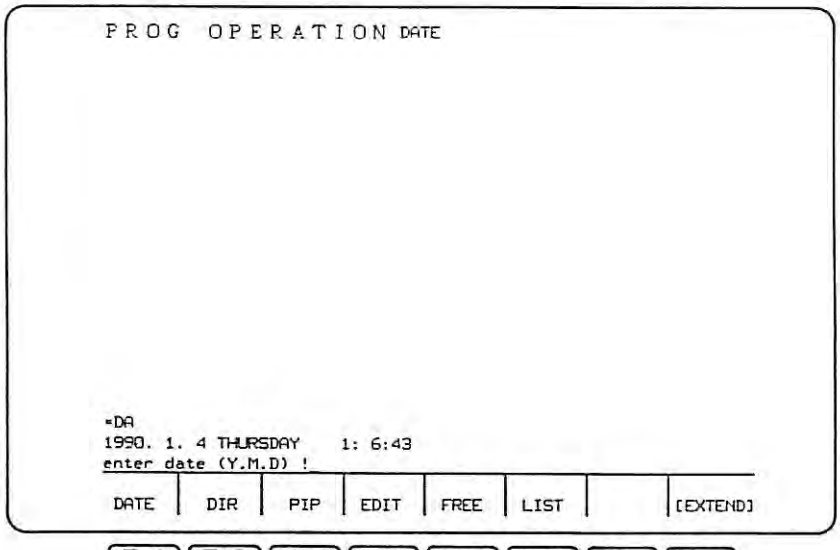
1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Press the function key [F1] (DATE).

The message below appears on the console line of the CRT, and the control is ready for date entry.

=DA
1990.1.4 THURSDAY 1:6:43
enter date (Y.M.D)!

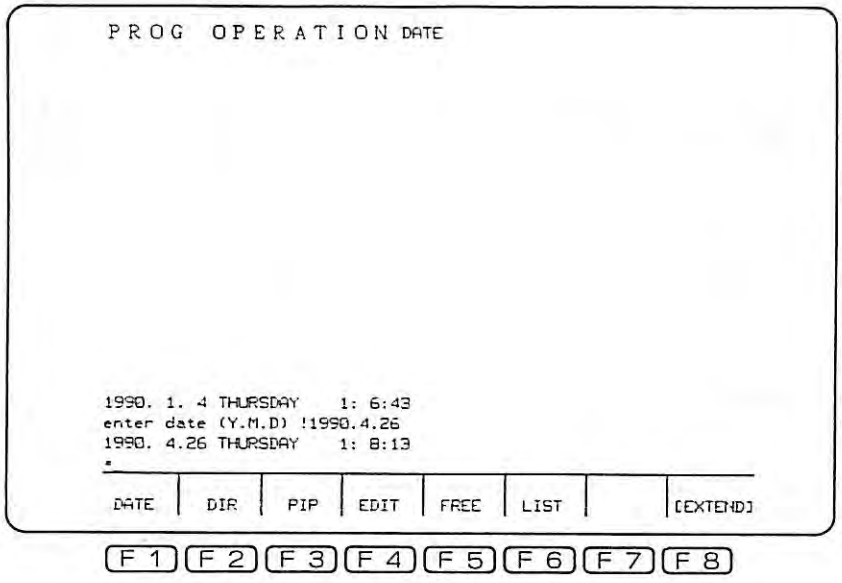


3) Through the keyboard, after the symbol "!" indicating, enter "year", "month", and "day" in that order, each separated by a period from each other.

Example: 1990.4.26

4) Press the **WRITE** key.

When the entered data is correct, the CRT shows the entered date along with a day of the week. If the entered data is not correct, the control requires the data again.



Note: *To enter the year, the lower two-digit data is acceptable.*

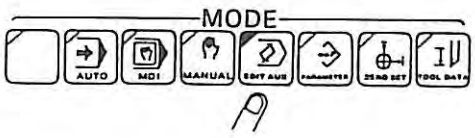
*When the **WRITE** key is pressed without data entry, no display is obtained.*

(2) Display of File Names

The operator can check the files stored in the bubble memory by making the CRT display them as a list. The list shows the file names, how each file uses the memory, and the registered date.

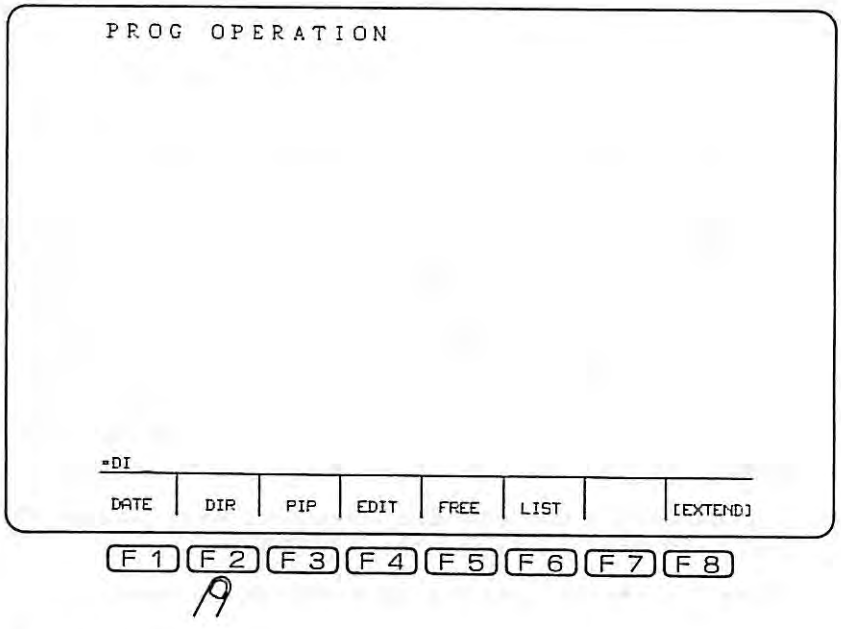
Procedure:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



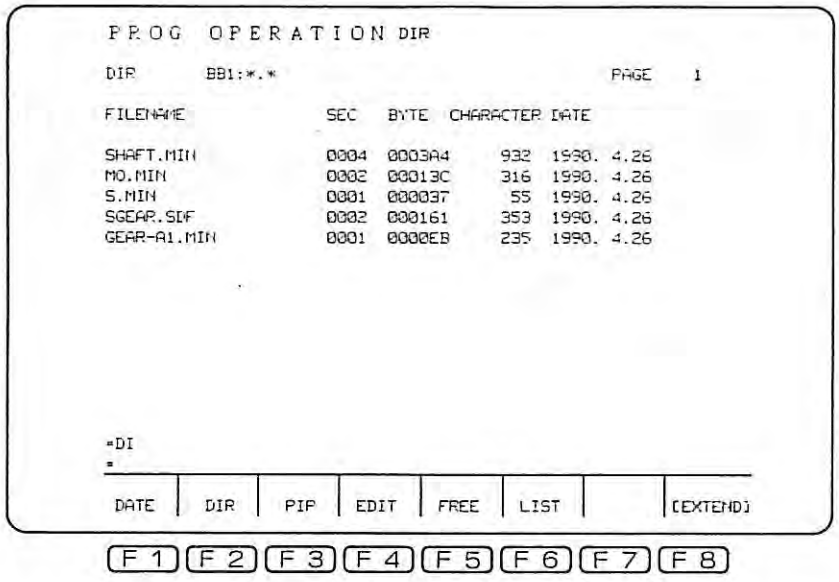
2) Press the function key [F2] (DIR).

The prompt "DI" appears on the console line of the CRT.



3) Press the **WRITE** key.

4) The CRT shows the list of files.



Note 1: *Up to 12 files can be displayed on one CRT display. If all the files cannot be shown on one CRT display, the symbol "=" indicating that the control is waiting for next instruction does not appear and the cursor remains as it is. In this case, press either the* ***BS*** *or the* ***WRITE*** *key. The* ***BS*** *key is used to turn the display page forward. The* ***WRITE*** *key turns pages continuously up to the end of the file display.*

Note 2: *When no file is saved in the bubble memory, the CRT shows the message* ***"file not found"****.*

Note 3: *The operator can check whether the required file is stored in the bubble memory or not, by entering the file name following "DI".*

*There, " * " and "?" can be used to enter the file name:*

**: Arbitrary character-string*
(can be used only once in file name and extended name)

?: Arbitrary one character

*Therefore, designating a file name containing " * " and "?" can produce a list showing the file names related to the entered data.*

Example 1: `=DI—*.MIN` **WRITE**

All the files assigned with extension MIN. are displayed.

Example 2: `=DI—SHAFT*.MIN` **WRITE**

All the files beginning with SHAFT and having extension MIN are displayed.

Example 3: `=DI—*.*` **WRITE**

All the files are displayed. This data entry has the same effect as if no data is entered for file name display.

Example 4: `=DI—???.SUB` **WRITE**

All the files having a file name with three or less than three characters and possessing the extension SUB are displayed.

Example 5: `=DI—SHAFT-1???.*` **WRITE**

All the files with a file name that begins with SHAFT-1 and suffixed by up to three characters are displayed.

Note 4: *Option C function*

Writing in of ";P" following the depression of the function key [F2] will result in a protect status display for each file.

`=DI;P` **WRITE**

With the operation above, the display will show the date followed by a code:

00 = File not protected
01 = File protected

(3) Display of Available Memory Capacity

The OSP500L-G has the capacity to store program data up to 30 meter tape long and the OSP5020L 60 meter tape long. However, the available capacity area decreases as files are stored. The control has the feature to show the remaining available memory capacity.

The display is obtained in "sectors" and "bytes": those two expressions have the same character and have the relation shown below.

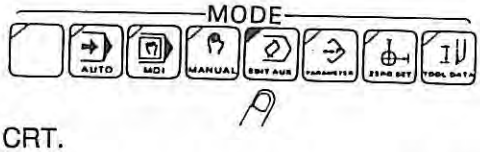
Number of bytes = Number of sectors × 252

One byte corresponds to one character. Since data is stored in the memory in the unit of a sector, data comprising only one or two characters occupy one sector. When the data comprises 253 characters, 252 characters occupy one sector and the last 253rd character occupies one sector.

Note that the bubble memory stores the data file name and others in addition to program data; actual available capacity might be smaller than the capacity expressed in the number of bytes.

Procedure:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Press the function key [F5] (FREE).

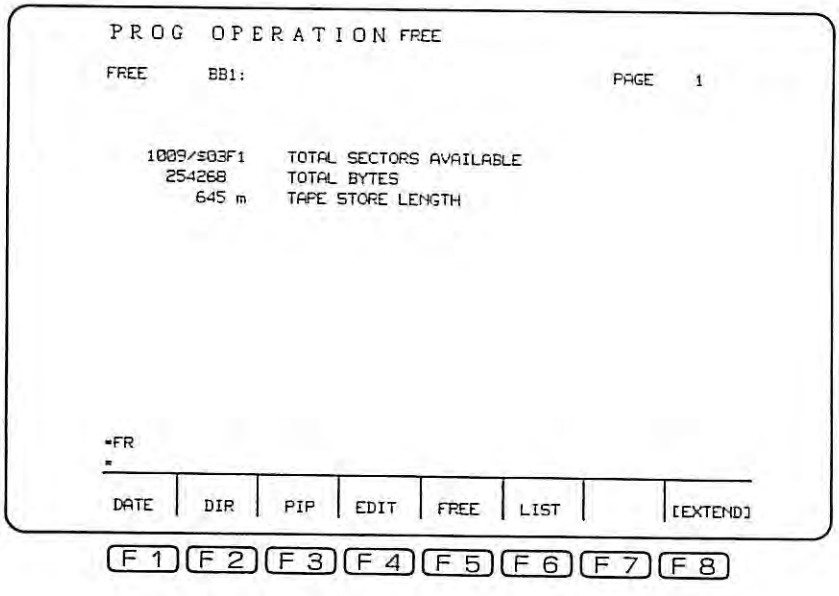
The prompt "FR" appears on the console line of the CRT.

Option code ";C" following the FR command is used to display the contiguous available sectors and bytes.

3) Press the **WRITE** key.

Now, the CRT displays the remaining capacity of the memory.

1009/$03F1 TOTAL SECTORS AVAILABLE
254268 TOTAL BYTES



Note: *$03F1 is hexadecimal notation of "1009".*

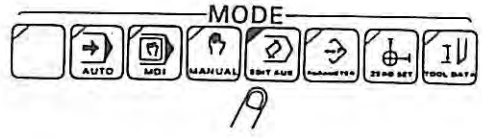
$$1009 = 3 \times 16^2 + 15 \times 16 + 1$$

(4) Listing Specified File

As explained in (2), file names can be checked using the function key [F2] (DIR). However, the contents of the files cannot be checked. This paragraph deals with the method used to check the contents of the file.

Procedure:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Press the function key [F6] (LIST).

The prompt "=L" appears on the console line of the CRT.

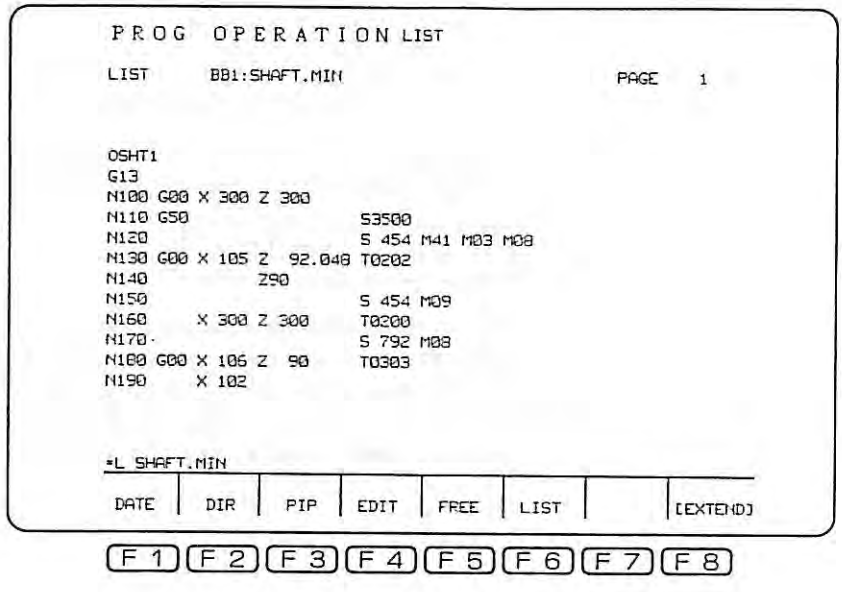
3) Enter the file name directly following "=L".

Example: =L—SHAFT.MIN

Entered file name

4) Press the **WRITE** key.

This displays the contents of file SHAFT-1.MIN on the CRT.



Note 1: *Up to 12 lines can be displayed on one CRT display. If all the contents cannot be shown on one CRT display, the symbol "=" indicating that the control is waiting for the next instruction does not appear and the cursor remains as it is. In this case, press either the* ***BS*** *or the* ***WRITE*** *key. The* ***BS*** *key is used to turn the display page forward. The* ***WRITE*** *key turns pages continuously up to the end of the file display.*

Note 2: *When the file name following "=L" is omitted, contents of file A.MIN are displayed.*

That is, (=L↵, [WRITE]) = (=L↵,A.MIN ***WRITE****)*

Note 3: *The operator can specify the device name used for output.*

The type of output device and corresponding code are as follows:

CRT screen . . . . . PN:
Printer (optional) . . . . . PR:*
*Printer (optional)** . . . . . CN0: - CN4:*

* *Printer connected through the data board Centronics interface*
** *Printer connected through RS232C interface*

CN0: . . . . . RS232C interface on main card 3
CN1: - CN4: . . . . . RS232C interface on RS board

To specify the output device, use a delimiter "," before the output device code and use ":" after it.

When no output device name is specified, the data is displayed on the CRT.

Example 1: `=L↵,PR:` ***WRITE***

Contents of A.MIN are output on the printer.

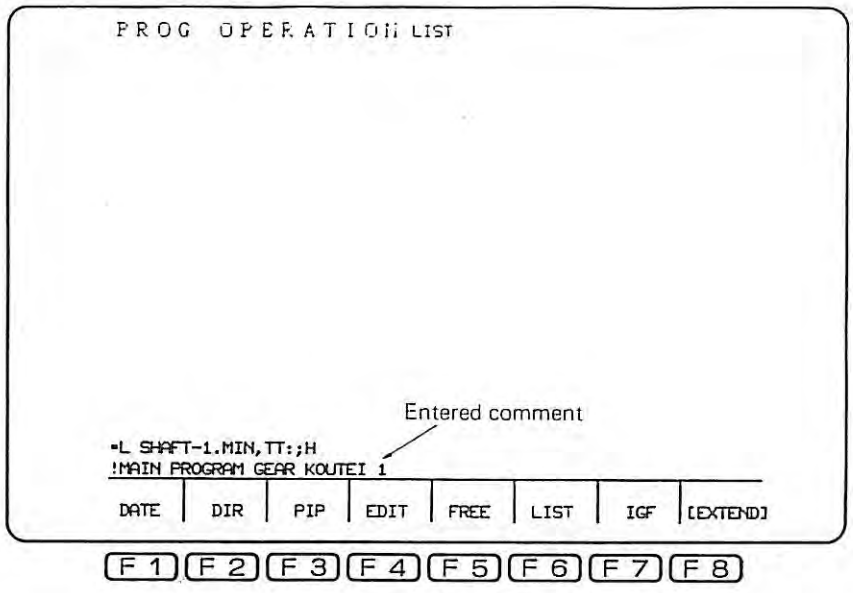
Example 2: `=L↵SHAFT-1.MIN,CN0:` ***WRITE***

Contents of the file SHAFT-1.MIN are output to the printer which is connected to channel CN0.

Note 4: *The use of ";H" (option code) statement following the commands for file listing permits the operator to add a comment at the beginning of the list page.*

Example: `=L↵GEAR-1.MIN,TT:;H` ***WRITE***

With the commands indicated above, "!" appears on the console line of the CRT, and the control waits for the entry of a comment. A comment up to 60 characters can be entered through the keyboard. Pressing the ***WRITE*** *key after that initiates the output of a list.*



Output using an RS232C device is as follows:

The entered comment is output on each output page.

```
L   BB1:  GEAR-1.MIN
MAIN PROGRAM GEAR KOUTEI 1

OGR-1
G14
G00          X800   Z150
N2    G42    X200   Z130   T010101   S200   M03
N3    G01           Z80
N4           X60    Z100
N5                  Z40
```

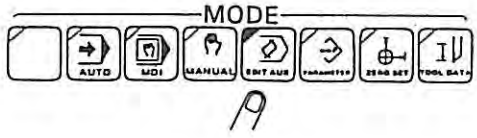
(5) Entry of Time

The control can time with reference to the set time, and allow the operator to check the time.

This feature can be effectively used to print the time at which the measuring cycle was executed when the machine is equipped with the optional measuring data printout specification.

Procedure:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.

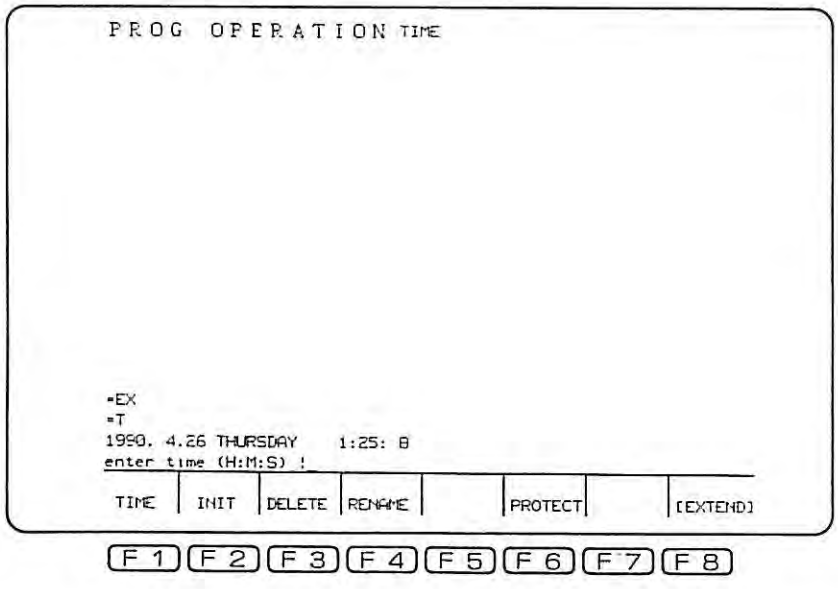


2) Press the function key [F8] (EXTEND).

3) Press the function key [F1] (TIME).

With this, the following display is obtained on the console line of the CRT, and the control is ready for the entry of time data.

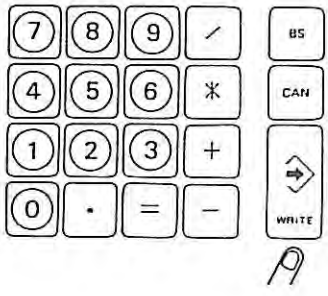
=T
1990.4.26 THURSDAY 1:25:8
enter time (H:M:S)!

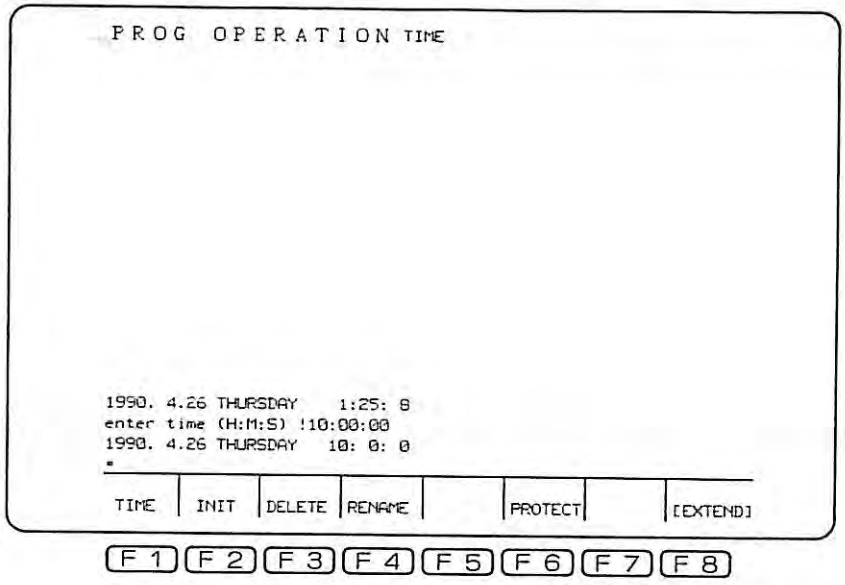


4) Enter hour, minute, and second data, each separated by " : " through the keyboard.

Example: 10:0:0

5) Press the **WRITE** key.





The CRT displays the entered time data including date and day of the week if the entry is correct.

When the set data is not correct, the CRT again displays the message **"enter time (H:M:S)!"**. Enter the correct data, then.

Note 1: *When the operator wants to check the time and when no setting is necessary, simply press the **WRITE** key after the completion of step 3). This terminates the time function.*

Note 2: *Range of numeral values entered is:*

- 0 through 23 for hour
- 0 through 59 for minute
- 0 through 59 for second.

Note 3: *After 23 hours 59 minutes 59 seconds, the time is restored to 0 hour 0 minute 0 second. Along with such change, the day of the month and day of the week also change.*

Note 4: *The time is counted from the moment the control is ready with the power turned on.*

(6) Clearing The Entire Stored Data

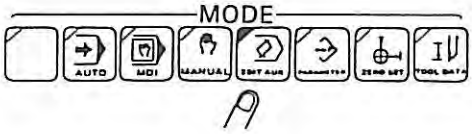
This is to initialize the bubble memory. Be sure to initialize it after turning on the power to the control for the first time after control tape loading.

The following step erases all the data stored in the bubble memory.

REMARKS

Procedure:

1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Press the function key [F8] (EXTEND).

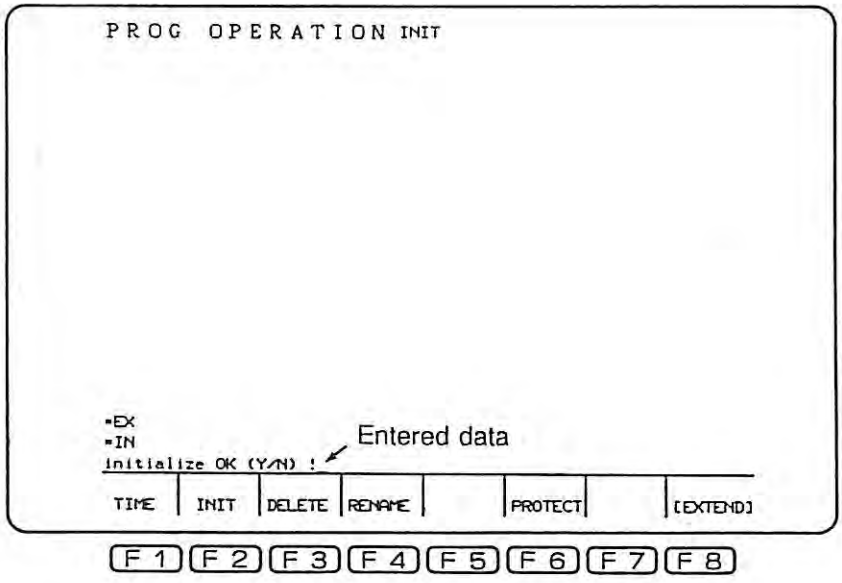
3) Press the function key [F2] (INIT).

The message below appears on the console line of the CRT, and the CRT waits for an answer from the operator.

= IN
initialize OK (Y/N)!

Enter "Y" through the keyboard to initialize the bubble memory. If not, enter "N".

4) Press the **WRITE** key.



When [Y] key is pressed in step 4), the bubble memory is initialized.

Pressing [N] does not initialize the bubble memory. This is used when the function key [F2] is pressed erroneously.

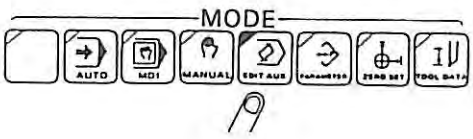
Note: *After the NC control software has been loaded, be sure to initialize the bubble memory.*

(7) Deleting Entire Files

Files stored in the bubble memory can be deleted.

Procedure:

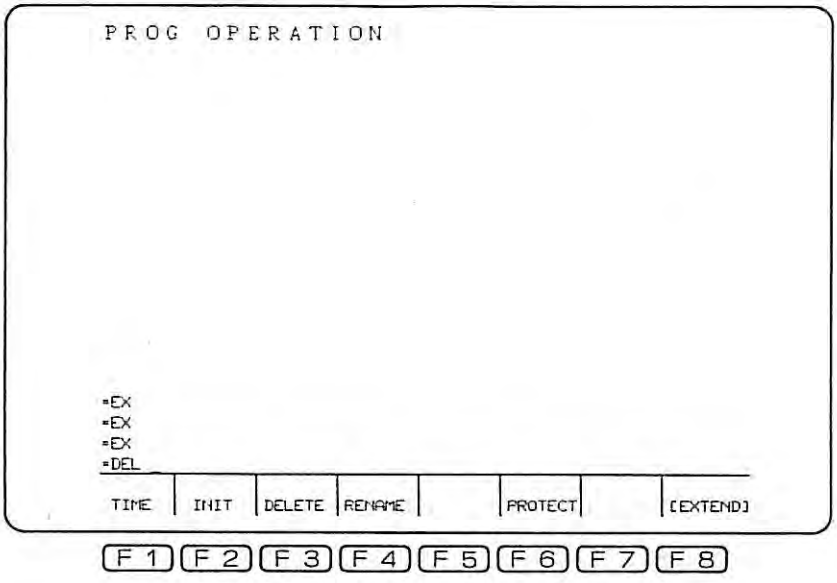
1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Press the function key [F8] (EXTEND).

3) Press the function key [F3] (DELETE).

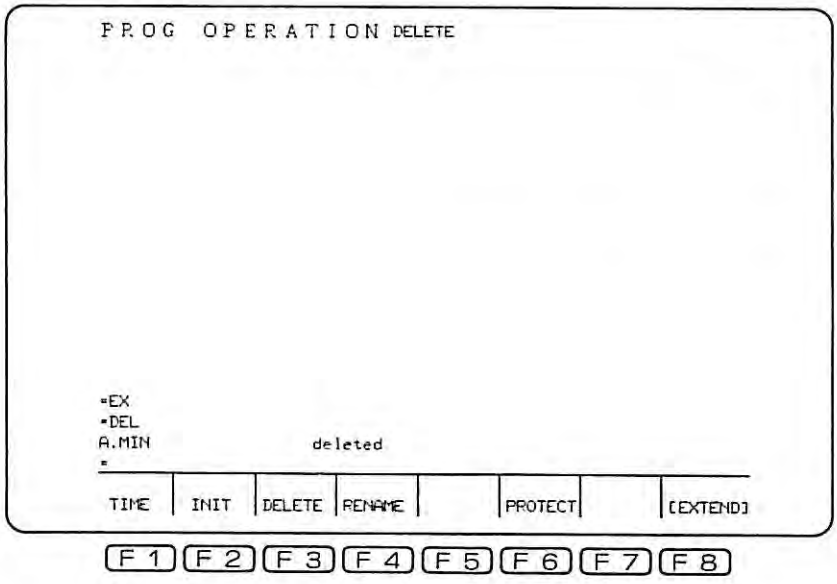
The prompt "DEL" appears on the console line of the CRT.



4) Press the **WRITE** key.

This deletes the file A.MIN and the following message appears on the console line of the CRT.

A.MIN deleted



Note 1: *If the name of the file to be deleted is other than A.MIN, designate the file name. When the extension of a file name is MIN, it can be omitted.*

Example: ABC.MIN . . . . . DEL ABC Omissible
ABC.SUB . . . . DEL ABC.SUB

Note 2: *When the file designated for deletion is not in the bubble memory, the message* ***"file not found"*** *appears on the console line of the CRT.*

Note 3: *" * " and "?" can be used for designating a file name.*

See Note 3 in (2).

Example:

*DEL *	*All the files with extension .MIN are deleted.*
*DEL * .SUB	*All the files with extension .SUB are deleted.*
DEL SHAFT.*	*All the main files beginning with SHAFT are deleted.*
*DEL *.*	*All the files are deleted.*
*DEL ???.SUB	*Main files with a file name containing up to three characters and with the extension. SUB are deleted.*

Note 4: *Option for file deletion*

By specifying ";V" after the name of the file to be deleted, the operator can visually check the file name on the console line of the CRT; he can determine whether that file should be deleted or not.

Example: *DEL*.*;V*

Pressing the ***WRITE*** *key after entering as above will display* ***SHAFT-A1.MIN delete OK (Y/N)!*** *on the console line.*

Press the ***WRITE*** *key after entering "Y", and the file is deleted.*

Entering "N" followed by the ***WRITE*** *key does not delete it.*

The CRT then displays:

SHAFT-A2.MIN delete OK (Y/N)! *on the console line.*

Repeat as explained above.

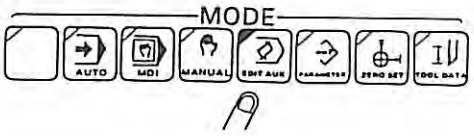
*In this example, " *.* " has specified all the files, requesting the operator to determine for all the files whether the file is to be deleted or not.*

(8) Renaming File

The file name of the stored program can be changed.

Procedure:

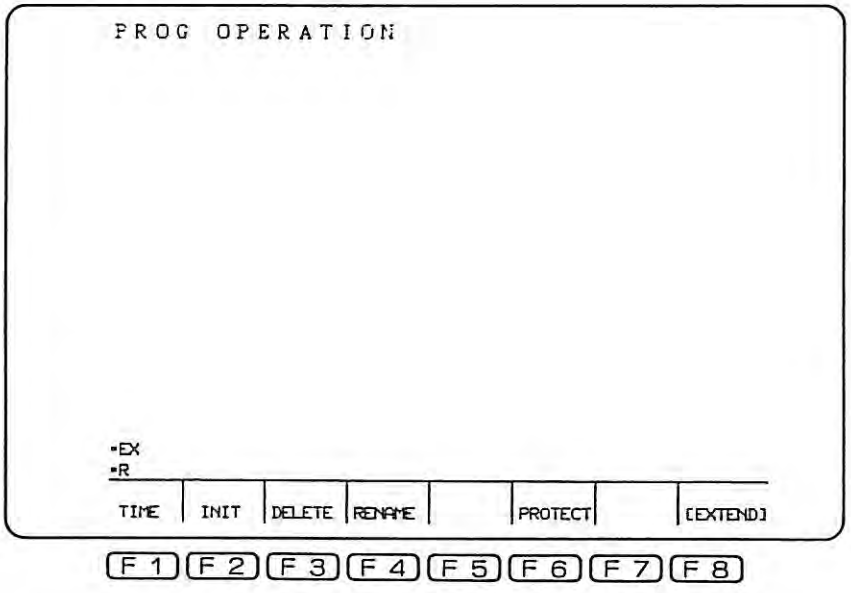
1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Press the function key [F8] (EXTEND).

3) Press the function key [F4] (RENAME).

The prompt "R" appears on the console line of the CRT.



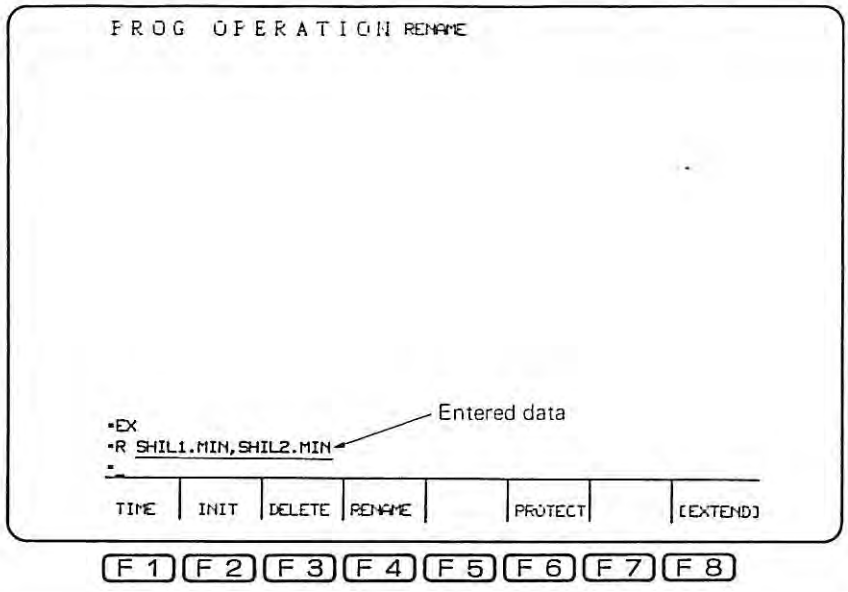
4) Enter the file names, old file name and new file name separated by a delimiter "," through the keyboard.

Example:

SHIL1,SHIL2	Changing SHIL1.MIN to SHIL2.MIN.
SHIL1.SUB,SHIL1.SSB . . .	Changing SHIL1.SUB to SHIL1.SSB.
SHIL1.SUB,SHIL2.SUB . . .	Changing SHIL1.SUB to SHIL2.SUB.

5) Press the **WRITE** key.

This completes the change of file name.



Note 1: *When the file name specified is not found in the bubble memory, the message **"file not found"** appears on the console line of the CRT.*

Note 2: *When the newly assigned file name is already in the bubble memory, the CRT shows the message **"file exist"**.*

Note 3: *" * " and "?" cannot be used in specifying the file name and extension.*

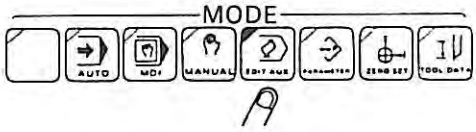
*If used, the message **"file name error"** appears on the CRT.*

(9) Protection of Specified Files

The file specified is protected from unexpected writing, editing, and deletion operations.

Procedure:

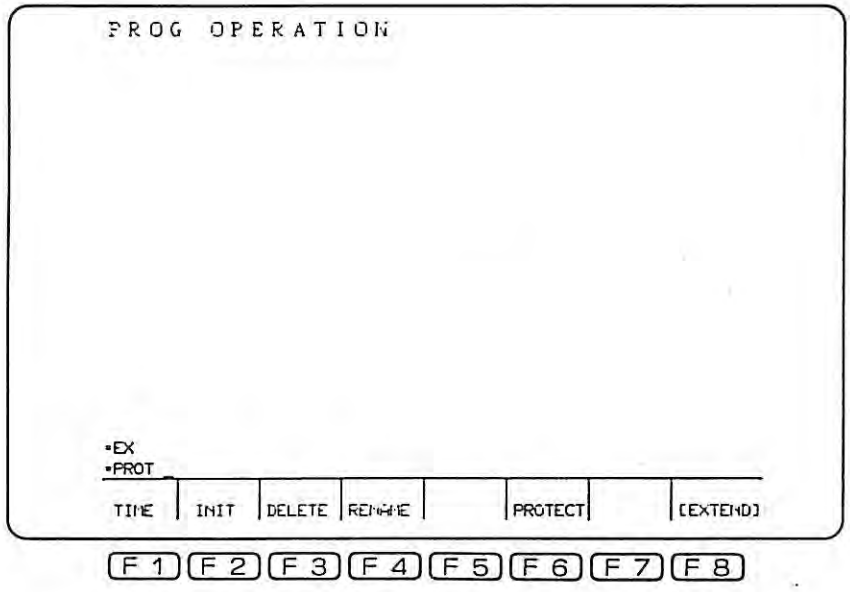
1) Select the PROG OPERATION mode by pressing the **EDIT AUX** key.



2) Press the function key [F8] (EXTEND).

3) Press the function key [F6] (PROTECT).

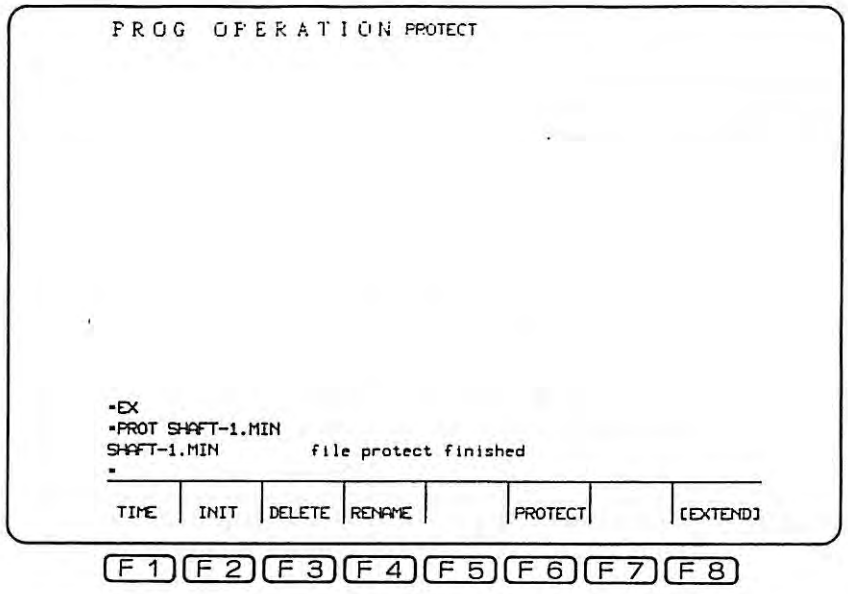
The prompt "=PROT" will be displayed on the console line.



4) Key in the names of files to be protected.

SHAFT-1.MIN, for example.

5) Press the **WRITE** key.



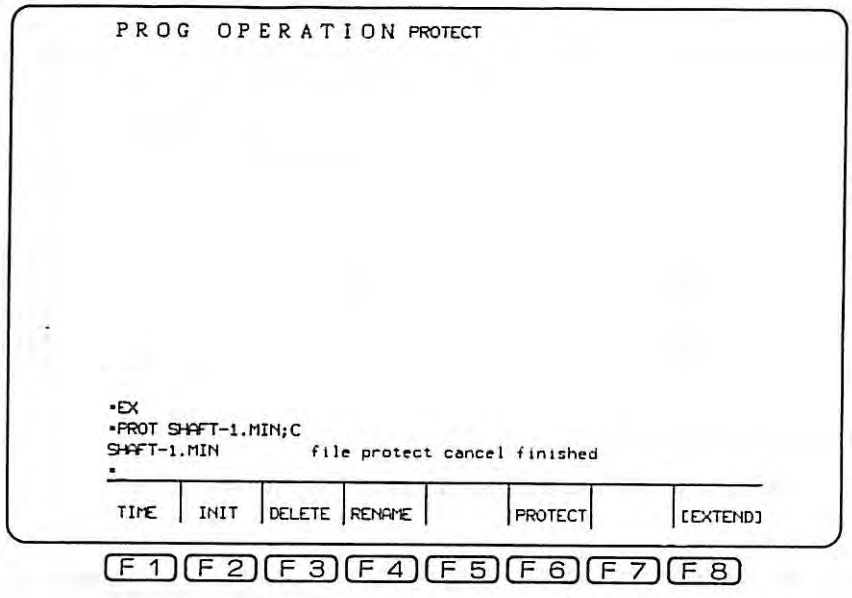
This completes the file protection operation.

Note: Memory initialization operation erases file-protected programs also.

Canceling Protection:

Key in ";C" after the file name in step 4).

SHAFT-1.MIN;C, for example.



Note: The procedure to display the file protected status is given in Note 4 in (2), "Display of File Names" in 2-3.

3. Operation in Parameter Mode

(1) Parameter

To position axes or edit a part program using the control, the axis travel range and tape output code are predetermined in accordance with the functions and the specifications of the OSP5020L/OSP500L-G and the machine. However, it is sometimes necessary to change the matching conditions.

The data necessary to control NC functions is called parameter; the OSP5020L/ OSP500L-G employs the following parameters:

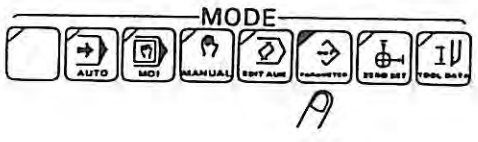
Type of Parameter	Contents	Example	CRT Display Page
User parameter	Parameter determined for respective workpiece types	Variable soft-limit positions Droop amount	1
Chuck/tailstock spindle	Parameter setting for barrier	Chuck gripping direction setting Setting chuck barrier	1
Common variable	System common variable	V1 - V32	1
System parameter	Parameter determined depending on machine	Stroke end positions, backlash, etc.	Varies depending on the specification
Optional parameter (long word)	Functional variable	Unit amount of rapid feedrate, etc.	1
Optional parameter (word)		Spindle oscillation speed, power save timer, etc.	3
Optional parameter (bit)		Edit function Position of decimal points, etc.	1
Turret indexing angle	Indexing angle when turret rotating	NC turret (LB15-Ⅱ series)	1
Spindle control parameter	Parameter determined by the high precision VAC specification	Spindle with the high precision VAC motor specification	Varies depending on the specification

(2) Outline of Parameter Setting

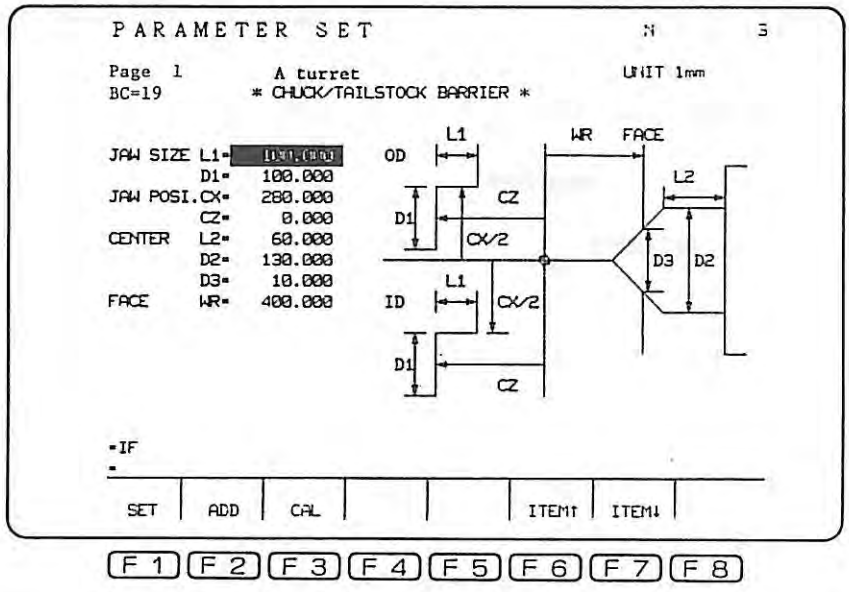
Example: For setting "500" at the optional parameter (word) No. 9

Procedure:

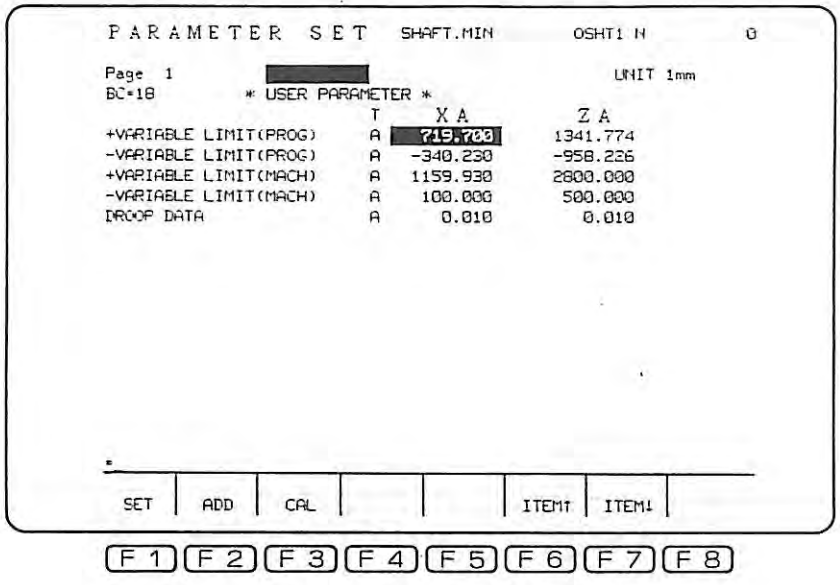
1) Select the PARAMETER SET mode by pressing the **PARAMETER** key.



Chuck/Tailstock Spindle Display:

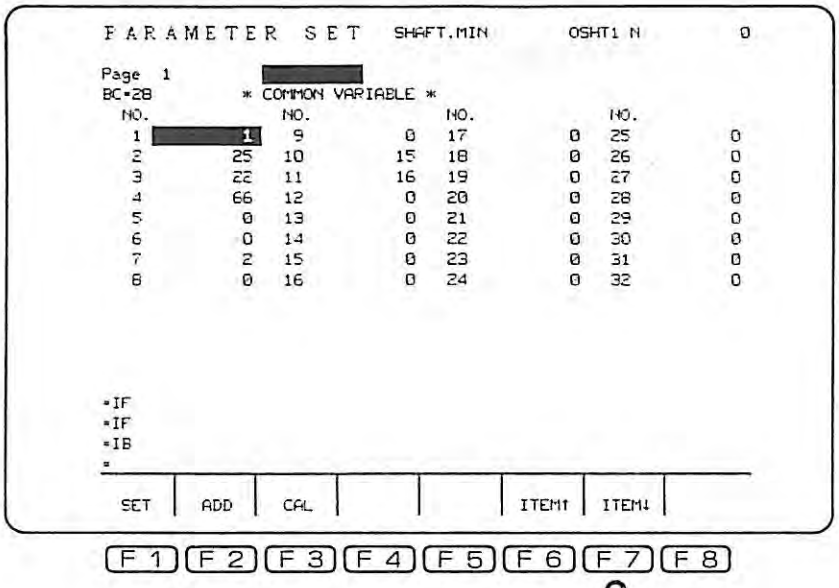


User Parameter Display:

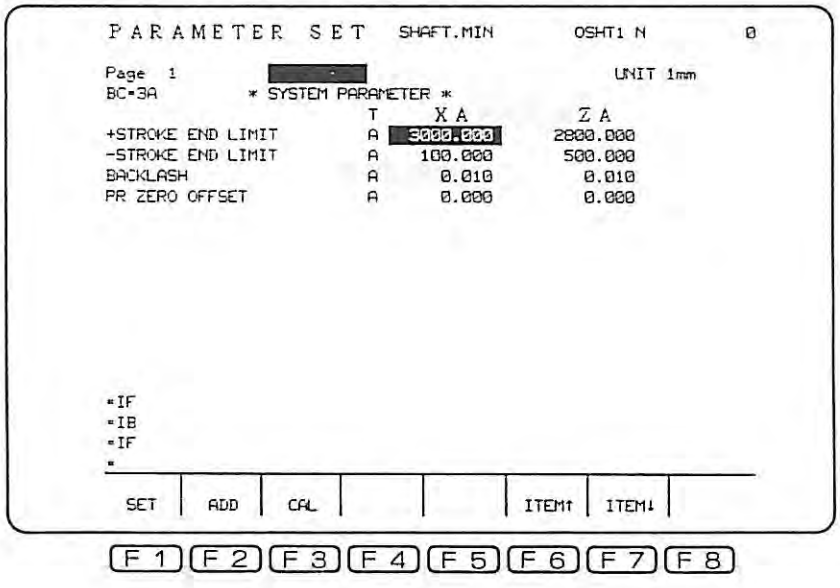


2) Select the type of parameter to set by pressing the function key [F6] (ITEM↑) or [F7] (ITEM↓); lists of parameters are displayed on the CRT.

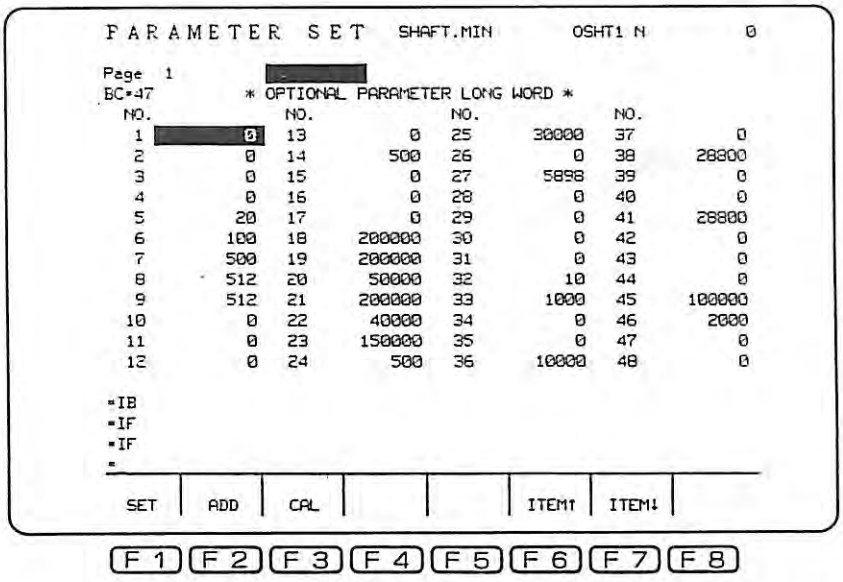
If necessary, press the  or  key to change the page within the same parameter category.



A

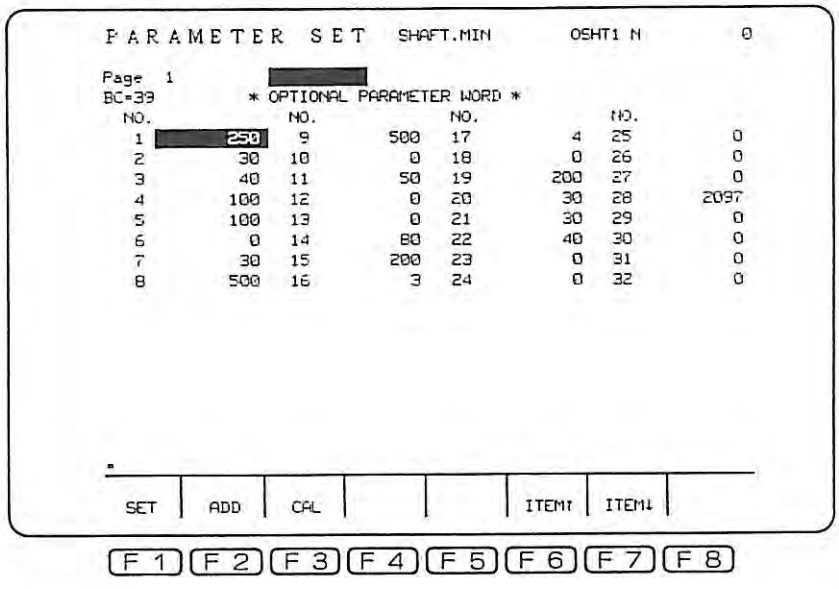


3) Move the cursor to the desired data position.

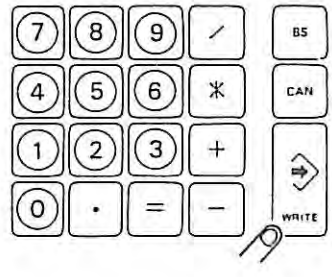


4) Press the function key [F1] (SET).

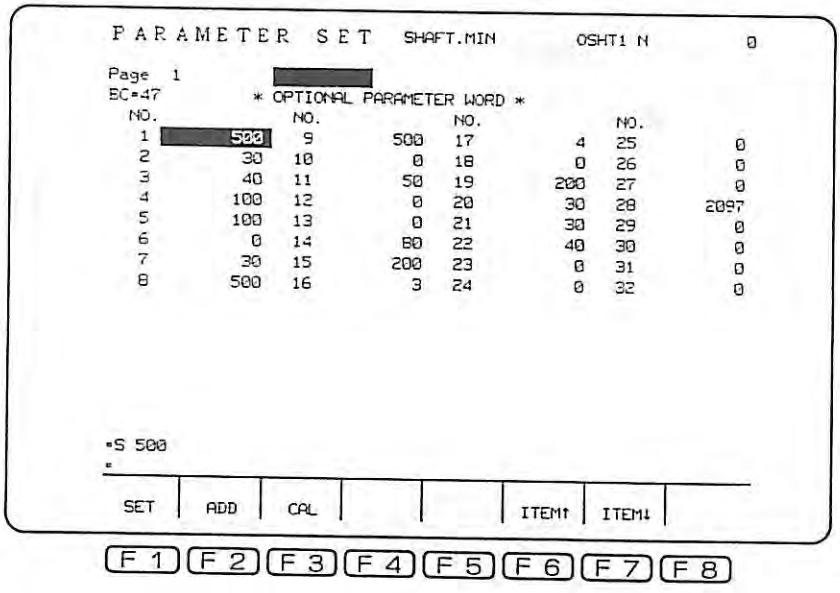
5) Enter the required new data, "500" for instance, through the keyboard.



6) Press the **WRITE** key.



This replaces the data of the selected parameter number with the newly entered data. The CRT displays new data.



Data setting is possible using the function keys and ten key pads. The keyed-in numerical value may be set as keyed-in or added to the value stored in memory depending on the function key pressed.

The example above shows the case in which the function key [F1] (SET) is used.

(3) Parameters Requiring Power Reapplication

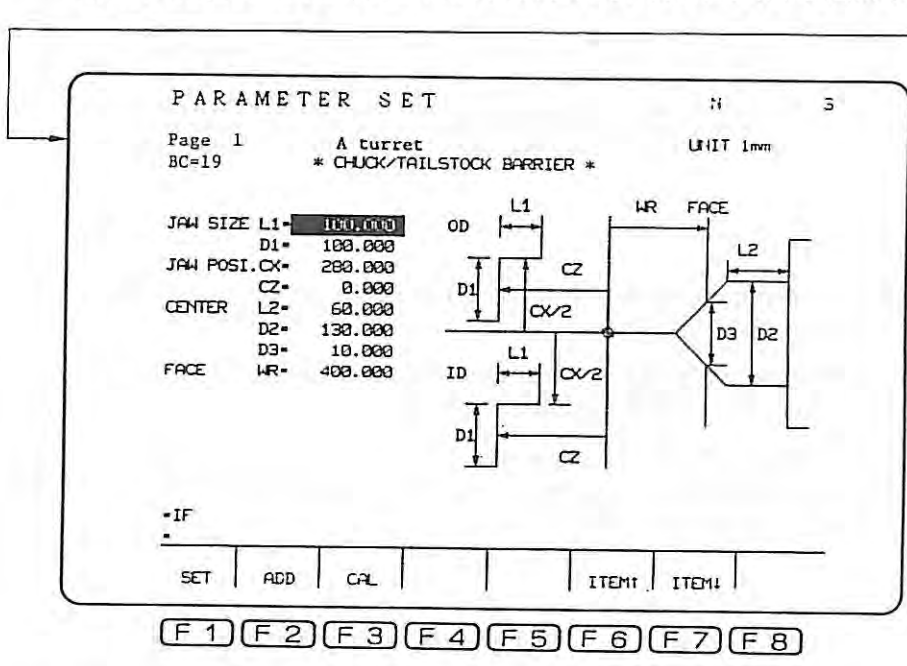
The droop data, the RS232C related parameters, etc. require power supply to be turned off once and then turned on again after setting the parameter data. Such parameters are called the power reapplication parameters.

When these data is set, the screen displays the message **"ALARM-D W01 ;power on effective parameter set"** on the upper area. In this case, check the counter data of the backup counter when the **WRITE** key is pressed after setting the data and turn off power supply when the counter displays the same counter data. For example, if the backup counter data is "15" when the parameter data is set, turn off power supply after making sure that the counter displays the same value "15" again. If power is turned off before the counter shows "15" again, the parameter data might not be set even if power supply is turned on again.

After reapplying power, confirm that the parameters have been changed to the new setting.

Note: *Backup counter*

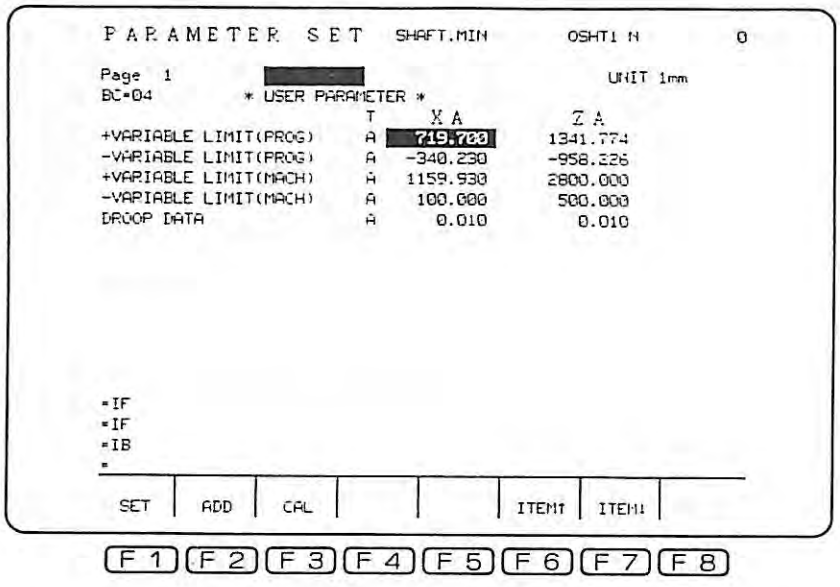
The backup counter data is displayed at the upper left area of the CRT on the TOOL DATA SET, ZERO SET, and PARAMETER SET screens in the form of "BC=__".



Parameters used with OSP5020L/OSP500L-G are explained in details hereafter:

3-1. User Parameter

User parameters are determined depending on the type of workpieces.



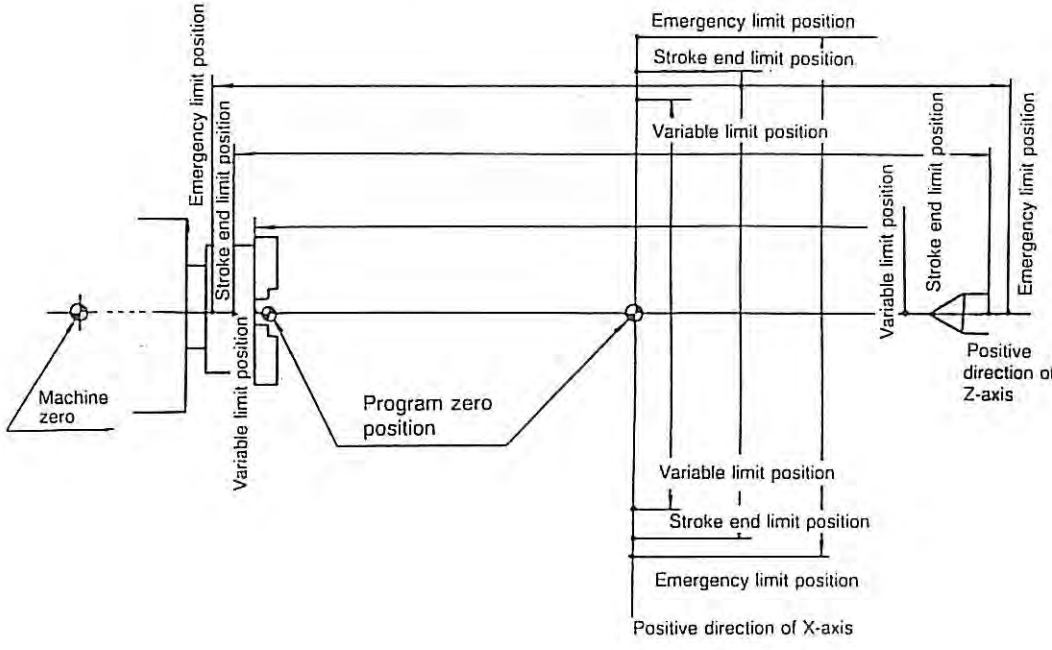
(1) Variable Soft-Limit

Variable soft-limit determines the axis travel range on the X- and Z-axis according to the size of the workpiece to be turned.

This determines the turret indexing positions, and inhibits positioning within a dangerous area to protect the chuck and the tailstock from unexpected collision with the cutting tools.

For the variable soft-limit, two positions are set on the respective axes both in the positive and negative directions. However, they cannot be set outside the stroke end limit positions set by the system parameter (detailed in 3-3.).

Relations of various limit positions set on the machine:



Emergency Limit Position (Hard-Limit Position):

If the limit switch confirming the emergency limit position is tripped, servo power for X- and Z-axis drive is turned OFF and the entire machine operation is stopped.

The limit switch for emergency limit position is not actuated during normal machine operation. However, if this limit is actuated, power supply to the X- and Z-axis drive systems is turned OFF and the machine operation is stopped immediately.

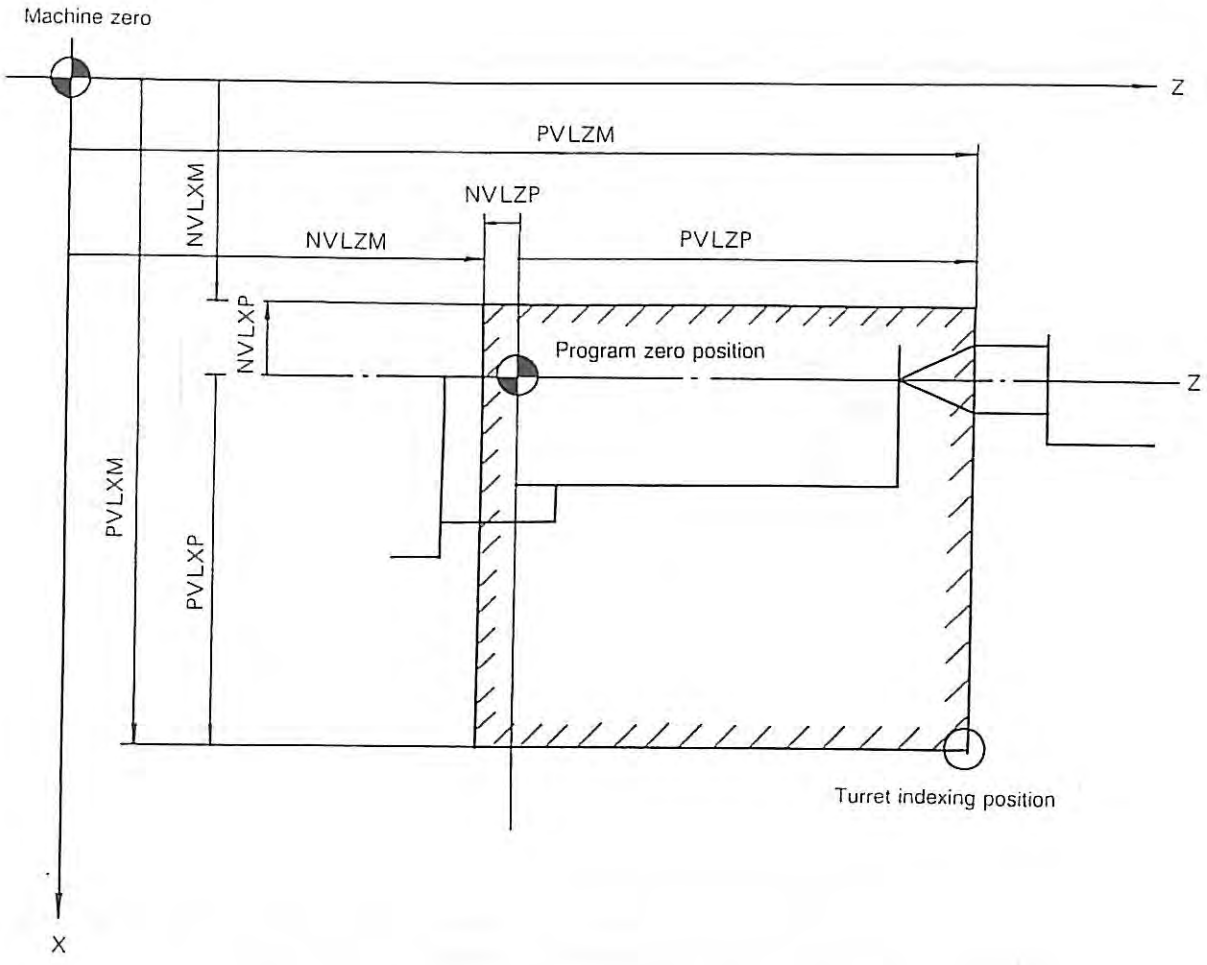
Stroke End Limit Position (Soft-Limit Position):

The stroke end limit positions set the nominal axis travel of each machine. (For the setting procedure, refer to 3-3.)

Variable Soft-Limit:

These limit positions are variable by setting parameters as desired to meet the size of the workpiece to be turned. However, they cannot be set outside the stroke end limit positions.

Turret indexing is carried out at the stroke end limit position in the positive direction.



Axis / CRT Display	X-axis	Z-axis
+ VARIABLE LIMIT (PROG)	PVLXP	PVLZP
− VARIABLE LIMIT (PROG)	NVLXP	NVLZP
+ VARIABLE LIMIT (MACH)	PVLXM	PVLZM
− VARIABLE LIMIT (MACH)	NVLXM	NVLZM

Fig. 4-5 Variable Soft-Limit

【 Coordinate System for Variable Soft-limit 】

As seen in Fig. 4-5, coordinates of the soft-limit positions can be designated both on the coordinate systems having the origin at the machine zero or the program zero.

The relationship between the variable soft-limit expressed on those two different coordinate systems is:

[Coordinates of the variable soft-limit position on the coordinate system of the machine zero] = [Coordinates of the variable soft-limit position on the coordinate system of the program zero] + [Zero offset value]

Therefore, setting the variable soft-limit positions on either of these two coordinate systems can automatically set them on another coordinate system. It is advisable to set them on the coordinate system of the program zero.

There are three methods to set the variable soft-limit positions;

a) Playback method

Use this method when the variable soft-limit positions are unknown.

b) Numerical data entry method

Use this method when the variable soft-limit positions are known.

c) Data renewal method

Use this method to change the currently stored stroke limit position data.

Detailed procedure for the respective methods is explained below:

a) Playback Method – Variable soft-limit positions are unknown

This method is used to set the variable soft-limit positions for the first time, or re-set them after changing the tooling.

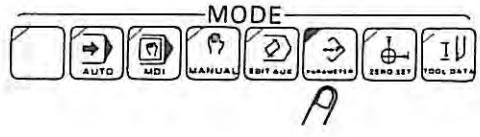
Procedure:

The explanation below is provided assuming the setup of the chuck and tooling, and the setting of various data such as zero offsets and tool offsets are all complete.

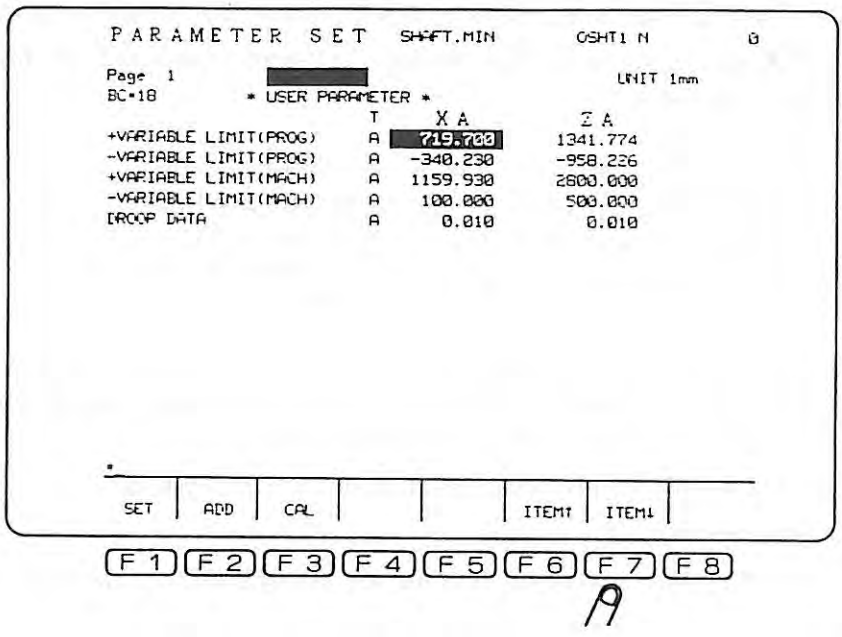
1) Move the axis to the desired variable stroke end position manually, observing the following points:

- To move the turret to the variable stroke end position in the positive direction, index the turret face mounting the longest tool (drill, boring bar, etc.); move the turret at a position where the indexed tool does not interfere with the workpiece or the tailstock.
- To move the turret to the variable stroke end position in the negative direction, index the turret face mounting the standard tool; move the turret at a position where the indexed tool does not interfere with the workpiece and the chuck.
- To set the variable stroke end positions after loading control software or outside the currently set stroke end positions, first set the stroke end positions referring to the data in the OSP CONTROL CARD which is contained inside the OSP box according to the method detailed in b).

2) Select the PARAMETER SET mode by pressing the **PARAMETER** key.

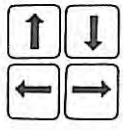


3) If the CRT display is not as shown below, press the function key [F7] (ITEM) until this display appears.

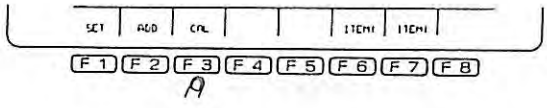


4) Move the cursor to the position where the data is to be set:

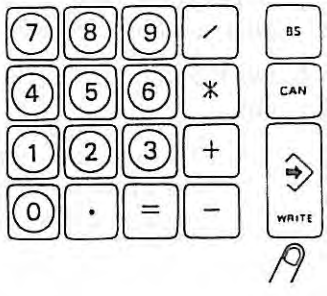
Positive/Negative
X/Z



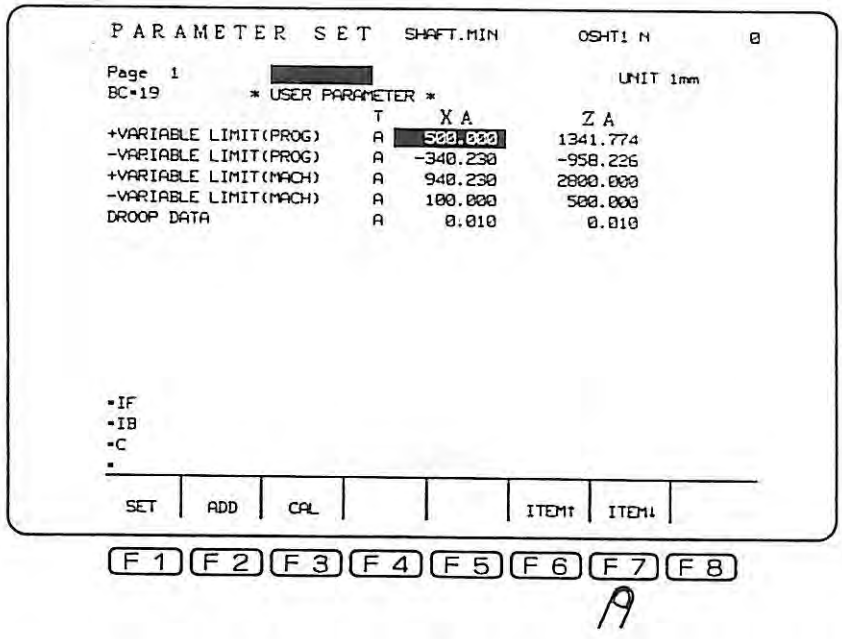
5) Press the function key [F3] (CAL).



6) Press the **WRITE** key.



This sets the current turret position as the variable stroke end position. The CRT displays the newly set data.



9

Note: *When the variable stroke end position is to be set at a point away from the current axis position by a certain distance, enter such distance data through the keyboard after step 6), press the function key [F2](ADD), and then press the* ***WRITE*** *key.*

To set the variable stroke end position at a point −50 mm away from the position where the turret is placed, proceed as:

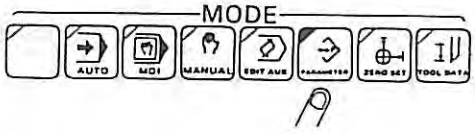
[F2] (ADD), [−][5][0], ***WRITE***

b) Numerical Data Entry Method

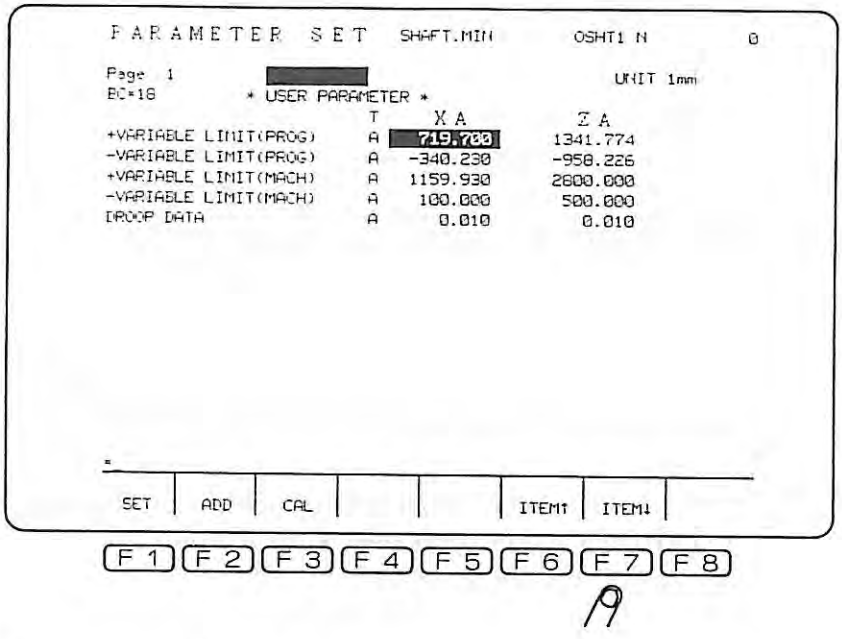
Set the variable stroke end positions when their coordinates are known beforehand:

Procedure:

1) Select the PARAMETER SET mode by pressing the **PARAMETER** key.

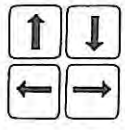


2) If the CRT display is not as shown below, press the function key [F7] (ITEM↓) until this display appears.



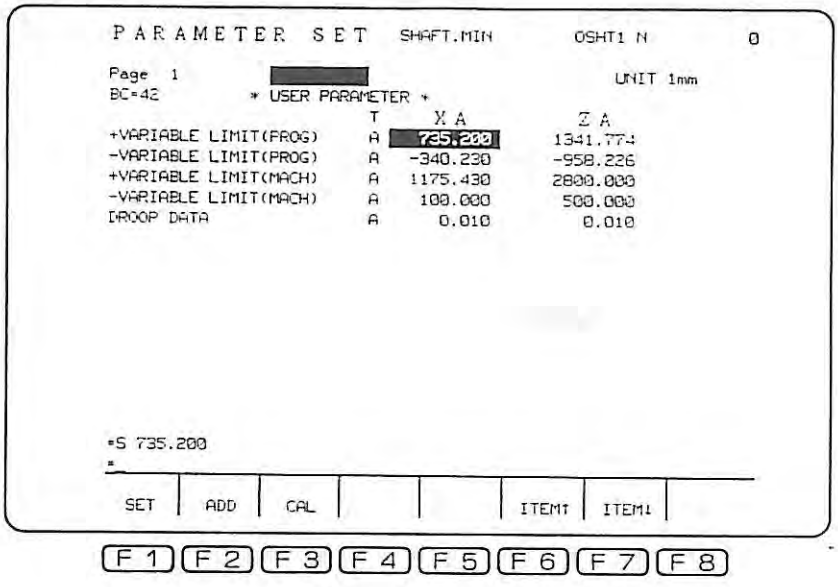
3) Move the cursor to the position where the data is to be set:

Positive/Negative
X/Z

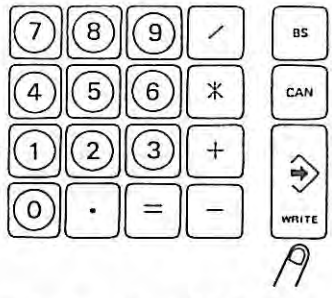


4) Press the function key [F1] (SET) and enter the known variable stroke end position data from the upper digit through the keyboard.

Example: [F1] [7][3][5][.][2][0][0]



5) Press the **WRITE** key.

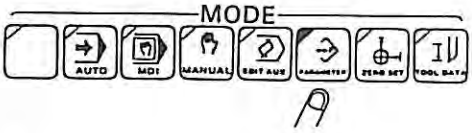


This sets the entered data as the variable stroke end position data and the CRT displays it.

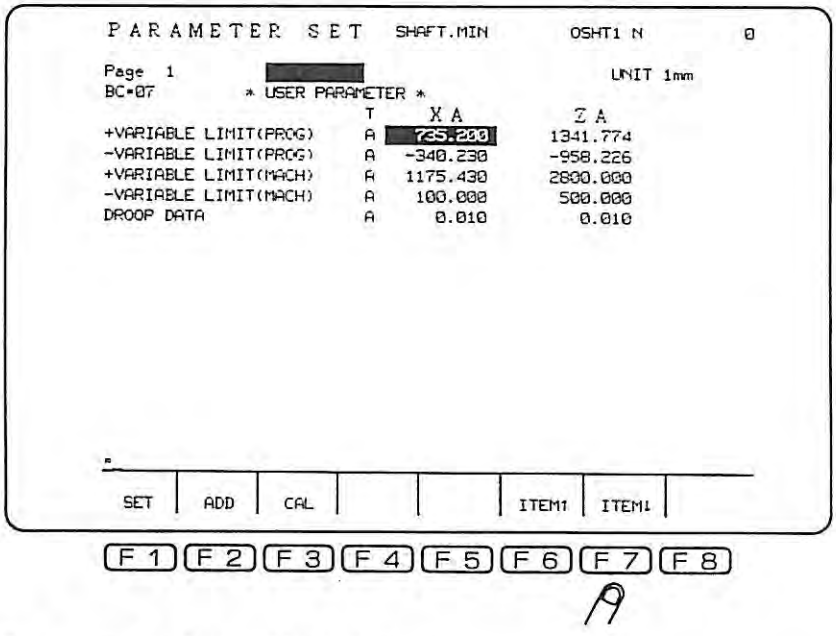
c) Data Renewal Method
This method is used for slightly changing the soft-limit data, which has been set.

Procedure:

1) Select the PARAMETER SET mode by pressing the **PARAMETER** key.

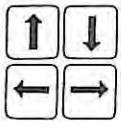


2) If the CRT display is not as shown below, press the function key [F7] (ITEM ↓) until this display appears.



3) Move the cursor to the position where the data is to be set:

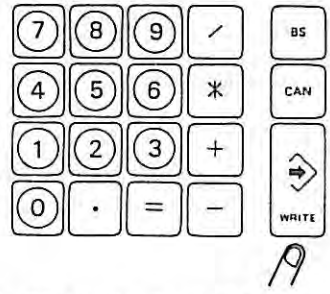
Positive/Negative
X/Z



4) Press the function key [F2] (ADD) and enter the amount to be added.

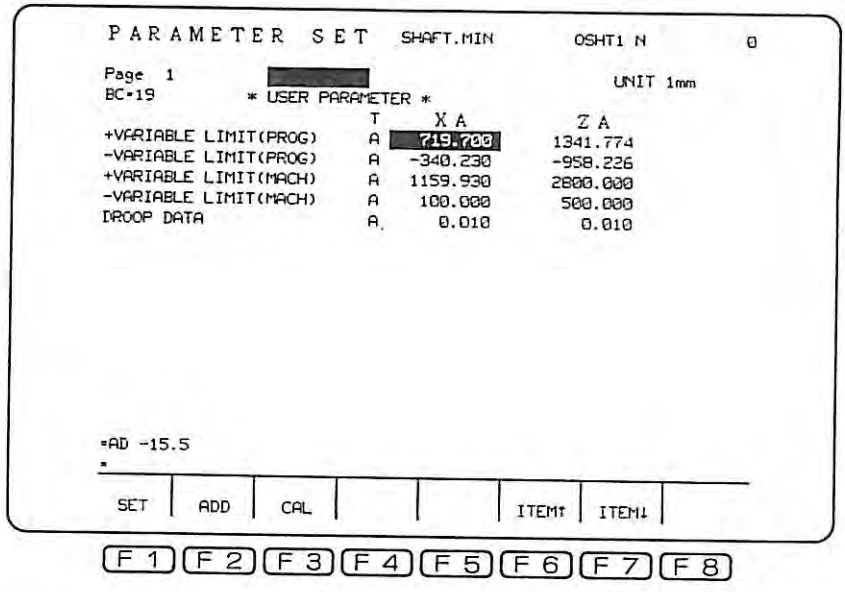
Example: [F2][−][1][5][.][5]

5) Press the **WRITE** key.



This adds "−15.5" to the currently stored data, "735.2" in this example, and the result of the addition, "719.7" is then stored.

735.2 + (−15.5) = 719.7



Note 1: *After carrying out the operations below, set the stroke end limit positions and the variable soft-limit positions without fail. Otherwise the X- and Z-axis do not move at all, or they move only up to a certain point.*

- *Loading of control software No. 3,*
- *Replace of memory board,*
- *Replace of OSP position encoder (X and/or Z)*
  (Setting should be made only on the replaced axis.)

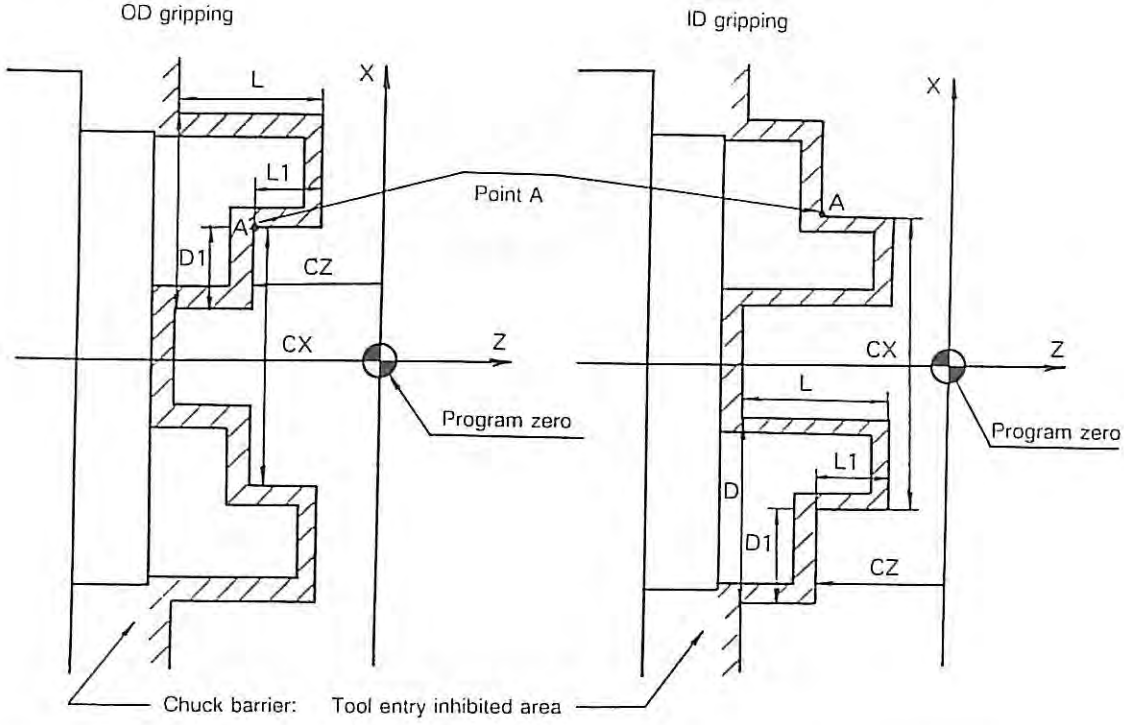
Note 2: *When the stroke end limit position is changed, the variable soft-limit position is also changed to the same position.*

(2) Chuck

Selection of chuck ID/OD mode

a) Chuck barrier

The chuck barrier function can set the inhibited area for tool entry around the chuck, that cannot be set by variable soft-limit position data.
Activation or deactivation of the chuck barrier function can be selected by programming proper M codes. Therefore, tool motion with the chuck barrier function active can be checked when required.



Symbol	Description	Method
L	Chuck jaw length	Optional parameter (long word) No. 18
D	Chuck jaw size	Optional parameter (long word) No. 19
L1	Chuck jaw gripping length	Chuck/tailstock barrier
D1	Chuck jaw gripping face width	Chuck/tailstock barrier
CX	Chuck gripping diameter	Chuck/tailstock barrier
CZ	Distance from programming zero	Chuck/tailstock barrier

Fig. 4-6

[Coordinate System for Chuck Barrier Setting]

The chuck barrier is set using the coordinate system which has an origin at the program zero. Therefore, the chuck barrier is shifted according to the change of program zero which is shifted according to the change of workpiece.

Setting Procedure:

1) The chuck barrier can be set only after the setup and setting of tool offsets are complete.
2) After selecting the manual mode, select the standard tool which has a zero tool offset value.
3) Move the tool to Point A and note down the actual position data display of both X and Z.
4) After selecting the parameter mode, display the CHUCK/ TAILSTOCK BARRIER page by pressing the function key [F7] (ITEM ↓).
5) Move the cursor to JAW POS. CX data, key in the value known in step 3) and press the function key [F1] (SET). After that move it to JAW POS. CZ and enter the value in the same manner as entering JAW POS. CX data. With this,the coordinates of the current turret position are stored in the NC memory.
6) Locate the cursor at JAW SIZE L1 and D1 data position and enter the data using the keyboard and the function key [F1] (SET).

Note 1: *If a point defining the chuck barrier is set outside the variable soft-limit position, the variable soft-limit is effective as the chuck barrier setting point.*

Note 2: *Loading Control Software No. 3 clears the chuck barrier setting point data to zero.*

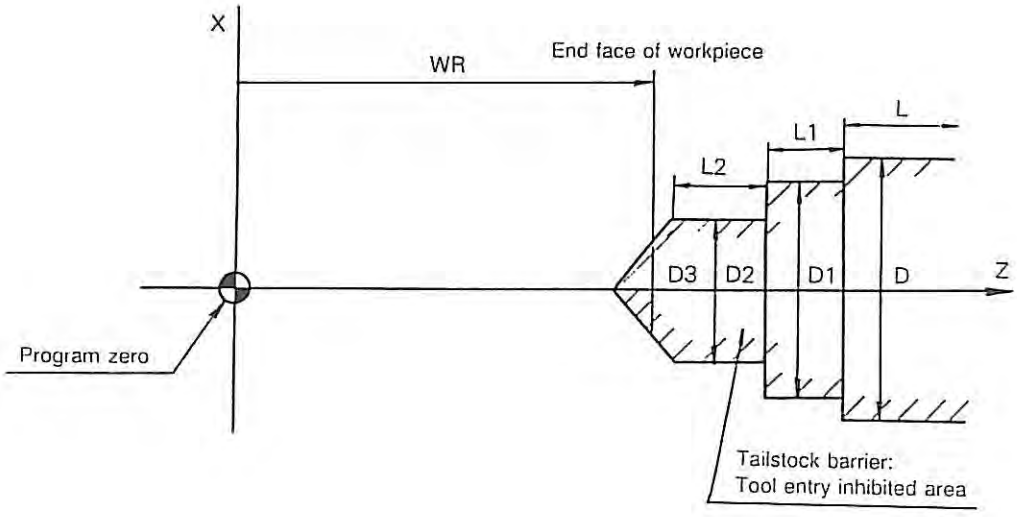
(3) Tailstock

Selection of CHUCK/CENTER WORK mode

a) Tailstock barrier

The tailstock barrier can set the inhibited area for tool entry around the tailstock, that cannot be set by variable soft-limit position data.

Activation or deactivation of the tailstock barrier function can be selected by programming proper M codes. Therefore, tool motion with the tailstock barrier function active can be checked when required.



Symbol	Description	Method
L	Chuck jaw length	Optional parameter (long word) No. 20
D	Tailstock spindle diameter	Optional parameter (long word) No. 21
L1	Tailstock spindle length (1)	Optional parameter (long word) No. 22
D1	Tailstock spindle diameter (1)	Optional parameter (long word) No. 23
L2	Tailstock spindle length (2)	Chuck/tailstock barrier
D2	Tailstock spindle diameter (2)	Chuck/tailstock barrier
D3	Tailstock spindle center diameter	Chuck/tailstock barrier
WR	Tailstock spindle position (Z)	Chuck/tailstock barrier

Fig. 4-7 Tailstock Barrier

Setting Procedure:

1) The tailstock barrier can be set only after the setting of the tailstock and tool offset are completed with the setup including workpiece chucking and turret completed.
2) After selecting the manual mode, select the standard tool which has a zero tool offset value.
3) Move the tool to the end face of the workpiece. Note down the actual position data of Z-axis.
4) After selecting the parameter mode, display the CHUCK/TAILSTOCK BARRIER page by pressing the function key [F7] (ITEM ↓).
5) Move the cursor to WR data and enter the value noted down after pressing the function key [F1] (SET).
6) Set the tailstock barrier data at CENTER L2, CENTER D2, and CENTER D3 by entering the proper data after pressing the function key [F1] (SET).

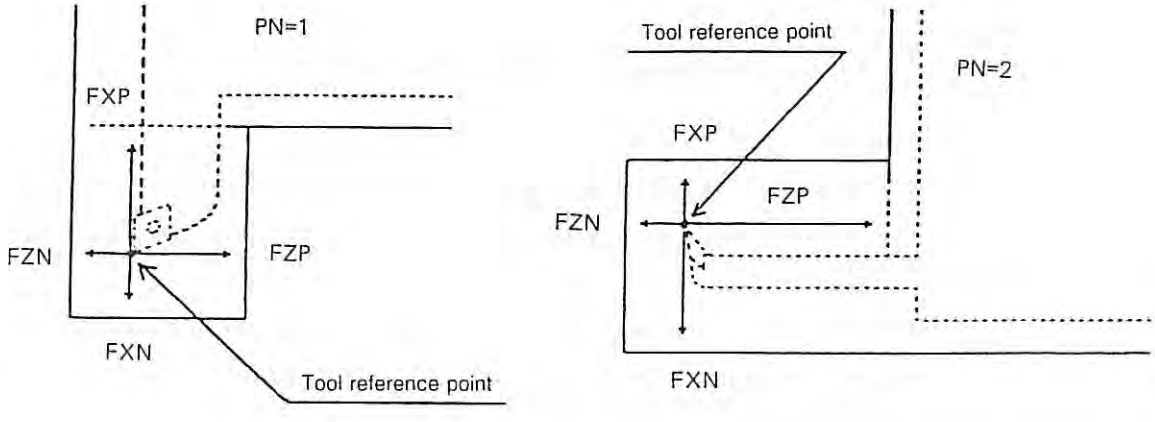
(4) Tool Interference Area (for the graphic specification)

The procedure to set the tool interference area differs depending on whether the graphic specification is selected or not.

Explanation below is given assuming the graphic specification is selected.

No special operations is required to set the tool interference area data since the tool interference area data is registered with the tool shape data. Selection of the tool shape in the TOOL DATA setting mode automatically selects the interference area setting data. However, if the change of the registered data is required, it is possible in the PARAMETER SET mode.

Tool interference area pattern and tool interference area setting data are registered with the tool kind number (They are common to both of turrets A/B). The tool interference area pattern is selected from the OD tool pattern and the ID tool pattern. The tool interference area is defined by four lines. See the figure below.

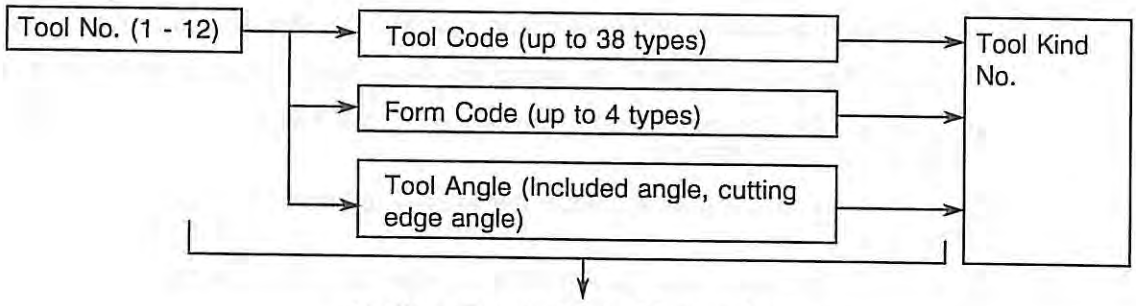


(a) Pattern No. 1 (for OD tools)

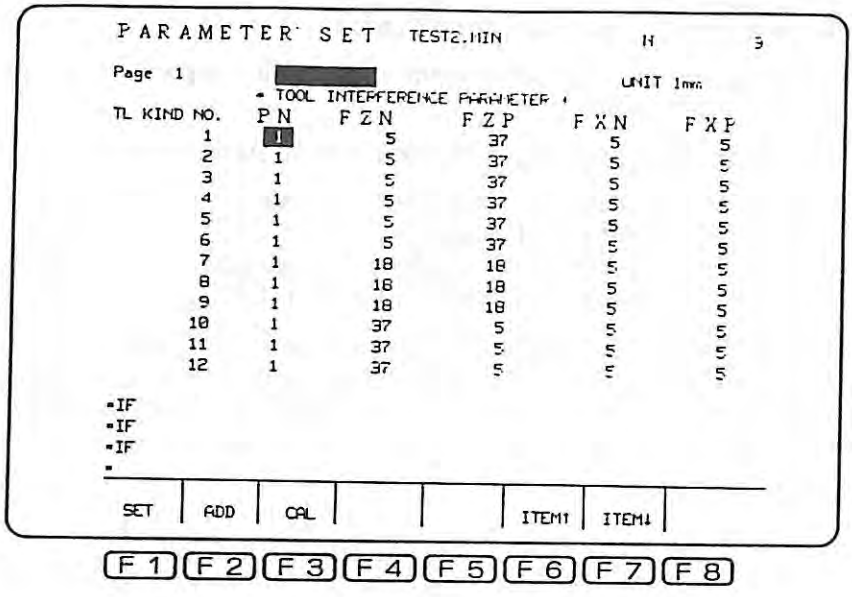
(b) Pattern No. 2 (for ID tools)

Fig. 4-8 Tool Interference Area Pattern and Area Defining Data

The tool kind number is determined from the tool code and form code for individual tool numbers, and the tool angle. For details, refer to Section 7, "Barrier Check Function" in the Programming Manual (Publication No. 3277-E).



These are set in the tool form selection



Setting Procedure:

1) Select the PARAMETER SET mode by pressing the **PARAMETER** key.
2) Press the function key [F7] (ITEM↓) to display the *TOOL INTERFERENCE PARAMETER* page.
3) Press the  or  key to display the page which contains the desired tool kind number. One page contains 12 tool numbers and a total of five pages are available (1 to 12, 13 to 24, 25 to 36, 37 to 48 and 49 to 60).
4) Use the  or  key to place the cursor on a desired tool kind number.
5) Use the  or  key to place the cursor on the data to be changed.
6) Press the function key [F1] (SET) and key in the desired data.
7) Press the **WRITE** key.

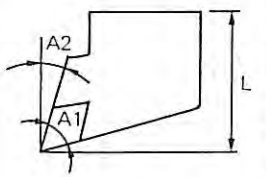
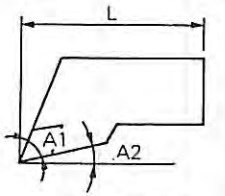
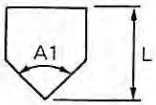
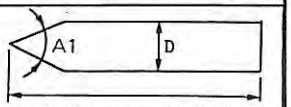
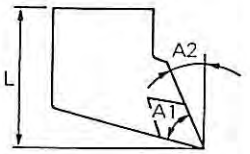
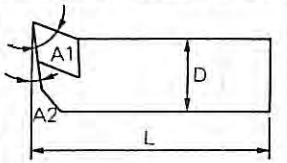
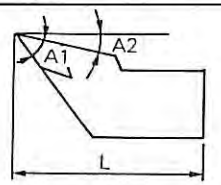
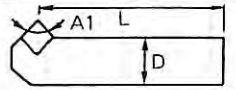
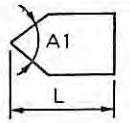
Repeat steps 3) through 4) to change the tool interference area setting data. In this operation, A and B TURRET selection may be ignored since the data is registered with the tool kind number.

Table 4-1 Tool Kind Number Selection Chart by Tool Code and Form Code

Tool Code (1-26) \ Form Code (1-4)		1	2	3	4
1	ROUGH OD ←	(1),2,3,4,5,6	(7),8,9	(22),23,24,25,26,27	
2	ROUGH ID ←	(36),37,38,39,40,41	(42)	(46),47,48	
3	ROUGH FACE ↓	(1),2,3,4,5,6	(22),23,24,25,26,27	(49),50,51	
4	ROUGH OD →	(10),11,12,13,14,15	(7),8,9		
5	ROUGH ID →	(43)	(46),47,48		
6	ROUGH FACE ↑	(36),37,38,39,40,41	(16),17,18,19,20,21	(49),50,51	
7	FINISH OD ←	(1),2,3,4,5,6	(7),8,9	(22),23,24,25,26,27	
8	FINISH ID ←	(36),38,39,40,41	(42)	(46),47,48	
9	FINISH FACE ↓	(1),2,3,4,5,6	(22),23,24,25,26,27		
10	FINISH OD →	(10),11,12,13,14,15	(7),8,9		
11	FINISH ID →	(43)	(46),47,48		
12	FINISH FACE ↑	(36),37,38,39,40,41	(16),17,18,19,20,21	(49),50,51	
13	THREAD OD ←	(30)	(7),8,9		
14	THREAD ID ←	(44)			
15	THREAD FACE ↓	(31)	(49),50,51		
16	THREAD OD →	(30)	(7),8,9		
17	THREAD ID →	(44)			
18	THREAD FACE ↑	(31)	(49),50,51		
19	GROVE OD ↓	(28)	(7),8,9	(30)	
20	GROVE ID ↑	(45)	(44)		
21	GROVE FACE ←	(29)	(31)		
22	DRILL, HSS	(33),34,35			
23	DRILL, CARBIDE	(33),34,35			
24	DRILL, CENTER	(32)			
25	RECESS OD	(1),2,3,4,5,6	(22),23,24,25,26,27		
26	RECESS ID	(36),37,38,39,40,41			

Numbers in ( ) in the table above are the tool kind number displayed as a default on the guidance page for the tool form selection in TOOL DATA setting operations.

Table 4-2 Tool Kind Number Selection Chart by Tool Angle

Tool Kind No.	Conditions	Remarks	Tool Kind No.	Conditions	Remarks
1	78° ≦ A1 ≦ 90°	 A1 : Tool angle A2 : Edge angle L : Sticking out	22	78° ≦ A1 ≦ 90°	 A1 : Tool angle A2 : Edge angle L : Sticking out
2	60° ≦ A1 < 78°		23	60° ≦ A1 < 78°	
3	54° ≦ A1 < 60°, 3° < A2 < 90°		24	54° ≦ A1 < 60°, 3° < A2 < 90°	
4	54° ≦ A1 < 60°, 0° ≦ A2 ≦ 3°		25	54° ≦ A1 < 60°, 0° ≦ A2 ≦ 3°	
5	0° < A1 < 54°, 3° < A2 < 90°		26	0° < A1 < 54°, 3° < A2 < 90°	
6	0° < A1 < 54°, 0° ≦ A2 ≦ 3°		27	0° < A1 < 54°, 0° ≦ A2 ≦ 3°	
7	60° ≦ A1 < 180°	 A1 : Tool angle L : Sticking out	33	120° < A1 ≦ 180°	 A1 : Tool angle D : Drill diameter L : Sticking out
8	54° ≦ A1 < 60°		34	90° < A1 ≦ 120°	
9	0° < A1 < 54°		35	0° < A1 ≦ 90°	
10	78° ≦ A1 ≦ 90°	 A1 : Tool angle A2 : Edge angle L : Sticking out	36	78° ≦ A1 ≦ 90°	 A1 : Tool angle A2 : Edge angle L : Holder length D : Holder dia.
11	60° ≦ A1 < 78°		37	60° ≦A1 < 78°	
12	54° ≦ A1 < 60°, 3° < A2 < 90°		38	54° ≦ A1 < 60°, 3° < A2 < 90°	
13	54° ≦ A1 < 60°, 0° ≦ A2 ≦ 3°		39	54° ≦ A1 < 60°, 0° ≦ A2 ≦ 3°	
14	0° < A1 < 54°, 3° < A2 < 90°		40	0° < A1 < 54°, 3° < A2 < 90°	
15	0° < A1 < 54°, 0° ≦ A2 ≦ 3°		41	0° < A1 < 54°, 0° ≦ A2 ≦ 3°	
16	78° ≦ A1 ≦ 90°	 A1 : Tool angle A2 : Edge angle L : Sticking out	46	60° ≦ A1 < 180°	 A1 : Tool angle D : Holder dia. L : Holder length
17	60° ≦ A1 < 78°		47	54° ≦ A1 < 60°	
18	54° ≦ A1 < 60°, 3° < A2 < 90°		48	0° < A1 < 54°	
19	54° ≦ A1 < 60°, 0° < A2 < 3°		49	60° ≦ A1 < 180°	 A1 : Tool angle L : Sticking out
20	0° < A1 < 54°, 3° < A2 < 90°		50	54° ≦ A1 < 60°	
21	0° < A1 < 54°, 0° ≦ A2 ≦ 3°		51	0° < A1 < 54°	

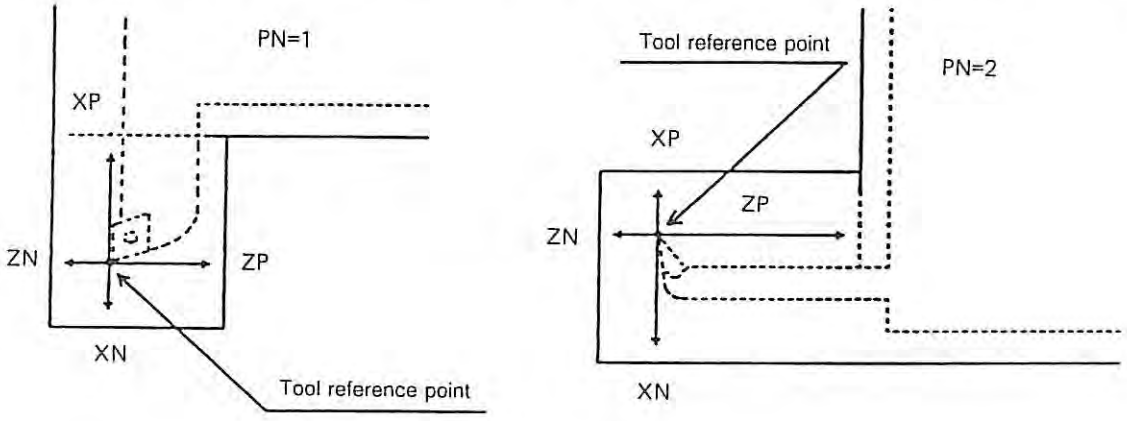
(5) Tool Interference Area (for the character specification)

The procedure to set the tool interference area differs depending on whether the graphic specification is selected or not.

Explained below is the procedure for the character display specification.

The data to define the tool interference area is set in the TOOL DATA SET mode.

Enter the tool interference area pattern and tool interference area setting data for each of the tools (Enter for each of turrets A/B). The tool interference area pattern is selected from the OD and ID tool patterns. The tool interference area is defined by four lines. See the figure below.



(a) Pattern No. 1 (for OD tools)

(b) Pattern No. 2 (for ID tools)

Fig. 4-9 Tool Interference Area Pattern and Area Defining Data

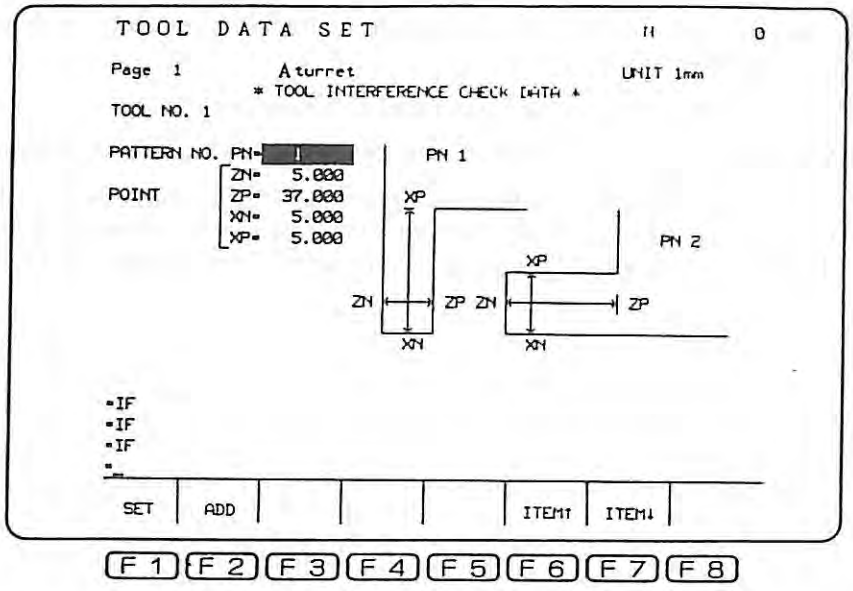
PN: Tool interference area pattern selection number
1 = for OD tools
2 = for ID tools

ZN: Position of the area defining line in the negative direction of Z-axis in reference to the tool reference point

ZP: Position of the area defining line in the positive direction of Z-axis in reference to the tool reference point

XN: Position of the area defining line in the negative direction of X-axis in reference to the tool reference point

XP: Position of the area defining line in the positive direction of X-axis in reference to the tool reference point



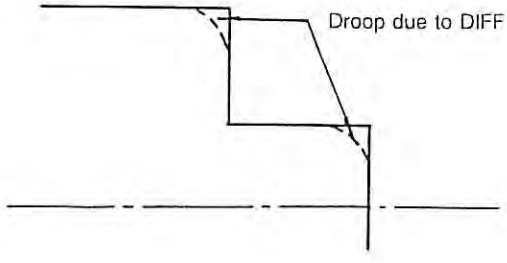
Setting Procedure:

1) Select the TOOL DATA SET mode by pressing the **TOOL DATA** key.
2) Press the function key [F7] (ITEM ↓) to display the *TOOL INTERFERENCE CHECK DATA* page. See the CRT above.
3) Press the key to select turret A.
4) Press the or key to display the page listing the tool number for which the tool interference area is to be set.
5) Press the or key to designate the data to be set.
6) Press the function key [F1] (SET) and key in the desired data.
7) Press the **WRITE** key.

Repeat steps 3) through 7) to set the necessary tool interference area setting data.
For the tools on turret B, set the data in the similar manner.

(6) Droop

Any servo system basically has a servo error, delay (DIFF) to the function generation. Especially in high-speed cutting, droop due to servo error (DIFF) results when a corner is cut. Such droop amount can be decreased by properly designating commands and also by setting the required parametric data.



Setting Procedure:

1) After pressing the **PARAMETER** key, select the USER PARAMETER page on the CRT.
2) After moving the cursor to "Droop - X", set the desired droop amount and press the function key [F1] (SET). Then set the desired amount on Z-axis after moving the cursor to "Droop - Z", and press the function key [F1].

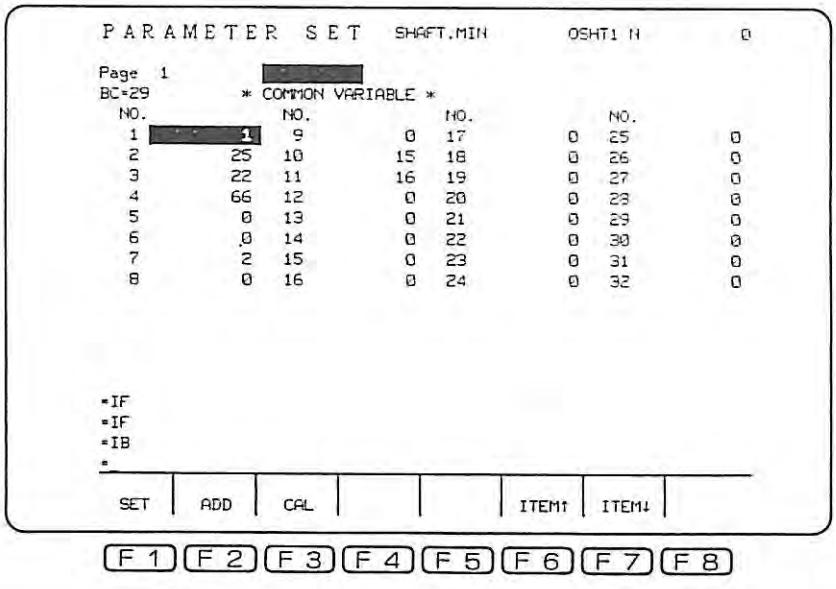
   The larger the set amount is, the larger the droop amount becomes.

Note: *Loading Control Software No. 3 sets droop amount to 10 μm.*

3-2. Common Variables

The OSP5020L/OSP500L-G can use 32 common variables, V1 through V32, which can be used both for turrets A and B, The use of these variables can be determined by the user independently of the system.

Procedure:

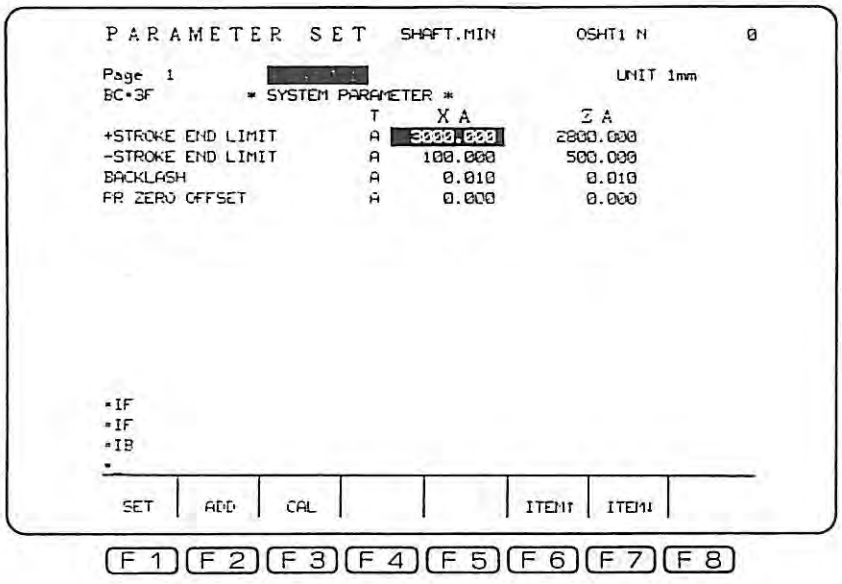


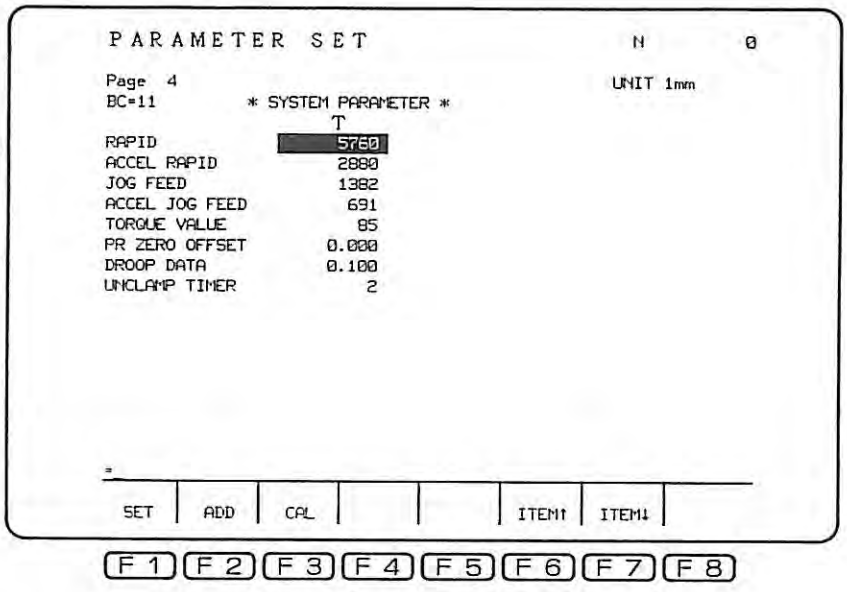
1) After pressing the **PARAMETER** key, select the COMMON VARIABLE page on the CRT.
2) Move the cursor to the position to be set the data or changed the variable No.
3) Set the required data using the function keys [F1] (SET) or [F2] (ADD).

Note: *Since the data for common variables is stored as data with floating zero, display right to the decimal point may not be the same as set due to rounding.*

3-3. System Parameter

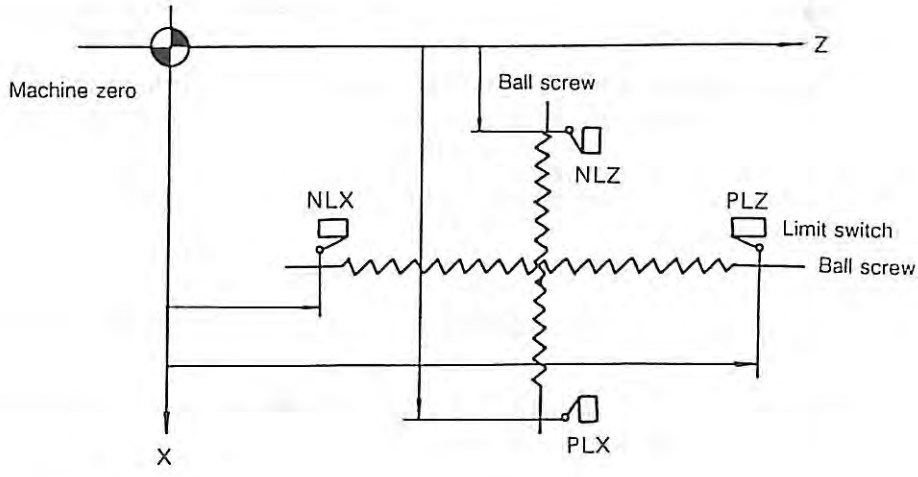
System parameter are the parameters determined by the data specific to the respective machines. Data should be set for turrets A and B, independently.





(1) Stroke End Limit

The stroke end limit determines the limit of axis travel. With this, the ball screw is protected from overrun. However, the machine is equipped with stroke end protective limit switches, which mechanically protect the machine; if any of these limit switches is actuated, the power to the machine, with the exception of to the control, is shut off, and machine operation is brought to a complete stop. Set the stroke end positions inside the area determined by those limit switches.



Axis / CRT Display	X-axis	Z-axis
+ STROKE END LIMIT	PLX	PLZ
− STROKE END LIMIT	NLX	NLZ

Fig. 4-10

The stroke end limit positions are factory-set and these data are recorded on the OSP CONTROL CARD.

NEVER CHANGE THESE DATA AFTER INSTALLATION.

Setting Procedure:

1) After pressing the **PARAMETER** key, select the SYSTEM PARAMETER page on the CRT.
2) After moving the cursor to "+STROKE END LIMIT - XA", set the data recorded on the OSP CONTROL CARD using the function key [F1] (SET).
3) Set the stroke end limit data of −X, +Z and −Z in the same manner.
4) If the stroke end limit data is unknown, record the actual position data of the turret, and set it as the stroke end data for both the negative and positive directions on respective axes.
5) Actually measure the available axis travel from the current turret position in both the positive and negative directions.
6) Set the measured axis travel in the positive direction using the function key [F2] (ADD): the data must be positive.

   Set the measured axis travel in the negative direction using the function key [F2] (ADD): the data must be negative.

(2) Backlash

Backlash elimination function serves to compensate for lost motion on each axis when the axis feed direction is reversed from the negative to the positive direction.

That is, the set backlash compensation value is subtracted from the value read by the OSP position encoder when axis motion is reversed from negative to positive, and the result of the subtraction is handled as the true position data.

The backlash compensation data is set before delivery; however, it can be changed if desired using a parameter.

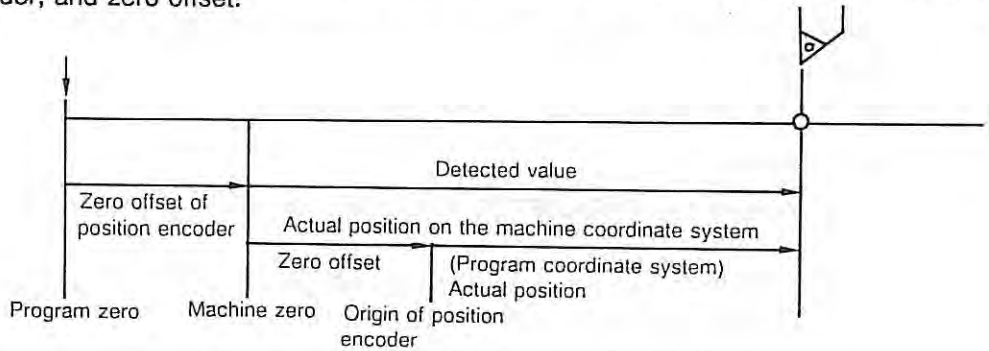
Procedure:

1) After pressing the **PARAMETER** key, select the SYSTEM PARAMETER page on the CRT.
2) Move the cursor to "BACKLASH - XA", and set the desired backlash compensation data using the function key [F1] (SET).
3) Set the backlash compensation data of Z-axis in the same manner.

Note: Loading Control Software No. 3 sets the backlash compensation data at "10 μm". (15 m for X-axis of the LC10)

(3) Zero Offset of Position Encoder

Zero offset of the position encoder is to compensate the current location which has not been compensated (to be referred to as detected value hereinafter) to make it into the actual position on the machine coordinate system. The diagram below expresses the relationship between the origin of the position encoder, machine zero, program zero, offset of position encoder, and zero offset.



Note: Origin of the position encoder indicates the position when the value of the position encoder is 0.

Since the value of the machine zero can be set arbitrarily by the position encoder zero offset, this function is effective in the following cases.

When the position encoder is changed:

Set the offset value of the position encoder zero position so that the position of machine zero does not change before and after the position encoder is changed. Resetting the position encoder offset value permits the values that are set in relation with the machine zero such as travel end limit, variable soft-limit and zero offset to be used without resetting.

When the detected value exceeds 8 digits in decimal numbers within the machine travel range (movable range):

This can take place depending on machine models. However, it is possible to set the machine zero so that the output value from the position encoder does not exceed 8 digits in decimal numbers by taking advantage of position encoder zero offset.

Procedure:

1) After pressing the **PARAMETER** key, select the SYSTEM VARIABLE page on the CRT.
2) Move the cursor to the X (Y) of the position encoder offset.
3) Set the data by pressing the function key [F1] (SET) (or [F2] (ADD) or [F3] (CAL)).

Note 1: *0 is set for both X and Z right after control software loading.*

Note 2: *Wait for more than 3 minutes after data setting before turning off and then on power again.*

Note 3: *When the CAL function is used, the input data should indicate where the present position should be located on the machine coordinate system.*

Note 4: *Setting range is within 9 digits in decimal numbers and the negative values cannot be set.*

Note 5: *When no setting is conducted (setting value is 0), the position encoder origin and the machine zero share the same point.*

Note 6: *No setting can be carried out for the C-axis and W-axis.*

(4) Unit Amount and Acceleration Unit Amount

Special attention must be paid when changing or setting the unit amount or acceleration unit amount since they are fundamental amount in axis control.

What is "unit amount"?

On the OSP5020L/OSP500L-G, axis motion is controlled as the control calculates RCON every 12.8 msec. The change of variation amount in the repetitive calculation is called "unit amount".

For feed axes, the feed unit amount is expressed as follows:

Feed unit amount (μm/12.8 msec.)

$$= \frac{X \text{ m/min} \times 10^6}{60 \times 10^3 \text{ msec}} \times 12.8 \text{ msec}$$

where, X: feedrate (m/min)

For example, when the feedrate is 10 m/min, the unit amount is 2,134 μm/12.8 msec.

For the turret indexing axis (T-axis), the change of variation amount is set in units of feed unit amount (0.001°/12.8 ms).

For example, with the LB15 Ⅱ, the relation between the rapid feed unit amount and the turret indexing speed is as indicated below:

Rapid Feed Unit Amount	Turret Indexing Speed
5,760	75 rpm
1,382	20 rpm

This setting is effective only for the NC turret.

What is "acceleration unit amount"?

On the OSP5020L/OSP500L-G, automatic acceleration and deceleration is available for starting and ending rapid feed or manual feed. Acceleration or deceleration means the change of unit amount described above.

While the axis is at a still, the unit amount is "zero". However, when feeding is started, the unit amount increases every 12.8 msec until it reaches the feed unit amount. The variation of the unit amount is called "acceleration unit amount".

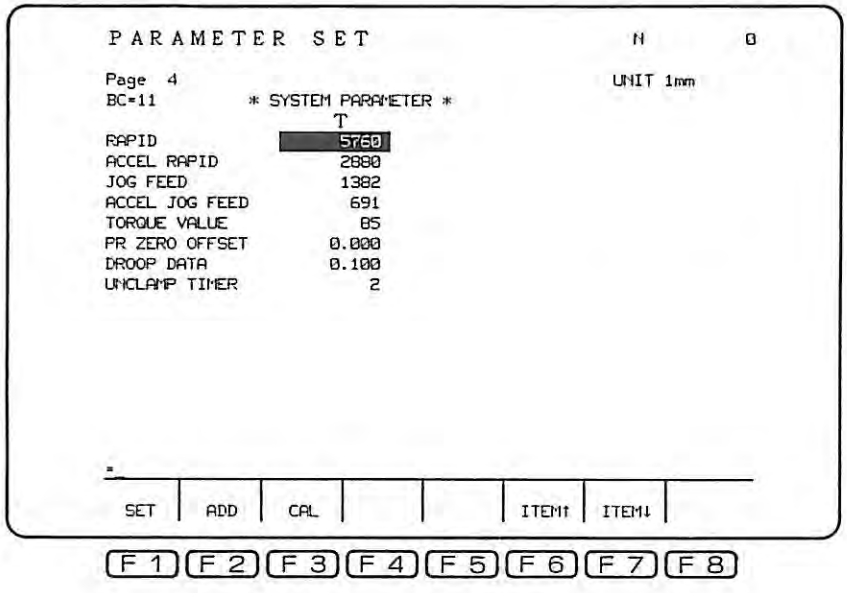
The time required for acceleration and deceleration for feeding can be calculated from the equation below:

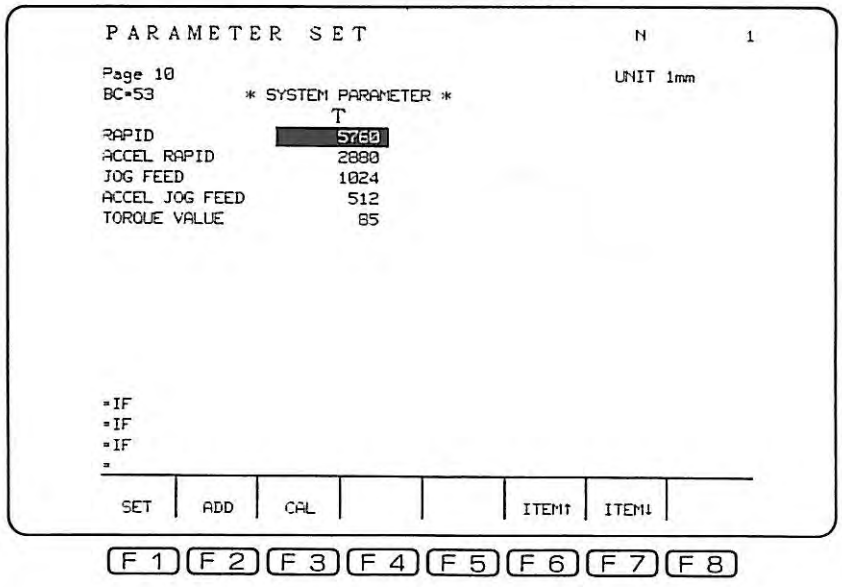
Acceleration/deceleration time (msec)

$$= \frac{\text{Feed unit amount}}{\text{Feed acceleration unit amount}} \times 12.8 \text{ msec}$$

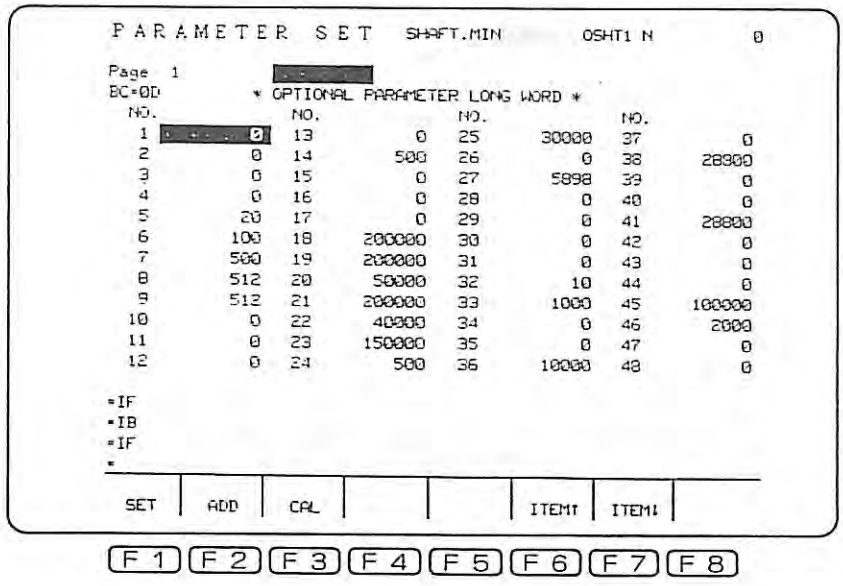
CAUTION

If the acceleration unit amount is changed to a value larger than standard one, it could adversely affect motor and machine performance. Therefore, be sure to contact Okuma when a change of the acceleration unit amount is desired.




PARAMETER SET
N 1
Page 10
UNIT 1mm
BC=53 * SYSTEM PARAMETER *
T
RAPID 5760
ACCEL RAPID 2880
JOG FEED 1024
ACCEL JOG FEED 512
TORQUE VALUE 85
=IF
=IF
=IF
=
SET
ADD
CAL
ITEM↑
ITEM↓
F1 F2 F3 F4 F5 F6 F7 F8

3-4. Optional Parameter (Long Word)



Optional parameter (long word) indicates the fundamental amounts.

Loading Control Software No. 3 sets the standard data. If the data needs to be changed, proceed very carefully since it is a very delicate element of NC operation.

Standard values of the optional parameter (long word) are indicated below:

( ) : 0.1μm or 0.00001 in. specification

Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
1					
2					
3					
4					
5	Allowable error in circular interpolation	μm	20 (200)	200 (2000)	2
6	Relieving amount in LAP - bar turning	μm	100 (1000)	1000 (10000)	1
7	Pecking amount in grooving in "mm/min feed" mode and drilling cycle	μm	500 (5000)	10000 (100000)	1
8	Unit amount of chamfering in thread cutting cycle	μm/12.8 msec	512 (1867)	1024 (5120)	21 (105)
9	Unit amount in dry run	μm/12.8 msec	512 (1867)	1024 (5120)	21 (105)

( ) : 0.1μm or 0.00001 in. specification

Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
10					
11					
12					
13					
14	Pecking amount in drilling cycle for multi-machining model	μm	500 (5000)	10000 (100000)	1
15					
16					
17					
18	Jaw dimension L	μm	200000 (2000000)	9999999 (99999999)	1000 (10000)
19	Jaw dimension D	μm	200000 (2000000)	9999999 (99999999)	1000 (10000)
20	Tailstock dimension L	μm	50000 (500000)	9999999 (99999999)	1000 (10000)
21	Tailstock dimension D	μm	200000 (2000000)	9999999 (99999999)	1000 (10000)
22	Tailstock dimension L1	μm	40000 (400000)	9999999 (99999999)	1000 (10000)
23	Tailstock dimension D1	μm	150000 (1500000)	9999999 (99999999)	1000 (10000)
24	Sensor protection distance	μm	500 (5000)	30000 (300000)	0
25	Allowable distance of touch setter for measurement	μm	30000 (300000)	9999999 (99999999)	0
26					
27					
28	ATC second barrier range, XA-axis (LR15M-ATC)	μm	0	9999999	0
29	ATC second barrier range, ZA-axis (LR15M-ATC)	μm	0	9999999	0
30	ATC first barrier range, XA-axis (LR15M-ATC)	μm	0	9999999	0
31	ATC first barrier range, ZA-axis (LR15M-ATC)	μm	0	9999999	0
32	Tool offset, nose R compensation Fixed amount addition/ deduction	μm (1/1000 in.)	0	1000 (10000)	0
33	Tool offset, nose R compensation and size catcher Limit value for addition	μm	1000 (10000)	1000 (10000)	0

( ) : 0.1μm or 0.00001 in. specification

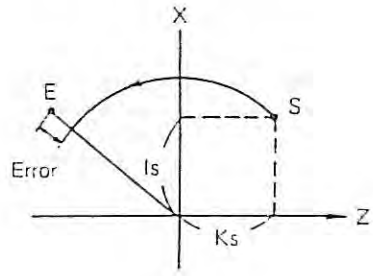
Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
34	ATC barrier range, XB-axis (LR15M-ATC)	μm	0	9999999 (99999999)	0
35	ATC barrier range, ZB-axis (LR15M-ATC)	μm	0	9999999 (99999999)	0
36	2nd spindle zero shift (set automatically with the chuck closed in the synchronous mode)	0.001°	0	359999 (3599999)	0
37					
38					
39					
40	2nd spindle orientation zero offset	0.001°	0	359999 (3599999)	0
41	2nd spindle acceleration unit amount	0.001°/ 12.8 msec²	9600	57600 (576000)	288 (2880)
42	Sub spindle backlash	0.001°	0	1000 (10000)	−1000 (−10000)
43	2nd spindle zero offset	0.001°	0	359999 (3599999)	0
44	Spindle orientation zero offset (for the high precision VAC)	0.001°	0	359999 (3599999)	0
45	Interference allowance between ZB-axis and W-axis	μm	0	99999999	0
46	LAP clearance amount (LAP 4 specification)	μm	2000	99999999	0
47					
48					
49	A side chuck barrier DS (FTL)	μm	0	9999999 (99999999)	0
50	B side chuck barrier DS (FTL-III)	μm	0	9999999 (99999999)	0
51	B side chuck barrier L (FTL-III)	μm	200000 (2000000)	9999999 (99999999)	1000 (10000)
52	B side chuck barrier D (FTL-III)	μm	200000 (2000000)	9999999 (99999999)	1000 (10000)
53	Interference-free distance PD (FTL-III)	μm	9999999 (99999999)	9999999 (99999999)	40000 (400000)
54	A side Z-axis limit position (Interference limit with respect to the turret) (FTL)	μm	0	99999999 (999999999)	0
55	B side Z-axis limit position (Interference limit with respect to the turret) (FTL-III)	μm	0	99999999 (999999999)	0

Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
56					
57					
58					
59					
60	Interference-free distance (FTL)	μm	0	99999999	0
61	Unloading position (FTL)	μm	0	99999999	0
62	Loading position (FTL)	μm	0	99999999	0
63	Robot answer area (FTL)	μm	0	99999999	0
64	Simple tow-along positive limit	μm	0	99999999	0
65	W-axis designated position	μm	0	99999999	0
66	No work confirmation timer	msec	50	999999	0
67					
68					
69					
70					
71					
72					
73					
74					
75					
76					
77					
78					
79					
80					
81					
82					
83					
84					
85					
86					
87					
88					
89					
90					

<Meaning of Optional Parameter (Long Word)>

Parameter #5	Allowable error in circular interpolation

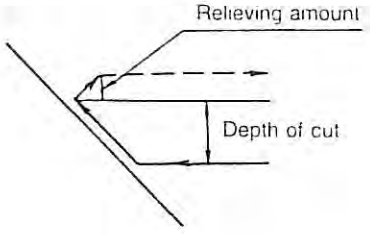
When commanding an arc, its center is dimensioned with reference to the arc starting point (S) using I and K words (Is, Ks). The programmed coordinates of the end point (E) might not correctly lie on the arc generated by the circular interpolation commands due to an error in the commanded values. The allowable error amount is indicated by parameter.



Standard setting is 20 μm.

Parameter #6	Relieving amount in LAP - bar turning

In bar turning operation in the LAP, the axes, after the completion of cutting with the specified depth of cut, shift away from the workpiece by the set relieving amount and then move to the starting point for next cutting. This amount is set with a parameter. Both the X- and the Z-axis relieve the same amount from the workpiece.



Standard setting is 0.1 mm.

Parameter #7	Pecking amount in grooving in "mm/min feed" mode and drilling cycle

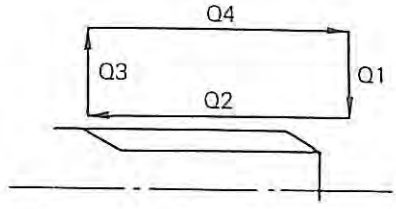
With the compound drilling and grooving fixed cycle, if pecking amount is not designated in the program, the amount set in this parameter is taken as the pecking amount.

Standard setting is 500 μm.

Parameter #8 Unit amount of chamfering in thread cutting cycle

In thread cutting cycle, thread cutting is performed in path Q2 and chamfering is performed in path Q3. Unit amount in path Q3 is set as a parametric data.

Standard setting is 512 μm/12.8 msec and feedrate in chamfering with this setting is 2.4 m/min.



Parameter #9 Unit amount in dry run

With the dry run function activated, all feed commands in a program are executed in the same feedrate. The value set in parameter #9 is the unit amount for dry run operation. Standard setting is 512 μm/12.8 msec and the feedrate with this setting is 2.4 m/min.

Parameter #14 Pecking amount in drilling cycle for multi-machining model

This sets the tool pecking amount in the grooving and drilling cycles.
Standard setting is 500 μm.

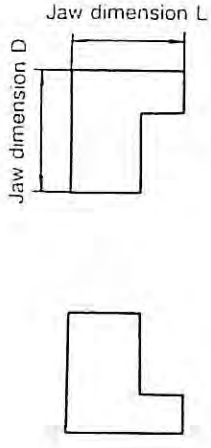
Parameter #18 - #23

Chuck and tailstock dimensions

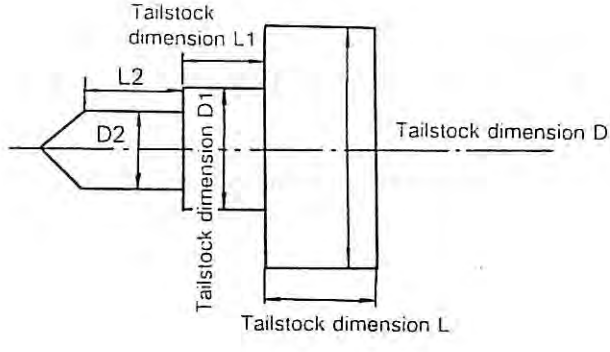
Chuck and tailstock dimensions are set on the display (graphic animated display). The dimensions set on this page are used as the data for establishing the chuck and tailstock barrier.

Parameter #18 ··· Jaw dimension L (200000 μm)
Parameter #19 ··· Jaw dimension D (200000 μm)
Parameter #20 ··· Tailstock dimension L (50000 μm)
Parameter #21 ··· Tailstock dimension D (200000 μm)
Parameter #22 ··· Tailstock dimension L1 (40000 μm)
Parameter #23 ··· Tailstock dimension D1 (150000 μm)

Values in parentheses are standard setting values.



Chuck



Tailstock

Parameter #24	Sensor protection distance

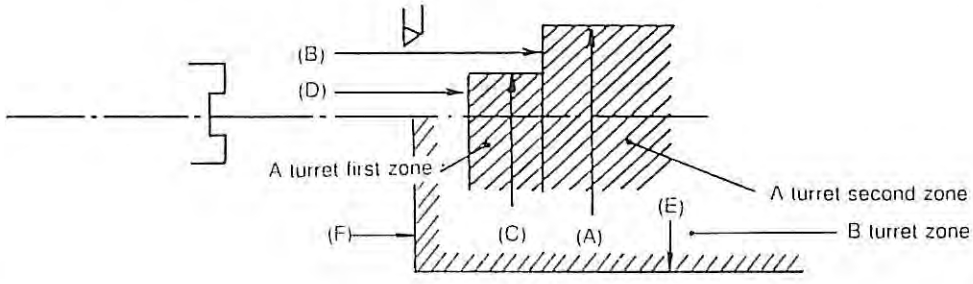
Movable distance after the contact with the sensor
Standard setting is 0.5 mm.

Parameter #25	Allowable distance of touch setter for measurement

Allowable moving distance between the touch setter measurement start and the contact with the sensor.
Standard setting is 30 mm.

Parameter #28 - #31, 34, 35	ATC barrier range

ATC barrier can be set to prevent interference between the ATC unit and the A or B turret. ATC barrier is set with the first, and the second, and B turret zone setting points.



(A) Parameter #28 (X-axis second zone)
(B) Parameter #29 (Z-axis second zone)
(C) Parameter #30 (X-axis first zone)
(D) Parameter #31 (Z-axis first zone)
(E) Parameter #34 (X-axis B turret zone)
(F) Parameter #35 (Z-axis B turret zone)

Parameter #32	Tool offset, nose R compensation Fixed amount addition/deduction

The parameter is used to set an amount used for carrying out addition or deduction of a fixed amount to compensate the offset data.
Standard setting is 0 (in units of 1 μm).

Parameter #33	Tool offset, nose R compensation and size catcher Limit value for addition

This parameter is used to set the limit value when adding the value for tool offset, tool nose radius compensation, or size catcher. This limit is ignored when "1" is set at the optional parameter (bit) No. 8.

Procedure:

1) After pressing the **PARAMETER** key, select the OPTIONAL PARAMETER (LONG WORD) page by pressing the function key [F7] (ITEM ↓).

2) Move the cursor to the data to be set or changed.

3) Using the function key [F1] (SET) and the keyboard switches, set the desired data.

Parameter #36	2nd spindle zero shift

The delay angle of the 2nd spindle in reference to the main spindle is set when a workpiece is transferred from the 2nd spindle (sub spindle or pickoff spindle) to the main spindle in the synchronous mode.
(The value is set automatically with the chuck closed in the synchronous mode.)

This is one of the parameters to determine the orientation position of the 2nd spindle.

Parameter #40	2nd spindle orientation zero offset

The zero offset value for 2nd spindle (sub spindle or pickoff spindle) orientation is set.

The zero point for 2nd spindle orientation is obtained as follows:

Parameter No. 36 + parameter No. 40 + parameter No. 43

Parameter #41	2nd spindle acceleration unit amount

The acceleration time is set. The value is set in the same unit as with the M-spindle (Parameters #27 and 38).

With the initial setting, 9,600, the acceleration time is 0.1 sec at a spindle speed of 1,000 rpm.

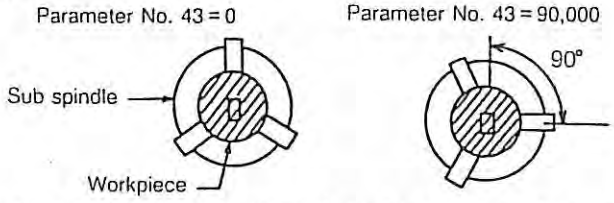
Parameter #42	Sub spindle backlash

The backlash amount of the sub spindle (opposing spindle) is set.

Parameter #43	2nd spindle zero offset

The zero offset value for the 2nd spindle (sub spindle or pickoff spindle) is set. This is the position of the 2nd spindle in reference to the main spindle at workpiece transfer in the synchronous mode.

Example



This is one of the parameters to determine the orientation position of the 2nd spindle

Parameter #44	Spindle orientation zero offset

With the spindle orientation specification, the zero point of C-axis commands are set.

The setting of this parameter is effective only for models equipped with the spindle high precision VAC. For models equipped with the standard VAC, the setting at the parameter (word) No. 12 is effective.

The spindle high-precision VAC is used on the following models:
LR10, FTL, LR15-M, LR25-M

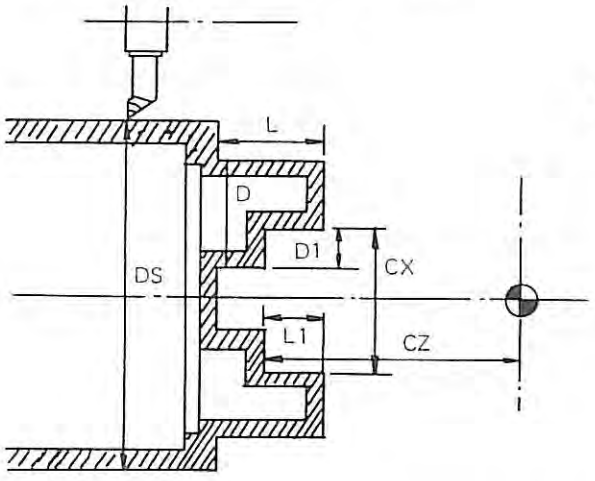
Parameter #46	LAP clearance amount (LAP4 specification)

When cutting is performed in the AP mode IV or V with LAP4 specification machine models, cutting is started after the cutting tool has been positioned at a point which is away from the blank material shape by the amount set at this parameter. For details, refer to Section 1, "Lathe Auto-programming Function (LAP4)" in the OSP5020L/OSP500L-G Special Function Manual No. 1 (Publication No. 3289-E).

Parameter #49	A side chuck barrier DS (for FTL only)

With FTL models, on which the headstock moves as the Z-axis, the cutting tool might move to the position as illustrated below.

To prevent the tool from colliding with the headstock, the spindle diameter (ram diameter) is set as the DS value.

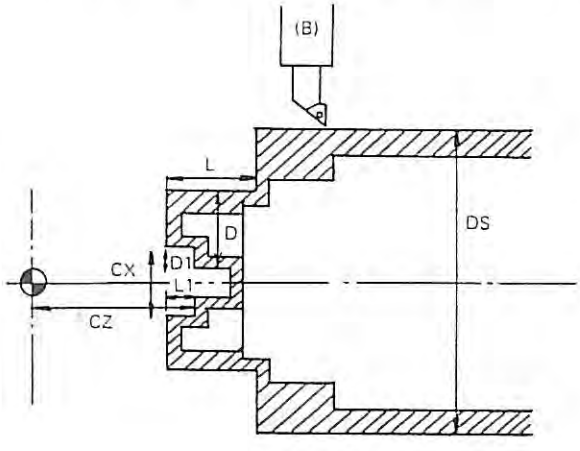


Chuck barrier -
Tool entry inhibited area

L, D, D_1, L1, CX, and CZ are the same as standard specification.

Parameter #50	B side chuck barrier DS (for FTL-Ⅲ only)
Parameter #51	B side chuck barrier L (for FTL-Ⅲ only)
Parameter #52	B side chuck barrier D (for FTL-Ⅲ only)

The chuck barrier value for the opposing spindle (B-spindle) is set.



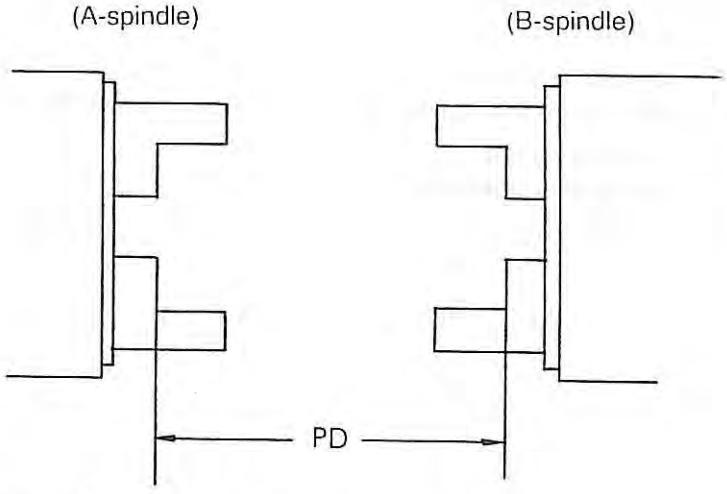
Parameter No. 50: Spindle (ram) diameter
Parameter No. 51: Jaw dimension D
Parameter NO. 52: Jaw dimension L

The values for D1, L1, CX and CZ and CZ are set at the user parameter.

Parameter #53 Interference-free distance PD (for FTL-Ⅲ only)

The interference-free distance between two spindles is set so that they do not interfere each other.

An alarm occurs when the actual distance between two spindles is smaller than the PD value.



Parameter #54 A side Z-axis limit position
(Interference limit with respect to the turret) (for FTL only)

When the A-spindle moves in the Z-axis negative direction exceeding the value set at this parameter, an alarm occurs so that the A-spindle does not interfere with the turret.

An alarm does not occur when the turret station set at the parameter (bit) No. 41 or No. 42 is indexed.

Parameter #55 B side Z-axis limit position
(Interference limit with respect to the turret) (for FTL-Ⅲ only)

When the B-spindle moves in the Z-axis negative direction exceeding the value set at this parameter, an alarm occurs so that the B-spindle does not interfere with the turret.

An alarm does not occur when the turret station set at the parameter (bit) No. 45 or No. 46 is indexed.

Parameter #60 - #63 Automatic operation of FTL and robot

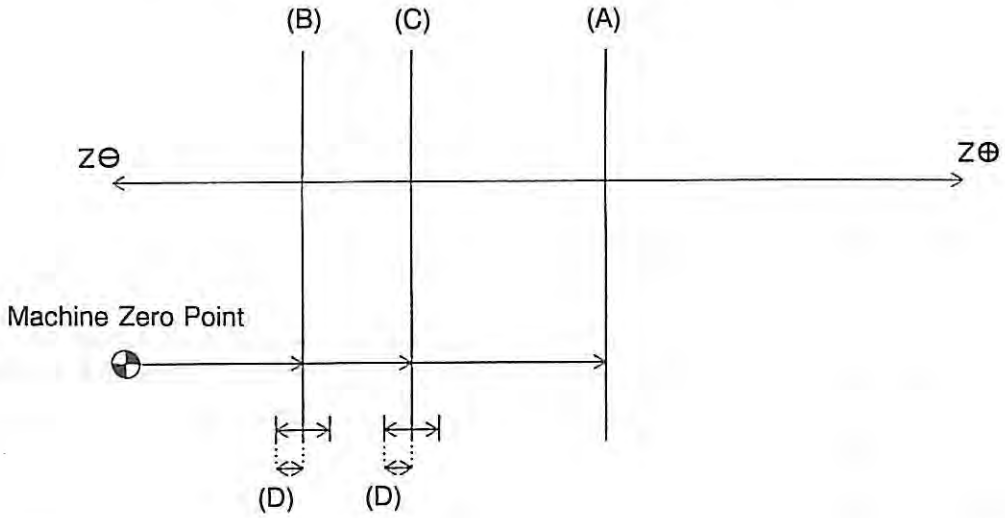
During automatic operation of the FTL and the robot, the Z-axis is advanced and retracted for loading/unloading workpieces. Therefore, completion of Z-axis positioning at the loading position, unloading position, and at an Interference-free position must be informed to the robot. Set these position using the following parameters.

(A) Interference-free position (retract position) ..... Parameter (long word) No. 60

(B) Unloading position (workpiece transfer position) ..... Parameter (long word) No. 61

(C) Loading position (workpiece blank receive position) ..... Parameter (long word) No. 62

(D) Answer area ..... Parameter (long word) No. 63



Parameter #64 Simple tow-along positive limit (for the LB15-Ⅱ and the simple tow-along specification only)

In pulse handle feed, the tailstock which is joined to the saddle does not move exceeding the position designated by this parameter. (This is to prevent the interference between the tailstock and the machine cover)

This parameter is effective only when tailstock joint is ON.

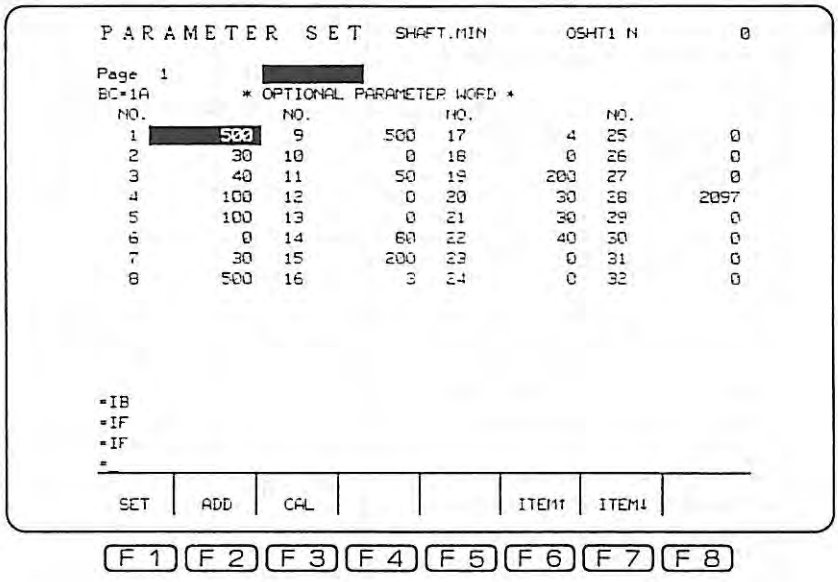
Parameter #65 W-axis at designated position (Machine with sub spindle)

For the machine with the sub spindle, if the W-axis is at the position designated by this parameter, the W-axis at the designated position EC output signal is turned on. This is to inform the robot or loader that the W-axis is at a potion where entry of the robot or loader is enabled.

Parameter #66 No workpiece confirmation time (msec)

IF the workpiece confirmation EC input signal is not turned on within the time set at this parameter after the execution of M120 (no workpiece confirmation command), an alarm of level B will occur.

3-5. Optional Parameter (Word)



Optional parameter (word) is used to control miscellaneous operations such as chuck open/close and spindle start/stop.

The Control Software No. 3 sets the data to the standard values. If necessary, set or change the data to the required value.

Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
1	Tape feed	Number of holes	250	10000	1
2	Spindle oscillation speed (motor speed)	rpm	30	500	1
3	Spindle jog speed (motor speed)	rpm	40	500	1
4	Spindle speed override in machine lock operation	%	100	1000	1
5	Chuck clamp timer	10 msec	100	1000	0
6	Chuck unclamp timer	10 msec	0	1000	0
7	Power save timer	minute	30	10000	0
8	SMW chuck clamp timer	10 msec	500	3000	0
9	SMW chuck unclamp timer	10 msec	500	3000	0
10	Alarm checking turret number for in-process gauging sensor protect	Number	0	12	0

Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
11	Feedrate in gauging cycle 1 (skip feed)	mm/min	50	500	1
12	Zero position for spindle orientation	0.1°	0	3599	0
13					
14	Tailstock spindle in-position timer	0.1 sec	80	2000	0
15	Feedrate in gauging cycle 2	mm/min	200	500	1
16	Lubrication motor operation interval (motor OFF time)	min	3 or 5	59	1
17	Overload detection time (load monitoring immune time)	0.1 sec	4	50	0
18	Post-process gauging data read timer	10 msec	0	6000	0
19	Chucking error detection air output timer	0.1 sec	30	6000	0
20	Lubrication motor ON time	sec	15 or 30	60	1
21					
22					
23	ATC magazine backlash	μm	0	100	– 100
24	ATC zero offset	μm	0	4999	– 4999
25	Machine number in FMS	Number	0	9999	0
26	STM time over timer	0.1 sec	0	6000	0
27	Cycle time over timer	sec	0	14400	0
28	Acceleration/deceleration coefficient for acceleration/deceleration of precision lathe	None	2097	32767	0
29					
30	Number of automatically recognized large diameter tools for ATC	pcs.	0	63	0
31	Not used				
32	Selection of display of servo processor internal variables	None	0	9999	0
33					

Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
34	RS232C busy time (CN0)	sec	30	9999	1
35	RS232C busy time (CN1)	sec	30	9999	1
36	RS232C busy time (CN2)	sec	30	9999	1
37	RS232C busy time (CN3)	sec	30	9999	1
38	RS232C busy time (CN4)	sec	30	9999	1
39	RS232C baud rate (CN0)	Baud	4800	19200	110
40	RS232C baud rate (CN1)	Baud	4800	19200	110
41	RS232C baud rate (CN2)	Baud	4800	19200	110
42	RS232C baud rate (CN3)	Baud	4800	19200	110
43	RS232C baud rate (CN4)	Baud	4800	19200	110
44	Punch device number	Number	0	5	0
45	Gauging data print device number	Number	0	5	0
46	DNC input/output device number	Number	0	4	0
47	Number of tools on turret A	pcs.	0	12	0
48	Number of tools on turret B	pcs.	0	12	0
49	Number of dots for scale frame shifting	dots	4	16	1
50	Output dot interval in INT	dots	2	4	2
51	Margin for automatic determination	%	100	100	0
52	Spindle orientation angle for C-axis connection	0.1°	0	3599	0
53					
54	Read device number	Number	0	5	0
55	Display distance of tool contour on the front view	%	50	100	10
56	Turret designated position (A turret)	Number	0	12	0
57	Timer for waiting time for command response from the host computer in the DNC mode	sec	30	999	1
58	Number of average load monitoring value collection	times	16	80	4
59	Overload alarm duration	×12.8 msec	10	999	1
60	Tool breakage alarm duration	×12.8 msec	5	999	1

Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
61	Lower end of load trace display	%	0	195	0
62	Upper end of load trace display	%	100	200	5
63	Automatic setting of the first limit level	%	110	200	100
64	Automatic setting of the second limit level	%	120	200	100
65	Percent value at the maximum input of the load monitoring spindle	%	0	500	0
66	Percent value at the maximum input of the M-tool spindle	%	0	500	0
67					
68					
69					
70	The number of bytes in a block in the program reverse transmission operation using DNC-B protocol A	byte	0	4000	0
71	Spindle orientation completion confirmation timer	10 msec	20	9999	0
72	Spindle torque limit value	%	100	100	40
73	Allowable chuck rotation speed	rpm	0	Max. spindle speed (depends on machine models)	0
74	Allowable door interlock spindle speed	rpm	50	500	1
75	C-axis center override	%	100	1000	0
76	2nd spindle torque limit value	%	100	100	40
77	Allowable 2nd spindle chuck rotation speed	rpm	0	Max. 2nd spindle speed	
78	Post-process gauging: RS232C method/size catcher method device number		0	4	0
79	2nd spindle chuck clamp timer		100	1000	0
80	2nd spindle chuck unclamp timer		0	1000	0
81	Pickoff chuck mounted turret station number	Number	0	16	1

Parameter No.	Description	Unit	Initial Value	Maximum Value	Minimum Value
82		Number	0	16	1
83	2nd spindle jog speed	rpm	40	500	1
84	2nd spindle SMW chuck clamp timer	10 msec	500	3000	0
85	2nd spindle SMW chuck unclamp timer	10 msec	500	3000	0
86	2nd spindle chucking error detection air output timer	0.1 sec	30	6000	0
87	2nd spindle oscillation speed	rpm	30	500	1
88	Torque monitoring delay time	10 msec	0	9999	0
89	Upper limit of torque limit command	%	0	100	1
90					
91					
92			0		
93	Turret designated position (B turret)	Number	0	12	0
94	LR15M-ATC magazine positioning pin IN signal output position	parts	0	2560	0
95	LR15M-ATC magazine rotation signal output position	parts	0	2560	0
96	LR15M-ATC magazine deceleration brake ON signal output position	parts	0	2560	0

<Meaning of Optional Parameter (Word)>

Parameter #1	Tape feed

In tape punch operation with the information transferred from memory in the program operation mode, tape feed holes must be punched before punching out the program data. The length of the leader section which contains only feed holes can be set as a parametric data of parameter #1.

Standard setting is 250 feed (approx. 60 cm).

Parameter #2 Spindle oscillation speed

Spindle oscillation speed can be set in terms of motor speed.

Standard setting is 30 rpm.

Parameter #3 Spindle jog speed

Spindle jog speed can be set in terms of motor speed.

Standard setting is 40 rpm..

Parameter #4 Spindle speed override in machine lock operation

During the machine lock mode, simulation is conducted using the programmed spindle speeds. It is possible to override the programmed spindle speeds.

Standard setting is 100%.

Parameter #5 Chuck clamp timer

The duration of the chuck clamp signal, i.e., from chuck clamped confirmation limit switch signal ON to chuck clamp signal turning OFF, can be set.

Standard setting is one second. The timer is set in units of 10 msec; setting is 100 for one second.

Parameter #6 Chuck unclamp timer

The duration of chuck unclamp signal, i.e., from chuck unclamped confirmation limit switch signal ON to chuck unclamp signal turning OFF, can be set.

Standard setting is zero.

The timer is set in units of 10 msec.

Parameter #7 Power save timer

The power save timer sets the time to activate the power save functions after one complete automatic cycle is finished with SINGLE BLOCK ineffective, or after machine operation is stopped due to alarm condition (type A) in continuous automatic mode operation.

The power save function is effective on:

Hydraulic power unit pump motor	1500 W
Guideway lubrication motor	5 W
Headstock lubrication motor	75 W
Work lamp	200 W
Total	1780 W

To cancel the power save mode, reset the control.

Standard setting is 30 minutes.

The unit of timer setting is "minute".

Parameter #8 SMW chuck clamp timer

This is the chuck clamp timer for SMW chuck.

Standard setting is five seconds.

The unit of timer setting is 10 msec; 500 for five seconds.

Parameter #9 SMW chuck unclamp timer

This is the chuck unclamp timer of SMW chuck.

Standard setting is five seconds.

The unit of timer setting is 10 msec; 500 for five seconds.

Parameter #10 Turret station number for in-process gauging sensor protect alarm check (only for the in-process gauging specification)

This sets the condition of sensor protect alarm check. If the setting is "0", monitoring of sensor protect alarm is always executed. When a number other than 0 is set, monitoring is executed only while the turret station with the designated number is indexed.

Parameter #11 Feedrate in gauging cycle 1

This sets the feedrate from the approach point to the contact point in the gauging cycle using the touch sensor. The feedrate in the second gauging cycle in the case of two-contact gauging cycle and the feedrate to be employed in the one-contact gauging cycle. The unit of data setting is mm/min.

The standard setting is 50 and the data setting range is 1 through 500.

Parameter #12 Zero position for spindle orientation

For the spindle orientation specification, zero position of C-axis command is set.

(The setting is not effective for models equipped with the high-precision VAC spindle motor. The setting at the parameter (long word) No. 44 is effective for these models.)

Parameter #14 Tailstock spindle in-position timer

The tailstock spindle advance command is completed after the in-position signal has been input.

Standard setting is 8 seconds.

Parameter #15 Feedrate in gauging cycle

This sets the feedrate from the approach point to the contact point in the first gauging cycle for the two-contact gauging cycle. The unit of data setting is mm/min.

The standard setting is 200 and the data setting range is 1 to 500.

Parameter #16 Lubrication motor operation interval (motor OFF time)

The operation interval of the way lubrication pump motor for the LB/LR series lathes is set. For other models, this parameter is not effective.

Standard setting is 3 or 5 seconds. (3 seconds for LR15 and LR25)

Parameter #17 Overload detection time (load monitoring immune time)

Effective for the overload detection specification. When an overload state continues exceeding the time duration set by this parameter, an alarm occurs.

During monitoring or automatic setting, collection of cutting load is suspended for a period set at this parameter after the feedrate is switched from rapid to cutting feed.

Parameter #18 Post-process gauging data read timer

Gauging data is read after the elapse of the time duration set by this parameter after the gauging cycle start command was output.

Parameter #19 Chucking error detection air output timer

The time duration in which chucking error detection air is kept on is set.

Parameter #20 Lubrication motor ON timer

For the lubrication motor controlled by software, the lubrication motor ON time is set with this parameter.

Parameter #23 ATC magazine backlash

For the ATC specification, lost motion of the ATC magazine is compensated by setting the proper backlash amount at this parameter.

Parameter #24 ATC magazine zero offset

For the ATC specification, the offset data used for aligning the ATC magazine position is set by this parameter.

Parameter #25 Machine number in FMS

The machine number of the subject machine when it is used in the FMS

Parameter #26 STM time over timer

The maximum time duration in which S, T and M functions may be executed (optional)

Parameter #27 Cycle time over timer

The maximum time duration in which one cycle may be executed (optional)

Parameter #28 Acceleration/deceleration coefficient for acceleration/deceleration of precision lathe

Acceleration/deceleration coefficient = 1/time constant × sampling time × 2^{16}

1/time constant = KV value

Initial setting : $10 \times 3.2 \times 10^{-3} \times 2^{16} = 2097$
Setting range : 0 - 32767

Parameter #30 Number of automatically recognized large-diameter tools for ATC

1 - (set value − 1) : Standard tool
Set value - 63 : Large-diameter tool
Initial setting : 0 (all tools are standard size tools)
Setting range : 0 - 63

Parameter #31 Workpiece pushing error detection air output timer

Parameter #32 Selection of display of servo processor internal variables

Upper two digits : Display data 1 (RSVPVAR1)
Lower two digits : Display data 2 (RSVPVAR2)

Parameter #34 - #38

RS232C busy time

Duration of time to wait for the response after the output of the request signal is set for individual channels. Setting unit is second. If no response is received within the set duration of time, an alarm occurs.

Parameter #34 . . . CN0: (RS232C interface on main card 13)
Parameter #35 . . . CN1:
Parameter #36 . . . CN2:
Parameter #37 . . . CN3: } (RS232C interface on RS board)
Parameter #38 . . . CN4:

Standard setting is 30 sec.

Parameter #39 - #43

RS232C baud rate

These parameters set baud rate for individual channels.

Parameter #39 . . . CN0: (RS232C interface on main card 13)
Parameter #40 . . . CN1:
Parameter #41 . . . CN2:
Parameter #42 . . . CN3: } (RS232C interface on RS board)
Parameter #43 . . . CN4:

Standard setting is 4,800 (bit/second).

Parameter #44 Punch device number

This sets the device used for punching by the number.

No. 0 . . . . . CN0: (RS232C interface on main card 13)
1 . . . . . CN1:
2 . . . . . CN2: } (RS232C interface on RS board)
3 . . . . . CN3:
4 . . . . . CN4:
5 . . . . . PP: (Parallel interface on punch)

Standard setting is 0.

Parameter #45 Gauging data print device number

The device used for printing the gauging data is designated by the number.

No. 0	. . . . .	CN0:	(RS232C interface on main card 13)
1	. . . . .	CN1:	(RS232C interface on RS board)
2	. . . . .	CN2:	
3	. . . . .	CN3:	
4	. . . . .	CN4:	
5	. . . . .	PP:	Centronics (data board)

Standard setting is 0.

Parameter #46 DNC input/output device number

The device used for inputting/outputting data in DNC mode operation is designated by the number.

No. 0	. . . . .	CN0:	(RS232C interface on main card 3)
1	. . . . .	CN1:	(RS232C interface on RS board)
2	. . . . .	CN2:	
3	. . . . .	CN3:	
4	. . . . .	CN4:	

Standard setting is 0.

Parameter #47, #48 Number of tools on turret

These parameters set the number of tools which can be set on the turret.

According to this setting, the turret rotation direction is automatically selected to choose the shorter path if bit data at bit 7 of parameter (bit) No. 4 is '1 '.

Parameter #47 . . . Number of tools on turret A
Parameter #48 . . . Number of tools on turret B

Parameter #49 Number of dots for scale frame shifting

This sets the number of dots the scale setting frame is shifted for each pushing of the enlarge/contract key or cursor key if the scale setting is carried on the graphic animated display page.

Standard setting is 4.

Parameter #50 Output dot interval in INT

This sets the minimum number of dots the tool is moved on the graphic animated display. When the setting number is small, the display will be smooth.

Standard setting is 2.

Parameter #51 Margin for automatic determination

On the graphic animated display, when automatically setting the scale from the data in the read part program, this sets the scale expansion ratio in reference to the read-in data. The unit of setting is percent.

Standard setting is 100.

Parameter #52 Spindle orientation angle for C-axis connection

Unit of setting is 0.1 deg.

Initial setting : 0
Setting range : 0 - 3599

Parameter #54 Read device number

The device used for tape reading is specified by the coded number.

No. 0 . . . . . TR:
1 . . . . . CN0:
2 . . . . . CN1:
3 . . . . . CN2:
4 . . . . . CN3:
5 . . . . . CN4:

Standard setting is 0.

Parameter #55 Display distance of tool contour on the front view

On the graphic display of the compound fixed cycle, the distance between the two adjacent contour lines to be drawn is set in percent in relation to the cutter diameter.

Standard setting is 50 (%).

Parameter #56 Turret designated position (turret A)

When the turret is indexed in the position designated by this parameter, its position is output to EC. (entering condition of robot or loader)

Parameter #57 Timer for waiting time for command response from the host computer in the DNC mode

This is the parameter used for the DNC-C specification. Time duration between the output of the request for sending NC program and the reception of its answer from the host computer in the INPUT and VERIFY functions in the DNC mode.

Standard setting is 30 (sec).

Parameter #58 Number of average load monitoring value collection

Cutting load of individual motors is calculated as the average of the number of data sets which is preset at this parameter. Since data is collected at intervals of 12.8 msec, an average value of cutting load is calculated every 204.8 msec if setting is 16.

Parameter #59 Overload alarm duration

If the detected cutting load exceeds the first limit level for a period calculated as the overload alarm duration from the value set at this parameter continuously, it generates an overload alarm. The overload alarm value is calculated as indicated below:

Setting × 12.8 msec = Overload alarm duration

When the setting is 10 msec, the overload alarm value is calculated as 128 msec.

Parameter #60 Tool breakage alarm duration

If the detected cutting load exceeds the second limit level for a period calculated as the tool breakage alarm duration from the value set at this parameter continuously, it generates a tool breakage alarm. The tool breakage alarm value is calculated in the same manner as the overload alarm duration.

Parameter #61 Lower end of load trace display

This parameter determines the display range of vertical axis for the load trace display (broken line chart) and it sets the lower end value in units of percents.

The display range is set in combination with the parameter No. 62 indicated below. If No. 61 = 0 and No. 62 = 150, the display range of the vertical axis is 0% to 150%.

Parameter #62 Upper end of load trace display

This parameter determines the display range of vertical axis for the load trace display and it sets the upper end value in units of percents.

Parameter #63 Automatic setting of the first limit level

The first limit level in the load monitoring and automatic setting mode (model-cutting) is calculated by the following equation:

$$\text{Reference level (\%) (max. value within monitoring part)} \times \frac{\text{Setting value}}{100} = \text{First limit level (\%)}$$

Parameter #64 Automatic setting of the second limit level

The second limit level in the automatic setting mode of the load monitoring is calculated by the following equation:

$$\text{Reference level (\%) (max. value within monitoring part)} \times \frac{\text{Setting value}}{100} = \text{Second limit level (\%)}$$

Parameter #65 Percent value at the maximum input of the load monitoring spindle

After loading the control software, the optimum value must be set in accordance with the spindle drive motor capacity and type as indicated below.

Type	Motor Capacity (kW)	Models	Setting Value (%)
DC motor	55	LC50 spindle	185
	45	LC40, LC50 spindle	185
	37	LC40, LC50, LH55 spindle	193
	22	LC30, LC40 spindle	182
	15	LC30 spindle	182
	11	LC30 spindle	184
VAC motor	3.7	LB9, FTL10 spindle LC40M-ATC M-spindle LCR35-M/LR45-M M-spindle	180
	5.5	LB15, LP15, FTL15, LR10, LR45-M spindle	164
	7.5	LB15, LP15, LS30-N, LB9, LB12	180
	11	LR15, LS30-N, LB25	164
	15	LR15, LH35, LR25	180
	22	LR25, LR35	164
	30	LR35	150
	37	LR45, LH55	150

Initial value : 0

Parameter #66 Percent value at the maximum input of the M-tool spindle

After loading the control software, the optimum value must be set in accordance with the M-spindle drive motor capacity and type as indicated above.

Parameter #70 The number of bytes in a block in the program reverse transmission operation using DNC-B protocol A

When transmitting the NC program to the host computer in protocol A, this parameter designates the number of bytes at the data portion of [DAT] command. That is, the units of data transmission is set. If this is not set (set to 0), the value follows the parameter Nb sent from the host computer.

Initial value : 0
Setting range : 0 - 4000

Parameter #71 Spindle orientation completion confirmation timer

This sets the duration in which spindle orientation is assumed to have completed after the spindle position is set within 4° from the target position.

Initial value : 20

Note: Setting unit is 10 msec; the setting value "20" indicates 200 msec.

Parameter #72 Spindle torque limit value

The motor torque limit value in accelerating or decelerating the high-precision VAC spindle speed is set.

Initial value : 100 %

Parameter #73 Allowable chuck rotation speed

This parameter sets the allowable chuck speed.
(Set a value without fail when the chuck has been replaced.)

Parameter #74 Allowable spindle speed in door interlock mode

When the door interlock function is on, spindle jog, oscillation, and orientation are allowed even when the door is open. However, if the spindle speed in these operations exceeds the speed set at this parameter, an alarm occurs.

Parameter #75 C-axis center override

This sets the C-axis feedrate override for feed operation only with C-axis, among the special operations in the G101 contour generation mode.

Parameter #76 2nd spindle torque limit value

The motor torque limit value in accelerating or decelerating the high-precision VAC 2nd spindle speed is set.

Initial value : 100%

Parameter #77 Allowable 2nd spindle chuck rotation speed

The allowable chuck rotation speed for the 2nd spindle is set. Set a value without fail when the chuck has been replaced.

Parameter #78 Post-process gauging:
RS232C method/size catcher method device number

The RS232C channel which is used for gauging is set by the number.

Setting:

0: CN0: (RS232C interface on main card 13)

1: CN1: (RS232C interface on RS board)

2: CN2: (RS232C interface on RS board)

3: CN3: (RS232C interface on RS board)

4: CN4: (RS232C interface on RS board)

Initial value : 0

Parameter #79 2nd spindle chuck clamp timer

The duration of the 2nd spindle chuck (sub spindle chuck, pickoff chuck) clamp signal is set. For details, refer to Parameter #5.

Parameter #80 2nd spindle chuck clamp timer

The duration of the 2nd spindle chuck (sub spindle chuck, pickoff chuck) unclamp signal is set. For details, refer to Parameter #6.

Parameter #81 Pick off chuck mounted turret station number

This sets the turret station number of turret A where a pickoff where a pickoff chuck is mounted.

Parameter #83 2nd spindle jog speed

The motor speed at 2nd spindle (sub spindle, pickoff spindle) jog operation is set.

Parameter #84 2nd spindle SMW chuck clamp timer

This is the chuck clamp timer for SMW chucks.

For details, refer to Parameter #8.

Parameter #85 2nd spindle SMW chuck unclamp timer

This is the chuck unclamp timer for SMW chucks.

For details, refer to Parameter #9.

Parameter #86 2nd spindle chucking error detection air output timer

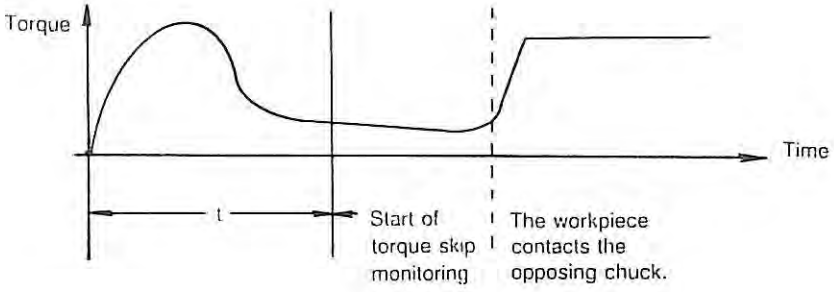
The time duration in which 2nd spindle (sub spindle, pickoff spindle) chucking error detection air is kept on is set.

Parameter #87	2nd spindle oscillation speed

The 2nd spindle (sub spindle, pickoff spindle) oscillation speed is set in terms of motor speed.

Parameter #88	Torque monitoring delay time

When a workpiece is transferred between the main spindle and the 2nd spindle (sub spindle, pickoff spindle), the time from torque skip feed ON (G22) to the start of torque skip condition monitoring is set.



Time "t" in the diagram to the left is set. (The time is set in units of 10 ms.)

Parameter #89	Upper limit value of torque limit command

The upper limit value of the P command in G29 (torque limit command) is set.

Parameter #93	Turret designated position (turret B)

When the turret is indexed in the position designated by this parameter, its position is output to EC. (entering condition of robot or loader)

Parameter #94	LR15M-ATC magazine positioning pin IN signal output position

This parameter sets the position where the pin IN signal is turned on by the number of parts (encoder value) in reference to the magazine positioning pin IN position.

Parameter #95	LR15M-ATC magazine rotation signal output position

This parameter sets the position where the magazine rotation output signal is turned off by the number of parts (encoder value) in reference to the magazine positioning pin IN position.

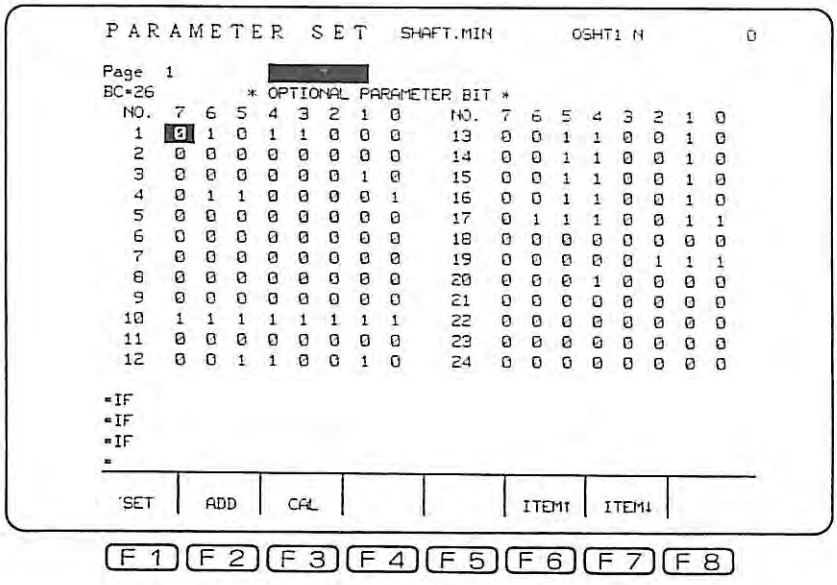
Parameter #96	LR15M-ATC magazine deceleration brake ON signal output position

This parameter sets the position where the brake ON signal is turned on by the number of parts (encoder value) in reference to the magazine positioning pin IN position.

Procedure:

1) In the PARAMETER SET mode, press the function key [F7] (ITEM ↓) to select the OPTIONAL PARAMETER (WORD).
2) Move the cursor to the data of the parameter number to be changed or set.
3) Set the new parametric data using function key [F1] (SET) and the keyboard.

3-6. Optional Parameter (Bit)



The optional parameter (bit) controls the turning on and off of respective functions in program operations, such as data transfer.

One set of data is made up of eight coded numerals, "1" and "0", and each digit is called "bit".

There are two types of data, one which means with a single bit and the other several bits.

Eight bits in one set of data specified with a parameter number are assigned with the numbers "0" through "7"; the leftmost digit is assigned with "7", the second leftmost with "6", ...

Example: No. 3 0 0 0 0 0 0 1 0

bit 7 bit 4 bit 0

No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
1	Type special code ignore	Tape special code alarm	Tape special code read	Tape read verify	Tape delimiter % (ER) code	Tape TV check	Tape code parity recognition	Tape code ISO
2	Output space code instead of NULL code when punching	Output of only LF for record end code	Verify data output at punching in DNC-A mode	No file name output at punching in DNC mode	No file name output at punching	Dry run active in rapid feed		Single block reading in
3		M75 cutting feed pattern designation	Tool offset addition limit cancel			10 μm unit	Decimal point command mm unit	Inch command
4	Automatic turret rotation direction control for shorter path	Externally input variable held	Alarm occurrence by tool life expiration	G30 F command feed	← Tool life management → Result of gauging cycle - NG	Tool wear amount	Cutting time	Number of machined workpieces
5	MOP specification*	Creation of gauging data print file	Shape designation in compound fixed cycle	No alarm display at an occurrence of error during print out of gauging data	Gauging data print ON	FTL simultaneous start of L and R possible		Pitch error compensation ignore*
6	MOP tool life management combined use	Overload alarm in tool life management	Coupled device alarm check designation	Load monitor interruption request effective for tool retraction cycle	Cycle stop completion of loader rot	Omission of selection of the identical program in external program selection	Switching program to be selected in external program	Mode condition for program start
7	ATC 0 tool time reduction	ATC dummy tool 0 tool check ignore (0: ON)	Robot interlock release	Overload detection alarm A	Tool gauging sensor head interlock release	Cycle stop completion of robot lot	Spindle orientation by M04	Spindle orientation by M03
8	No confirmation of program selection in coupled device	← MOP automatic setting → Tool data	Current pattern	Tool path data	Loader interlock release (0: ON)	Coupling external device alarm C	CEJ matic NG processing	Display of MSB
9	M236/M237 hold output	M234/M235 hold output	M232/M233 hold output	M230/M231 hold output	M107/M108 hold output	M105/M106 hold output	M103/M104 hold output	M101/M102 hold output
10	← Data related with graphic animated display → Blank actually cut in animated display	Tool contour in front view	Tailstock	Chuck	Blank material	Cutting feed tool path	Rapid feed tool path	Manual feed tool movement
11	File finish code	← IBM format data code →		IBM format file directory 1 file/sector type	FMS communi-cation error reset enable	IBM format without extension in output file name	Cycle time over check	STM time over check
12	← RS232C (CN0:) → File name read	DC code control type 2*	DC code control*	8 bit JIS*	Even parity*	Parity check yes*	Ready signal no*	Stop bit 1 bit*

Parameters in rectangle indicate those set at initial values.
Blank blocks indicate parameters not used.

When the setting of the parameters marked with * has been changed, it is necessary to once turn off the power and then turn it on again.

No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
13	File name read	DC code control type 2*	DC code control*	RS232C (CN1:) 8 bit JIS*	Even parity*	Parity check yes*	Ready signal no*	Stop bit 1 bit*
14	File name read	DC code control type 2*	DC code control*	RS232C (CN2:) 8 bit JIS*	Even parity*	Parity check yes*	Ready signal no*	Stop bit 1 bit*
15	File name read	DC code control type 2*	DC code control*	RS232C (CN3:) 8 bit JIS*	Even parity*	Parity check yes*	Ready signal no*	Stop bit 1 bit*
16	File name read	DC code control type 2*	DC code control*	RS232C (CN4:) 8 bit JIS*	Even parity*	Parity check yes*	Ready signal no*	Stop bit 1 bit*
17		Data related with hour meter output: Single block off	Feedrate override 100%	Spindle speed override 100%	Axis moving	Spindle in rotating	RUN lamp on	Automatic operation mode
18	Specification code	Panel output 1,2,3	Panel input 1,2,3	EC output (extended)1,2	EC output 1,2	EC input (extended)1,2	EC input 1,2	Axis data
19	Front view for multi-machining model No display*	Actual position display No CD display	NC operation monitor setting disable	Tow-along tailstock XB interlock release	Spindle not stopping in M02,30,00,01	Spindle jogging in M155	Spindle jogging in M51	Spindle jogging in M89
20	C-axis zero shift amount after coordinate conversion effective	Automatic limit for C-axis feedrate in contour generation face machining	Spindle jogging in M219	Spindle jogging in M127	Alarm lamp off: Alarm A	Alarm B	Alarm C	Alarm D
21				CNC slave station: CN 4:*	CN 3:*	CN 2:*	CN 1:*	CN 0:*
22								
23	DNC-B parameter (protocol A)							
24	DNC-B parameter (protocol A)							
25	Touch setter (B) standard sensor: Sensor 4	Sensor 3	Sensor 2	Sensor 1	Touch setter (A) standard sensor: Sensor 4	Sensor 3	Sensor 2	Sensor 1

Parameters in rectangle indicate those set at initial values.
Blank blocks indicate parameters not used.

When the setting of the parameters marked with * has been changed, it is necessary to once turn off the power and then turn it on again.

No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
26	← EIA code pattern to be replaced with ASCII " = " code →							
27	← EIA code pattern to be replaced with ASCII " * " code →							
28	← EIA code pattern to be replaced with ASCII " [" code →							
29	← EIA code pattern to be replaced with ASCII "] " code →							
30	← EIA code pattern to be replaced with ASCII " $ " code →							
31	← Irregular code →							
32	← Correct code (ISO) corresponding to irregular code →							
33	Touch setter double tooling turret 11 - 20		← Touch setter A interlock release → Parts catcher at retract end	Coupled device at retract position	Z-axis at positive limit	Chuck unclamp	Tailstock at retract end	Spindle stop
34	Mirror image coordinate select at spindle side*	Mirror image coordinate select at sub spindle side*	Post-process gauging number designating method		Parts catcher interlock release	S command cancel by M02/M30	Tapping mode change G184	Spindle orientation pin IN not output
35	Sensor protect alarm check (select condition)	Z-EC coordinate system selection 0: right (outside) 1: left (inside)	Robot/loader type-D (E) 0: cycle start effective 1: cycle start ineffective	← Sensor head interlock cancel → W-axis positive limit end	2nd spindle chuck unclamp	M-tool stop	EC extension output 3, 4 display	EC extension input 3, 4 display
36	←		Cam turning →					
		Drift compensation effective	XS-axis data reversed	Collecting deviator	← XS-axis control unit change →		Spindle P.G. resolving power	Digital filter effective
37	← FDOS → Backed up always	Backed up one time						LR15M-ATC arbitrary operation mode
38				← Designation of switching EC output position - by parameter or fixed → Additional air blow 4 M219/M218	Additional air blow 3 M219/M218	Additional air blow 2 M155/M154	Additional air blow 1 M51/M50	Air blow M88/M89
39				← Designation of switching EC output position - fixed at turret G code or spindle G code → Additional air blow 4 M219/M218	Additional air blow 3 M219/M218	Additional air blow 2 M155/M154	Additional air blow 1 M51/M50	Air blow M88/M89
40					NC turret pulse handle at B side	NC turret pulse handle at A side	High-speed turret clamp control	High-speed turret, low speed

Parameters in rectangle indicate those set at initial values.
Blank blocks indicate parameters not used.

When the setting of the parameters marked with * has been changed, it is necessary to once turn off the power and then turn it on again.

No. \ bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
41	← Interlock at A turret Z-axis interference position (FTL) →							
	16	15	14	13	12	11	10	9
42	← Interlock at A turret Z-axis interference position (FTL) →							
	8	7	6	5	4	3	2	1
43	Coolant output designation 0···1st spindle 1···2nd spindle		← SMW chuck switching →		Switching the high pressure coolant output 0···1st spindle 1···2nd spindle		2nd spindle orientation in the M04 direction	2nd spindle orientation in the M03 direction
			2nd spindle (FTL)	1st spindle (FTL)				
44			2nd spindle jog air blow M219	2nd spindle not stopping in M00, M01	2nd spindle jog air blow M127	2nd spindle jog air blow M155	2nd spindle jog air blow M51	2nd spindle jog air blow M89
45	← Interlock at B turret Z-axis interference position (FTL) →							
	16	15	14	13	12	11	10	9
46	← Interlock at B turret Z-axis interlock position (FTL) →							
	8	7	6	5	4	3	2	1
47	← Designation of switching EC output position - by parameter or fixed →							
	Additional coolant 2 M175/M174	Additional coolant 1 M145/M144	Coolant M08/M09					
48	← Designation of switching EC output position - fixed at turret G code or →							
	Additional coolant 2 M175/M174	Additional coolant 1 M145/M144	Coolant M08/M09					

< Meaning of Optional Parameter (bit) >

Parameter #1	Program operation

Bit 0 ··· Tape Code ISO

Bit 1 ··· Tape code parity recognition

Bits "0" and "1" are used to specify the coding system of tape punch out and tape verify.

	Bit 1	Bit 0	Operation Condition
A	1	1	In the READ and VERIFY modes, parity check is automatically performed disregarding the tape code, whether ISO or EIA. In the PUNCH mode, ISO code is selected.
B	1	0	In the READ and VERIFY modes, parity check is automatically performed disregarding the tape code, whether ISO or EIA. In the PUNCH mode, EIA code is selected.
C	0	1	The control selects ISO code for READ and VERIFY operation. (Tape code nonconformity results in alarm.) In the PUNCH mode, ISO code is selected.
D	0	0	The control selects EIA code for READ and VERIFY operation. (Tape code nonconformity results in alarm.) In the PUNCH mode, EIA code is selected.

Standard setting is A.

Bit 2 ··· Tape TV check

The number of characters in one block is checked: from the character preceded by LF (EOB) character to the next LF (EOB) character. The number of characters in one block must be even.

	Bit 2	Operation Condition
A	0	No TV check is performed in READ mode operation. In the PUNCH mode, the number of characters in one block is not adjusted.
B	1	In READ mode operation, the number of characters in one block is checked and if the number of characters in one block is odd, an alarm results. In the PUNCH mode, the number of characters in one block is adjusted so that one block contains an even number of characters.

Standard setting is A.

Bit 3 ··· Program end code, % or ER (Program delimiter)

"%" or "ER" code can be used as a delimiter of programs on a tape instead of using feed holes.

	Bit 3	Operation Condition
A	0	The control accepts "%" or "ER" code as the delimiter of a program.
B	1	The control accepts feed holes as the delimiter of a program.

Standard setting is B.

Note: *Tape data between the first CR (or LF) and the following one is ignored.*

Bit 4 ··· Tape read verify

Program data is verified when it is read.

	Bit 4	Operation Condition
A	0	Verify is not made after the completion of tape reading in.
B	1	Verify is made after the completion of tape reading in.

Standard setting is B.

Note: *File name is not verified.*

Bit 5 ··· Tape special code read

Bit 6 ··· Tape special code alarm

Bit 7 ··· Tape special code ignore

These bits determine whether the control triggers an alarm state, on ignores or accepts the special codes when they are read from the tape.

	Bit 7	Bit 6	Operation Condition
A	*	1	Alarm is constituted when a special code is read in READ mode operation.
B	1	0	A special code when read in READ mode operation is ignored.
C	0	0	A special code when read in READ mode operation is accepted.

**0 or 1*

Standard setting is A.

Note: *For acceptable standard codes, refer to the Programming Manual.*

Parameter #2 Automatic operation

Bit 0 ··· Single block reading in

Even when the operation is executed in the SINGLE BLOCK mode, blocks containing only the commands not calling for actual machine operation (NOEX command) are continuously executed to the block that contains the actual machine operation command when the CYCLE START button is pressed. This type of reading operation is handled as the single block function.

	Bit 0	Operation Condition
A	0	Blocks not calling for actual machine operation are continuous executed even in the SINGLE BLOCK mode.
B	1	The SINGLE BLOCK function is active on the blocks calling for no actual machine operation.

Standard setting is A.

Bit 1 ··· Not used

Standard setting is 0.

Bit 2 ··· Dry run active in rapid feed

Even when the DRY RUN function is active, rapid feedrate is not changed. However, it is possible to activate the dry run function at the rapid feedrate by setting proper parametric data.

	Bit 2	Operation Condition
A	0	Rapid feedrate is not changed in the DRY RUN mode.
B	1	Rapid feedrate is changed to dry run speed in the DRY RUN mode.

Bit 3 ··· No file name output at tape punch out

This specifies whether a file name is to be output or not in tape punch out operation.

=0 Output
=1 Not output

Standard setting is 0.

Bit 4 ··· No file name output at punching out in DNC mode

This specifies whether a file name is to be output or not at file output.

=0 Output
=1 Not output

Standard setting is 0.

Bit 5 ··· Verify data output at punching out in DNC-A mode

This specifies whether the data for verifying is to be output or not after punch out in the DNC mode.

= 0 Not output
= 1 Output

Standard setting is 0.

Bit 6 ··· Output of only LF for record end code

= 0 CR, LF output
= 1 Only LF output

Standard setting is 0.

Bit 7 ··· Output of space code instead of NULL code at punching in slave station

= 0 NULL code output
= 1 Space code output

Parameter #3 Unit System

Bit 0 ··· Inch command
Bit 1 ··· Decimal point mm unit command
Bit 2 ··· 10 μm unit

Combination of bits 0 through 2 can select a desired unit system for programming.

	Bit 2	Bit 1	Bit 0	Operation Condition
A	*	1	0	1 mm unit system
B	1	0	0	10 μm unit system
C	0	0	0	1 μm unit system
D	*	1	1	1 inch unit system
E	*	0	1	1/10000 inch unit system
F	1	1	0	0.1 m unit system (for precision lathe only)
G	1	0	1	1/100000 inch unit system (for precision lathe only)

* 1 or 0 (usually "0" is set)

Standard setting is A.

If inch/metric switchable feature is not selected, bit 0 cannot be set to "1".

Settings of F and G are allowed for precision lathe (LP15) only.

Bit 3 and 4
··· Not used

Standard setting is 0.

Bit 5 ··· Addition of tool offset or tool nose R compensation value exceeding 1 mm (or 0.1 inches) (optional parameter (word) No. 33) is effective or ineffective

= 0 Ineffective (when exceeding 1 mm)
= 1 Effective (even when exceeding 1 mm)

Bit 6 ··· Designation of infeed pattern in the M75 mode

The infeed pattern of thread cutting in the M75 mode is designated.

= 0 Infeed pattern 3
= 1 Infeed pattern 4

} Refer to the "Thread Cutting Compound Fixed Cycle" in the Programming Manual.

Parameter #4 Tool Life Management (optional)

Bit 0 ··· Number of machined workpieces
Bit 1 ··· Total cutting time
Bit 2 ··· Tool wear amount
Bit 3 ··· Result of gauging cycle - NG

Any tool life expiration determination factor can be selected from bits 0 through 2. Combination with bit 3, alarm signal output is available when tool life is determined to have expired in terms of the tool life factor selected by bits 0 through 2.

	Bit 3	Bit 2	Bit 1	Bit 0	Tool Life Management and Alarm Signal Output
A	0	0	0	1	Tool life is managed in terms of the number of machined workpieces.
A'	1	0	0	1	Tool life is managed in terms of the number of machined workpieces. Alarm signal output available.
B	0	0	1	0	Tool life is managed in terms of total cutting time.
B'	1	0	1	0	Tool life is managed in terms of total cutting time. Alarm signal output available.
C	0	1	0	0	Tool life is managed in terms of tool wear amount.
C'	1	1	0	0	Tool life is managed in terms of tool wear amount. Alarm signal output available.

Standard setting is A.

Bit 4 ··· G30 F command feed

= 0 Feed by the value set at parameter (word) No. 11
= 1 F command feed

Bit 5 ··· Alarm occurrence by tool life expiration

= 0 Cycle stop
= 1 Stop by alarm B

Standard setting is 1.

Bit 6 ··· Holding externally input variable

= 0 No processing
= 1 External outputs 41 through 48 are available with external input 1 through 8 turned on.

Standard setting is 1.

Bit 7 ··· Automatic turret rotation direction control for shorter path

= 0 Automatic control is not effective.
= 1 Turret rotation direction is automatically determined independent of the turret rotation direction command (M86, M87) programmed so that turret indexing path will be shorter.

Settings of parameters No. 47 and 48 are necessary.

Standard setting is 0.

Parameter #5 Data related with graphic display and gauging data printout

Bit 0 ··· Pitch error compensation ignore

With the ball screw pitch error compensation and the inductosyn pitch error compensation specifications, this specifies whether the pitch error compensation is carried out or not.

= 0 Compensation carried out
= 1 Compensation not carried out

Standard setting is 0.

Bit 1 and 2 ··· Not used

Standard setting is 0.

Bit 3 ··· Gauging data printout

= 0 Gauging data printed out
= 1 Gauging data not printed out

Standard setting is 0.

Bit 4 ··· No alarm display at an occurrence of error during printing out of gauging data

This specifies whether alarm message (982 measure data out) is displayed or not when an error occurs during printing out of gauging data.

= 0 Alarm message displayed
= 1 Alarm message not displayed

Standard setting is 0.

Bit 5 ··· Shape designation in the compound fixed cycle

= 0 Starting point of cutting
= 1 Starting point of G00 movement

Bit 6 ··· Creation of gauging data print file

= 0 Printout on device specified by parameter (word) No. 45
= 1 File created

Bit 7 ··· MOP specification

= 0 MOP specification not executed
= 1 MOP specification executed

Parameter #6 Data related with cycle starting

Bit 0 ··· Program start mode requirements

= 0 Auto mode
= 1 Cycle start in other than the automatic mode is possible except for MDI and manual.

Standard setting is 0.

Bit 1 ··· Switching program to be selected in external program selection

This specifies whether a program selected using the external program selection function is a main program or a schedule program.

= 0 Main program
= 1 Schedule program

Standard setting is 0.

Bit 2 ··· Omission of selection of identical program in external program selection

This specifies whether the program identical to the one presently selected is to be loaded again or not when it is selected using the external program selection function.

= 0 Loaded
= 1 Not loaded

Standard setting is 0.

Bit 3 ··· Cycle stop at completion of loader lot

= 0 Cycle stop not activated
= 1 Cycle stop activated

Bit 4 ··· Load monitor interruption request effective for tool retract cycle

With the machine with the tool retraction cycle specification, whether the interruption request from the load monitor is made effective or not is determined by the setting of this parameter.

= 0 Effective
= 1 Not effective

Bit 5 ··· Coupled device alarm check designation

For the machine with the coupled device specification, whether the alarm check for the coupled device is always made or only when the coupled device is controlled in the automatic mode is determined by the setting of this bit.

= 0 Always checked
= 1 Checked only in the automatic mode

Bit 6 ··· Overload alarm in tool life management

= 0 No alarm
= 1 Alarm occurs

Bit 7 ··· MOP tool life management combined use

= 0 Combined use
= 1 Not combined use

Parameter #7 Data related with spindle rotation direction in spindle orientation

Bit 0 ··· Spindle orientation by M03

= 0 No processing
= 1 After spindle has been stopped, spindle orientation is carried out in the M03 direction

Standard setting is 0.

Bit 1 ··· Spindle orientation by M04

= 0 No processing
= 1 After spindle has been stopped, spindle orientation is carried out in the M04 direction

Standard setting is 0.

Bit 2 ··· Cycle stop at completion of robot lot

= 0 Cycle stop not activated
= 1 Cycle stop activated

Standard setting is 0.

Bit 3 ··· Tool gauging sensor head interlock ON/OFF

= 0 Interlock ON
= 1 Interlock OFF

Standard setting is 0.

Bit 4 ··· Overload detection alarm A

= 0 Alarm C at overload detection
= 1 Alarm A at overload detection

Standard setting is 0.

Bit 5 ··· Robot interlock ON/OFF

= 0 Interlock ON
= 1 Interlock OFF (spindle rotation, turret movement)

Standard setting is 0.

Bit 6 ··· ATC dummy tool 0 tool check ignore

= 0 Sub arm operation executed for 0 tool
= 1 Sub arm operation ignored for 0 tool

Standard setting is 0.

Bit 7 ··· Dummy tool interlock cancel

= 0 An alarm occurs if the axis is moved without dummy tool for either L or M tool on turret A
= 1 Axis movement without dummy tool possible

Standard setting is 0.

Parameter #8 MSB

Bit 0 ··· Display of MSB

= 0 No display
= 1 MSB display at program selection

Standard setting is 0.

Bit 1 ··· No alarm occurrence for NG state of CEJ matic

This allows the operator to control an occurrence of alarm when the control is equipped with the tool life management function using the CEJ matic gauging function.

= 0 Alarm
= 1 No alarm occurrence and NG flag in the tool life management table is set ON.

Standard setting is 0.

Bit 2 ··· External device alarm C

= 0 Alarm A at an occurrence of alarm with coupled external device
= 1 Alarm C at an occurrence of alarm with coupled external device

Standard setting is 0.

Bit 3 ··· Loader interlock

= 0 Loader interlock ON
= 1 Loader interlock OFF

Bits 4 through 6

··· MOP automatic setting

This selects the model machining data to be stored during the MOP automatic setting mode.

Bit 4: Tool path data
Bit 5: Current pattern
Bit 6: Tool data

= 0 Data is ignored.
= 1 Data is stored.

Bit 7 ··· Coupled device program selection without confirmation

This bit data allows the program to be selected without confirmation when a program selection command is given to the coupled device. In this case, by setting "1" to this bit, the program actually selected by the coupled device is not confirmed whether its program number is the same as the one designated.

= 0 Confirmed
= 1 Not confirmed

Standard setting is 0.

Parameter #9 Data related with M code

Bit 0 ··· M101/M102 hold output

= 0 Output not held
= 1 Output held

Standard setting is 0.

Bit 2 ··· M105/M106 hold output

= 0 Output not held
= 1 Output held

Standard setting is 0.

Bit 3 ··· M107/M108 hold output

= 0 Output not held
= 1 Output held

Standard setting is 0.

Bit 4 ··· M230/M231 hold output

= 0 Output not held
= 1 Output held

Standard setting is 0.

Bit 5 ··· M232/M233 hold output

= 0 Output not held
= 1 Output held

Standard setting is 0.

Bit 6 ··· M234/M235 hold output

= 0 Output not held
= 1 Output held

Standard setting is 0.

Bit 7 ··· M236/M237 hold output

= 0 Output not held
= 1 Output held

Standard setting is 0.

Note: As long as output of these M codes is held, the state does not change even if the NC is reset.

Parameter #10 Data related with graphic animated display

This specifies whether the display element on the graphic animated display in the operation mode is to be displayed or not for individual display elements.

Bits 0 through 7

··· = 0 Corresponding display element is not displayed.
= 1 Corresponding display element is displayed.

Standard setting is 1.

Display element:

Bit 0 Manual tool movement path
Bit 1 Rapid feed tool path
Bit 2 Cutting feed tool path
Bit 3 Blank material
Bit 4 Chuck
Bit 5 Tailstock
Bit 6 Tool contour in front view
Bit 7 Progress of cutting

Parameter #11 Data related with time over check

Bit 0 ··· STM time over check

= 0 Time over check not executed
= 1 Time over check executed in accordance with parameter (word) No.26

Bit 1 ··· Cycle time check over

= 0 Time over check not executed
= 1 Time over check executed in accordance with parameter (word) No.27

Bit 2 ··· IBM format without extension in output file name

= 0 Extension MIN is replaced with blank
= 1 All extension are replaced with blank

Bit 3 ··· FMS communication error reset

= 0 FMS communication error reset is impossible.
= 1 FMS communication error reset is possible.

Bit 4 ··· IBM format file directory 1-file/sector type

In the IBM format, file directory is usually written in the 2-file/sector. However, it is possible to change this into 1-file/sector only for side 1, cylinder 0 (when 2DD floppy disk is used.).

= 0 File directory 2-file/sector
= 1 File directory 1-file/sector

Bits 5 and 6

··· IBM format data code

= 00 ISO
= 01 EBCDIC
= 10 ASCII
= 11 EBCDIC

Standard setting is 00.

Bit 7 ··· File end code

This specifies the file end code in file output in IBM format.

= 0 NULL
= 1 %

Parameter #12

Data related with communication channel CN0: (channel 0)
(Standard setting is 1 for only bit 4.)

Bit 0 ··· RS232C stop bit check

= 0 Stop bit 2
= 1 Stop bit 1

Bit 1 ··· Availability of ready signal for RS232C interface

= 0 Ready signal available
= 1 Ready signal not available

Bit 2 ··· RS232C parity check

= 0 Parity check not carried out
= 1 Parity check carried out

Bit 3 ··· RS232C even parity

= 0 Odd parity
= 1 Even parity

Bit 4 ··· RS232C 8 bit JIS

= 0 7 bit JIS (ISO code; JIS7 + parity)
= 1 8 bit JIS

Note: *This parameter is used to select the length of data to be output to a printer using the LIST command or measured data printout function.*

Bit 5 and 6

··· Designation of DC code control

	Bit 6	Bit 5	Operation Condition
A	0	0	No DC code control
A'	1	0	No DC code control
B	0	1	Standard DC code control
C	1	0	DC code control type 2

Bit 7 ··· File name read

= 0 File name not read when data is read
= 1 File name is read with the data

Parameter #13	Data related with communication channel CN1: (channel 1)
Parameter #14	Data related with communication channel CN2: (channel 2)
Parameter #15	Data related with communication channel CN3: (channel 3)
Parameter #16	Data related with communication channel CN4: (channel 4)

Contents of parameters #13 through #16 are the same as that of parameter #12.

Parameter #17	Data related with hour meter output

The hour meter operation signal is output when the conditions corresponding to the bits designated are met.

= 0 Condition ignored
= 1 Condition specified

Conditions:

Bit 0 Automatic operation mode
Bit 1 RUN lamp on
Bit 2 Spindle in rotation
Bit 3 Axis moving
Bit 4 Spindle speed override 100%
Bit 5 Feedrate override 100%
Bit 6 Single block OFF
Bit 7 Not used

Standard setting of bits 0, 1, 4, 5 and 6 is 1.

Parameter #18	Data related with display of operation mode

This sets the check data display page assigned to the bits designated.

= 0 Page displayed
= 1 Page not displayed

Standard setting is 0.

Display page:

Bit 0 Axis data
Bit 1 EC input 1, 2
Bit 2 EC input (extended) 1, 2
Bit 3 EC output 1, 2
Bit 4 EC output (extended) 1, 2
Bit 5 Panel input 1, 2, 3
Bit 6 Panel output 1, 2, 3
Bit 7 Specification code

Parameter #19 — Spindle jog operation, NC work counter, display of actual position

Bits 0 through 2 ··· Spindle jog

= 0 Spindle jog not carried out during air blow
= 1 Spindle jog carried out during air blow

Bit 0 At M89
Bit 1 At M51
Bit 2 At M155

Bit 3 ··· Spindle does not stop with M02, M30, M00, or M01. (optional)

With the special spindle stop specification:

= 0 Spindle stop with M02, M30, M00 and M01
= 1 No spindle stop with M02, M30, M00 or M01

Bit 4 ··· Tow-along tailstock XB-axis interlock cancel (optional)

Cancellation of positive limit interlock on the XB-axis with the tow-along tailstock specification

= 0 Interlock effective
= 1 Interlock cancel

Bit 5 ··· NC operation monitor setting disabling bit

= 0 Setting enabled
= 1 Setting disabled

Bit 6 ··· CA spindle position display disabling bit

= 0 Setting enabled
= 1 Setting disabled

Bit 7 ··· No front view for multi-machining model

This specifies whether the front view is displayed in graphic function for multiple-machining model.

= 0 Front view is displayed.
= 1 Front view is not displayed.

Parameter #20 — Related to alarm lamp of each alarm

Bit 0 ··· = 0 Alarm lamp is turned on at an occurrence of alarm of level D.
= 1 Alarm lamp is not turned on at an occurrence of alarm of level D.

Bit 1 ··· = 0 Alarm lamp is turned on at an occurrence of alarm of level C.
= 1 Alarm lamp is not turned on at an occurrence of alarm of level C.

Bit 2 ··· = 0 Alarm lamp is turned on at an occurrence of alarm of level B.
= 1 Alarm lamp is not turned on at an occurrence of alarm of level B.

Bit 3 ··· = 0 Alarm lamp is turned on at an occurrence of alarm of level A.
= 1 Alarm lamp is not turned on at an occurrence of alarm of level A.

Standard setting is 0000.

Bit 4 ··· Spindle jog while air blow designated by M127 is active.

0 = Spindle jogs
1 = Spindle does not jog

Bit 5 ··· Spindle jog while air blow designated by M219 is active.

0 = Spindle jogs
1 = Spindle does not jog

Bit 6 ··· = 0 C-axis feedrate for contour generation facing is automatically limited.
= 1 C-axis feedrate for contour generation facing is not automatically limited.

Bit 7 ··· C-axis zero shift amount after coordinate conversion effective

= 0 C-axis zero shift amount ineffective
= 1 C-axis zero shift amount effective

Parameter #21 Related with CNC slave station control

Bit 0 ··· = 1 CN0: is the slave station.
= 0 CN0: is not the slave station.

Bit 1 ··· = 1 CN1: is the slave station.
= 0 CN1: is not the slave station.

Bit 2 ··· = 1 CN2: is the slave station.
= 0 CN2: is not the slave station.

Bit 3 ··· = 1 CN3: is the slave station.
= 0 CN3: is not the slave station.

Bit 4 ··· = 1 CN4: is the slave station.
= 0 CN4: is not the slave station.

Standard setting is 0000.

Bits 5 through 7

··· Not used

Standard setting is 0.

Parameter #22 Not used

Parameter #23 Related with DNC-B parameter

Parameter #24 Related with DNC-B parameter

Data of the 41st to 44th bytes in [SAT] and [SET] commands which are transferred to and from the host computer.

Bit 0 ··· = 0 Buffer memory is not cleared by NC reset.
= 1 Buffer memory is cleared by NC reset.

Standard setting is 0.

Not decided for other bits. (Standard setting is all 0.)

Parameter #25 Related to touch setter

This specifies the sensor to be referenced to. (When the position of the reference sensor has changed, the variation amount should be added to the sensor on the other side. Addition is not carried out if it is not the reference sensor.)

Bit 0 ···	= 1	Sensor 1 on A side is taken as the standard sensor.
	= 0	Sensor 1 on A side is not taken as the standard sensor.
Bit 1 ···	= 1	Sensor 2 on A side is taken as the standard sensor.
	= 0	Sensor 2 on A side is not taken as the standard sensor.
Bit 2 ···	= 1	Sensor 3 on A side is taken as the standard sensor.
	= 0	Sensor 3 on A side is not taken as the standard sensor.
Bit 3 ···	= 1	Sensor 4 on A side is taken as the standard sensor.
	= 0	Sensor 4 on A side is not taken as the standard sensor.
Bit 4 ···	= 1	Sensor 1 on B side is taken as the standard sensor.
	= 0	Sensor 1 on B side is not taken as the standard sensor.
Bit 5 ···	= 1	Sensor 2 on B side is taken as the standard sensor.
	= 0	Sensor 2 on B side is not taken as the standard sensor.
Bit 6 ···	= 1	Sensor 3 on B side is taken as the standard sensor.
	= 0	Sensor 3 on B side is not taken as the standard sensor.
Bit 7 ···	= 1	Sensor 4 on B side is taken as the standard sensor.
	= 0	Sensor 4 on B side is not taken as the standard sensor.

Standard setting is all 0.

Parameters #26 through #32

Parameters related with input/output of irregular codes

The irregular codes in the EIA or ISO code can be read and punched out. For details, refer to 2-1. "Transfer of Main Program Data".

Parameter #33 Touch setter arm interlock

This sets the ON/OFF state of interlocks when the touch setter arm advances or retracts.

= 0 Interlocks checked
= 1 Interlocks released

Conditions:

Bit 0 Spindle stop
Bit 1 Tailstock at retract end
Bit 2 Chuck unclamp
Bit 3 Z-axis at positive limit
Bit 4 Coupled device at retract position
Bit 5 Parts catcher at retract end
Bit 6 Not used
Bit 7 This sets which tools are offset, tools 1 through 10 or tools 11 through 20, when the gauging cycle using a touch setter is executed with double tooling turret specification models.

= 0 Tools 1 through 9
= 1 Tools 11 through 20

Parameter #34 Spindle orientation pin IN

Bit 0 ··· This specifies whether or not the pin IN (brake ON) is carried out with spindle orientation specification models.

= 0 Pin IN (brake ON) is output
= 1 Pin IN (brake ON) is not output

Bit 1 ··· = 0 G184: Float tapping mode
= 1 G184: Synchronized tapping mode

Bit 2 ··· This specifies whether or not the currently effective S command is canceled by M02 or M30.

= 0 S command not canceled by M02 or M30
= 1 S command canceled by M02 or M30

Note: *S commands for the 1st spindle, 2nd spindle, and M-tool spindle can be canceled by setting "1".*

S commands cannot be canceled by resetting the NC or by switching the operation mode.

Bit 3 ··· The machine is provided with the following interlock:

When a ZB-axis move command is designated while the parts catcher is not at the retract end, an alarm will be activated. Or when the parts catcher is placed at a position other than the retract position while the ZB-axis is moving, an alarm will be activated.

This parameter can set whether or not this interlock is kept effective or released.

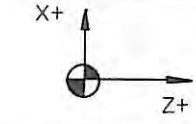
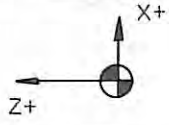
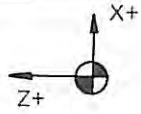
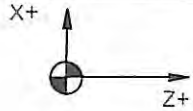
= 0 Interlock effective
= 1 Interlock released

Bit 5 ··· With the post-process gauging cycle using the RS232C, the method to designate the gauging number is set.

= 0 Direct designation method
= 1 Indirect designation method

Bits 6 and 7

··· Whether or not the mirror image function is activated in the Z-axis direction for models with the 2nd spindle (sub spindle, pickoff spindle, or opposing spindle) is set.

	Main spindle	2nd spindle
= 0		
= 1		

Bit 6 : Main spindle
Bit 7 : 2nd spindle

Procedure:

1) After pressing the function key [F7] (ITEM ↓), select the OPTIONAL PARAMETER (BIT) page.
2) Move the cursor to the data of the parameter number to be changed or set.
3) Set the new parametric data using function key [F1] (SET) and the keyboard.

Parameter #35

Bit 0 ··· 03 and 04 of EC extended input screen

Bit 1 ··· 03 and 04 of EC extended output screen

Bit 2 through 4

··· The following conditions on bit 2, 3, and 4 are recognized as the interlock condition when the touch setter arm advancing/retracting/sliding.

Bit 2 ··· M-tool spindle stop in combined machine

Bit 3 ··· Chuck unclamp at sub spindle side

Bit 4 ··· W-axis at positive limit end
= 0 Interlock ON
= 1 Interlock OFF

Bit 5 ··· Selects the condition for cycle start the robot/loader interface.
= 0 Type-C cycle start condition for robot/loader interface effective
= 1 Type-C cycle start condition for robot/loader interface ineffective

Bit 6 ··· Selects the coordinate system for side machining.
= 0 Coordinate system at the right hand of the Z-axis as the vertical axis
= 1 Coordinate system at the left hand of the CE-axis as the vertical axis

Bit 7 ··· Selects the condition for sensor protect alarm check.
(touch setter specification)
= 0 Checks sensor protect alarm always
= 1 Checks sensor protect alarm when the touch setter is not at the retract end

Parameter #36 Parameter related with cam turning

This is the parameter related with cam turning which is available on the LB9-CAM, LR15-CAM, etc.

Parameter #37 Parameter related with ATC arbitrary operation and FDOS

Bit 0 ··· LR15M-ATC arbitrary operation

= 0 Normal operation mode (ATC arbitrary operation ineffective)
= 1 Arbitrary operation mode (ATC arbitrary operation effective)

Bits 1 through 5

··· Not used

Bits 6 and 7

··· Backup method at FDOS RUN

Bit 7	Bit 6	Backup method
0	0	Not backed up
1	0	Backed up always (disregarding the setting at bit 6)
0	1	Backed up only one time
1	1	Backed up always (disregarding the setting at bit 6)

Parameters #38 Parameter related with EC output position for air blow

Bit 0 through 4

··· These bit data specifies whether the EC output position corresponding to the air blow ON/OFF M code is switched by the parameter or it is fixed.

= 0 Switched by parameter
= 1 Fixed

Note 1 : These bit data is used in combination with parameter (bit) No. 39, bit 0 to bit 4.

Parameter (bit) No. 39, bit 0 to bit 4:

i) When "0" is set at parameter (bit) No. 38, bit 0 to bit 4:

These bit data specifies whether the EC output position is switched by turret designation G codes (G13/G14) or spindle designation G codes (G140/G141).

= 0 Switched by turret designation G codes (G13/G14)
= 1 Switched by spindle designation G codes (G140/G141)

ii) When "1" is set at parameter (bit) No. 38, bit 0 to bit 4:

These bit data specifies whether switching of the EC output position is performed at the 1st spindle side or it is performed at the 2nd spindle side, irrespective of the selected coordinate system.

= 0 At the 1st spindle side
= 1 At the 2nd spindle side

	Air blow M89/M88	Additional air blow 1 M51/M50	Additional air blow 2 M155/M154	Additional air blow 3 M127/M126	Additional air blow 4 M219/M218
Parameter (bit) No. 38	bit 0	bit 1	bit 2	bit 3	bit 4
Parameter (bit) No. 39	bit 0	bit 1	bit 2	bit 3	bit 4

The following tables show the combination of parameter bit data and EC output positions for air blow.

Output Pattern

Parameter (bit) No. 38	Parameter (bit) No. 39		M89/M88	M51/M50	M155/M154	M127/M126	M219/M218
	"1"	"0"					
"0"	G140	G13	1st spindle air blow	1st spindle additional air blow	1st spindle additional air blow B	1st spindle additional air blow C	1st spindle additional air blow D
	G141	G14	2nd spindle air blow	2nd spindle additional air blow	2nd spindle additional air blow B	2nd spindle additional air blow C	2nd spindle additional air blow D

Parameter (bit) No. 38	Parameter (bit) No. 39	M89/M88	M51/M50	M155/M154	M127/M126	M219/M218
"1"	"0"	1st spindle air blow	1st spindle additional air blow	1st spindle additional air blow B	1st spindle additional air blow C	1st spindle additional air blow D
	"1"	2nd spindle air blow	2nd spindle additional air blow	2nd spindle additional air blow B	2nd spindle additional air blow C	2nd spindle additional air blow D

Note 2: *Spindle jog motion by air blow which is activated by turning on parameter bit data is dependent on the combination of parameter (bit) No. 38, bit 0 to bit 4 and parameter (bit) No. 39, bit 0 to bit 4.*

Spindle jog motion is not performed when parameter bit data is OFF.

1st spindle jog by M89	Optional parameter (bit) No. 19, bit 0
1st spindle jog by M51	Optional parameter (bit) No. 19, bit 1
1st spindle jog by M155	Optional parameter (bit) No. 19, bit 2
1st spindle jog by M127	Optional parameter (bit) No. 20, bit 4
2nd spindle jog by M89	Optional parameter (bit) No. 44, bit 0
2nd spindle jog by M51	Optional parameter (bit) No. 44, bit 1
2nd spindle jog by M155	Optional parameter (bit) No. 44, bit 2
2nd spindle jog by M127	Optional parameter (bit) No. 44, bit 3
2nd spindle jog by M219	Optional parameter (bit) No. 44, bit 5

i) When "0" is set at parameter (bit) No. 38, bit 0 to bit 4:

- Spindle jog is not performed when "0" is set at parameter (bit) No. 39, bit 0 to bit 4.
- Spindle jog is performed when "1" is set at parameter (bit) No. 39, bit 0 to bit 4.

ii) When "1" is set at parameter (bit) No. 38, bit 0 to bit 4:

- The 1st spindle performs jog motion when "0" is set at parameter (bit) No. 39, bit 0 to bit 4.
- The 2nd spindle performs jog motion when "1" is set at parameter (bit) No. 39, bit 0 to bit 4.

Parameter #39

Bit 0 through 4

··· Refer to the explanation for optional parameter (bit) No. 38, bit 0 to bit 4.

Parameter #40 Parameter related with high-speed turret (for adjustment)

Bit 0 ··· When "1" is set, the turret can be rotated only in the low-speed mode, at a speed which is approximately 1/10 to 1/20 (depending on machine models) of the normal rotation speed.

Bit 1 ··· The turret is only unclamped and is not rotated when "1" is set. The turret can be rotated manually.

Bit 2 ··· When "1" is set, the manual operation mode is established. This unclamps the turret and allows the turret to be rotated using the pulse handle. The setting of the **PULSE HANDLE** keys on the machine operation panel is ignored.

Bit 3 ··· When "1" is set, the manual operation mode is established. This unclamps the turret and allows the turret to be rotated using the pulse handle. The setting of the **PULSE HANDLE** keys on the machine operation panel is ignored.

After the completion of operation, set "0" and designate an T command.

Parameters #41, #42, #45, and #46

Interlock at the Z-axis interference position with FTL models

The turret indexing position with which the spindle can move in the Z-axis negative direction exceeding the position set at the parameter (long word) No. 54 and 55 is set.

=0 The spindle can not move (an alarm occurs).
=1 The spindle can move (an alarm does not occur).

The correspondence between the bit number and the turret station

Parameter No.	Turret	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
41	A	–	–	–	–	12	11	10	9
42	A	8	7	6	5	4	3	2	1
45	B	–	–	–	–	12	11	10	9
46	B	8	7	6	5	4	3	2	1

–: No turret station available

Example: When "1" is set at the parameter (bit) No. 42, bit 3, the spindle can move in the Z-axis negative direction exceeding the Z-axis interference position while T4 of the turret A is indexed.

Parameter #43 Parameter related with 2nd spindle orientation

Bit 0 and 1

··· Availability and direction of 2nd spindle orientation (sub spindle, pickoff spindle) is set.

Refer to Parameter #7.

Bit 2 ··· Not used

Bit 3 ··· Switching the high pressure coolant output
=0 1st spindle side
=1 2nd spindle side

Bit 4 ··· Designation of main spindle chuck (for FTL only)

The type of main spindle chuck, hydraulic or SMW, is designated.

=0 Hydraulic chuck
=1 SMW chuck

Bit 5 ··· Designation of 2nd spindle chuck (for FTL only)

The type of 2nd spindle chuck, hydraulic or SMW, is designated.

=0 Hydraulic chuck
=1 SMW chuck

Bit 6 ··· Not used

Bit 7 ··· Coolant output designation

Whether coolant output for the 2nd spindle is controlled independent of the main spindle or it is controlled by the coolant output signal for the main spindle is set.

=0 The coolant output signal is controlled independently.
=1 The coolant output signal is controlled by the coolant output signal for the main spindle.

Parameter #44	Parameter related with spindle jog while 2nd spindle air blow is active

= 0 Spindle does not jog.
= 1 Spindle jogs.

Bit 0 ··· Air blow designated by M89
Bit 1 ··· Air blow designated by M51
Bit 2 ··· Air blow designated by M155
Bit 3 ··· Air blow designated by M127

Bit 4 ··· Whether or not the 2nd spindle stops when M00 or M01 is designated is set.

= 0 Spindle stops.
= 1 Spindle does not stop.

Bit 5 ··· Air blow designated by M219

Bits 6 and 7

··· Not used

Parameter #47	Air blow output designation

Bit 5 through 7

··· These bit data specifies whether the EC output position corresponding to the coolant ON/OFF M code is switched by the parameter or it is fixed.

= 0 Switched by parameter
= 1 Fixed

Note: These bit data is used in combination with parameter (bit) No. 48, bit 5 to bit 7.

Parameter (bit) No. 48, bit 5 to bit 7:

i) When "0" is set at parameter (bit) No. 47, bit 5 to bit 7:

These bit data specifies whether the EC output position is switched by turret designation G codes (G13/G14) or spindle designation G codes (G140/G141).

= 0 Switched by turret designation G codes (G13/G14)
= 1 Switched by spindle designation G codes (G140/G141)

ii) When "1" is set at parameter (bit) No. 47, bit 5 to bit 7:

These bit data specifies whether switching of the EC output position is performed at the 1st spindle side or it is performed at the 2nd spindle side, irrespective of the selected coordinate system.

= 0 At the 1st spindle side
= 1 At the 2nd spindle side

	Coolant M08/M09	Additional coolant 1 M145/M144	Additional coolant 2 M175/M174
Parameter (bit) No. 47	bit 5	bit 6	bit 7
Parameter (bit) No. 48	bit 5	bit 6	bit 7

The following tables show the combination of parameter bit data and EC output positions for coolant.

Output Pattern

Parameter (bit) No. 47	Parameter (bit) No. 47		M08/M09	M145/M144	M175/M174
	"1"	"0"			
"0"	G140	G13	1st spindle coolant	1st spindle additional coolant 1	1st spindle additional coolant 2
	G141	G14	2nd spindle coolant	2nd spindle additional coolant 1	2nd spindle additional coolant 2

Parameter (bit) No. 47	Parameter (bit) No. 48	M08/M09	M145/M144	M175/M174
"1"	"0"	1st spindle coolant	1st spindle additional coolant 1	1st spindle additional coolant 2
	"1"	2nd spindle coolant	2nd spindle additional coolant 1	2nd spindle additional coolant 2

Parameter #48

Bit 5 through 7

··· Refer to the explanation for optional parameter (bit) No. 47, bit 5 through 7.

Procedure:

1) After pressing the function key [F7] (ITEM ↓), select the OPTIONAL PARAMETER (BIT) page.

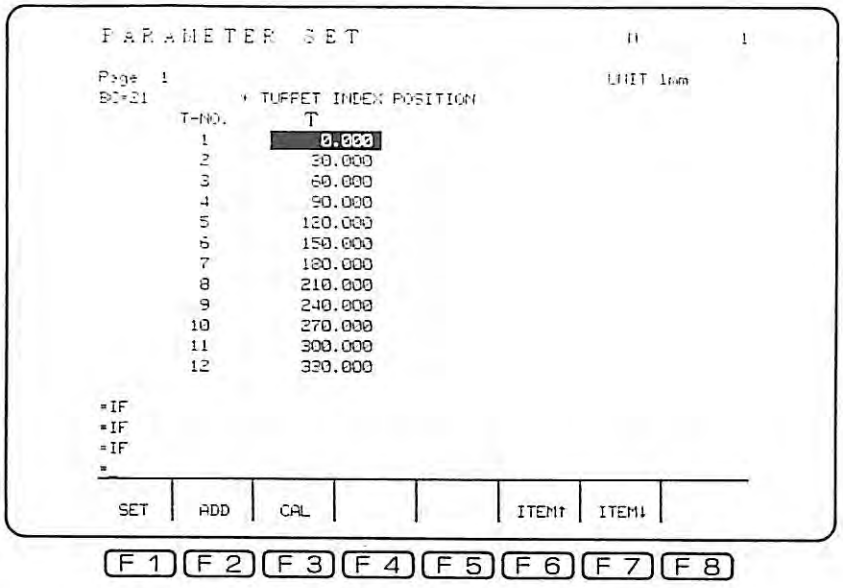
2) Move the cursor to the data of the parameter number to be changed or set.

3) Set the new parametric data using function key [F1] (SET) and the keyboard.

3-7. Turret Indexing Angle

With the LB15-Ⅱ, the turret indexing angle can be set using the turret indexing angle setting screen. Since a turret indexing angle is initially set by Okuma prior to the shipment, do not attempt to change it.

Note: The set value is compensated and a new value is set each time the turret is clamped. (clamp position compensation function)



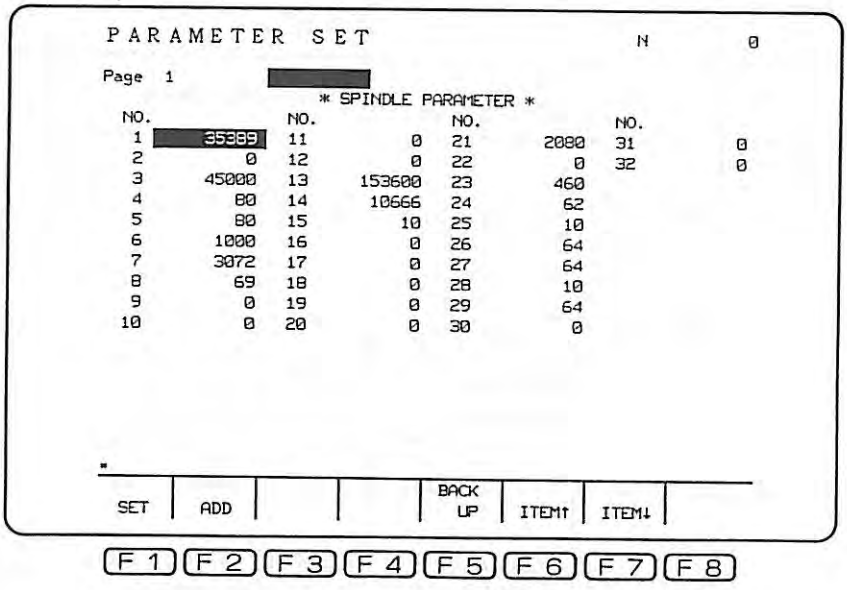
3-8. Spindle Control Parameter

The spindle control parameter setting screen is provided for machine models equipped with the high-precision VAC spindle motor. Prior to the shipment from the Okuma, optimal valves are initially set at spindle control parameters.

Do not change the initial values.

Note 1: *The spindle control parameter becomes effective when the power has been once turned off and then back on again.*

Note 2: *The number of parameter sets is equal to the number of gear states and winding states. A spindle control parameter screen is provided for each parameter set. Press* [P↑] *and* [P↓] *keys to change the page.*



Spindle Control Parameter List

No.	Contents	Effective Command	Setting Range	Use	Data Format
1	Spindle control acceleration/deceleration unit amount	SET, ADD	0 to 4294967295	Acceleration/deceleration processing for the spindle control mode	0.001°/12.8 msec or 0.0001°/12.8 msec
2	Spindle index starting angle (θ1)	SET, ADD	1 to 4294967295	Spindle orientation	0.001° or 0.0001°
3	Spindle deceleration starting angle (θ2)	SET, ADD	1 to 4294967295	Spindle orientation	0.001° or 0.0001°
4	Spindle creep starting angle (θ3)	SET, ADD	1 to 4294967295	Spindle orientation	0.001° or 0.0001°
5	Spindle index complete angle (θ4)	SET, ADD	1 to 4294967295	Spindle orientation	0.001° or 0.0001°
6	Spindle position loop gain setting angle after spindle index (θ5)	SET, ADD	0 to 4294967295	Spindle orientation	0.001° or 0.0001°
7	Spindle index starting velocity (V2)	SET, ADD	−2147483648 to 2147483647	Spindle orientation	0.001°/12.8 msec or 0.0001°/12.8 msec
8	Spindle creeping velocity (V3)	SET, ADD	Spindle orientation	0.001°/12.8 msec or 0.0001°/12.8 msec	
9					
10					
11	Backlash compensation amount	SET, ADD	−2147483648 to 2147483647	Backlash compensation for position control	0.001° or 0.0001°
12	Lost motion compensation amount	SET, ADD	0 to 4294967295	Lost motion compensation for the C-axis	0.001° or 0.0001°
13	Ftp maximum	SET, ADD	−2147483648 to 2147483647	Tolerance control	0.001°/12.8 msec or 0.0001°/12.8 msec
14	Ft1 maximum	SET, ADD	−2147483648 to 2147483647	Tolerance control	0.001°/12.8 msec or 0.0001°/12.8 msec
15	Fti minimum	SET, ADD	−2147483648 to 2147483647	Tolerance control	0.001°/12.8 msec or 0.0001°/12.8 msec
16					
17					
18					
19					
20	Attachment backlash compensation amount	SET, ADD	−2147483648 to 2147483647	0.001° or 0.0001°	

No.	Contents	Effective Command	Setting Range	Use	Data Format
21	KV value	SET, ADD	1 to 65535	Position control	52 × KV value
22	Torque feed forward amount	SET, ADD	0 to 2047	Torque feed forward for the C-axis	
23	Tp maximum	SET, ADD	1 to 256	Tolerance control	6.4 msec × n
24	T1 maximum	SET, ADD	1 to 256	Tolerance control	6.4 msec × n
25	T1 minimum	SET, ADD	1 to 256	Tolerance control	6.4 msec × n
26	T2 maximum	SET, ADD	1 to 128	Tolerance control	0.8 msec × n
27	T2 minimum	SET, ADD	1 to 128	Tolerance control	0.8 msec × n
28	Time constant for linear acceleration (1st step) T1	SET, ADD	1 to 256	Acceleration/deceleration control for C-axis control	6.4 msec × n
29	Time constant for linear acceleration (2nd step) T2	SET, ADD	1 to 128	Acceleration/deceleration control for C-axis control	0.8 msec × n
30					
31	CH1 number	SET, ADD	—	Analog voltage monitor for service and on-site adjustment	
32	CH2 number	SET, ADD	—	Analog voltage monitor for service and on-site adjustment	

LIST OF PUBLICATIONS

Publication No.	Date	Edition	
3272-E-R1	September 1990	7th	A613*
3272-E-R2	July 1991	8th	
3272-E-R3	November 1991	10th	A614*
3272-E-R4	January 1993	11th	

This manual may be at variance with the actual product due to specification or design changes.
Please also note that specifications are subject to change without notice.
If you require clarification or further explanation of any point in this manual, please contact your OKUMA representative.

9309-06-100 ©