CNC SYSTEMS

OSP5020L
OSP500L-G

SPECIAL FUNCTIONS MANUAL
(No. 2) (6th Edition)

OKUMA

Publication No. 3290-E-R1
(September 1990)

CNC SYSTEMS

OSP5020L
OSP500L-G

SPECIAL FUNCTIONS MANUAL
(No. 2) (6th Edition)

LE51-007-R5

TABLE OF CONTENTS

PAGE

PAGE

PAGE

PAGE

PAGE

PAGE

PAGE

PAGE

SECTION 1 PROGRAMMABLE TAILSTOCK FUNCTION (Tow-Along System)

1. Overview

The tailstock body can be moved and positioned using the motion of the saddle (ZA-axis), not requiring a tailstock body drive axis. The tailstock body is connected to the saddle with the joint pin. See Fig. 1-1.

When the tailstock body is connected to the saddle, it is unclamped automatically, allowing positioning. Once the joint pin retracts to disconnect the tailstock body from the saddle, it is clamped against the guideway. These operations, disconnection, and unclamp, are all confirmed by the limit switches LS1, LS2 and LS3, respectively.

After the tailstock has been connected, it is towed by the saddle (ZA-axis) according to the W-axis command and is positioned to the commanded position. Tailstock motion is defined as W-axis motion, and it is controlled by the position encoder and the axis drive motor for the ZA-axis only while the tailstock body is connected with the saddle. The machine origin of the tailstock body (W-axis) is at the same point as the machine origin of the ZA-axis as shown in Fig. 1-1; the independent program zero for the W-axis is also available to simplify programming.

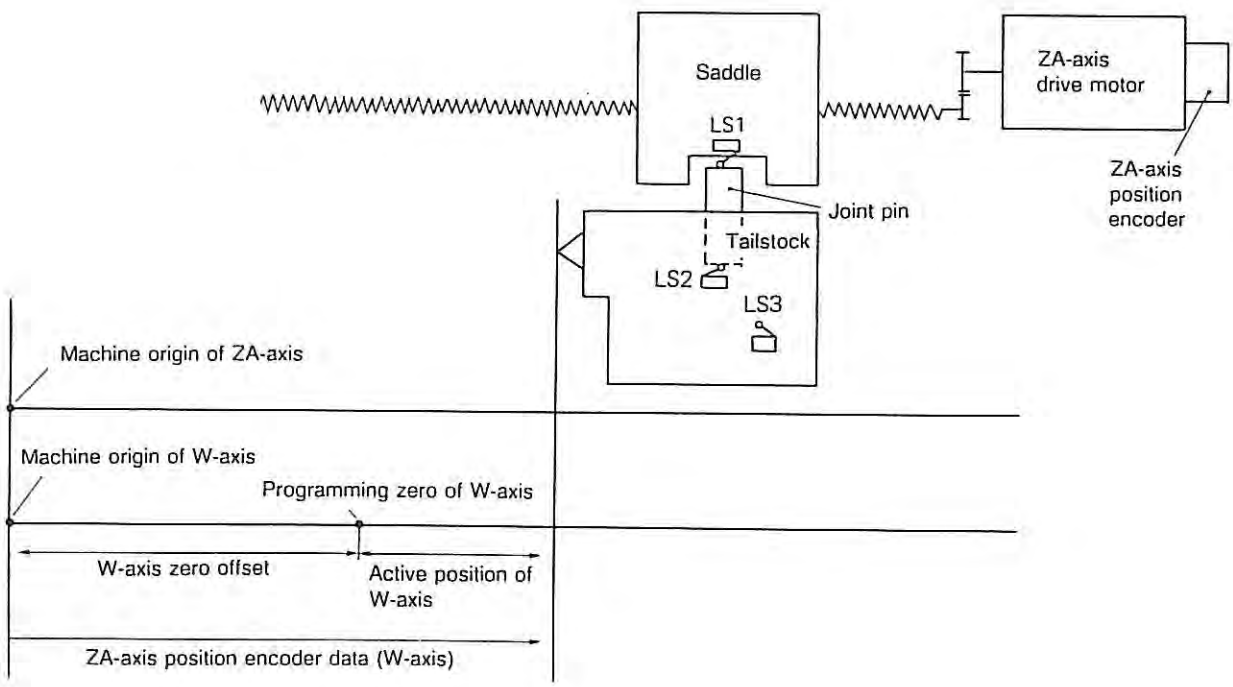


Fig. 1-1

2. Setting Initial Position

Set the initial position for tailstock positioning when connecting the tailstock body with the saddle for the first time, after loading the control software, or when the position encoder of the ZA-axis has been replaced. This setting is necessary for storing the active position data of the tailstock body to the NC memory.

Take the following steps:

(1) Move the X-axis to the travel end to avoid interferences during setting.

(2) Retract the tailstock spindle fully. Keep the spindle at a still.

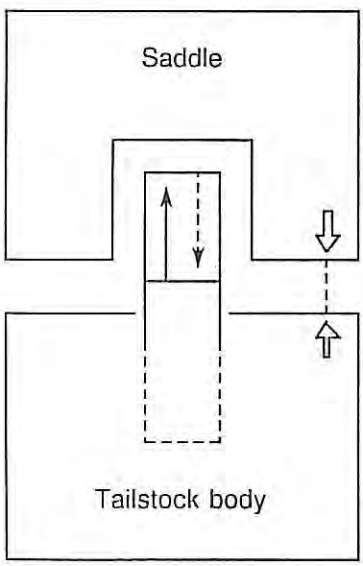
(3) Feed the ZA-axis to the position where the tailstock body can be connected with the saddle. Match marks are provided on the saddle and the tailstock body for visual confirmation of joint position.

(4) After selecting the MANUAL mode, press the **TAILSTOCK UNCLAMP/JOINT ON** pushbutton switch to connect the tailstock body with the saddle. When it has been connected, the TAILSTOCK UNCLAMP/JOINT ON indicating lamp on the operation panel illuminates.

(5) Move the ZA-axis in the MANUAL mode to tow the tailstock body.

(6) Press the **TAILSTOCK CLAMP/JOINT OFF** pushbutton switch. When the tailstock body is disconnected from the saddle, the TAILSTOCK CLAMP/JOINT OFF indicating lamp illuminates.

This completes the setting of the initial position data.



Note: When connecting or disconnecting the tailstock to or from the saddle manually, align the match marks.

Fig. 1-2

3. Manual Positioning of Tailstock Body

See Fig. 1-6.

To position the tailstock body manually, follow the steps below:

(1) Move the X-axis to the travel end position in the positive direction.

(2) Retract the tailstock spindle fully. Keep the spindle at a still.

(3) Press the **JOINT POSITION RETURN** pushbutton switch on the operation panel. This feeds the saddle to the joint position at the rapid feedrate.

(4) Press the **TAILSTOCK UNCLAMP/JOINT ON** pushbutton switch. The tailstock body is connected to the saddle and unclamped with the TAILSTOCK UNCLAMP/JOINT ON indicating lamp illuminates.

(5) Manually feed the Z-axis to bring the tailstock body to the desired position.

100% setting; 2.4 m/min

(6) Move the saddle in the opposite direction as much as a half of the backlash amount to ease disconnection of the tailstock body form the saddle. (Movement of approx. 1 mm will be sufficient.)

(7) Press the **TAILSTOCK CLAMP/JOINT OFF** pushbutton switch on the operation panel.

The tailstock body is disconnected from the saddle and then clamped. Check to be sure that the TAILSTOCK CLAMP/JOINT OFF indicating lamp on the operation panel is on.

Note: *Unit amount of acceleration/deceleration for manual W-axis motion can be changed as desired with a parameter. Refer to 5. "Parameter Setting".*

4. Tailstock Positioning by Programmed Commands

Commands Used in Program:

W . . . . . W-axis command (Incremental commands allowed)
G152 . . . G code calling for tailstock positioning
M188 . . . Joint OFF
M189 . . . Joint ON

These G and M codes are effective only in the commanded block.

Example of Program:

```
N001  G00  X1000  Z200  CR
N002  G152 W100         CR
N003  G00  X____  Z____ CR
  :
  :   Cutting Program
  :
```

Explanation of Each Block:

N001: The X-axis retracts to the travel end position.

N002: Tailstock positioning cycle (1) through (5) is performed.

(1) The Z-axis moves to the position where the tailstock body was previously connected to the saddle at the rapid feedrate.

(2) The tailstock body is connected to the saddle. In two seconds after the tailstock body joint confirmation, the saddle starts movements to tow the tailstock body to W100 position at 2.4 m/min.

(3) The saddle moves so that the joint pin is located at the center of the joint section.

(4) The tailstock body is disconnected from the saddle and the tailstock body is clamped.

(5) The Z-axis returns to the position where the Z-axis was located before the execution of the tailstock positioning cycle. (Z200 in N001, in this case).

In the single block mode, cycles (1) through (5) are executed as one block command.

N003: Cutting program follows.

CAUTION

(1) The W-axis movement command must be specified with G152 in the same block as G152 W100.

(2) Conditions for W-axis command execution are:

- *The X-axis is at the travel end in the positive direction. (For two-saddle models, both the XA-axis and the XB-axis must be at the travel end in the positive direction.)*
- *The tailstock spindle is at the retract end.*
- *The spindle is at a still.*

(3) The W-axis command cannot be designated in the LAP or nose radius compensation mode.

(4) M codes commanding joint ON/OFF, M188 and M189, are automatically generated by the control during the execution of the G152 cycle and it is not necessary to designate those M codes.

(5) To move the Z-axis in the G00 mode after the completion of tailstock joint designated by M189, designate M190 in the same block as G00. If M190 is designated in the G00 block, an alarm will be generated.

(6) Axis feedrate in the G00 mode with the tailstock jointed is the same as the W-axis feedrate.

(7) Axis movement designated by G codes other than G152 and G00 is not possible when the tailstock is jointed.

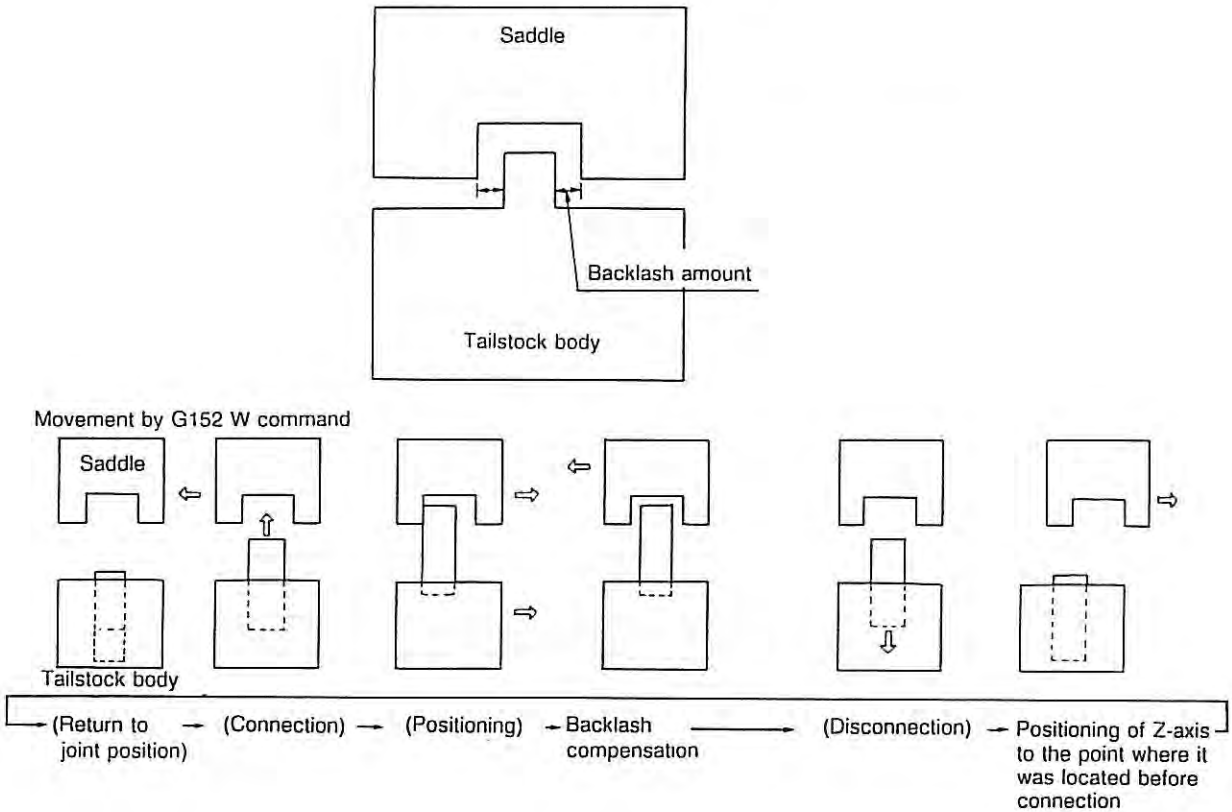


Fig. 1-3

5. Parameter Setting

(1) Soft Limit and Travel End Limit

On the W-axis, soft limit and travel end limit are set both in the positive and negative directions as on the X- and Z-axis, determining travel range of the W-axis. See Fig. 1-4.

They can be set in the same manner as setting those limits of the X- and Z-axis.

As with the X- and Z-axis, soft limits of the W-axis cannot set outside the travel end limit.

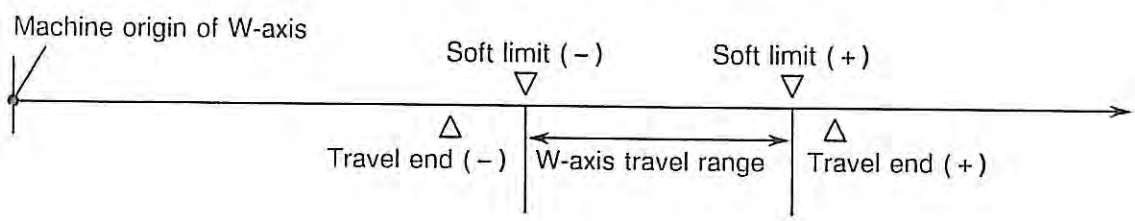
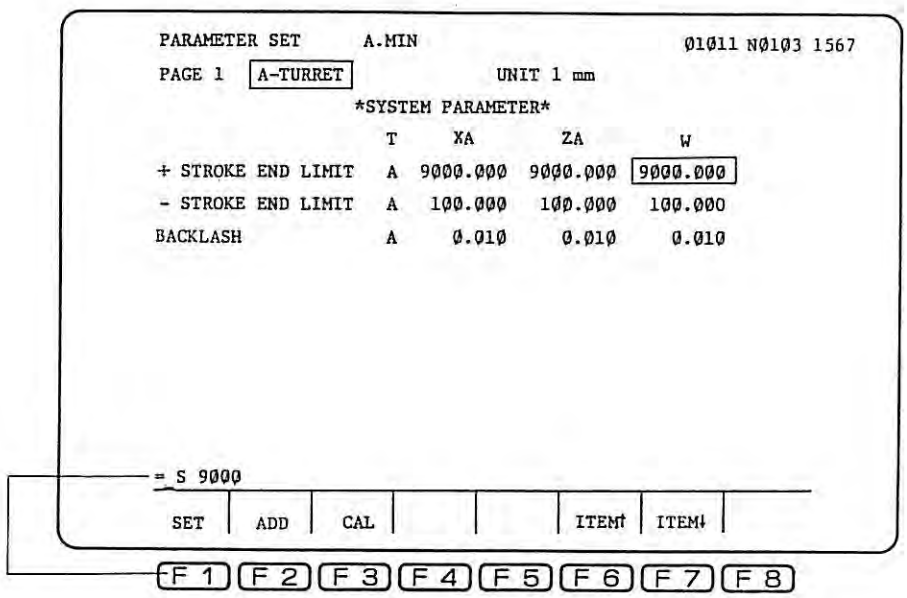


Fig. 1-4

Note: Set the soft limits of W-axis inside those of ZA-axis.

a) Procedure for setting travel end limit



1) Display the SYSTEM PARAMETER page of the PARAMETER SET mode pages.

   For two-saddle models, press the **A** key on the NC operation panel to select A-turret.

2) Move the cursor to W data of +STROKE END LIMIT and −STROKE END LIMIT.

3) Press the function key [F1](SET) and set the STROKE END LIMIT values of W-axis noted on the NC management data card at the back of the tape reader box.

4) If the values to be set are not recorded on the card, first set the actual position of the axes, which are known from the position display data, at the parameters STROKE END LIMIT.

5) Measure the movable distances of the W-axis both in the positive and negative directions from the actual position.

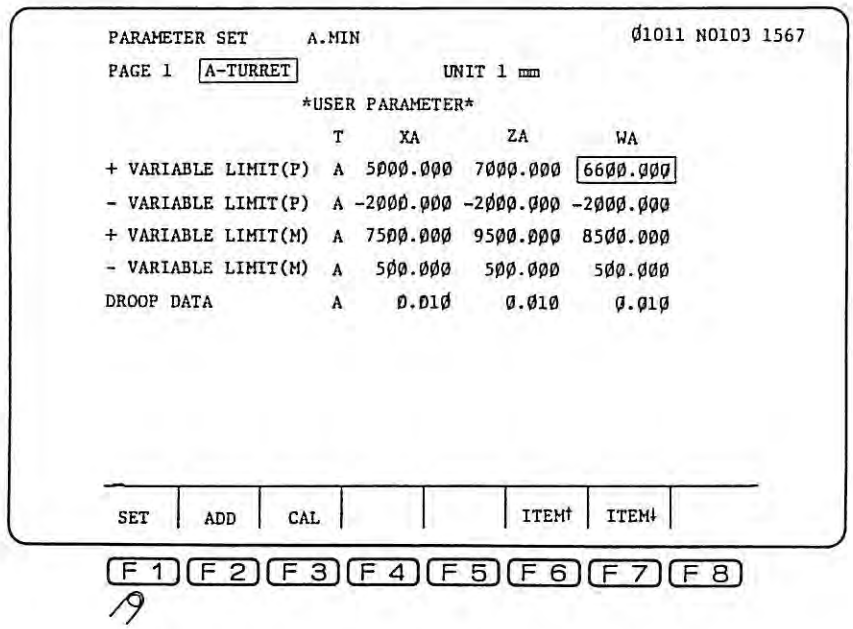
6) Set the measured distances using the function key [F2] (ADD).

   Movable distance in the positive direction : [F2] and (plus value)

   Movable distance in the negative direction : [F2] and (minus value)

Note: *If travel end values must be changed, contact your local Okuma service representative before changing the data.*

b) Procedure for setting soft limit (variable limit)



1) Display the USER PARAMETER page of the PARAMETER SET mode pages.

   For two-saddle models, press the **A** key on the NC operation panel to select A-turret.

2) Move the cursor to W data of parameters +/– variable limit (P/M).

3) After pressing the function key [F1] (SET), key in numerical values through the keyboard.

4) Press the **WRITE** key.

Note 1: *The operations indicated below require the setting of both travel end limit and soft limit. If the setting is not made, the W-axis will fail to move or its movement is limited within a certain range.*

- *Loading of NC control software No. 3*
- *Replacement of memory board*
- *Replacement of OSP position encoder of Z-axis*

Note 2: *When the travel end limit data is changed, the soft limit data is also changed at the same values.*

(2) Backlash

As shown in Fig. 1-5, there is a backlash between the saddle and the tailstock body for joint. Therefore, it is necessary to preset the backlash amount to insert the joint pin at the center of the joint hole correctly and also to eliminate positioning error caused by different joint movement direction.

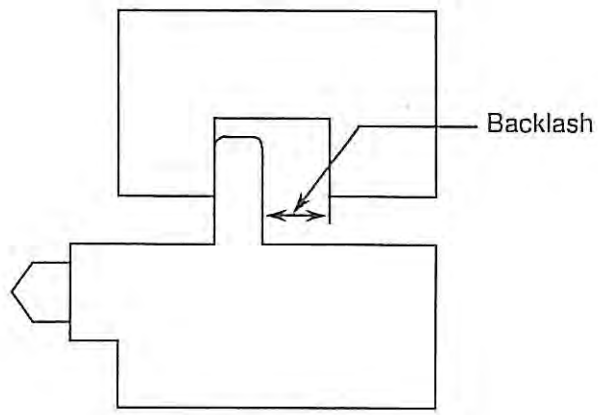
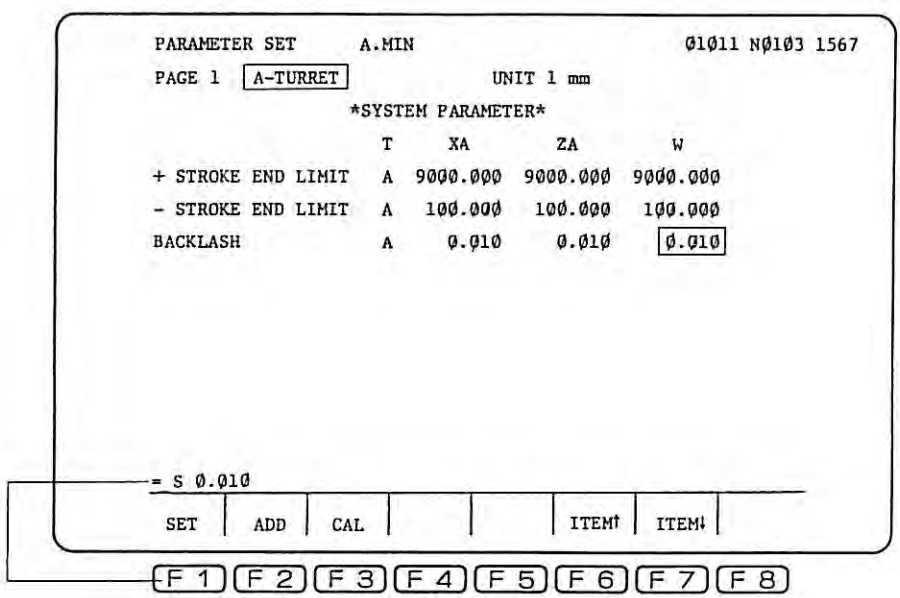


Fig. 1-5

Backlash amount must be set within a range of 0 mm to 10 mm in units of 1 μm. The loading of the NC control software clears the setting to 0 mm.

The procedure for setting the backlash amount is explained below.



1) Display the SYSTEM PARAMETER page of the PARAMETER SET mode pages.

   For two-saddle models, press the **A** key on the NC operation panel to select A-turret.

2) Move the cursor to W data of the parameter BACKLASH.

3) After pressing the function key [F1] (SET), key in the backlash data and then press the **WRITE** key.

(3) Feed Unit Amount and Acceleration Unit Amount for Manual W-axis Movement

Use the system parameters to set the unit amount and acceleration unit amount of manual feed or rapid feed.

Feedrate is determined by the designated unit amount. The relationship is shown in the expression described below.

$$\text{Unit amount } (\mu\text{m}/12.8 \text{ msec}) = \frac{X \text{ (m/min)} \times 10^6}{60 \times 10^3 \text{ (msec)}} \times 12.8 \text{ (msec)}$$

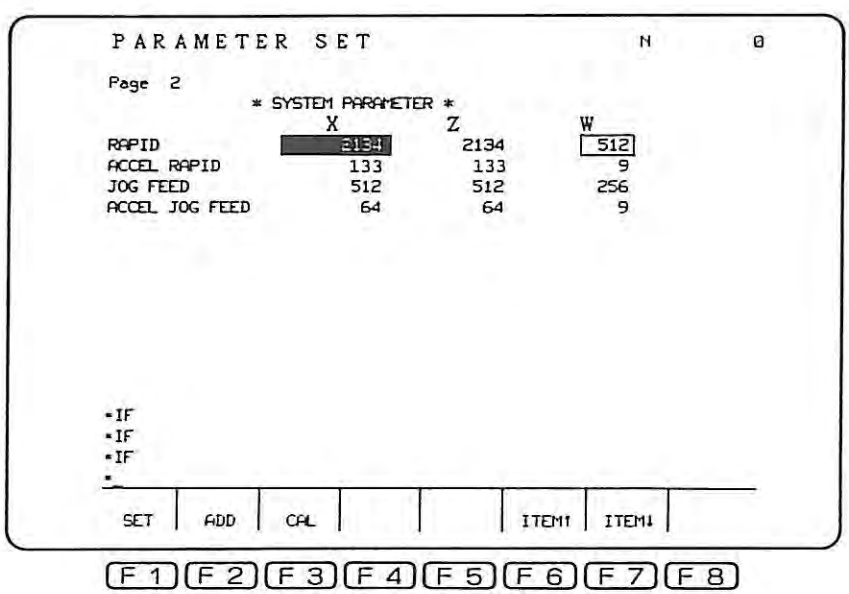
"X" equals feedrate and its unit is m/min.

When starting or ending feeding, automatic acceleration and deceleration are carried out and the acceleration/deceleration time is determined by the acceleration unit amount. Their relationship is as follows:

$$\text{Acceleration/deceleration time (msec)} = \frac{\text{Unit amount}}{\text{Acceleration unit amount}} \times 12.8 \text{ (msec)}$$

Note: *For the attempt to change these values, indications of Okuma must be obtained. If they are changed without our technical support, the axis feed systems might be damaged.*

The following is the setting method:



1) Display the SYSTEM PARAMETER page of the PARAMETER SET mode page.
2) Move the cursor to the desired position on the W column.
3) Press the function key [F1] (SET) and key in the value through the keyboard. Then press the **WRITE** key.

(4) Cancellation of Interlock on the XB-axis Limit

With two-saddle models, it is impossible to move the tailstock when the XB-axis is not at the positive limit position. However, this interlock can be canceled by setting proper parameter data.

Bit 4 of parameter (bit) data No. 19 (initial value 0)

0 ··· Interlock valid
1 ··· Interlock cancel

6. System Parameters

System variables indicated below are used for setting and reading out of zero offset, and plus and minus soft limit of W-axis. (These variables are available when user task 2 function is selected.)

(1) <VZOFW> ··· Zero OFfset of W-axis

When setting a parameter, designate as VZOFW = 1234.567

(2) <VPVLW> ··· PositiVe Limit on W-axis

When setting a parameter, designate as VPVLW = 2000.000

(3) <VNVLW> ··· NegatiVe Limit on W-axis

When setting a parameter, designate as VNVLW = 100.000

Note 1: *Numerical data of these parameters are referenced to the origin of the machine coordinate system (machine origin).*

Note 2: *For two-saddle models, these parameters can be designated only when the G13 mode (A-turret) is established.*

7. Alarm Messages

ALARM-A

141 Tow-along tailstock connect

G00, G01, G02, G03, G30, G31, G32, G33, G34 or G35 command is designated while the tailstock body is connected.

Index	:	None
Character-string	:	None
Code	:	None

142 Tow-along tailstock condition

M188 (Tailstock disconnect), or M189 (Tailstock connect) is designated with G152 calling for tailstock towing mode while the spindle speed is zero or the spindle is at a still.

M189 or G152 is designated while the spindle rotation command is active.

M188, M189, or G152 is designated while the center work is not elected.

M188, M189, or G152 is designated while the tailstock spindle is not at the retract end, or the turrets (both A-turret and B-turret) are not at the travel end in the positive direction.

Index : None

Character-string : None

Code :

1	M188, M189, or G152 is designated while the spindle is not in zero speed.
2	M188, M189, or G152 is designated while the spindle is not at a still.
3	M188, M189, or G152 is designated while the spindle rotation command is active.
4	M188, M189, or G152 is designated while center work is not selected.
5	M188, M189, or G152 is designated while the tailstock spindle is not at the retract end.
6	M188, M189, or G152 is designated while the turrets (both A-turret and B-turret) are not at the travel end in the positive direction.
7	M189 or G152 is designated when either both of joint on and off signals are on or neither joint on nor joint off signal is off.

143 Tow-along tailstock clamp/unclamp

Neither joint ON nor joint OFF signal is derived for more than three seconds.

Both joint ON and joint OFF signals are derived for more than three seconds.

The joint ON signal is derived at other than travel end of X-axis or tailstock spindle retract end.

Index : None

Character-string : None

Code :

None	Neither joint ON nor joint OFF signal is derived for more than three seconds.
3	Both joint ON and joint OFF signals are derived for more than three seconds.
4	Joint ON signal is derived at other than travel end of X-axis.
5	Joint ON signal is derived at other than tailstock spindle retract end.

144 W-axis plus var. limit over

W-axis command exceeding the positive soft limit (variable limit) is designated.

Index : None

Character-string : None

Code : None

145 W-axis minus var. limit over

W-axis command exceeding the negative soft limit (variable limit) is designated.

Index : None

Character-string : None

Code : None

ALARM-B

530 Two-along tailstock movement: condition

G152 (tow-along tailstock positioning cycle) is designated in other than G13 (A-turret) mode.

G152 is designated without a W command.

G152 is designated while tool nose radius compensation or LAP mode is active.

Index : None

Character-string : None

Code :

1 ..... G152 was designated in other than G13 mode.

2 ..... G152 was designated without a W command.

3 ..... G152 was designated during tool nose radius compensation mode.

4 ..... G152 was designated during LAP mode.

531 Two-along tailstock movement: no spec.

G152 was designated although the control has no tow-along tailstock specification.

Index : None

Character-string : None

Code : None

532 W-axis minus var. limit over

A W command exceeding the soft limit (variable limit) in the negative direction is designated.

Index : None

Character-string : None

Code : None

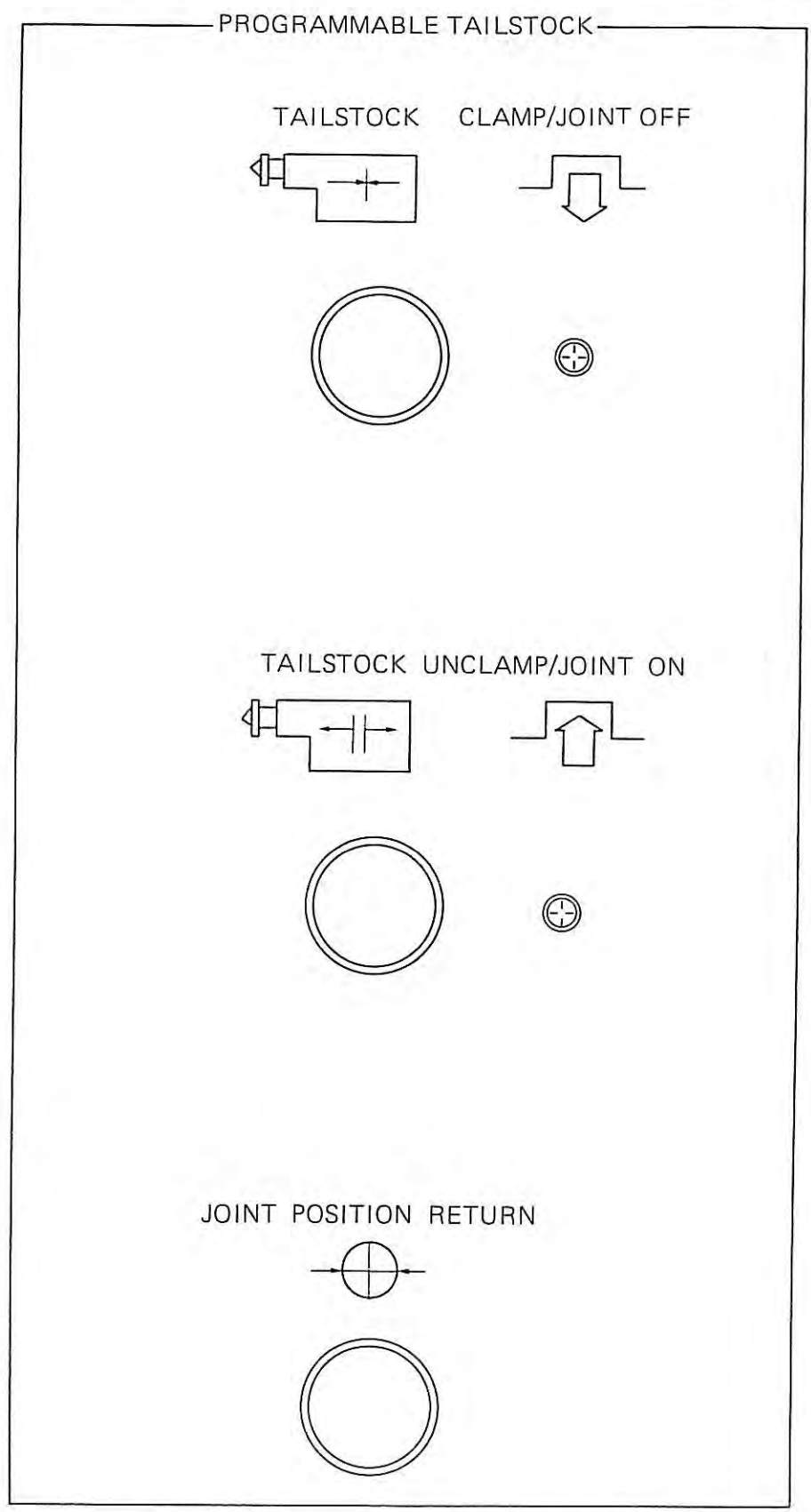


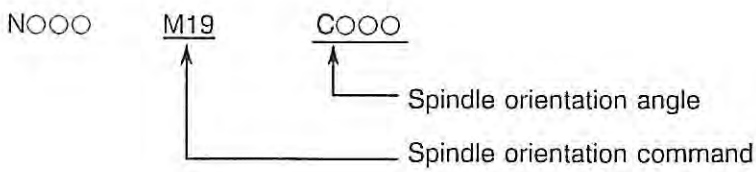
Fig. 1-6 Operation Panel for Programmable Tailstock

SECTION 2 SPINDLE ORIENTATION FUNCTION

1. Overview

This function stops the spindle at a desired angular position specified by a C command preceded by M19.

2. Programming Format



(1) The spindle orientation angle is specified by a C command.

(2) The zero position of the spindle orientation function is set using a parameter.

The zero position is set at the parameter (word) No. 12 in increments of 0.1 deg. Setting range is 0 deg. to 359.9 deg. and initial value is 0 deg. (For models equipped with the VAC Ⅱ, the zero position is set at the parameter (long word) No. 44.)

(3) The angle is measured in the M03 direction taking the marker signal position as a reference position, 0°.

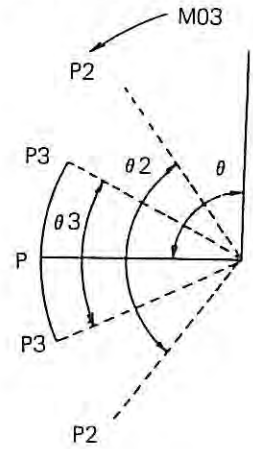
(4) Programmable range of the spindle orientation angle is from 0 deg. to 359.999 deg.

(5) When no C command is designated following M19, the control assumes C command as C = 0° and spindle orientation is made to that angular position.

(6) After the zero position of the spindle orientation function has been established, C commands are referenced to the set zero position.

Note: *For the pin type spindle orientation function, the programmable range of spindle orientation angle is predetermined depending on the machine specifications. In this case, the data of the orientation zero position might be other than 0 deg. and the value specific to individual machine might be set. Therefore, it is necessary to write down the zero position value since loading the NC control software clears the set value to zero.*

3. Contents of Parameters



θ : Spindle orientation angle

θ2: Spindle deceleration starting angle

θ3: Spindle creep rotation starting angle

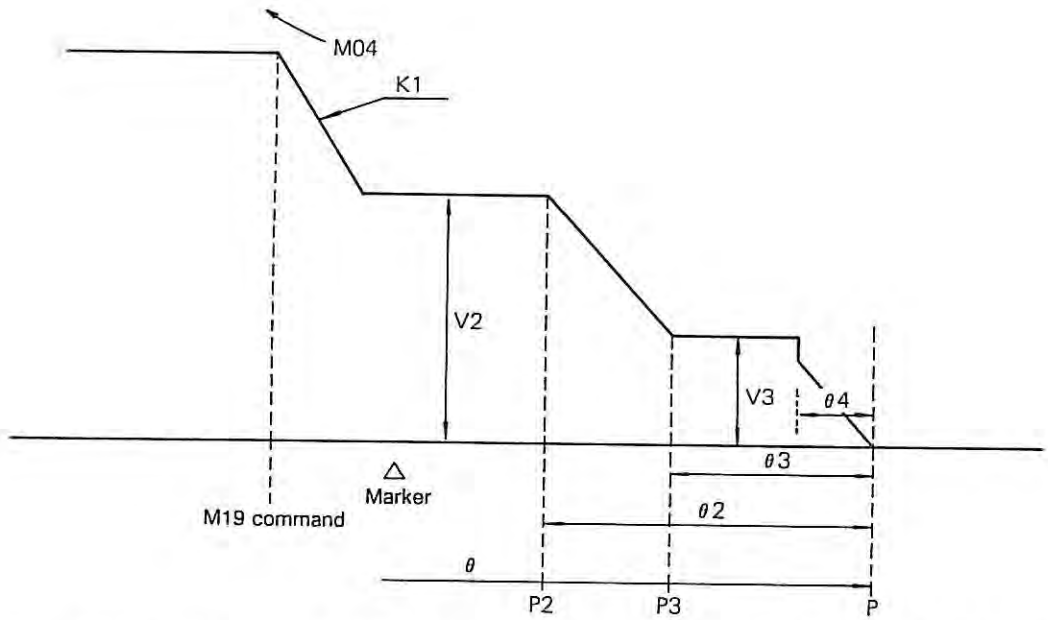
θ4: Angle where the spindle speed becomes zero

V2: Spindle orientation starting speed

V3: Spindle creep speed

K1: Deceleration rate for decelerating spindle speed to V2

P : Designated spindle orientation position



θ1 and T1 in addition to θ2, θ3, θ4, V2, V3, and K1 explained in the illustration above are all set by the parameters for spindle orientation.

θ1: The angle to determine the execution mode of M19 spindle orientation function when M19 is commanded with the spindle at a still. If the present angular position is within the angle set by θ1 from the target orientation angle, the spindle rotates by one turn. (If θ1 = 0, however, the spindle does not rotate by one turn even the actual position and the target position coincide each other.)

T1: Duration of time the control determines until the spindle orientation is completed, after the spindle speed has reached zero.

Note: This value is set by parameter (word) No. 71. (Unit: 10 msec)

θ5: Parameter used to output spindle speed in proportion to positional offset from the target position after the spindle has been brought into range within θ4 from the target position.

4. M19 Command Programmed While Spindle Is at a Still

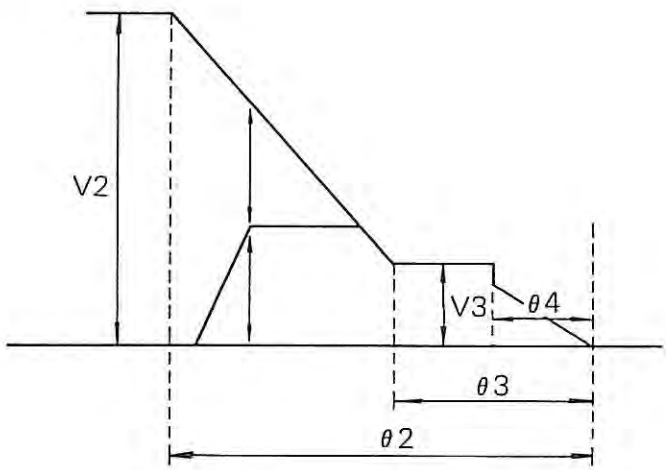
The mode for executing the M19 command depends on the spindle angular position referenced to the orientation target angle.

When present angular position is:

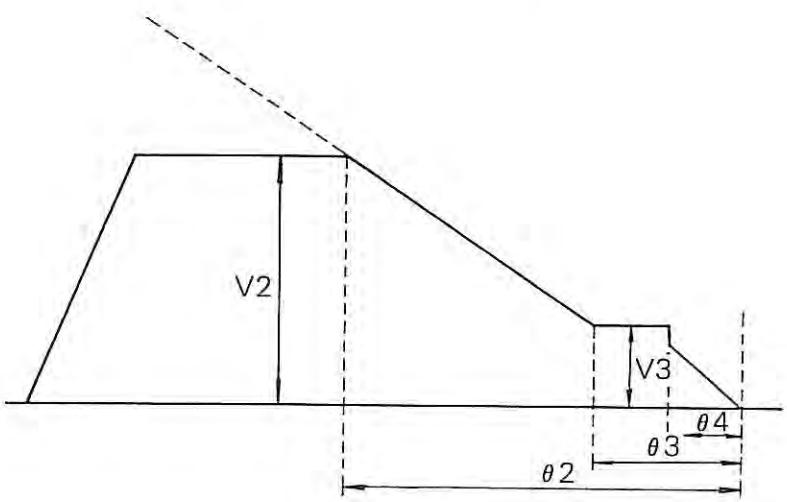
Within $\theta 4$ ··· M19 is completed without the spindle rotation.

Within $\theta 3$ ··· Orientation is accomplished at a creep speed.

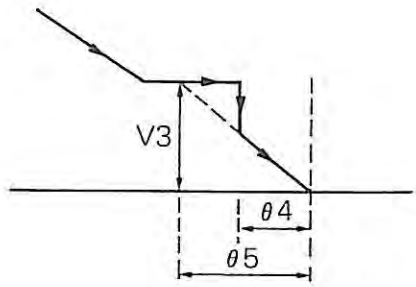
Within $\theta 2$ ··· The spindle is accelerated at a rate K1. When the spindle speed reaches the half the speed determined by present spindle angular position $\theta 2$ and V2, the spindle maintains that speed until the speed drops below the speed determined by $\theta 2$ and V2. After that, the spindle speed changes as in conventional spindle orientation operation.



Over $\theta 2$ ··· The spindle is accelerated at a rate K1. When the half the spindle speed determined by $\theta 2$ and V2 is larger than the speed V2, the spindle rotates at V2. After that, the spindle speed changes as in conventional spindle orientation operation.

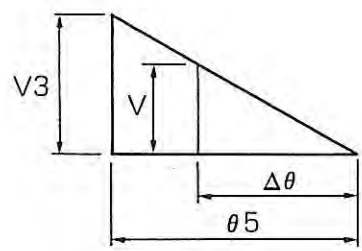


Control after the actual position enters within $\theta 4$ range:



After the spindle has been brought into the range within $\theta 4$ from the target position, positioning is carried out at a spindle speed calculated in the following formula:

$$V = \frac{V3}{\theta 5} \times \triangle\theta \ (\triangle\theta = \text{angle from the target point})$$



5. Data Setting

The screen for setting the parameters stated on the previous page varies depending on the spindle drive unit used on the machine.

1) Parameter Data not Influenced by the Spindle Drive Unit

   Spindle orientation complete timer . . . . . parameter (word) No. 71 (unit: 10 ms)

   Spindle orientation direction . . . . . parameter (bit) No. 7, bits 0 and 1

2) Standard VAC Drive Unit or DC Drive Unit

   Spindle orientation zero point . . . . . parameter (word) No. 12 (unit: 0.1°)

The parameters stated in the previous page must be set for respective spindle speed ranges, M41, M42, M43, and M44.

Set the data on the SPINDLE ORIENTATION page of the PARAMETER SET mode pages.

No.1 ··· M41
No.2 ··· M42
No.3 ··· M43
No.4 ··· M44
— Set K1. (1 - 32767, Unit: rev/sec)

No.5 ··· M41
No.6 ··· M42
No.7 ··· M43
No.8 ··· M44
— Set θ2. (1 - 9999, Unit: 0.1 deg.)

No.9 ··· M41
No.10 ··· M42
No.11 ··· M43
No.12 ··· M44
— Set V2. (1 - 999, Unit: rpm)

No.13 ··· M41
No.14 ··· M42
No.15 ··· M43
No.16 ··· M44
— Set θ3. (1 - 9999, Unit: 0.1 deg.)

No.17 ··· M41
No.18 ··· M42
No.19 ··· M43
No.20 ··· M44
— Set V3. (−99 - 999, Unit: rpm)
If the set value is negative, rpm is calculated by dividing the absolute value of the setting by 10
Example: −5 → 0.5 rpm, 5 → 5 rpm

No.21 ··· M41
No.22 ··· M42
No.23 ··· M43
No.24 ··· M44
— Set θ4. (1 - 9999, Unit: 0.1 deg.)

No.25 ··· M41
No.26 ··· M42
No.27 ··· M43
No.28 ··· M44
— Set θ5. (0 - 9999, Unit: 0.1 deg.)

No.29 ··· M41
No.30 ··· M42
No.31 ··· M43
No.32 ··· M44
— Set θ1. (0 - 9999, Unit: 0.1 deg.)

3) VACII Drive Unit

Spindle orientation zero point . . . . . parameter (long word) No. 44 (unit: 0.001°)

The screen page for setting the parameters other than the above varies depending on the selected spindle speed range as indicated below.

M41 1st page

M42 2nd page

M43 3rd page

M44 4th page

When the data has been set, first turn off the power supply and then turn it back on again. The set data becomes effective.

Parameter No.	Contents	Setting Range	Setting Unit	Remarks
1	K1	0 - 99999999	0.001°/12.8 msec^2	0.0001°/12.8 msec^2 (0.1 μm specification)
2	θ1	1 - 99999999	0.001°	0.0001° (0.1 μm specification)
3	θ2	1 - 99999999	0.001°	0.0001° (0.1 μm specification)
4	θ3	1 - 99999999	0.001°	0.0001° (0.1 μm specification)
5	θ4	1 - 99999999	0.001°	0.0001° (0.1 μm specification)
6	θ5	0 - 99999999	0.001°	0.0001° (0.1 μm specification)
7	V2	0 - 99999999	0.001°/12.8 msec	0.0001°/12.8 msec (0.1 μm specification)
8	V3	0 - 99999999	0.001°/12.8 msec	0.0001°/12.8 msec (0.1 μm specification)
11	Backlash compensation amount	—1000 - 1000	0.001°	0.0001° (0.1 μm specification)

Do not change the data other than those listed above.

Setting Examples:

a) Spindle acceleration/deceleration unit amount K1

To stop the spindle which is rotating at "S" rpm in "T" seconds, the time K1 is obtained using the following equation.

$$K1 = \frac{983.04\ S[rpm]}{T[sec]}$$

For example, to stop the spindle which is rotating at 1,000 rpm in 0.5 seconds,

$$K1 = 983.04 \times \frac{1{,}000}{0.5} = 1{,}966{,}080$$

b) V2 or V3 speed

When the spindle speed is "S", V2 or V3 can be obtained using the following equation.

$$V2 \text{ or } V3 = 76.8S$$

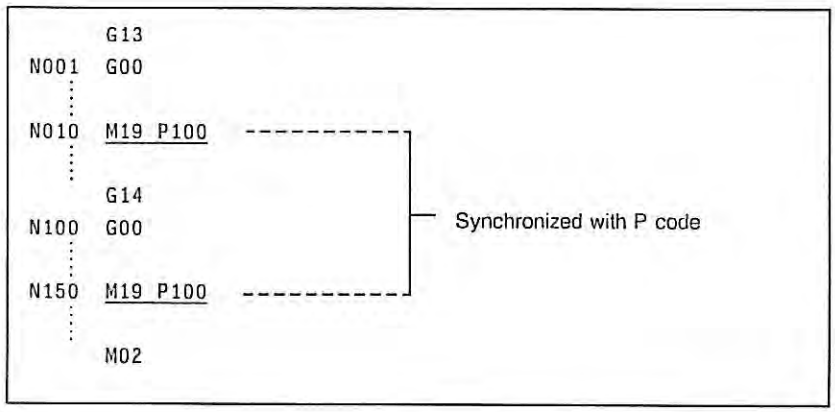
For example, when the spindle speed is 100 rpm,

$$V2 \text{ or } V3 = 76.8 \times 100 = 7{,}680$$

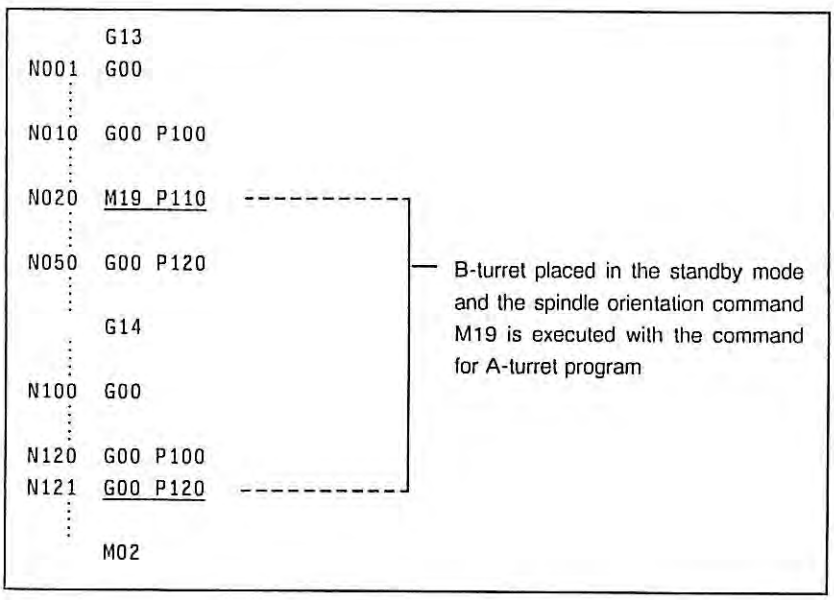
Note: Multiply the values in a) and b) by ten for 0.1 μm control models.

6. Program Examples for Two-Saddle Models

(1) Example 1



(2) Example 2



7. Remarks

(1) When designating M19 while in the simultaneous 4-axis control mode, synchronize the operation of the turrets A and B using a P code.

(2) When sequence return operation is performed passing the block containing M19, spindle orientation is not executed.

(3) If the spindle is rotated in the manual operation intervention mode after the completion of M19, automatic return from the manual operation intervention mode is impossible.

For example 1 in 6. "Examples of Program for Two-Saddle Models", sequence return to sequence N010, or to N150 and later sequences is possible. However, sequence return to sequence N050 or to N121 and later sequences in example 2 is impossible.

SECTION 3 GRAPHIC CRT DISPLAY

1. Overview

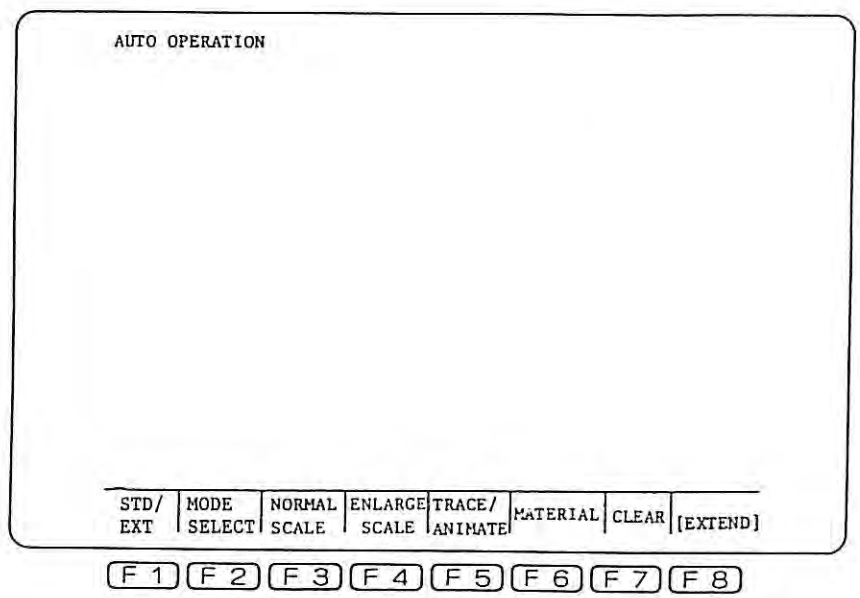
This function displays the progress of the program in animation in addition to the programmed tool paths on CRT.

The OSP5020L has the color graphic CRT on which workpieces, cutting tools, etc. are differentiated by colors.

The OSP500L-G has the monochrome graphic CRT on which workpieces, cutting tools, etc. are differentiated by patterns, line types, and brightness.

2. Function Keys Used for Graphic Display Operation

The graphic display is possible in the auto, MDI and manual operation modes.



(1) F1 (STD/EXT GRAPHIC)

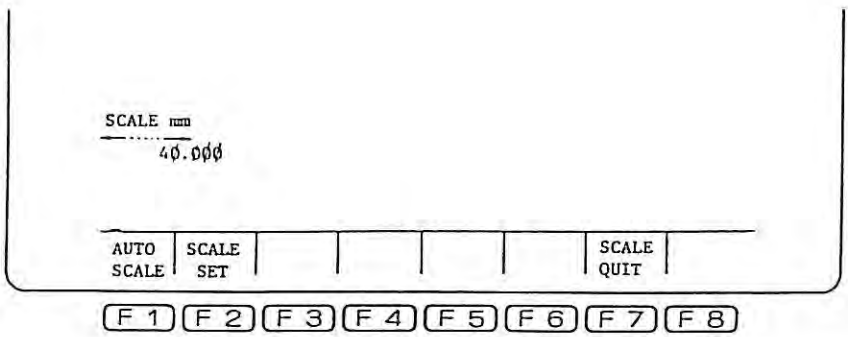
The graphic display mode in the scale set on the NORMAL SCALE function is selected.

Or, the graphic display mode in the scale set on the ENLARGE SCALE function is selected.

Each time this key is pressed, the display mode is switched between normal and enlarged.

Note: *Switching between the standard and enlarged graphic display modes during the execution of a program is impossible.*

(2) F3 (NORMAL SCALE)



This function selects the unit length of an axis on the graphic display. On the graphic display, a dotted line with arrow marks at both ends is displayed with SCALE indication and scale value. This represents the scale length.

The standard scale is set in two different methods such as

a) Automatic determination
b) Arbitrary setting

The procedure for setting the standard scale is explained below. Note that the standard graphic page must be displayed for setting the standard scale.

a) Automatic determination

The graphic scale is automatically determined by pressing the function key [F1] (AUTO SCALE). When this key is pressed, the program is read up to the end of the program (M02) and the scale is determined automatically so that the machining area can be displayed on the CRT.

Remarks:

1) Before executing the automatic determination, select the desired program.
2) If following commands are used in a part program, they are executed when the program is read by the pressing of the function key [F1] (AUTO SCALE).

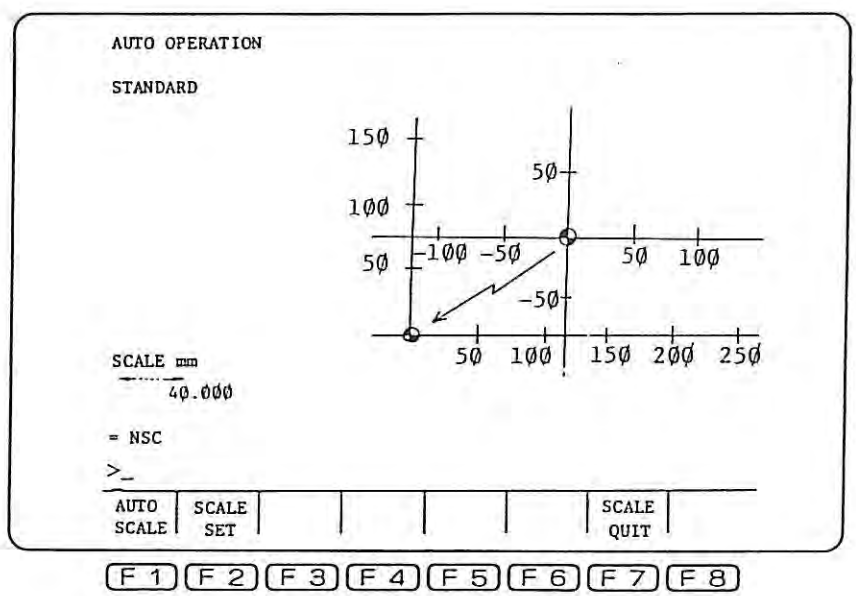
   READ, WRITE, GET, PUT, DELETE, SAVE and DEF

3) If output variables, system variables, and common variables are used in a left member of expression in a part program, they are rewritten by the automatic determination operation.
4) If the scheduled operation mode is selected, automatic determination of the scale value is possible only for the program selected by the first PSELECT command.
5) If an alarm occurs during the reading of a program for automatic determination, or if the program reading is looped due to the statements GOTO, IF and other program execution order designating commands and automatic determination of the scale is not completed, then escape such state by resetting the control.
6) The drawing area automatically determined is calculated from the following formula.

$$\left(\begin{array}{l}\text{Operating area in the}\\ \text{cutting G code mode}\end{array}\right) \times (100 + n)\%$$

Here n is a factor to allow margin in the tool path drawing operation and set by optional parameter (word) No. 51. ($0 \leqq n \leqq 100$)

b) Arbitrary determination

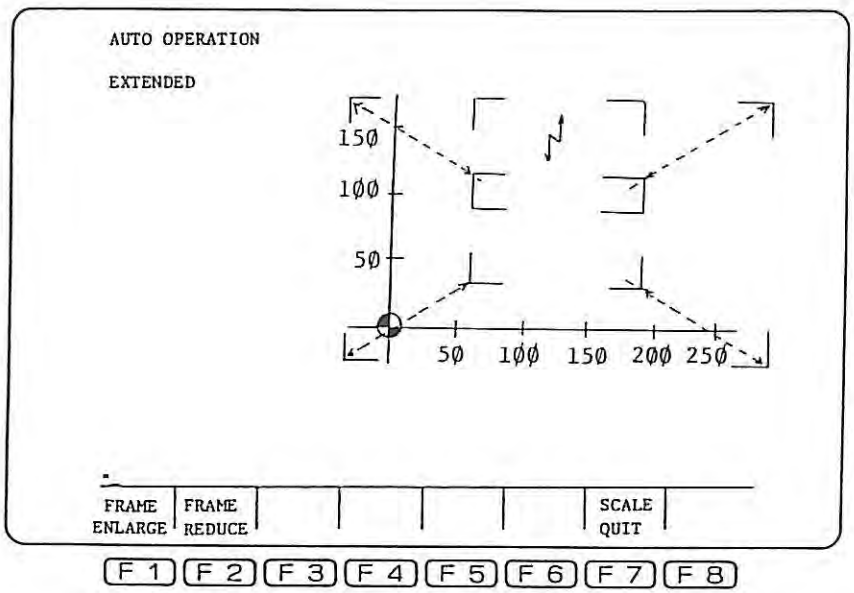


The scale value can be directly entered through the keyboard after pressing the function key [F2] (SCALE SET). In this scale setting, setting range is from 12.5 mm to 1,250 mm.

The position of the coordinate axes can be set at a required position using the cursor after setting the scale value.

The standard scale setting operation ends by pressing the function key [F7] (SCALE QUIT).

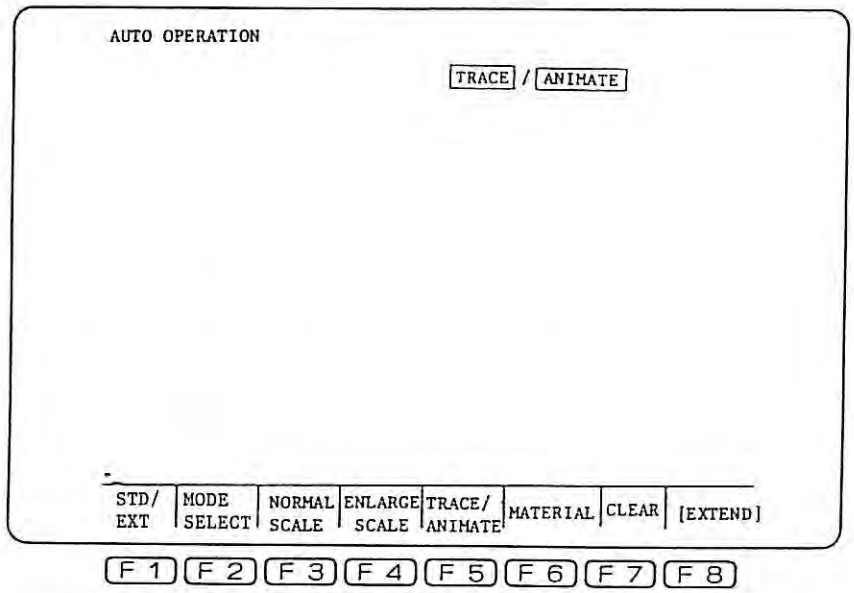
(3) F4 (ENLARGE SCALE)



The display area to be zoomed up from the standard graphic display or enlarged graphic display can be set as needed.

Pressing the function key [F1] (FRAME ENLARGE) reduces the frame indicating the display area. Pressing the function key [F2] (FRAME REDUCE) enlarges the display area indicating frame. Setting the position of the display area frame is made using the cursor key. The enlarge scale setting function terminates when the function key [F7] (SCALE QUIT) is pressed.

(4) F5 (TRACE/ANIMATE)



The graphic display mode is selectable from the three modes indicated below by pressing the function key [F5] (TRACE/ANIMATE). Note that selection of the graphic display mode must be made before starting the operation. The selected mode cannot be changed during the operation.

a) TRACE/ANIMATE mode

In this mode, tool shape, chuck shape, blank shape and tailstock spindle shape are displayed in addition to programmed tool paths. The blank shape is actually removed in accordance with the progress of the part program.

b) TRACE mode

In this mode, only the programmed tool paths are drawn.

c) ANIMATE mode

In this mode, tool shape, chuck shape, blank shape and tailstock spindle shape are displayed and blank shape is actually removed in accordance with the progress of the part program.

Display of the tool paths is not available.

(5) F6 (MATERIAL)

The blank shape, chuck shape and tailstock spindle shape registered are displayed.

(6) F7 (CLEAR)

Tool paths, blank shape, chuck shape and tailstock spindle shape displayed on the CRT are all cleared.

3. General Precautions for Animation Mode Display

(1) Scale Value Setting Range

Setting ranges of the scale value are indicated below:

Normal scale . . . . . 12.5 mm to 1,250 mm
Enlarge scale . . . . . 0.4 mm to 1,250 mm

(2) Relationship between Animation Display and Scale Value

Available animation display differs depending on the scale value setting and the relationship between the set scale value and animation display available is summarized in the table below.

Function / Scale Value	Tool Shape Display	Blank Removal Display	Tool Path Display
1,250 mm - 5 mm	○	○	○
5 mm - 2.5 mm	×	○	○
2.5 mm - 0.4 mm	×	×	○

(3) Animation Drawing Speed

The animation drawing function has a limit in the drawing speed (feedrate of 1 meter per min. will be the maximum drawing speed at a scale value setting of 25 mm).

This means simulation with the animation drawing in the machine lock mode automatically overrides the specified feedrate (animation speed override). In actual machining operation, however, animation speed override is impossible and if program is executed at a feedrate for which animation drawing is impossible, the control displays the message "**ANIM SPEED OVER**" and erases animation drawing. In this case, only programmed paths are displayed on the CRT.

*Note: To clear the message "**ANIM SPEED OVER**", press the function key [F7] (CLEAR) after the part program has been completed.*

(4) Parameter Data for Selecting the Elements of Animation and Tool Paths Drawing

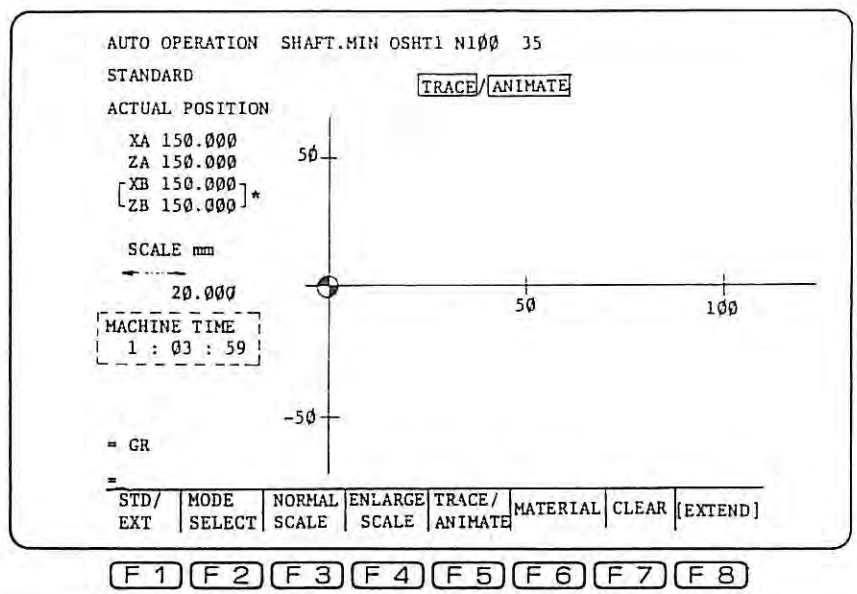
For the display or drawing of animation and tool path, display elements can be selected by a parameter data - optional parameter (bit) No. 10.

Bit	ON/OFF
Bit 0	Jog feed path drawing
Bit 1	Rapid feed path drawing
Bit 2	Cutting feed path drawing
Bit 3	Blank shape drawing
Bit 4	Chuck shape drawing
Bit 5	Tailstock shape drawing
Bit 6	Tool shape drawing
Bit 7	Blank removal drawing

4. High Speed Drawing

Programmed tool paths can be drawn at a high speed in the machine lock and dry run on mode. Setting of the single block function is effective. In this programmed path drawing operation, since the tool path in the G00 rapid feed mode is not the same as the actual movements, this must be taken into consideration.

5. Display Page



*1 Data in [ ] is available for two-saddle models.

*2 Data in [ ] is available for multi-machining specs.

6. Machine Time Display

Actual program execution time in the automatic and MDI mode operation is counted and displayed. While the control is in the slide hole mode or the program stop mode, such time is not counted.

In the high speed drawing mode (machine lock and dry run on), machine time display is not available. If machine time calculation is required through the execution of the high speed drawing operation, optional function for this operation is necessary.

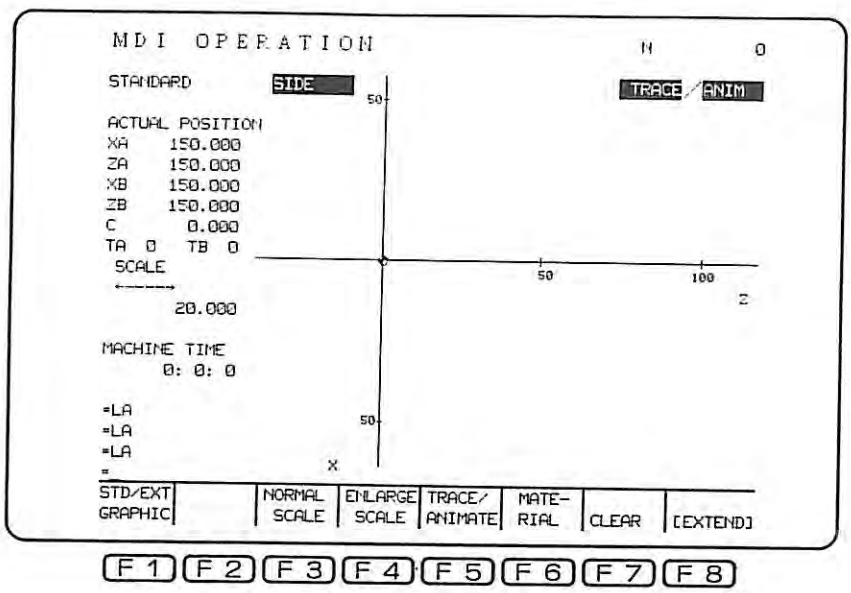
7. Additional Functions for Multi-machining Models

The available function keys are basically the same as used for standard models.

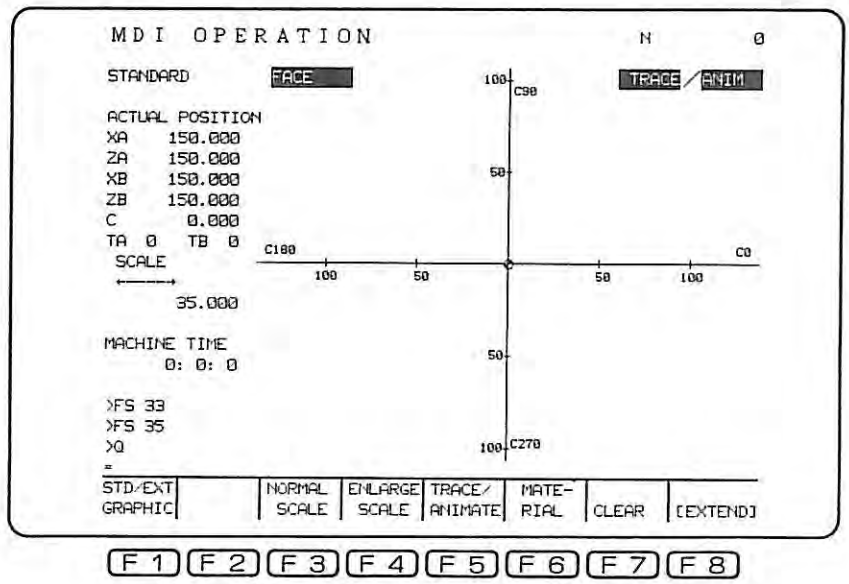
(1) Standard/Enlarged Graphic Display

a) Standard graphic display mode

For the blank display, the page key is used for switching the view angle from the side to the front and from the front to the side. This display change is possible even during the progress of cutting operations. Note that the page key is not effective for other than the multi-machining models.



Side View



Front View

The front view is displayed in the coordinate system on which the C-axis angle is fixed as shown above. The indications on the scale represent X-axis values (in radius).

b) Enlarged graphic display mode

In the enlarged graphic display mode, switching the view angle between the front and side views is possible as in the standard graphic display mode.

(2) Standard Scale Setting

Independent scale setting on the side and front views is possible. For setting the scale for these two views, first display the required view. When the same scale setting is used for both of the displays in common, then the setting is necessary only from either of the displays. In this case, specific scale value is set both for the front and side views.

Setting procedure is the same as used with standard models.

(3) Enlarged Scale Setting

In the enlarged graphic display mode, independent scale setting for the different two displays - front and side. As in the scale setting from the standard display mode, display the required view and setting should be done from that display.

(4) Switching between Trace and Animation Displays

As with the standard models, the function key [F5] (TRACE/ANIMATE) is used for selecting the three different display modes.

a) Trace/Animate

Side View:

Tool shape, chuck shape, blank shape and tailstock spindle shape are displayed. Tool paths are also displayed in cyan and the blank is removed as the progress of cutting.

While the C-axis is connected (only for A-turret tools for two-saddle model), the tool paths are displayed in magenta and removal of blank will not occur.

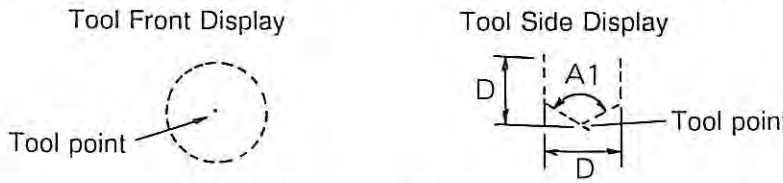
Front View:

The front view of the blank shape is displayed and the tool tip point is indicated by red marker. Tool path is also displayed in magenta. During the cutting, the tool outline is displayed in the specified interval*1. Removal of blank does not occur.

The marker and tool outline are displayed only while the C-axis is connected.

Display of blank shape, tool path and tool outline is dependent on the setting of parameter data explained in 3. (4).

*1: The tool outline is drawn in dots in magenta in accordance with the tool mounting direction, tool diameter (D) and cutting angle (A1) when the revolving tool (M-tool) is selected (tool classification code: 27-38) as the tool shape.



The interval of the tool outline drawing is the distance the tool point has moved from the previous tool outline display position as indicated below:

$$\text{Drill diameter} + \frac{n}{100}$$

"n" represents the factor to designate the tool outline drawing interval and is set at optional parameter (word) No. 55 within a range of 10 to 100.

The tool outline is drawn on the display, in addition to the conditions indicated above, when the feedrate is changed from the rapid traverse to cutting feedrate or the commanded point is reached (with C-axis connected).

b) Trace

Side View:

The display is only tool paths, chuck shape and tailstock spindle shape. As in case a) above, tool path is drawn in magenta while the C-axis is in connection (only for A-turret for two-saddle model) and in cynic in other cases.

Front View:

The tool point position is identified by red marker and tool outline is drawn in magenta in the specified.

c) Animate

Side View:

Tool shape, chuck shape, blank shape and tailstock spindle shape are displayed. Tool path display is not available. Removal of blank occurs in accordance with the progress of cutting. Note that when the C-axis is in connection (only for A-turret for two-saddle model), removal of the blank does not occur.

Front View:

Same as in case a) above, but the tool path is not displayed.

(5) Blank Drawing Function Key

Side View:

Blank shape, chuck shape and tailstock spindle shape registered are displayed.

Front View:

Blank shape registered is displayed.

This function key is effective only for the page currently displayed. That is, pressing this key while the side view is being displayed displays only the side view of the blank and front view of the blank cannot be display. This is the same for the front view display page.

(6) Delete Function Key

Side View:

Tool paths, blank shape, chuck shape, the tailstock spindle shape and tool shape displayed on the side view page are all deleted.

Front View:

The tool outline drawing and the blank shape displayed on the front view page are deleted.

This function key is also effective only for the page currently displayed.

8. Additional Functions for Sub Spindle Models

The available function keys are basically the same as used for standard models.

(1) [F1] (STD/EXT GRAPHIC)

The front and side views of the 1st and 2nd spindles can be displayed in the pattern which is selected by pressing the function key [F2] (MODE SELECT). The front and side views of the 1st and 2nd spindles can also be displayed in the enlarged mode in the pattern which is selected by pressing the function key [F2] (MODE SELECT).

Each time this key is pressed, the display mode is switched between normal and enlarged. Note however, the display mode cannot be switched while machine operation is in progress.

(2) [F2] (MODE SELECT)

The display pattern for the standard/enlarged graphic display in each operation mode can be selected by pressing this function key. Select the display pattern before starting machine operation.
The display mode cannot be switched while machine operation is in progress.

a) Graphic display pattern

Display pattern	Display screen	
1	Side view of the 1st spindle	Front view of the 1st spindle
2	Side view of the 2nd spindle	Front view of the 2nd spindle
3	Side view of the 1st spindle	Side view of the 2nd spindle
4	Front view of the 1st spindle	Front view of the 2nd spindle

In display patterns 1 and 2, switching between the side view and the front view is possible using page keys even during machine operation.

In display patterns 3 and 4, switching between the 1st spindle and the 2nd spindle is not possible using page keys during machine operation. It is switched automatically by GG140/G141 during operation. In this case, the graphic display of the spindle which is selected for the turret selected by the **A** key or **B** key.

b) Display patterns available when the multi-machining specification is not selected

When the multi-machining specification is not selected, patterns 1 through 3 can be selected by pressing the function key [F2] (MODE SELECT). The front views for patterns 1 and 2 cannot be selected using the page keys.

c) Display patterns available when the multi-machining specification is selected only for the main spindle

Patterns 1 through 3 can be selected by pressing the function key [F2] (MODE SELECT). The front view for pattern 2 cannot be selected using the page keys.

d) Display patterns available when the multi-machining specification is selected for both the main and sub spindles

Patterns 1 through 4 can be selected by pressing the function key [F2] (MODE SELECT).

(3) Standard Scale Setting

Independent scale setting on the side and front views is possible. For setting the scale for these two views, first display the required view.

Independent scale setting for the 1st and 2nd spindles is not possible.

When the same scale setting is used for both of the displays in common, then the setting is necessary only from either of the displays. In this case, specific scale value is set both for the front and side views.

Setting procedure is the same as used with standard models.

(4) Enlarged Scale Setting

In the enlarged graphic display mode, independent scale setting for the different two displays - front and side. As in the scale setting from the standard display mode, display the required view and setting should be done from that display.

(5) Switching between Trace and Animation Displays

As with the standard models, the function key [F5] (TRACE/ANIMATE) is used for selecting the three different display modes.

a) Trace/Animate

b) Trace

c) Animate

For details, refer to 2. and 7.

(6) Blank Drawing Function Key

Side View:

Blank shape, chuck shape and tailstock spindle shape registered are displayed.

Front View:

Blank shape registered is displayed.

This function key is effective only for the page currently displayed. That is, pressing this key while the side view is being displayed displays only the side view of the blank and front view of the blank cannot be display. This is the same for the front view display page.

(7) Delete Function Key

Side View:

Tool paths, blank shape, chuck shape, the tailstock spindle shape and tool shape displayed on the side view page are all deleted.

Front View:

The tool outline drawing and the blank shape displayed on the front view page are deleted.

This function key is also effective only for the page currently displayed.

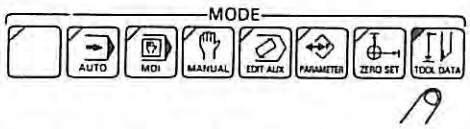
9. Tool Form Selection

The procedure to set the tool form used in animation display is explained below.

The operation to set the tool form means that the data necessary to select the form pattern appropriate to the tool to be used from the tool form patterns already defined.

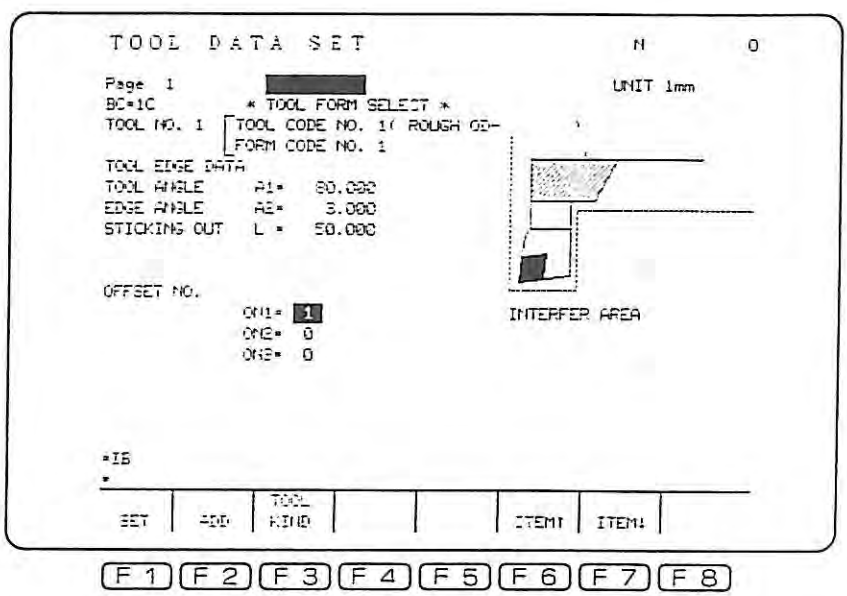
Tool form data must be set in advance for all tools that are used in the program. However, when a tool animation data command (commanded using system variables) is designated in a program such as the program made using the IGF function, the tool form data is automatically set when the program is executed. Thus the setting of tool form data is not necessary in this step.

1) Select the TOOL DATA SET mode.



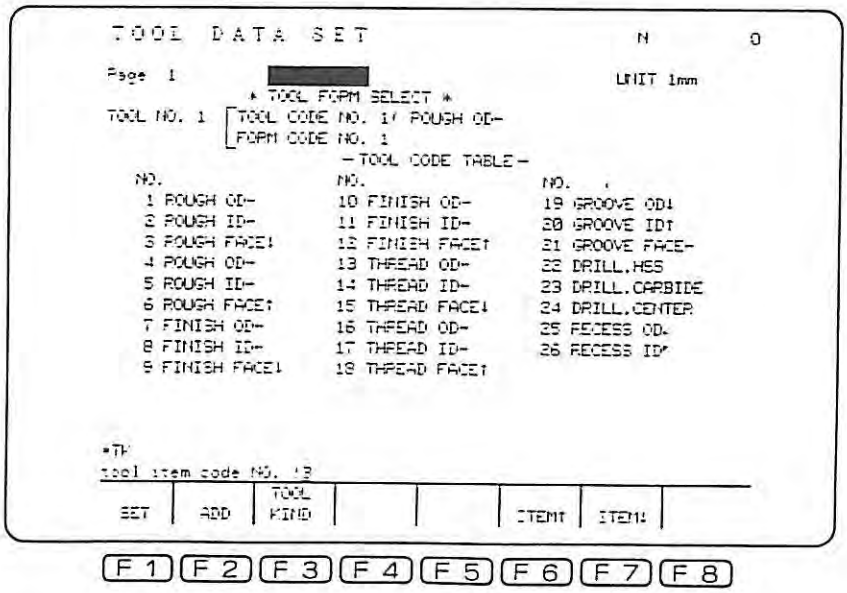
2) Press the function key [F7] (ITEM ↓) to display the TOOL FORM SELECT page.

3) Using the PAGE keys [P↑] [P↓], display the screen for the tool number to be set.



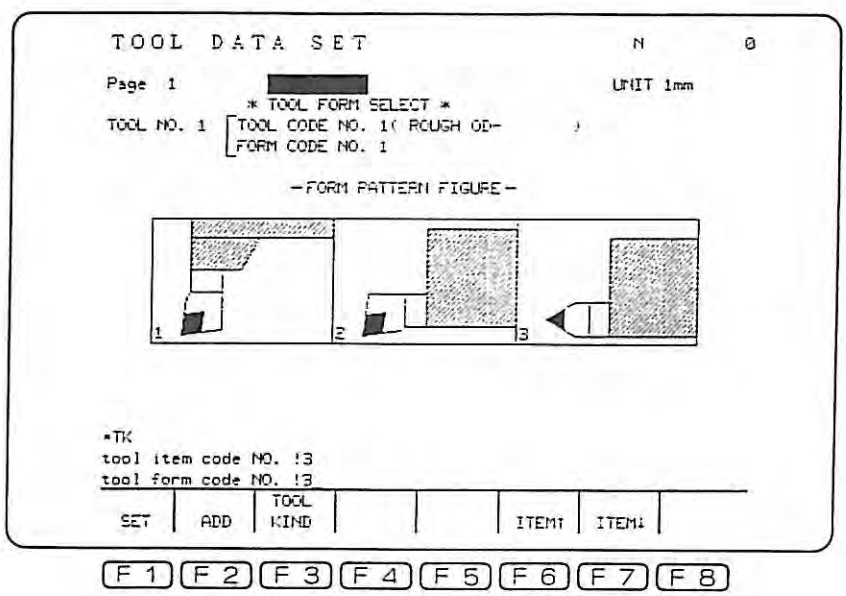
Note: *In the initial state, an OD turning tool is set as the tool form.*

4) Press the function key [F3] (TOOL KIND) to display the page from which the tool code number can be set.



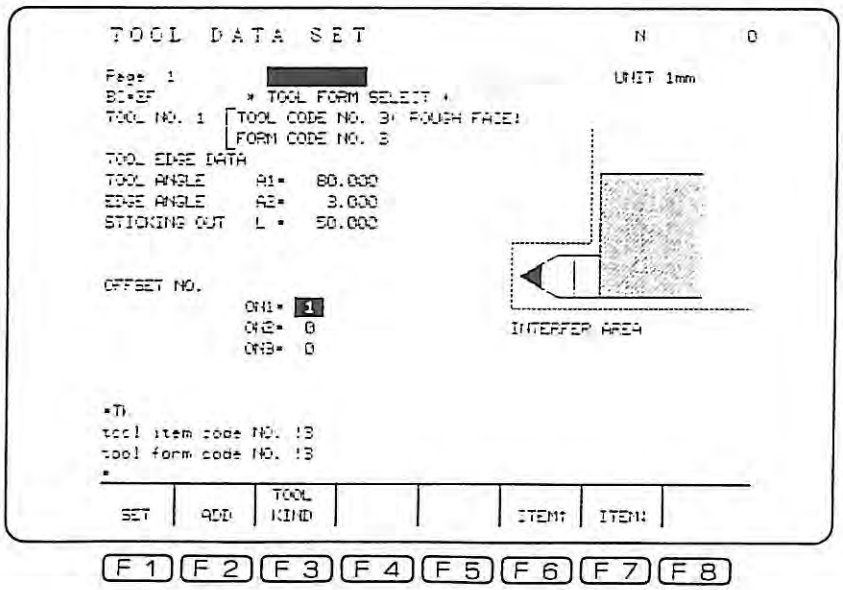
5) Input TOOL CODE NO.

If keys [3] and **WRITE** are pressed, the display indicated below is displayed.



6) Input the FORM CODE NO.

If form code "3" is designated, the display will change into the page as shown below.



7) From this page, set the TOOL EDGE DATA by locating the cursor at the required data position.

TOOL ANGLE A1
EDGE ANGLE A2
STICKING OUT L

Note: *The tool interference area is automatically created according to the selected tool form. For the automatic interference area creation, values registered with parameters are used. If required, change these values.*

8) Input the offset number of this tool by locating the cursor at the required OFFSET NO (ON1 - ON3) data position.

This number is used to designate the tool form for which graphic guide is given on the tool offset setting screen. For the system having the tool life management specification, ON1 - ON3 corresponds to the offset group 1 - 3, respectively. On the tool life management table, tool offset numbers identical to those set in the tool life management table are automatically set.

10. Drawing Blank Shape

The user graphic commands (UGC) are provided so that arbitrary blank shapes may be drawn on the color graphic CRT.

(1) Definition Format (One-spindle models)

The format used for blank shape (side view) drawing is indicated below.

DEF WORK . . . . . Declaration of blank definition

A group of user graphic commands used for drawing blank shape

END . . . . . End of blank shape definition

DRAW . . . . . Blank shape defined is drawn.

The format used for blank shape (front view) drawing is indicated below. (This is effective only for multiple machining specification models.)

DEF WORKF . . . . . Declaration of blank definition

A group of user graphic commands used for drawing blank shape

END . . . . . End of blank shape definition

DRAW . . . . . Blank shape defined is drawn.

(2) Definition Format (Two-spindle models)

The format used for blank shape (side view) drawing is indicated below.

Commands to register the workpiece side view:

DEF WORK [spindle designation*1] . . . . . Declaration of blank definition

A group of user graphic commands used for drawing blank shape

END . . . . . End of blank shape definition

DRAW . . . . . Blank shape defined is drawn.

*1 Spindle designation

M	:	Main spindle
S	:	Sub spindle (pick-off spindle)
Not designated	:	Main spindle
Others	:	Alarm occurs.

Commands to register the workpiece front view:

When the multi-machining specification is selected only for the main spindle, the commands to register the workpiece front view are the same as those used for one-saddle multi-machining models.

When the multi-kmachining specification is selected for both the main and sub spindles, designates as follows.

DEF WORK [spindle designation*2] . . . . . Declaration of blank definition

A group of user graphic commands used for drawing blank shape

END . . . . . End of blank shape definition

DRAW . . . . . Blank shape defined is drawn.

*2 Spindle designation

M	:	Main spindle
S	:	Sub spindle (pick-off spindle)
Not designated	:	Main spindle
Others	:	Alarm occurs.

Workpiece Copy Command:

When the workpiece copy command is designated, the workpiece on the spindle which is opposite to the specified spindle is copied.

Command: "WCPY"

The workpiece copy command is effective only when pattern 3 or pattern 4 is selected as the display pattern.

(3) User Graphic Commands (UGC)

The user graphic commands are largely classified into four types such as drawing point setting, line drawing, circle drawing and painting.

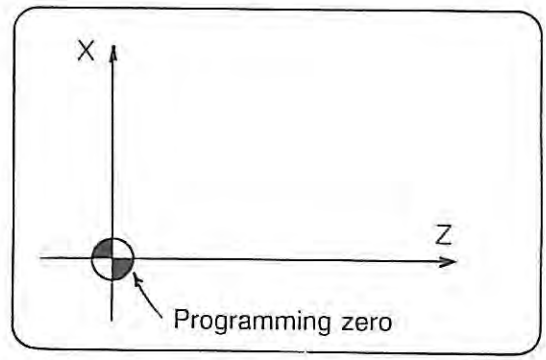
The explanation below provides the format of the representative commands.

1) Coordinate System for Drawing and Setting Unit

The coordinate system used for drawing the blank shape is the Z-X coordinate system having the programming zero as the origin.

The unit system used is:

1 mm for metric system, and 0.1 inches for inch system



2) Setting Drawing Point

POINT [Z_0, X_0]

Note: The command underlined may be used instead of fully giving command characters. POINT, for instance, can be commanded by the underlined characters PO.

The POINT command simply sets the starting point (Z_0, X_0) for the drawing and no actual drawing operation occurs.

This sets the last reference point (LP) for the drawing.

3) Drawing Straight Line

LINE [Z_1, X_1]<, line-code>

A straight line is drawn from the last reference point (LP) of the drawing up to the commanded end point (Z_1, X_1). The end point is referenced to the LP. That is, the command must be given in a relative value. After the execution of the command, the LP is established at the end point of this command.

Line code:
0 = Solid line ————
1 = Dotted line ·····
2 = Short dashes line - - - - - - - -
3 = Long dashes line —— —— ——
4 = Alternate short and long dashes line - - —— - - ——
5 = Alternate long and two short dashes line —— ·· ——
6 = No axis display
7 = Erasing

Default is "0 (solid line)".

4) Circle Drawing

CIRCLE [Z_1, X_1], [Z_c, X_c] <,rotation-direction> <,line-code>

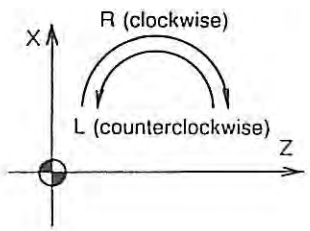
An arc is drawn in the following conditions:

Starting point : LP
End point : (Z_1, X_1)
Center : (Z_c, X_c)

The coordinates of the end point and the center are referenced to the LP. After the execution of the command, the LP is established at the end point of this command.

Rotation Direction:

R: Clockwise
L: Counterclockwise
Default is "R (clockwise)".



Line code: Eight codes as used for drawing a straight line.
(0 through 7) (Refer to 3).

5) Painting

PAINT (or PF) <number-of-vertexes> <,vertex-coordinates> <,tile-pattern>

The PAINT command is used for painting triangles, rectangles and circles defined on the absolute coordinate system.

Number of Vertexes:

0: Circles or sectors
3: Triangles
4: Rectangles

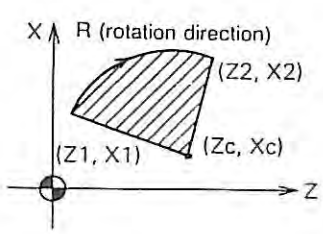
Vertex Coordinates:

The designation of coordinates depends on the number of vertexes specified.

Number of vertex = 0

[Z_1, X_1], [Z_2, X_2] [Z_c, X_c] <,rotation-direction>

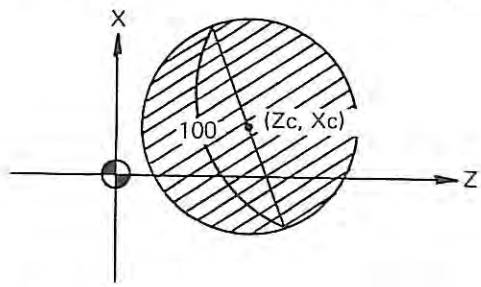
Z_1, X_1 : Start point
Z_2, X_2 : End point
Z_c, X_c : Center



When the number of vertexes is zero (0), designation of diameter is also possible.

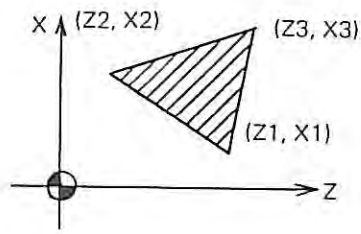
[Zc, Xc], 100, D

Zc, Xc : Center
100 : Diameter
D : Code to indicate diameter designation



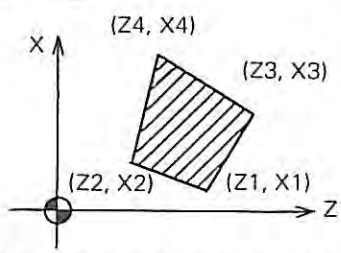
Number of vertexes = 3

$[Z_1, X_1],[Z_2, X_2],[Z_3, X_3]$
$(X_1 \leq X_2 \leq X_3)$



Number of vertexes = 4

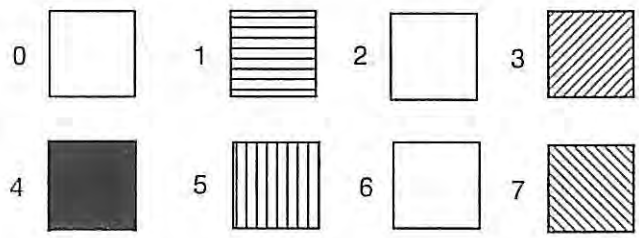
$[Z_1, X_1],[Z_2, X_2],[Z_3, X_3],[Z_4, X_4]$
$(X_1 \leqq X_2 \leqq X_3 \leqq X_4)$



Note that the coordinates of vertexes must be given in absolute values.

Tile Pattern:

0: No drawing (all dots off)
1: All dots painted; on every other horizontal line
2: Dots alternately on and off; on every other horizontal line
3: Oblique line; from upper right to lower left
4: All dots painted
5: All dots painted; on every other vertical line
6: Dots alternately on and off; on every other vertical line
7: Oblique line; from upper left to lower right



Default is "4".

PAINTI <number-of-vertexes> <,reference-point-coordiantes>
<,vertex-coordinates> <,tile-pattern>

The PAINTI command is used for painting triangle, rectangles and circles defined on the incremental coordinate system in reference to the reference point specified.

Number of Vertexes:

0: Circles or sectors
3: Triangles
4: Rectangles

Reference Point Coordinates:

The coordinates of the reference point for defining a shape;

$[Z_0, X_0]$

The values must be given in absolute values.

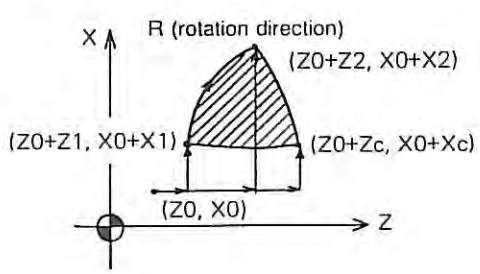
Vertex Coordinates:

The designation of coordinates depends on the number of vertexes specified.

Number of vertexes = 0

$[Z_1, X_1]$, $[Z_2, X_2]$ $[Zc, Xc]$ <,rotation-direction>

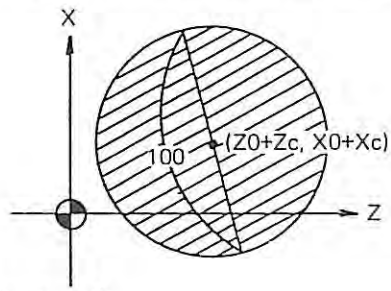
Z_1, X_1 : Start point
Z_2, X_2 : End point
Z_c, X_c : Center



When the number of vertexes is zero (0), designation of diameter is also possible.

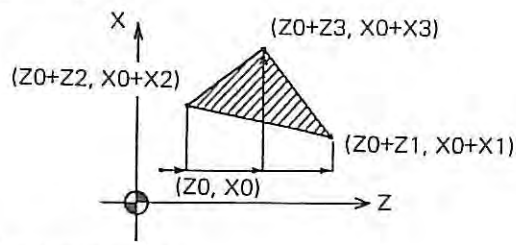
[Zc, Xc], 100, D

Zc, Xc : Center
100 : Diameter
D : Code to indicate diameter designation



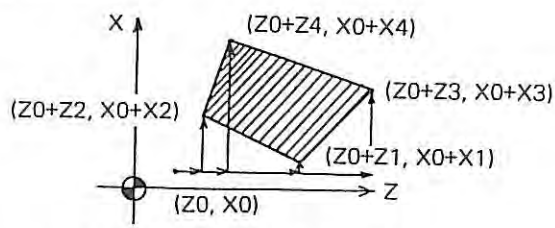
Number of vertexes = 3

$[Z_1, X_1],[Z_2, X_2],[Z_3, X_3]$
$(X_1 \leqq X_2 \leqq X_3)$



Number of vertexes = 4

$[Z_1, X_1],[Z_2, X_2],[Z_3, X_3],[Z_4, X_4]$
$(X_1 \leqq X_2 \leqq X_3 \leqq X_4)$



Tile Pattern: Eight patterns, 0 through 7.

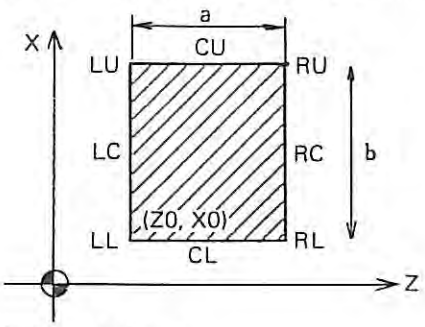
PAINTS <reference-point-position>,[Z_0, X_0],[a, b]
<,tile-pattern>

The PAINTS command is used for painting a rectangle having lengths of a and b.

Reference Point Position:

This specifies the position of the reference point on the rectangle being painted.

LL: Left lower
LC: Left center
LU: Left upper
CU: Center upper
RU: Right upper
RC: Right center
RL: Right lower
CL: Center lower



Reference Position [Z_0, X_0]:

The coordinates must be specified in absolute values.

Tile Pattern: Eight patterns, 0 through 7.

PAINTP <reference-point-position>,[Z_0, X_0],[a, b]
<,angle-code> <,tile-pattern>

The PAINTP command is used for painting rectangles and triangles.

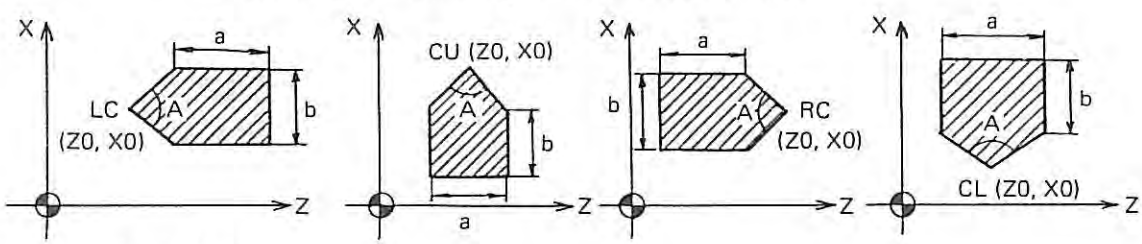
Reference Point Position:

This specifies the position of the reference point on the rectangle and triangle being painted.

LC: Left center
CU: Center upper
RC: Right center
CL: Center lower

Reference Position [Z_0, X_0]:

The coordinates must be specified in absolute values.



Angle Code:

0: Angle A of triangle is 30°
1: Angle A of triangle is 60°
2: Angle A of triangle is 120°

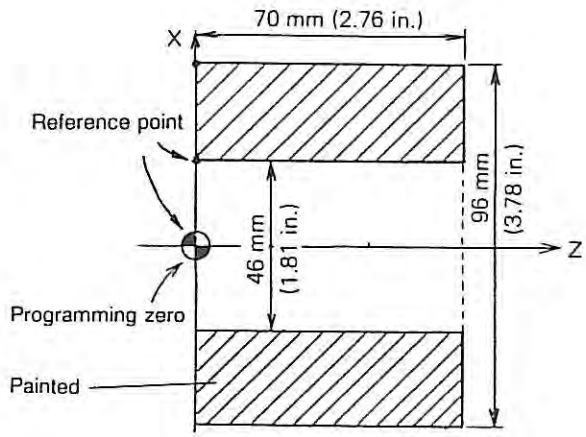
Default is "0 = 30°".

Tile Pattern: Eight patterns, 0 through 7.

(4) Example of Painting

a) Blank material viewed from side

This item explains the procedure to paint the blank shape indicated below.



This painting operation uses the PAINTS command.

Reference point $[Z_0, X_0]$ = [0, 0]
Dimensions [a, b] = [70, 96]

```
PAINTS LC, [0, 0], [70, 96], 4
```

The opening in the blank should be defined using the tile pattern "0" which represents all dots off.

Reference point $[Z_0, X_0]$ = [0, 0]
Dimensions [a, b] = [70, 46]

```
PAINTS LC, [0, 0], [70, 46], 0
```

Therefore, the painting is programmed as indicated below:

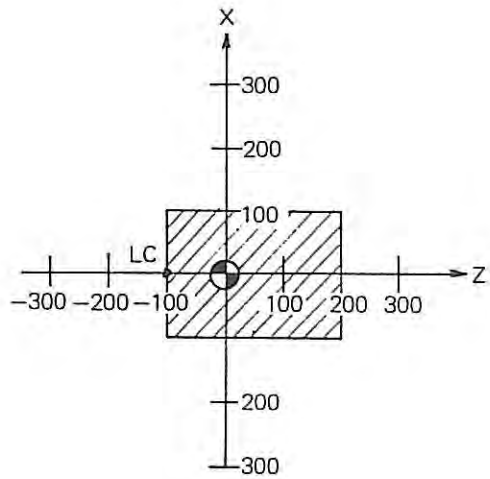
```
DEF WORK
PAINTS LC,  [0, 0], [70, 96], 4
PAINTS LC,  [0, 0], [70, 46], 0
END
DRAW
```

Using simple commands, the same painting is programmed as indicated below:

```
DEF WORK
PS LC,  [0, 0], [70, 96]
PS LC,  [0, 0], [70, 46], 0
END
DRAW
```

Example Program 1

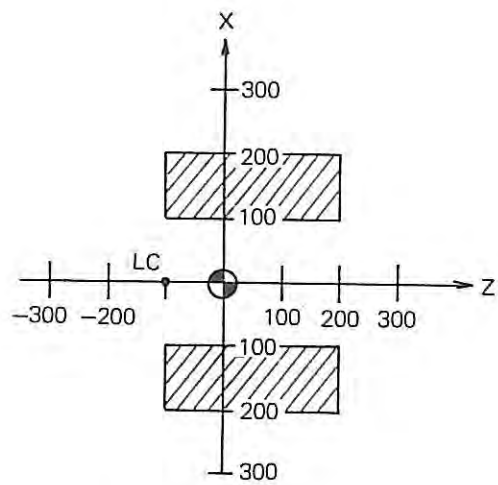
```
Reference Point  . . . .  LC
DEF WORK[M]
PS LC,  [−100, 0], [300, 200]
END
```



Example Program 2

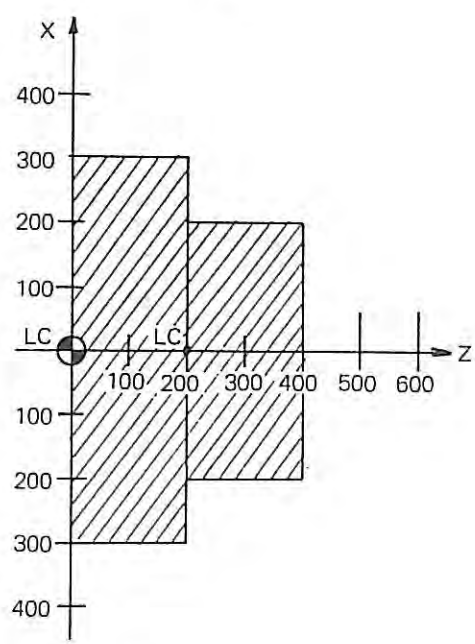
```
Reference Point  . . . .  LC
DEF WORK[M]
PS LC,  [−100, 0], [300, 400]
PS LC,  [−100, 0], [300, 200], 0 (*)
END
```

Note: *0 (*) indicates drawing not to be done.*



Example Program 3

```
Reference Point  . . . .  LC
DEF WORK
PS LC,  [0, 0], [200, 600]
PS LC,  [200, 0], [200, 400]
END
```



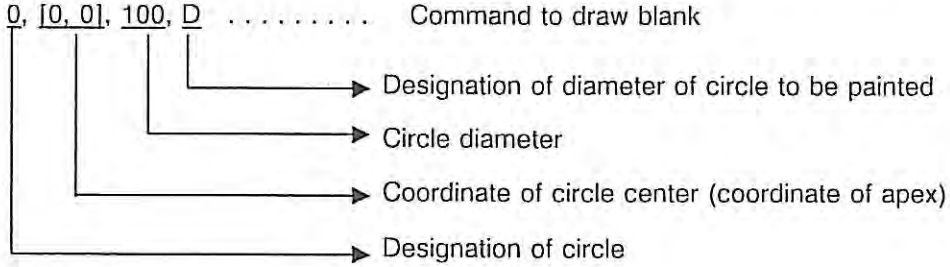
Tile patterns in the examples 1, 2 and 3 are painting of all dots.

b) Blank material viewed from front (only for multiple machining specification)

Description of Commands:

DEF WORKF . . . . . Definition statement - Front view of blank

PF 0, [0, 0], 100, D . . . . . Command to draw blank



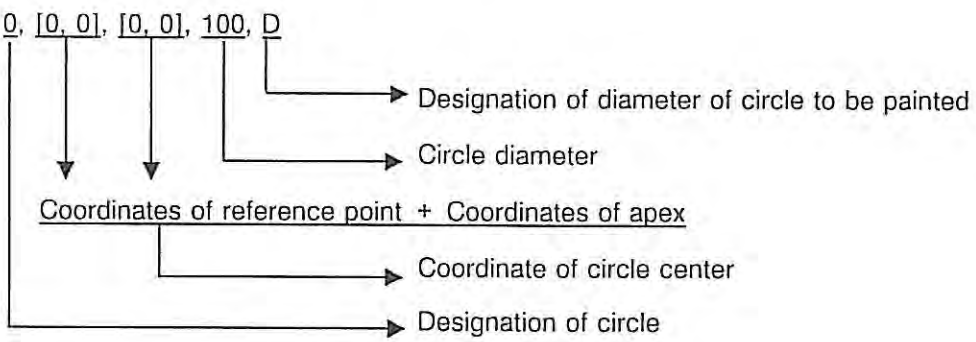
END . . . . . End statement

⇩

or

DEF WORKF

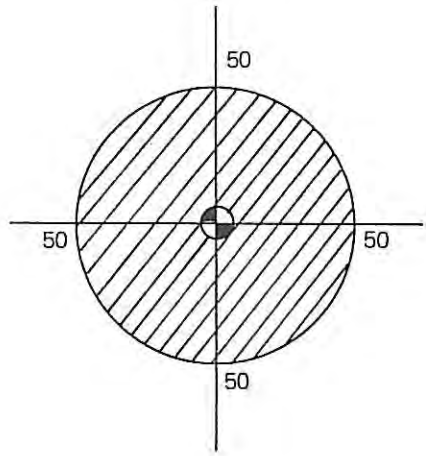
PI 0, [0, 0], [0, 0], 100, D



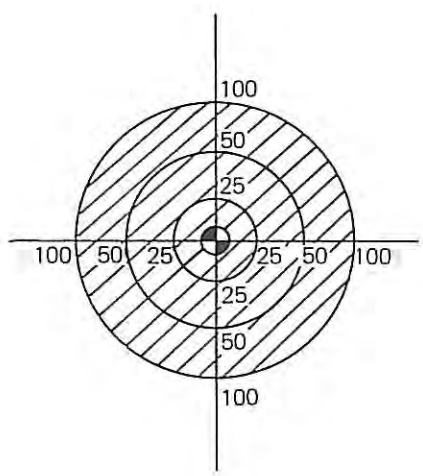
END

Example Program

```
DEF WORKF
PF 0, [0, 0], 100, D
END
```

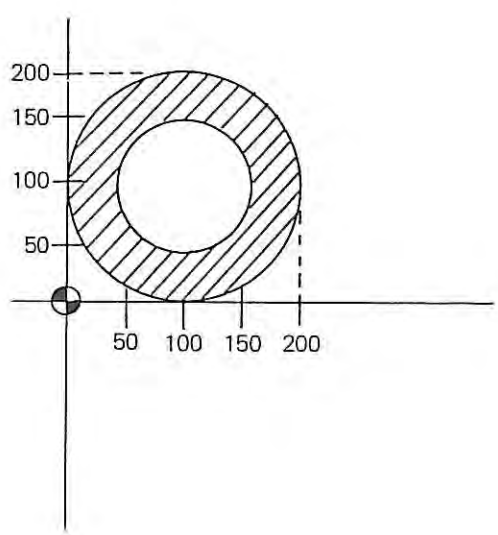



```
DEF WORKF
PF 0, [0, 0], 200, D
PF 0, [0, 0], 100, D
PF 0, [0, 0], 50, D
END
```




```
DEF WORKF
PI 0, [100, 100], [0, 0], 200, D
      [100, 100], [0, 0], 100, D, 0 (*)
END
```

Note: 0 (*) indicates drawing not to be done.

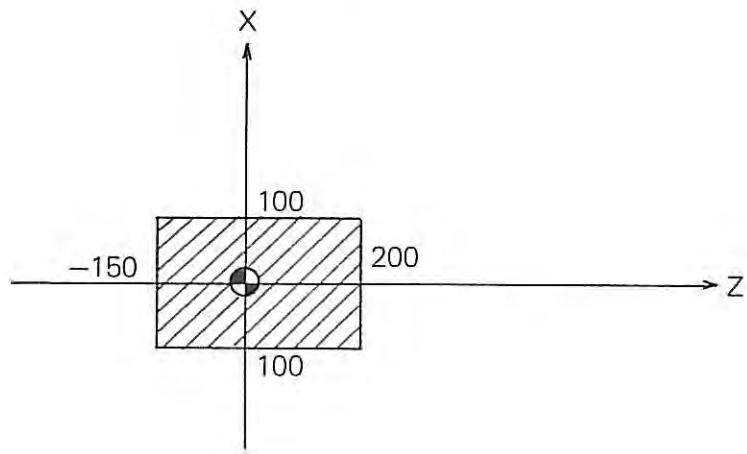


11. How to Set Drawing Positions of Chuck and Tailstock

(1) Single-spindle

To set the chuck and tailstock positions on the CRT screen corresponding to the workpiece material, input proper data at JAW SIZE (L1, D1), JAW POSI. (CX, CZ), CENTER (L2, D2), CENTER (D3) and FACE(WR) on the *CHUCK/ TAILSTOCK BARIER* page in the PARAMETER SET mode.

Example:

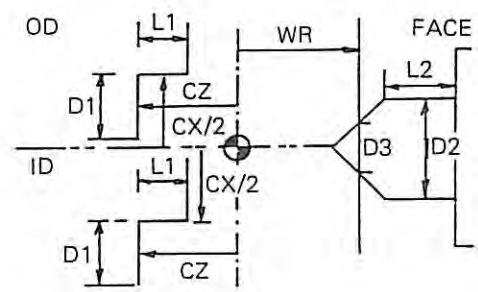


PARAMETER SET
PAGE

A TURRET
CHUCK/TAILSTOCK

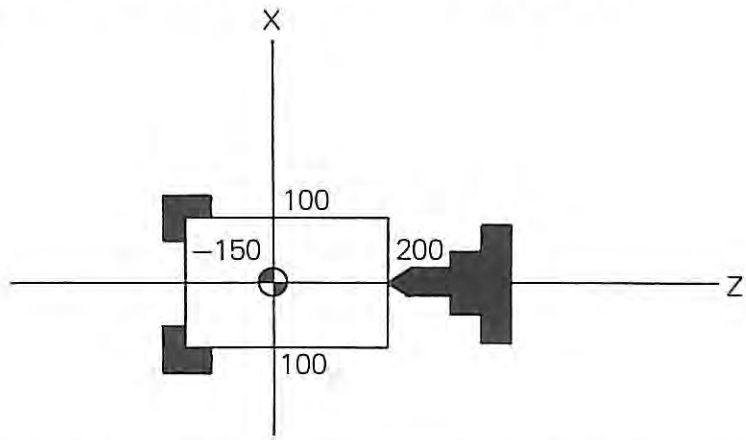
UNIT 1 mm

JAW SIZE	L1 =
	D1 =
JAW POSI.	CX =
	CZ =
CENTER	L2 =
	D2 =
	D3 =
FACE	WR =



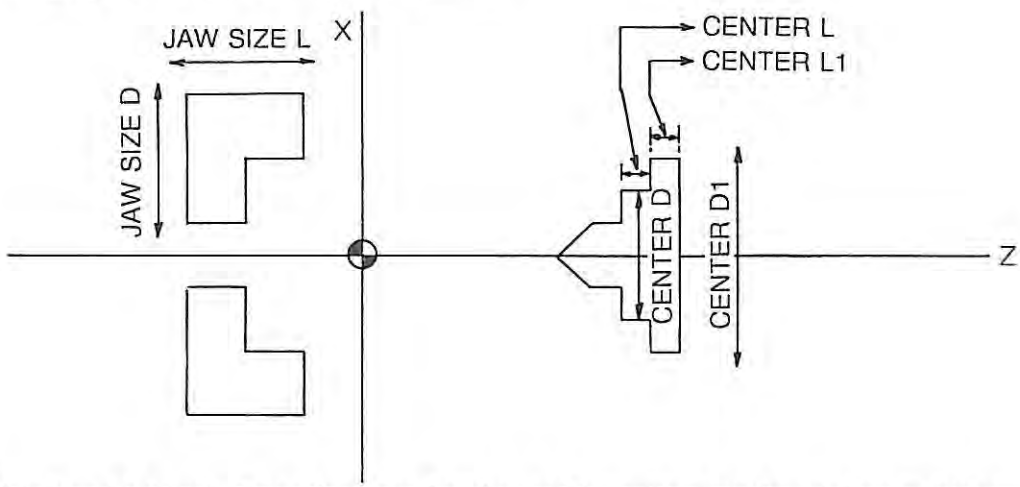
When the following data is input at the *CHUCK/TAILSTOCK BARIER* page in reference to the blank material illustrated above, the chuck and tailstock are drawn as below.

L1 = 50
D1 = 50
CX = 200
CZ = −150
L2 = 50
D2 = 50
D3 = 20
WR = 200



Note: *If the blank material shape is not symmetric in reference to Z-axis, the chuck and the tailstock cannot hold the material correctly.*

Changing Chuck and Tailstock Dimensions:

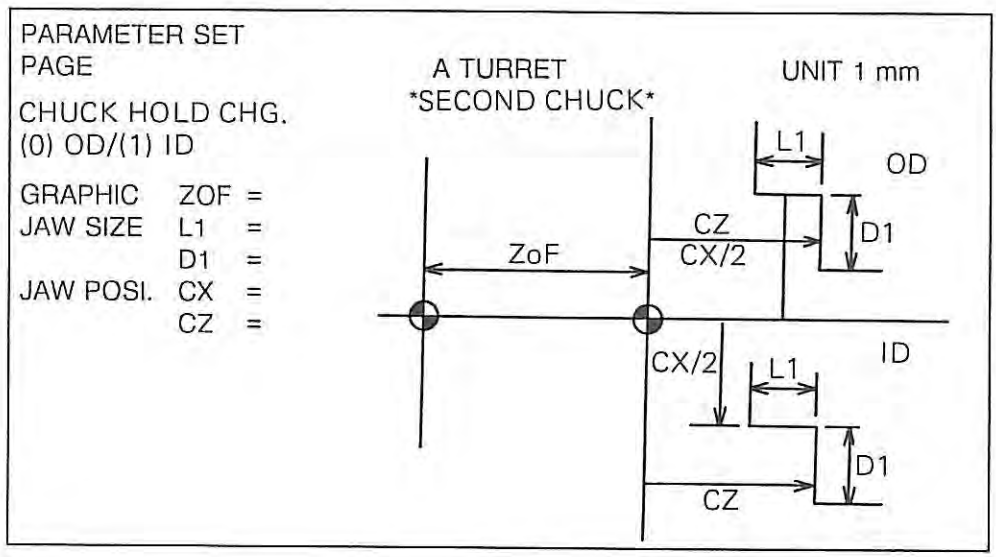


Chuck and tailstock dimensions can be changed by setting required values at parameters No. 18 through No. 23 of optional parameter (long word) which can be displayed in the parameter set mode.

No. 18 . . . . . JAW SIZE L
No. 19 . . . . . JAW SIZE D
No. 20 . . . . . CENTER L
No. 21 . . . . . CENTER D
No. 22 . . . . . CENTER L1
No. 23 . . . . . CENTER D1

(2) Two-Spindle Models

To set the chuck and tailstock positions on the CRT screen corresponding to the workpiece material, input proper data on the SUB CHUCK screen, the second page of the CHUCK/TAILSTOCK BARIER as screen.



CHUCK HOLD CHG. . . Set whether the workpiece ID is chucked or the workpiece OD is chucked.

GRAPHIC ZOF . . . . . Set the offset amount from the graphic zero point of the main spindle chuck.

. . . . . Setting unit: 1 mm

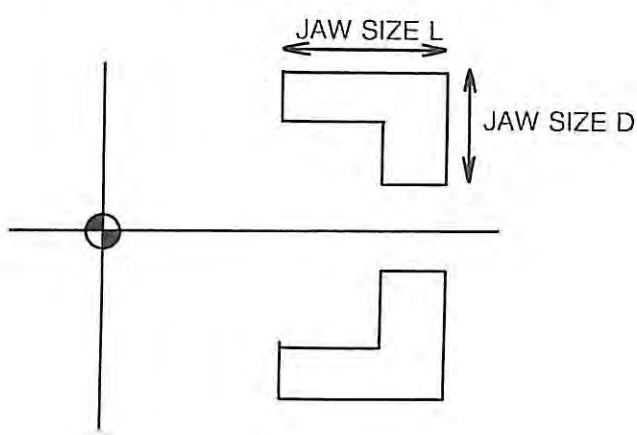
. . . . . Setting range: 0 - 9999.999

JAW SIZE L1 . . . . . Length of the jaw on the sub spindle chuck

JAW SIZE D1 . . . . . Step on the jaw on the sub spindle chuck

JAW POSI. CX . . . . . Gripping diameter of the sub spindle chuck

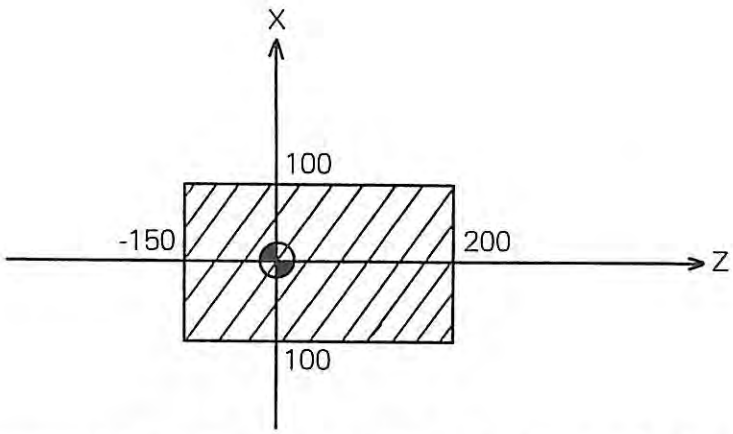
JAW POSI. CZ . . . . . Distance from the program zero



To change the values of JAW SIZE L and JAW SIZE D, change the settings at the optional parameter (long word).

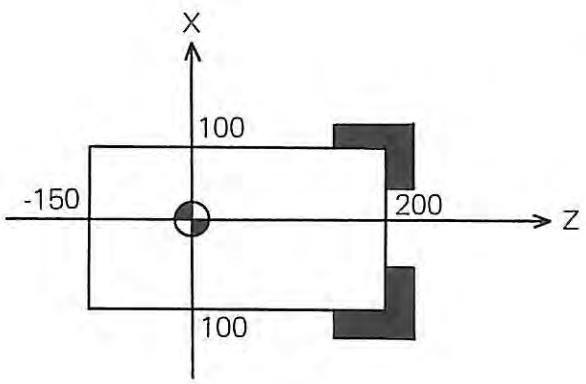
No. 18 . . . . . JAW SIZE L
No. 19 . . . . . JAW SIZE D
} The data for the main spindle chuck is used in common.

Example:



The drawing as illustrated below is displayed when the following sub chuck data is set on the parameter setting screen.

ZOF = 0
L1 = 50
D1 = 50
CX = 200
CZ = 200
L = 80
D = 80



12. System Variables for Animation Display

The tool shape data, chuck barrier data and tailstock barrier data can be commanded in the same manner as conventional part program using the system variables.

(1) Tool Shape Definition

The seven system variables indicated below are used for defining the tool shape.

Variable Name	Contents	Data Size
VTLIN[I]	Tool classification code	1 to 26
VTLFN[I]	Tool shape code	1 to 4
VTLA1[I]	Tool angle	0 to 360.000
VTLA2[I]	Edge angle	−360.000 to 360.000
VTLL[I]	Holder length/Sticking out length/Drill length	0 to 9999.999
VTLD[I]	Holder diameter/Drill diameter	0 to 9999.999
VTLW[I]	Tool width	0 to 9999.999

The index [I] of the system variable is designated by the tool number.

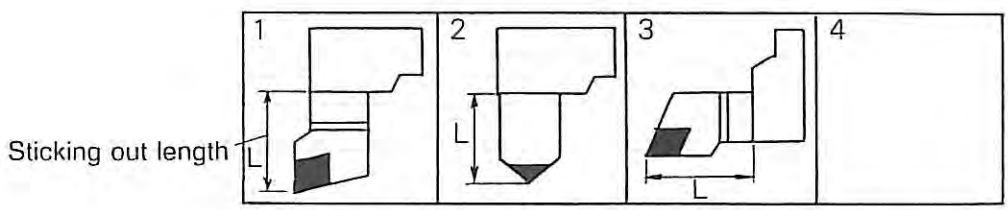
The tool number indicates the station number of the turret.

For the tool life management specification, the system variable name is assigned in the format VGR**[I]. The index [I] is the tool group number.

Tool Classification Code :
1 = ROUGH OD←
2 = ROUGH ID←
3 = ROUGH FACE ↓
4 = ROUGH OD→
5 = ROUGH ID→
6 = ROUGH FACE ↑
7 = FINISH OD←
8 = FINISH ID←
9 = FINISH FACE ↓
10 = FINISH OD→
11 = FINISH ID→
12 = FINISH FACE ↑
13 = THREAD OD←
14 = THREAD ID←
15 = THREAD FACE ↓
16 = THREAD OD→
17 = THREAD ID→
18 = THREAD FACE ↑
19 = GROOVE OD ↓
20 = GROOVE ID ↑
21 = GROOVE FACE←
22 = DRILL HSS
23 = DRILL CARBIDE
24 = DRILL CENTER
25 = RECESS OD↙
26 = RECESS ID↖
27 = M-DRILL ↓
28 = M-DRILL←

29 = BORING ↓
30 = BORING←
31 = TAP ↓
32 = TAP←
33 = REAMER ↓
34 = REAMER←
35 = END MILL ↓
36 = END MILL←
37 = FACE MILL ↓
38 = FACE MILL←

Tool Shape Code: 1, 2, 3, 4 — Tool shape number which corresponds to the tool shape usable for the tool selected by the tool classification code.



(ROUGH OD←, FINISH OD←)

Example:

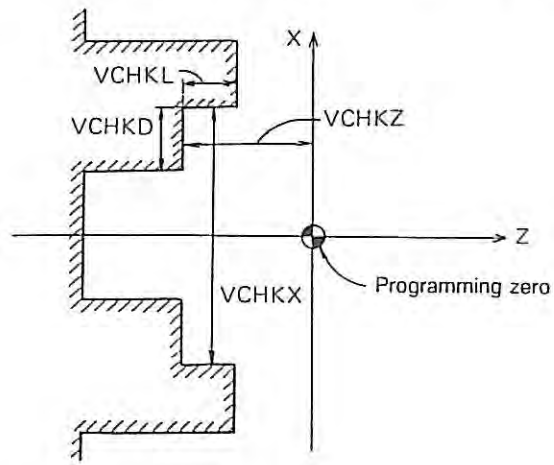
ROUGH OD
TOOL NO. TN = 2
TOOL ANGLE A1 = 80°
EDGE ANGLE A2 = 3°
STICKING OUT L = 40 mm

VTLIN[2] = 1
VTLFN[2] = 1
VTLA1[2] = 80.000
VTLA2[2] = 3.000
VTLL[2] = 40.000

(2) Chuck Barrier Definition

The four system variables indicated below are used for defining the chuck barrier.

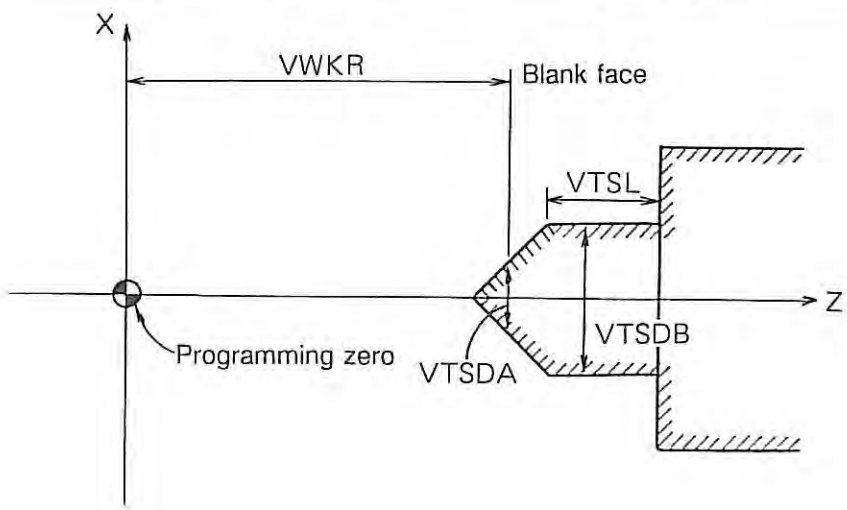
Variable Name	Contents	Data Size
VCHKL	Chuck jaw length	0 to 9999.999
VCHKD	Step on chuck jaw	0 to 9999.999
VCHKX	Gripping diameter	−9999.999 to 9999.999
VCHKZ	Distance from program zero	−9999.999 to 9999.999



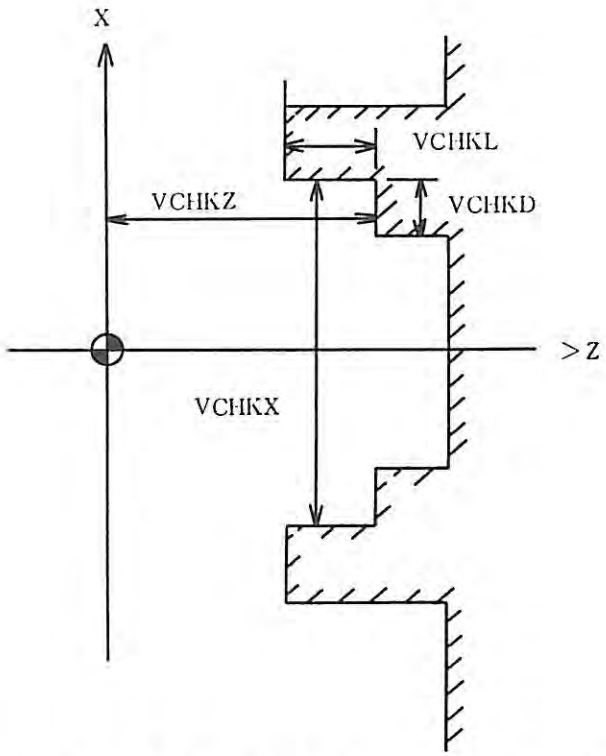
(3) Tailstock Barrier Definition

The four system variables indicated below are used for defining the tailstock barrier.

Variable Name	Contents	Data Size
VWKR	Blank face position	−9999.999 to 9999.999
VTSL	Sticking out length of tailstock spindle	0 to 9999.999
VTSDA	Center hole diameter	0 to 9999.999
VTSDB	Tailstock spindle diameter	0 to 9999.999



(4) Sub Chuck



The system variables which are used for the main spindle chuck are used in common for the sub spindle chuck. The selection of the spindle, sub or main, is made by designating a G code.

Example:

```
G13
G140
VCHKL =  ┐
VCHKD =  │
VCHKX =  ├ Main chuck
VCHKZ =  ┘
   :
   :
G141
VCHKL =  ┐
VCHKD =  │
VCHKX =  ├ Sub chuck
VCHKZ =  ┘
   :
   :
```

13. Blank Material Drawing Method

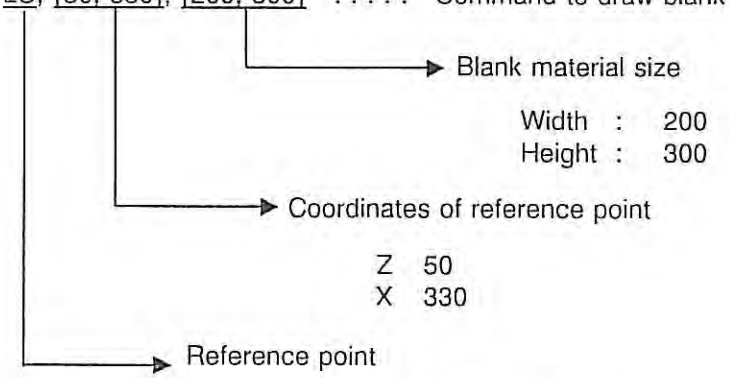
To draw blank material, designate the blank material drawing commands and display the shape (square, rectangle) of the material by pressing the material shape key.

(1) Blank Material Viewed from Side

Description of Commands:

DEF WORK . . . . . Definition statement

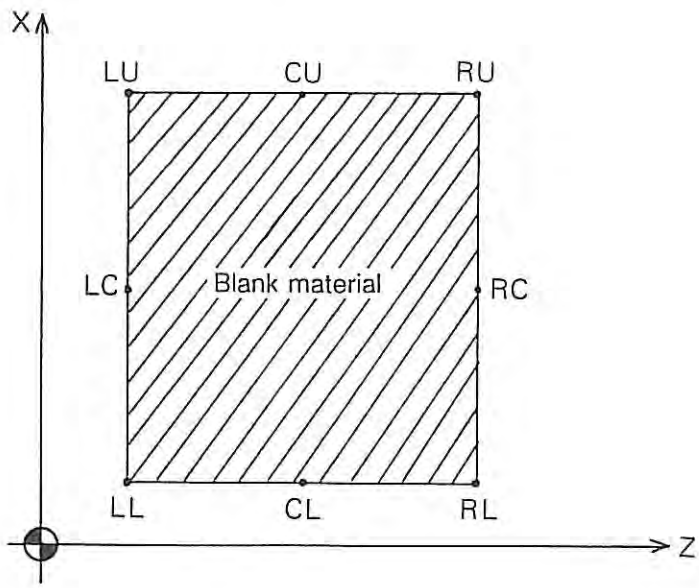
PS LU, [50, 330], [200, 300] . . . . . Command to draw blank



END . . . . . End statement

Note: To draw blank material, at least the above indicated three statements must be written.

Reference Points:



Minimum Setting Unit: 1 mm (0.1 in.)

SECTION 4 CYCLE TIME CALCULATION FUNCTION

1. Overview

This function calculates the length of time required for executing a part program under the specified conditions without actually operating the machine.

2. Cycle Time Calculation Process

(1) In High Speed Graphic Drawing Mode

Cycle time is calculated during the control is in the high speed graphic drawing mode. The high speed graphic drawing mode means that both the machining lock and dry run functions are turned on.

(2) Counting Actual Program Execution Time during Automatic or MDI Mode Operations

While the control is operating under either the automatic or the MDI mode, execution time of a part program is calculated (counted) as the program is executed on the graphic display, the final shape is drawn as in the high speed graphic drawing mode.

a) In the automatic mode

The cycle time is calculated as the total of one-block execution time by adding the program execution time for each block. Therefore, this calculation function can be activated independently of the setting of the single block function whether it is on or off. Pressing the **CYCLE START** pushbutton switch resets the counter to zero and counting the program execution time starts from 0 second.

b) In the MDI mode

Each time the **CYCLE START** pushbutton switch is pressed after entering the commands of one block, the length of time required for executing the entered block is counted and added to the accumulated count data. This counting occurs when the operation mode is switched to the MDI from other modes. Counter is automatically reset to zero at the instant the operation mode is switched.

Note: *The MACHINE TIME data displayed before switching the operation mode to the auto or MDI indicates the cycle time having been counted. And this data is automatically cleared to zero when the* ***CYCLE START*** *pushbutton switch is pressed.*

(3) Influence of Override Setting

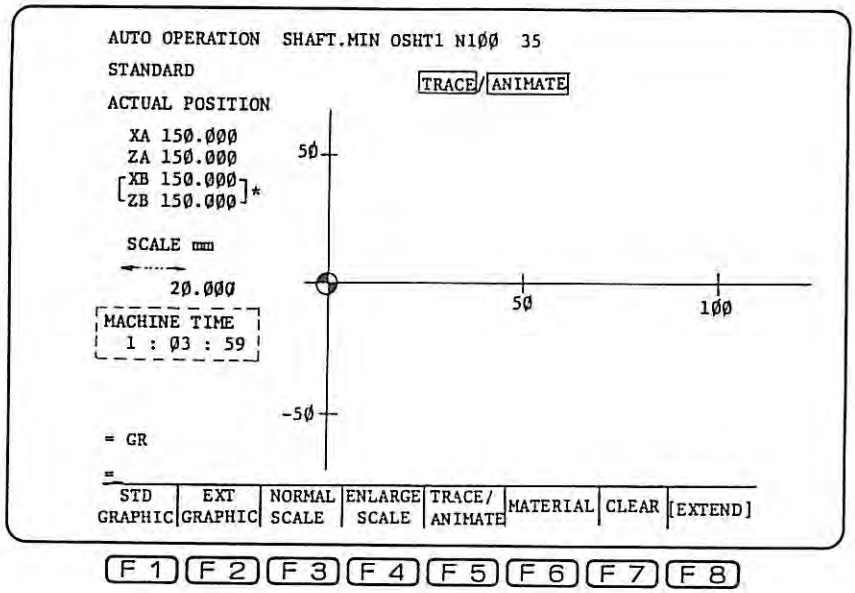
The cycle time is counted taking the setting of the override value into account. In other words, the override function is effective for the cycle time counting function. For the rapid feedrate, the setting of the override is ignored and fixed at 100%.

(4) Cycle Time Counting for Simultaneous 4-axis Cuts

For the two-saddle models, synchronizing command P is effective for counting the cycle time.

3. CRT Display of Machine Time Data

The calculated (counted) cycle time is displayed on the graphic display.



* The data in [ ] is available for the two-saddle models.

The unit of the MACHINE TIME data is hours, minutes and seconds.

4. Operation Procedure

(1) Select the high speed graphic drawing mode (MACHINE LOCK and DRY RUN ON).

(2) Select the graphic display mode by pressing an appropriate function key.

(3) Follow the steps for operating the machine in normal machine operation mode.

SECTION 5 OVERLOAD DETECTION

1. Overview

This function monitors load applied to X- and Z-axis using meter relays.

Limit current value and detection time are preset. If the current value of the axis drive motor exceeds the preset limit level for a duration longer than the preset time (detection time), an alarm C occurs and the control stops in the cycle stop mode. Alarm level can be switched from alarm C to alarm A by setting appropriate parameter (bit) data.

2. Setting Monitor Range

The sequences in which load applied to the axis drive motor is to be monitored can be set by M codes, M135 and M134.

M134: End of thrust monitoring
M135: Start of thrust monitoring

These M codes are designated for respective turrets, independently. (For two-saddle model)

Programmed M codes are reset when the control is reset.

3. Program Examples

3-1. For One-Saddle Model

Load on X- and Z-axis drive motors is monitored between blocks N002 and N004.

```
N001   G00    X500   Z1000
N002   M135
N003   G01    X800   Z700   F10    S200   M41    M03
N004   X1000
N005   M134
 :
 :
```

3-2. For Two-Saddle Model

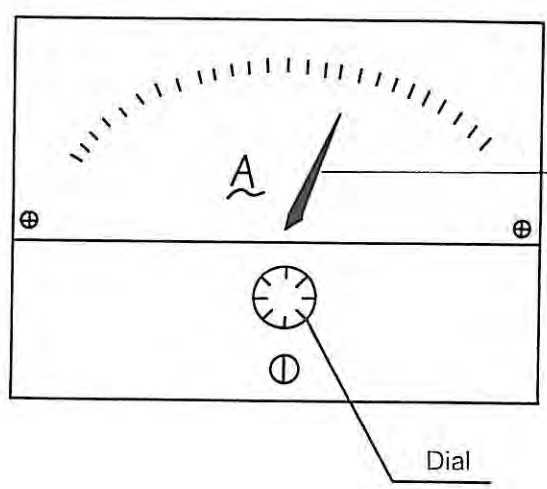
Load on XA- and ZA-axis drive motor is monitored between blocks N003 and N005 and that on XB- and ZB-axis between blocks N103 and N105.

```
N001   G13
N002   G00    X500   Z1000
N003   M135
N004   G01    X800   Z700   F10    S200   M41 M03 P10
N005   X1000
N006   M134
 :
 :
N101   G14
N102   G00    X700   Z500
N103   M135
N104   G01    X900   Z200   F10    S200   M41 M03 P10
N105   X1200
N106   M134
 :
 :
```

Note: *The Z-axis thrust (X-axis thrust) monitoring specification is also available. These specifications monitor the thrust of only Z- or X-axis.*

4. Setting Limit Current Value

The limit level of current can be set for X- and Z-axis independently using meter relays. For two-saddle models, limit level can be set for four axes (XA, ZA, XB and ZB) independently using meter relays.



Meter Relay

5. Setting Detection Time

Detection time is set in the following procedure.

(1) Press the **PARAMETER** key on the operation panel.

(2) Press the function key [F7] (ITEM ↓) to display ***OPTIONAL PARAMETER WORD*** page.

(3) Locate the cursor to No. 17.

(4) After pressing the function key [F1] (SET), enter desired detection time.

(5) Press the **WRITE** key.

Detection time is set in increments of 0.1 seconds within a range of 0 to 50 seconds. When "30" is set, detection time is set to "3" seconds. For two-saddle models, the set value is effective for both turrets.

Note: *If the time less than 0.7 seconds is set, the monitoring feature will possibly be affected by motor starting current.*

6. Setting Alarm Level

Overload detection alarm level, initially set at level C, can be changed to level A by the parameter (bit) data setting.

Parameter (bit) data No. 7	Alarm level
Bit 4 = 0	Alarm C
Bit 4 = 1	Alarm A

Note: *Alarm C . . . . An occurrence of an alarm does not interrupt the execution of a program and the program is executed up to the end of the program (M02) even if an alarm occurs.*

Alarm A . . . . An occurrence of an alarm stops axis movements, spindle rotation and coolant supply.

Procedure for setting alarm level:

(1) Press the **PARAMETER SET** key on the operation panel.

(2) Press the function key [F7] (ITEM ↓) to display the ***OPTIONAL PARAMETER BIT*** page.

(3) Locate the cursor at No. 7.

(4) Locate the cursor at bit 4 position.

(5) After pressing the function key [F1] (SET), key in "0" or "1".

(6) Press the **WRITE** key.

7. Alarm Messages

ALARM-C

914 ZA-axis overload

ZA-axis overload detection relay has been kept actuated due to overload for more than the time set by a parameter (optional parameter (word) data 17).

Index : None

Character-string : None

Code : None

915 ZB-axis overload

ZB-axis overload detection relay has been kept actuated due to overload for more than the time set by a parameter (optional parameter (word) data 17).

Index : None

Character-string : None

Code : None

917 XA-axis overload

XA-axis overload detection relay has been kept actuated due to overload for more than the time set by a parameter (optional parameter (word) data 17).

Index : None

Character-string : None

Code : None

918 XB-axis overload

ZB-axis overload detection relay has been kept actuated due to overload for more than the time set by a parameter (optional parameter (word) data 17.)

Index : None

Character-string : None

Code : None

ALARM-A

183 XA-axis overload

XA-axis overload detection relay has been kept actuated due to overload for more than the time preset by a parameter (optional parameter (word) data 17).

Index : None

Character-string : None

Code : None

184 XB-axis overload

XB-axis overload detection relay has been kept actuated due to overload for more than the time preset by a parameter (optional parameter (word) data 17).

Index : None

Character-string : None

Code : None

185 ZA-axis overload

ZA-axis overload detection relay has been kept actuated due to overload for more than the time preset by a parameter (optional parameter (word) data 17).

Index : None

Character-string : None

Code : None

186 ZB-axis overload

ZB-axis overload detection relay has been kept actuated due to overload for more than the time preset by a parameter (optional parameter (word) data 17).

Index : None

Character-string : None

Code : None

1137 Y-axis overload

On the Y-axis specification, Y-axis overload detection relay has been kept actuated due to overload for more than the time preset by a parameter (optional parameter (word) data 17).

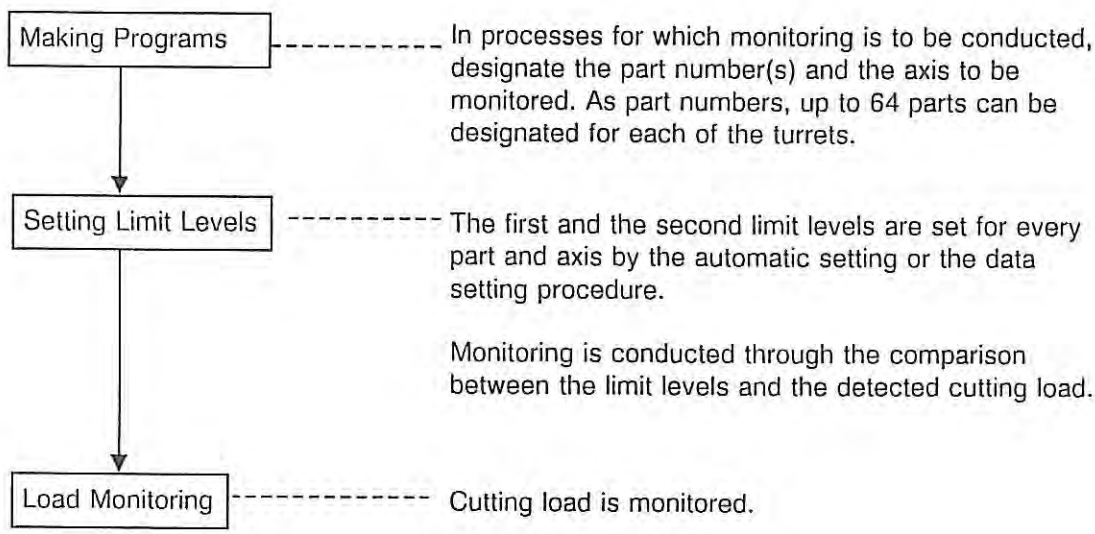
SECTION 6 LOAD MONITOR A/B

1. Overview

This function monitors load applied to the motors driving axes, spindle and M-tool (rotary tool) spindle. Note that monitoring function for the spindle and M-tool spindle drive motors is optional.

Load monitoring is conducted in accordance with the monitoring intervals (to be referred to as "part" hereinafter) and monitoring axis command both designated in a part program. If the cutting load detected exceeds the preset first limit level, an alarm of level C (overload alarm) occurs and if it exceeds the preset second limit level, an alarm of level A (tool breakage alarm) occurs.

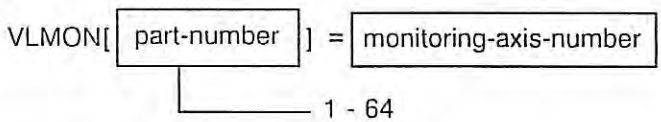
2. Overview of Operations



3. Programs

Programs related to load monitoring are written using system variables VLMON.

3-1. Format of System Variable VLMON



Monitoring-axis-number:

0 . . . . . Monitoring axis OFF
1 . . . . . X-axis ON
2 . . . . . Z-axis ON
4 . . . . . C-axis ON
8 . . . . . Spindle ON
16 . . . . . M-tool spindle ON
32 . . . . . W-axis ON
64 . . . . . Sub spindle ON

To monitor the load of two or more axes simultaneously, designate the total of the monitoring-axis-number of the axes to be monitored.

Example 1: To monitor the XA-axis, ZA-axis, sub spindle, and W-axis simultaneously, designate as follows:

VLMON[ ] = 1 + 2 + 32 + 64

Example 2: To monitor the XB-axis, ZB-axis, M-tool spindle, and C-axis simultaneously, designate as follows:

VLMON = 1 + 2 + 4 + 16

3-2. Monitoring ON/OFF Command

To turn the monitoring function on and off within a part program where a monitoring is required, designate the system parameter VLMON. Program Example 1 shows the case in which the monitoring is turned on and then off for monitoring part 1 for Z-axis.

Program Example 1:

```
N010 VLMON[1]=2  . . . . . .   Monitoring ON
   :
   :   Cutting program
   :
N020 VLMON[1]=0  . . . . . .   Monitoring OFF
```

3-3. Monitoring on Multiple Parts

Up to 64 parts can be designated for individual turrets to be monitored.

To change the part, first turn off monitoring in the previous part and then turn on monitoring in a new part.

In the similar manner, turn off monitoring in the previous part before changing the monitoring axis.

Program example 2 shows the case in which the monitoring is conducted in the following order.

Monitoring part 1 = Z-axis

Monitoring part 2 = X-axis

Monitoring part 3 = Z-axis

Program Example 2:

```
N009                          G00  X   Z
N010    VLMON[1]=2            G01      Z   F
   :                                :
   :   Cutting program
   :                                :
N020    VLMON[1]=0            G00  X   Z
N021    VLMON[2]=1            G01  X   Z   F
   :                                :
   :   Cutting program
   :                                :
N030    VLMON[2]=0            G00  X   Z
N031    VLMON[3]=2            G01  X   Z   F
   :                                :
   :   Cutting program
   :                                :
N040    VLMON[3]=0            G00  X   Z
```

Note 1: *Although there is no relationship between the part numbers and the tool numbers, programs can be made easier to read by designating them with a certain relationship between them.*

Tool 1	Rough cutting	Part 1
	Finish cutting	Part 11
Tool 2	Rough cutting	Part 2
	Finish cutting	Part 22

Note 2: *In the LAP cycle, it is impossible to turn on and off the monitoring for individual parts. When monitoring is required only for cutting feeds, use the rapid feed ignoring function, explained detail in 3-5.*

Note 3: *When MONITORING ON/OFF command is specified in a block containing axis motion commands, the specified command becomes effective from the designated block.*

3-4. Programming on Two-saddle Models

Cutting load monitoring is possible on two saddles independently by designating the load monitoring system variable at G13 and G14.

Program Example 3 shows the case in which the monitoring is conducted as indicated below:

Monitoring part 1 = X-axis for G13

Monitoring part 1 = Z-axis for G14

Program Example 3:

```
N001   G13
N009   G00                  XZ
N010   VLMON[1]=1           G01  X   Z   F
   :                            :
   :   Cutting program
   :                            :
N020   VLMON[1]=0           G00  X   Z
N001   G14
N009   G00                  XZ
N010   VLMON[1]=2           G01  X   Z   F
   :                            :
   :   Cutting program
   :                            :
N020   VLMON[1]=0           G00  X   Z
N021   M02
```

3-5. Rapid Feed Ignoring Function (M215, M216)

For a program in which cutting feed and rapid feed appear alternately or for the LAP cycle, if monitoring is required only for cutting feed blocks, the monitoring ON/OFF command must designated before and after the monitoring required blocks frequently.

The rapid feed ignoring function is provided to facilitate such programming. That is, the rapid feed ignoring function designated by the M216 command allows the monitoring to be ignored for all rapid feed blocks within the monitoring area.

Program Example 4:

```
N009    M216  (rapid feed ignore)
N010    G00   X   Z
N011    VLMON[1]=3   G01 X   Z   F
   :              :
        Cutting program
   :              :
N020    VLMON[1]=0   G00 X   Z
N021    VLMON[2]=1   G01 X   Z   F
   :              :
        Cutting program
   :              :
N030    VLMON[2]=0   G00 X   Z
N031    M215  (rapid feed ignoring function cancel)
```

Within the monitoring area, rapid feed blocks are not monitored in the range beginning with the M216 command and ending with the M215 command.

These M commands are also effective for automatic setting of the monitoring data; in this setting process, rapid feed sequences are excluded for monitoring operation. For details, refer to 4. "Setting of Limit Levels".

3-6. Load Trace Display Clear Command (LCLEAR)

The load trace (broken line display) in the load trace display mode (to be discussed later) can be cleared by pressing the function key [F7] (CLEAR) or changing the display screen. In addition to these operations, the system supports the command to clear the load trace.

To clear the load trace in a program, designate the LCLEAR command. This command can also be input through the keyboard.

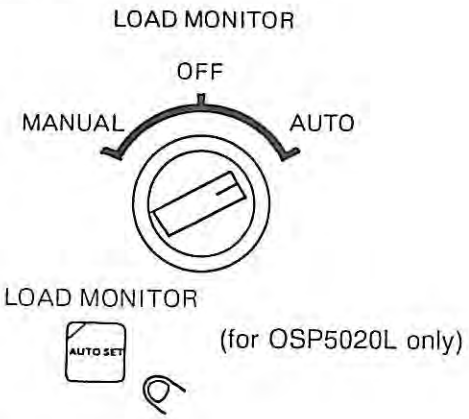
4. Setting of Limit Levels

Limit levels can be set in two methods. One is an automatic setting through the execution of model cutting and the other is a direct setting in which the limit level data is set from the TOOL DATA SET mode, LOAD MONITOR screen. These two methods are explained in detail below.

4-1. Automatic Setting Method

(1) Place the **LOAD MONITOR** switch on the option panel at the AUTO SET.

(2) Carry out trial cutting using a load monitoring program made as explained in 3.



With the steps above, limit levels are automatically calculated and set in reference to the maximum value, which is taken as the reference load, of the cutting load (note 2) for a designated part and designated axis. 110% (note 1) of the reference load is set as the first limit level and 120% (note 1) of the reference load is set as the second limit level.

If the monitoring is conducted for the same part with the same axis more than one time, the limit levels are set in reference to the maximum value of the cutting load during the cycle for the last command.

As explained above, all the limit levels for the part and axis to be monitored can be automatically set in reference to the outcome of the actual cutting.

When carrying out the automatic setting, the following points must be attended to:

a) Model cutting must be conducted under the conditions same as during actual cutting.

b) If any of the feedrate, spindle speed, feedrate override value and spindle speed override value differs between the model cutting and actual cutting to be monitored, monitoring cannot be conducted correctly.

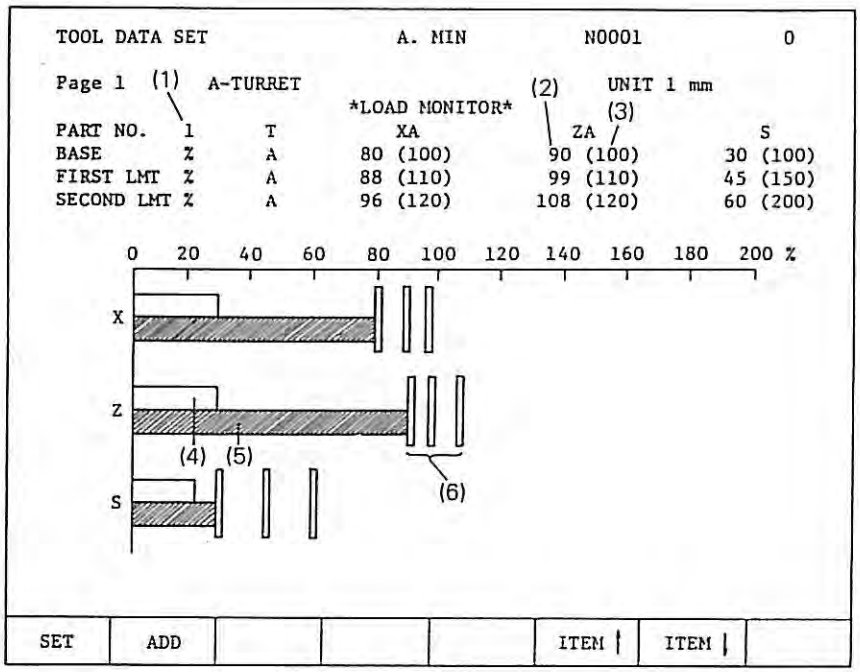
c) If simultaneous 2-saddle cutting is conducted, since cutting conditions during model cutting using only one saddle might differ from those used in actual 2-saddle cutting, do not conduct model cutting in one-saddle mode.

Note 1: *Changeable by the setting of optional parameter (word) data.*

Note 2: *As cutting load of individual drive motors, average value of 16 samples collected in sampling intervals of 12.8 msec is obtained. Note that the number of samples to be collected is changeable (4 to 80 samples) by optional parameter (word) data.*

4-2. Direct Setting of Limit Levels

Direct setting of the reference level and limit levels through the keyboard is possible at the TOOL DATA SET mode, LOAD MONITOR screen when the reference level and limit levels are known or the data already set is to be changed.



(1) PART NO.

This indicates the monitoring area number; changeable within a range from "1" to "64" using the page keys.

(2) Setting Value

The data for the BASE (reference level), FIRST LMT (first limit level) and SECOND LMT (second limit level) is set in terms of percent referenced to the motor rating value.

(3) Ratio to Reference Level

Values in ( ) are percent values of the first and the second limit levels in reference to the BASE level. Setting at these columns is not possible.

(4) Load in Bar Chart

These bars indicate the current load conditions.

(5) Peak Value in Bar Chart

This bar indicates the peak value monitored in a specific part (part being monitored).

The peak value is held until the monitoring of the next part begins. Note that it is cleared when power is turned off.

(6) Setting Levels

Three vertical bars indicate the set levels:

Reference level in dotted line (blue)

First limit level in broken line (yellow)

Second limit level in solid line (red)

These values are set in reference to the motor continuous rating, which is taken as 100%.

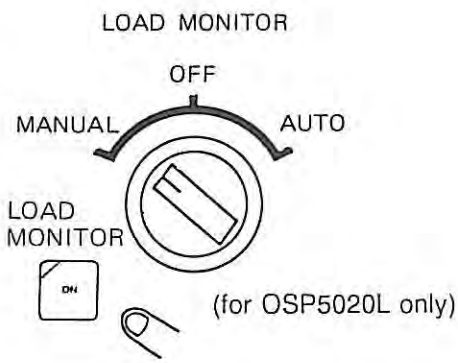
Data setting is conducted in the following steps:

1) Select the TOOL DATA SET mode.
2) Display the LOAD MONITOR screen by pressing the function keys [F6] and [F7].

   On the multiple-machining models, select the display screen between the C-axis and M-tool spindle screen and the feed axis and spindle screen.
3) Press the page key to select the part.
4) Locate the cursor at the data setting column required.
5) Set the required data.

5. Load Monitoring

After placing the **LOAD MONITOR** switch on the option panel at MONITOR and start the program. If monitoring is not required, place this switch at OFF.



The function conducts the load monitoring within the designated part and for the designated axis as specified in a program. If the detected load exceeds the first limit level for a preset length of time continuously, a level C alarm "Load monitor over load" occurs. If the detected load exceeds the second limit level for a preset length of time continuously, a level A alarm "Load monitor tool break down" occurs.

By making the function immune for a preset length of time (0.4 sec) from the instant the cutting starts (feedrate is changed from rapid to cutting), the peak load detected at the beginning of cutting may be removed from the object of monitoring. This immune period may be changed by setting proper parameter data within a range of 0 to 5 seconds. The limit level calculations are not conducted in the automatic setting mode.

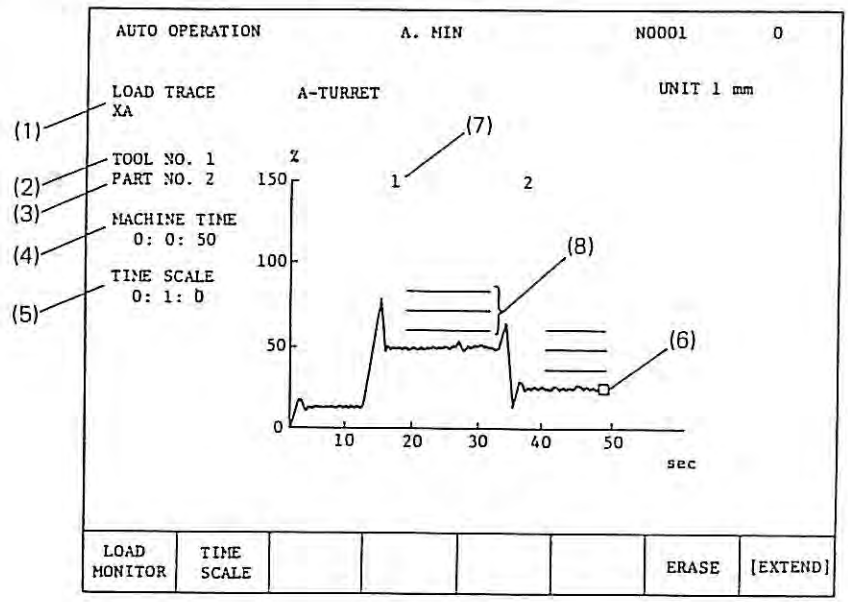
Note : As cutting load of individual drive motors, average value of 16 samples collected in sampling intervals of 12.8 msec is obtained. Note that the number of samples to be collected is changeable (4 to 80 samples) by optional parameter (word) data.

6. Load Monitoring Graphic Display (Load Trace Display)

The loaded condition and monitored conditions of the motor are displayed on the CRT screen in the form of broken line chart. This graphic display is available in the AUTO, MDI and MANUAL OPERATION modes.

Note that in the MANUAL OPERATION mode, load trace is not drawn. This trace is drawn only while the program is being executed.

CRT display and operation procedures are explained below:



(1) Display Axis

The axis is selected by pressing the PAGE key.

(2) TOOL NO.

This indicates the tool number presently selected.

(3) PART NO.

This indicates the monitor area (part) number presently selected.

(4) MACHINE TIME

This indicates total cutting time from the cycle start.

(5) TIME SCALE

This indicates the time scale from the left edge to the right edge.

(6) Marker

This displays the present load point. The load trace is drawn where the load point has passed.

(7) Part No.

This displays the part number of the monitoring area in which the monitoring has been turned on.

(8) Setting Level Lines

Three lines--reference level line, first limit line, second limit line--are displayed while the monitoring is on.

- Reference level line in dotted line (blue)
- First limit level line in broken line (yellow)
- Second limit level line in solid line (red)

Note 1: *The vertical axis should be set in advance by a parameter.*

Note 2: *When the display axis is switched, previous tool path trace is erased.*

Note 3: *The setting level lines in (8) are not displayed in the automatic setting mode. In this case, the red line which indicates the monitoring area is displayed at the lowest line (on the horizontal axis).*

Operation Procedure:

1) Select either of the MDI and AUTO OPERATION modes.
2) Press the function key [F1] (LOAD MONITOR). If the function LOAD MONITOR is not assigned to the function key [F1], press the function key [F8] (EXTEND) once.
3) Press the function key [F2] (TIME SCALE) to set the time axis. This sets the scanning time from the left edge to the right edge of the graph.

   [F2] [0] [:] [2] [:] [0] **[WRITE]**

   The above key operations set 2 min. (Setting range: 20 sec to 1 hour)
4) For two-saddle models, select the turret by pressing the turret selection key.
5) Select the axis to be displayed by the **PAGE** key.

7. Load Monitoring Function Used in Combination with the Tool Life Management Function

When the tool life management function is used in combination with the load monitoring function, automatic turret indexing is possible without causing the overload alarm (alarm C) when the cutting load exceeds the first limit level.

When the cutting load exceeds the first limit level, the NG flag in the tool life management information table is turned on. Then, the TLID command (refer to **Section 18**, "Tool Life Management") is output to index the turret.

Whether the overload alarm occurs or the NC flag is turned on when the first limit level is exceeded is determined by the setting of optional parameter (bit) data.

Optional parameter (bit) No. 6, bit 6

0 . . . NG flag is turned on and alarm does not occur.
1 . . . Alarm C (overload in load monitoring)

Note 1: *When the setting of optional parameter (bit) No. 6 is "0", setting of optional parameter (bit) No. 4 bit 3* must be "1".*

* *Tool life management - life determination condition NG in gauging*

Note 2: *Even when NG flag ON is selected, level A alarm occurs if load detected exceeds the second limit level.*

8. Setting of Percent Value at the Maximum Input of Spindle and M-Tool Spindle Drive Motors

After the control software has been loaded, it is necessary to set the proper values at the optional parameter (word) depending on the motor capacity and type. Setting should be made at No. 65 and No. 66 for the spindle and the M-tool spindle, respectively. If the value at these numbers is zero, an alarm of level D occurs (W02). However, if the spindle overload monitoring specification is not selected, setting of the former parameter is unnecessary. If the spindle overload monitoring and multiple-machining specification are not selected, it is not necessary to set the M-tool spindle parameter.

Type	Motor Capacity kW (hp)	Models	Setting Value (%)
DC Motor	55 (75)	LC50 spindle	185
	45 (60)	LC40, LC50 spindle	185
	37 (50)	LC40, LC50, LH55 spindle	193
	22 (30)	LC30, LC40 spindle	182
	15 (20)	LC30 spindle	182
	11 (15)	LC30 spindle	184
VAC Motor	3.7 (5)	LB9, FTL10 spindle, LC40M-ATC M-tool spindle, LR35-M/LR45-M-tool spindle	180
	5.5 (7.5)	LB15, LP15, FTL15, LR10, LR45-M-tool spindle	164
	7.5 (10)	LB15, LP15, LS30N, LB9, LB12	180
	11 (15)	LR15, LS30N, LB25	164
	15 (20)	LR15, LH35, LR25	180
	22 (30)	LR25, LR35	164
	30 (40)	LR35	150
	37 (50)	LR45, LH55	150

9. Parameters

Optional Parameter (Word)

No. 17 . . . . Initial Value : 10 (Setting Range: 0 - 50)

Function : Load monitoring immune time

This value is set in units of 0.1 sec. 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.

No. 58 . . . . Initial Value : 16 (Setting Range: 4 - 80)

Function : 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.

No. 59 . . . . Initial Value : 10 (Setting Range: 1 - 999)

Function : Overload alarm duration

If the cutting load detected 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 value x 12.8 msec = Overload alarm duration

When the setting is 10 msec, the overload alarm value is calculated as 128 msec.

No. 60 . . . . Initial Value : 5 (Setting Range: 1 - 999)

Function : Tool breakage alarm duration

If the cutting load detected 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.

No. 61 . . . . Initial Value : 0 (Setting Range: 0 - 195)

Function : 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%.

No. 62 . . . . Initial Value : 100 (Setting Range: 5 - 200)

Function : 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.

No. 63 . . . . Initial Value : 110 (Setting Range: 100 - 200)

Function : 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:

$$\text{Reference level (\%) (max. value within monitoring part)} \times \frac{\text{Setting value}}{100} = \text{First limit level (\%)}$$

No. 64 . . . . Initial Value : 120 (Setting Range: 100 - 200)

Function : Automatic setting of the second limit level

The second limit level in the load monitoring and automatic setting mode (model cutting) is calculated by the following equation:

$$\text{Reference level (\%) (max. value within monitoring part)} \times \frac{\text{Setting value}}{100} = \text{Second limit level (\%)}$$

No. 65 . . . . Initial Value : 0 (Setting Range: 0 - 500)

Function : 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 in the table in 8., "Setting of Percent Value at the Maximum Input of Spindle and M-Tool Spindle Drive Motors". If the setting is "0", an alarm of level D (W02) occurs.

No. 66 . . . . Initial Value : 0 (Setting Range: 0 - 500)

Function : Percent value at the maximum input of the load monitoring M-tool spindle

After loading the control software, the optimum value must be set in accordance with the spindle drive motor capacity and type as indicated in the table in 8., "Setting of Percent Value at the Maximum Input of Spindle and M-Tool Spindle Drive Motors". If the setting is "0", an alarm of level D (W02) occurs.

Note that the setting of this parameter is unnecessary if the spindle monitoring specification and multiple-machining specification are not selected.

10. Alarm Messages

ALARM-A

256 Load monitor tool break down

During monitoring, cutting load has exceeded the second limit level continuously for a preset length of time.

Index : TURRET

Code : None ..... Z-axis
1 ..... X-axis
2 ..... C-axis
3 ..... Spindle
4 ..... M-tool spindle
5 ..... W-axis
6 ..... Sub spindle

257 Load monitor command execution

An attempt to execute the monitoring command while the monitoring is in progress.

Cancel the monitoring command first before designating the monitoring in the next monitoring part.

Index : TURRET

Code : None

ALARM-C

989 Load monitor overload

During monitoring, cutting load has exceeded the first limit level continuously for a preset length of time.

Index : TURRET

Code : Same as Alarm-A 256

ALARM-D

W02 Load monitor spindle parameter non set

Spindle input max. value parameter (optional parameter (word) No. 65, 66) is 0.

Index : None

Character-string : None

Code : None

SECTION 7 TAPE CONVERSION FUNCTION (OSP3000L → OSP5020L/OSP5000L-G/OSP500L-G)

1. Overview

This function is used to convert programs prepared for OSP3000L and stored in a bubble memory into those applicable to OSP5020L/OSP5000L-G/OSP500L-G.

Items converted are:

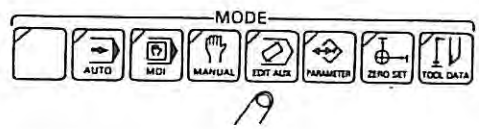
(1) Simultaneous 4-axis control program
(2) Constant speed cutting program
(3) Address characters specifying variable pitch
(4) Endless operation
(5) Nose radius compensation (end up) function
(6) LAP

2. Operation

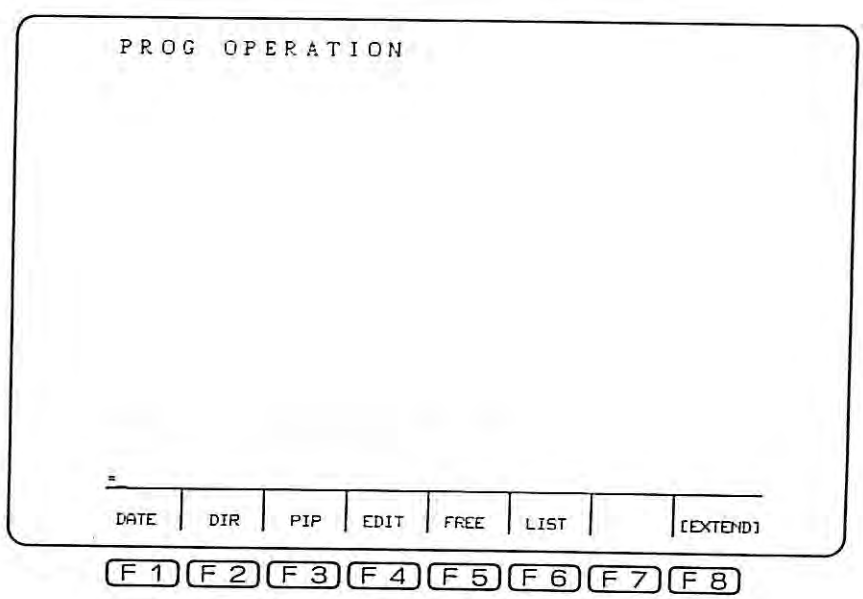
For the store programs prepared for OSP3000L, refer to the instructions provided in Section 4, 2-1. (1) of the Operation Manual for OSP5020L/OSP5000L-G/OSP500L-G (Publication No. 3272-E).

Conversion of the program can be made in the following procedure:

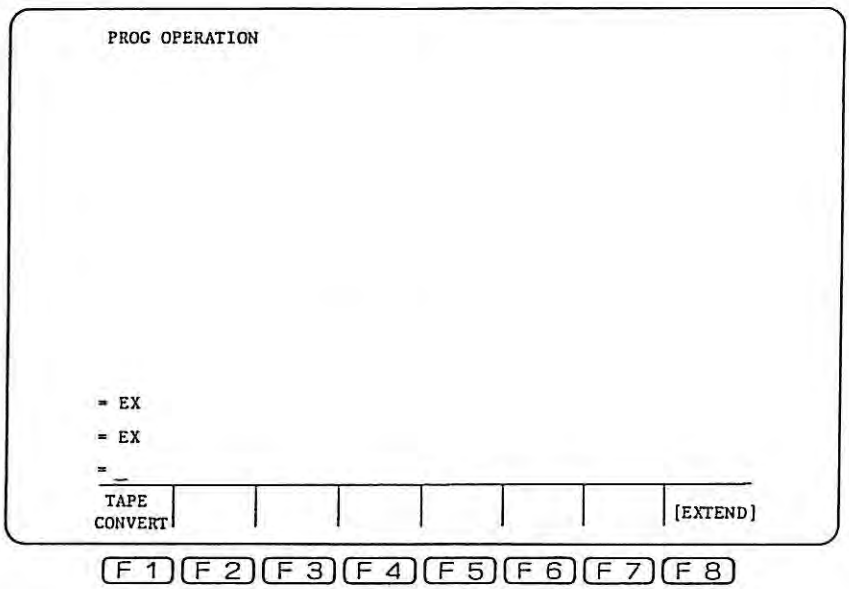
1) Press the **EDIT AUX** key to select the program operation mode.



2) The CRT display is shown below:



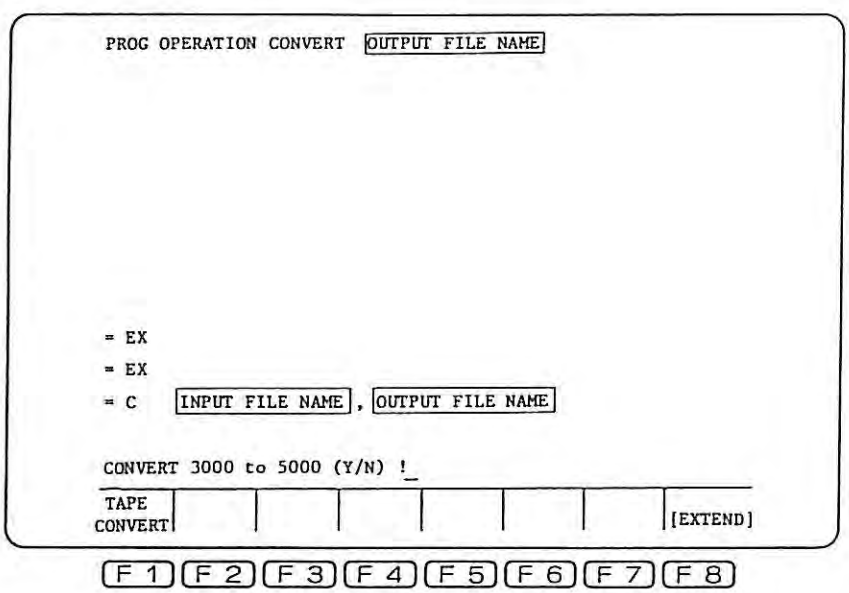
3) Press the function key [F8] (EXTEND) two times and the CRT display on the function line changes as shown below:



4) Press the function key [F1] (TAPE CONVERT).
The CRT shows "C = ".

5) Enter "input-file-name" and "output-file-name" through the keyboard.
<input-file-name>, <output-file-name>

6) Press the **WRITE** key. The CRT display changes as shown below:



7) When conversion is to be made, key in "Y" and press the **WRITE** key. When conversion is not required, key in "N" instead of "Y" and press the **WRITE** key.

CAUTION

(1) When only "input file name" is designated, a program for OSP5020L/OSP5000L-G/OSP500L-G assigned with the same file name is created and the program for OSP3000L is erased.

(2) For other cases, designate both "input file name" and "output-file-name".

(3) When the input file assigned with the designated file name is not found, message "file not found" appears on the CRT and tape conversion operation terminates.

(4) If the output file assigned with the file name presently designated already exists, message "file exist over write (Y/N)!" appears. Pressing the WRITE key after keying in "Y" erases the old file and tape conversion is executed.

Pressing the WRITE key after keying in "N" terminates the tape conversion operation without executing conversion.

(5) To check the results of the converted program(s), refer to the instruction in Section 4, 2-3. (4) of the Operation Manual for OSP5020L/OSP5000L-G/OSP500L-G (Publication No. 3272-E).

(6) When a new command value to be generated in tape conversion process cannot be calculated, "?" is entered at the location where such new data is to be entered.

3. Contents of Conversion

3-1. Simultaneous 4-Axis Control Program

(1) Simultaneous 4-axis Control Part

a) M codes calling for start and end of simultaneous 4-axis control, M11 and M10, are deleted.

b) For Program Stop and Optional Stop M codes, M00 and M01, P code with the numeral value "1" larger than the value of the P code just preceding it is added for respective turrets to synchronize execution of these M codes.

c) P codes already designated for respective turrets are renewed in the manner that the synchronization is maintained.

(2) Independent Control Part

a) For G codes selecting turret, G13 and G14, P code with the numeral value "10" larger than the value of the P code just preceding it is added.

b) The first P code, appearing in the independent cutting mode after completing simultaneous 4-axis control, is assigned with the numeral value "10" larger than the value of the P code having the largest value in the preceding simultaneous 4-axis control program.

Note 1: *G13 and G14 should be designated in one block without other commands.*

Note 2: *M00 and M01 should be designated for both turrets A and B.*

Example:

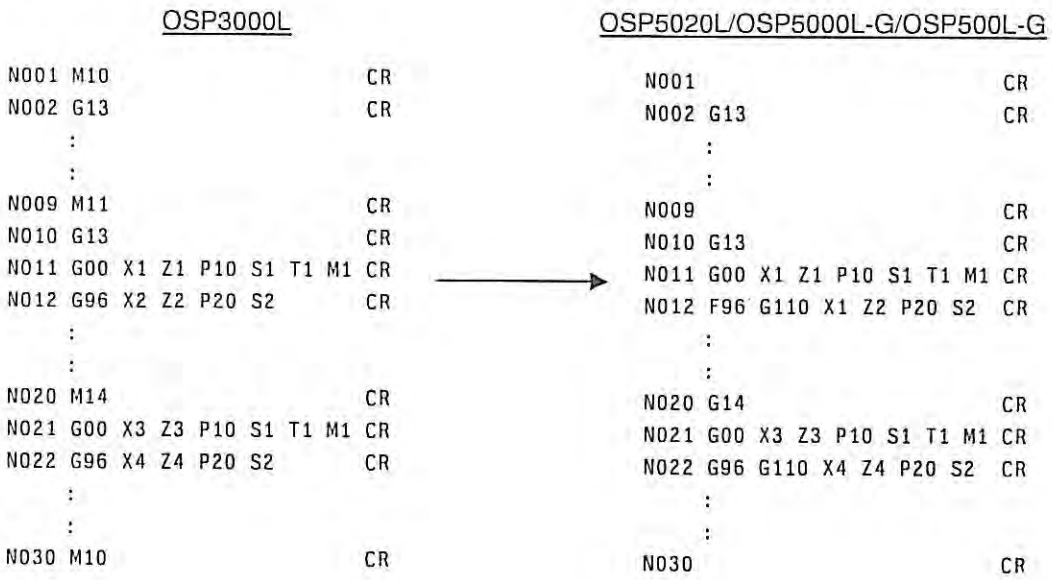
OSP3000L		OSP5020L/OSP500L-G/OSP5000L-G	
N001 M11	CR	N001	CR
N002 G13	CR	N002	CR
N003 G00 X1 Z1 P1 M1 T1 S1	CR	N003 G00 X1 Z1 P10 M1 T1 S1	CR
	:		:
	:		:
N014 M01	CR	N014 M01 P11	CR
N015 G14	CR →	N015 G14	CR
N016 G00 X1 Z1 P1 M1 T2 S1	CR	N016 G00 X1 Z1 P10 M1 T2 S1	CR
	:		:
	:		:
N024 M01	CR	N024 M01 P11	CR
N025 M10	CR	N025	CR
N026 G14	CR	N026 G14	CR
N027 G00 X2 Z2 T3 S2 M2	CR	N027 G00 X2 Z2 T3 S2 M2 P21	CR

3-2. Constant Cutting Speed Program

In simultaneous 4-axis control program, the turret on which constant cutting speed (CCS) is effective is determined by the designated turret selection command effective just before entering simultaneous 4-axis control and G code indicated below is added following G96 calling for the CCS operation.

G110 . . . . . CCS on turret A (G13)
G111 . . . . . CCS on turret B (G14)

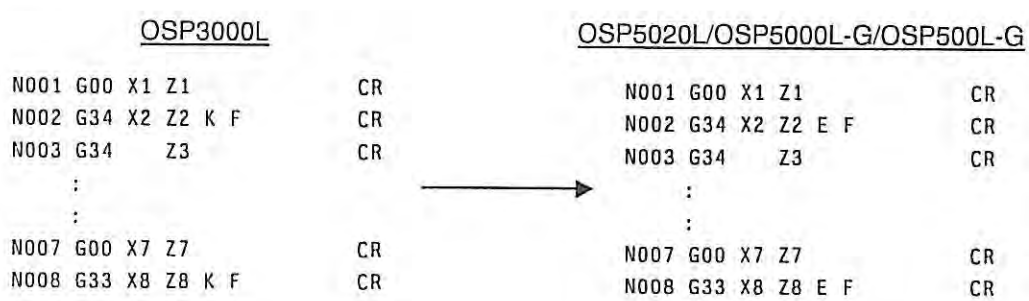
Example:



3-3. Address Characters Specifying Variable Pitch

The address character used to command pitch variation amount per pitch in cutting variable pitch thread is converted from "K" to "E".

Example:



3-4. Endless Operation

In tape conversion, code M31 is converted into code M02.

Example:

OSP3000L		OSP5020L/OSP5000L-G/OSP500L-G
N001 G13 CR	→	N001 G13 CR
:		:
:		:
N009 M31 CR		N009 M02 CR

3-5. Nose Radius Compensation Function

For G40 G00 calling for end up of the nose radius compensation function programmed for the OSP3000L, imaginary point (I, K) is created from X2, Z2 commanded in G40 G00 block and X1, Z1 provided in the preceding block with tape conversion.

I = X2 − X1

K = Z2 − Z1

Example:

OSP3000L		OSP5020L/OSP5000L-G/OSP500L-G
N009 G01 X1 Z1 CR	→	N009 G01 X1 Z1 CR
N010 G40 G00 X2 Z2 CR		N010 G40 G00 X2 Z2 I K CR

Note: G40 and G00 must be designated in the same block.

3-6. LAP

(1) The address character "L" expressing stock amount is converted into "U" or "W": "U" in longitudinal cutting and "W" in transverse (end face) cutting.

(2) In a program for OSP3000L, cutting conditions for roughing cycle are specified along with the commands specifying final contour. For OSP5020L/OSP5000L-G/OSP500L-G, contour defining commands containing both final contour and rough cut conditions are separately provided from the finish cut conditions; in actual cutting stage, finish cut conditions are called for from the contour defining commands.

Example:

OSP3000L

```
N001 G00 Xt Zt              CR
N002     Xs Zs Fr Sr Tr Mr  CR
N003 G85                    CR
N004 G00 Xt Zt L (U W) D    CR
N005     Xa Za Ff Sf Tf Mf  CR
N006 G01 Xb Zb              CR
N007     Xc Zc              CR
N008 G80                    CR
N009 G00 Xt Zt              CR
```

→

OSP5020L/OSP5000L-G/OSP500L-G

```
N001 G00 Xt Zt              CR
N002     Xs Zs Fr Sr Tr Mr  CR
N003 G85 NLA01 U D          CR
N004 G00 Xt Zt              CR
Sf   Tf  Mf                 CR
G87  NLA01                  CR
NLA01    G81                CR
N005     Xa Za Ff           CR
N006 G01 Xb Zb              CR
N007     Xc Zc              CR
N008 G80                    CR
N009 G00 Xt Zt              CR
```

(3) Thread cutting cycle commanded by G33 is performed in a compound fixed cycle and continuous thread cutting cycle by G34 and G35 is performed in LAP.

For G33 thread cutting cycle which is converted into compound thread cutting cycle, G71 or G72 is added depending on thread cutting direction, longitudinal or transverse.

Example:

OSP3000L

```
N021 G00 X3 Z3 P10 S1 T1 M1  CR
N002     Xs Zs               CR
N003 G86                     CR
N004          D  E           CR
N005 G00 Xa Za               CR
N006 G34 Xb Zb K F           CR
N007     Xc Zc               CR
N008 G00 Xd Zd               CR
N009 G80                     CR
```

→

OSP5020L/OSP5000L-G/OSP500L-G

```
N001 G00       S1 T1 M1      CR
N002     Xs Zs               CR
N003 G88 NLA01 H  D          CR
NLA01       G81              CR
N005 G00 Xa Za               CR
N006 G34 Xb Zb E  F          CR
N007     Xc Zc               CR
N008 G00 Xd Zd               CR
N009 G80                     CR
```

Example:

OSP3000L

```
N001 G00       S  T  M      CR
N002     Xs Zs               CR
N003 G86                     CR
N004     D  E0               CR
N005 G33 Xb Zb (I K) F       CR
N006 G80                     CR
N007     Xt Zt               CR
```

→

OSP5020L/OSP5000L-G/OSP500L-G

```
N001 G00          S  T  M  CR
N002     Xs Zs               CR
N003 G71 Xb Zb I D H E1 F    CR
N007     Xt Zt               CR
```

Example:

OSP3000L

```
N001 G00          S  T  M      CR
N002     Xs Zs                 CR
N003 G86                       CR
N004 G82                       CR
N005     D  E0                 CR
N006 G33 Zb Xb   (I  K) F      CR
N007 G80                       CR
N008     Xt Zt                 CR
```

OSP5020L/OSP5000L-G/OSP500L-G

```
N001 G00       S  T  M      CR
N002     Xs Zs              CR
N003 G72 Xb Zb K D H E1 F   CR
N008     Xt Zt              CR
```

(4) With OSP5020L/OSP5000L-G/OSP500L-G, grooving and drilling cycles are performed not in LAP but in compound fixed cycle.

When grooving and drilling cycles are performed in compound fixed cycle, G code is converted into G73 or G74 depending on direction of cutting, longitudinal or transverse.

Example:

OSP3000L

```
N001 G00 Xt Zt S1 T1 M1   CR
N002     Xs Zs            CR
N003 G83                  CR
N004     D  E             CR
N005 G01 Xa Za Fr         CR
N006 G80                  CR
N007 G00 Xt Zt            CR
```

OSP5020L/OSP5000L-G/OSP500L-G

```
N001 G00 Xt Zt S1 T1 M1   CR
N002     Xs Zs            CR
N003 G73 Xa Za K  D  Fr   CR
N007 G00 Xt Zt            CR
```

Example:

OSP3000L

```
N001 G00 Xt Zt S1 T1 M1   CR
N002     Xs Zs            CR
N003 G83                  CR
N004 G82                  CR
N005     D  E             CR
N006 G01 Xa Za Fr         CR
N007 G80                  CR
N008 G00 Xt Zt            CR
```

OSP5020L/OSP5000L-G/OSP500L-G

```
N001 G00 Xt Zt S1 T1 M1   CR
N002     Xs Zs            CR
N003 G74 Xa Za I D Fr     CR
N004 G00 Xt Zt            CR
```

Note: When stock "L" or "U" or "W" is "0", finish cut cycle (G87) is not carried out.

4. Restrictions on Tape Conversion Function (OSP3000L→OSP5020L/OSP5000L-G/OSP500L-G)

The OSP5020L/OSP5000L-G/OSP500L-G has the tape conversion function that converts a part program for OSP3000L into one compatible to OSP5020L/OSP5000L-G/OSP500L-G. However, this function cannot convert all OSP3000L part programs; some minor (not explained in the Programming Manual) or special specifications cannot be converted.

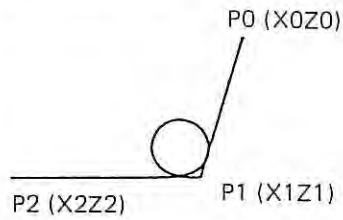
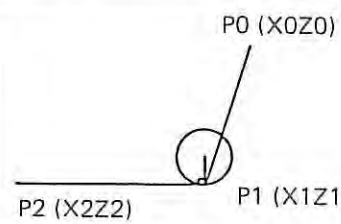
Generally, program items not explained in this section are output in the original OSP3000L form even after passing through the tape convert.

Therefore, specifications not listed as convertible among those modified according to the change of the control from OSP3000L to OSP5020L/OSP5000L-G/OSP500L-G are not accepted or executed by OSP5020L/OSP5000L-G/OSP500L-G.

Inconvertible Specifications:

Specifications	OSP3000L	OSP5020L/OSP5000L-G/ OSP500L-G
Programming Format Command value "0"	Value "0" is omissible and just "X" or "Z" is acceptable.	Value "0" is not omissible and program must be made as "0" and "Z0".
+/− sign	"+" and "−" signs may be placed at any position before the next address character. See example below: X100−Z1+00	The signs must be placed right before the numeral value as below: X−100 Z+100
Space	Spaces may be placed at any position. See example below: X10 0 Z 1 00	Spaces must not be placed between numerals or between numerals and characters: X10 0Z 100 (not permissible)
M02	Leading zero is omissible and "M2" is acceptable.	Leading zero is not omissible and the code must be given as "M02".
/ (slash)	The slash code may be placed at any position in a block. The commands after the slash code are ignored. N001 G00 X100 /Z100	The slash code must be placed either at the beginning of each block or right after a sequence number. Otherwise, it is interpreted as a division calculation. N001 /G00 X100 Z100

Specifications	OSP3000L	OSP5020L/OSP5000L-G/ OSP500L-G
Programming Format % code	Commands may be programmed in the first sequence following the % code. % N001 G00 ...	The % code must be followed by an end of block command, LF or CR. % CR (LF) N001 G00 ...
Thread Cutting 8 times F command	Combined with M27, a thread lead is programmed following address character F in 8 times the actual lead. FOOO M27	Codes M27 and M28 cannot be programmed since numerical values smaller than 0.001 mm are programmable. FO.OOOOO
10 times F command	Combined with M28, a thread lead is programmed following address character F in 10 times the actual lead. FOOO M28	
Designation of reference axis of thread lead	For a thread having an angle of 45 deg. or larger to Z-axis, its lead is referenced to X-axis.	Code M27 is used to change reference axis of thread lead from Z-axis to X-axis.
Simultaneous 4-axis Cutting G13 and G14	Axis movement commands and other commands can be programmed in the same block containing G13 or G14 selecting the turret used for cutting.	Both G13 and G14 must be specified in a block independently.
M00 and M01	These M codes may be programmed only for one turret.	These M codes must be programmed for both turrets A and B.
Mirror Image G13 and G14	Axis movement commands and other commands can be programmed in the block containing G13 or G14.	Both G13 and G14 must be specified in a block independently.

Specifications	OSP3000L	OSP5020L/OSP5000L-G/ OSP500L-G
Nose Radius Compensation Start-up N0 G00 X0 Z0 N1 G42 X1 Z1 N2 G01 X2 Z2 F1	 With the commands in block N1, the cutting tool is positioned so that the tool nose comes to contact both straight lines P0P1 and P1P2.	 With the commands in block N1, the cutting tool is positioned so that the tool nose comes to contact the straight line P1P2 at point P1.
LAP Sequence branch	Sequence to which jump or branch is made after the completion of rough or finish cutting cycle is specified following address character Q.	Since commands of contour or shape definition, rough cutting cycle, and finish cutting cycle are independently programmed, no sequence branch function is available.

Inconvertible special specifications of OSP3000L

- Parameter program (user task)
- Tool life management
- Gauging functions (work gauging, tool gauging, post-process gauging)
- Robot
- Programmable tailstock function
- Bar feeder
- Multiple machining model (C-axis linkage, C-axis commands, fixed cycle)
- Scheduled operation
- Overload detection on Z-axis

Other functions available only on a specific model and machine designed for custom-use generally cannot be converted.

SECTION 8 GAUGING DATA PRINTOUT FUNCTION

1. Overview

This function prints out the results of work gauging, master ring gauging, and tool gauging cycle.

Information covered in this section includes:

(1) Conditions for data printout

(2) Contents of printout data

(3) Printout format

(4) Interface for RP-80 link

a) Parameter setting at OSP5020L/OSP500L-G

b) Switch setting at RP-80 or SP-500

c) Connection of RP-80 or SP-500 to OSP5020L/OSP500L-G

2. Conditions for Data Printout

(1) The gauging data is printed out when the gauging cycles such as work gauging cycle, master ring gauging cycle, and tool gauging cycle have been executed.

(2) The term printout includes two types of data output form as printing out as hard copy and creation of a file*. Parameter (bit) data is used to select whether the gauging data is to be printed out or not.

Parameter (bit) No. 5	Gauging Data Print Out
Bit 3 = 1	ON
Bit 3 = 0	OFF

Furthermore, selection of printout form is possible by parameter (bit) data.

Parameter (bit) No. 5	File Creation/Printer
Bit 6 = 1	File creation
Bit 6 = 0	Printer

* File creation: Gauging data is stored in the bubble memory in the format used for printing out the data as hard copy. The file name is A.MDF. Printout and file creation cannot take place simultaneously.

(3) Data printout is turned on and off by the MAN/AUTO selector switch on the operation panel and print out is possible when the switch is set it in the AUTO position.

(4) Interface type for print out is selectable between the RS232C (serial) and centronics (parallel) by the setting of parameter (word) data.

Parameter (word) No. 45 =	0	CN0: (TT:)	RS232C
	1	CN1:	
	2	CN2:	
	3	CN3:	
	4	CN4:	
	5		Centronics

(5) Interface boards required for individual output ports are indicated below.

Centronics	.....	Data board
RS232C CN0: (TT:)	.....	Main card 3 or 13
CN1: - CN4:	.....	RS board

3. Contents of Print Out Data

Print out data includes the following items:

(1) TOFF. NO.

Tool offset number for which gauging cycle has been executed.

(2) COMMAND

Target value of gauging cycle

(3) ACTUAL

Result of gauging cycle

(4) LAST-DATA

Tool offset value (zero offset value) in the previous gauging cycle

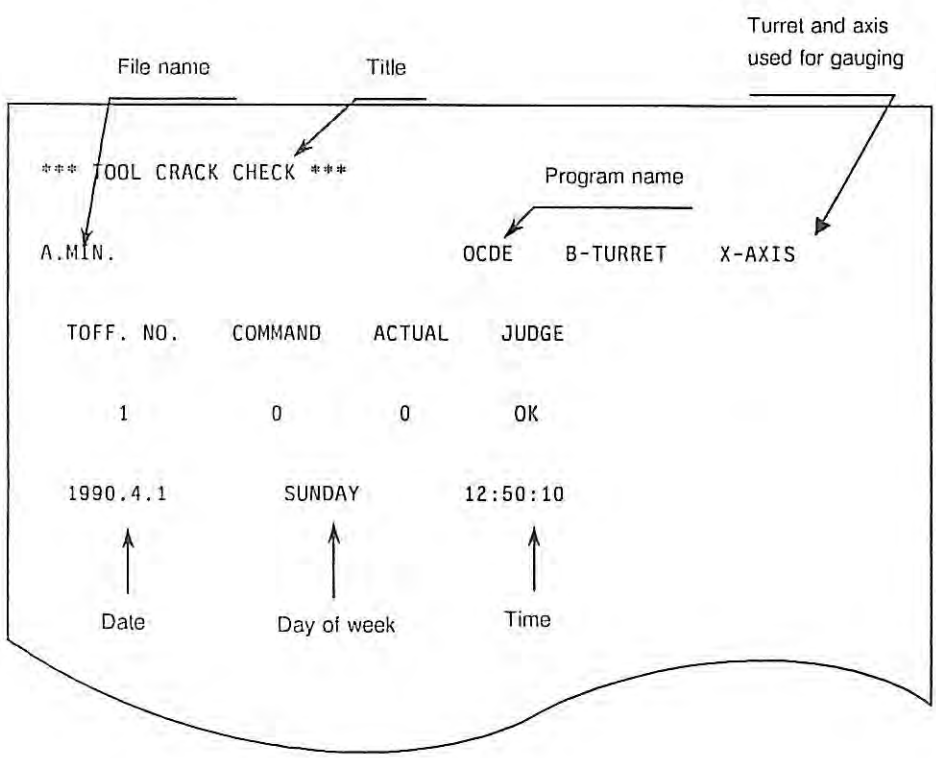
(5) NEW-DATA

Tool offset value (zero offset value) compensated for in accordance with the result of present gauging cycle

(6) JUDGE

Evaluation results (OK, ±OK, ±NG)

On the CRT, title, gauging file name, program name, turret, axis, date and time are displayed in addition to the gauging data indicated above.



4. Printout Format

(1) Work Gauging Cycle

```
*** TOOL OFFSET COMPENSATION ***

A.MIN.                          OABC    A-TURRET    X-AXIS

  TOFF. NO.  COMMAND   ACTUAL   LAST-DATA   NEW-DATA   JUDGE

      1      100.000   0.000      0.000       0.000      OK

 1990.4.1          SUNDAY          12:10:05
```

(2) Master Ring Gauging Cycle

```
*** ZERO OFFSET COMPENSATION ***

A.MIN.                          OBCD    A-TURRET    Z-AXIS

  COMMAND   ACTUAL   LAST-DATA   NEW-DATA   JUDGE

  100.000   0.000      0.000       0.000      OK

 1990.4.1          SUNDAY          12:50:10
```

(3) Tool Gauging Cycle

```
*** TOOL CRACK CHECK ***

A.MIN.                  OCDE    B-TURRET    X-AXIS

  TOFF. NO.  COMMAND   ACTUAL   JUDGE

   10        200.000    1.500     NG

 1990.4.1          SUNDAY          12:50:20
```

5. Interface for RP-80 Link

Type RP-80 or SP-500 printer is the standard printer for printing out gauging data. If a printer of other types is used, the printer must meet the conditions indicated below.

5-1. Parameter Setting at OSP5020L/OSP500L-G

(1) Main Card 3 or 13

a) Baud rate setting

Baud rate at CN0: (TT:) is set at the parameter (word) data No. 39. Set the baud rate at 2400. Although the baud rate can be set in a range of 110 to 19,200 at this parameter, the baud rate actually set is as indicated below:

Setting at parameter (word) No. 39, X	Baud rate actually set
X = 110	110
110 < X ≦ 150	150
150 < X ≦ 300	300
300 < X ≦ 600	600
600 < X ≦ 1200	1200
1200 < X ≦ 2400	2400
2400 < X ≦ 4800	4800
4800 < X ≦ 9600	9600
9600 < X ≦ 19200	19200

The initial setting value after the loading of NC control program software is 4,800 baud.

b) Communication condition setting

Character construction, parity and other conditions for data communication are set at the parameter (bit) data No. 12.

	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
No. 12	*	*	*	8 bit JIS	Even parity	Parity check	Ready signal none	Stop bit 1 check

Bits 7 through 5 marked by an asterisk (*) are not used to set the conditions of print out function.

Bits 5, 4, and 1 are initially set.

Initial setting for CN0: is: No parity
Ready signal not used
Stop bit 2 bits
8 bit JIS

(2) RS Board

a) Baud rate setting

Baud rate at CN1: through CN4: are set at parameter (word) data No. 40 through No. 43. Setting range of baud rate is from 110 to 19,200 and initial setting is 4,800 baud. Set the baud rate at 2,400.

No. 40 . . . . . CN1:
No. 41 . . . . . CN2:
No. 42 . . . . . CN3:
No. 43 . . . . . CN4:

b) Communication condition setting

Character construction, parity and other conditions for data communication for CN1: through CN4: are set at parameter (bit) data No. 13 through No. 16.

	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
No. 13 (CN1:)	*	*	*	8 bit JIS	Even parity	Parity check	Ready signal none	Stop bit 1 check
No. 14 (CN2:)	*	*	*	8 bit JIS	Even parity	Parity check	Ready signal none	Stop bit 1 check
No. 15 (CN3:)	*	*	*	8 bit JIS	Even parity	Parity check	Ready signal none	Stop bit 1 check
No. 16 (CN4:)	*	*	*	8 bit JIS	Even parity	Parity check	Ready signal none	Stop bit 1 check

Bits 7 through 5 marked by an asterisk (*) are not used to set the conditions of print out function.

Bits 5, 4 and 1 are intially set.

When the parameter (word) and/or parameter (bit) data have been changed, turn off power supply once about three minutes after the setting of new data. Turn on power supply again then.

5-2. Switch Setting at RP-80

(1) Control Board

Switch No.	Setting	Function
SW1-1	OFF	Paper out detection
SW1-2	OFF	Buzzer ON *1
SW1-3	ON	Paper feed 1/6 inches per line
SW1-4	OFF	Print mode selection
SW1-5	OFF	
SW1-6	OFF	Standard
SW1-7	OFF	Print character mode selection
SW1-8	OFF	Normal (Pica)

Switch No.	Setting	Function
SW2-1	OFF	Character zero "0"
SW2-2	ON	SLCT IN internally fixed
SW2-3	OFF	Automatic carriage return by LF
SW2-4	OFF	1 inch skip over perforation ineffective

*1 With a mechanical error, the buzzer will sound for about three seconds disregarding of the setting of SW1-2. The SW1-2 is used to turn on the buzzer in response to the BEL code and paper out.

(2) RS232C Option Interface Board

Switch No.	Setting	Function
SW1-1	ON	Baud rate 2400 BPS
SW1-2	ON	
SW1-3	OFF	
SW1-4	OFF	
SW1-5	OFF	Data buffer 1936 bytes
SW1-6	OFF	
SW1-7	ON	No parity check
SW1-8	ON	Even parity

Switch No.	Setting	Function
SW2-1	OFF	8 bit data length
SW2-2	ON	Reverse channel = mark with the serial data input inhibited condition
SW2-3	OFF	
SW2-4	ON	Reverse channel effective

5-3. Switch Setting at RP-80 II

(1) Control Board

Switch No.	Setting	Function
SW1-1	OFF	Paper out detection
SW1-2	OFF	Not used
SW1-3	ON	Paper feed 1/6 inches per line
SW1-4	OFF	Print mode selection
SW1-5	OFF	
SW1-6	ON	Standard
SW1-7	OFF	Print character mode selection
SW1-8	OFF	Normal (Pica)

Switch No.	Setting	Function
SW2-1	OFF	Character zero "0"
SW2-2	ON	SLCT IN internally fixed
SW2-3	OFF	Automatic carriage return
SW2-4	OFF	1 inch skip over perforation ineffective

(2) RS232C Option Interface Board (Code No. 8143)

Switch No.	Setting	Function	Baud rate (BPS)	SW 1-1	SW 1-3	SW 1-4	SW 1-7
SW1-1	ON	Baud rate selection (Refer to the table to the right.)	75	ON	ON	ON	ON
			110	ON	OFF	ON	ON
SW1-2	OFF	8 bit data length	134.5	ON	ON	OFF	ON
			150	ON	OFF	OFF	ON
SW1-3	OFF	Baud rate selection (Refer to the table to the right.)	200	OFF	ON	ON	ON
			300	OFF	OFF	ON	ON
SW1-4	ON	Baud rate selection (Refer to the table to the right.)	600	OFF	ON	OFF	ON
			1,200	OFF	OFF	OFF	ON
SW1-5	ON	Even Parity	1,800	ON	ON	ON	OFF
			2,400	ON	OFF	ON	OFF
SW1-6	OFF	No parity check	4,800	ON	ON	OFF	OFF
			9,600	ON	OFF	OFF	OFF
SW1-7	OFF	Baud rate selection (Refer to the table to the right.)	300	OFF	ON	ON	OFF
			1,200	OFF	OFF	ON	OFF
SW1-8	ON	Serial I/F	2,400	OFF	ON	OFF	OFF
			9,600	OFF	OFF	OFF	OFF

5-4. Switch Setting at SP-500

(1) Control Board

Switch No.	Function	ON	OFF	Setting
SW1-1	–	–	–	OFF
SW1-2	Character grade selection	NLQ	Draft	OFF
SW1-3	Not used	–	–	OFF
SW1-4	one inch/perforation	Effective	Not effective	OFF
SW1-5	Non-paper detector	Not effective	Effective	OFF
SW1-6	Paper feed amount selection	1/8"	1/6"	OFF
SW1-7	Reduced mode selection	Reduced character	Normal character	OFF
SW1-8	Print character mode selection	Elite (12 CPI)	Pica (10 CPI)	OFF
SW2-1	Character zero selection	0	0	OFF
SW2-2	Buzzer selection	Sounds	Does not sound	OFF
SW2-3	Emphasized mode selection	Emphasized character	Normal character	OFF
SW2-4	Automatic carriage return by CR (AUTO FEED XT)	Effective	Not effective	OFF

CAUTION

Never operate DIP switches (1 to 3) which are not in use.

(2) RS232C Option Interface Board

a) Code No. 8143

Switch No.	Setting	Function	Baud rate (BPS)	SW 1-1	SW 1-3	SW 1-4	SW 1-7
SW1-1	ON	Baud rate selection (Refer to the table to the right.)	75	ON	ON	ON	ON
			110	ON	OFF	ON	ON
SW1-2	OFF	8 bit data length	134.5	ON	ON	OFF	ON
			150	ON	OFF	OFF	ON
SW1-3	OFF	Baud rate selection (Refer to the table to the right.)	200	OFF	ON	ON	ON
			300	OFF	OFF	ON	ON
SW1-4	ON	Baud rate selection (Refer to the table to the right.)	600	OFF	ON	OFF	ON
			1,200	OFF	OFF	OFF	ON
SW1-5	ON	Even Parity	1,800	ON	ON	ON	OFF
			2,400	ON	OFF	ON	OFF
SW1-6	OFF	No parity check	4,800	ON	ON	OFF	OFF
			9,600	ON	OFF	OFF	OFF
SW1-7	OFF	Baud rate selection (Refer to the table to the right.)	300	OFF	ON	ON	OFF
			1,200	OFF	OFF	ON	OFF
SW1-8	ON	Serial I/F	2,400	OFF	ON	OFF	OFF
			9,600	OFF	OFF	OFF	OFF

* Same as RP-80 II .

Baud rate is 2,400 BPS with the above setting.

b) Code No. 8145

	SW1	Setting
1	Baud rate	ON
2		ON
3		OFF
4		OFF
5	Data buffer	ON
6		ON
7	No parity check	ON
8	Even parity	OFF

	SW2	Setting
1	8 bit data length	OFF
2		ON
3	Signal polarity	OFF
4	Reverse channel	ON

c) Code No. 8148

DIP Switch 1 (SW1) Setting:

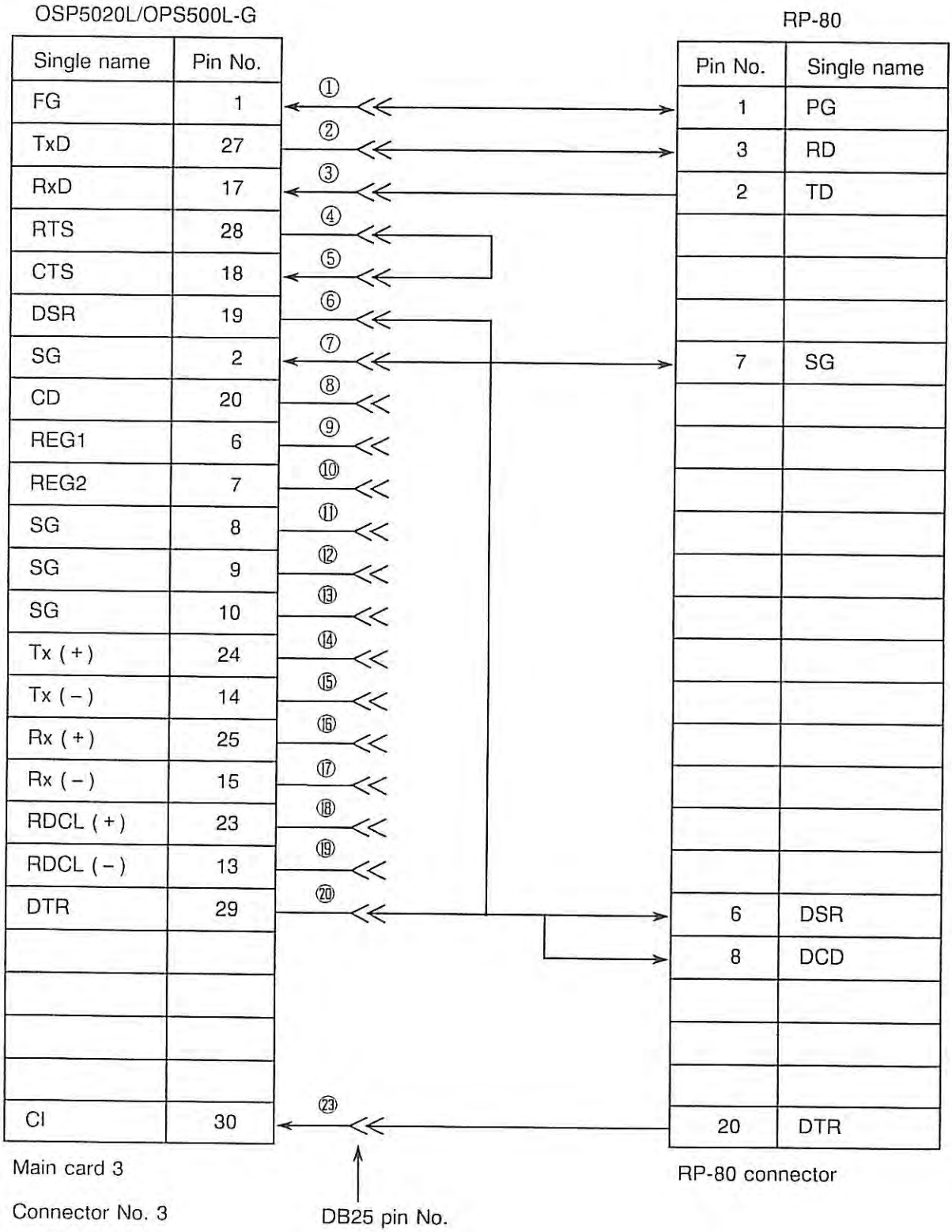
Switch No.	Function	ON	OFF	Setting
SW1-1	Data length	7 bit	8 bit	OFF
SW1-2	Setting of parity check	Refer to Parity Check Setting Table.		OFF
SW1-3				OFF
SW1-4	Flag polarity selection	Negative	Positive	OFF
SW1-5	Bit rate selection 3	Refer to Bit Rate Setting Table.		OFF
SW1-6	Bit rate selection 2			ON
SW1-7	Bit rate selection 1			ON
SW1-8	Bit rate selection 0			OFF

DIP Switch 2 (SW2) Setting:

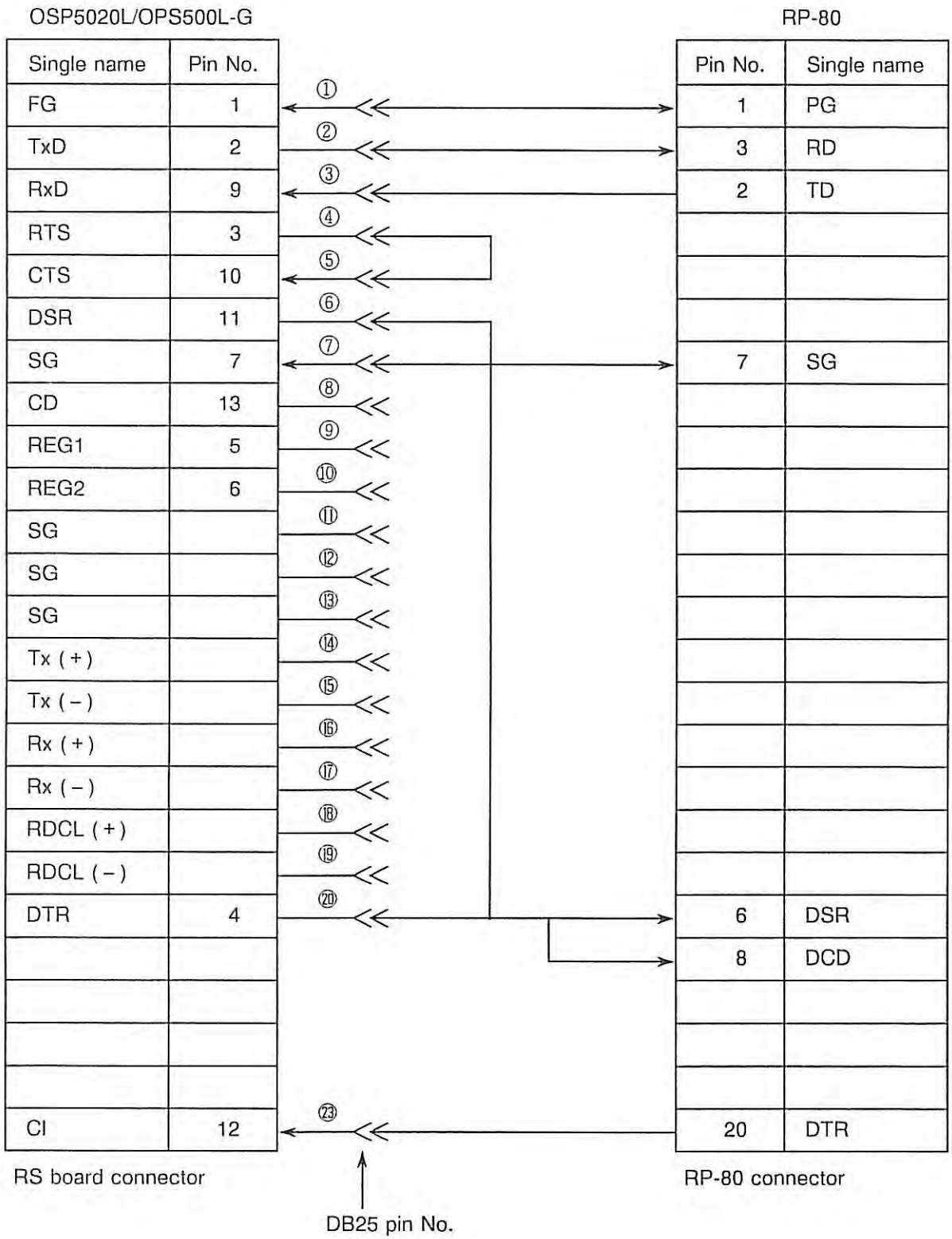
Switch No.	Function	ON	OFF	Setting
SW2-1	I/F board effective/not effective selection	Effective	Not effective	ON
SW2-2	Buffer operation	Effective	Not effective	ON
SW2-3	Buffer full recovery 1	Refer to Buffer Recovery Timing.		OFF
SW2-4	Buffer full recovery 0			OFF
SW2-5	Self test setting	Refer to Self Test Setting Table.		OFF
SW2-6				OFF

5-5. Connection of RP-80 to OSP5020L/OSP500L-G

(1) Main Card 3 or 13 CN0: (TT:)



(2) RS Board CN1: - CN4:

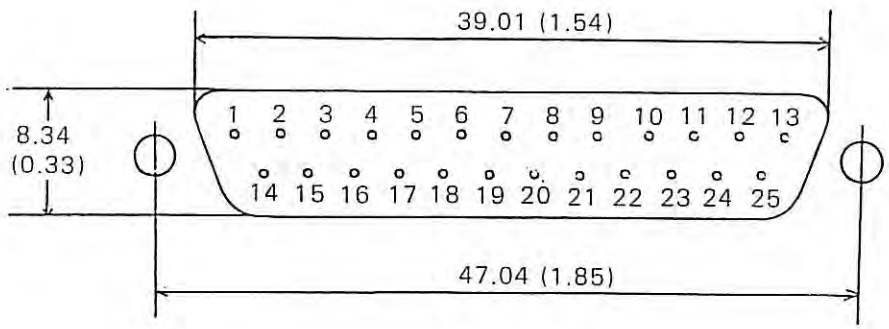


Connection to OSP5020L/OSP500L-G

Connections between OSP5020L/OSP500L-G and I/O Device

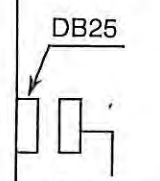
OSP5020L/OSP500L-G Control Box

Connector at OSP5020L/OSP500L-G: DB25 (male)



1	2	3	4	5	6	7	8	9	10	11	12	13
FG	TXD	RXD	RTS	CTS		SG						

14	15	16	17	18	19	20	21	22	23	24	25
						DTR			CI		



J.A.E. Connector: DB25 (standard shell)

Lock : D20418-J (hexagon type)

SECTION 9 SLIDE HOLD FUNCTION DURING THREAD CUTTING CYCLE (G34, G35, G112, G113)

1. Overview

Pressing the **SLIDE HOLD** pushbutton switch during the execution of the G34, G35, G112 and G113 thread cutting cycles retracts the cutting tool from the workpiece by the amount commanded by retraction amount (L)* and stops the cycle there.

The direction of tool retraction from the workpiece is determined from the direction of the thread cutting and the sign of a retraction amount command. After the thread cutting cycle has been stopped, the operation resumption is impossible until the control is reset.

* In X-axis direction, the retraction amount is two times the distance specified by L.

The axis to be taken as the thread lead reference axis is selected by a designated M code command.

M code	Reference Axis
M26	Z-axis
M27	X-axis

When the control is reset, the control is placed in the M26 mode.

Retraction amount command: G34, G35 L
G112, G113 CL

2. Program Examples

(1) Example 1 - Longitudinal Thread Cutting

See Fig. 9-1.

Program

```
N001  G00   X1   Z1                          CR
N002        X2   Z2   M03   M42   S10        CR
N003  G34   X3   Z3   F1    E1    L1         CR
N004        X4   Z4               L2         CR
N005        X5   Z5               L3         CR
N006  G00   X1   Z1                          CR
```

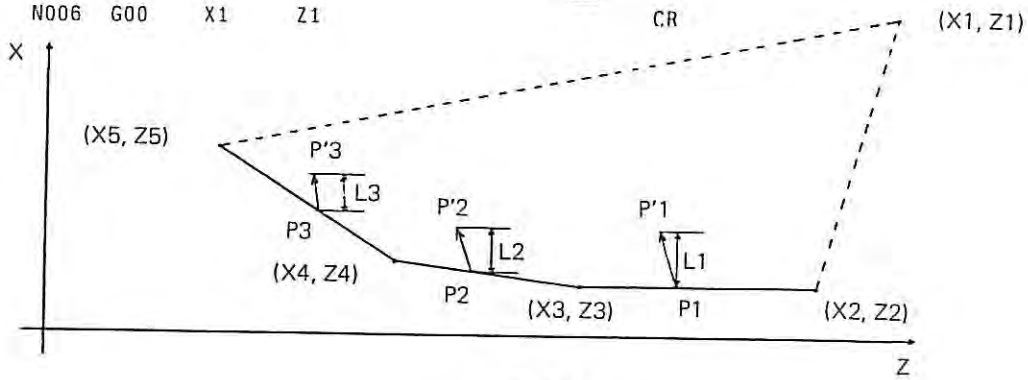


Fig. 9-1

P1 - P3 : SLIDE HOLD is pressed
P1' - P3' : Tool retracted point

(2) Example 2 - Longitudinal Thread Cutting

See Fig. 9-2.

Program

In this example, retraction amounts L1, L2 and L3 in example 1 is changed to −L1, −L2 and −L3. The cutting tool retracts in the direction opposite to the retraction direction in Example 1.

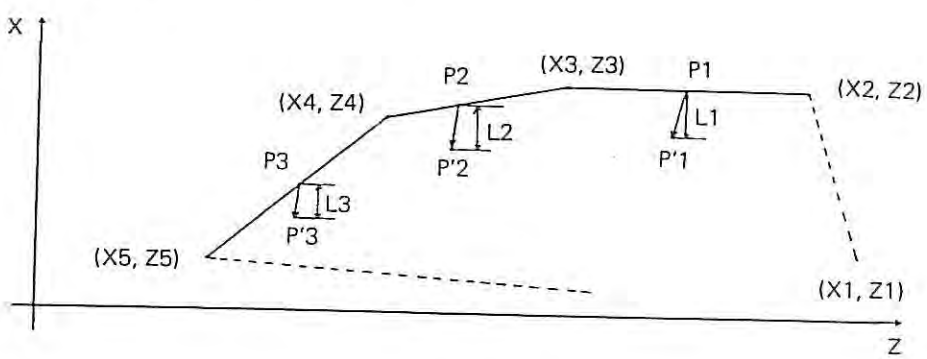


Fig. 9-2

(3) Example 3 - Face Thread Cutting Cycle

See Fig. 9-3.

Program

```
N001  G00   X1   Z1                              CR
N002        X2   Z2   M03   M42   S10           CR
N003  G34   X3   Z3   F1    E1    L1    M27     CR
N004        X4   Z4                L2    M27     CR
N005        X5   Z5                L3    M27     CR
N006  G00   X1   Z1                              CR
```

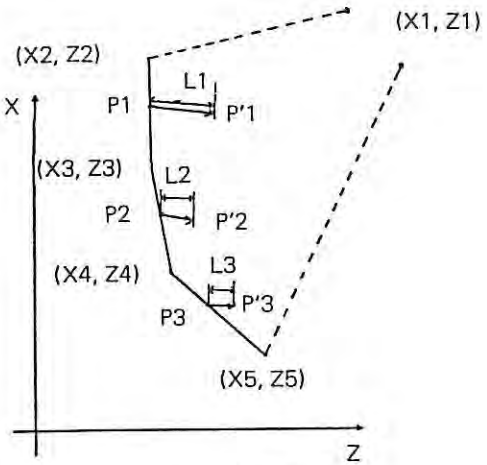


Fig. 9-3

(4) Example 4 - Face Thread Cutting Cycle

See Fig. 9-4.

Program

In this example, retraction amounts L1, L2 and L3 in example 3 is changed to −L1, −L2 and −L3. The cutting tool retracts in the direction opposite to the retraction direction in Example 3.

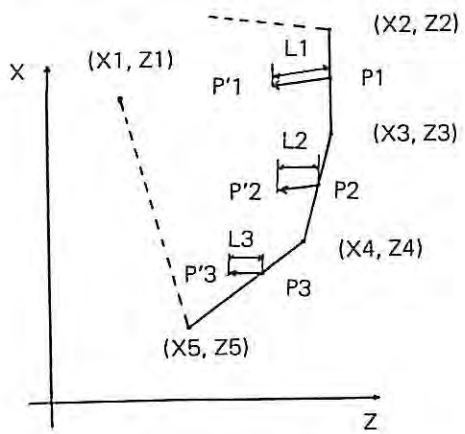


Fig. 9-4

P1 -P3 : SLIDE HOLD is pressed
P1' -P3' : Tool retracted point

3. Precautions

(1) The cutting tool may be retracted both in the positive and the negative direction in reference to the thread cutting reference axis. Therefore, the direction of tool retraction should be specified by the sign of an L code command.

(2) If the tool reaches the commanded end point during the retraction caused by the pressing of the **SLIDE HOLD** pushbutton switch, the retraction is made up to the commanded end point and then along the axis of tool retraction until the specified tool retraction amount is reached.

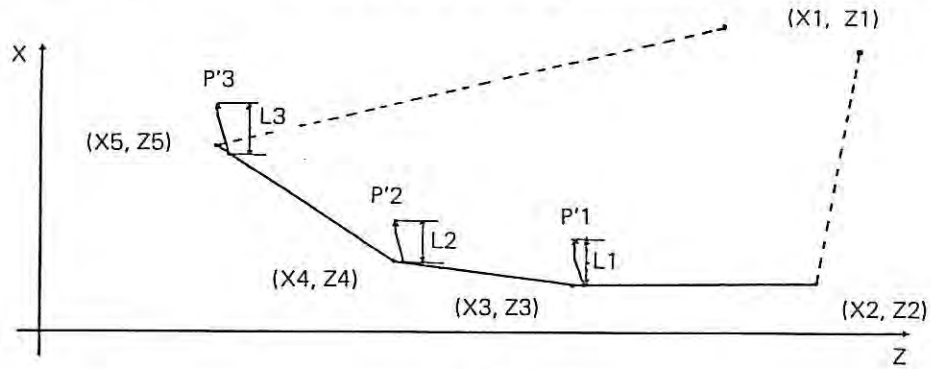


Fig. 9-5

(3) G112 and G113 are optional thread cutting commands on an arc.

The illustrations given are all for G34/G35 mode. Only the differences are that the tool path for thread cutting is an arc and that the CL is used for the retraction amount command. Tool retraction occurs in the same manner as in G34/G35 mode.

Example: Longitudinal cutting

```
N001  G00    X1   Z1                                        CR
N002         X2   Z2    M03   M42   S10                   CR
N003  G113   X3   Z3    I1    K1    F1    E1    CL1 CR
N004  G112   X4   Z4    I2    K2                CL2 CR
N005  G112   X5   Z5    I3    K3                CL3 CR
N006  G00    X1   Z1
```

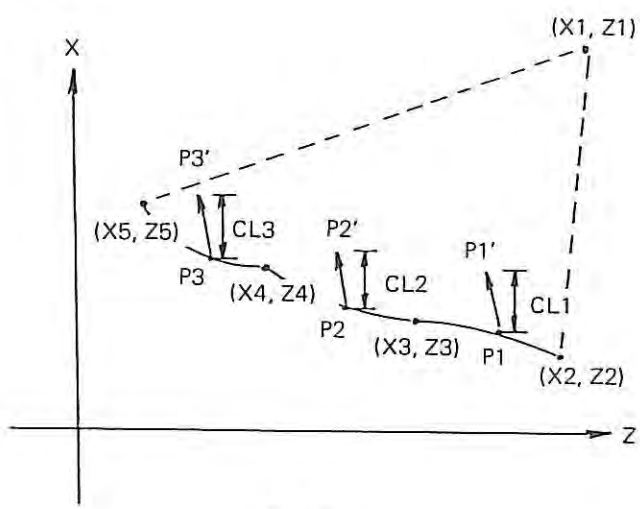


Fig. 9-6

SECTION 10 PITCH ERROR COMPENSATION FOR X, Z AND C AXES

1. Overview

The pitch error compensation function is provided to improve positioning accuracy of X, Z and C axes.

Information covered in this section includes:

(1) Data Setting

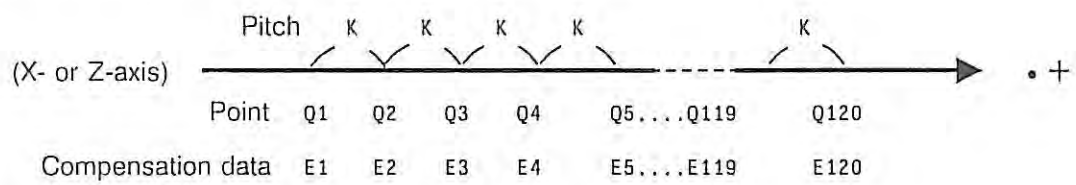
(2) Compensation Method

2. Data Setting

2-1. Data to be Set

(1) X and Z axes

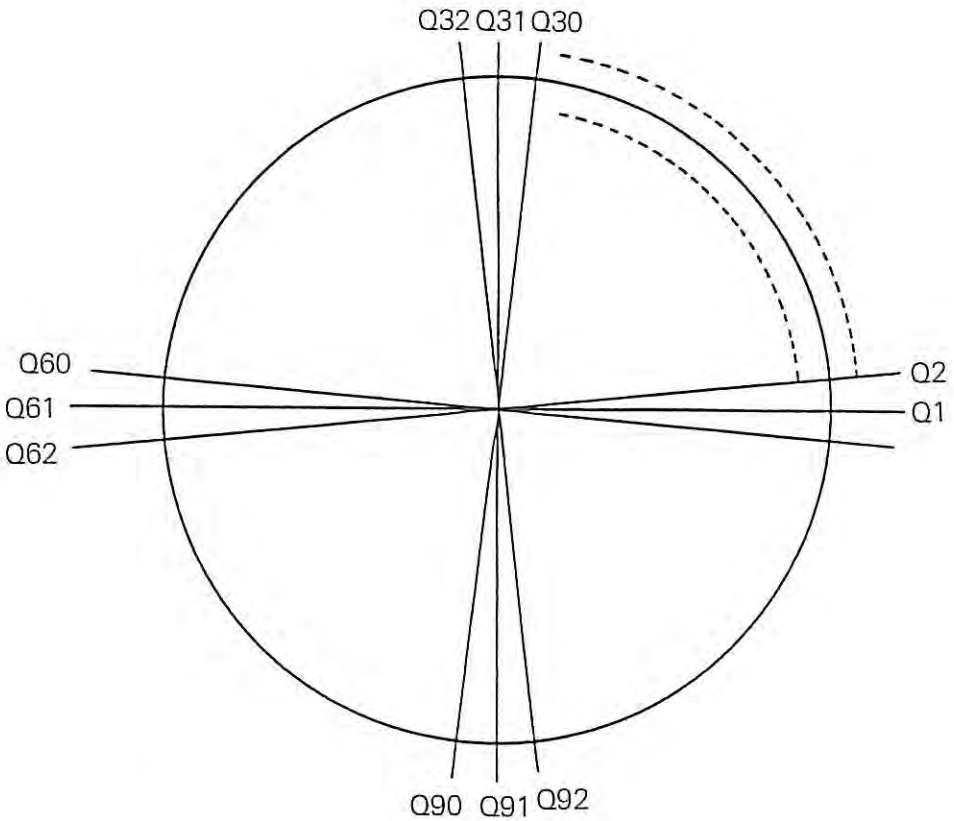
First set the compensation points by the pitch data, which establishes the 120 equally spaced points with the travel end point in the negative direction taken as the reference. Then, set the compensation data at each compensation point.



(2) C-axis

Pitch error compensation points are established at 120 points, each spaced by 3 degrees. Pitch error compensation data should be set for each of these points.

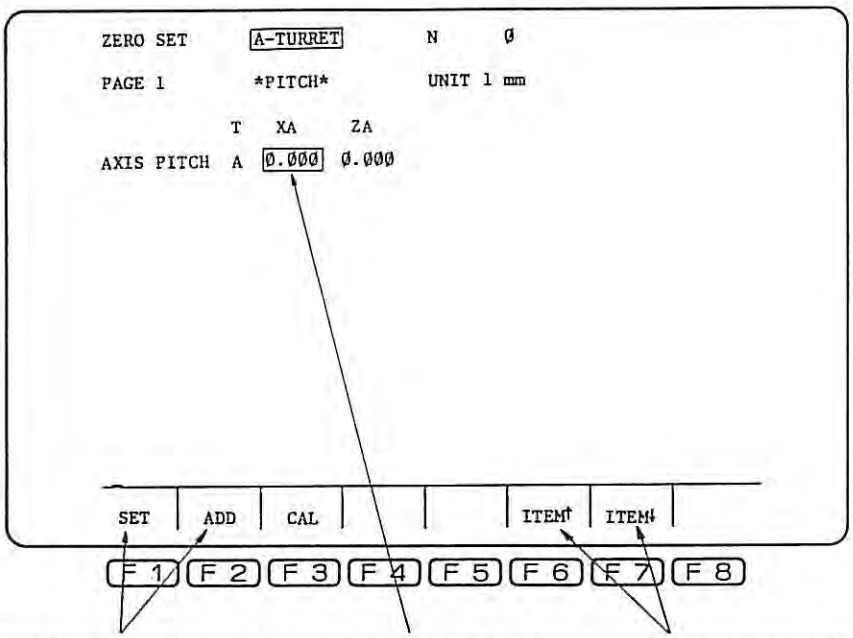
Set the compensation data for the Y-axis at the CT-axis (turret rotation axis) since the Y-axis is the coordinate axis of the CT-axis and the X(Z)-axis. (for LR15M-ATC-Y and LR45M-ATC-Y only)



Note: *For C-axis and CT-axis, pitch error compensation intervals are fixed at 3°.*

2-2. Data Setting Procedure

(1) Setting in ZERO SET Mode



Press function key [F1] or [F2], key in pitch data and press **WRITE**.

Locate the cursor on the axis for which the pitch data is to be set.

Press function key [F6] or [F7] to display the ***PITCH*** page.

Fig. 10-1

AXIS PITCH setting procedure:

a) Press the **ZERO SET** key and the function key [F7] (ITEM ↓) to display the first page (PAGE 1) of ZERO SET.

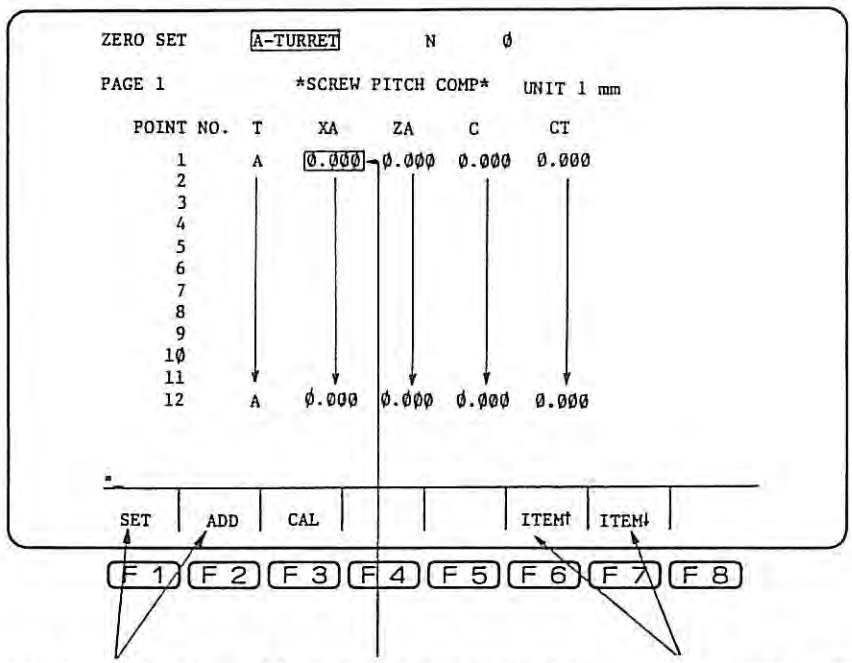
b) For two-saddle models, select the turret by pressing the **A** or **B** key on the operation panel.

c) Move the cursor at the axis for which pitch data is set.

d) Press the function key [F1] (SET) and key in the pitch data.

e) Press the **WRITE** key.

Turret selection is possible by pressing **A** and **B** keys on the operation panel.



Press function key [F1] or [F2], key in compensation data and press **WRITE**.

Move the cursor at the data location where compensation data is to be set.

Press [F6] or [F7] to display the ***SCREW PITCH COMP*** page.

* CT-axis is displayed only for the Y-axis specification.

Fig. 10-2

SCREW PITCH COMP data setting procedure:

a) Press the **ZERO SET** key and the function key [F7] (ITEM ↓) to display the SCREW PITCH COMP page.

b) For two-saddle models, select the turret by pressing the **A** or **B** key on the operation panel.

c) Move the cursor at the data setting location.

d) Press the function key [F1] (SET) and key in the compensation data.

e) Press the **WRITE** key.

f) One SCREW PITCH COMP page displays a total of 12 sets of compensation data. To display the desired point number, use the page keys and Pages are from PAGE 1 to PAGE 10. (Points No. 1 to No. 120)

Turret selection is possible by pressing the **A** and **B** keys on the operation panel. However, compensation data of C-axis is set only at the A-turret side.

(2) Data Setting using Program

Pitch and compensation data can be set using system variables.

VPCHX : X-axis pitch data
VPCHZ : Z-axis pitch data
VPFVX[*] : X-axis compensation data
VPFVZ[*] : Z-axis compensation data
VPFVC[*] : C-axis compensation data
VPFVT[*] : Y-axis compensation data (only for LR15M-ATC-Y and LR45M-ATC-Y)

* point number (1 through 120)

Program examples:

a) One saddle model

```
VPFVX[1] = 0.001
VPFVX[2] = 0.002
:
:
VPFVX[120] = 0.002
VPFVZ[1] = 0.001
VPFVZ[2] = 0.002
:
:
VPFVZ[120] = 0.002
VPFVC[1] = 0.001
VPFVC[2] = 0.002
:
:
VPFVC[120] = 0.002
VPCHX = 20.0
VPCHZ = 10.0
M02
```

X-axis compensation data

Z-axis compensation data

C-axis compensation data

X-axis pitch data

Z-axis pitch data

b) Two-saddle model (two-turret model)

```
G13
VPFVX[1] = 0.001
VPFVX[2] = 0.002
    :
    :
VPFVX[120] = 0.002
VPFVZ[1] = 0.001
VPFVZ[2] = 0.002
    :
    :
VPFVZ[120] = 0.002
VPFVC[1] = 0.001
VPFVC[2] = 0.002
    :
    :
VPFVC[120] = 0.002
VPCHX[1] = 20.0
VPCHZ[2] = 10.0

G14

VPFVX[1] = 0.001
VPFVX[2] = 0.002
    :
    :
VPFVX[120] = 0.002
VPFVZ[1] = 0.001
VPFVZ[2] = 0.002
    :
    :
VPFVZ[120] = 0.002
VPCHX = 20.0
VPCHZ = 10.0
M02
```

A turret selection

XA-axis compensation data

ZA-axis compensation data

C-axis compensation data
(The data can be designated only at A-turret side.)

XA-axis pitch data
ZA-axis pitch data

B turret selection

XB-axis compensation data

ZB-axis compensation data

XB-axis pitch data
ZB-axis pitch data

Select and read the program made in the format indicated above in the automatic mode. This sets the pitch error compensation data.

2-3. Data Setting Range

Setting and display ranges of the pitch, point and compensation data are indicated below:

(1) X-/Z-axis Pitch

0 mm
2.000 mm (0.0787 inch) $\leqq$ VPCHX, VPCHZ $\leqq$ 65.000 mm (2.5590 inch)

(2) Point Data

$1 \leqq$ point number $\leqq 120$

(3) Screw Pitch Error Compensation Data

X-/Z-axis : -0.999 mm $\leqq$ VPFVX, VPFVZ $\leqq +0.999$ mm
(-0.0393 inch) ($+0.0393$ inch)

C-axis : $-0.999° \leqq$ VPFVC $\leqq +0.999°$

CT-axis : $-0.999° \leqq$ VPFVT $\leqq +0.999°$

2-4. Compensation Data Backup Function

The pitch compensation data is lost when the power is turned off if it is simply set. Therefore, the pitch compensation data must be stored in the system bubble memory (BB0:) using the backup function. The procedure to backup the set data is explained below:

(1) Backup of Compensation Data

a) After selecting the program operation mode, press the function key [F8] (EXTEND) to display the function name PCDW for the function key [F6].

b) Pressing the function key [F6] displays the prompt =PCDW_ on the console line. Press the **WRITE** key.

c) The prompt "**compensation data backup OK (Y/N)!**" appears on the console line.

d) Pressing the "Y" key and **WRITE** keys starts searches of the data file (stored in the system bubble memory before shipment) from the system bubble memory and stores the compensation data of all axes. If the data file is not found, the processing terminates with the error message "**2596 not found pitch comp. data fail**" displayed.

e) Pressing the "N" key instead of the "Y" key in step d) above ends the backup function without backup operation.

(2) Confirmation of Compensation Data File

The compensation data file registers the machine model name and machine number to indicate that the data file is specifically prepared for the machine. The identification data (machine model name and machine number) can be referenced by the following operations.

a) After selecting the program operation mode, press the function key [F8] (EXTEND) to display the function name PCDW for the function key [F6].

b) Press the "N" key and then the **WRITE** key following the prompt = PCDW_;. The data file in the system bubble memory is searched for and the machine model name and the machine number in the data file are displayed on the console line in the following format:

```
=PCDW_; N
MACHINE NAME=xxxx
MACHINE NO.=xxxx
```

If the data file is not found in the system bubble memory, error message "**2596 not found pitch comp. data fail**" is displayed.

(3) Remarks

a) Compensation processing is not executed if pitch data "0" is set for both X and Z axes.

b) When "1" is set at bit 0 of optional parameter (bit) data No. 5, no compensation is carried out for all axes. It is recommended to set this bit data at "1" for measuring the compensation data. The change of parameter data requires the power to be turned off once three minutes after setting the new data.

c) For X and Z axes, pitch error compensation is not effective for points outside the 120 pitch compensation points.

d) With two-saddle models, C-axis compensation data can be set only from A-turret.

e) With the mirror image specification, pitch interval data and pitch compensation data become effective instantaneously when they have been set or changed. If a tool is present between the pitch compensation effective area when the data is set or changed, the tool will move in accordance with the set or changed data. If the difference between the new and old data is too much, an alarm (102 ALARM-A APA velocity) might be generated. Therefore, never change or set the compensation pitch point and compensation data during the progress of cutting.

For the models other than those with the mirror image specification, the data cannot be renewed unless the power is turned off after setting and storing the compensation data in the system bubble memory in accordance with the operations indicated above.

Note that data backup operation is required for the mirror image specification.

f) For models with the Y-axis specification, the pitch error in the Y-axis direction is compensated using the CT-axis (turret rotation axis). Since Y-axis pitch is indicated by distance while CT-axis by angle, it is necessary to convert the distance into an angle to obtain the compensation value.

g) Once the compensation data has been stored in the system bubble memory, it is not lost even when power is turned off or the control is reset.

h) When the control software program is loaded, the compensation data is all cleared to the initial setting (0). This, therefore, requires the compensation data to be reset and stored again.

2-5. Alarm Messages

ALARM-P

73 Pitch comp. data file read

For the ball screw pitch error compensation specification or inductosyn compensation specification, the pitch error compensation data file (LAT 06---POL) is not found when power is turned on. Or the size of the data file is different from the specification.

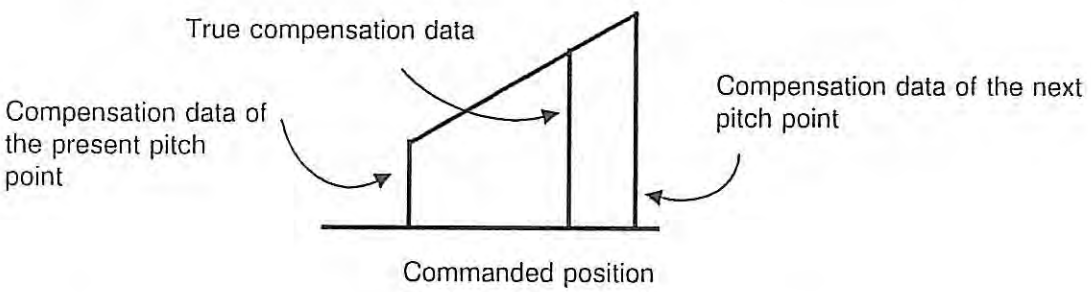
Index	:	None	
Character-string	:	None	
Code	:	FFFFFFFF	Compensation file not found
		Other than FFFFFFFF	Wrong compensation data file size (in units of sectors)

3. Compensation Method

The pitch error compensation is active for the detected values.

Detected value after compensation = Detected value + Compensation value

When the commanded point falls between the two pitch error compensation points, the true compensation data is calculated from the compensation data set for the present and the next pitch points using the linear interpolation method. The calculated values smaller than 0.001 mm are rounded off due to position detection unit resolution.



SECTION 11 NC OPERATION MONITOR

1. Overview

The NC operation monitor has two functions: the NC work counter and the NC hour meter.

With the NC work counter, the function has a total of four NC work counters and individual counters are displayed on the CRT. These counters can be used independently for different purposes.

With the NC hour meter, the function accumulates NC power on time, NC operation time, spindle rotating time, cutting time and external input time and the accumulated data is displayed on the CRT.

This section covers the following items:

(1) Contents of display and count-up conditions for NC work counter

(2) Alarm conditions of NC work counter

(3) Contents of display and count-up conditions for NC hour meter

(4) Alarm conditions of NC hour meter

2. Contents of Display and Count-up Conditions of NC Work Counter

(1) Contents of Display

In the PARAMETER SET mode, press the function key [F7] (ITEM ↓) to display the NC WORK COUNTER page. When the NC HOUR METER page is displayed, change the display by pressing **PAGE** key. See page 122.

(2) Count-up Conditions

The work counter A (or B, C, or D) counts the M02 and M30 commands, in other words, it counts the number of machined workpieces.

The count-up conditions of these counters are identical and counting does not occur during the operation in the dry run or machine lock mode.

The four work counters can be used for different purposes. Work counter A for counting the lot numbers and work counter B for counting the number of machined workpieces, for instance.

The maximum data set table is 99999999 both for COUNT and SET.

Count values and setting values of the work counters A through D can be set or read by the following system variables.

VWKCC[1]	Count value of work counter A
VWKCC[2]	Count value of work counter B
VWKCC[3]	Count value of work counter C
VWKCC[4]	Count value of work counter D
VWKCS[1]	Setting value of work counter A
VWKCS[2]	Setting value of work counter B
VWKCS[3]	Setting value of work counter C
VWKCS[4]	Setting value of work counter D

Example:

When cutting two workpieces with the same main program: (using work counter A)

```
N001
 (
 )          } Cutting of the first workpiece

N100 VWKCC[1]=VWKCC[1]+1

N020
 (
 )          } Cutting of the second workpiece

N200 VWKCC[1]=VWKCC[1]+1

M02
```

3. Alarm Conditions for NC Work Counter

When any of the counter data (COUNT) reaches the set value, an alarm occurs and operation of the control is inhibited.

ALARM-C

951 Work counter over X

Index	:	None
Character-string	:	None
Code	:	X= 1 ..... COUNT data of work counter A reaches the SET data
		2 ..... COUNT data of work counter B reaches the SET data
		3 ..... COUNT data of work counter C reaches the SET data
		4 ..... COUNT data of work counter D reaches the SET data

To cancel the alarm state,

(1) reduce the COUNT data to any value smaller than the SET data,

(2) increase the SET data to any value greater than the COUNT data,

or

(3) clear the SET data to "0".

After carrying out any of the operations indicated above, press the **RESET** pushbutton switch on the NC operation panel. Simple NC reset operation cannot cancel the alarm state.

Note: If the SET data is set to "0", alarm check is not carried out.

4. Contents of NC Hour Meter Display and Count-up Conditions

(1) Contents of Display

The NC WORK COUNTER page and the NC HOUR METER page are switched by pressing the or keys. (See page 122.)

(2) Count-up Conditions

On the NC HOUR METER page, the data is indicated in units of "hour: minute: second".

a) POWER ON TIME

This indicates the accumulated power on time of the control.

b) NC RUNNING TIME

This indicates the accumulated time in which the control has been running. With a two-saddle model, counting occurs when either of the two turrets is operating. Note that counting will not occur during the machine lock or dry run mode operation.

c) SPINDLE REVOLUTION

This indicates the accumulated spindle revolution time. In the machine lock mode operation, counting will not occur since the spindle is not rotating in this operation mode.

d) CUTTING TIME

This indicates the accumulated time in which axis has been fed in a cutting feedrate. With a two-saddle model, counting occurs when either of the two turrets is fed at a cutting feedrate. Note that counting will not occur during the machine lock or dry run mode operation.

e) EXTERNAL INPUT TIME

This indicates the accumulated time in which the external input signal is on.

The maximum values for both COUNT and SET are 99999:59:59. When the COUNT data exceeds 99999:59:59, the display jumps to 99999:99:99.

5. Alarm Conditions of NC Hour Meter

When any of the counter data (COUNT) reaches the set value, an alarm occurs and operation of the control is inhibited.

ALARM-C

952 Running hour over

Index	:	None
Character-string	:	None
Code	:	X= 1 ..... COUNT data of POWER ON TIME reaches the SET data.
		2 ..... COUNT data of NC RUNNING TIME reaches the SET data.
		3 ..... COUNT data of SPINDLE REVOLUTION TIME reaches the SET data.
		4 ..... COUNT data of CUTTING TIME reaches the SET data.
		5 ..... COUNT data of EXTERNAL INPUT TIME reaches the SET data.

To cancel the alarm state,

(1) reduce the COUNT data to any value smaller than the SET data,

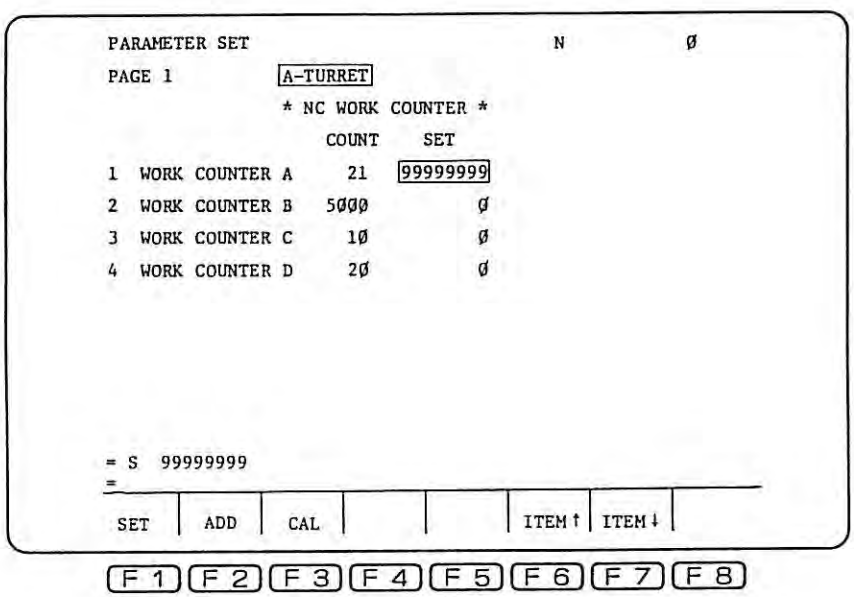
(2) increase the SET data to any value greater than the COUNT data,

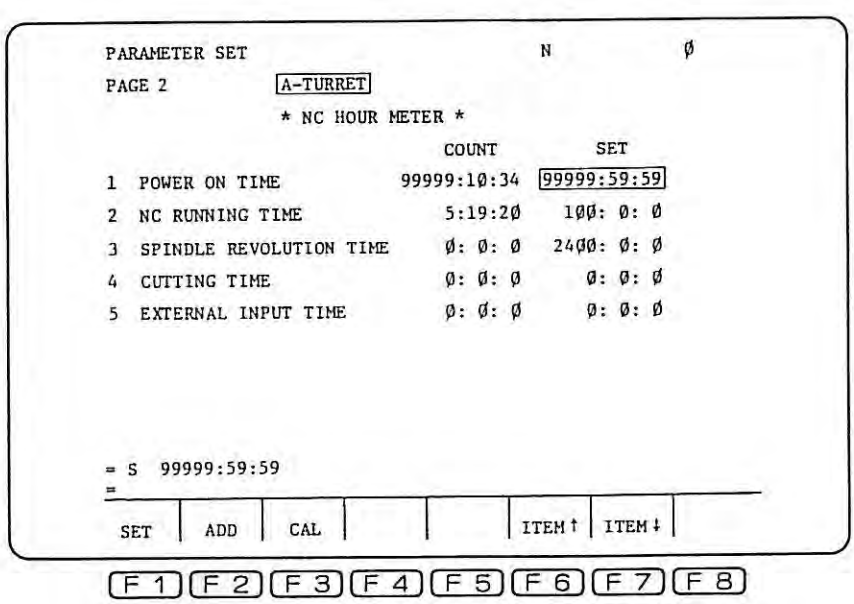
or

(3) clear the SET data to "0:0:0".

After carrying out any of the operations indicated above, press the **RESET** pushbutton switch on the NC operation panel. Simple NC reset operation cannot cancel the alarm state.

Note: If the SET data is set to "0:0:0", alarm check is not carried out.





6. NC Operating Time Printout

The NC operating time can be printed out by connecting a printer to the NC.

(1) Parameter Setting (in common with gauging data printout)

- Output device designation : Parameter (word) No. 45

  0 . . . . CN0: main card 3 or 13
  1 . . . . CN1: } RS board
  2 . . . . CN2: }
  3 . . . . CN3: }
  4 . . . . CN4: }
  5 . . . . Centronics interface - data board

- Data printout : Parameter (bit) No. 5, bit 3

  0 . . . . Not printed out
  1 . . . . Printed out

- Data printout file creation : Parameter (bit) No. 5, bit 6

  0 . . . . Printed out by the device designated by the parameter (word) No. 45
  1 . . . . A file is created in the bubble memory.

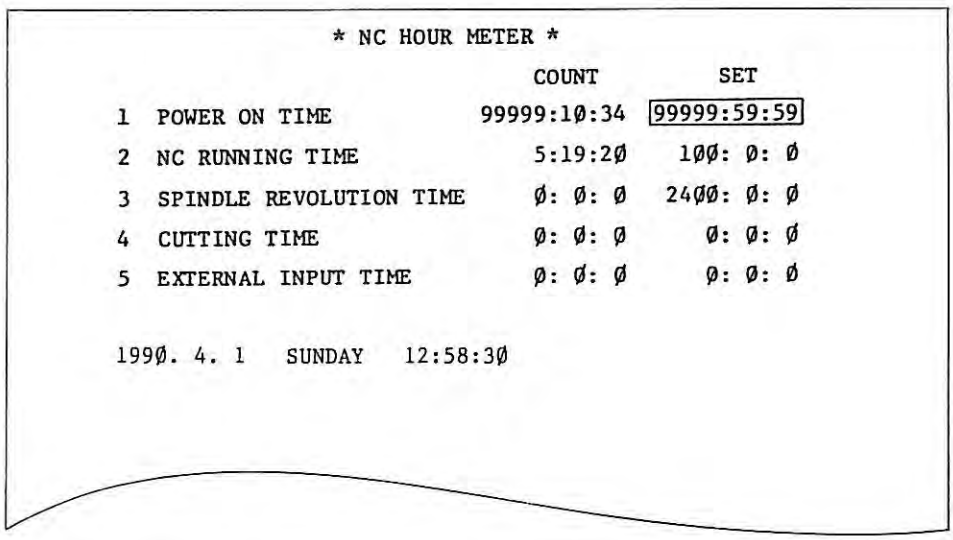
(2) Operation

1) Select the MDI mode.
2) Press the function key [F1] (DATA INPUT).
3) Key in as follows and press the **WRITE** key:

   [P][R][N][T][SP][2][0]

   The NC operating time is printed out in the format shown in (3).
4) To print out the NC operating time during automatic machine operation, designate the command in step 3) in the program. Each time the command is executed, the data is printed out.

(3) Format



```
                * NC HOUR METER *
                                        COUNT          SET
1  POWER ON TIME                   99999:10:34   99999:59:59
2  NC RUNNING TIME                     5:19:20     100: 0: 0
3  SPINDLE REVOLUTION TIME             0: 0: 0    2400: 0: 0
4  CUTTING TIME                        0: 0: 0       0: 0: 0
5  EXTERNAL INPUT TIME                 0: 0: 0       0: 0: 0

1990. 4. 1   SUNDAY   12:58:30
```

CAUTION

The date of printout cannot be displayed correctly unless the date and time are set in the program operation mode.

SECTION 12 TAPE DATA INPUT/OUTPUT FUNCTION (Standard for OSP5020L Only)

1. Overview

In the PROG OPERATION mode, the D-PIP (data transmission) function permits the transmission of the tool data and parameters.

Data Transmit Function:

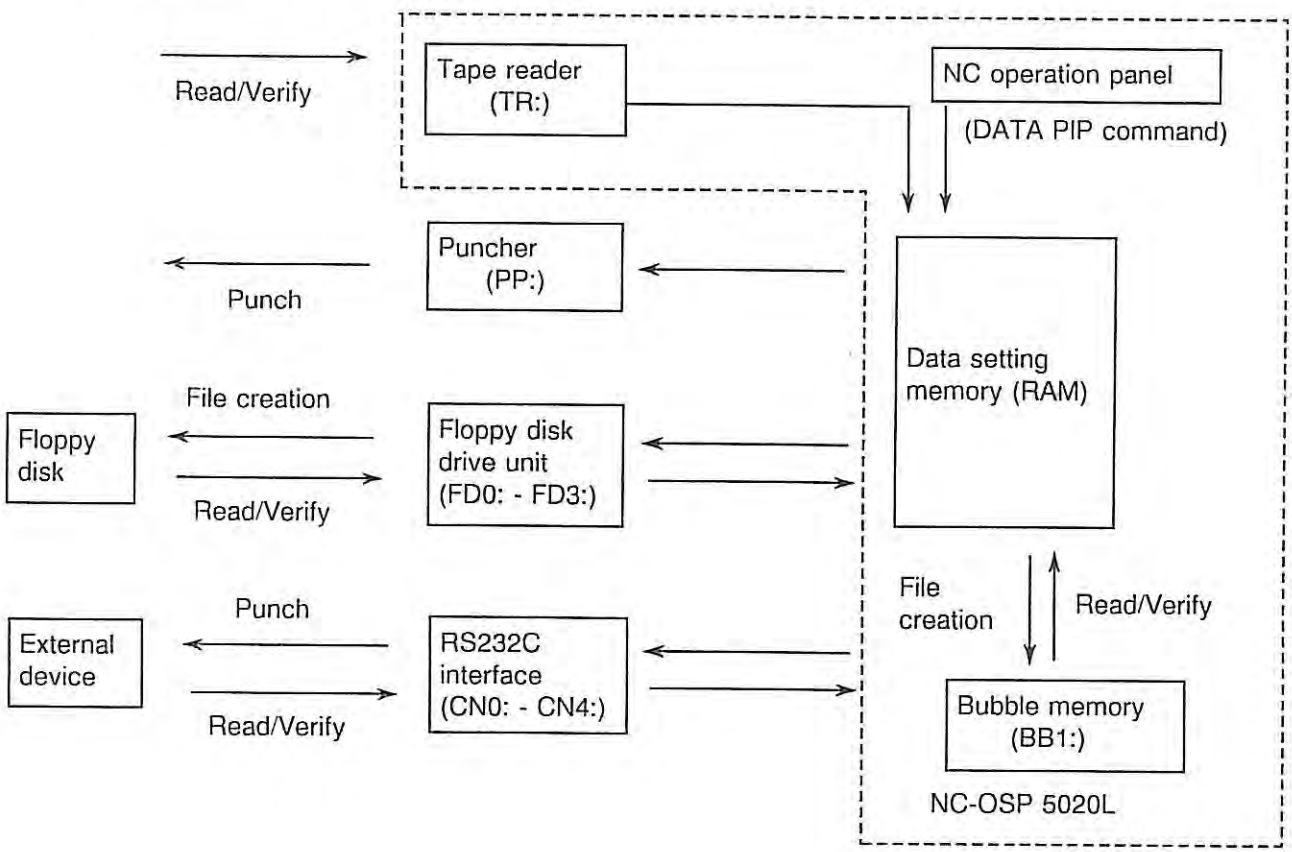
Data Input (READ) . . . . . The setting data tape, or the data file is read and stored in the NC.

Data Output (PUNCH) . . . . . Data stored in the NC is punched out on a tape. Also used to create a bubble memory or floppy disk data file.

Data Verify (VERIFY) . . . . . The data content of the tape, bubble memory, or floppy disk data file is checked against the NC memory data.

Transfer Complete (QUIT) . . . . . End of data transmit function

Data Transmit Function Layout:



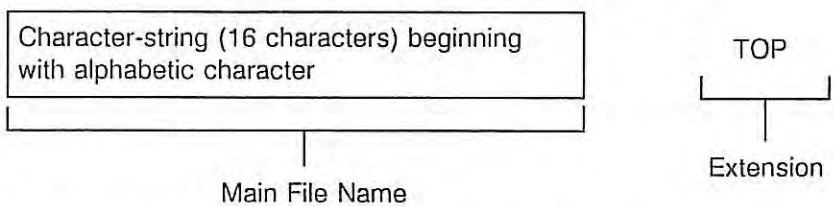
2. Function Application

The primary purpose of the data transmit function is to permit the tool data, zero point, and parameter setting data in the NC memory to be punched and stored on paper tape.

3. Input/Output Tape Format of Setting Data

3-1. File Names

File names that can be processed by the D-PIP function are as follows:



When the file name is omitted, the system adopts the file name A.TOP.

3-2. Tape Format Table

Setting Mode	Data Type	Data ID	Data No. (N)	Description	Remarks
TOOL DATA SET	Tool offset	T1	1 - 32 (96)	X (axis), Z (axis)	
	Nose R compensation	T2	1 - 32 (96)	X (axis), Z (axis), Q (nose radius pattern)	
	Tool interference	T3	1 - 12 (96)	A (pattern), B (minus), C (Z plus), D (X minus), E (X plus)	Effective only for 2-saddle specification, with no graphics
	Tool life control tool table	T4	1 - 12 (96)	G (group number), A (set workpiece quantity), B (actual machined quantity), C (set machining time), D (actual machining time), E (set wear amount), F (actual wear amount), I (offset No. 1), J (offset No. 2), K (offset No. 3), H (tool NG flag), L (tool life flag)	
	Tool life control group table	T5	1 - 12 (24)	S (selected tool No.), L (tool group life flag)	
	External mes auto compensation data	T6	1 - 8	I (input data No.), C (compensation amount), Q (tool offset No.), G (tool group No.), F (tool offset group), A (axis), S (skip counter), D (compensation ignore counter)	
	CEJ matic compensation data	T7	1 - 12	Q (tool offset No.), F (tool offset group No.)	

In the Data No. column, the numeric value in ( ) applies for the ATC specification.

Setting Mode	Data Type	Data ID	Data No. (N)	Description	Remarks
TOOL DATA SET	Tool form select	T10	1 - 12 (96)	A (tool angle), B (edge angle), C (sticking out), D (drill diameter), E (tool width), I (offset No. 1), J (offset No. 2), K (offset No. 3), S (classification code), L (form code)	Effective with graphic specification
	ATC tool form	T11	1 - 96	I (1st position tool's offset No. 1), J (1st position tool's offset No. 2), K (1st position tool's offset No. 3), A (2nd position tool's offset No. 1), B (2nd position tool's offset No. 2), C (2ndposition tool's offset No. 3), S (2nd position tool's tool classification code), L (2nd position tool's tool form code)	
	Load monitor base	T12	1 - 64	X (axis), Z (axis), C (axis), S (spindle), M (M-tool spindle), W (axis), B (sub spindle)	M can be designated for the B-turret.
	Load monitor first limit	T13	1 - 64	X (axis), Z (axis), C (axis), S (spindle), M (M-tool spindle), W (axis), B (sub spindle)	M can be designated for the B-turret.
	Load monitor second limit	T14	1 - 64	X (axis), Z (axis), C (axis), S (spindle), M (M-tool spindle), W (axis), B (sub spindle)	M can be designated for the B-turret.
	Post-process gauging, RS232C method	T15	1 - 12	I (gauging point), C (turret), Q (axis), G (G flag), F (group No./ offset No.), A (offset group No.), S (Reference value)	

In the Data No. column, the numeric value in ( ) applies for the ATC specification.

Setting Mode	Data Type	Data ID	Data No. (N)	Description	Remarks
ZERO SET	Zero point	O1	1 - 2	X (axis), Z (axis), W (axis), C (axis)	N1: ZERO OFFSET, N2: ZERO SHIFT
	Pitch	O2	1	X (axis), Z (axis)	
	Screw pitch compensation	O3	1 - 120	X (axis), Z (axis), C (axis)	
	2nd spindle zero point	O4	1 - 2	X (axis), Z (axis), C (axis)	N1: ZERO OFFSET, N2: ZERO SHIFT
PARAMETER SET	User parameter	P1	1 - 4, 9	X (axis), Z (axis), W (axis), C (axis)	N1: + VARI. LIMIT P N2: – VARI. LIMIT P N3: + VARI. LIMIT M N4: – VARI. LIMIT M N9: DROOP DAT
	Common variable	P2	1 - 32	R (data)	
	System parameter	P3	1 - 10	X (axis), Z (axis), W (axis), C (axis)	N1: + STROKE END LIMIT N2: – STROKE END LIMIT N3: BACKLASH N4: PR CONNECT CMP N5: + SENSOR POSITION (PROG) N6: – SENSOR POSITION (PROG) N7: + SENSOR POSITION (MACH) N8: – SENSOR POSITION (MACH) N9: STANDARD RING

Setting Mode	Data Type	Data ID	Data No. (N)	Description	Remarks
PARAMETER SET	System parameter	P3	1 - 10	X (axis), Z (axis), W (axis), C (axis)	N10: ATC TOOL CHANGE POS. N11: Position encoder offset N12: Rapid feedrate unit amount N13: Rapid feedrate acceleration/ deceleration unit amount N14: Manual feedrate unit amount N15: Manual feedrate acceleration/d eceleration unit amount
	Turret position compensation	P4	1 - 12 (20)	X (axis), Z (axis)	
	Optional parameter long word	P5	1 - 96	R (data)	
	Optional parameter word	P6	1 - 96	R (data)	
	Optional parameter bit	P7	1 - 48	R (data)	
	Spindle orientation parameter	P8	1 - 32	R (data)	
	Measure counter	P9	1 - 32	R (data)	
	NC work counter	P10	1 - 4	Q (actual), R (full count)	

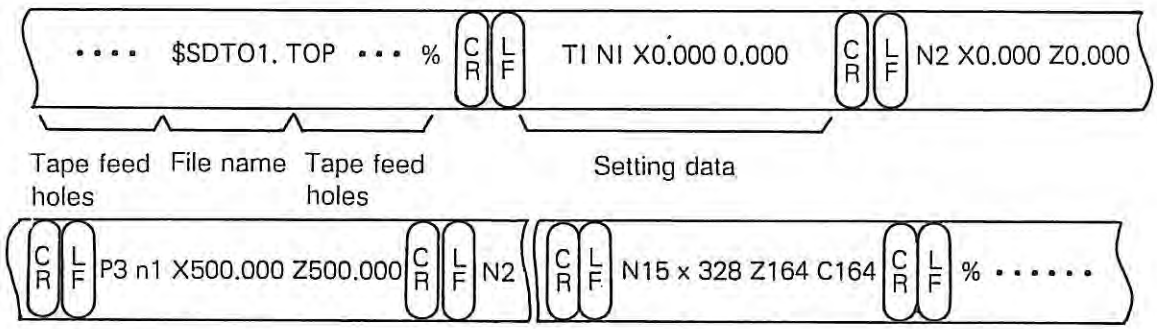
In the Data No. column, the numerical value in ( ) is effective for double-tooling turret specification.

Setting Mode	Data Type	Data ID	Data No. (N)	Description	Remarks
PARAMETER SET	NC hour meter	P11	1 - 5	Q (actual), R (full count)	N1: POWER ON TIME N2: NC RUNNING TIME N3: SPINDLE REVOLUTION TIME N4: CUTTING TIME N5: EXTERNAL INPUT TIME
	Tool interference parameter	P12	1 - 80	A (pattern), B (Z minus), C (Z plus), D (X minus), E (X plus)	Effective with graphic specification and 2-saddle specification
	Chuck/ tailstock barrier	P13		A (jaw dimension L1), B (jaw dimension D1), C (jaw position CX), D (jaw position CZ), E (center dimension L2), F (center dimension D2), I (center position D3), J (workpiece end-face WR)	
	Home position	P14	1 - 8	X (axis), Z (axis)	
	2nd spindle chuck barrier	P15		A (jaw dimension L1), B (jaw dimension D1), C (jaw position CX), D (jaw position CZ), E (graphic zero offset ZOF)	
	2nd spindle user parameter	P16	1 - 4, 9	X (axis), Z (axis), C (axis)	N1: + VARI. LIMIT P N2: – VARI. LIMIT P N3: + VARI. LIMIT M N4: – VARI. LIMIT M N9: DROOP DAT

Setting Mode	Data Type	Data ID	Data No. (N)	Description	Remarks
PARAMETER SET	2nd spindle system parameter	P17		X (axis), Z (axis)	N1: + STROKE END LIMIT N2: – STROKE END LIMIT N3: BACKLASH N4: PR CONNECT CMP N5: + SENSOR POSITION (PROG) N6: – SENSOR POSITION (PROG) N7: + SENSOR POSITION (MACH) N8: – SENSOR POSITION (MACH)

Note: The data in the above table is for all system specifications. Therefore, all data cannot always be handled by the tape data input/output function.

3-3. Actual Tape Format




```
T1N1X0.000Z0.000
N2X0.000Z0.000
N3X0.000Z0.000
  ⋮       ⋮

T2N1X0.000Z0.000Q0
N2X0.000Z0.000Q0
N3X0.000Z0.000Q0
  ⋮       ⋮

T4N1G0A0B0C0:0D0:0E0.000F0.000I0J0K0H0L0
N2G0A0B0C0:0D0:0E0.000F0.000I0J0K0H0L0
N3G0A0B0C0:0D0:0E0.000F0.000I0J0K0H0L0
  ⋮       ⋮

01N1X12563.256Z8965.235C0.000
N2X0.000Z0.000C0.000
  ⋮       ⋮

P3N1X13063.256Z9465.235
N2X12463.256Z8865.235
N3X0.010Z0.010C0.100
  ⋮       ⋮

P2N1R0
N2R0
N3R0
  ⋮       ⋮

P1N1X500.000Z500.000
N2X-100.000Z-100.000
N3X13063.256Z9465.235
  ⋮       ⋮

P4N1X0.000Z0.000
N2X0.000Z0.000
N3X0.000Z0.000
  ⋮       ⋮
```

Tool offset

Nose radius compensation

Tool life management data table

Zero point

User parameters

Common variables

User parameters

Turret position compensation

[Supplement]

On 2-saddle models, the A/B turrets are distinguished by the following G codes:

G13 . . . . Turret A
G14 . . . . Turret B

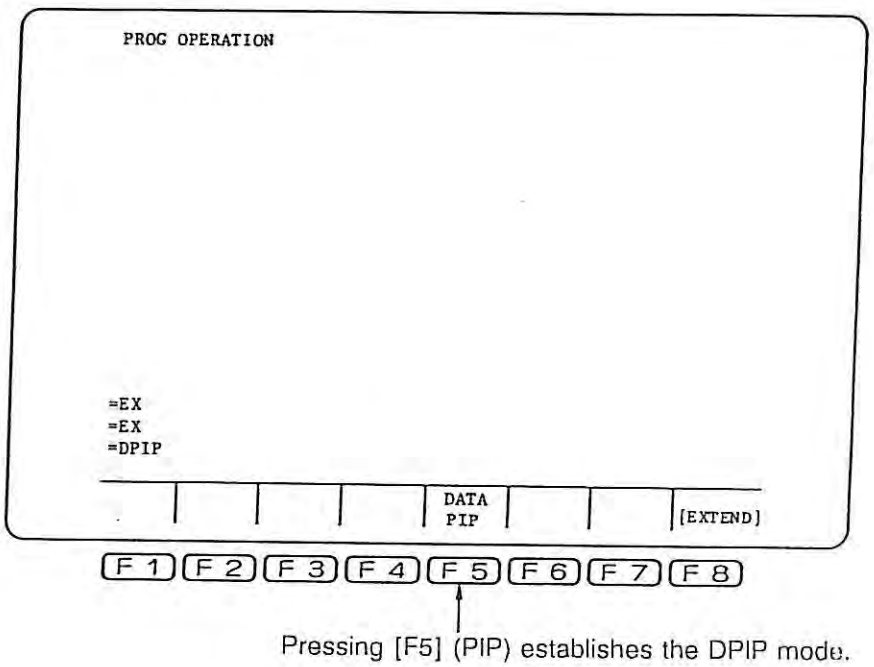
If this G code is not designated, the system will select the G13 code.

```
G13
T1 N1 X0.000  Z0.000 CR      ┐
          :                  │ Turret A data
          :                  ┘
G14
T1 N1 X0.000  Z0.000 CR      ┐
          :                  │ Turret B data
          :                  ┘
```

4. Operation Procedures

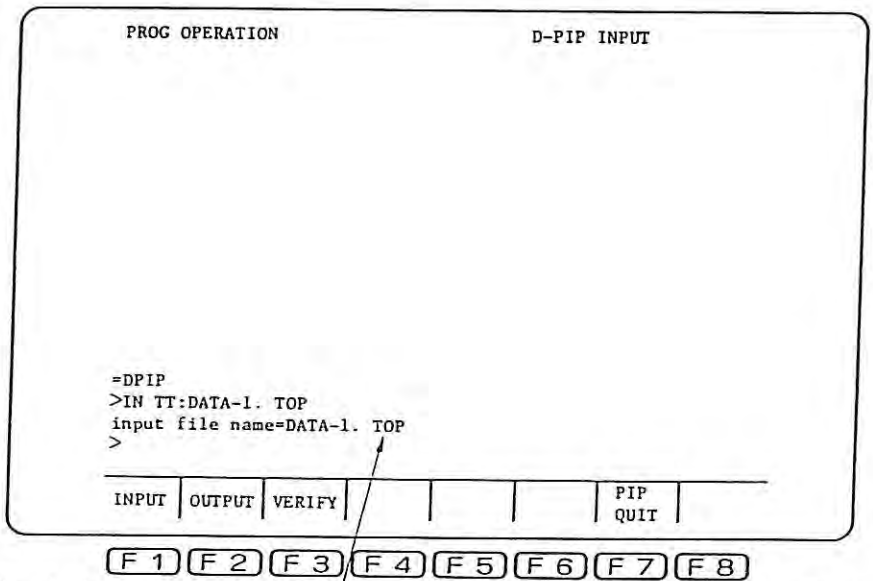
1) Select the PROG. OPERATION mode.

2) Press the function key [F8] (EXTEND). The function DATA PIP will appear above [F5] as shown below.



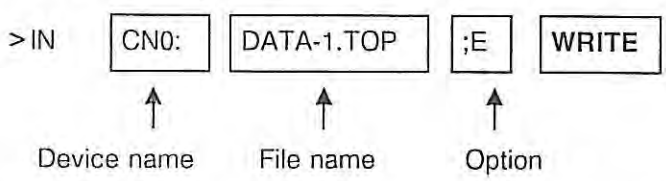
3) Press the function key [F5] (DATA PIP).

4-1. Data Input (READ)



If the device name is BB1: or FD0: through FD3:, device name is not displayed.

(1) Press the function key [F1] (READ), and use the panel to designate the data input device, input file name, and option.



(2) If the device name is omitted, the system will select the tape input device designated by parameter (word) No. 54.

(3) The following device names can be designated:

TR:, CN0:, (or TT:), CN1:, CN2:, CN3:, CN4:, BB1:, FD0:, FD1:, FD2:, FD3:

TR: . . . . . Tape reader
Standard equipment

CN0: . . . . . RS232C interface channel 0
"TT:" can also be used.
Optionally requires main card 3 and external device.

CN1: - CN4: . . . . RS232C interface channel 1 - 4
Optionally requires RS board and external device.

BB1: . . . . . User bubble memory
Standard : 30 m
Optional : 60 m, 160 m, 320 m, 640 m, 1,280 m, 2,560 m, 3,840 m, 5,120 m (200 ft, 525 ft, 1,050 ft, 2,100 ft, 4,200 ft, 8,400 ft, 12,600 ft, 16,800 ft)

FD0: - FD3: . . . . Floppy disk drive unit channel 0 - 3
Optionally requires a floppy board and drive unit.

(4) If the file name is omitted when using device names BB1:, FD0: - FD3:, the system will adopt and input "A.TOP" as the file.

(5) For tape input devices (TR:, CN0: - CN4:), the file name on the tape will be displayed on the console line.

(6) When using tape input devices, if the designated file name does not match the tape file name, the **"2585 Tape file name not same ERROR"** message will be displayed on the console line, and data input will end.

(7) Option designations

;V ..... When reading is complete, the tape returns to the beginning and re-reads the data, comparing the tape or file contents with the data which was input. This option is not effective for the RS232C (with device names TT:, CN0: - CN4:).

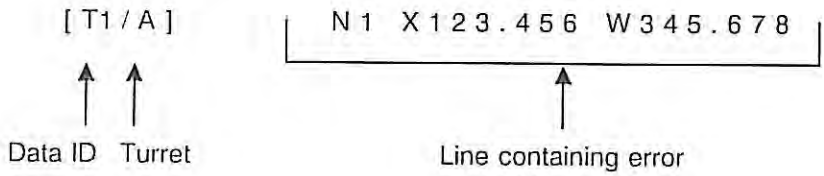
;E ..... Effective for tape, RS232C input
Indicates that the tape is using EIA code.

;I ..... Effective for tape, RS232C input
Indicates that the tape is using ISO code.

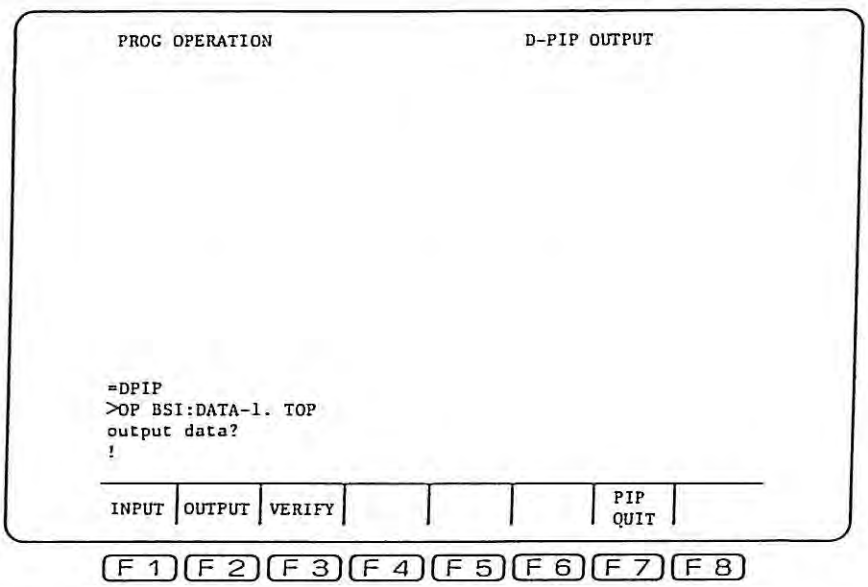
Note: *";E" and ";I" designation cannot be made for sector devices (BB1:, FD0: - FD3:).*

*When the ;V comparison is made, if the data matches, "**all same data**" will be displayed on the console line. If a data mismatch is found, "**Tape read verify ERROR**" will be displayed, and the VERIFY operation will end.*

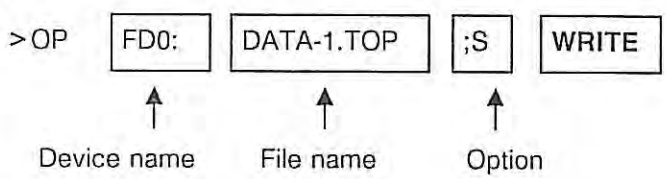
(8) If the data format is incorrect, or the tape contains illegal data, **"Tape file format ERROR"** will be displayed on CRT console line and the data line which contains the error will be displayed as follows:



4-2. Data Output (PUNCH)



(1) Press the function key [F2] (PUNCH), and use the panel to designate the data output device, output file name, and option.



(2) If the device name is omitted, the system will select the tape output device designated by parameter (word) No. 44.

(3) The following device names can be designated:

PP:, CN0:, (or TT:), CN1:, CN2:, CN3:, CN4:, BB1:, FD0:, FD1:, FD2:, FD3:

PP: . . . . . FACIT puncher
Optionally requires a main card 3 and puncher.

CN0: . . . . . RS232C interface channel 0
"TT:" can also be used.
Optionally requires main card 3 and external device.

CN1: - CN4: . . . . RS232C interface channel 1 - 4
Optionally requires RS board and external device.

BB1: . . . . . User bubble memory
Standard : 30 m
Optional : 60 m, 160 m, 320 m, 640 m, 1,280 m, 2,560 m, 3,840 m, 5,120 m (200 ft, 525 ft, 1,050 ft, 2,100 ft, 4,200 ft, 8,400 ft, 12,600 ft, 16,800 ft)

FD0: - FD3: . . . . Floppy disk drive unit channel 0 - 3
Optionally requires a floppy board and drive unit.

(4) If the output file name is omitted, the system will select "A.TOP" as the file name.

(5) When item (1) is executed, **"Is punch data?"** is displayed on the console line. When an exclamation mark "!" is displayed on the next line, the "waiting for input" mode is established.

(6) Designate output data by using keyboard.

a) ! **WRITE**

All data is output.
(Output varies according to system specification.)

b) !T **WRITE**

Tool data is output.

c) !T, O **WRITE**

Tool data and zero point data are output.

d) !T1, T2, O1, P1 **WRITE**

Tool offset data, nose radius compensation data, zero point data, and user parameter data are output.

Note 1: *If an incorrect command, or a command containing illegal data is given, "**2587 Data command ERROR**" will be displayed on the console line, and data output will end.*

Note 2: *If no data settings are made after loading the NC software, **OVERFLOW** will be displayed on the CRT. If the data output command is given when in this condition, the "**Output data over flow**" error will occur, and data output will be impossible. Therefore, be sure the data setting is made prior to data output.*

(7) Option designations

;A . . . . . Output occurs for the A turret side of the designated data. (For 2-saddle, 2-turret specification)

;B . . . . . Output occurs for the B turret side of the designated data. (For 2-saddle, 2-turret specification)

;S . . . . . Output occurs with a space.

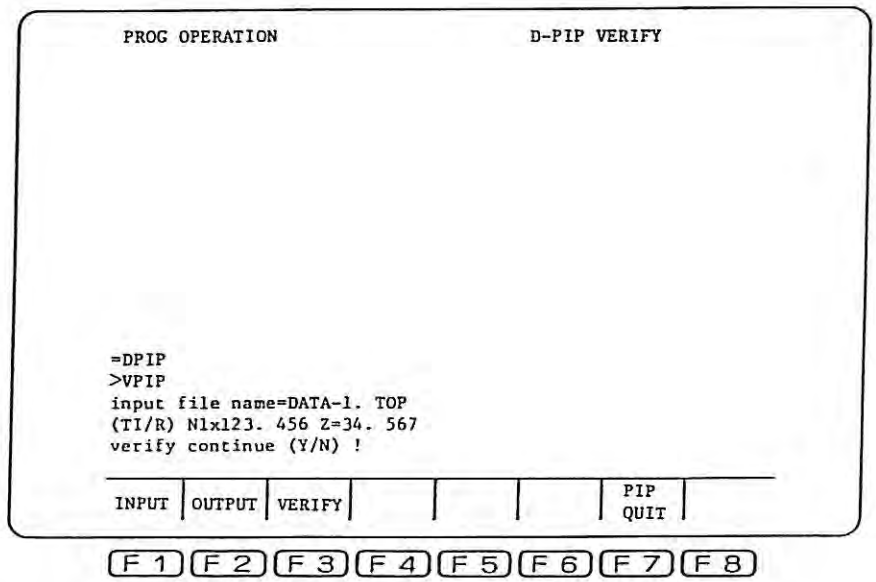
;E . . . . . Effective for tape, RS232C output

;I . . . . . Effective for tape, RS232C output
Indicates that tape output uses ISO code.

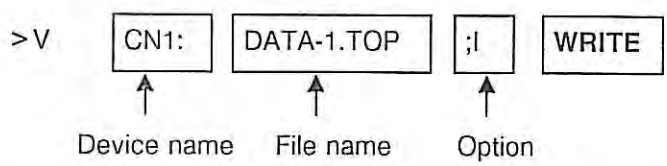
Note: *[;E] and [;I] designation cannot be made for sector devices (BB1:, FD0: - FD3:).*

4-3. Data Verify

Verification of the Data File Content and Data Stored in the NC



(1) Press the function key [F3] (VERIFY), and use the panel to designate the data input device, input file name, and option.



(2) If the device name is omitted, the system will select the tape input device designated by parameter (word) No. 54.

(3) The following device names can be designated:

TR:, CN0:, (or TT:), CN1:, CN2:, CN3:, CN4:, BB1:, FD0:, FD1:, FD2:, FD3:

TR: . . . . . Tape reader
Standard equipment

CN0: . . . . . RS232C interface channel 0
"TT:" can also be used.
Optionally requires main card 3 and external device.

CN1: - CN4: . . . . RS232C interface channel 1 - 4
Optionally requires RS board and external device.

BB1: . . . . . User bubble memory

Standard : 30 m
Optional : 60 m, 160 m, 320 m, 640 m, 1,280 m, 2,560 m, 3,840 m, 5,120 m (200 ft, 525 ft, 1,050 ft, 2,100 ft, 4,200 ft, 8,400 ft, 12,600 ft, 16,800 ft)

FD0: - FD3: . . . . Floppy disk drive unit channel 0 - 3
Optionally requires a floppy board and drive unit.

(4) If the file name is omitted when the device name is BB1:, FD0: - FD3:, the system searches for the file name "A.TOP", and the verify function occurs accordingly.

(5) When tape input devices (TR:, CN0: - CN4:) are used, the file name on the tape will be displayed on the console line.

(6) When tape input devices are used, if the designated file name does not match the file name on the tape, "**2585 Tape file name not same ERROR**" will be displayed on the console line, and the VERIFY function will end.

(7) If a data mismatch is found during the VERIFY function, the data line which contains the error is displayed on the console line, with the message "**verify continue? (Y/N)**". Key in "Y" and press **WRITE** key to continue the VERIFY function with the next data. Key in "N" and press **WRITE** key to end the VERIFY function. If no data mismatch is found, "**all same data**" will be displayed on the console line.

(8) Option designations

;E . . . . . Effective for tape, RS232C input
Indicates that the tape is using EIA code.

;I . . . . . Effective for tape, RS232C input
Indicates that the tape is using ISO code.

Note: [;E] and [;I] designation cannot be made for sector devices (BB1:, FD0: - FD3:).

4-4. Precautions on Tape Read-in, Punchout and Verifying Operations

There are two coding systems: EIA and ISO; selection of the coding system can be made by

(1) Parameter setting, or

(2) Designating the desired coding system each time, read-in, punch out or verifying operation is made.

(1) Parameter Setting

Two parameters (optional parameter (bit) No. 1, bit 0 and bit 1) 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 (bit 1)	Tape Code ISO Code (bit 0)	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 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 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.

This parameter setting is effective when neither option code "E;" (EIA) nor "I;" (ISO) is designated in READ, VERIFY and/or PUNCH operation.

(2) ISO or EIA Designation for Respective Program Operation - READ, VERIFY and PUNCH

By designating "E;" (EIA) or "I;" (ISO) each time 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.

5. Error Messages

2584 NC LOB memory load

Failure in attempt to read the load object file from the bubble memory in the program operation.

Character-string : File name attempted to read

Code :
- 1 ..... File to be loaded is not found.
- 2 ..... File attribute is not LOB1.
- 3 ..... Address for loading is wrong.

2585 Tape file name not same

In the data input/output function, the file name input or designated in the verify operation does not match the file name punched on tape.

Character-string : File name on tape

Code : None

2586 File format

In the data input/output function, the data format of the tape through the input or verify operation is incorrect.

Character-string : Data record (block) which caused an error

Code :
- 1 ..... File record (block) data does not end correctly.
- 2 ..... The first character of the file is not an alphabetic character, or the number of digits in the data is too large.
- 3 ..... The data does not have data ID (Tn, On, Pn) at the beginning.
- 4 ..... Data number Nn is not used at the beginning or the place following the data ID in the record data.
- 5 ..... Data which cannot be set is contained in the tape data.
- 6 ..... On the 2-saddle model, turret designation G code is wrong.
- 7 ..... Data ID Tn, On, or Pn is not correct.
- 8 ..... Data number Nn is not correct.

2587 Data command

In the data input/output function, data ID designation is wrong in the designation of the output data.

Character-string : Data ID which is designated

Code : 1 ..... Designation method is wrong.

2 ..... A character other than T, O or P is designated as data ID.

3 ..... Data ID number not contained in the data to be output is designated.

2588 Input data over flow

In the data input/output function, setting is impossible as the data input is too small or too large.

Character-string : Record in which error has occurred

Code : None

2589 Stroke end limit over

In the data input/output function, variable limit data input is outside the travel end limit range.

Character-string : Record in which error has occurred

Code : None

2590 Time data

In the data input/output function, time data input is not correct.

Character-string : None

Code : 1 ..... Hour:minute data is wrong.

2 ..... Hour:minute:second data is wrong.

2591 Data not same

In the data input/output function, verify error occurs during the input verify operation.

Character-string : Record in which error has occurred

Code : None

2592 Tool entry

In the data input/output function, S (selection tool number) of T5 (tool life management group information) has not been registered.

Character-string : Record in which error has occurred

Code : None

2593 Output data over flow

In the data input/output function, the data to be output is too large.

Character-string : None

Code : Data to be output (hexadecimal)

2594 Output data under flow

In the data input/output function, the data to be output is too small.

Character-string : None

Code : None

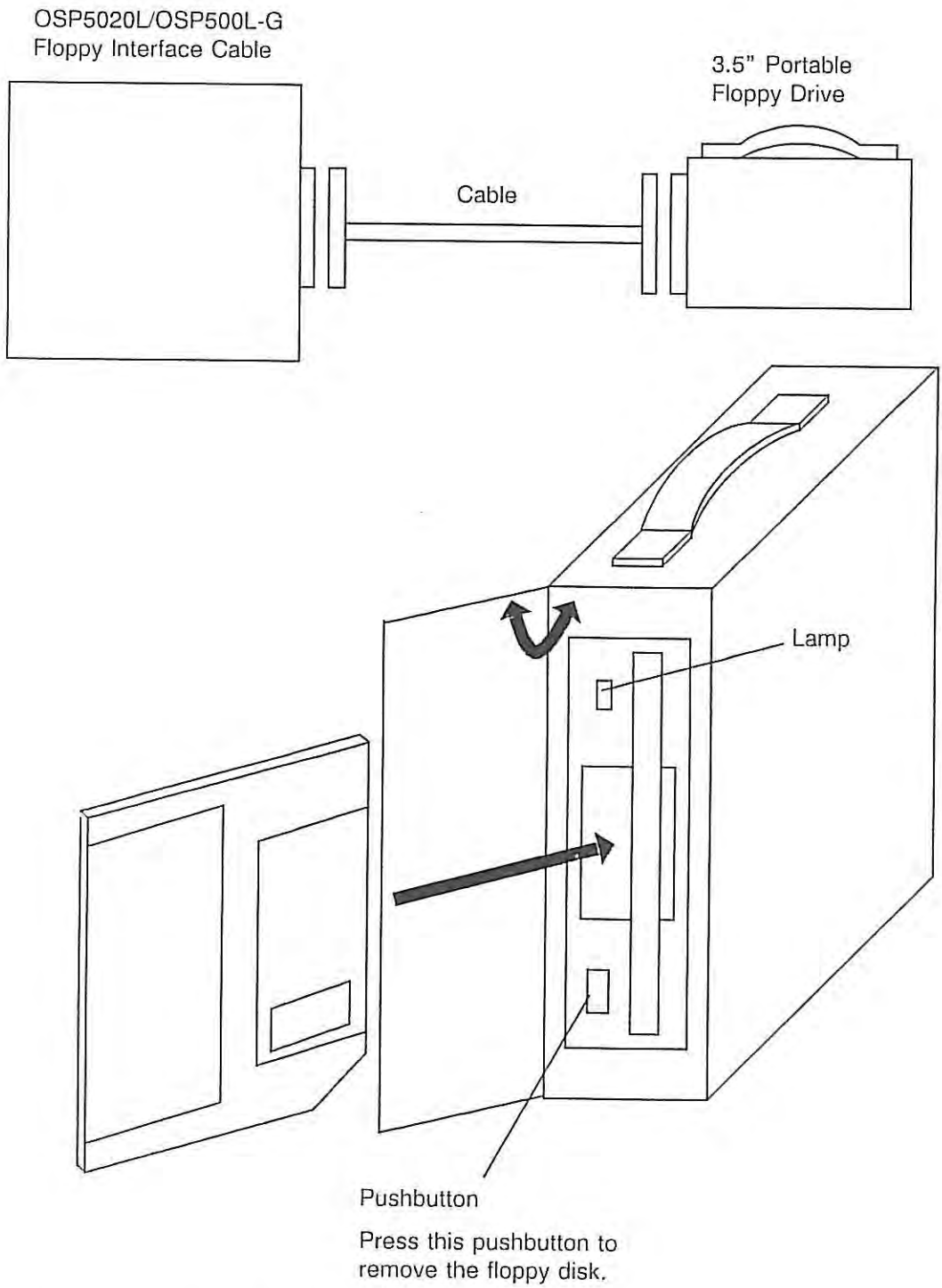
SECTION 13 PROGRAM INPUT/OUTPUT FUNCTIONS BY FLOPPY DISKS (OSP FORMAT)

1. Overview

Ordinarily, paper tape is used as the storage medium for NC machining programs. This function allows the use of floppy disks instead of paper tape. Both 3.5" and 8" floppy disks can be used in OSP format.

2. Configuration

2-1. 3.5" Portable Floppy Disk Drive



For the sector device name, use FD0:.

(1) Floppy Disk Setting

Push in the floppy disk as indicated above till it is inserted completely.

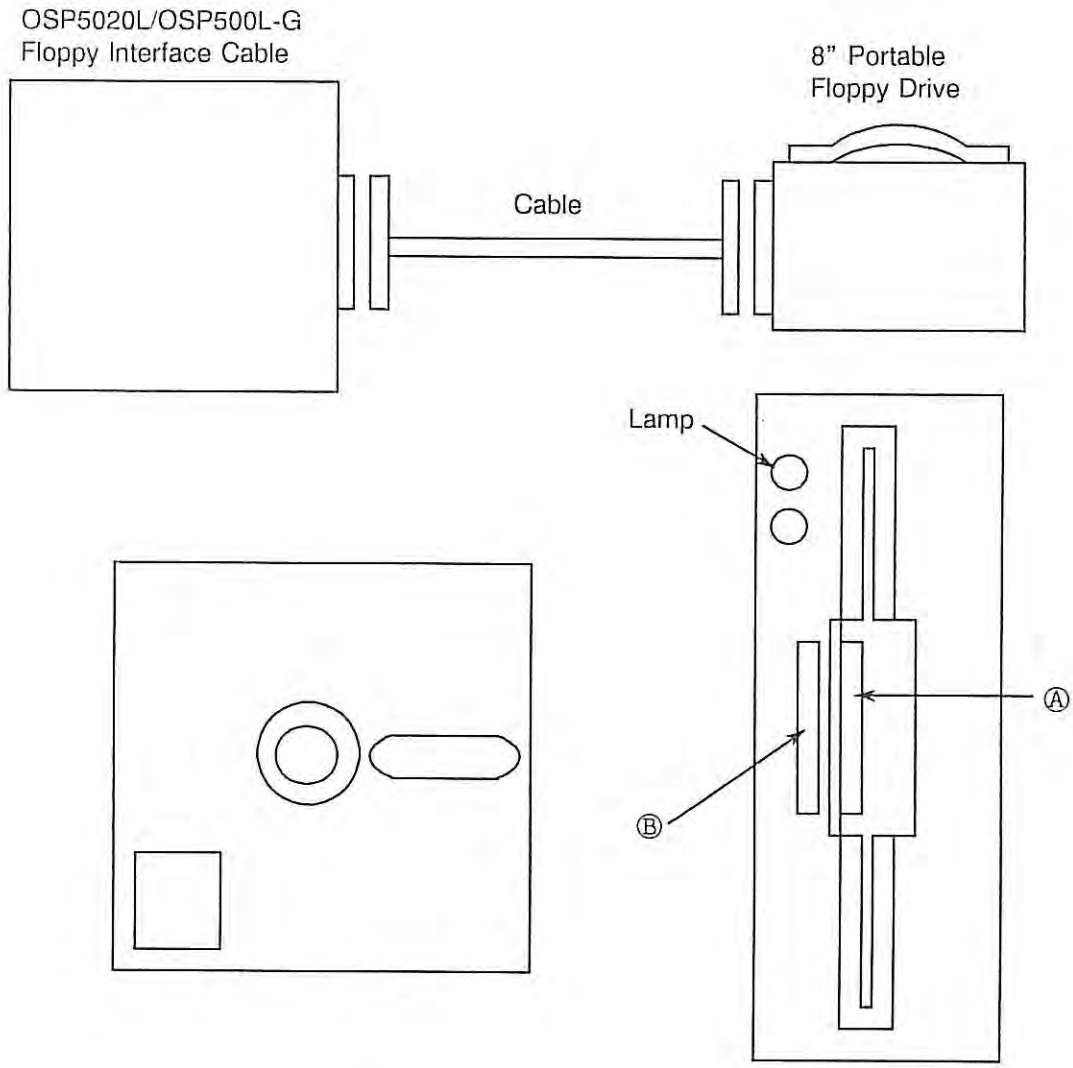
(2) Floppy Disk Removal

Press the pushbutton.

Note 1: *Before connecting/disconnecting the cable to/from the floppy disk interface of OSP5020L, turn off the power.*

Note 2: *The lamp lights while floppy disk read/write operation is being executed. It does not light when a floppy disk is inserted.*

2-2. 8" Portable Floppy Disk Drive



For the sector device name, use FDD:.

(1) Floppy Disk Setting

Press Ⓑ on the drawing above and insert and set a floppy disk.

Press Ⓐ to the left and close the cover.

(2) Floppy Disk Removal

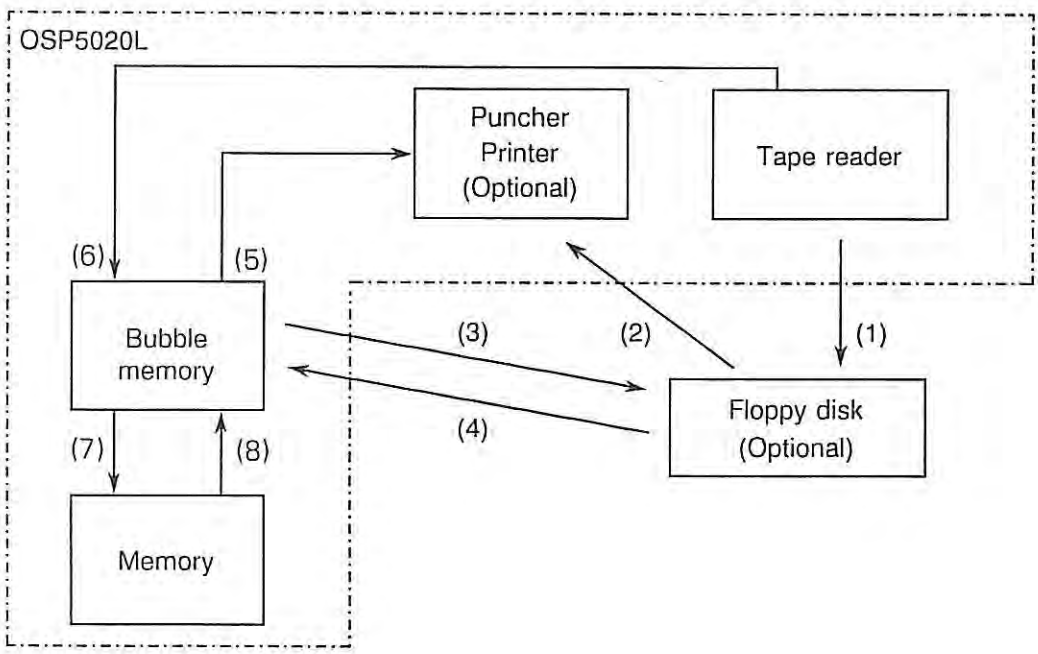
Press Ⓑ.

Note: *Before connecting/disconnecting the cable to/from the floppy disk interface of OSP5020L, turn off the power.*
The lamp lights when the floppy disk is correctly set.

3. Data Input/Output Functions by Floppy Disks

3-1. Function Overview

The data input/output operations using the floppy disk are represented by (1) through (4) in the following diagram. (2) and (7) are for tape puncher specifications (optional), and (8) through (10) are standard specifications for the OSP5020L/OSP500L-G.



(1) In the program operation mode, the NC machining program stored on the paper tape is written directly onto the floppy disk from the tape reader by the read command in the PIP mode.

(2) In the program operation mode, the NC machining program stored in the floppy disk is directly output to the puncher to create a punched tape for the NC machining program by the punch command in the PIP mode.

Similarly, the NC machining program stored in the floppy disk is directly output to the printer to create a process sheet by the list command.

Similarly, the names of the NC machining program files stored in the floppy disk are directly output to the printer to create a file name list by the directory command.

(3) In the program operation mode, the NC machining program stored in the bubble memory of the NC unit is copied onto the floppy disk by the copy command in the PIP mode.

(4) In the program operation mode, the NC machining program stored in the floppy disk is copied into the bubble memory of the NC unit by the copy command in the PIP mode.

(5) In the program operation mode, the NC machining program stored in the bubble memory of the NC unit is output to the puncher to create a punched tape of the NC machining program by the punch command in the PIP mode.

Similarly, the NC machining program stored in the bubble memory is directly output to the printer to create a process sheet by the list command.

Also in the same way, the names of the NC machining program files stored in the bubble memory are directly output to the printer to create a file name list by the directory command.

(6) In the program operation mode, the NC machining program punched onto the paper tape is read into the bubble memory of the NC unit from the tape reader by the read command in the PIP mode.

(7) In the program operation mode, the NC machining program stored in the bubble memory of the NC unit is transferred to the edit buffer by the edit command.

In the automatic operation mode, the NC machining program stored in the bubble memory is transferred to the operation execution buffer by the program selection command.

(8) In the program operation mode, the NC machining program in the edit buffer is returned to the bubble memory by the edit command.

3-2. Input/Output Relationship

The data stored in the input device is transferred to the output device by inputting the operation command.

Commands marked by an asterisk (*) are options.

Command \ External Device		Sector Device	Tape Reading Device	Tape Punching Device	Printout Device
F2 DIR		Input			Output
F3 PIP	F1 READ	Output	Input		
	F2 PUNCH	Input		Output	
	F3 VERIFY	Input/Output	Input		
	F4 COPY	Input/Output			
F5 FREE		Input			Output
F6 LIST		Input			Output
F7 IGF*		Input/Output			
F2 INIT		Output			
F3 DELETE		Output			
F4 RENAME		Output			
F5 PROTECT		Output			
F1 TAPE CONVERT*		Input/Output			
F5 DATA PIP*		Input/Output	Input	Output	Output
F6 IGF CONVERT*		Input/Output			

Connectable External Devices and Their Symbolic Designations:

Sector device	BB1:	User bubble memory
	FD0:	Floppy disk 0 (optional)
	FD1:	Floppy disk 1 (optional)
	FD2:	Floppy disk 2 (optional)
	FD3:	Floppy disk 3 (optional)

In more than one floppy disk drive unit is required, contact us.

Tape reader	CN0:	For RS232C Channel 0 tape reader
	CN1:	For RS232C Channel 1 tape reader
	CN2:	For RS232C Channel 2 tape reader
	CN3:	For RS232C Channel 3 tape reader
	CN4:	For RS232C Channel 4 tape reader

Tape punch	CN0:	For RS232C Channel 0 tape punch
	CN1:	For RS232C Channel 1 tape punch
	CN2:	For RS232C Channel 2 tape punch
	CN3:	For RS232C Channel 3 tape punch
	CN4:	For RS232C Channel 4 tape punch

Printer	CN:	Console
	PN:	Operation panel CRT
	CN0:	For RS232C Channel 0 printer
	CN1:	For RS232C Channel 1 printer
	CN2:	For RS232C Channel 2 printer
	CN3:	For RS232C Channel 3 printer
	CN4:	For RS232C Channel 4 printer

When the name of a connectable device is not designated, the respective default device name is assumed.

BB1: . . . . . for sector device
TR: . . . . . for tape reader (changeable by NC optional parameter (word) No. 54)
CN0: . . . . . for tape punch (changeable by NC optional parameter (word) No. 44)
PN: . . . . . for printer

* Default device for the tape reader and tape punch can be changed as required by parameter settings.
* When the output NC machining program name is omitted, the same name as the input NC machining program is used.
* When the input NC machining program name is omitted, the name of the input NC machining program becomes "A.MIN". However, if the NC machining program name is punched on the paper tape, then that will be used as the name of the input NC machining program.

3-3. Types of Floppy Disks

3.5" Floppy Disks:

Type	Maker	OSP Format
2DD 80 tracks 135TPI	TDK Sumitomo 3M Hitachi Maxell	9 sector/track 512 byte/sector
2HD 80 tracks 135 TPI	TDK Sumitomo 3M Hitachi Maxell	18 sector/track 512 byte/sector

8" Floppy Disks:

IBM P/N 2305830 (Hitachi Maxell FD1-128) (IBM DISKETTE 1)	Single-sided single-density: 77 cylinders/disk 1 track/cylinder 26 sectors/track 128 bytes/sector
IBM P/N 2736700 (Hitachi Maxell FD2-256) (IBM DISKETTE 2)	Double-sided single-density: 77 cylinders/disk 2 track/cylinder 15 sectors/track 256 bytes/sector
IBM P/N 1766872 (Hitachi Maxell FD2-256D) (IBM DISKETTE 2D)	Double-sided double-density: 77 cylinders/disk 2 track/cylinder 26 sectors/track 256 bytes/sector

As files are generated according to the OSP format (a file control system exclusive to the OSP5020L/OSP500L-G), floppy disks containing IBM format files generated on other computers cannot be read.

When reading floppy disks generated by IBM format into the OSP5020L/OSP500L-G, specially select floppy disk input/output function (optional) for IBM formatted disk.

New floppy disks must be initialized before they can be used.

The data stored in the floppy disk is stored by file. More than one file can be stored in a single floppy disk. Multi-volume specification which allows storing of a single program to be stored in more than one floppy disk is optionally available in the IBM format.

The data storage capacity of each floppy disk corresponds to the tape length as listed below.

Floppy Disk Type	3.5" Floppy Disks		8" Floppy Disks		
	2DD	2HD/2HC	FD1-128	FD2-256	FD2-256D
Data storage capacity (Tape length) [m]	1,840	3,770	650	1,470	2,550

【 Cautions for Handling Floppy Disks 】

(1) Do not directly touch or wipe the area of the disk that is exposed.

(2) Keep floppy disks away from magnets etc.

(3) Do not use ball point pens or pencils to write onto the floppy disk covers. Use a soft felt tip pen to write on them.

(4) Do not use clips to fasten floppy disks.

(5) Keep floppy disk in their protective envelopes during storage.

(6) Keep floppy disks out of the direct sunlight or hot places.

4. Explanation of Operations

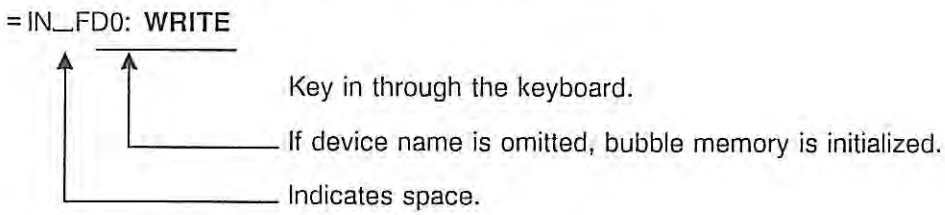
4-1. INIT: Initialize

New floppy disks which have not been formatted to the OSP format must be initialized before use.

Note: *As the data already stored on disks is deleted when initializing is carried out, take care not to initialize by mistake disks which may be required later.*

4-1-1. 3.5" Floppy Disks

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).
4) The input "FD0" after "=IN" and press the **WRITE** key.



5) In response to the input above, the system prompt

   initialize OK (Y/N)!

   appears on the screen.

   Press "Y" and then **WRITE** key.

6) The system prompt "**formatting OK (Y/N)!**" appears on the screen.

   Press "Y" and **WRITE** key.

7) Then following system prompt appears on the screen.

   format (2DD-720K = 0, 2HD-1.44M = 1)!

   Key in "0" or "1", according to the type of the floppy disk. For example, when a 2HD floppy disk is used, key in "1" and press the **WRITE** key. Initialization is started.

   In the case of OSP-formatted floppy disk, key in "N" and press the **WRITE** key in step 6) above. The following message is displayed and operation is ended.

   FDO: OSP format (2HD-1.44M) . . . . for a 2HD floppy disk

4-1-2. 8" Floppy Disks

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

   "=" is displayed on the screen.

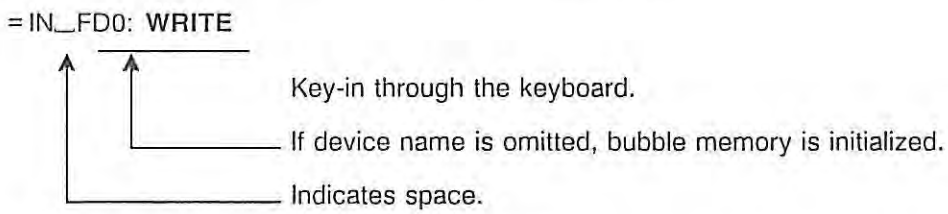
2) Press the function key [F8] (EXTEND).

   "=EX" is then displayed.

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

   This changes the display to "=IN"

4) Then input "FD0:" after "=IN" and press the **WRITE** key.



5) In response to the input above, the system prompt

   FD0: OSP format (FD2-256D) ← Floppy disk type (double-sided double-density)

   initialize OK (Y/N)!

   appears on the screen.

   Press "Y" and then **WRITE** key.

Note 1: *When a new floppy disk which has not been initialized by the OSP5020L/OSP500L-G is to be initialized, the floppy disk is usually written in the IBM format and therefore the system prompt in response to the initialization command input in step 3) and 4) above will be as indicated in step 5):*

FD0: IBM format (FD2-256D)

initialize OK (Y/N)!

*Keying "Y" and pressing the **WRITE** key in response to the prompt above initializes the floppy disk to the OSP format.*

Note 2: *If the optional floppy I/O function in the IBM format is selected, keying "Y" and press the **WRITE** key in response to the prompt displayed on the screen in step 5) above, will display the prompt below.*

type (OSP=O, IBM=I)!

*In this case, press "O" and then **WRITE** key, and the floppy disk is initialized to the OSP format.*

Note 3: *When initializing the floppy disk, the OSP automatically reads and judges which type of floppy disk is being used; FD1-128, FD2-256 or FD2-256D.*

However, if the data for floppy disk type identification has been destroyed, the system prompt displayed in response to the input of the initialization command will be as follows:

FD0: ??? format (FD?-???)

2162 floppy disc initialize

And thus the initialization of the floppy disk will be impossible.

For the floppy disks written in the format other than the IBM format or the OSP format will cause the display of the same message.

In such a case, initialize the floppy disk using the option code ";F", which allows the designation of the floppy disk type.

Example:

For designating the double-sided double-density floppy disk type

=IN FD0:;F ***WRITE***

initialize OK (Y/N)!

Press "Y" and then ***WRITE*** *key.*

formatting (Y/N)!

Press "N" and then ***WRITE*** *key.*

format (FD2-128 = 0, FD2-256 = 1, FD2-256D = 2, FD1-128 = 3)!

Press "2" and the ***WRITE*** *key.*

(Do not use FD2-128 since it has a special formatting system).

Note 4: *If the formatting data (data in the index field) of the floppy disk has been destroyed, the following error message is displayed and initialization will be impossible.*

2014 floppy disk read/write

This occurs if the floppy disk has been placed near magnet, the floppy disk has been scratched, or the floppy disk used in the device which employs different formatting system.

In this case use the option code ";F" also.

Press "Y" and the ***WRITE*** *key following the system prompt appearing on the screen.*

formatting (Y/N)!

Note that floppy disks are consumables and if a problem is with the floppy disk itself, then the initialization of such floppy disk is impossible.

Note that for the floppy disk having been formatted, read/write operations is possible only in the floppy disk drive used for formatting and these operations using other devices is not guaranteed. Therefore, formatting should be avoided unless otherwise absolutely necessary.

4-2. DEL: Delete

By designating the name of the file stored in the floppy disk, the file can be deleted.

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

   " = " is displayed on the screen.

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

   " = EX" is then displayed on the screen.

3) Press the function key [F3] (DELETE).

   This changes the display to " = DEL".

4) Key in "FD0:ABC.MIN" after " = DEL" and press **WRITE** key.

   "FD0:ABC.MIN deleted" appears on the screen showing that deletion is completed. By this operation, the file "ABC.MIN" was deleted.

Note: Do not forget to leave a space between " = DEL" and "FD0".

4-3. RENAME: Name Change

This command permits the change of a file name.

Procedure:

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

   " = " is displayed on the screen.

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

   " = EX" is then displayed on the screen.

3) Press the function key [F4] (RENAME).

   This changes the display to " = R".

4) Key in "FD0:ABC.MIN,DEF.MIN" after " = R" and press the **WRITE** key.

By the above operation, the file name "ABC.MIN" is renamed "DEF.MIN".

The contents of the file are not effected by changing of the file name.

4-4. DIR: Directory

This command displays all file names stored in the floppy disk, and prints out a list of file names on the printing device.

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

   " = " is displayed on the screen.

2) Press the function key [F2] (DIR).

   " = DI" is then displayed on the screen.

3) Key in "FD0:,CN0:" after " = DI" and press the **WRITE** key.

   Default of output device name is PN: (CRT on operation panel).

By the above operation, a list of all the files stored in the floppy disk is output to the printer (optional) connected to the NC unit (CN0:).

Option codes can also be used with this command. Input of ";P" displays file protected conditions.

4-5. FREE: Remaining Floppy Disk Area

By this command, the remaining size of area on the floppy disk and the maximum size of the contiguous vacant area can be displayed.

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

   " = " is displayed on the screen.

2) Press the function key [F5] (FREE).

   " = FR" is then displayed on the screen.

3) Key in "FD0:,CN0:" after " = FR" and press the **WRITE** key.

   Default of output device name is PN: (CRT on operation panel). Size of vacant area is displayed on the CRT in this case.

The remaining writing space is output to the printer (optional).

Option codes can be used with this command. Input of ";C" displays the maximum size of the contiguous vacant area. Therefore, in order to display the capacity in the contiguous vacant area key in "FD0:,CN0:;C" after " = FR" and press the **WRITE** key.

4-6. LIST: List of Files

By this command, a designated list of files stored on the floppy disk can be displayed on the operation panel CRT or output to the printer.

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

   " = " is displayed on the screen.

2) Press the function key [F6] (LIST).

   " = L" is then displayed on the screen.

3) Key in "FD0:ABC.MIN,PT:" after " = L" and press the **WRITE** key.

   Default of the output device name is PN: (CRT on operation panel).

By the above operation, the list of file "ABC.MIN" is printed out by the printer (optional).

4-7. READ: Reading from Tape Reader

By designating this command, the NC machining program stored in the punched tape is read onto the floppy disk.

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

   " = " is displayed on the screen.

2) Place the punched NC machining program tape into the tape reader.

3) Press the function key [F3] (PIP) for input/output operations between the OSP and peripheral equipment.

   " = PIP > " is then displayed on the screen.

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

   This changes the display to " > R".

5) Key in "FD0:ABC.MIN" after " > R" and press the **WRITE** key.

By the above operation, the NC machining program on the punch tape is read from the tape reader under the name of "ABC.MIN".

The following option codes can be used with this command.

Option Code	Meaning
;E	EIA code designation
;I	ISO code designation
;V	Verify designation
;A	File designation (used when designating the destination file to which the program is stored)
;C	When read error has occurred, continuous reading can be carried out. Symbol "!" is displayed or punched for character which causes an error.

4-8. Output of Files to the Tape Punch

By specifying the file name, the data stored in the floppy disk is punched out onto the paper tape.

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

   " = " is displayed on the screen.

2) Press the function key [F3] (PIP) for input/output operations between the OSP and peripheral equipment.

   " = PIP > " is then displayed on the screen.

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

   This changes the display to " > P".

4) Key in "FD0:ABC.MIN,PP:" after " > P" and press the **WRITE** key.

If an output device name is not designated, the device assigned by the parameter setting is selected.

By the above operation, all data stored in the file "ABC.MIN" is output to an optional tape punch.

The following option codes can be used with this command.

Option Code	Meaning
;E	EIA code designation
;I	ISO code designation

4-9. Verify

By designating this command, the NC machining program punched out onto the paper tape is checked with the program stored in the floppy disk to see that the file name, program data etc. are matching.

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

   "=" is displayed on the screen.

2) Place the punched NC machining program tape into the tape reader.

3) Press the function key [F3] (PIP) for input/output operations between the OSP and peripheral equipment.

   "=PIP >" is then displayed on the screen.

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

   This changes the display to ">V".

5) Key in "FD0:ABC.MIN" after ">V" and press the **WRITE** key.

By the above operation, the system checks whether the data of file "ABC.MIN" punched onto the paper tape matches that of the stored in the floppy disk. The device name for reading the tape, such as tape reader (TR:), may be omitted.

When verification is completed, the following is displayed:

end of tape

end of file

all same data

When the data does not match, data causing mismatch flashes on the CRT and verification stops temporarily.

The following option codes can be used with this command:

Option Code	Meaning
;E	EIA code designation
;I	ISO code designation

4-10. Read from Bubble Memory

By designating the file stored in the bubble memory, the NC program of that file name is read onto the floppy disk.

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

   " = " is displayed on the screen.

2) Press the function key [F3] (PIP) for input/output operations between the OSP and peripheral equipment.

   " = PIP > " is then displayed on the screen.

3) Press the function key [F4] (COPY).

   This changes the display to " > CO".

4) Key in "ABC.MIN,FD0:" after " > CO" and press the **WRITE** key.

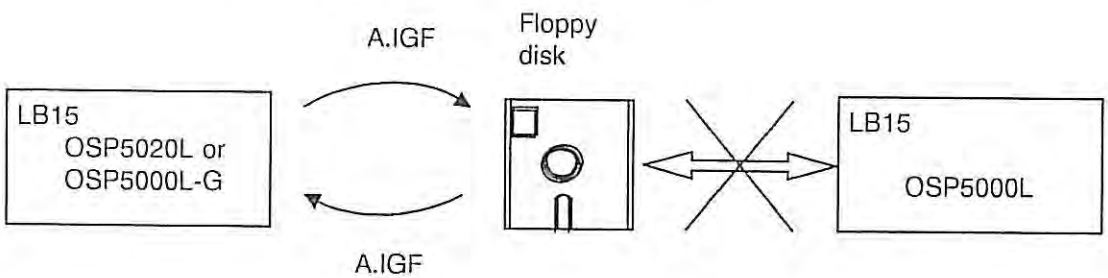
By this operation, the data of file "ABC.MIN" stored in the bubble memory is transferred to the floppy disk under the same file name.

The following option codes can be used with this command.

Option Code	Meaning
;A	File designation (used when designating the destination file to which the program is stored)
;V	VERIFY (The prompt "Y/N" will appear on the screen.)

Note: *As IGF data files conform strictly to IGF operations and specifications, IGF data files cannot be transferred when IGF specifications of the IGF-L2, L2A and L3 etc. differ or when the type of machines are different.*

As a rule, IGF data files generated on the same machines only can be used, however, when an addition has been made to the IGF specifications, data files generated prior to the change in the specification cannot be used.



4-11. Data Transmission to Bubble Memory

By designating the file stored in the floppy disk, the NC program of that file name is read into the bubble memory.

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

   "=" is displayed on the screen.

2) Press the function key [F3] (PIP) for input/output operations between the OSP and peripheral equipment.

   "=PIP >" is then displayed on the screen.

3) Press the function key [F4] (COPY).

   This changes the display to ">CO".

4) Key in "FD0:ABC.MIN" after ">CO" and press the **WRITE** key.

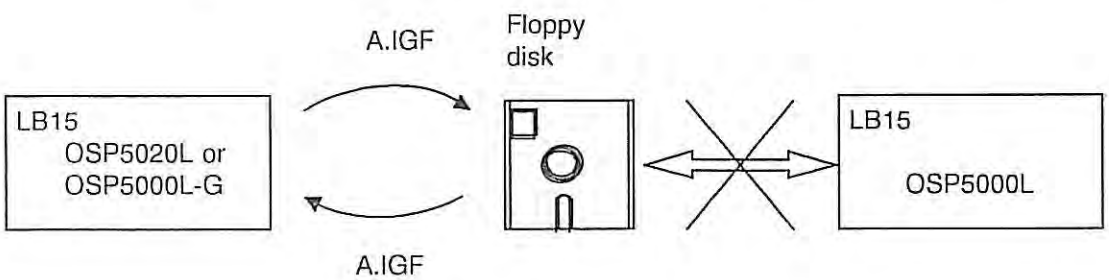
By this operation, the data of file "ABC.MIN" stored in the floppy disk is transferred to the bubble memory under the same file name.

The following option codes can be used with this command.

Option Code	Meaning
;A	File designation (used when designating the destination file to which the program is stored)
;V	VERIFY (The prompt "Y/N" will appear on the screen.)

Note: *As IGF data files conform strictly to IGF operations and specifications, IGF data files cannot be transferred when IGF specifications of the IGF-L2, L2A and L3 etc. differ or when the type of machines are different.*

As a rule, IGF data files generated on the same machines only can be used, however, when an addition has been made to the IGF specifications, data files generated prior to the change in the specification cannot be used.



4-12. IGF Floppy Disk Files (optional)

NC machining programs generated based on the IGF data stored in the floppy disk are stored in the floppy disk.

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

   "=" is displayed on the screen.

2) Press the function key [F7] (IGF).

   "=IGF" is then displayed on the screen.

3) Press the function key [F1] (GRAPHIC EDIT).

   This changes the display to ">GE".

4) Key in "FD0:A.IGF,FD0:B.IGF" after ">GE" and press the **WRITE** key.

   "FD0:A.IGF" is the input IGF data.

   "FD0:B.IGF" is the output IGF data.

   The NC machining program is generated based on the IGF data of file "A.IGF" stored in the floppy disk and the IGF data of file "B.IGF" is stored in the floppy disk.

   When the output file name is omitted the same as the input file name "A.IGF" is used as the output file name. The old IGF data is replaced with the new IGF data.

5) The NC machining program is generated by the IGF operation.

   Refer to OSP5020L/OSP500L-G Operation Manual.

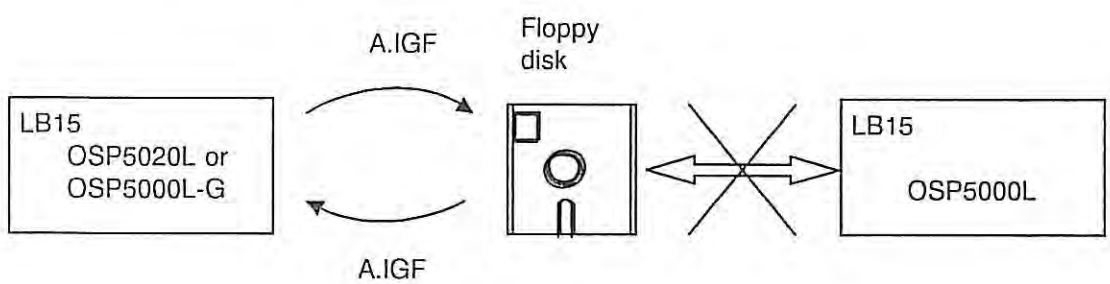
6) When generating an IGF program, give it a name as indicated below.

   ! FD0:ABC.MIN,O1234 **WRITE**

   By the above operation, the NC machining program "ABC.MIN" generated by IGF is given the program name "O1234".

Note: *As IGF data files conform strictly to IGF operations and specifications, IGF data files cannot be transferred when IGF specifications of the IGF-L2, L2A and L3 etc. differ or when the type of machines are different.*

As a rule, IGF data files generated on the same machines only can be used, however, when an addition has been made to the IGF specifications, data files generated prior to the change in the specification cannot be used.



4-13. IGF Floppy Disk File Conversion (optional)

NC machining programs generated by IGF stored on the floppy disk are converted to a general NC machining program format for use in the OSP5000 or OSP3000 series, and stored in the floppy disk.

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

   "=" is displayed on the screen.

2) Press the function key [F7] (IGF).

   "=IGF" is then displayed on the screen.

3) Press the function key [F6] (IGF CONVERT).

   This changes the display to "=IGFC".

4) Key in "FD0:ABC.MIN,FD0:ABC0.MIN" after "=IGFC" and press the **WRITE** key.

   "FD0:ABC.MIN" is the input file name.

   "FD0:ABC0.MIN" is the output file name.

By the above operation, the NC machining program "ABC.MIN" stored in the floppy disk is converted to general NC machining program format and stored in the floppy under the name "ABC0.MIN".

The following option codes can be used with this command.

Option Code	Meaning
;A	Conversion to OSP5000 program
;B	Conversion to OSP3000 program
;S	NC machining programs are generated with vacant included.
;T	The NC machining program is converted for use in turret B.

Refer to the OSP5020L/OSP500L-G Operation Manual for Interactive Graphic MDI Function (IGF), Section 8, "Conversion Function" for further details of the conversion.

Note: *Always carry out IGF conversion on the machine which was used to generate the NC machining program.*

4-14. Floppy Disk File Conversion (optional)

The NC machining program for the OSP3000 series stored in the floppy disk is converted to the OSP5000 series NC machining program format, and is stored in the floppy disk.

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

   "=" is displayed on the screen.

2) Press the function key [F8] (EXTEND) twice.

3) Press the function key [F1] (TAPE CONVERT).

   This changes the display to "=C".

4) Key in "FD0:A.MIN,FD0:A0.MIN" after "=C" and press the **WRITE** key.

"FD0:A.MIN" is the input file name.

"FD0:A0.MIN" is the output file name.

By the above operation, the NC machining program "A.MIN" for use with the OSP5020/OSP3000 series stored on the floppy disk is converted to the program for use with the OSP5020/OSP5000 series, and is stored in the floppy disk under the name "A0.MIN".

Refer to Section 7, "Tape Conversion Function (OSP5000L → OSP5020L/OSP5000L-G/OSP500L-G)" for details of the conversion.

4-15. Floppy Disk File Protection

By designating this command, files stored in the floppy disk are protected so that they cannot be written over, edited or deleted. In effect, by this command the file is made into a read only file.

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

"=" is displayed on the screen.

2) Press the function key [F8] (EXTEND) twice.

3) Press the function key [F5] (PROTECT).

This changes the display to "=PROT".

4) Key in "FD0:A.MIN" after "=PROT" and press the **WRITE** key.

When this operation is completed, the display **"A.MIN file protection end"** appears on the screen, denoting that the NC machining program "A.MIN" stored on the floppy disk has a protection label written for it.

In the following cases, the files are protected and the display **"A.MIN file write protection"** appears on the screen.

In the case of output file name is "FD0:A.MIN" in the following operations:

[F1] . . . . . READ
[F4] . . . . . COPY
[F4] . . . . . EDIT
[F3] . . . . . DELETE
[F4] . . . . . RENAME
[F5] . . . . . DATA PIP

The following option codes can be used with this command.

Option Code	Meaning
;C	Release of the protection label for a designated file
;V	VERIFY (The prompt Y/N will appear on the screen.)

4-16. Floppy Disk File Data Transfer (PIP) (optional)

By this command, the tool offset, zero offset and parameter values stored in the bubble memory are stored in the floppy disk file or vice versa.

Refer to Section 12, "Tape Data Input/Output Function" for further details.

5. Combined Use of 3.5" Floppy Disk Drive and 8" Floppy Disk Drive

Combined use of the 3.5"floppy disk drive and the 8" floppy disk drive is possible, with the restrictions indicated below.

(1) The sector device name must be "FD0:".

(2) The 3.5"floppy disk drive and the 8" floppy disk drive cannot be connected to the NC at the same time. Connect either the 3.5" floppy disk drive or the 8" floppy disk drive.
(from floppy interface version FDCBII)

SECTION 14 CALENDAR TIMER

1. Overview

Conventionally, warming up of the machine has been carried out by the operator. This calendar timer is provided to automatically carry out the warming up of the machine without requiring operator's attendance. At the preset time, external calendar timer signal is input to the control to automatically turn on power supply. The control then automatically selects and starts the program for warming up the machine.

2. Warm-Up Program

As explained above, the program for warming up the machine (hereafter referred to as warm-up program) is selected by the calendar timer signal. This warm-up program must be prepared at the same level of main programs and assigned with file name "W.WAF". Prepare this program in advance.

Note: *The file name must be W.WAF and the use of other file name is not allowed. If this file is not found in the memory, an alarm occurs.*

Examples of warm-up programs:

Example 1: Warm-up of only the spindle

```
N001 M185 M157          ;Cancels interlock (*)
N002 M03  S1000 M42     ;Selects gear range or winding and starts spindle
N003 G04  F1800         ;Spindle rotation for 30 minutes in the dwell mode
N004 M05                ;Spindle stops
NEND M02
```

Example 2: Warm-up including axis movements

```
N001 M185 M157
N002 G00  X1000 Z1000
N003 M03  S1000 M42   T0101
N004 G01  F1    X500  Z300
 :
 :
NEND M02
```

* Since the warm-up program is processed in the same level as main programs, interlock conditions of spindle rotation might prevent the execution of the warm-up program. For instance, the spindle cannot rotate unless the chuck is clamped. The M codes, M185 and M157, designated in block N001 are used for canceling the interlock conditions. (M185 and M157 are optional. When these M codes are not available on the machine, warm up the machine with the workpiece clamped so that it is not prevented by interlock conditions.) For details of interlock condition canceling M codes, refer to Sections 25 and 31.

Note: *When the calendar timer is used in combination with the automatic power shut-off specification, set the switches as indicated below. The calendar timer is set automatically after the execution of automatic power shut-off.*

Switch	Setting
Calendar timer ON/OFF	ON
Auto power shut-off ON/OFF	ON

3. Alarm Messages

An alarm described below will occur if the conditions necessary for executing warm-up operation are not satisfied.

ALARM-C

954 Warm-up operation

Code :

1 ..... Warm-up program not found

2 ..... Calendar timer signal is input at other than power on timing.

3 ..... Other than automatic mode selected when starting warm-up program, or machine lock, single block or independent A/B turret operation mode selected.

To reset the alarm state, turn off the calendar timer ON/OFF switch and press the **RESET** pushbutton switch.

4. M Codes for Canceling Interlock Functions

M185 Spindle rotation possible with chuck unclamped

M184 Cancel of M185

M157 Spindle rotation possible with tailstock spindle at retraction end for center-work

M156 Cancel of M157

SECTION 15 EDIT INTERLOCK FUNCTION

1. Overview

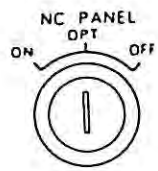
This function enables the operations of the controls on the NC operation panel excluding edit operations when the **NC PANEL** switch is placed in the "OPT" position.

The relationship between the setting of the **NC PANEL** switch and the available operations when the edit interlock function is selected is described below.

(1)

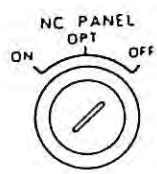
All operations using the switches on the NC and machine operation panels are possible.

(2)



The edit operation using the switches on the NC operation panel is impossible. All other operations are allowed.

(3)



Operations using the switches on the NC operation panel are impossible.

Setting the **NC PANEL** switch in the ON or the OFF position has the same effect as the setting at these positions for the control not supported by the edit interlock function.

SECTION 16 WORK COUNTER

1. Overview

There are three types of work counters.

(1) Workpiece count only

The work counter counts the designated M02 commands, and the number of finished workpieces is accumulated.

(2) Cycle stop

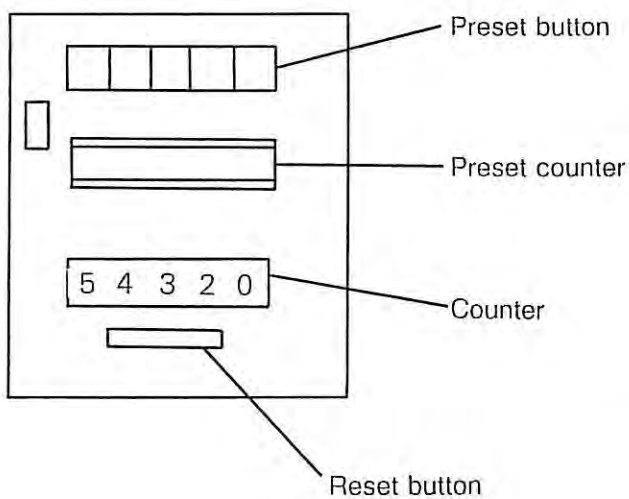
Machining cycle is stopped when the preset number has been reached.

(3) Cycle start disabled

Cycle start of the NC is disabled when the number of finished workpieces has reached the number which is set at the preset counter of work counter.

Restart from the slide hold mode and cycle start in the MDI mode are possible.

The counter output is not made in the machine lock mode, and is not counted up.



Preset Counter (Example)

SECTION 17 CYCLE TIME REDUCTION FUNCTION

There are many possible ways of reducing the cycle time even by one second. For NC specifications, the following M codes enable the simultaneous execution of two functions to cut cycle time.

M63 . . . . . Ignoring spindle rotation answer

M64 . . . . . Ignoring answer signals for M codes other than spindle rotation control

M65 . . . . . Ignoring answer signals for T commands

M66 . . . . . Free turret indexing position

M62 . . . . . Confirmation M answer ignored by M64

1. Ignoring Spindle Rotation Answer, M63

Answer signals related with to spindle rotation control, such as spindle rotation answer for M03, M04 and M05, spindle in neutral position answer for M40, gear range match answer signal for M41 through M44, spindle orientation answer for M19, and constant speed arrived answer for S commands are all confirmed before the start of axis movements. However, when M63 is designated in the block containing the above indicated M and/or S command, answer signal of the designated command is confirmed after the completion of axis movements, thus permitting axis movement commands to be simultaneously executed with the above indicated commands.

Example:
```
N100 G00 X50 Z50 P100 M03 S1000 M63 CR
N101          Z10                   CR
```

With the commands in block N100, axis movement commands are executed simultaneously with the spindle start command, M03, and spindle rotation answer and constant spindle speed arrived answer signals are confirmed only after the execution of the axis movement commands in that block.

2. Ignoring Answer Signals for M Codes Other Than Spindle Rotation Control, M64

For M codes which are associated with answer signals except those relating to spindle rotation control, answer signals indicating that the designated M commands have been completed are confirmed before the start of axis movements. However, when M64 is designated in the block containing such M codes, execution of the commands in the subsequent block starts without waiting for answer signals.

M codes to which M64 is effective are:
(For special M codes, consult your local Okuma representative.)

M55, M56	Tailstock spindle retract/advance
M76, M77	Unloader retract/advance
	Work catcher retract/advance
	Swing arm loader retract/advance
M83, M84	Chuck close and open
M90, M91	Front door close and open
M92, M93	Bar feeder retract/advance
M94, M95	Loader loading/unloading
M101, M102 M103, M104 M105, M106 M107, M108 M230, M231 M232, M233 M234, M235 M236, M237	Spare

When confirmation of answer signals of these M codes is required, program M62 and the axis movement command is executed after the answer signal has been confirmed.

Example:
```
N100  G00  X50  Z50  M56  M64  CR
N101            Z30            CR
N103  G01       Z 5  F0.5 M62  CR
```

With the commands in block N100, axis movement commands are simultaneously executed with the tailstock spindle advance command, and the answer signal for M56 is confirmed in block N103.

3. Ignoring Answer Signals for T Commands, M65

When a T command is designated, axis movement commands in the same block are executed after the programmed turret indexing has been completed. However, when M65 is designated in the block containing a T command, axis movement commands in the same block are executed simultaneously with the T command and then the commands in the following sequence are executed.

When it is necessary to confirm the completion of turret indexing, program the same T command without M65.

Example:
```
N100  G00  X100  Z100  T0101  M65  M66  CR
N101             Z 30  T0101            CR
```

With the commands in block N100, positioning at a point (X100, Z100), is executed simultaneously with turret indexing; completion of the turret indexing is confirmed in block N101.

4. Free Turret Indexing Position, M66

Usually, the turret is indexed only at the positive travel end (variable soft limit) position of either X- or Z-axis. However, when M66 is designated with a T command, turret indexing can be performed at any position.

Example:
```
N100  T0200  M66  CR
N101  T0300       CR
```

With the commands in block N100, the turret can be indexed even when the turret is not positioned at the variable soft limit in the positive direction. Since, M66 is active only in the programmed block, it is not effective in block N101 and turret indexing for T0300 command is executed at a predetermined turret indexing position.

SECTION 18 TOOL LIFE MANAGEMENT

1. Overview

The tool life management function automatically selects the spare tool of the tool whose service life has been expired. The tools managed by this function are first grouped and the service life of the tool used is checked based on the predetermined tool life condition.

The tool life conditions which can be used for the tool life management function are indicated below.

a) Number of the machined workpieces
b) Accumulated cutting time
c) Accumulated tool wear amount
d) Tool gauging data

When all tools registered in the same group have been used to the life, the control stops in the alarm B state or in the cycle stop state by the external cycle stop signal. Note that the cycle stop using the external cycle stop signal is optional.

Information covered in this section includes:

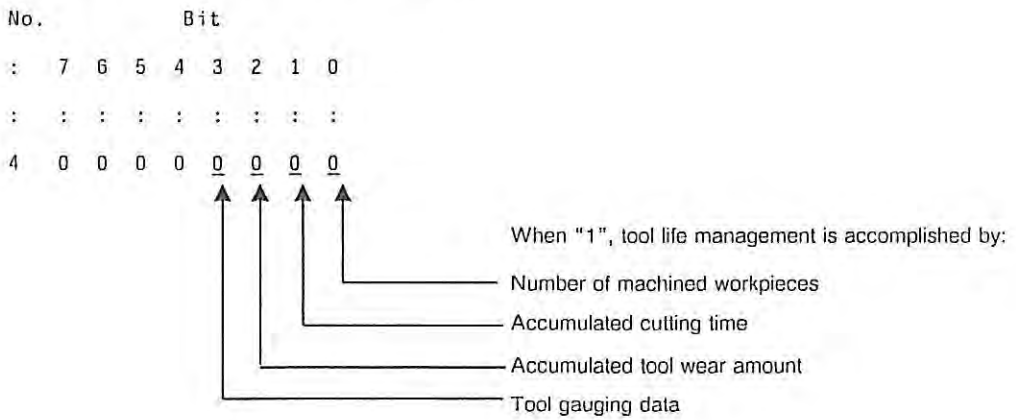
(1) Selection of tool life conditions
(2) Setting tool life data and display of tool life data
(3) Programming
(4) Processing after all spare tools have been used up
(5) Alarm messages

2. Selection of Tool Life Conditions

There are four factors selectable to determine expiration of tool service life for activating automatic spare tool selection function.

a) Number of the machined workpieces
b) Accumulated cutting time
c) Accumulated tool wear amount
d) Tool gauging data

The operator can select the factor to be used to determine expiration of tool service life by parameter setting: Optional Parameter (bit) No. 4.



3. Setting Tool Life Data and Display of Tool Life Data

3-1. Data Setting Procedure

(1) Press the function key [F7] (ITEM ↓) in the TOOL DATA SET mode so that the CRT displays the *TOOL LIFE CONTROL TOOL TABLE A* page. Note that this page is not displayed unless the number of machined workpieces has been selected as the factor for determining tool service life.

Read the following instructions with referring to display examples.

a) Registration of tool - designation of tool group number (TOOL-G)

After moving the cursor to the TOOL-G data of the required tool number, key in the tool group number and press the function key [F1] (SET).

b) Setting workpiece number (SET-N)

In the same manner as in a), set the tool life in terms of the number of workpieces.

c) Registration of tool offset number (TOOL OFFSET NO.)

Set the tool offset number in the same manner as in a).

Max. three tool offset numbers can be set.

d) Setting cutting time (SET-TIME)

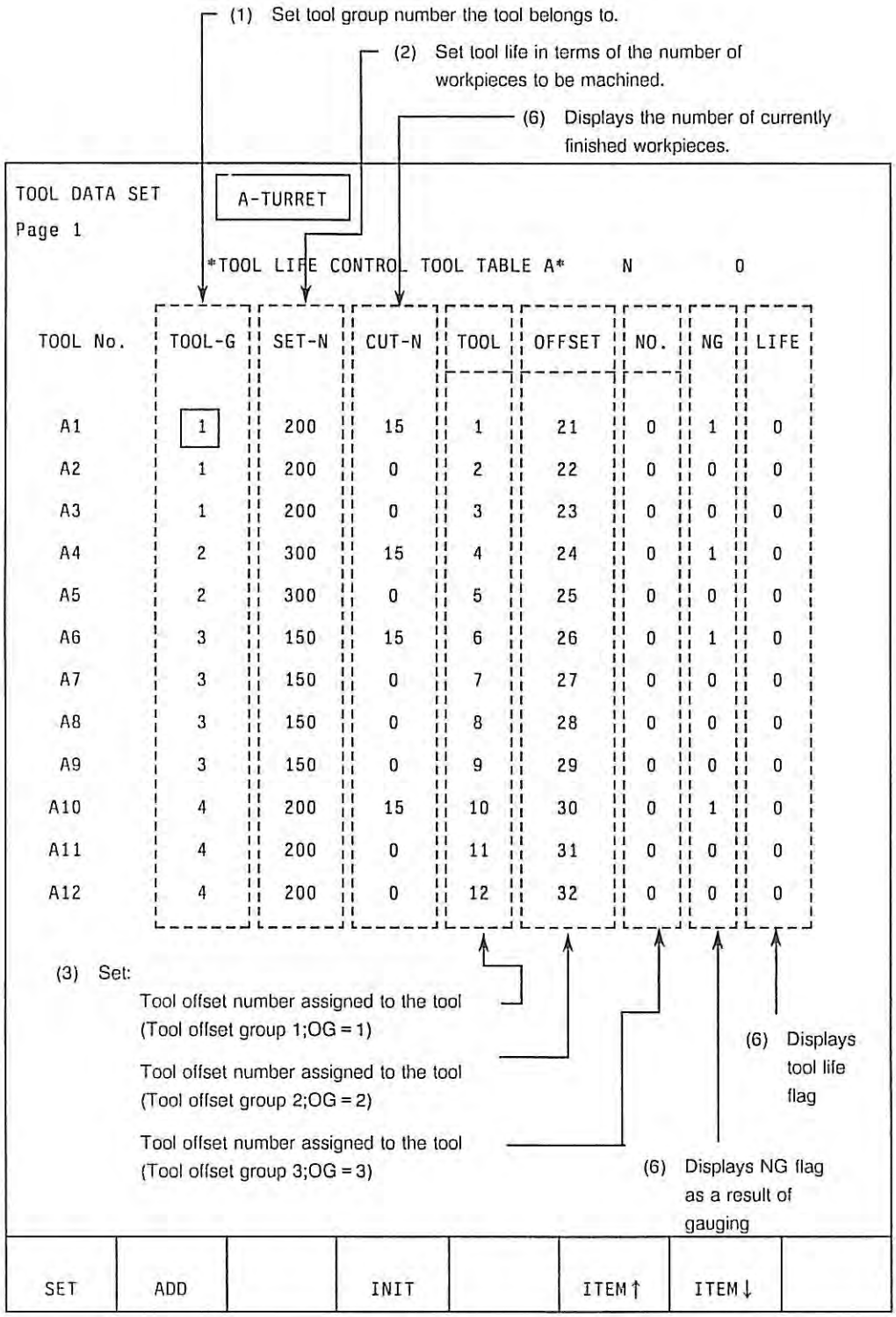
Set the tool life in terms of cutting time in the same manner as in a). Note that this setting is possible from the *TOOL LIFE CONTROL TOOL TABLE B* page which is not displayed unless the accumulated cutting time has been selected as the factor for determining tool service life.

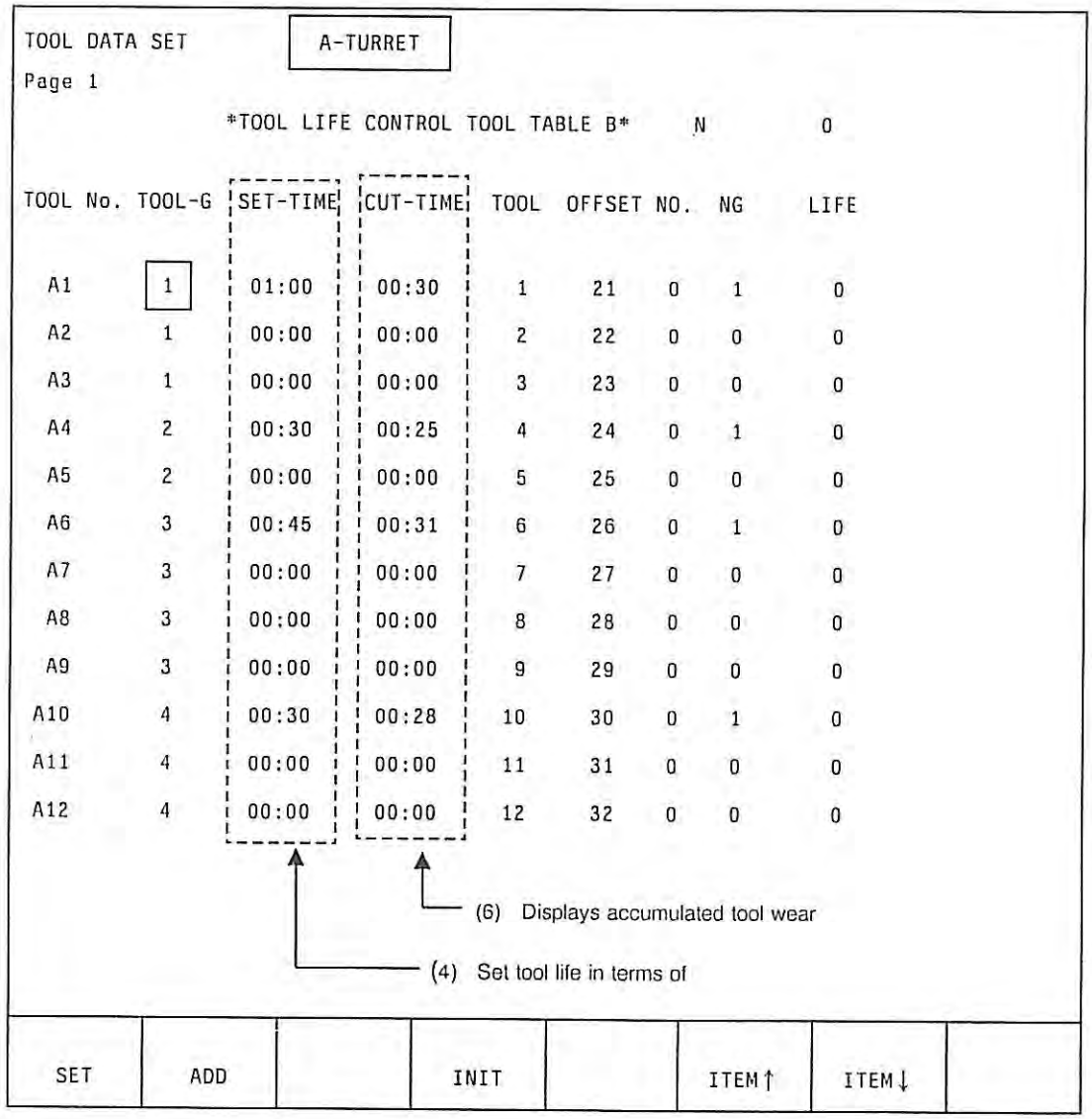
e) Setting tool wear amount

Set the tool life in terms of tool wear amount in the same manner as in a). Note that this setting is possible from the *TOOL LIFE CONTROL TOOL TABLE C* page which is not displayed unless the accumulated tool wear has been selected as the factor for determining tool service life.

f) NG flag and LIFE flag determined based on the results of the counted workpiece number, cutting time, accumulated tool wear amount, or the results of tool gauging cycle are set and displayed while the machine is operating; it is unnecessary to set the data for them.

These data are cleared when the tool life control data tables are initialized. (See 3-2.)


(1) Set tool group number the tool belongs to.
(2) Set tool life in terms of the number of workpieces to be machined.
(6) Displays the number of currently finished workpieces.
TOOL DATA SET
A-TURRET
Page 1
TOOL LIFE CONTROL TOOL TABLE A N 0
TOOL No.
TOOL-G
SET-N
CUT-N
TOOL
OFFSET
NO.
NG
LIFE
A1 1 200 15 1 21 0 1 0
A2 1 200 0 2 22 0 0 0
A3 1 200 0 3 23 0 0 0
A4 2 300 15 4 24 0 1 0
A5 2 300 0 5 25 0 0 0
A6 3 150 15 6 26 0 1 0
A7 3 150 0 7 27 0 0 0
A8 3 150 0 8 28 0 0 0
A9 3 150 0 9 29 0 0 0
A10 4 200 15 10 30 0 1 0
A11 4 200 0 11 31 0 0 0
A12 4 200 0 12 32 0 0 0
(3) Set:
Tool offset number assigned to the tool (Tool offset group 1;OG = 1)
Tool offset number assigned to the tool (Tool offset group 2;OG = 2)
Tool offset number assigned to the tool (Tool offset group 3;OG = 3)
(6) Displays tool life flag
(6) Displays NG flag as a result of gauging
SET
ADD
INIT
ITEM↑
ITEM↓


TOOL DATA SET
A-TURRET
Page 1
TOOL LIFE CONTROL TOOL TABLE B
N
0
TOOL No. TOOL-G SET-TIME CUT-TIME TOOL OFFSET NO. NG LIFE
A1 1 01:00 00:30 1 21 0 1 0
A2 1 00:00 00:00 2 22 0 0 0
A3 1 00:00 00:00 3 23 0 0 0
A4 2 00:30 00:25 4 24 0 1 0
A5 2 00:00 00:00 5 25 0 0 0
A6 3 00:45 00:31 6 26 0 1 0
A7 3 00:00 00:00 7 27 0 0 0
A8 3 00:00 00:00 8 28 0 0 0
A9 3 00:00 00:00 9 29 0 0 0
A10 4 00:30 00:28 10 30 0 1 0
A11 4 00:00 00:00 11 31 0 0 0
A12 4 00:00 00:00 12 32 0 0 0
(6) Displays accumulated tool wear
(4) Set tool life in terms of
SET
ADD
INIT
ITEM↑
ITEM↓

TOOL DATA SET [A-TURRET]

Page 1

TOOL LIFE CONTROL TOOL TABLE C UNIT 1 mm

TOOL No.	TOOL-G	SET-ABR	CUT-ABR	TOOL	OFFSET NO.	NG	LIFE
A1	[1]	2.000	0.400	1	21	0	1	0
A2	1	2.000	0.000	2	22	0	0	0
A3	1	2.000	0.000	3	23	0	0	0
A4	2	1.000	0.100	4	24	0	1	0
A5	2	1.000	0.000	5	25	0	0	0
A6	3	1.500	0.500	6	26	0	1	0
A7	3	1.500	0.000	7	27	0	0	0
A8	3	1.500	0.000	8	28	0	0	0
A9	3	1.500	0.000	9	29	0	0	0
A10	4	1.000	0.200	10	30	0	1	0
A11	4	1.000	0.000	11	31	0	0	0
A12	4	1.000	0.000	12	32	0	0	0

(6) Displays accumulated tool wear amount.

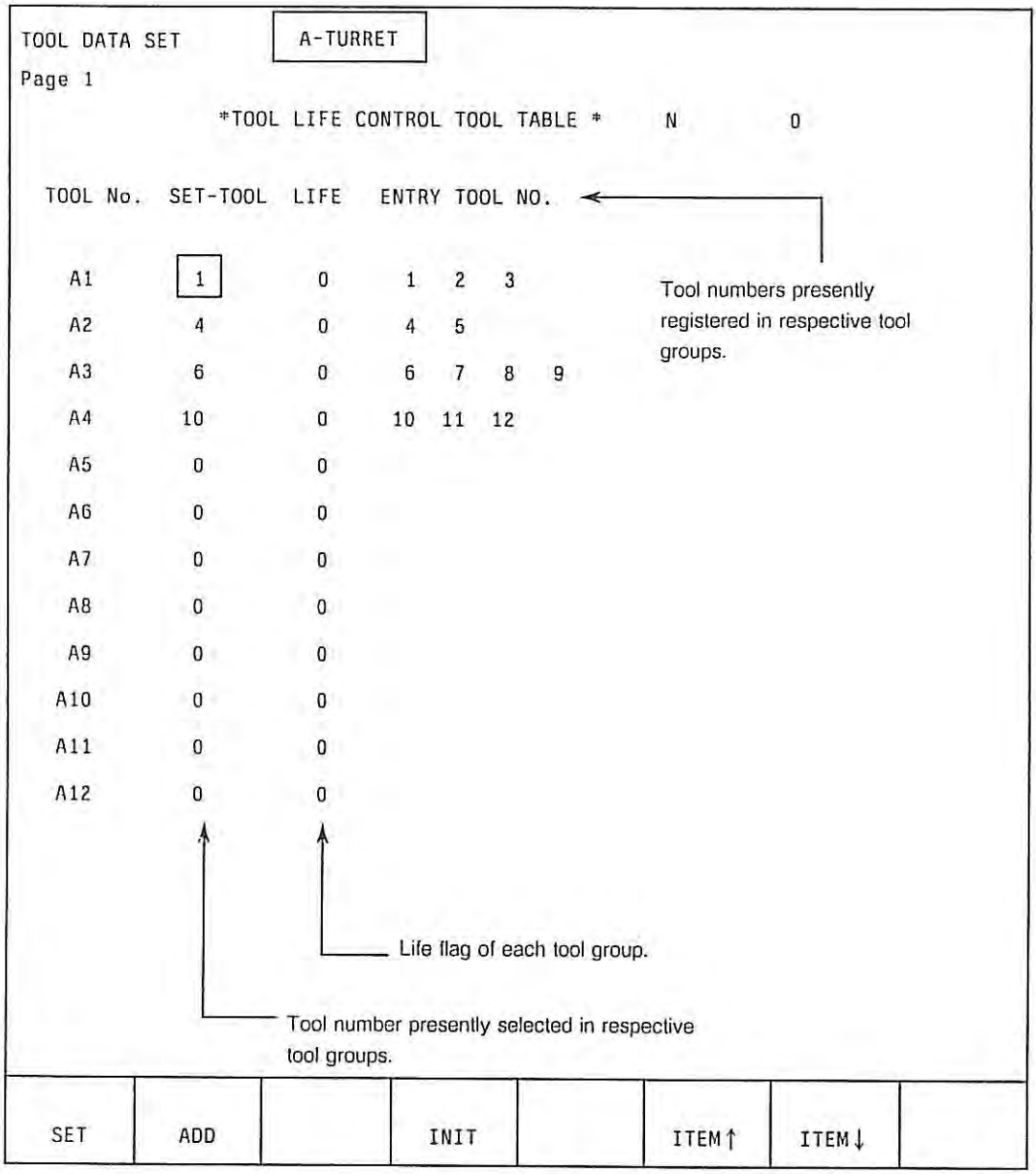
(5) Set tool life in terms of tool wear amount.

SET	ADD		INIT		ITEM↑	ITEM↓	

(2) Press the function key [F7] (ITEM ↓) to display the *TOOL LIFE CONTROL GROUP TABLE screen*.

For this page, it is unnecessary to set the data.

When the tool life control pages are initialized, the tool assigned with the smallest tool number in each group is automatically selected and the LIFE flag of each tool group is cleared to zero.



(3) Data Setting Range

Setting range of individual data is as follows:

a) Tool Life Control Tool Tables

Tool group No.	0 to 12	(0: not registered)
	0 to 25	(ATC specification)
Workpiece number	0 to 9999	
Cutting time	00 hours 00 minute to 99 hours 59 minutes	
Tool wear amount	0 to 999.999 mm	
Tool offset No.	0 to 32	(0: not registered)
NG flag	0 or 1	
. . . . .	0:	Serviceable
	1:	Unacceptable
Tool life flag	0 or 1	
. . . . .	0:	Serviceable
	1:	Life expired

b) Tool Life Control Group Table

Selected tool	0 to 12 (turret A)	
	0 to 12 (turret B)	(0: not registered)
Life flag	0 or 1	
. . . . .	0:	Serviceable
	1:	Life expired

3-2. Initializing Tool Life Control Table

After designating tool group number for respective tool numbers and tool offset numbers and setting life conditions, it is necessary to initialize tool life control tables.

This clears all the data - current workpiece number, cutting time, tool wear amount, NG flag, and life flag, and automatically sets the tool assigned with the smallest number of a group.

After changing all the tools in setup, the tool life control tables must be initialized.

Follow the steps indicated in the next page to initialize them.

TOOL DATA SET [A-TURRET]

Page 1

TOOL LIFE CONTROL TOOL TABLE A

TOOL No.	TOOL-G	SET-N	CUT-N	TOOL	OFFSET NO.		NG	LIFE
1	1	200	15	1	21	0	1	0
2	1	200	0	2	22	0	0	0
3	1	200	0	3	23	0	0	0
4	2	300	15	4	24	0	1	0
5	2	300	0	5	25	0	0	0
6	3	150	15	6	26	0	1	0
7	3	150	0	7	27	0	0	0
8	3	150	0	8	28	0	0	0
9	3	150	0	9	29	0	0	0
10	4	200	15	10	30	0	1	0
11	4	200	0	11	31	0	0	0
12	4	200	0	12	32	0	0	0

Initialized (Y/N)

Tool group No. (0 = All group) !

SET	ADD		INIT		ITEM↑	ITEM↓	

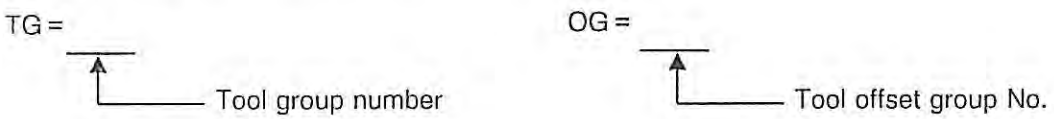
Initializing Procedure:

1) Press the function key [F4] (INIT).
2) Prompt "**Initialize (Y/N)!**" is displayed.

   To initialize the tool life control table, key in "Y" and press the **WRITE** key. If initialization is not required, key in "N" and press the **WRITE** key.
3) When "Y" is entered, prompt "**Tool group No. (0 = All group)!**" is displayed. Key in the tool group number(s) and press the **WRITE** key. To initialize the data for all tool groups, key in "0" and press the **WRITE** key.
4) The operation above initializes the tool life control table.

4. Programming

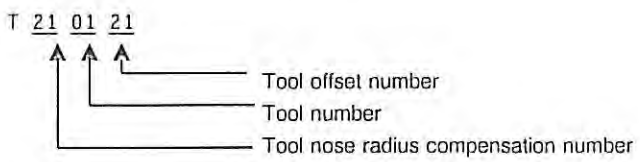
To specify a tool group in a program, program as follows:



Tool group No. . . . . 1 to 12
Tool offset group No. . . . . 1 to 3

Commands TG = 1 and OG = 2, for example, specify Tool No. 1 and Tool Offset No. 21 (T0121).

Note: For the tool nose radius compensation number, the number identical to the specified tool offset number is selected.



To perform automatic spare tool selection after judging tool service life:

Program "TLID" in the first block of the program or in the block preceding the one containing M02. See below:

Each time "TLID" is executed, the number of the machined workpieces and the cutting time are accumulated. When the tool life has been reached, a tool life flag is built.

When the TG command in the program is executed, the newest tool in the group with tool flag "0" is indexed.

```
G13                    G14
N001  TLID             N100  TLID
N002 ...               N101  ...
   :                      :
   :                      :
or
                       G14
G13                       :
   :                      :
   :                   N197  M100
N097  M100             M198  TLID
N098  TLID             M199  M02
N099  M02
```

Note: *When no TLID command is designated, automatic spare tool selection by the tool life management function is not performed.*

When programming TLID in the block preceding the one containing M02, and when it is necessary to synchronize the execution of TLID on turret A and B, program M100 in the blocks preceding the ones containing TLID.

5. Processing After All Spare Tools Have Been Used Up

When all the spare tools are used up, either alarm B occurs to stop the machine or cycle stop by the external cycle stop signal is activated. Note that the cycle stop function using the external cycle stop signal is optional.

The operator can select which mode - alarm stop or cycle stop - is to be activated by entering a proper parameter data: Optional parameter (bit) No. 4.

```
No.
 :
 :
 4     0 0 0 0 0 0 0 0
         ↑
         └──────────── 0:  Cycle stop
                       1:  Alarm stop (Alarm B)
```

Note: *The cycle stop is activated by the signal output from the external coupled device such as a loader. Therefore, cycle stop is not effective when the machine is operated independently without using external coupled device even when bit data "0" is set.*

The cycle stop function using the external cycle stop signal is optional.

6. Alarm Messages

ALARM-B

525 Tool life control: spare tool none

There is no spare tool in the commanded tool group.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the tool group having no spare tool

526 Tool life control: tool group

Numerical value of tool group command TG is: TG < 1 or TG > 12

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the designated TG

527 Tool life control: no T-entry

Tools are not registered in the designated tool group.

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF

528 Tool life control: tool offset group

Illegal tool offset group number is designated.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the designated tool offset group number

529 Tool life control: no T-offset

Tool offset number is not registered for the designated tool offset group.

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF

SECTION 19 PHASE MATCHING FUNCTION FOR THREAD CUTTING OPERATIONS

1. Overview

To finish thread already roughly cut on other machine, it becomes necessary to match the phase of thread. This function makes this phase matching process easier.

2. M Codes

M193 . . . . . Cancels M194 (thread phase matching function).

This M code is always used with M194 as a pair and must be specified at the end of thread cutting cycle called out by M194.

It must be specified in a block without other commands.

M194 . . . . . When this command is specified, phase mismatch amount at the thread cutting starting point is calculated and necessary offset is execute at the start and end points of thread cutting.

This M code is specified in the same block containing the G code calling out a thread cutting fixed cycle.

G31, G33 (Fixed thread cutting cycle: Longitudinal)
G32 (Fixed thread cutting cycle: End face (Traverse))
G71 (Compound fixed thread cutting cycle: Longitudinal)
G72 (Compound fixed thread cutting cycle: Traverse)

M195 . . . . . Cancels M196 (command to obtain phase mismatch amount in manual operation).

This M code is always used with M196 as a pair and must be specified after the completion of phase matching by mid-auto manual operation.

It must be specified in a block without other commands.

M196 . . . . . When this command is specified, manually fed axis movement amount to match thread cutting phase is stored in memory.

This mode is effective until M195 is specified or the control is reset.

The code must be specified in a block without other commands, preceding the one containing M00.

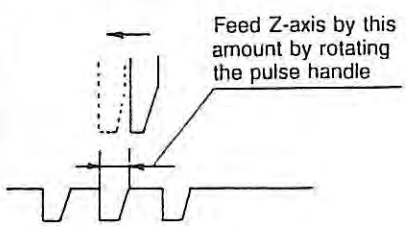
M197 . . . . . Cancels the data stored as thread cutting phase matching amount.

Since the data stored as phase matching amount is not cleared by the NC reset operation, it is necessary to specify this command to clear the stored data.

It must be specified in a block without other commands.

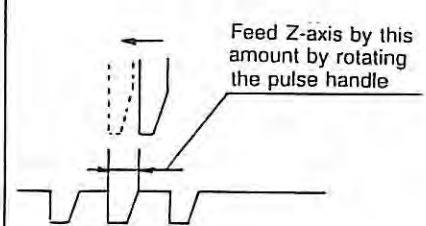
3. Operations

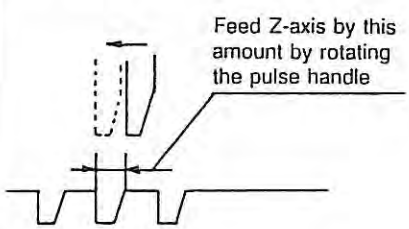
The operation procedure to match thread cutting phase is explained below taking G33 (longitudinal thread cutting fixed cycle) as an example.

Operations	Program
1) Carry out spindle orientation at the 0° position. In this operation, setting of zero point for the spindle orientation (parameter (word) No. 12) may not be zero.	1) M19
2) Bring the thread cutting tool at a rapid feedrate near the thread portion with which the phase must be matched. In this operation, Z command value should be a multiple of thread lead from the thread cutting starting point.	2) G00 X90 Z80 T101
3) Program the M code to store axis movement amount in the mid-auto manual operation by which the axis is fed to match the phase for thread cutting.	3) M196
4) Stop the program execution.	4) M00
5) After the execution of the program has been stopped, press the **MID-AUTO MANUAL** pushbutton switch to allow manual operations. Feed Z-axis with the pulse handle to align the cutting tool with the groove of the thread already cut. The axis feed amount in this operation is stored at system variables VTHRX or VTHRZ as the thread cutting phase matching amount when the sequence restart operation is carried out.	5) Manual operation 
6) After the cutting tool edge and the thread groove are correctly aligned, press the **SEQ. RESTART** pushbutton switch to return the axes to the point where manual operation has been interrupted. Then press the **CYCLE START** pushbutton switch.	6) Manual operation
7) Cancel the M196 mode after the manually fed axis movement amount has been stored as the thread cutting phase matching amount (the data is stored when the **SEQ. RESTART** pushbutton switch is pressed). Do not forget to specify this M code. Otherwise, manually operated axis movement, if attempted again during the thread cutting fixed cycle, is stored in memory again and thus phase matching cannot be executed correctly.	7) M195
8) Feed the thread cutting tool up to the thread cutting starting point at a rapid feedrate.	8) G00 X100 Z130

Operations	Program
9) If phase matching amount is calculated by M194 before the spindle reaches the fixed speed, correct phase matching amount cannot be calculated. Therefore, buffer reading of the next block of commands is temporarily delayed by specifying M100 until the spindle reaches the fixed speed.	9) M100
10) Start the thread cutting fixed cycle. Specify M194 in the block containing G33. This executes the phase matching in the thread cutting operation.	10) G33 X88.8 Z40F10 M194 X88.6 X88.4 X88.2 X88
11) After the completion of thread cutting fixed cycle, thread cutting phase matching should be canceled by specifying M193. Note that M193 is always used with M194 in a pair to cancel the M194 mode.	11) M193
12) Proceed to the next process.	12) G00 X200 Z200

When a workpiece on which phase matching is required at two places, follow the steps below:

Operations	Program
1) Carry out spindle orientation at 0° position. In this operation, setting of zero point for the spindle orientation (parameter (word) No. 12) may not be zero.	1) M19
2) Bring the thread cutting tool at a rapid feedrate near the first thread portion with which the phase must be matched. In this operation, Z command value should be a multiple of thread lead from the thread cutting starting point.	2) G00 X90 Z100 T101
3) Program the M code to store axis movement amount in the mid-auto manual operation by which the axis is fed to match the phase for thread cutting.	3) M196
4) Stop the program execution.	4) M00
5) After the execution of the program has been stopped, press the **MID-AUTO MANUAL** pushbutton switch to allow manual operations. Feed Z-axis with the pulse handle to align the cutting tool with the groove of the thread already cut. The axis feed amount in this operation is stored at system variables VTHRX and VTHRZ as the thread cutting phase matching amount when the sequence restart operation is carried out.	5) Manual operation 

Operations	Program
6) After the cutting tool edge and the thread groove are correctly aligned, press the **SEQ. RESTART** pushbutton switch to return the axes to the point where manual operation has been interrupted. Then press the **CYCLE START** pushbutton switch.	6) Manual operation
7) Save the axis movement amount at the first thread portion to common variable (V1).	7) V1 = VTHRZ
8) Bring the thread cutting tool at a rapid feedrate near the second thread portion with which the phase must be matched. In this operation, Z command value should be a multiple of thread lead from the thread cutting starting point.	8) G00 X90 Z30
9) Stop the program execution.	9) M00
10) After the execution of the program has been stopped, press the **MID-AUTO MANUAL** pushbutton switch to allow manual operations. Feed Z-axis with the pulse handle to align the cutting tool with the groove of the thread already cut. The axis feed amount in this operation is stored at system variables VTHRX and VTHRZ as the thread cutting phase matching amount when the sequence restart operation is carried out.	10) Manual operation 
11) After the cutting tool edge and the thread groove are correctly aligned, press the **SEQ. RESTART** pushbutton switch to return the axes to the point where manual operation has been interrupted. Then press the **CYCLE START** pushbutton switch.	11) Manual operation
12) Save the axis movement amount at the second thread portion to common variable (V2).	12) V2 = VTHRZ
13) Cancel the M196 mode after the manually fed axis movement amount has been stored as the thread cutting phase matching amount (the data is stored when the **SEQ. RESTART** pushbutton switch is pressed). Do not forget to specify this M code. Otherwise, manually operated axis movement, if attempted again during the thread cutting fixed cycle, is stored in memory again and thus phase matching cannot be executed correctly.	13) M195
14) Feed the thread cutting tool up to the first thread cutting starting point at a rapid feedrate.	14) G00 X100 Z130

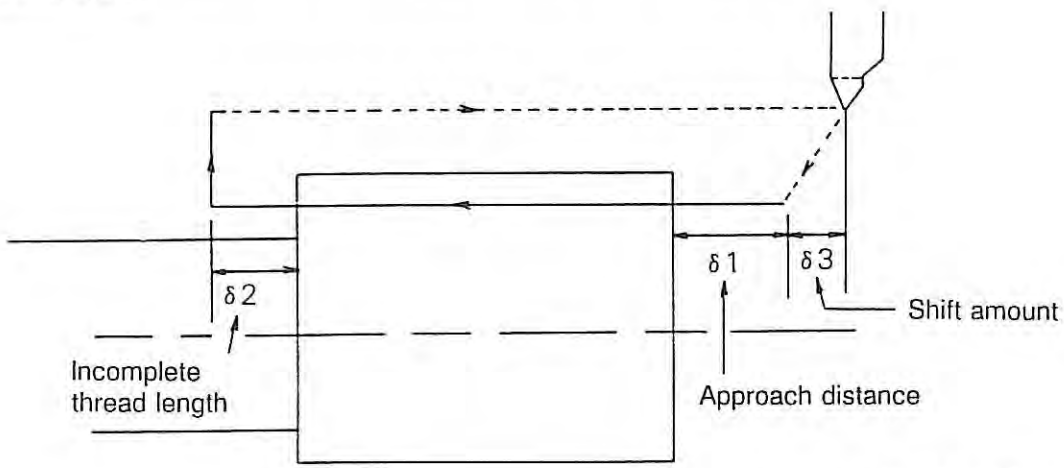
Operations	Program
15) If phase matching amount is calculated by M194 before the spindle reaches the fixed speed, correct phase matching amount cannot be calculated. Therefore, buffer reading of the next block of command is temporarily delayed by specifying M100 until the spindle reaches the fixed speed.	15) M100
16) Call out the axis feed amount at the first thread cutting portion saved at V1.	16) VTHRZ = V1
17) Start the thread cutting fixed cycle. Specify M194 in the block containing G33. This executes the phase matching in the thread cutting operation.	17) G33 X88.8 Z60 F10 M194 X88.6 X88.4 X88.2 X88
18) After the completion of thread cutting fixed cycle, thread cutting phase matching should be cancelled by specifying M193. Note that M193 is always used with M194 in a pair to cancel the M194 mode.	18) M193
19) Proceed to the next process.	19) G00 X200 Z200
20) Feed the thread cutting tool up to the second thread cutting starting point at a rapid feedrate.	20) G00 X100 Z50
21) If phase matching amount is calculated by M194 before the spindle reaches the fixed speed, correct phase matching amount cannot be calculated. Therefore, buffer reading of the next block of commands is temporarily delayed by specifying M100 until the spindle reaches the fixed speed.	21) M100
22) Call out the axis feed amount at the second thread cutting portion saved at V2.	22) VTHRZ = V2
23) Start the thread cutting fixed cycle. Specify M194 in the block containing G33. This executes the phase matching in the thread cutting operation.	23) G33 X88.8 Z0 F10 M194 X88.6 X88.4 X88.2 X88
24) After the completion of thread cutting fixed cycle, thread cutting phase matching should be cancelled by specifying M193. Note that M193 is always used with M194 in a pair to cancel the M194 mode.	24) M193
25) Proceed to the next process.	25) G00 X200 Z200

4. Precautions

(1) Always specify M193 at the end of the thread cutting fixed cycle carried out in the M194 mode. If the thread cutting cycle is executed next without specifying M193, the phase matching amount in the previous thread cutting cycle is called out and phase matching by this amount is carried out.

(2) Sequence restart from the sequence in the thread cutting fixed cycle and mid-auto manual operation in the thread cutting fixed cycle are possible. However, sequence restart from the block containing M194 is not possible. If this is required, carry out sequence restart from the preceding block containing M100.

(3) Allow margin before and after the thread cutting start and end points when programming thread cutting cycle. If thread cutting is programmed without approach distance ($\delta 1$) and incomplete thread portion ($\delta 2$), thread cannot be finished in correct pitches. Furthermore, the phase matching during thread cutting shifts the thread cutting starting point by the calculated amount. Therefore, such amount must also be taken into consideration for programming thread cutting cycle.



N: Spindle speed
P: Thread pitch
KV: Constant specific to each individual model (Refer to Section 3.1, "Programming Special Threads" in the Programming Manual.)

$$\delta_1 > \frac{N \times P}{KV \times 20}$$

$$\delta_2 > \frac{N \times P}{KV \times 60}$$

$$\delta_3 > (1.312 * 10^{-3}) \times N \times P + P$$

(4) This function is also usable for taper thread cutting in G31, G32, G33, G71 and G72 modes.

(5) When cutting threads at two different position in a single workpiece using the G32 (face thread cutting fixed cycle) or G72 (face thread cutting compound fixed cycle), change the operation procedure as indicated below.

(In case of G31, G33, G71)		(In case of G32, G72)
7) V1 = VTHRZ	→	V1 = VTHRX
12) V2 = VTHRZ	(Change) →	V2 = VTHRX
16) VTHRZ = V1	→	VTHRX = V1
22) VTHRZ = V2	→	VTHRX = V2

(6) Setting for VTHRX and VTHRZ are cleared when the power is turned off.

SECTION 20 EXTERNAL PROGRAM SELECTION A (PUSHBUTTON METHOD)

1. Overview

This function selects the main and schedule programs by pressing the button corresponding to the desired program.

2. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule program is selected by the setting of the parameter (bit) data.

Parameter (Bit) No. 6	Main/Schedule Program
Bit 1 = 1	Schedule programs
Bit 1 = 0	Main programs

3. Inhibiting the Selection of the Same Program

By setting a proper parameter (bit) data, it is possible to inhibit the loading of the program selected if the program having been selected and stored in memory is selected again. This can save the program loading time.

Parameter (Bit) No. 6	Loading of the Same Program
Bit 2 = 1	Inhibited
Bit 2 = 0	Allowed

4. Program Selection

Selection of a desired program is possible by pressing the pushbutton switch corresponding to that program.

Selection of a program from P1.MIN (or SDF) to P16.MIN (or SDF) is possible. (P0.MIN cannot be selected.)

Note that the number of selectable programs differs depending on the specification.

When a program is selected, a lamp corresponding to the pressed button goes on, indicating that the program is selected.

5. Precautions

(1) Program selection is possible only in the automatic mode.

(2) While a program is being selected, pressing another program selection button is ignored.

(3) While the control is in the alarm state, program selection is not carried out.

(4) If the program designated for selection is not found, an alarm occurs.

(5) An attempt to select a program while the program having been selected is executed causes an alarm.

6. Alarm Messages

ALARM-C

944 Program select

An attempt to select a program in the following cases.

a) Program selected is not found:

Error message "**Not found main program file error 1**" or "**Not found schedule program file error 1**" is displayed.

b) External program selection without resetting the control after the MDI mode operation:

Error message "**Main program execution error 1 (or 2*)**" is displayed.

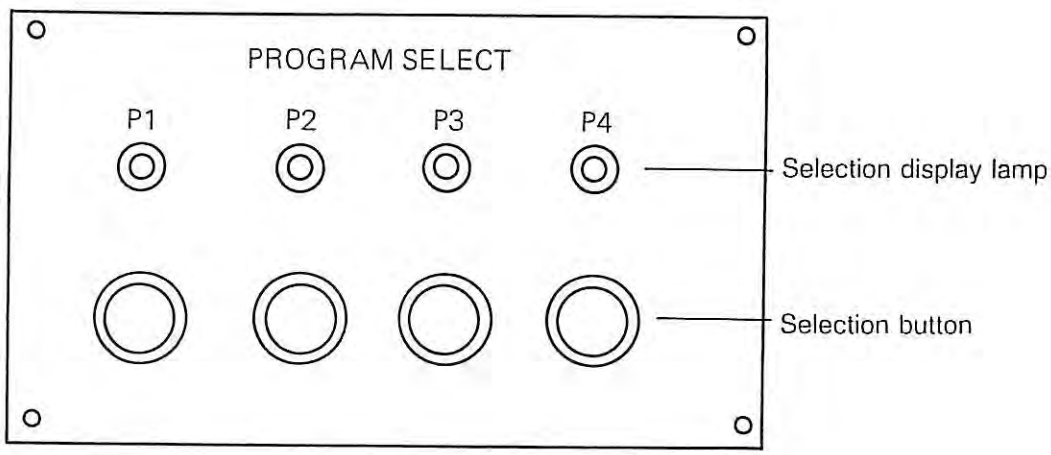
c) External program selection while the program selected is being executed:

Error message "**Main program execution error 1 (or 2*)**" is displayed when machining is carried out using main program, or "**Schedule program execution error 1 (or 2*)**" is displayed when the machining is carried out using the schedule program.

*: 1 when main program is selected and 2 when schedule program is selected.

Index	:	None
Character-string	:	None
Code	:	1

7. Panel Example (When the number of programs is four)



SECTION 21 EXTERNAL PROGRAM SELECTION B (ROTARY SWITCH METHOD)

1. Overview

This function selects the main and schedule programs using the rotary switch and the program selection button.

2. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule programs is selected by the setting of the parameter (bit) data.

Parameter (Bit) No. 6	Main/schedule Program
Bit 1 = 1	Schedule programs
Bit 1 = 0	Main programs

3. Inhibiting the Selection of the Same Program

By setting a proper parameter (bit) data, it is possible to inhibit the loading of the program selected if the program having been selected and stored in memory is selected again. This can save the program loading time.

Parameter (Bit) No. 6	Loading of the Same Program
Bit 2 = 1	Inhibited
Bit 2 = 0	Allowed

4. Program Selection

Selection of a desired program is possible by specifying the desired program using the rotary switch.

Selection of a program from P1.MIN (or SDF) to P16.MIN (or SDF) is possible.

Note that the number of selectable programs differs depending on the specification.

(P0. MIN cannot be selected.)

5. Precautions

(1) Program selection is possible only in the automatic mode.

(2) While a program is being selected, pressing the program selection button is ignored.

(3) While the control is in the alarm state, program selection is not carried out.

(4) If the program designated for selection is not found, an alarm occurs.

(5) An attempt to select a program while the program having been selected is executed causes an alarm.

6. Alarm Messages

ALARM-C

944 Program select

An attempt to select a program in the following cases.

a) Program selected is not found:

Error message **"Not found main program file error 1"** or **"Not found schedule program file error 1"** is displayed.

b) External program selection without resetting the control after the MDI mode operation:

Error message **"Main program execution error 1 (or 2*)"** is displayed.

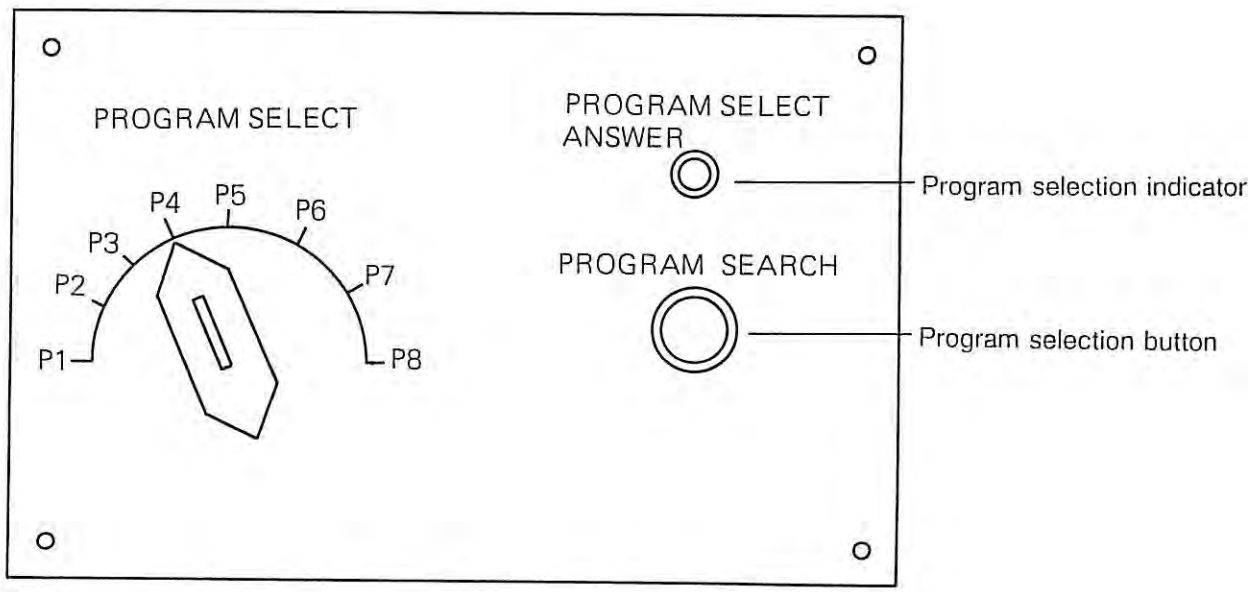
c) External program selection while the program selected is being executed:

Error message **"Main program execution error 1 (or 2*)"** is displayed when machining is carried out using main program, or **"Schedule program execution error 1 (or 2*)"** is displayed when the machining is carried out using the schedule program.

*: 1 when main program is selected and 2 when schedule program is selected.

Index	:	None
Character-string	:	None
Code	:	1

7. Panel Example (When the number of programs is four)



8. Lamp Indication

The following lamp indication is possible.

The SAME PROGRAM lamp lights up if the program number set by the rotary switch and the program number currently selected are the same. Here, the program number currently selected means the program number having been selected by the rotary switch setting and pressing the program selection button. This lamp indication is provided to prevent the operator from carelessly leaving the program selection button unpressed after setting the program number with the rotary switch.

9. Cycle Start Interlock Using Part Program

The input variable function available with user task 2 allows the check of the program to be executed if it is the one selected.

See the example below:

```
N0010 IF [VDIN [2001] NE 1] N0990
N0990 VDOUT [992] = ****
N1000 M02
```

When three blocks of a program above are written at the beginning of the program P1.MIN, command in N0010 checks the program number of the program started, i.e., P1.MIN, against the setting of or input from the rotary switch. If they do not match each other, program jumps to N0990 and message "ALARM-B User reserve code ****" is displayed on the CRT.

*Note: "****" represents a coded number consisting of up to four digits.*

(1) Checking EC Input Rotary Switch Setting by Input Variable

EC input	No. 23 bit 0	VDIN [2001]	P1.MIN
	No. 23 bit 1	VDIN [2002]	P2.MIN
	:		
	:		
	No. 23 bit 7	VDIN [2008]	P8.MIN
	No. 23 bit 0 - 7	VDIN [2009]	
EC input extend	No. 10 bit 0	VDIN [2011]	P9.MIN
	No. 10 bit 1	VDIN [2012]	P10.MIN
	:		
	:		
	No. 10 bit 7	VDIN [2018]	P16.MIN
	No. 10 bit 0 - 7	VDIN [2019]	

(2) Checking Panel Input Rotary Switch Setting by Input Variable

Panel input	No. 13 bit 0	VDIN [2021]	P1.MIN
	No. 13 bit 1	VDIN [2022]	P2.MIN
	:		
	:		
	No. 13 bit 7	VDIN [2028]	P8.MIN
	No. 13 bit 0 - 7	VDIN [2029]	

Examples:

a) For checking P4.MIN with panel input rotary switch:

```
N0010 IF [VDIN [2024] NE 1] N0990 . . . Is rotary switch input "4"?
                    :
                    :
N0990 VDOUT [992] = **** . . . . . ALARM-B User reserve code ****
N1000 M02
```

b) For checking P3.MIN with EC input rotary switch:

```
N0010 IF [VDIN [2003] NE 1] N0990 . . . . . Is rotary switch input "3"?
                    :
                    :
N0990 VDOUT [992] = **** . . . . . ALARM-B User reserve code ****
N1000 M02
```

SECTION 22 EXTERNAL PROGRAM SELECTION C (BCD SYSTEM)

1. Overview of External Program Selection C

This function selects a program in accordance with the program number input as the binary coded decimal number.

2. Types of Selectable Programs

P * * * * .MIN (main program)
P * * * * .SDF (schedule program)

Note: *P0000.MIN cannot be used. Therefore, selectable program range is from P0001.MIN to P9999.MIN.*

3. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule programs is selectable by the setting of the parameter (bit) data.

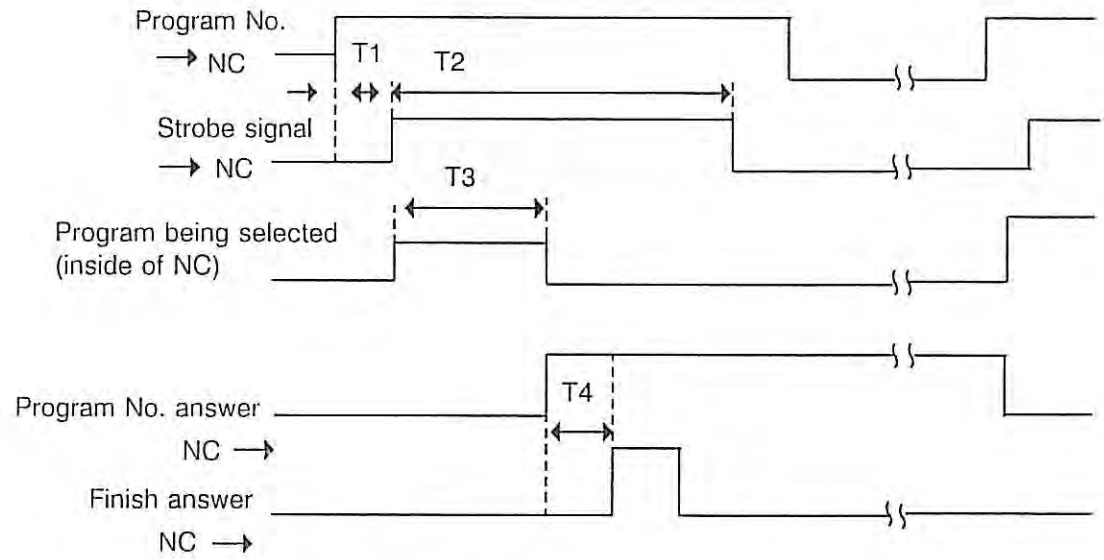
Parameter (Bit) No. 6	Main/schedule Program
Bit 1 = 1	Schedule programs
Bit 1 = 0	Main programs

4. Inhibiting the Selection of the Same Program

By setting proper parameter (bit) data, it is possible to inhibit the loading of the program selected if the program having been selected and stored in memory is selected again. This can save the program loading time.

Parameter (Bit) No. 6	Loading of the Same Program
Bit 2 = 1	Inhibited
Bit 2 = 0	Allowed

Signal Time Chart



T1: 1 msec or longer
T2: 100 msec or longer
T3: 1 to 5 sec
T4: 100 msec

Note 1: *The program number answer signal output from the NC is turned off when the next program selection command is given either as an BCD input external signal or a signal input from the operation panel.*

Note 2: *The program selection completion answer signal output from the NC is turned off when the strobe signal input to the NC is turned off. If the strobe signal is off when the NC outputs the completion answer signal, it remains ON for 25 msec.*

5. Precautions

(1) Program selection is possible only in the automatic mode.

(2) While a program is being selected, pressing the program selection button is ignored.

(3) While the control is in the alarm state, program selection is not carried out.

(4) If the program designated for selection is not found, an alarm occurs.

(5) An attempt to select a program while the program having been selected is executed causes an alarm.

6. Alarm Messages

ALARM-C

944 Program select

An attempt to select a program in the following cases.

a) Program selected is not found:

Error message **"Not found main program file error 1"** or **"Not found schedule program file error 1"** is displayed.

b) External program selection without resetting the control after the MDI mode operation:

Error message **"Main program execution error 1 (or 2*)"** is displayed.

c) External program selection while the program selected is being executed:

Error message **"Main program execution error 1 (or 2*)"** is displayed when machining is carried out using main program, or **"Schedule program execution error 1 (or 2*)"** is displayed when the machining is carried out using the schedule program.

*: 1 when main program is selected and 2 when schedule program is selected.

Index	:	None
Character-string	:	None
Code	:	1

SECTION 23 CYCLE TIME OVER CHECK FUNCTION

1. Overview

Duration of cycle time (beginning with the cycle start and ending with the execution of M02 or M30) is measured and if the measured time exceeds the parameter set cycle time, an alarm occurs.

2. Check ON Conditions

When a "1" is set for optional parameter bit No. 11, bit 1, the check function is made effective.

3. Cycle Time Setting

Set the cycle time for optional parameter (word) No. 27. Setting is made in units of seconds.

Maximum setting : 14,400 seconds

4. Alarm

If the measured cycle time exceeds the parameter set time, alarm C occurs and the following message is displayed on the screen.

ALARM-C 950 Cycle time over

SECTION 24 STM TIME OVER CHECK FUNCTION

1. Overview

Duration of S, T, M cycle time is measured and if the measured time exceeds the parameter set cycle time, an alarm occurs.

2. Check ON Conditions

When a "1" is set for optional parameter bit No. 11, bit 0, the check function is made effective.

When a "0" is set for optional parameter bit No. 11, bit 0, the check function is made ineffective.

3. S, T, M Cycle Time Setting

Set the S, T, M cycle time for optional parameter (word) No. 26. Setting is made in units of 0.1 seconds.

Maximum setting : 600 seconds

4. Alarm

If the measured S, T, M cycle time exceeds the parameter set time, alarm B occurs and the following message is displayed on the screen.

ALARM-B 574 STM time over

SECTION 25 CHUCK OPEN/CLOSE BY M CODES

1. M Codes Used for Chuck Open/Close

M83 . . . . . Chuck close
M84 . . . . . Chuck open

The M code status selected is not influenced by power turning on to or resetting the control.

For the two-saddle specification models, M83/M84 can be specified at either of G13 or G14.

2. Alarm Messages

ALARM-A

117 Chuck clamp

While the chuck is opening or closing, ID/OD chucking mode is switched, or insufficient chucking pressure state has continued for 2 seconds.

Index : None

Character-string : None

Code :

100 . . . . . Low chucking pressure state continued for 2 seconds (EC input No. 1, bit 6 = 0).

None . . . . . The OD chucking (Panel input No. 6, bit 0) and the ID chucking (Panel input No. 6, bit 1) signals are both off.

3 . . . . . The OD chucking (Panel input No. 6, bit 0) and the ID chucking (Panel input No. 6, bit 1) signals are both on.

118 Chuck condition illegal

Chuck open/close command (M83/M84) is specified while the spindle or the M-tool spindle is revolving, or an external interlock signal is ON.

This alarm does not occur with the SMW chuck or while the internal interlock release state is active.

Index : TURRET

Character-string : None

Code :

1 ..... The spindle zero speed signal is not ON (EC input No. 9, bit 3 = 0).

2 ..... The spindle is not in the stop state (higher than 5 rpm).

3 ..... The spindle rotation command is given.

4 ..... The external interlock signal is ON (EC input No. 9, bit 7 = 0).

5 ..... The M-tool spindle zero speed signal is not ON (EC input No. 22, bit 4 = 0).

6 ..... The M-tool spindle is not in the stop state.

7 ..... The M-tool spindle rotation command is given.

119 Chuck open

The spindle or the M-tool spindle rotation command is given while the chuck is not closed, or the chuck close state is not confirmed while the spindle or the M-tool spindle is revolving. Note that this error is ignored when the internal interlock release signal is ON.

Index : TURRET or none

Character-string : None

Code :

1 ..... The spindle rotation command is given.

2 ..... The chuck close state is not confirmed while the spindle is revolving.

3 ..... The M-tool spindle rotation command is given.

4 ..... The chuck close state is not confirmed while the M-tool spindle is revolving.

SECTION 26 CANCELLATION OF CHUCK OPEN/CLOSE INTERLOCK BY M CODES

1. Overview

While the spindle is rotating, chuck open/close is interlocked so that it will not be opened or closed during spindle rotation to assure safe operation. This interlock may be canceled by programming a proper M code.

2. M Codes Used for Canceling Chuck Open/Close Interlock

M184 . . . . . Chuck open/close interlock ON
M185 . . . . . Chuck open/close interlock OFF

When power supply to the control is turned on or when the control is reset, the control is in the M184 mode.

These M codes are effective only in the AUTO and MDI modes.

These M codes are effective when the front door is closed and the **AUTO ON/OFF** switch is set in the ON position (only for the automatic operation with automating equipment connected).

Program Example:

```
  :
  :
N0100  G00  X1000  Z1000
N0101  M185                ; Interlock OFF
N0102  M84                 ; Chuck open
  :
  :
N0110  M83                 ; Chuck close
N0111  M184                ; Interlock ON
  :
  :
```

Note 1: *When the door is not closed during chuck open/close interlock release operation, or when the **AUTO ON/OFF** switch is set in the OFF position (only for the automatic operation with automating equipment connected), the chuck open/close interlock becomes effective.*

Note 2: *When the front door is not closed or when the **AUTO ON/OFF** switch is set in the OFF position, the following alarm occurs if M185 is designated.*

ALARM-A

362 Chuck interlock cancel condition

M185 is designated when the front door is not closed or when the **AUTO ON/OFF** switch is set in the OFF position.

Index	:	TURRET
Character-string	:	None
Code	:	1

SECTION 27 CHUCK HIGH/LOW PRESSURE SELECTION BY M CODES

1. Overview

There are cases where chucking pressure is to be changed by designated commands meeting workpiece types. This function is used to change chucking pressure between high and low levels by designated M codes.

2. M Codes Used for Chuck High/Low Pressure Selection

M58 . . . . . Chucking pressure low
M59 . . . . . Chucking pressure high

Selected mode is not influenced by turning on the control or by resetting it.

For the two-saddle models, the M codes above may be designated at either G13 or G14 mode.

SECTION 28 INDEX CHUCK

1. M Codes Used for Indexing the Index Chuck

M54 . . . . . Index chuck index command

For the two-saddle models, the M codes above may be designated at either G13 or G14 mode.

2. Alarm Messages

ALARM-A

148 Index chuck

The signal from the index chuck indexing completion confirmation limit switch (Bit 7 of EC input #11) is not in during function generation of the NC.

Index	:	None
Character-string	:	None
Code	:	1

SECTION 29 CHUCKING ERROR DETECTION

1. Overview

In the automated operation including a loader, if a workpiece is machined while it is not clamped in chuck correctly, it will constitute hazardous conditions both to operators and the machine. To prevent such dangerous condition, the air is blown from the chuck face where a workpiece has been brought into contact. Chucking error is detected as the leakage of the air.

2. M Code Used for Detecting Chucking Error

M133 . . . . . Chucking error detection ON
M132 . . . . . Chucking error detection OFF
M131 . . . . . Chucking error air ON
M130 . . . . . Chucking error air OFF

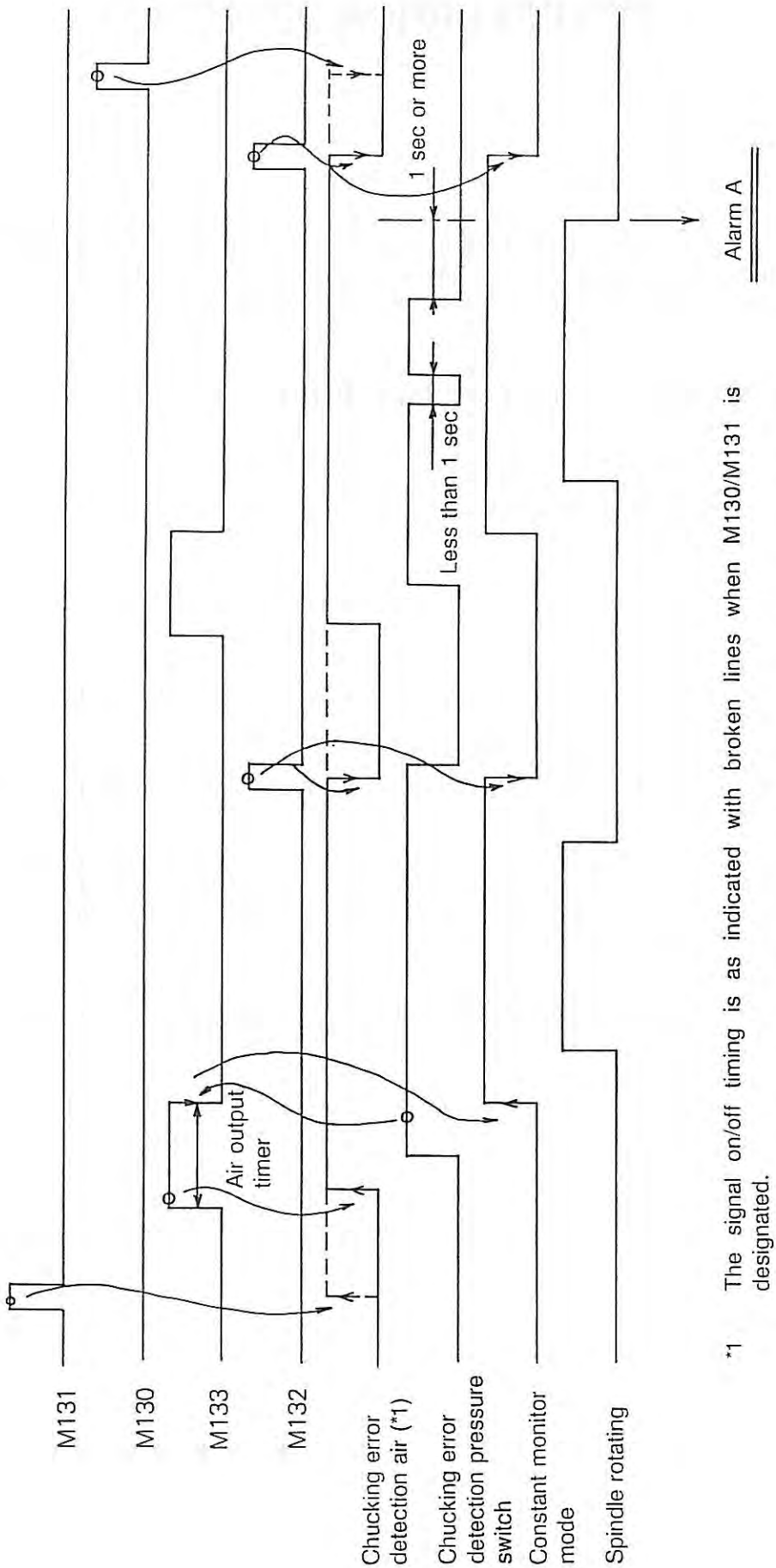
3. Functions

Correct workpiece seating when it is clamped in the chuck and workpiece offsetting during machining can be monitored.

When the M133 code is designated while the chucking error detection ON switch is set in the ON position, air is supplied from the chuck face. The workpiece is determined to have been set in the chuck correctly when there is no air leak.

While the spindle is rotating, offsetting of the workpiece in the chuck can be checked by monitoring the pressure switch. This monitoring mode is canceled by designating the M132 code. In the manual operation mode, air is automatically supplied when the chuck is closed to check correct setting or offsetting of a workpiece in the chuck.

The M131 and M130 codes are used to only turn on and off the supply of air for chucking error detection and, thus, can be used to blow foreign matter to prevent blocking of the discharge port.



Timing Chart Example in Automatic Operation

*1 The signal on/off timing is as indicated with broken lines when M130/M131 is designated.

4. Parameter Setting

The duration in which the air is blown is set by the Parameter (word) No. 19. The setting is possible up to 600 seconds in units of 0.1 seconds. Loading of the NC control software clears the setting to 30 seconds (setting: 30).

5. Alarm Message

ALARM-A

210 Chucking mistaken

The air pressure switch has been off for more than 0.5 sec while the spindle is rotating. Workpiece comes off the chuck.

ALARM-B

535 Chucking error

The air pressure switch is not actuated when the M133 chucking error detection ON command is executed.

Index	:	None
Character-string	:	None
Code	:	1

677 Chucking miss air OFF command enabled

The chucking error detection air pressure turning off command (M130) is designated in the constant chucking error monitoring mode.

SECTION 30 TAILSTOCK SPINDLE ADVANCE/RETRACT BY M CODES (WITH CONFIRMATION LIMIT SWITCHES)

1. M Codes Used for Chuck Open/Close

M55 . . . . . Tailstock spindle retract
M56 . . . . . Tailstock spindle advance

The M code status selected is not influenced by power turning on to or resetting the control.

For the two-saddle specification models, M55/M56 can be specified at either of G13 or G14.

Time duration after the input of the in-position signal after the execution of the tailstock spindle advance command (M56) to the completion of the tailstock spindle advance command can be set with parameter (word) data No. 14 in increments of 0.1 sec. When the NC control software is loaded, it is set at 0.8 seconds.

2. Alarm Messages

ALARM-A

138 Tailstock spindle advance answer

The in-position answer signal is not output within 5 seconds after the tailstock spindle advance command was given.

Index : None
Character-string : None
Code : 2

139 Tailstock spindle over advance

The tailstock spindle advanced exceeding the advance end during the execution of a program.
(EC input No. 6, bit 3 = 1, bit 4 = 0, bit 5 = 0)

Index : None
Character-string : None
Code : 1

140 Tailstock spindle condition illegal

The spindle rotation command is given when the tailstock spindle is not in-position with the setting of the switch on the operation panel in the CENTER WORK, or when the tailstock spindle is not in the retraction position with the switch setting in the CHUCK WORK. Or during the chuck work operation, the tailstock spindle retraction end position condition is lost.

Or the tailstock advance/retraction (M55, M56) command is specified in the chuck work setting. M55 or M56 is specified while the spindle is revolving or it is specified while the tailstock is other than the disconnect state (programmable tailstock specification).

Note that the alarm does not occur with the machine other than the center work specification, or when the interlock is released with M157 specified.

Index : TURRET or none

Character-string : None

Code :

1 ..... The spindle rotation command is given. The tailstock spindle advance/retraction M code is specified when the machine is in the chuck work setting.

2 ..... The tailstock spindle position has shifted. The tailstock spindle advance/retraction M code is specified while the spindle is revolving.

ALARM-B

464 Tailstock spindle over advance

The tailstock spindle advanced exceeding the advance end.
(EC input No. 6, bit 3 = 1, bit 4 = 0, bit 5 = 0)

Index : None

Character-string : None

Code : 1

SECTION 31 TAILSTOCK SPINDLE HIGH/LOW THRUST SELECTION BY M CODES

1. Overview

There are cases where tailstock spindle thrust is required to be changed by designated commands meeting workpiece types. This function is used for changing tailstock spindle thrust between high and low levels by designated M codes.

2. M Codes Used for Selecting High/Low Tailstock Spindle Thrust

M98 . . . . . Tailstock spindle thrust low
M99 . . . . . Tailstock spindle thrust high

Selected mode is not influenced by turning on the control or by resetting it.

For the two-saddle models, the M codes above may be designated at either G13 or G14 mode.

SECTION 32 CANCELLATION OF CENTER WORK INTERLOCK BY M CODES

1. Overview

The machine is provided with the interlock function to prevent spindle rotation unless the tailstock spindle is at the predetermined position (in-position) for center work operation or the tailstock spindle is at the retraction end for chuck work operation. This function cancels this interlock function by designating a proper M command to allow the spindle rotation independently of the tailstock position.

2. M Codes Used for Canceling Center Work Interlock

M156 . . . . . Center work interlock ON

M157 . . . . . Center work interlock OFF

When power supply to the control is turned on or when the control is reset, the control is in the M156 mode.

These M codes are effective only in the AUTO and MDI modes.

3. Program Example

```
  :
  :
N0010  G00  X500  Z500  M05
N0011  M55                    ;  Tailstock spindle retract
N0012  M157                   ;  Center work interlock OFF
N0013  M03
  :
  :
N0020  M05
N0021  M156                   ;  Center work interlock ON
N0022  M56                    ;  Tailstock spindle advance
N0023  G00  X50  Z50  M03
  :
  :
```

SECTION 33 TAILSTOCK SPINDLE ADVANCE/RETRACT DURING SPINDLE ROTATION BY M CODES

1. Overview of Tailstock Spindle Advance/Retraction during Spindle Rotation

When the **CENTER WORK/CHUCK WORK** selector switch on the operation panel is placed in the CENTER WORK position, an interlock function is activated to prevent tailstock spindle advance/retraction operation during the spindle rotation thus assuring the safety. This function cancels the interlock function to allow the tailstock spindle to advance or retract even while the spindle is rotating.

2. M Codes Used for Advancing and Retracting Tailstock Spindle

M166 . . . . . Tailstock spindle advance/retract interlock during spindle rotation ON
M167 . . . . . Tailstock spindle advance/retract interlock during spindle rotation OFF

When power supply to the control is turned on or when the control is reset, the control is in the M166 mode.

These M codes are effective only in the AUTO and MDI modes.

3. Program Example

```
  :
  :
N0100  G00   X500  Z500  M03
N0101  M167  M157            ;  Interlock OFF
N0102  M55                   ;  Tailstock spindle retract
  :
  :
N0110  M56                   ;  Tailstock spindle advance
N0111  M166  156             ;  Interlock ON
  :
  :
```

Note: When the tailstock advance/retract interlock function is canceled by designating M167, also cancel the center work interlock by designating M157.

SECTION 34 WORK REST CLAMP/UNCLAMP AND ADVANCE/RETRACT BY M CODES

1. M Codes Used for Clamping/Unclamping and Advancing/ Retracting Work Reset

M122 . . . . . Work rest retract
M123 . . . . . Work rest advance
M78 . . . . . Work rest unclamp
M79 . . . . . Work rest clamp

Selected mode is not influenced by turning on the control or by resetting it.

For the two-saddle models, the M codes above may be designated at either G13 or G14 mode.

Note: *The M122/123 codes cannot be designated when the work rest has no advance/retract function.*

SECTION 35 AUTOMATIC FRONT DOOR OPEN/CLOSE BY M CODES

1. Overview

Front door of the lathe is automatically opened or closed by M codes.

2. M Codes Used for Opening/Closing the Front Door

M90 . . . . . Cover close
M91 . . . . . Cover open

Selected mode is not influenced by turning on the control or by resetting it.

For the two-saddle models, the M codes above may be designated at either G13 or G14 mode.

SECTION 36 PARTS CATCHER ADVANCING/ RETRACTING BY M CODES

1. M Codes Used for Advancing/Retracting Parts Catcher

M76 . . . . . Parts catcher retract
M77 . . . . . Parts catcher advance

Selected mode is not influenced by turning on the control or by resetting it.

For the two-saddle models, the M codes above may be designated at either G13 or G14 mode.

2. Interlock

(1) Parts catcher advance/retract is possible only when the turret is located at the Z-axis travel end.

The turret can not be moved in the Z-axis direction when the parts catcher is advancing.

For two-saddle models, parts catcher advance/retract is possible only when the B-turret (G14) is located at the Z-axis travel end.

(2) Parts catcher advance/retract is not possible when the touch setter arm is not at the retract end. (for touch setter A/M specification)

(3) Parts catcher

Parts catcher advance/retract is not possible when the door is not closed. (for LB9, LB25, LR15 and LR25)

* For the 3rd turret interlock, refer to **Section 16**, "3rd Turret" of Special Functions Manual (No. 3).

3. Alarm

ALARM-A

274 Parts catcher interlock

Parts catcher advance/retract command is designated when parts catcher advance/retract conditions are not satisfied.

Index	:	None	
Character-string	:	None	
Code	:	2	Touch setter arm is not at the retract end.
		3	Door is not closed.
		4	Turret is not at the Z-axis travel end.

SECTION 37 BAR FEEDER INTERLOCK FUNCTION

1. M Codes Used for Operating Bar Feeder

M93 . . . . . Bar feeder start

There is an **AUTO CYCLE/INDEPENDENT** selector switch on the operation panel and selection of the INDEPENDENT position ignores code M93. The NC machine is independently controlled in this case.

In the AUTO CYCLE mode, M93 is processed as an effective command and bar feeder operation interlocked with the machining cycle is performed.

2. Top-Cut

When a new bar is loaded on the machine, the length and the end shape of such material are not uniform and vary from material to material. Therefore, it is necessary to cut off the uneven end face of the loaded bar before starting the first cutting cycle. This cutting off cycle is called "Top-Cut".

The program for top-cut cycle is provided preceding the machining cycle and is executed when the top-cut cycle initiation signal is derived from the bar feeder. Unless such a signal is presented, the top-cut is ignored.

3. Example of Program

```
N001  IF  VTOPC  NTOP
N002  GOTO  NCUT
NTOP  M84  ┐
N003  M93  ├---------- Top-cut program
  :        │
  :        ┘
NCUT  M84  ┐
N100  M93  ├---------- Machining program
  :        │
  :
```

* Program for top-cut cycle is executed when top-cut signal is present.
* Signal from the bar feeder ON . . . . . Machining program is executed.
  Signal from the bar feeder OFF . . . . . Top-cut program is executed.

4. Alarm Messages

ALARM-A

150 Coupling device illegal

ALARM-C

969 Coupling device illegal

An abnormality was detected with the bar feeder.

Index : None

Character-string : None

Code : None

The alarm level can be switched between Alarm A and Alarm C by parameter setting.

Bit 2 of parameter bit No. 8
- 0 → Alarm A
- 1 → Alarm C

Standard: 0 → Alarm A

In addition, whether the alarm occurrence is checked "always" or only in "AUTO CYCLE mode" can be determined by parameter setting.

Bit 5 of parameter (bit) No. 6
- 0 → Always monitored
- 1 → Only in AUTO CYCLE mode

Standard: 0 → Always monitored

5. Automatic Continuous Operation

Automatic continuous operation including the bar feeder is performed in accordance with a schedule program.

Provided below is an example of a program:

```
Schedule file name  . . . .  BFD.SDF

N001   PSELECT A.MIN,,,Q100
N002   END
```

Q100 → No. of cycle repetitions

With the program above, the machining program assigned with the file name A.MIN is repeated 100 times. After the designated file has been repeated 100 times, machine cycle stops.

To terminate the cycle using the preset counter, enter 9999 for Q command. This allows the cycle to be repeated as many times as set on the preset counter.

For operating the machine, select the schedule program.

SECTION 38 SPARE AIR BLOWER FUNCTION

1. M Codes Used for Turning ON/OFF Spare Air Blow

M50 . . . . . Spare air blower function OFF
M51 . . . . . Spare air blower function ON

Once activated, ON or OFF of the air blower function remains effective until another M code cancels the presently selected ON or OFF state.

When the control is reset or the power is turned ON, the control is placed in the M50 state.

Spindle jog operation may be executed while the spare air blower is on by setting proper parameter data.

Parameter (Bit) No. 19 (No. 44)	Spindle Jogging
Bit 1 = 0	Not executed
Bit 1 = 1	Executed

( ): Sub spindle (2nd spindle) jog operation

The spindle jog is stopped when the M50 command is executed.

2. M Codes Used for Turning ON/OFF Spare Air Blow

M154 . . . . . Spare air blower function OFF
M155 . . . . . Spare air blower function ON

Once activated, ON or OFF of the air blower function remains effective until neither M code cancels the presently selected ON or OFF state.

When the control is reset or the power is turned ON, the control is placed in M154 state.

Spindle jog operation may be executed while the spare air blower is on by setting proper parameter data.

Parameter (Bit) No. 19 (No. 44)	Spindle Jogging
Bit 2 = 0	Not executed
Bit 2 = 1	Executed

( ): Sub spindle (2nd spindle) jog operation

The spindle jog is stopped when the M154 command is executed.

SECTION 39 SPINDLE JOGGING FUNCTION WHEN AIR BLOWER IS COMMANDED

1. Overview

The spindle can be jogged simultaneously with the execution of the air blower command.

2. Operation

Whether or not the spindle jogs simultaneously with the execution of the air blow command is set with the setting for optional parameters (bit).

Optional parameter (bit) No. 19 (No. 44*):

Bit 0 . . . . . Spindle jogging simultaneous with the execution of air blower (M89)
Bit 1 . . . . . Spindle jogging simultaneous with the execution of spare air blower (M51)
Bit 2 . . . . . Spindle jogging simultaneous with the execution of spare air blower (M155)

Setting 1: Spindle jogs.

Setting 0: Spindle does not jog.

* Sub spindle

The above parameter setting is effective for manual air blower operation.

SECTION 40 ROBOT/LOADER INTERFACE FUNCTION

1. Overview

Two different types of robot/loader interface function are available:

Type-B NC master system for controlling robot/load operations using request signals (M signals) output from the NC.

Type-C Robot/loader master system where the robot/loader begins loading/unloading of the workpiece in response to the cycle completion signal and the NC starts machining on receipt of a cycle start signal from the robot/loader.

2. Type-B Robot/Loader Interface

The Type-B robot/loader interface provides an NC master system for controlling robot/loader operations using request signals (M signals) output from the NC.

The Type-B interface features the following functions.

(1) Request outputs from the NC (4 output signals)

(2) Auxiliary operation signal I/O functions

(3) Status signal I/O functions

(4) Door interlock function

2-1. Features

(1) During robot/loader servicing, turret rotation is possible.

(2) Automatic continuous machining possible using the NC schedule program.

(3) Besides the cutting program, the machining program also contains the robot/loader request M signal.

(4) Since the four robot/loader request signals are independent, various NC operations including tool gauging and axis feed can be conducted between requests using the program.

2-2. Functions

(1) Request output from the NC

NC request signals are output using the M signals listed below.

M180 . . . . . Request 1
M181 . . . . . Request 2
M182 . . . . . Request 3
M183 . . . . . Request 4

These M signals are output only in AUTO mode and ignored in the single (AUTO off) mode.

3. Type-C Robot/Loader Interface

With type-C robot/loader interface the robot/loader begins loading/unloading cycle in response to the cycle completion signal. The NC starts machining operations upon the reception of a cycle start signal from the robot/loader.

The type-C interface features the following functions.

(1) Output of cycle start and cycle completion signal from the robot/loader

(2) Auxiliary operation signal I/O functions

(3) Status signal I/O functions

(4) Door interlock function

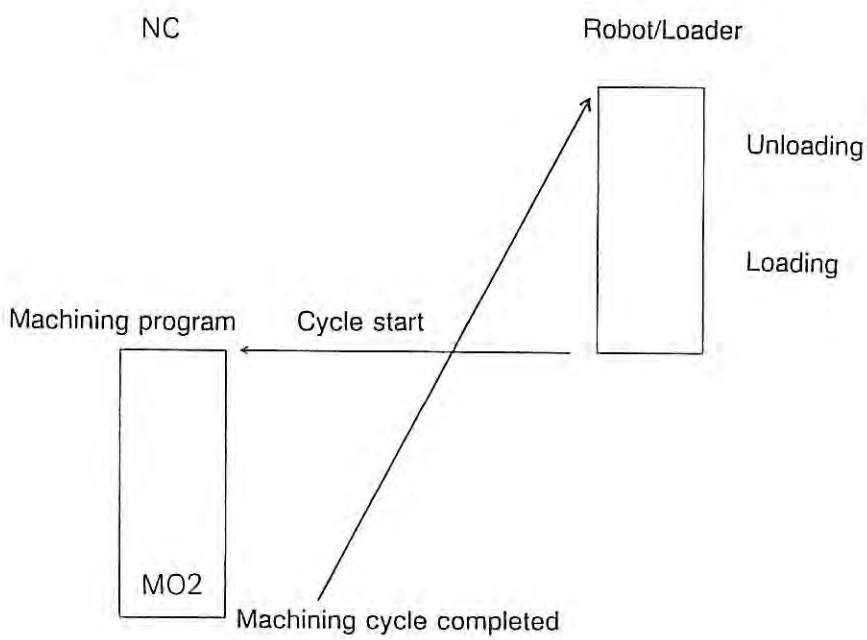
3-1. Features

(1) NC operations are disabled while the robot/loader is carrying out loading/unloading cycle.

(2) Only cutting program is required when making NC program.

(3) Schedule programs for the NC are not necessary.

(4) Control is comparatively simple since the NC and robot/loader control functions are clearly defined and separated.



4. Alarm Messages

ALARM-A

150 Coupling device illegal

ALARM-C

969 coupling device illegal

An abnormality was detected with the robot.

Index : None

Character-string : None

Code : None

The alarm level can be switched between Alarm A and Alarm C by parameter setting.

Bit 2 of parameter bit No. 8
- 0 → Alarm A
- 1 → Alarm C

Standard: 0 → Alarm A

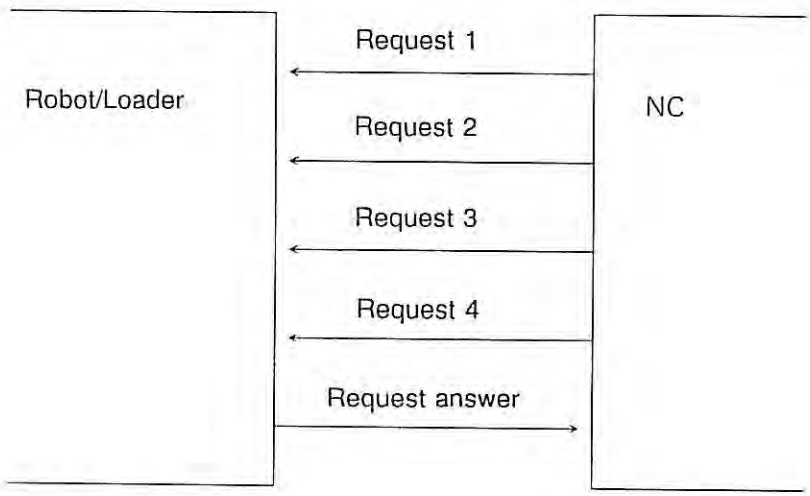
In addition, whether the alarm occurrence is checked "always" or "only in AUTO CYCLE mode" can be determined by parameter setting.

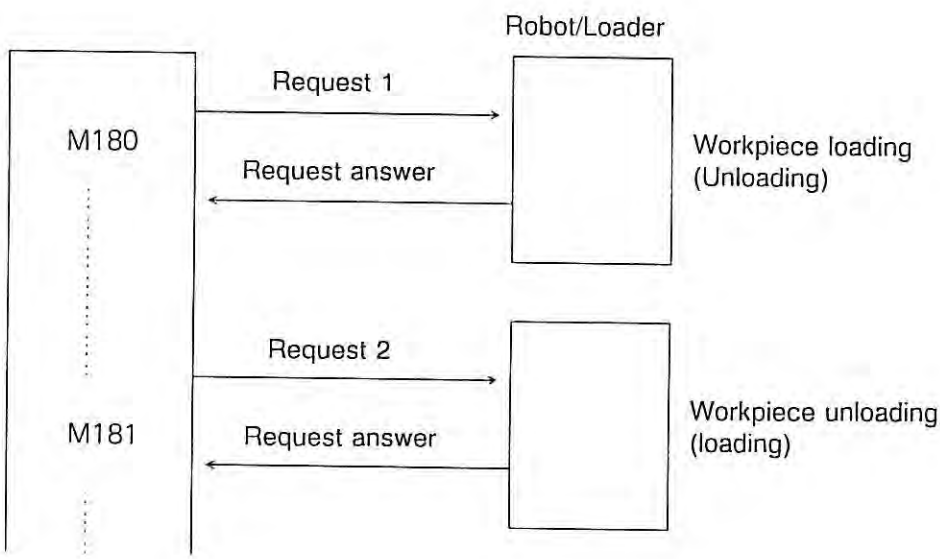
Bit 5 of parameter (bit) No. 6
- 0 → Always monitored
- 1 → Only in AUTO CYCLE mode

Standard: 0 → Always monitored

5. Operation of Type-B Robot/Loader Interface

In response to requests, request answers from the robot/loader are received.





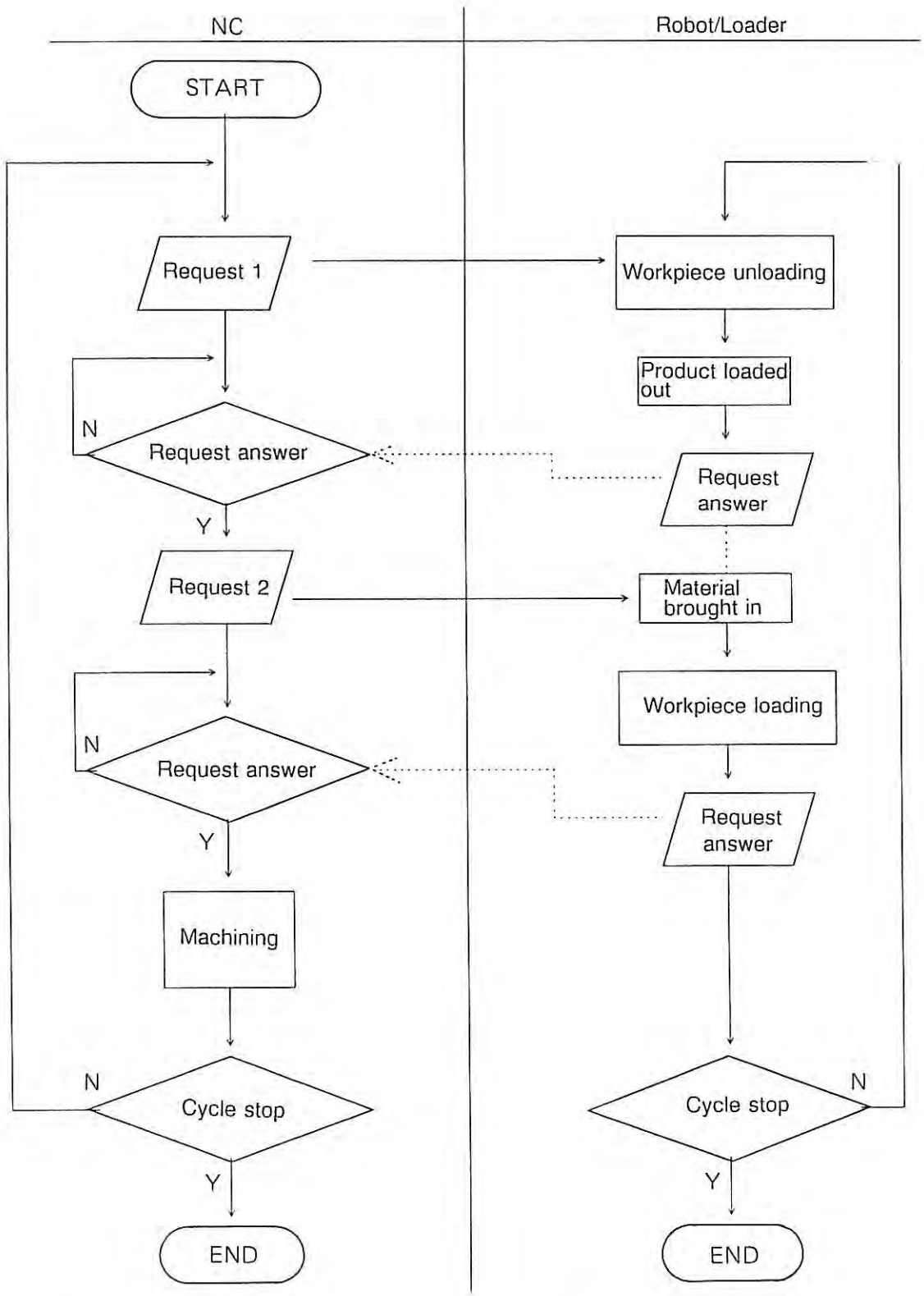
When four types of request are insufficient for operations that require more, select separately "Robot/Loader Program Selection Specifications" which permits BCD, two-digit and four-digit commands of program signals from the NC.

Program number is determined after consultation.

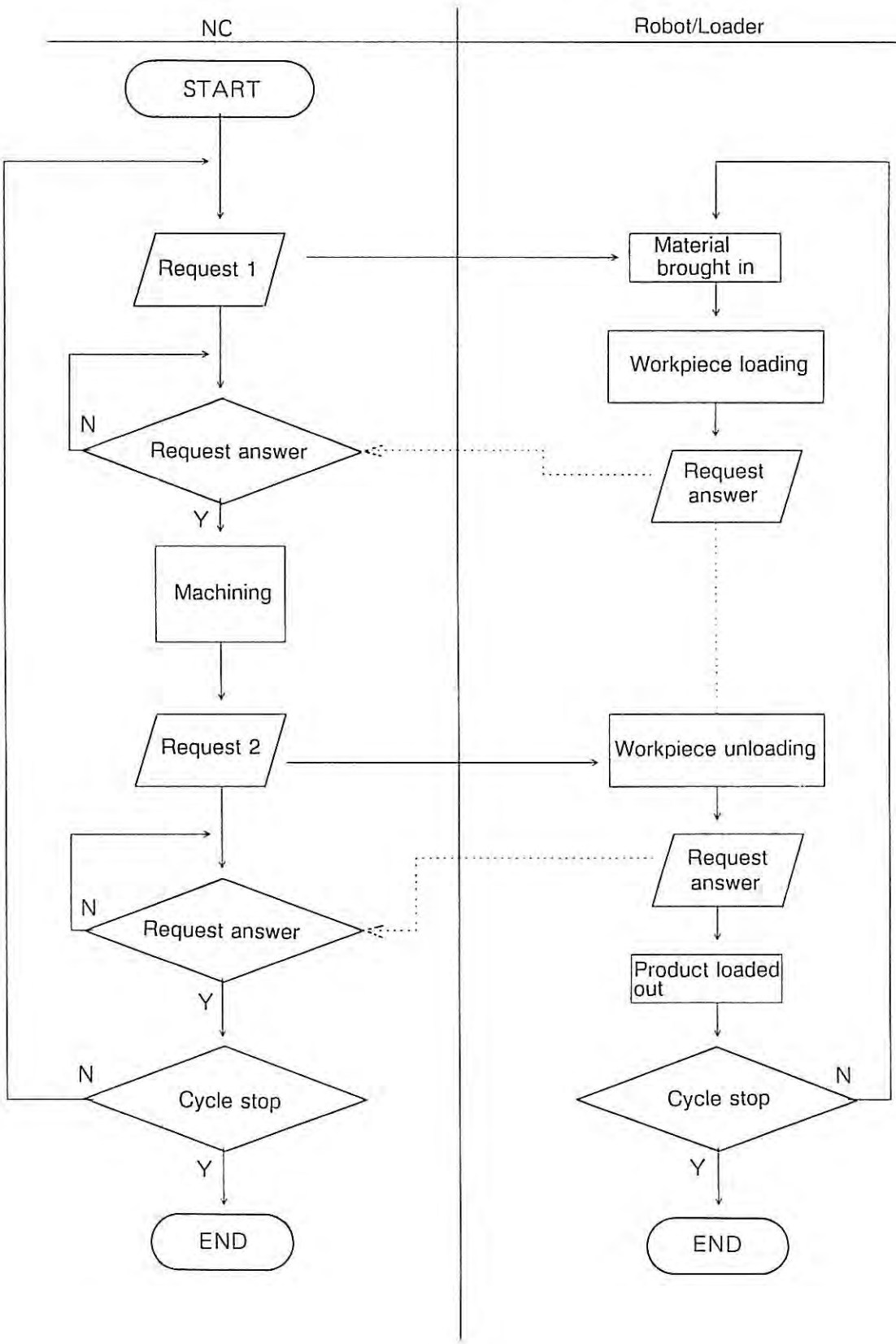
To execute (4), "For double-hand robots, and the operation starts and ends without the product (the machined workpiece) inside the NC machine" of 5-1. "User Task 2" must be selected.

5-1. Operation Sequence Diagram

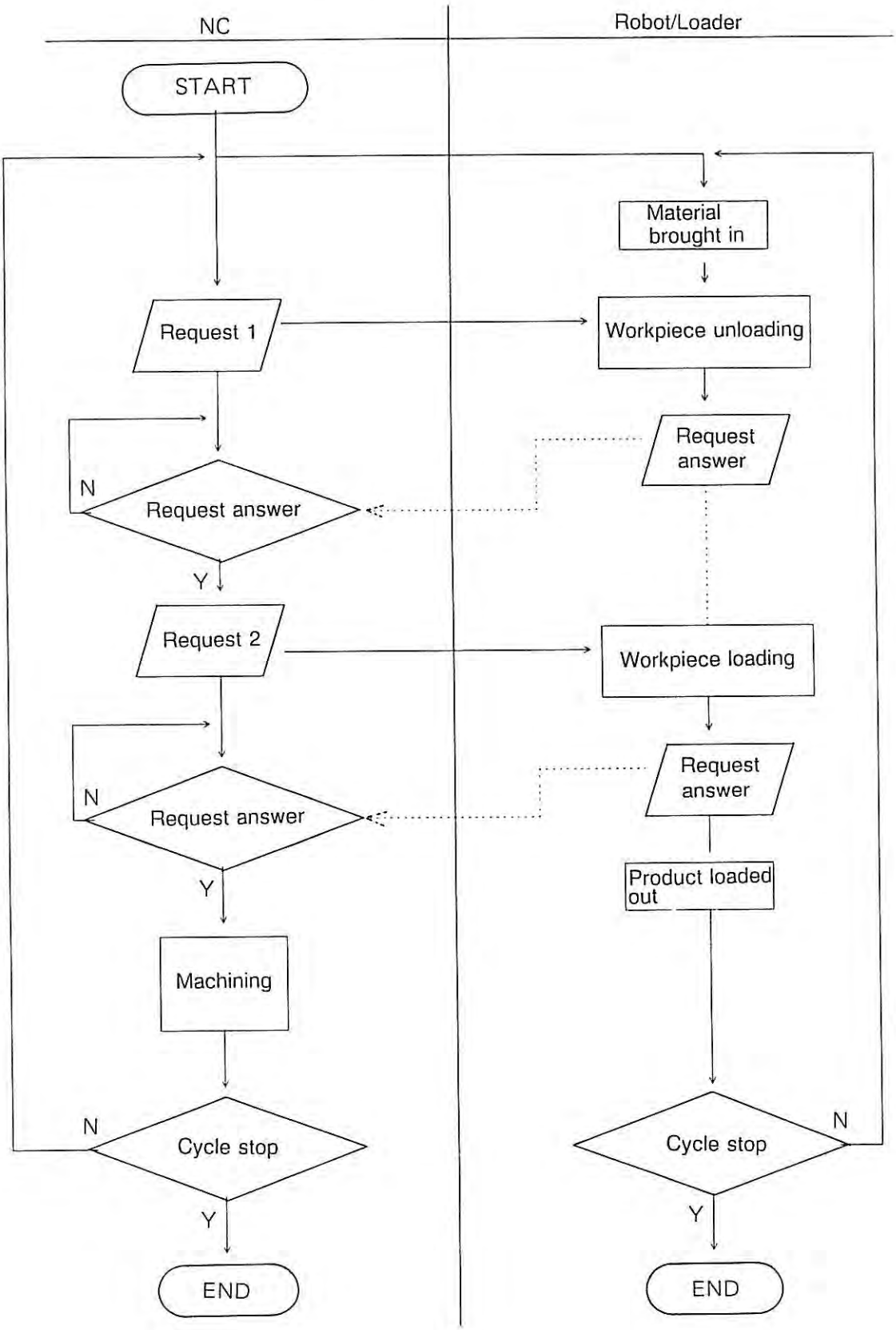
(1) For single-hand robots, and the operation starts and ends with the product (the machined workpiece) inside the NC machine:



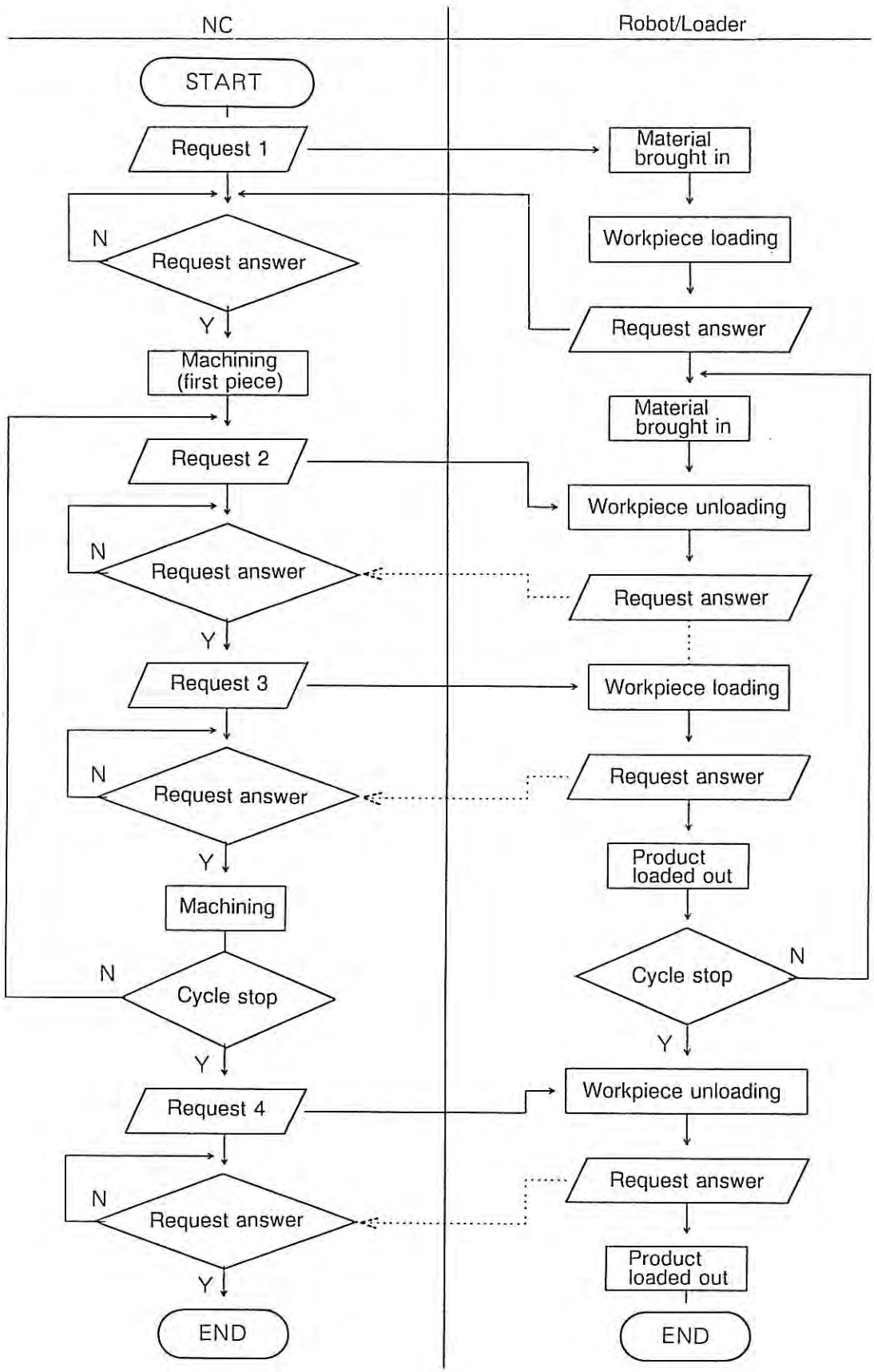
(2) For single-hand robots, and the operation starts and ends without the product (the machined workpiece) inside the NC machine.



(3) For double-hand robots, and the operation starts and ends with the product (the machined workpiece) inside the NC machine.

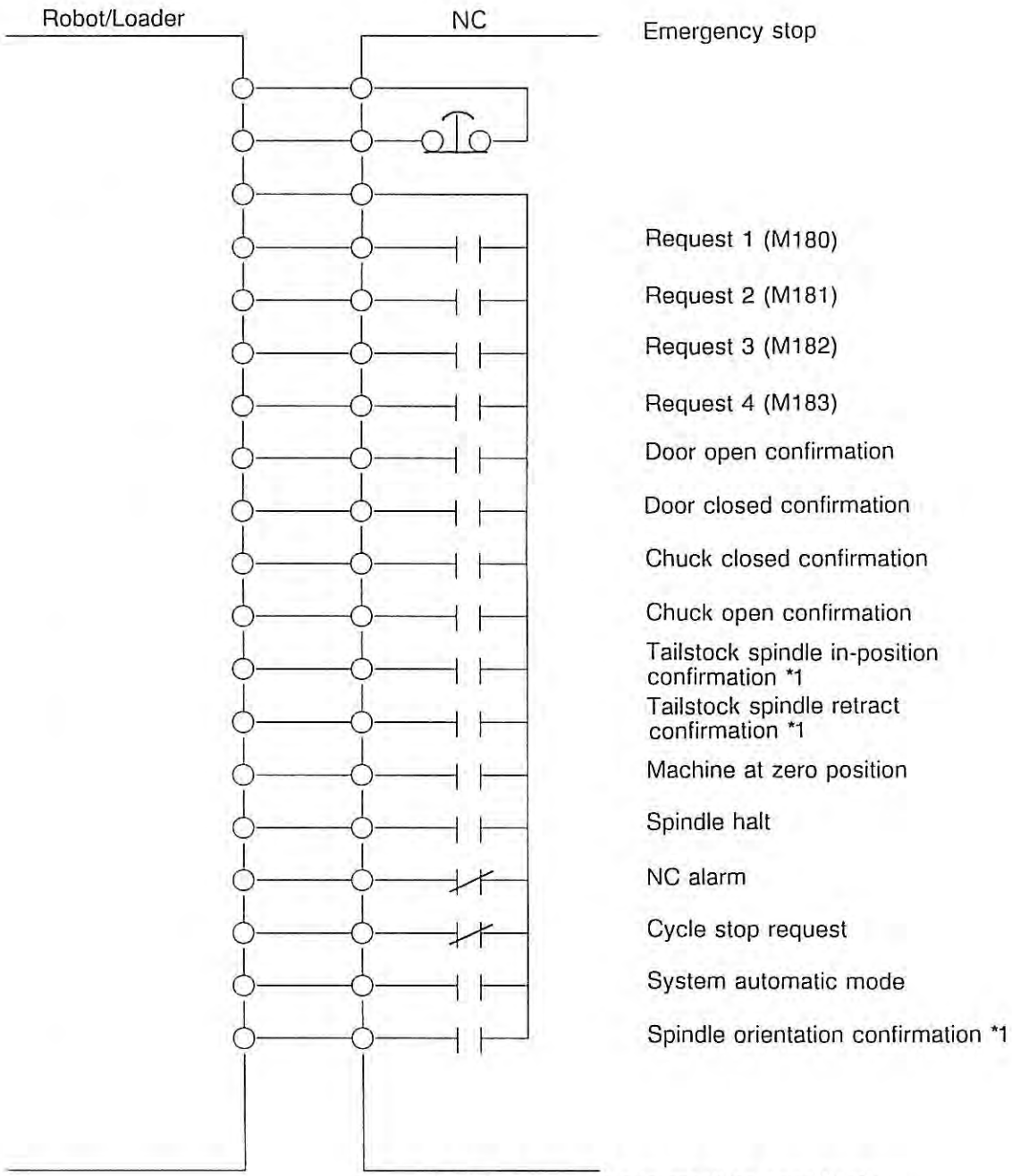


(4) For double-hand robots, and the operation starts and ends without the product (the machined workpiece) inside the NC machine.



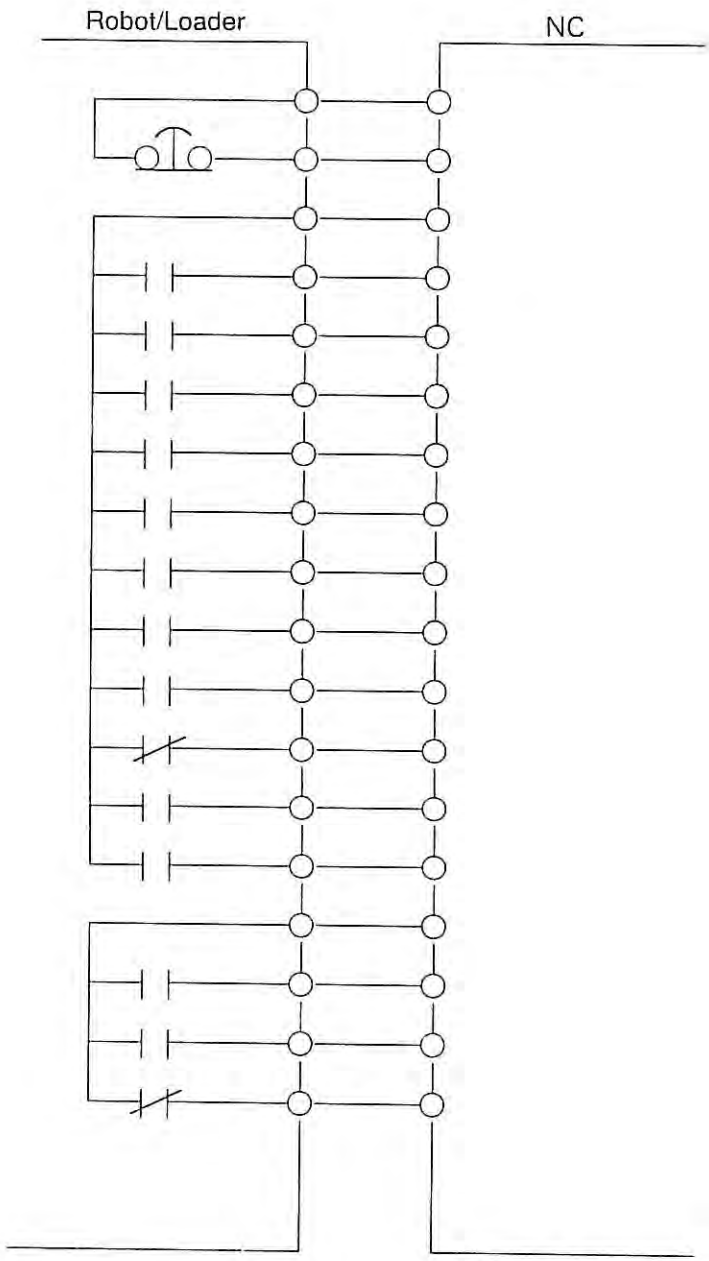
5-2. I/O Signals

(1) Outputs from NC to Robot/Loader



*1 Varies according to machine specifications

(2) Inputs from the NC Robot/Loader



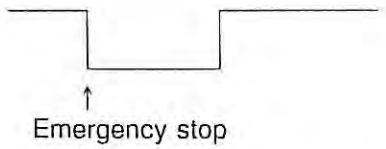
*1 Varies according to machine specifications

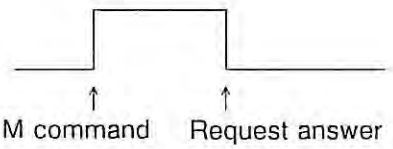
*2 This is used when servicing multiple robot/loader facilities having the sequential start button switch in the main operation panel.

5-3. Signals

(1) Outputs from the NC to the Robot/Loader

<table>
<tr><td>Emergency stop

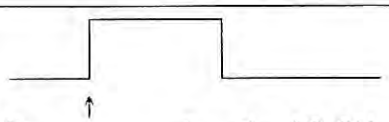
This signal is normally ON and set OFF when the emergency stop button on the NC operation panel is pressed. When this button is pressed, the NC is placed in the alarm state and stops entire operation. The robot/loader should be halted by turning the operational power supply off.</td><td> 
</td></tr>
<tr><td>Request 1 (M180)

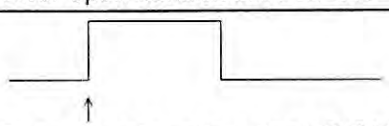
This signal is set to ON in the automatic operation mode when command M180 is output from the NC. Using this signal, conduct robot/loader operations corresponding to the request. The signal is set OFF with the request answer signal.</td><td rowspan="4"> 
</td></tr>
<tr><td>Request 2 (M181)

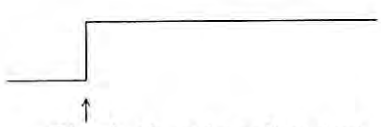
Same as Request 1. The M signal is M181.</td></tr>
<tr><td>Request 3 (M182)

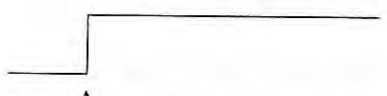
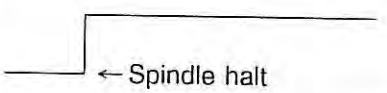
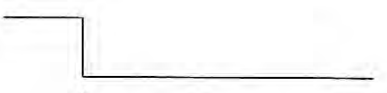
Same as Request 1. The M signal is M182.</td></tr>
<tr><td>Request 4 (M183)

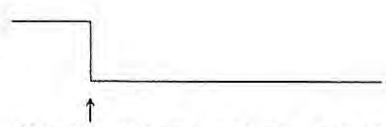
Same as Request 1. The M signal is M183.</td></tr>
<tr><td>Door open confirmation

This signal is set ON when the door open confirmation limit switch is actuated.</td><td> 
</td></tr>
<tr><td>Door closed confirmation

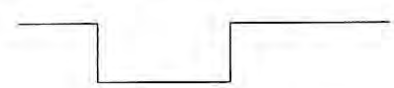
This signal is set ON when the door closed confirmation limit switch is actuated.</td><td> 
</td></tr>
<tr><td>Chuck closed confirmation

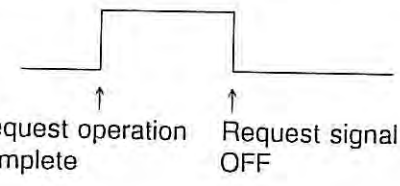
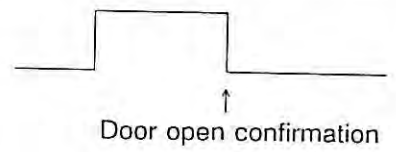
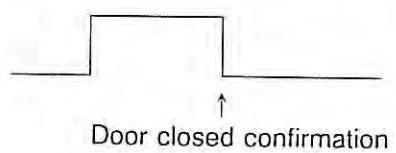
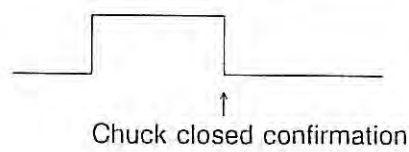
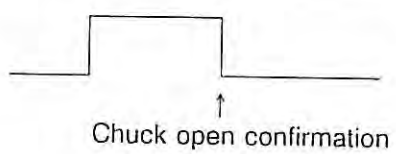
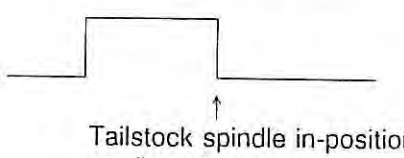
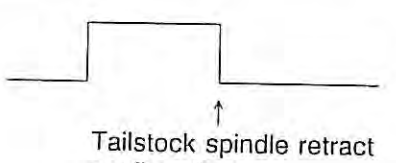
This signal is set ON when the chuck closed confirmation limit switch is actuated and the limit set by the soft timer is exceeded. The signal is set OFF with the next chuck open command. The soft timer count time can be set in 10 msec units using the optional parameter (word) No. 5.</td><td> 
</td></tr>
</table>

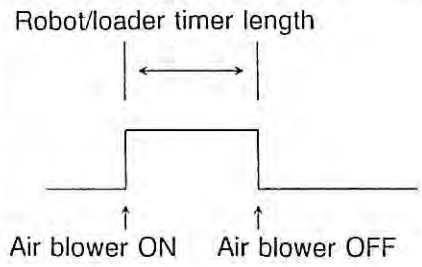
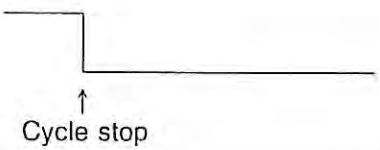
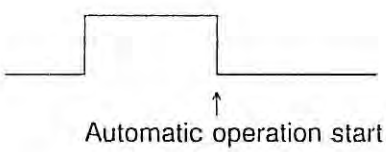
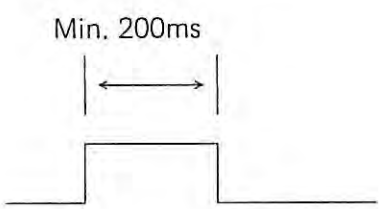
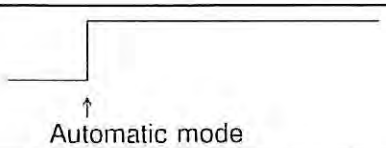
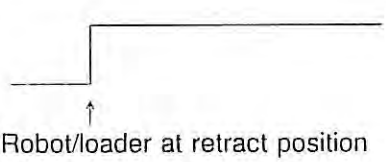
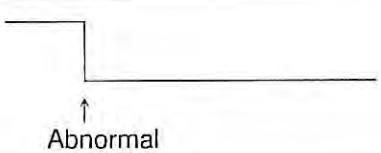
Chuck open confirmation This signal is set ON when the chuck open confirmation limit switch is actuated and the limit set by the soft timer is exceeded. The signal is set OFF with the next chuck closed command. The soft timer count time can be set in 10 msec units using the optional parameter (word) No. 6.	 ↑ Chuck opened confirmation LS & soft timer
Tailstock spindle in-position confirmation (For the machine equipped with tailstock) This signal is set ON when the tailstock spindle in-position confirmation limit switch and the tailstock spindle over-advanced confirmation limit switch are both actuated. The next tailstock spindle retract command sets the signal OFF.	 ↑ Tailstock spindle in-position confirmation LS & tailstock spindle over-advanced confirmation LS.
Tailstock spindle retract confirmation (For the machine equipped with tailstock) This signal is set ON when the tailstock spindle retract limit switch is actuated. The next tailstock spindle advanced command sets the signal OFF.	 ↑ Tailstock spindle retract confirmation LS
Machine at zero position This signal is set ON when the turret* is at the X and Z-axis plus (+) travel limits with the door open confirmation LS is ON. * Both A and B turrets for two-saddle specification	 ↑ X-axis tool rest plus (+) limit, Z-axis tool rest plus (+) limit, & door closed confirmation LS
Spindle halt This signal is set ON while the spindle is halted.	 ← Spindle halt
NC alarm (A, B, C) This signal, normally ON, is set to OFF when alarm A, B, and C of the five following alarms are generated --- the NC CPU alarm, and alarms P, A, B, and C. Alarm reset sets the signal back to ON.	 ↑ Alarm A, B, and C

Cycle stop request This signal, normally ON, is set to OFF when the following conditions exist. When this occurs, cycle stop the robot/loader. (1) This signal is set to OFF when the workpiece counter has been counted up. (For workpiece counter specifications) (2) This signal is set to OFF when tool life has reached its specified limits (For tool life management specifications.) *Note: When NC is not provided with the tool life management function, strap the terminals at the terminal block in the NC box.* (3) This signal is set to OFF if it has been determined that no more stock material is available or finished product becomes full on the out chute or similar equipment when the stock material and finished products are managed.	 ↑ Work counter count-up or tool life command
System automatic mode This signal is set to ON when an OSP mode other than manual operation has been selected with the system automatic mode switch located on the NC additional operation panel set to ON.	 ↑ System automatic operation ON & OSP in other than manual mode
Spindle orientation confirmation (For machines having this specification) This signal is set to ON when the spindle is stopped at the pre-set in-position when the spindle orientation command has been executed.	 ↑ Spindle orientation

(2) Inputs from the Robot/Loader to the NC

Emergency stop The NC comes to a complete stop when this signal is set to OFF. The message **"ALARM-A Emergency stop"** is displayed on the CRT screen.	 ↑ Emergency stop

Request answer The request signal being sent from the NC to the robot/loader is set to OFF when this signal is set to ON in the automatic operation mode. Set this signal to OFF after the request signal is set to OFF.	
Door open command The NC opens the door when this signal is set to ON and the robot/loader at-retract position signal is ON while in the automatic mode. Set the door open confirmation signal to OFF.	
Door close command The NC closes the door when this signal is set to ON and the robot/loader at-retract position location signal is ON while in the automatic mode. Set the door closed confirmation signal to OFF.	
Chuck close command The NC machine closes chuck when this signal is set to ON while in the automatic mode. Set this signal OFF in response to the chuck closed confirmation signal.	
Chuck open command The NC machine opens chuck when this signal is set to ON while in the automatic mode. Set this signal OFF in response to the chuck open confirmation signal.	
Tailstock spindle advance command (For the machine equipped with tailstock) The NC machine advances the tailstock spindle when this signal is set to ON while in the automatic mode. Set this signal OFF in response to the tailstock spindle in-position confirmation signal.	
Tailstock spindle retract command (For the machine equipped with tailstock) The NC retracts the tailstock spindle when this signal is set to ON while in the sequential mode. Set the tailstock spindle retract confirmation signal to OFF.	

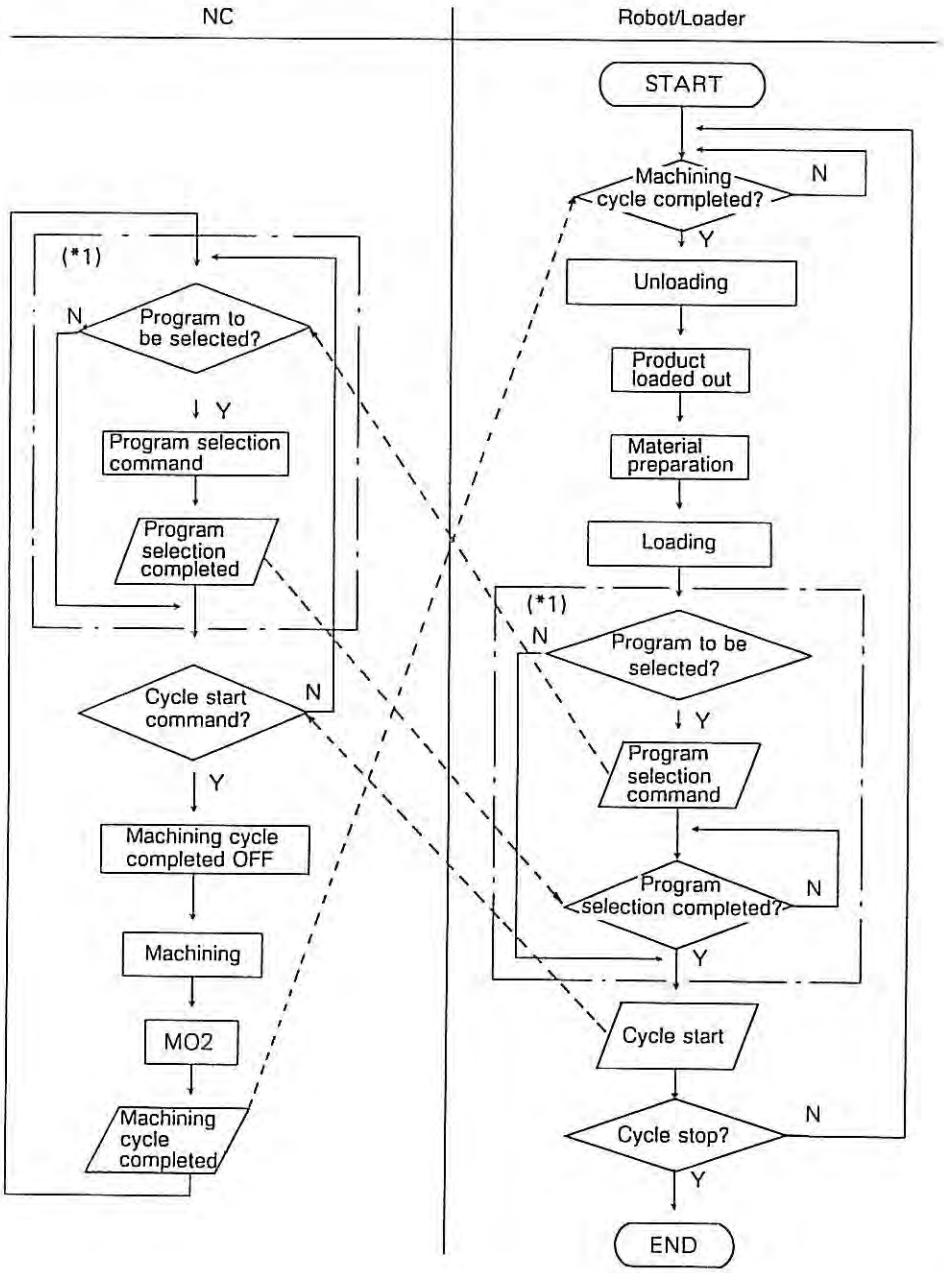
Air blower command The period of use of the air blower during workpiece loading and unloading is controlled by the robot/loader timer. The NC machine turns on the air blower and the spindle inching operation only during the period this signal is ON while in the automatic mode.	
Cycle stop When this signal is set to OFF by program end (M02) command in the NC machining program while in the automatic mode, the NC machine executes cycle stop.	
Automatic operation start (For automatic operation button furnished machines) The trailing edge of this signal starts NC operation in the automatic mode executing the selected program.	
Spindle orientation command (optional) If the spindle is to be stopped at the predetermined position while the robot/loader request signal is active, the spindle orientation is carried out when this signal is turned ON in the automatic mode. The orientation position can be set by parameter (word) No. 12 (0.1° units).	
Automatic mode The NC does not output the robot/loader request signal when this signal is OFF.	
Robot/loader at-retract position Set this signal ON when the robot/loader is in a position that does not interfere with the NC machine. When this signal is OFF, the NC door cannot be opened nor closed preventing the start of machine operation.	
Robot/loader abnormal The NC stops at the point this signal is set OFF and the alarm message "**ALARM-A EXTERNAL**" is displayed.	

6. Operation of Type-C Robot/Loader Interface

6-1. Operation Sequence Diagram

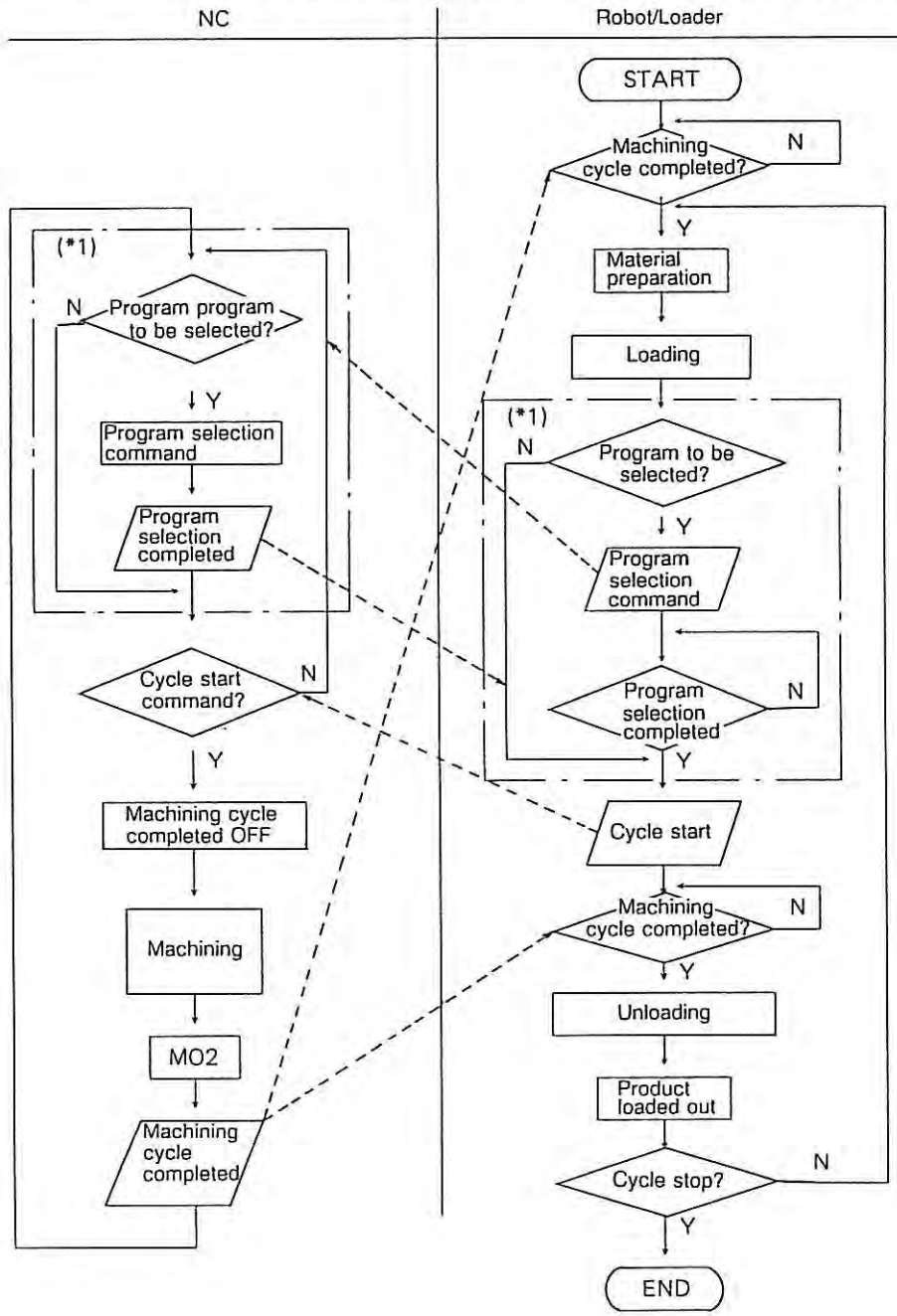
(1) For Single-Hand Robot/Loader

a) The operation begins and ends with the workpiece in the NC machine.



(*1) When a different part program is to be executed within one schedule, it becomes necessary to select separately external program selection specifications.

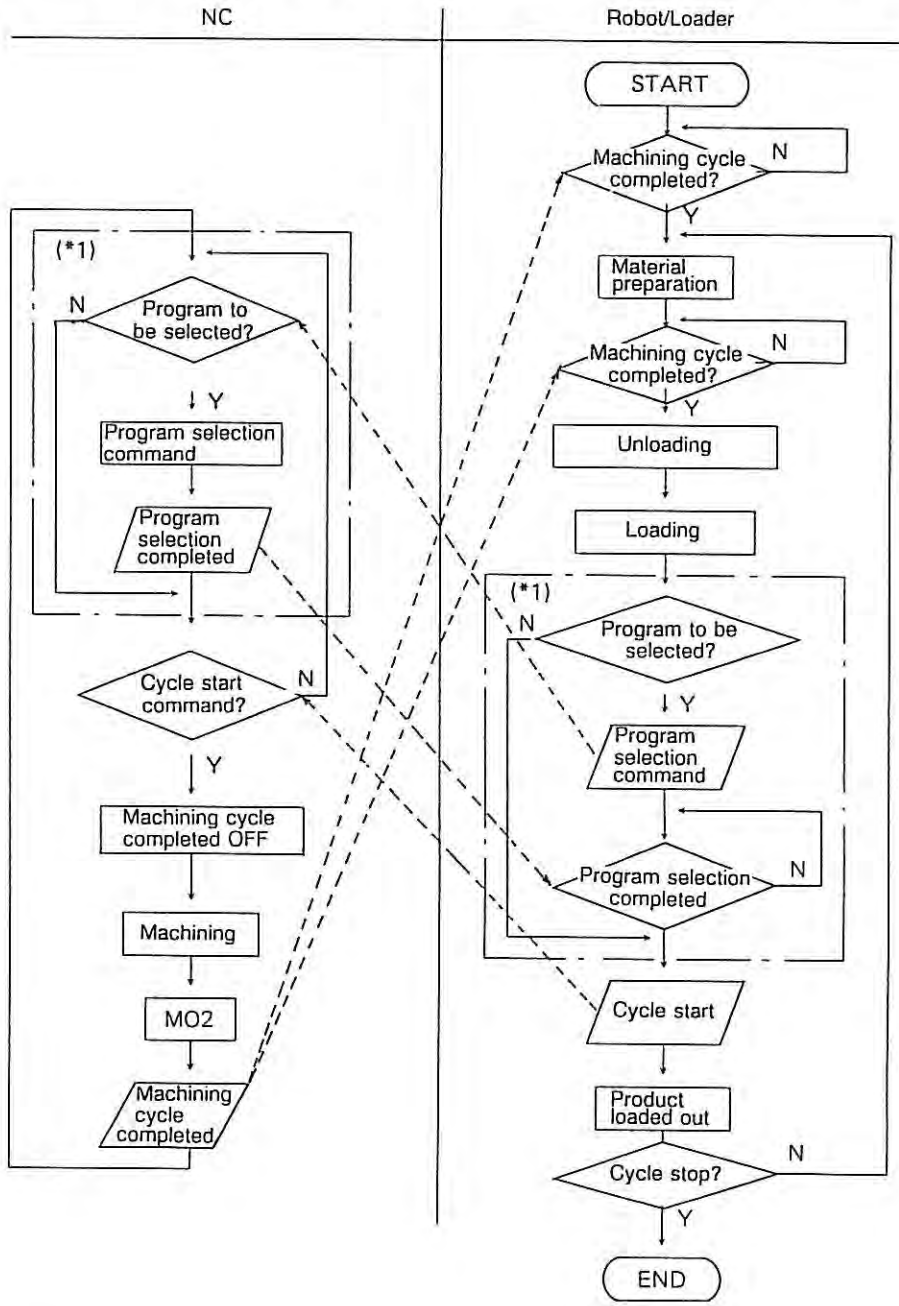
b) The operation begins and ends without the workpiece in the NC machine.



(*1) When a different part program is to be executed within one schedule, it becomes necessary to select separately external program selection specifications.

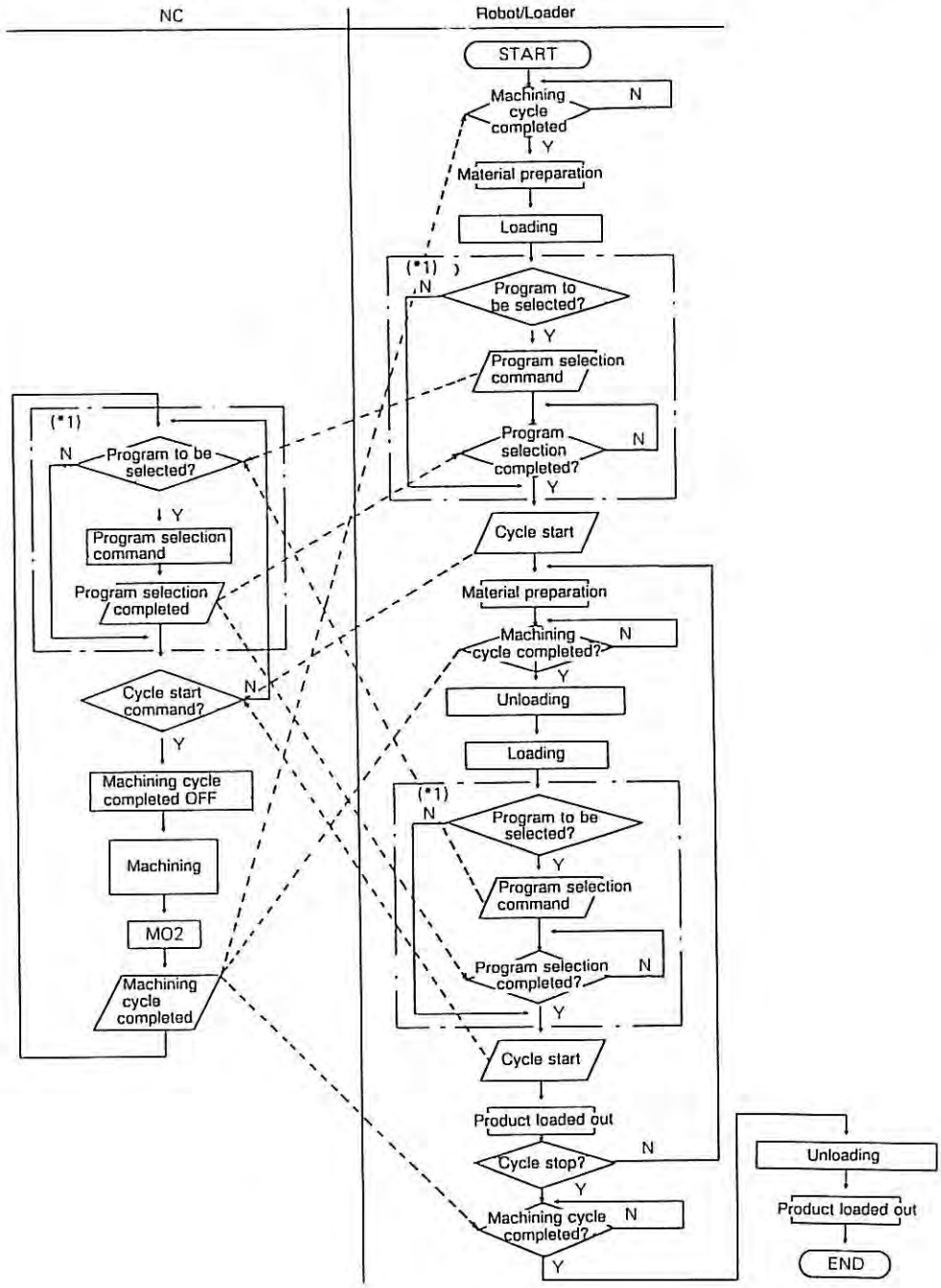
(2) For Double-Hand Robot/Loader

a) The operation begins and ends with the workpiece in the NC machine.



(*1) When a different part program is to be executed within one schedule, it becomes necessary to select separately external program selection specifications.

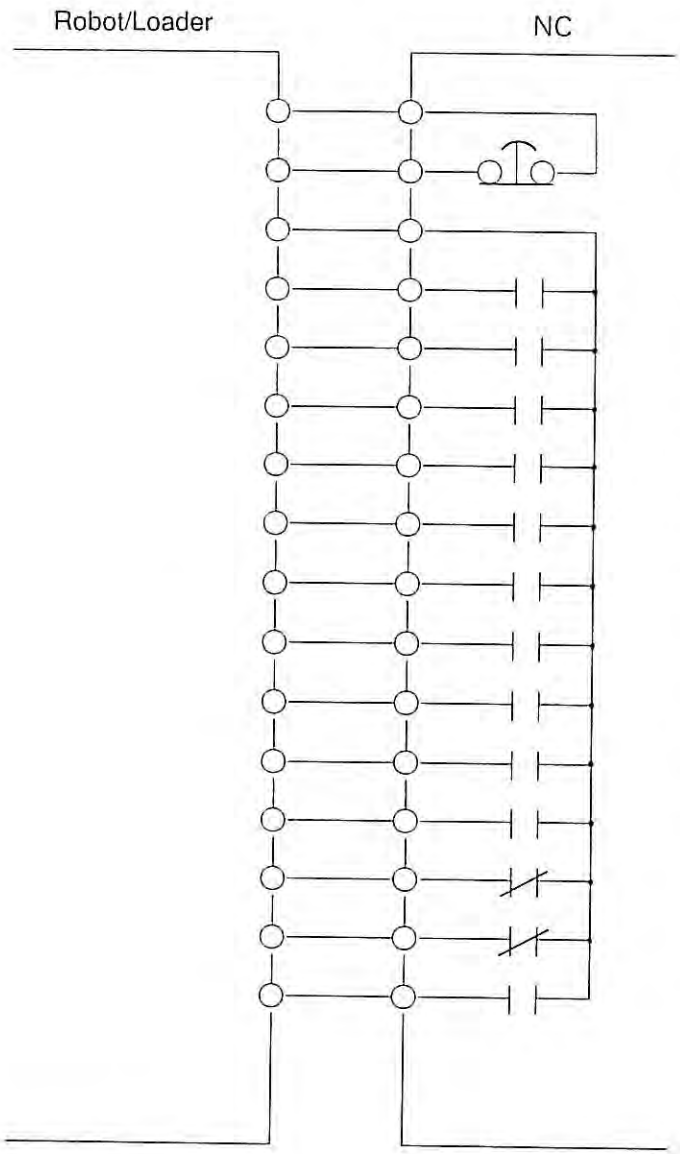
b) The operation begins and ends without the workpiece in the NC machine.



(*1) When a different part program is to be executed within one schedule, it becomes necessary to select separately external program selection specifications.

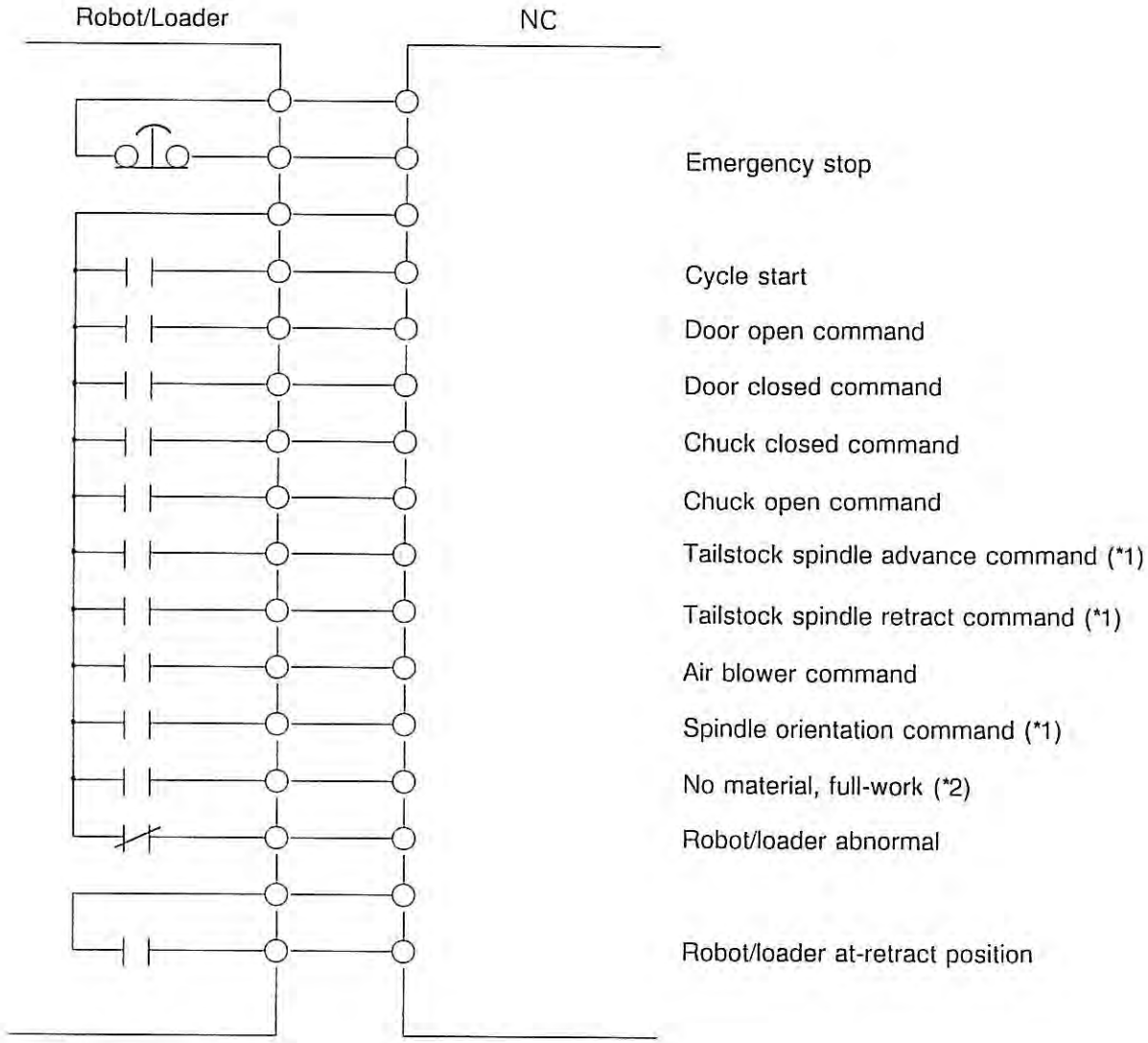
6-2. I/O Signals

(1) Outputs from NC to Robot/Loader



(*1) Varies according to machine specifications

(2) Inputs from the NC to the Robot/Loader

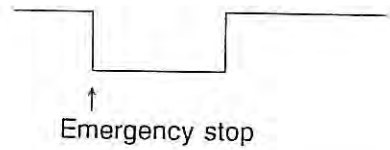
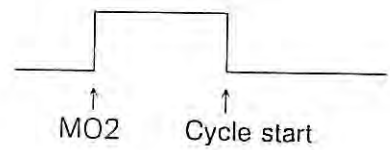
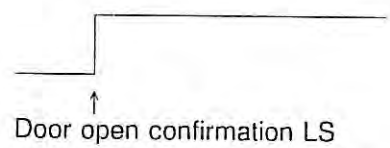
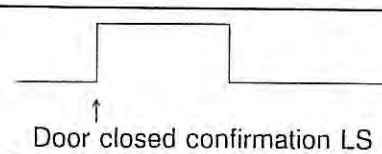
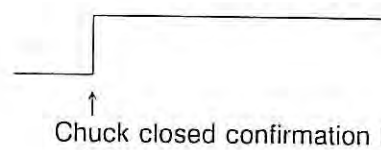


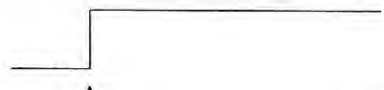
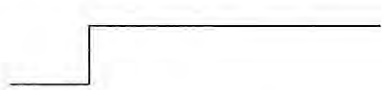
(*1) Varies according to machine specifications.

(*2) This signal is input when presence of material and product is managed by the robot/loader.

6-3. Signals

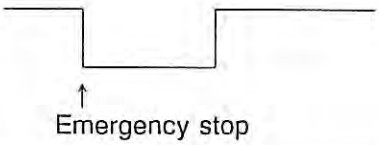
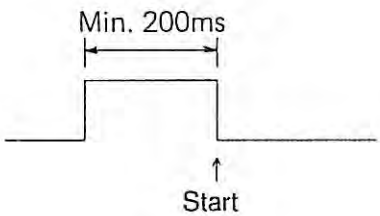
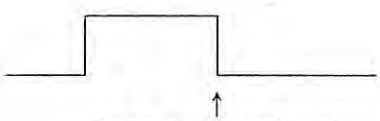
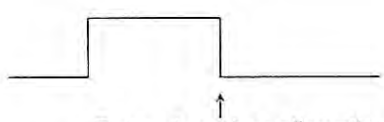
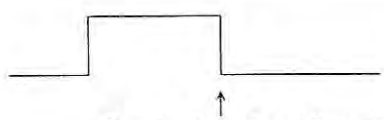
(1) Outputs from the NC to the Robot/Loader

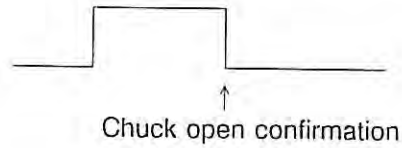
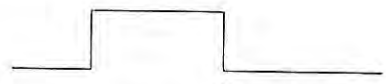
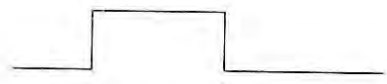
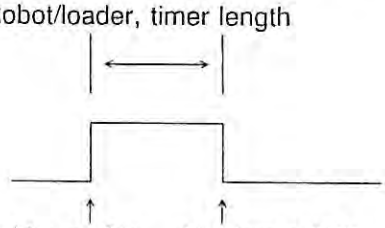
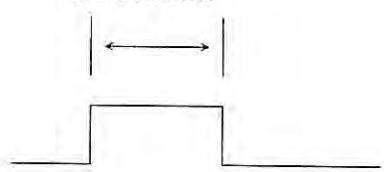
Emergency stop This signal is normally ON and set OFF when the emergency stop button on the NC operation panel is pressed. When this button is pressed, the NC is placed in the alarm state and stops entire operation. The robot/loader should be halted by turning the operational power supply off.	
Machining cycle completed This signal is set to ON by the NC machining completed code M02 (program end) and turned off by the leading edge of the cycle start signal from the robot/loader. The status of this signal is maintained whether applied power is removed or NC reset command is given. The robot/loader controller should check this signal at the trailing edge of the cycle start signal. This signal should be used as one of the robot/loader cycle start conditions in the automatic operations.	
Door open confirmation This signal is set ON when the door open confirmation limit switch is actuated. Use this as a condition for the entry of robot/loader into NC machine.	
Door closed confirmation This signal is set ON when the door closed confirmation limit switch is actuated.	
Chuck closed confirmation This signal is set ON when the chuck closed confirmation limit switch is actuated and the limit set by the soft timer is exceeded. The signal is set OFF with the next chuck open command. The soft timer count time can be set in 10 msec units using the optional parameter (word) No. 5.	

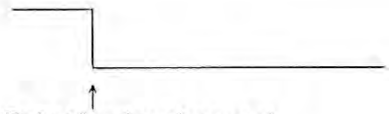
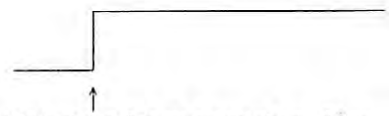
Chuck open confirmation This signal is set ON when the chuck open confirmation limit switch is actuated and the limit set by the soft timer is exceeded. The signal is set OFF with the next chuck closed command. The soft timer count time can be set in 10 msec units using the optional parameter (word) No. 6.	 ↑ Chuck opened confirmation LS & soft timer
Tailstock spindle in-position confirmation This signal is set ON when the tailstock spindle in-position confirmation limit switch and the tailstock spindle over-advanced confirmation limit switch are both actuated. The next tailstock spindle retract command sets the signal OFF.	 ↑ Tailstock spindle in-position confirmation LS & tailstock spindle over-advanced confirmation LS.
Tailstock spindle retract confirmation This signal is set ON when the tailstock spindle retract limit switch is actuated. The next tailstock spindle advanced command sets the signal OFF.	 ↑ Tailstock spindle retract confirmation LS
Machine at zero position This signal is set ON when the turret* is at the X and Z-axis plus (+) travel limits. Use this as a condition for the entry of robot/loader into NC machine. * Both A and B turrets for two-saddle specification	 ↑ X-axis tool rest plus (+) limit, Z-axis tool rest plus (+) limit
Spindle halt This signal is set ON while the spindle is halted. Use this as a condition for movements of the robot/loader to the chuck.	 ↑ Spindle halt
Spindle orientation This signal is set to ON when the spindle is stopped at the pre-set in-position when the spindle orientation command has been executed.	 ↑ Spindle orientation

NC alarm (A, B, C) This signal, normally ON, is set to OFF when alarm A, B, and C of the five following alarms are generated --- the NC CPU alarm, and alarms P, A, B, and C. Alarm reset sets the signal back to ON.	 Alarm A, B, and C
Cycle stop request This signal, normally ON, is set to OFF when the following conditions exist. When this occurs, cycle stop the robot/loader. (1) This signal is set to OFF when the workpiece counter has been counted up. (For workpiece counter specifications) (2) This signal is set to OFF when tool life has reached its specified limits (For tool life management specifications.) *Note: When NC is not provided with the tool life management function, strap the terminals at the terminal block in the NC box.*	 Work counter count-up or tool life command
Automatic mode This signal is set to ON when an OSP mode other than manual operation has been selected with the system automatic mode switch located on the NC additional operation panel set to ON.	 System automatic operation ON & OSP in other than manual mode

(2) Inputs from the Robot/Loader to the NC

Emergency stop The NC comes to a complete stop when this signal is set to OFF. The message **"ALARM-A Emergency stop"** is displayed on the CRT screen.	
Cycle start When the conditions listed below are all satisfied, the NC executes the selected machining program upon the trailing edge of this signal. (1) The system automatic ON/OFF switch is in the ON position. (2) The OSP mode selected is "Automatic operation". (3) Machining program selection is completed. (4) NC status is no-alarm. (5) NC is not in operation. (6) Robot/loader is in the retracted position.	
Door open command The NC opens the door when this signal is set to ON and the robot/loader at-retract position signal is ON while in the automatic mode. Set the door open confirmation signal to OFF.	
Door close command The NC closes the door when this signal is set to ON and the robot/loader at-retract position signal is ON while in the automatic mode. Set the door closed confirmation signal to OFF.	
Chuck close command The NC machine closes chuck when this signal is set to ON while in the automatic mode. Set this signal OFF in response to the chuck closed confirmation signal.	

Chuck open command The NC machine opens chuck when this signal is set to ON while in the automatic mode. Set this signal OFF in response to the chuck open confirmation signal.	 Chuck open confirmation
Tailstock spindle advance command The NC machine advances the tailstock spindle when this signal is set to ON while in the automatic mode. Set this signal OFF in response to the tailstock spindle in-position confirmation signal.	 Tailstock spindle in-position confirmation
Tailstock spindle retract command The NC machine retracts the tailstock spindle when this signal is set to ON while in the automatic mode. Set this signal OFF in response to the tailstock spindle retract confirmation signal.	 Tailstock spindle retract confirmation
Air blower command The period of use of the air blower during workpiece loading and unloading is controlled by the robot/loader timer. The NC machine turns on the air blower and the spindle inching operation only during the period this signal is ON while in the automatic mode.	Robot/loader, timer length  Air blower ON Air blower OFF
Spindle orientation command If the spindle is to be stopped at the predetermined position for conducting the loading/unloading operation, the spindle orientation is carried out when this signal is turned ON. The orientation position can be set by optional parameter (word) No. 12 (0.1° units).	Min. 200ms 
No material, full-work When this signal is set to ON the NC lights the operator call lamp (patolite, yellow). This is not necessary when there is an operator call lamp on the robot/loader. Provided upon request.	 No material, full-work

Robot/loader abnormal When this signal is set to OFF the NC lights the operator call lamp (patolite, red). This is not necessary when there is an operator call lamp on the robot/loader. Provided upon request.	 ↑ Robot/loader abnormal
Robot/loader at-retract position Set this signal ON when the robot/loader is in a position that does not interfere with the NC machine. When this signal is OFF, the NC door cannot be opened nor closed preventing the start of machine operation.	 ↑ Robot/loader at retract position

7. Connection Precautions

(1) This piece of equipment provides a non-voltage interface between the input and output.

A G2A-432A 24 V DC produced by OMRON is used in the NC for I/O signal interfacing.

Ratings		Contact		
Operation coil		Load / Items	Resistive load (cos o = 1)	Inductive load (cos o = 0.4 L/R = 7 ms)
Voltage	DC24V			
Current	45m A	Rated load	AC110V 0.3A DC24V 0.5A	AC110V 0.2A DC24V 0.3A
Coil resistance	530	Rated current	3A	
Operating voltage	< 80%	Maximum contact point voltage	AC250V DC125V	
Recovery voltage	< 10%	Maximum contact point	AC1A DC3A	AC0.75A DC1:5A
Dissipation (VA.W)	Approx. 1.1	Maximum open/close capacity	AC110VA DC70W	AC80VA DC36W

(2) I/O signals should be connected to terminals. Connect the signal wires to the NC control box when connecting the robot/loader.

(3) For the needs of independent operations of the NC machine, some terminals are strapped in the interface terminal. Remove the strap when connecting the robot/loader.

(4) Since automatic operation without workpieces during robot/loader adjustment is needed, an independent operation ON/OFF toggle switch should be provided in the robot/loader control box along with a circuit enabling operations without workpieces during independent operation.

SECTION 41 COOLANT HIGH/LOW PRESSURE SELECTION FUNCTION BY M CODES

1. Overview of Coolant High/Low Pressure Selection Function

There are cases where the coolant pressure is required to be changed by a designated command for the workpiece to be machined. This function can be effectively used in such cases.

2. M Codes Used for Selecting High/Low Coolant Pressure

M142 . . . . . Coolant pressure low
M143 . . . . . Coolant pressure high

Selected mode is not influenced by turning on the control or by resetting it.

For the two-saddle models, the M codes above may be designated at either G13 or G14 mode.

LIST OF PUBLICATIONS

Publication No.	Date	Edition
3290-E-R1	September 1990	6th

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.