OPERATING

Apr. 1988
EDITION 3-0

CONTENTS

1. OPERATING PANEL 1

2. OPTIONAL PANEL 5

3. MANUAL OPERATION SWITCH 6

4. OPERATIONAL PROCEDURE 8
4-1. Explanation of Expressional Symbols 8
4-2. Operating the Power ON/OFF 9
[1] Turning power ON 9
[2] Turning power OFF 9
4-3. Explanation of Indicated Location 10
[1] Display of FUNCTION 10
[2] Display of MODE 10
[3] Display of input data by using keys 11
[4] Display of program number and sequence number 11
[5] Display of alarm 12
[6] Indication of soft key 12
4-4. Procedure of Current Position Display 13
4-5. Manual ZERO RETURN 14
4-6. Positioning by Manual Operation 16
4-7. Display of the Amount of Program Memory Used 18
4-8. Storing Data of NC PROCESSOR to Memory 19
4-9. Storing Part Program by MDI 20
4-10. Writing of the Work Coordinate System 21
[1] In case of turning and boring tools 21
[2] In case of I.D tool, drill and reamer 23
[3] Alteration of work coordinate system 24
4-11. Display and Setting of Tool Nose-R Compensation 26
[1] Display of tool nose R compensation 26
[2] Setting of tool nose R compensation 27
[3] Command of the direction of the theoretical cutting edge ... 28
4-12. Returning to the Head of Program 29
4-13. Search Operation 30
[1] Program number search 30
[2] Sequence number search in AUTO mode 31
[3] Method of program search by address 32
4-14. MDI Operation 33
4-15. Display of Command Value 34
4-16. Memory Operation 36
[1] Memory operating procedure 36
[2] SINGLE BLOCK operation 37
[3] Starting cycle from half way of program 38
4-17. Setting and Display of Tool Wear Offset Value 39
[1] Display of tool wear offset value 40
[2] Setting of tool wear offset value 41
4-18. Editing Operation 43
[1] Designation of editing portion 43
[2] Altering a word 44
[3] Inserting a word 45
[4] Deleting a word 46
[5] Deleting a block 47
[6] Deleting for more than one block 48
[7] Deleting a program 49
[8] Deleting all program 49

4-19. Transferring Stored Program to NC PROCESSOR 50
4-20. Display and Setting of Input/Output Parameter
(Setting Parameter) 51
4-21. Display and Change of Parameter 52
[1] Display of parameter 52
[2] Setting and changing parameter 53
[3] Display of PC parameter and PC timer 56
[4] Setting and changing PC parameter 57
4-22. Graphic Function (Option) 58
[1] Setting of graphic parameter 58
[2] Magnification of screen 60
4-23. Conversational Programming with Graphic Function (Option) 62
[1] Programming 62
[2] Program for chamfering and corner radius 64
[3] Method of using multiple repetitive cycle (G70 to G73) 65
[4] Confirmation of program 67
[5] Editing of program 69
4-24. When Slide Activated Machine End Micro Switch 73
4-25. Alarm Display 74
[1] Alarm list 75

5. MAIN SPINDLE CHUCK SYSTEM 84
[1] BND-20S Main spindle chuck system (for USA) 84
[2] BND-20S Main spindle chuck system (Excluding USA) 85
[3] BND-20S Main spindle orientation collet chuck system 86
[4] BND-34S Main spindle chuck system (for USA) 87
[5] BND-34S Main spindle chuck system (Excluding USA) 88
[6] BND-34S Main spindle orientation collet chuck system 89

6. INSTALLATION AND REMOVAL OF COLLET CHUCK FOR MAIN SPINDLE
OF BND-20S, BND-34S 90

7. INSTALLATION AND REMOVAL OF RUBBER FLEX COLLET FOR BND-20S,
BND-34S MAIN SPINDLE 92

8. INSTALLATION AND REMOVAL OF SPRING COLLET CHUCK FOR MAIN SPINDLE
ORIENTATION OF BND-20S, BND-34S 94

9. POWER CHUCK FOR BND-20S, BND-34S MAIN SPINDLE 96

10. ADJUSTMENT OF MAIN SPINDLE CHUCK CLAMPING FORCE 103

11. HYDRAULIC OPERATING 105

12. PNEUMATIC 106

13. CENTRAL OIL SUPPLY 107

14. BND-20S SUB SPINDLE CHUCK SYSTEM 108
[1] Spring type work ejector and chuck system (BND-20S)
for USA 108
[2] Air cylinder type work ejector and chuck system (BND-20S)
for excluding USA 110

15. BND-34S SUB SPINDLE CHUCK SYSTEM 112
[1] Spring type work ejector and chuck system (for USA) 112
[2] Air cylinder type work ejector and chuck system
(for excluding USA) 114

16. SPECIFICATION OF SUB SPINDLE SPRING TYPE WORK EJECTOR AND CHUCK
SYSTEM 117

17. INSTALLATION AND REMOVAL OF SPRING COLLET AND SPRING TYPE WORK EJECTOR 118

18. INSTALLATION AND REMOVAL OF RUBBER FLEX COLLET AND SPRING TYPE WORK EJECTOR 122

19. SPECIFICATION OF AIR CYLINDER TYPE WORK EJECTOR AND CHUCK SYSTEM 127

20. INSTALLATION AND REMOVAL OF SPRING COLLET AND AIR CYLINDER TYPE WORK EJECTOR FOR BND-20S, BND-34S 128

21. INSTALLATION AND REMOVAL OF RUBBER FLEX COLLET AND AIR CYLINDER TYPE WORK EJECTOR FOR BND-20S, BND-34S 130

22. ADJUSTMENT OF CLAMP LENGTH OF THE AIR TYPE WORK EJECTOR 132

23. BAR FEEDING SYSTEM 133
[1] ASR F-20M/F-34M 134
[2] Construction of MS tube barfeeder for BND-20S, BND-34S 135
[3] Replacement part for PF-V2M/PF-V3M 146

24. ADJUSTING METHOD OF PART CATCHER 150

25. EXPLANATION OF ORIENTATION 151

26. MAIN SPINDLE POSITIONING 155

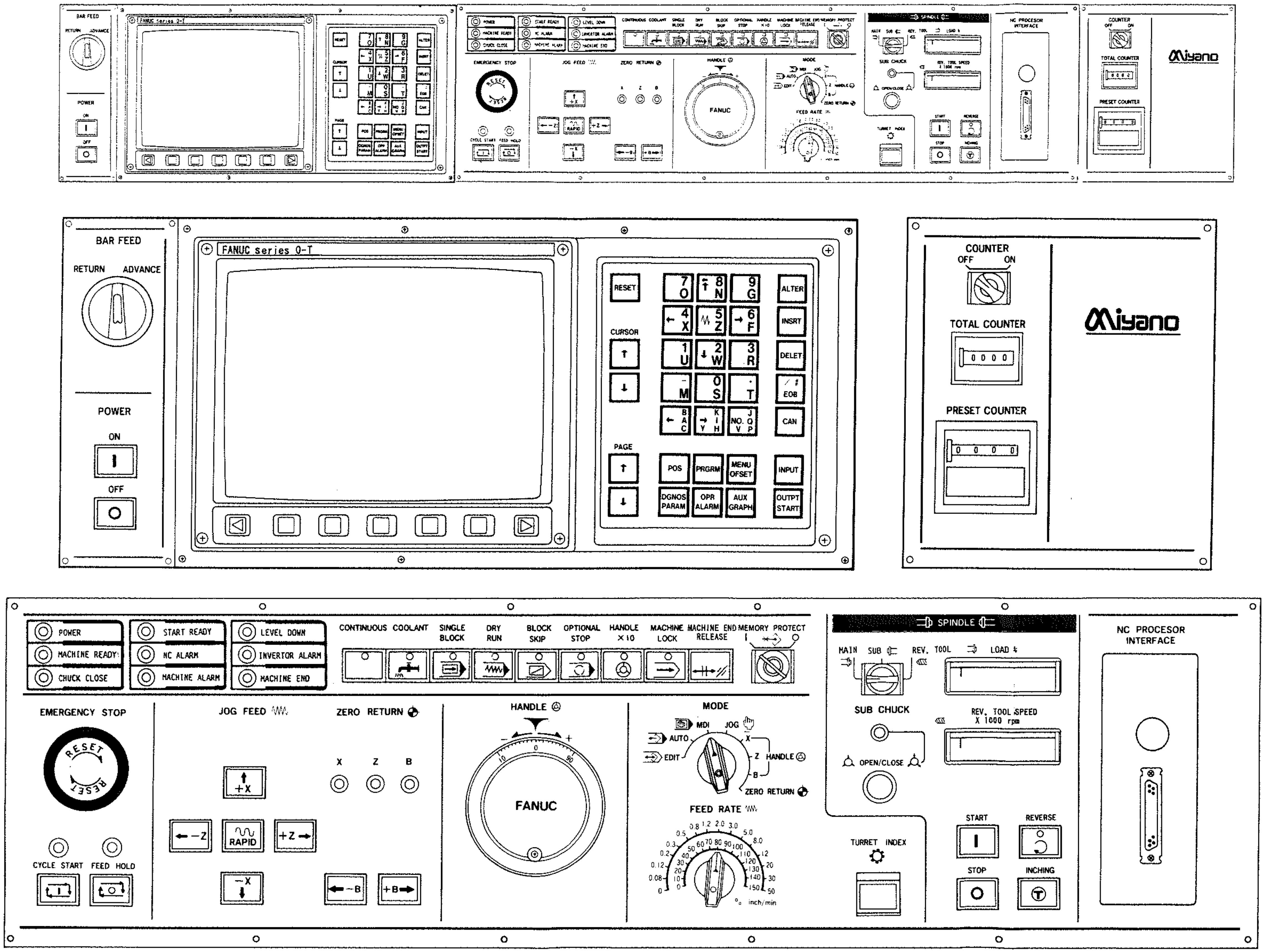
27. SUB SPINDLE POSITIONING 157

28. METHOD FOR CORRECTING PHASE BETWEEN THE MAIN SPINDLE AND SUB SPINDLE 159

29. METHOD OF MAKING "0° SETTING" OF SYNCHRONIZED OPERATION (Option) . 160

BND-S OTC OPE 90/7

1. OPERATING PANEL



CRT character display:
For displaying the contents of program or alarm, etc.

RESET key:
When this key is depressed,
Alarm state is released.
Automatic running stops and the machine stops.

DATA INPUT key:
For inputting the adress(alphabet) and numerical data

PROGRAM EDIT key:

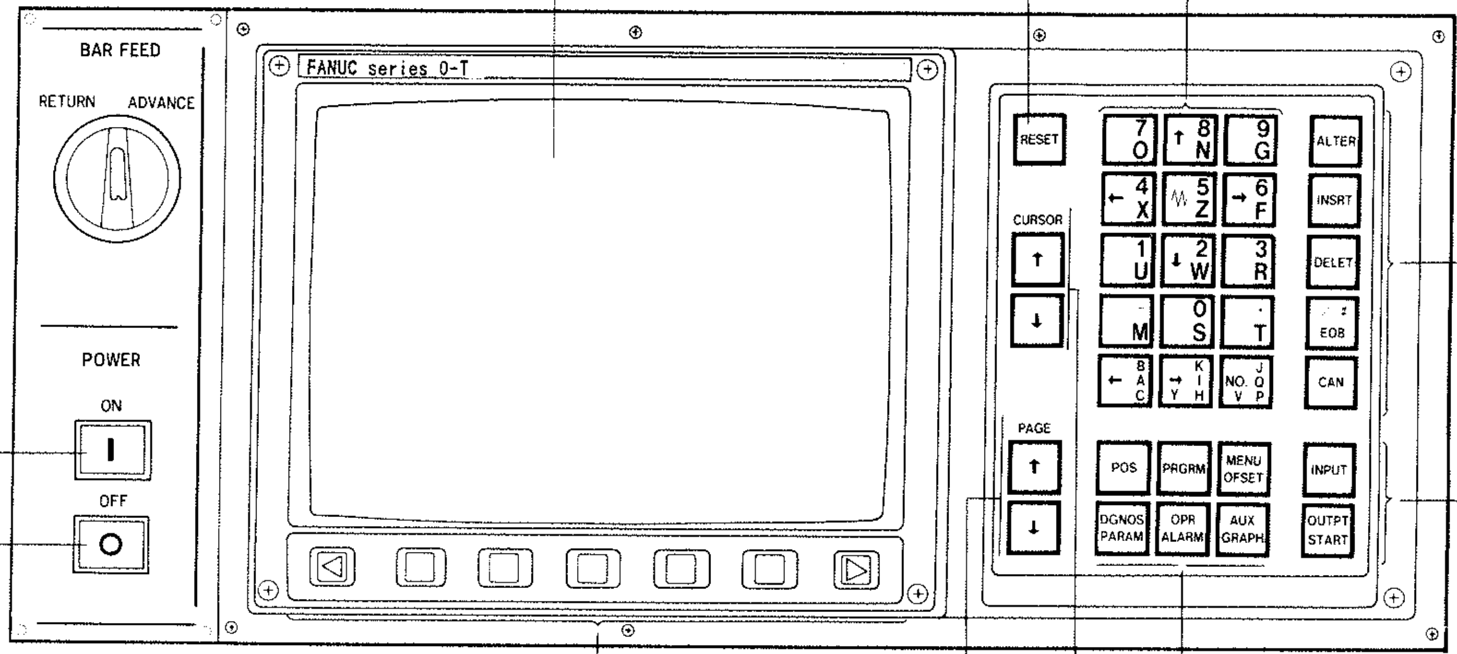
ALTER For modifying the word

INSRT For inserting the word

DELET For deleting the word

EOB For storing '/'(slash) code
For storing one block data in MDI operation

CAN For cancelling mis-input data



INPUT/OUTPUT key:

INPUT Press this key to input parameter or offset etc. and also to start inputting of NC PROCESSOR, or to input command data under MDI mode.

OUTPT START For outputting program parameter and offset etc.

POWER "OFF":
"OFF" switch for NC, servo and hydrauric power

POWER "ON":
"ON" switch for NC, servo and hydrauric power

FUNCTION key's:

POS For displaying the current position

PRGRM Conducts the following:
In EDIT mode........edit and display of the program in the memory
In MDI mode.........input and display of the MDI data
In automatic operation.....
display of command value

MENU OFSET Setting and display of offset value

DGNOS PARAM Setting and display of parameter and display of diagnostic data

OPR ALARM Alarm number display

AUX GRAPH Graphic display(Option)
(NOTE)
This function is valid only when optional graphic display are selected.

SOFT keys:
The soft keys have various functions, according to the applications.
The soft key functions are displayed at the bottom of the CRT screen.

Left-end soft key : This key is used in order to exit to the initial state (condition when the function button is depressed) when each feature has been operated via soft keys.

Right-end soft key : This key is used when operate functions which have not yet been displayed.

(NOTE)
This function is valid only when optional graphic display and figure conversational input are selected

CURSOR keys:
For specifing the starting block in memory operation, the block in editing operation, the word and etc.

PAGE key:
For changing the page of CRT display.

POWER:
When the no fuse breaker in NC box is turned ON, this lamp is turned on.

READY ON:
For indicating that the servo power is ready, hydraulic pump motor is ON and various operations are ready to perform.

CHUCK CLOSE:
For indicating that the chuck is closed.

START READY:
Lamp is turned on in the situation that automatic operation is possible.

NC ALARM:
For indicating when a trouble occurs in NC unit included servo system, or when data processing operation of program is mistaken.
In this case, message will be displayed on CRT screen.

MACHINE ALARM:
When a trouble occurs on machine side, this is turned on.

LEVEL DOWN:
When lubrication oil, coolant (O.P) hydraulic oil is not enough volume or air pressure becomes too low, this is turned on.

INVERTER ALARM:
Lamp is turned on when abnormality related to inverter unit occurs.

MACHINE END:
In case of emergency stop condition as slide systems reach machine end, lamp is turned on.

EDIT:
For storing reading out and editing the program

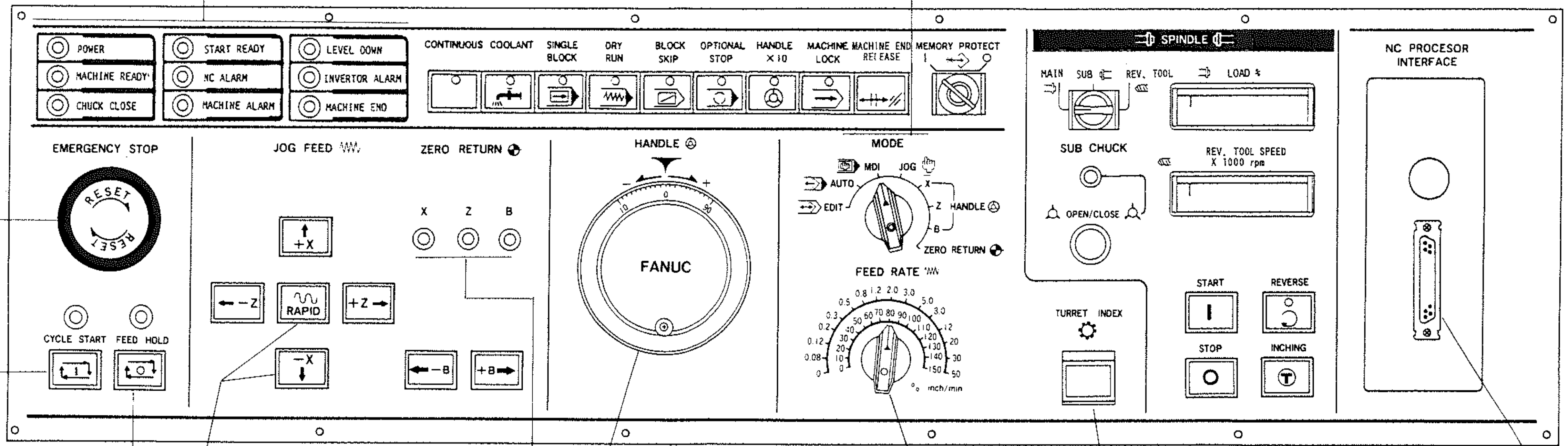
AUTO:
For running by memory

MDI:
For operating manual data input

JOG:
For operating various function manually

HANDLE:
For selecting the operating axis by handle

ZERO RETURN:
For using when operating the manual zero return



EMERGENCY STOP:
For stop the machine completely

CYCLE START:
When mode selection is selected in automatic cycle or MDI operation, if this button is pressed, automatic cycle is started and a lamp on the automatic cycle start button is turned on.

FEED HOLD:
When this button is pressed during automatic cycle, automatic cycle is suspended and the lamp on the feed hold button is turned on.
Operation can be restarted by pressing automatic cycle start button.

JOG FEED:
When a manual feed button of slide and a rapid feed button are pressed at the same time, the slide moves at rapid feed.

ZERO RETURN:
For indicating that the slide returns to the reference point

HANDLE
(manual pulse generator) :
In case of handle mode, selected axis moves by the amount of turning of this dial.

FEED RATE:
For setting the jog feed rate.
In case of automatic running, feed override is possible by this dial.
When this dial is 100% during automatic operation, this becomes the same as program command value.

NC PROCESSOR INTERFACE:
In case of input/output of program, offset and parameter by NC processor, this is used.

TURRET INDEX:
When this button is pressed during mode selection of JOG, handle and zero return, the turret is indexed.
Indexing direction is forward direction.
When this is pressed one time, index of one position is performed. When it is pressed continuously, continuous indexing is not performed.

CONTINUOUS Continuous operation or one cycle operation is selected.
ON : Continuous running.
OFF: One cycle running.

COOLANT For operating the coolant pump ON-OFF
ON : Effective in manual mode
In automatic mode, depending on the program.
OFF: Effective in any mode.

SINGLE BLOCK When this lamp is turned on during automatic operation, one block of program is executed and the machine stops.

DRY RUN For selecting the feed command on the program in either effective or ignored
ON : Both rapid feed and cutting feed speed is regarded as setting speed on the JOG FEED RATE dial.
OFF: Feed speed is depending on the program.

BLOCK SKIP
ON : A block with / is skipped.
OFF : It becomes the same as without /.

OPTIONAL STOP
ON : Program stops at the block containing M01.
OFF: It does not stop at the block containing M01.

HANDLE ×10
ON : A slide moves at the magnification of 10 times of dial indication scale during handle mode.
OFF : A slide moves at the magnification of one time of dial indication scale during handle mode.

MACHINE LOCK
ON : Without moving a slide, the program can be executed.
However, M, S and T codes are executed.

MACHINE END RELEASE
When the turret slide reaches to the machine end and emergency stop condition is realized, this is used for release.
While this button is pressed (Lamp turned on) continuously, this is valid.
Release procedure of machine end condition.
1. Push the machine end release button, and hold it. (Servo system and hydraulic pump is turned on.)
2. Move the slide the opposite direction by jog feed button or handle.
3. Release the machine end release button.

MEMORY PROTECT
I : The program cannot be changed.
O : The program can be changed.

LOAD %
This load meter indicates load of spindle driving motor during cutting.
When the pointer of loadmeter indicates less than 100%, it indicates that the load is withstandable continuously by spindle driving motor.

REV. TOOL SPEED X 1000 rpm
Displays number of rpm of revolving tools in manual and automatic modes.
Switch to select unit to be rotated in manual mode.

START For starting the spindle in manual mode.

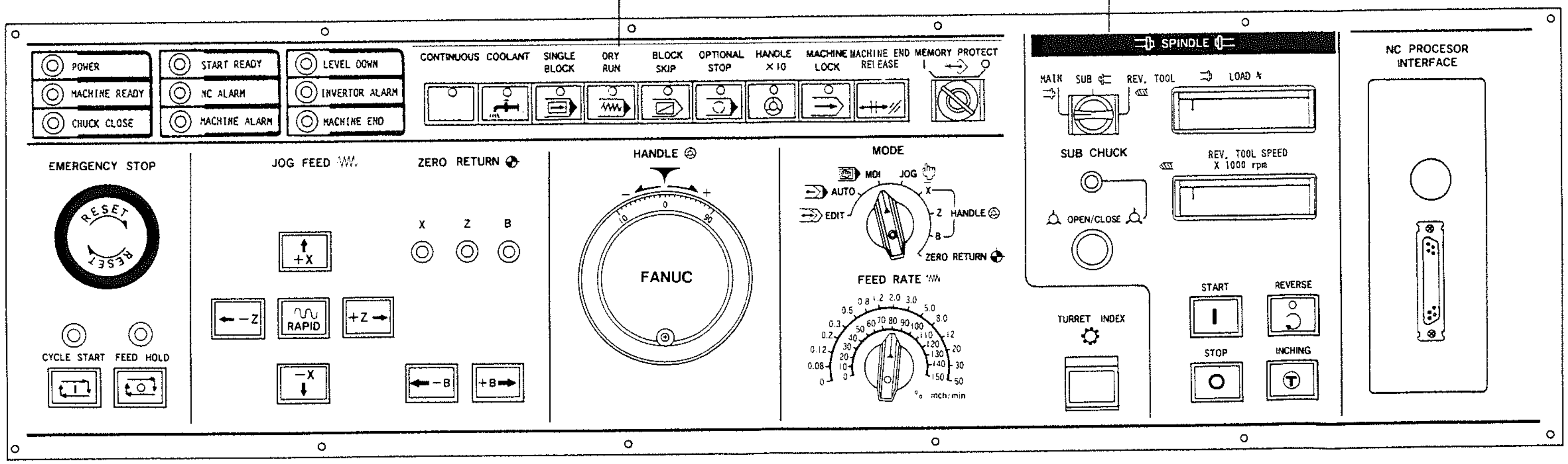
REVERSE
ON : Spindle reverse rotation under manual mode
OFF : Spindle forward rotation under manual mode

INCHING For inching the spindle in any mode

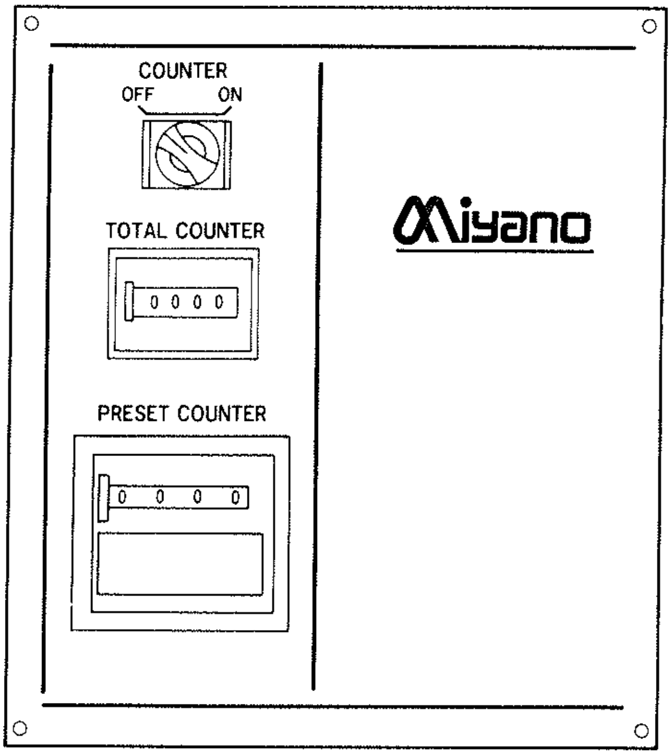
STOP For stop the spindle in manual mode

MAIN SUB REV. TOOL Spindle selection in manual mode, this switch selects main spindle or sub-spindle or revolving tool.

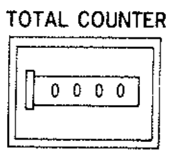
SUB CHUCK OPEN/CLOSE Push button for chuck open or close of sub-spindle.
Sub chuck indicator lamp turned on when chuck is closed.



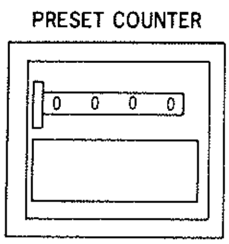
2. OPTIONAL PANEL



ON or OFF of total and preset counter is selected.
OFF: No count.
ON : Count.



Total number of work piece is counted.



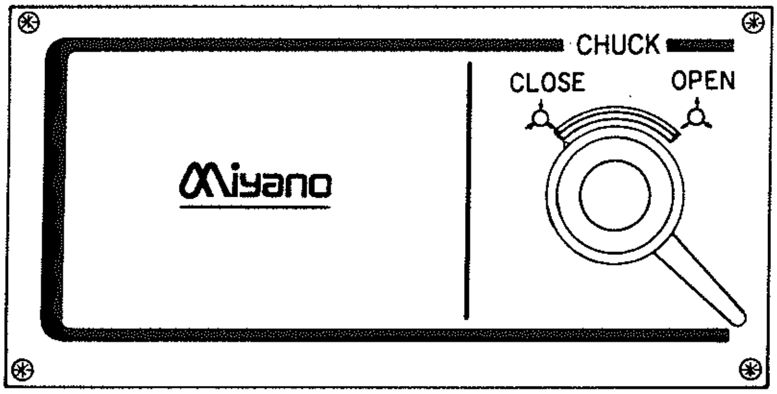
A preset value can be set by preset counter.

NOTE) An optional panel differs depending on the option specification.

3. MANUAL OPERATION SWITCH

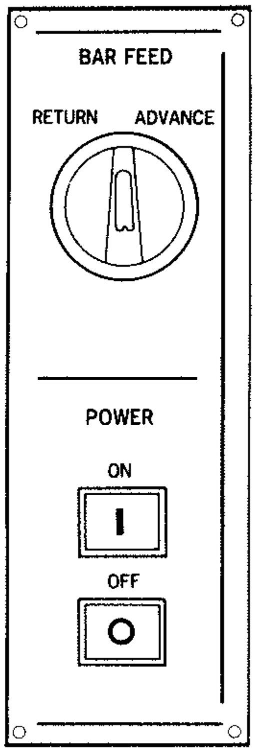
(1) Selector switch for "closing" and "opening" of main spindle chuck.

Selector switch for open/close of main spindle chuck. Effective in all modes. Chuck closing lamp is turned on and off by this switch.



(2) Bar feeding "return" "advance" selection switch

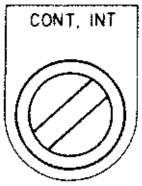
Bar feed and bar feed return are executed by this selection switch in manual mode.



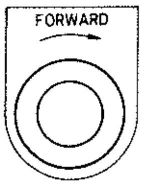
BND-S OTC OPE 90/7

(3) Switches relating to intermittent chip conveyor (option)

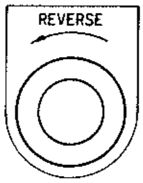
Cutting oil is deposited with chips and conveyor will carry to outside of machine when the chip conveyor is operated continuously. Operate intermittently when cutting oil is especially inhibited to be carried outside.



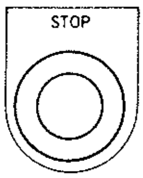
Selection switch for continuous and intermittent operation
This switch selects continuous or intermittent operation of conveyor.



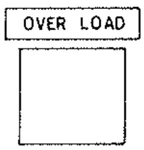
Forward direction start switch
Chip conveyor starts forward direction by pushing this button.



Reverse direction start switch
Chip conveyor starts reversely by pushing this button. It shall be used when chips are accumulated in the chip conveyor.



Stop push button switch
Chip conveyor stops by pushing this button.



Overload switch:
When a chip conveyor is overloaded, this lamp is turned on.

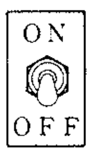
(4) Parts conveyor "ON" "OFF" toggle switch , start/stop of parts conveyor

1) In MDI and automatic operation

When parts conveyor toggle switch is selected in "ON" position and M213 is commanded, parts conveyor starts, and stops when M215 is commanded.

2) In jog mode

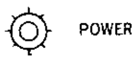
By turning "ON" the toggle switch of parts conveyor, the parts conveyor starts and stops when turn it "OFF".



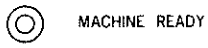
4. OPERATIONAL PROCEDURE

4-1 Explanation of Expressional Symbols

Expressional symbols in this manual are used as follows;



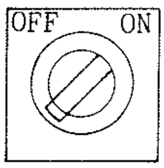
---------- Shown as indicating lamp is ON

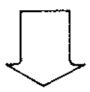


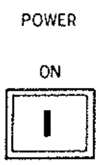
-------- Shown as indicating lamp is OFF

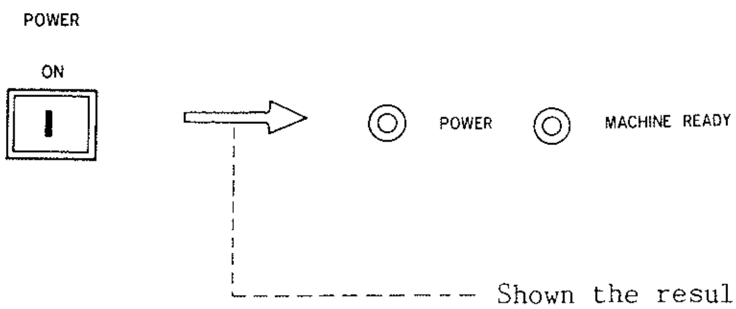
-------- Shown as depressing the switch in order left to right

--------- Shown as depressing both switches simultaneously.




---------- Shown the order of operation

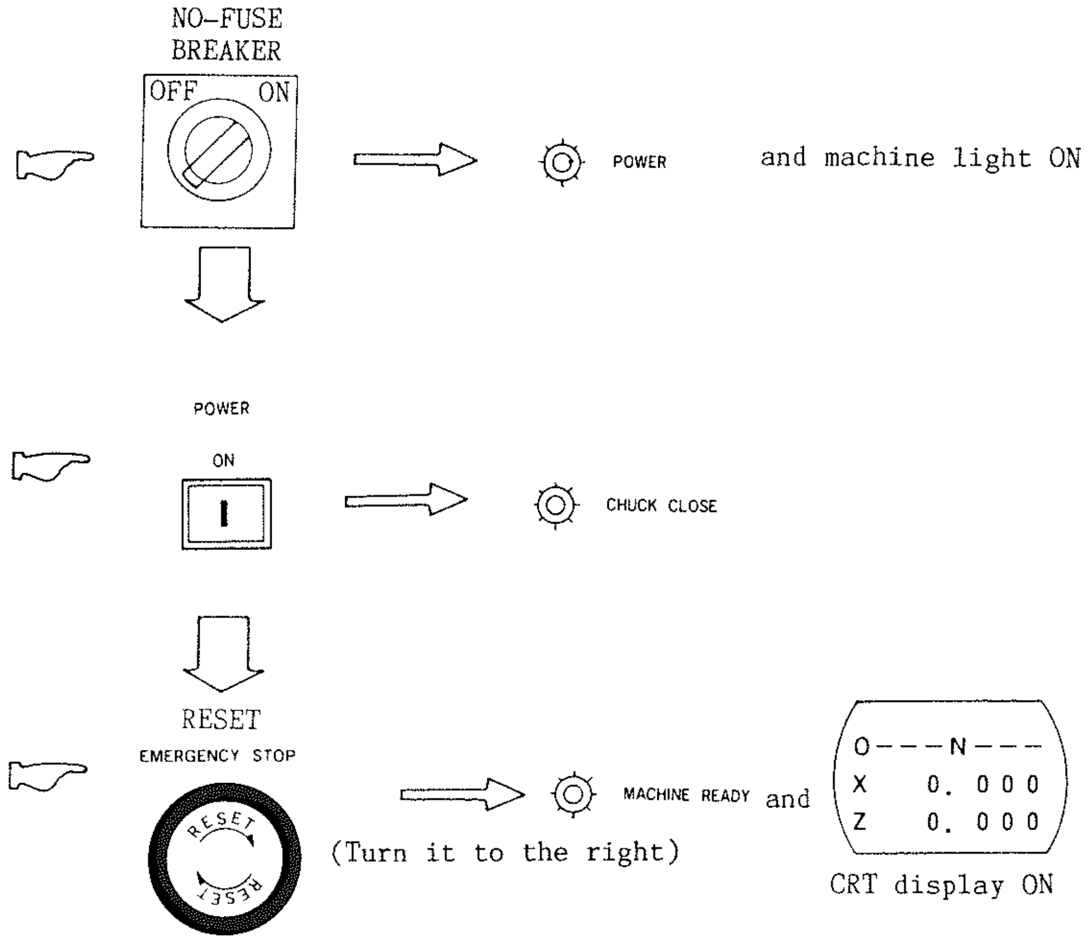




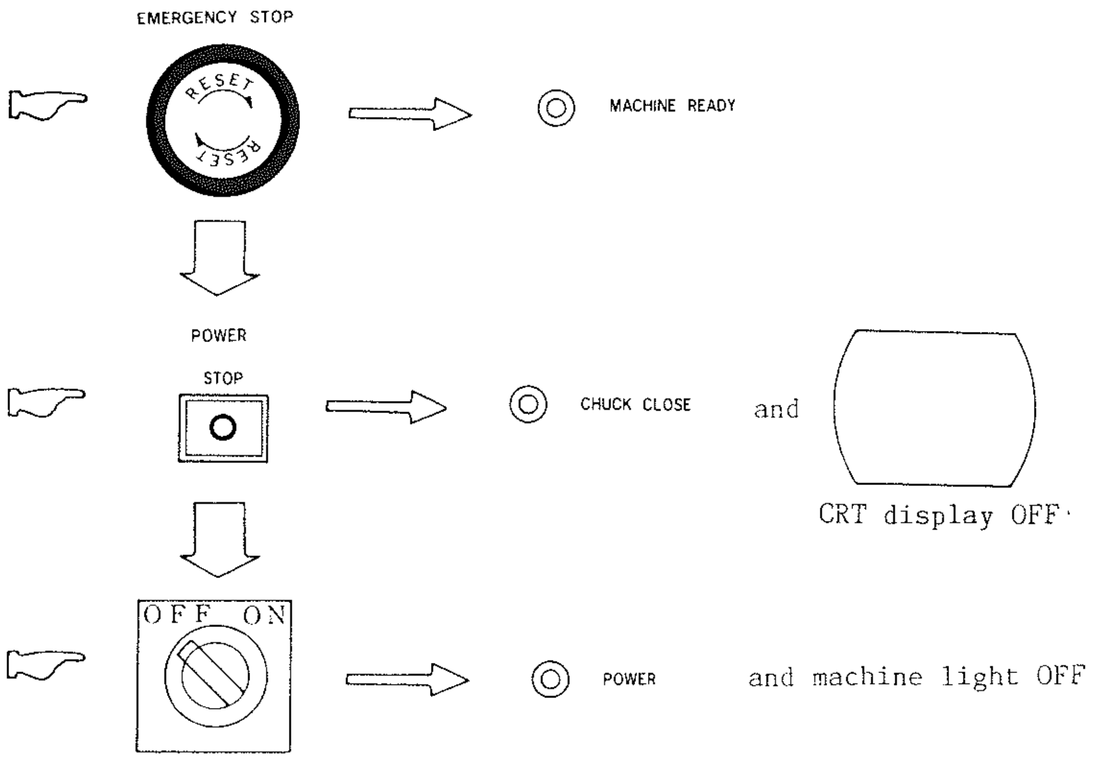
---------- Shown the result of operation

4-2 Operating the Power ON/OFF

[1] Turning power ON



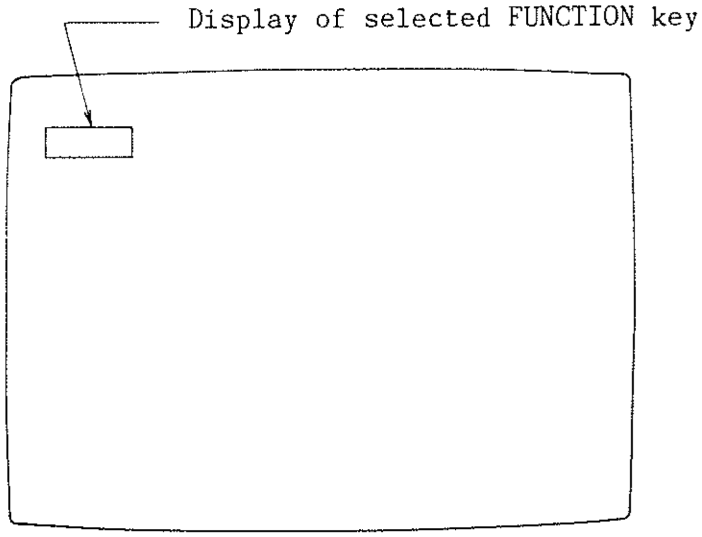
[2] Turning power OFF



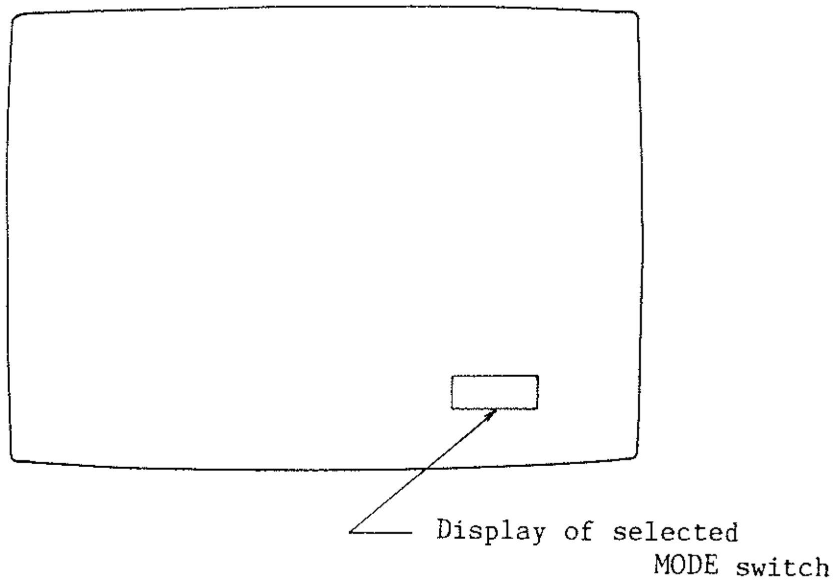
BND-S OTC OPE 90/7

4-3 Explanation of Indicated Location

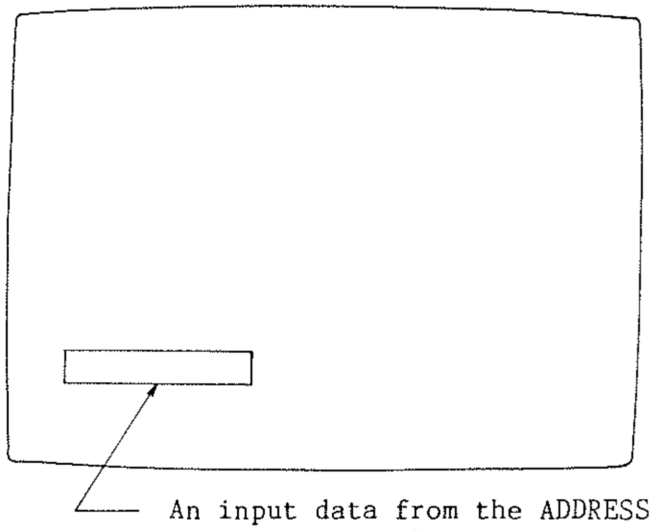
[1] Display of FUNCTION



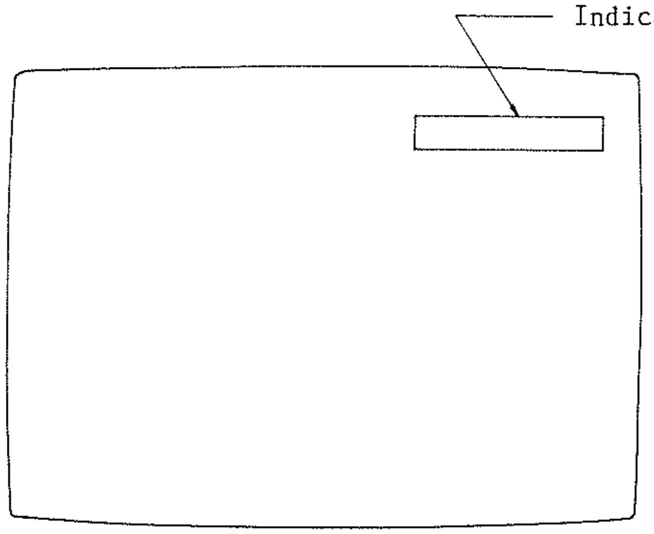
[2] Display of MODE



[3] Display of input data by using keys

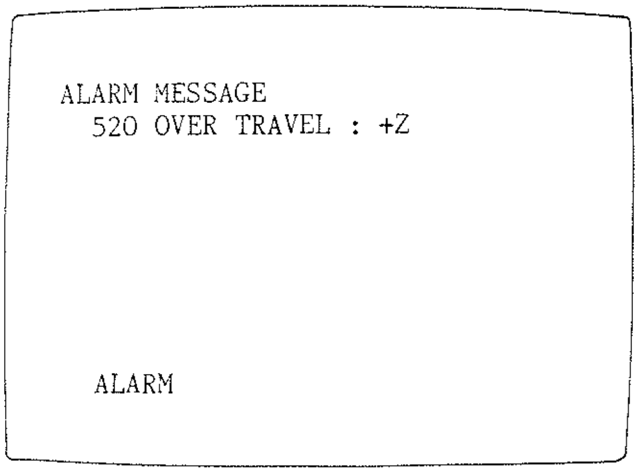


[4] Display of program number and sequence number

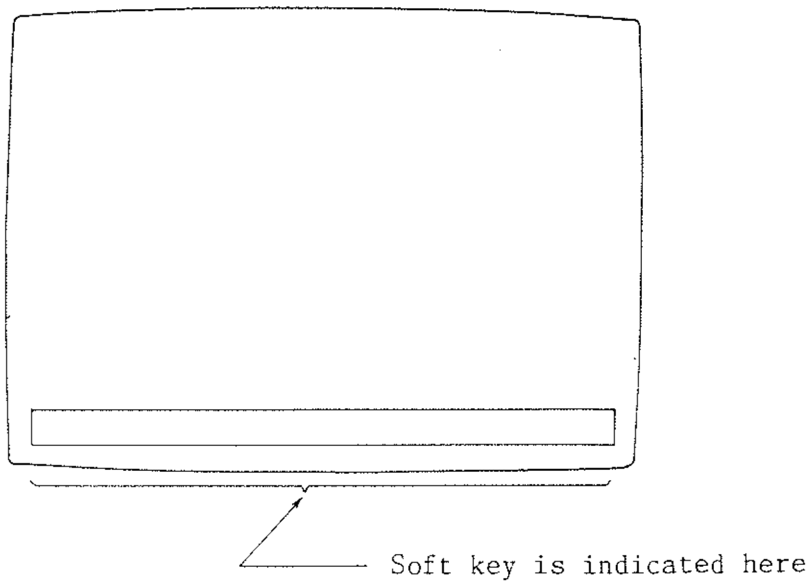


[5] Display of alarm

When an alarm occurs "ALARM" is displayed at bottom, press the ALARM button, the alarm message is displayed as shown below.



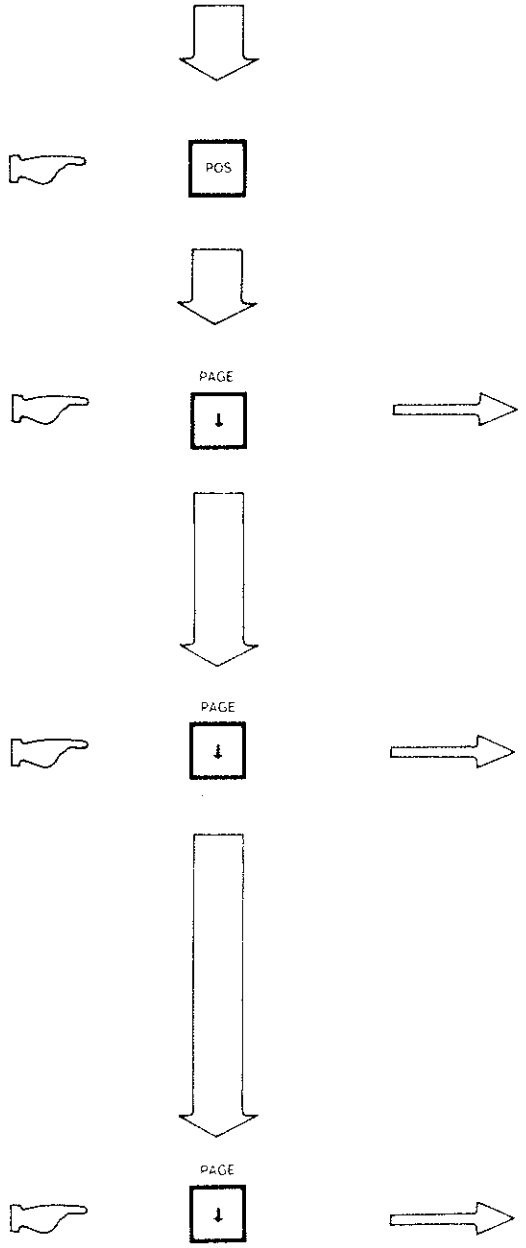
[6] Indication of soft key (Option)



(Note 1)
This function is valid only when optional graphic display and figure conversational input are selected.
(Note 2)
Although this explanation document is described as soft key display, display to this section is not plovided when above-mentioned option is not selected.

4-4 Procedure of Current Position Display

If the value of tool position is not set, set by MDI operation.



Position display in the work coordinate system

```
ACTUAL POSITION(ABSOLUTE)

   O----          N----
   X----
   Z----
   B----

                     PART COUNT
RUN TIME  H  M   CYCLE  TIME H  M  S
[ ABS ] [ REL ] [ ALL ] [     ] [     ]
```

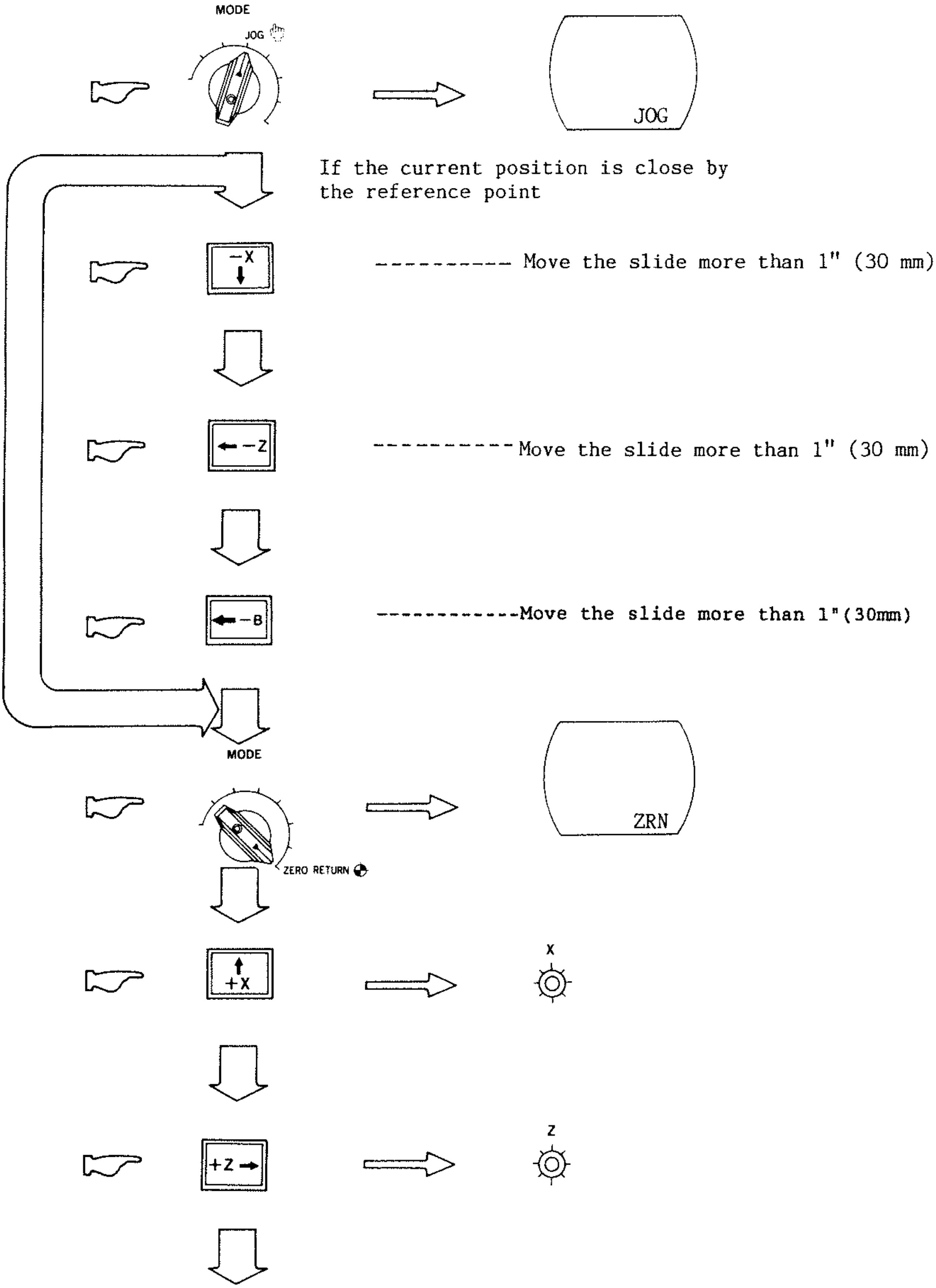
Position display in the relative coordinate system

```
ACTUAL POSITION(RELATIVE)

   O----          N----
   U----
   W----
   B----

                     PART COUNT
RUN TIME  H  M   CYCLE  TIME H  M  S
[ ABS ] [ REL ] [ ALL ] [     ] [     ]
```

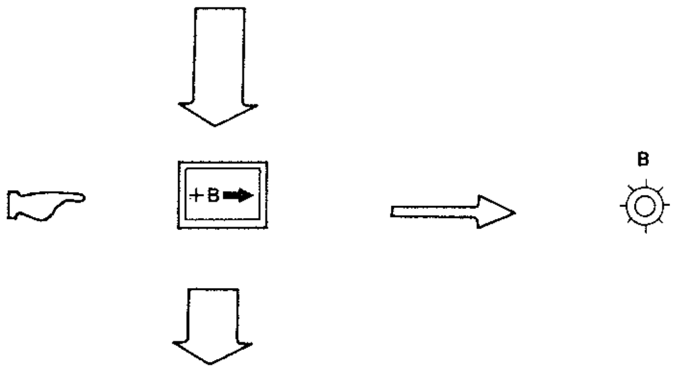
```
ACTUAL POSITION:             O0001 N0052
  (RELATIVE)               (ABSOLUTE)
  U     0.001              X     0.000
  W     0.001              Z     0.000
  B     0.000              B     0.000
  (MACHINE)            (DISTANCE TO GO)
  X     0.000              X     0.000
  Z     0.000              Z     0.000
  B     0.000              B     0.000
                     PART COUNT
RUN TIME  H  M   CYCLE  TIME H  M  S
[ ABS ] [ REL ] [ ALL ] [     ] [     ]
```

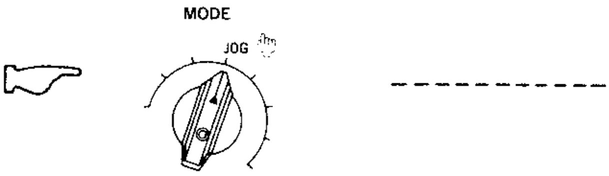
RELATIVE : Position in the relative coordinate system.
ABSOLUTE : Position in the work coordinate system.
MACHINE : Distance from the reference point.
DISTANCE TO GO : Residual movement amount for commanded amount.

4-5 Manual ZERO RETURN



BND-S OTC OPE 90/7



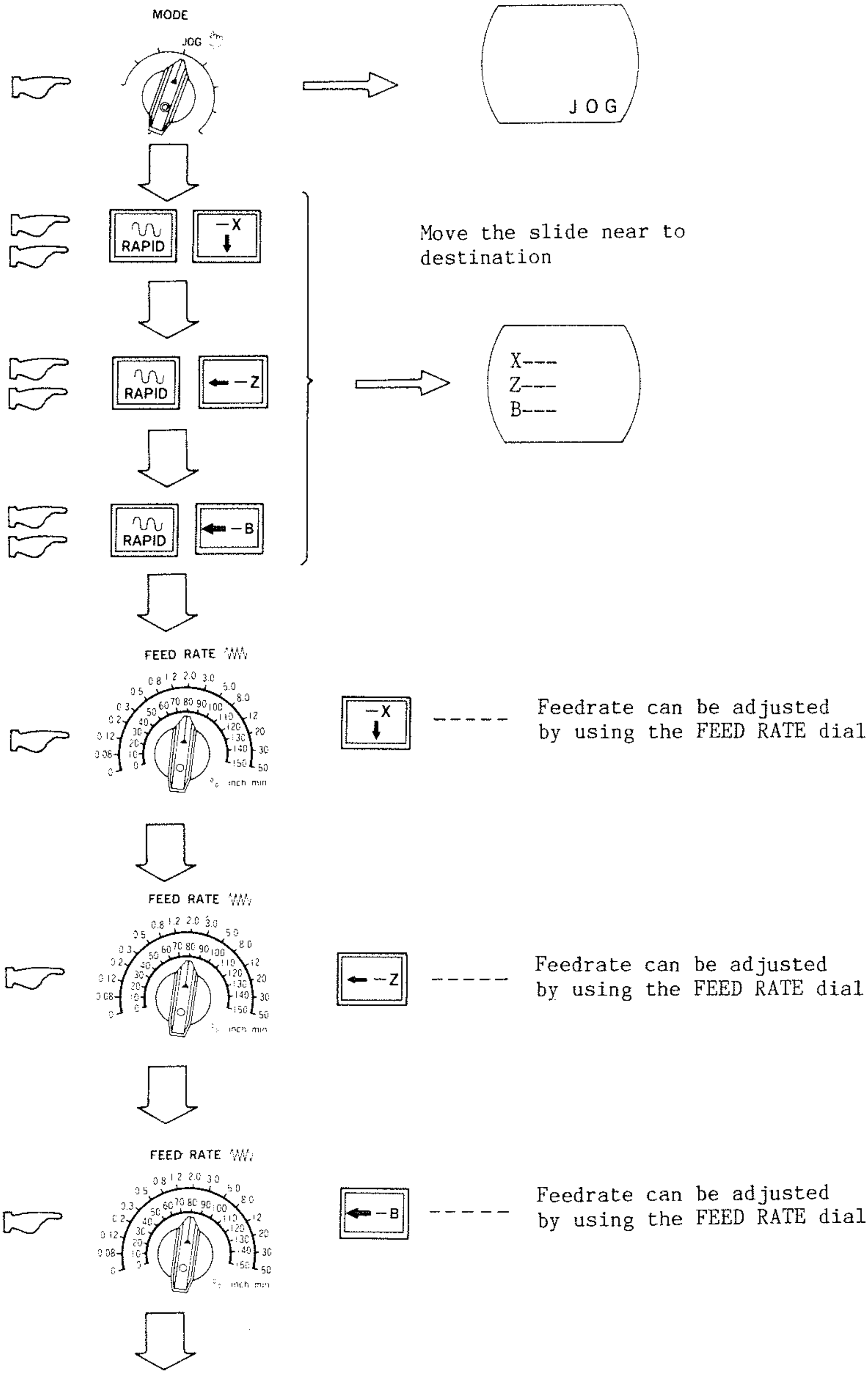


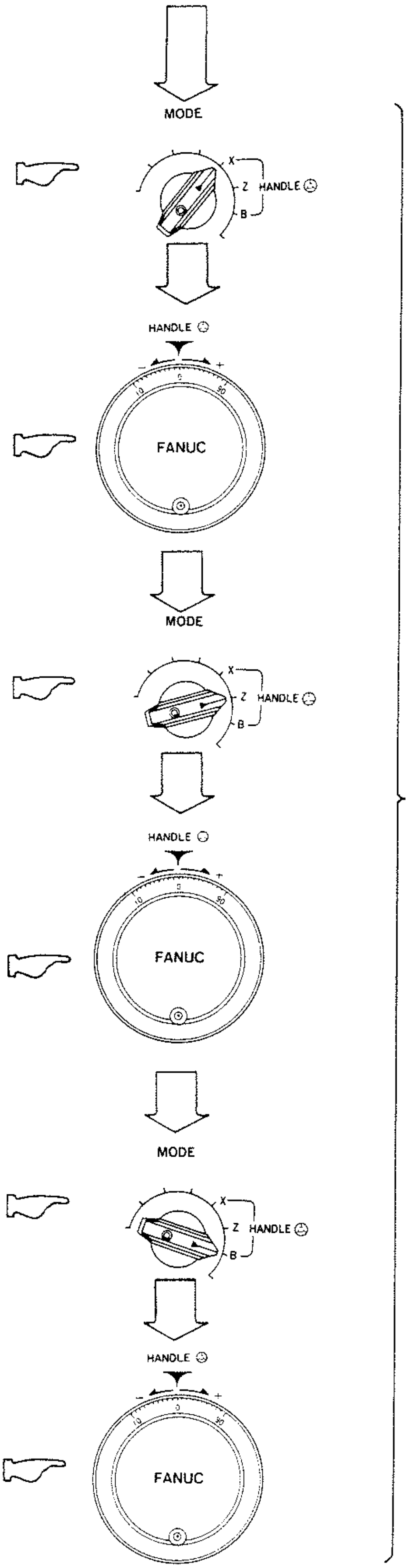
Set the switch other than ZERO RETURN mode

(Note)

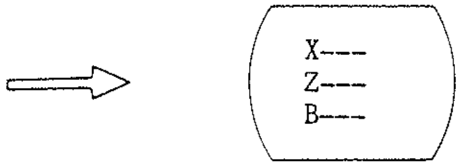
The manual ZERO RETURN must be performed after turning the power on. Otherwise, automatic running can not be performed.

4-6 Positioning by Manual Operation

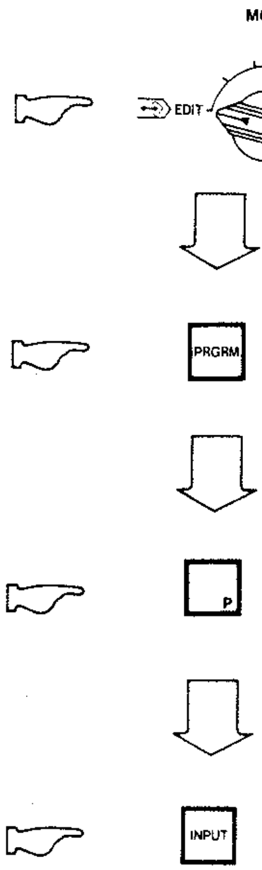




Each slide position can be confirmed by position display



4-7 Display of the Amount of Program Memory Used



PROGRAM NO. USED: The number of the programs registered (including the sub programs).

FREE: The number of programs which can be registered additionally.

MEMORY AREA USED: The capacity of the program memory in which data is registered (indicated by the number of characters)

FREE: The capacity of the program memory which can be used additionally (indicated by the number of characters).

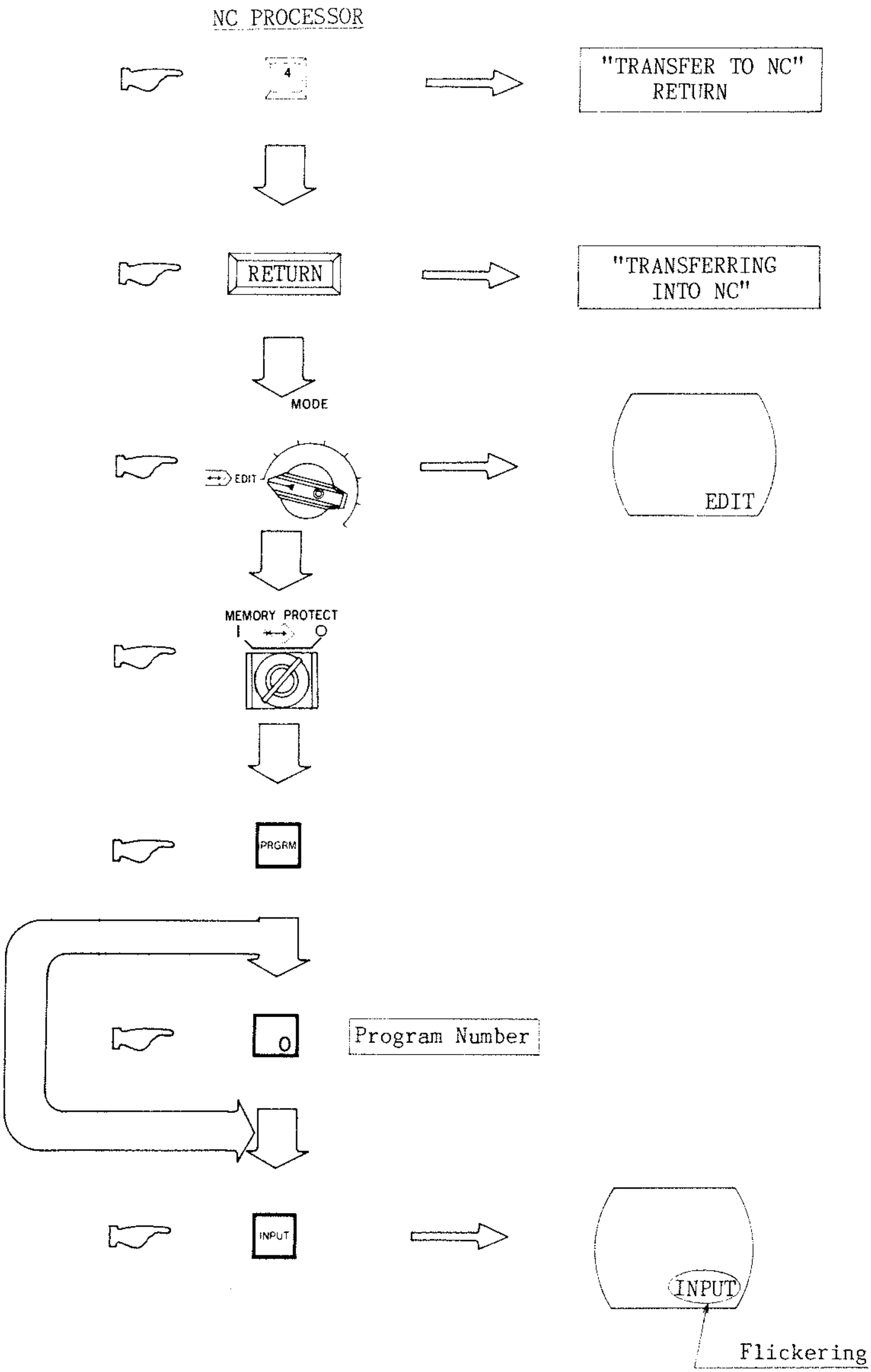
PROGRAM LIBRARY LIST : The nomenclature of the program can be indicated on the program list. In the program nomenclature indicating screen, the program number and the nomenclature of the program are displayed.

```
PROGRAM
   SYSTEM EDITION     0660-04
 PROGRAM NO. USED:    5 FREE:    58
 MEMORY AREA USED:   68 FREE:  4027
PROGRAM LIBRARY LIST
 O0001
 O0002
 O0003(PRO CHECK)
 O0010
 O0020

[PRGRM] [     ] [     ] [     ] [ IAP ]
```

(Note)
Program without program name is displayed only program number.

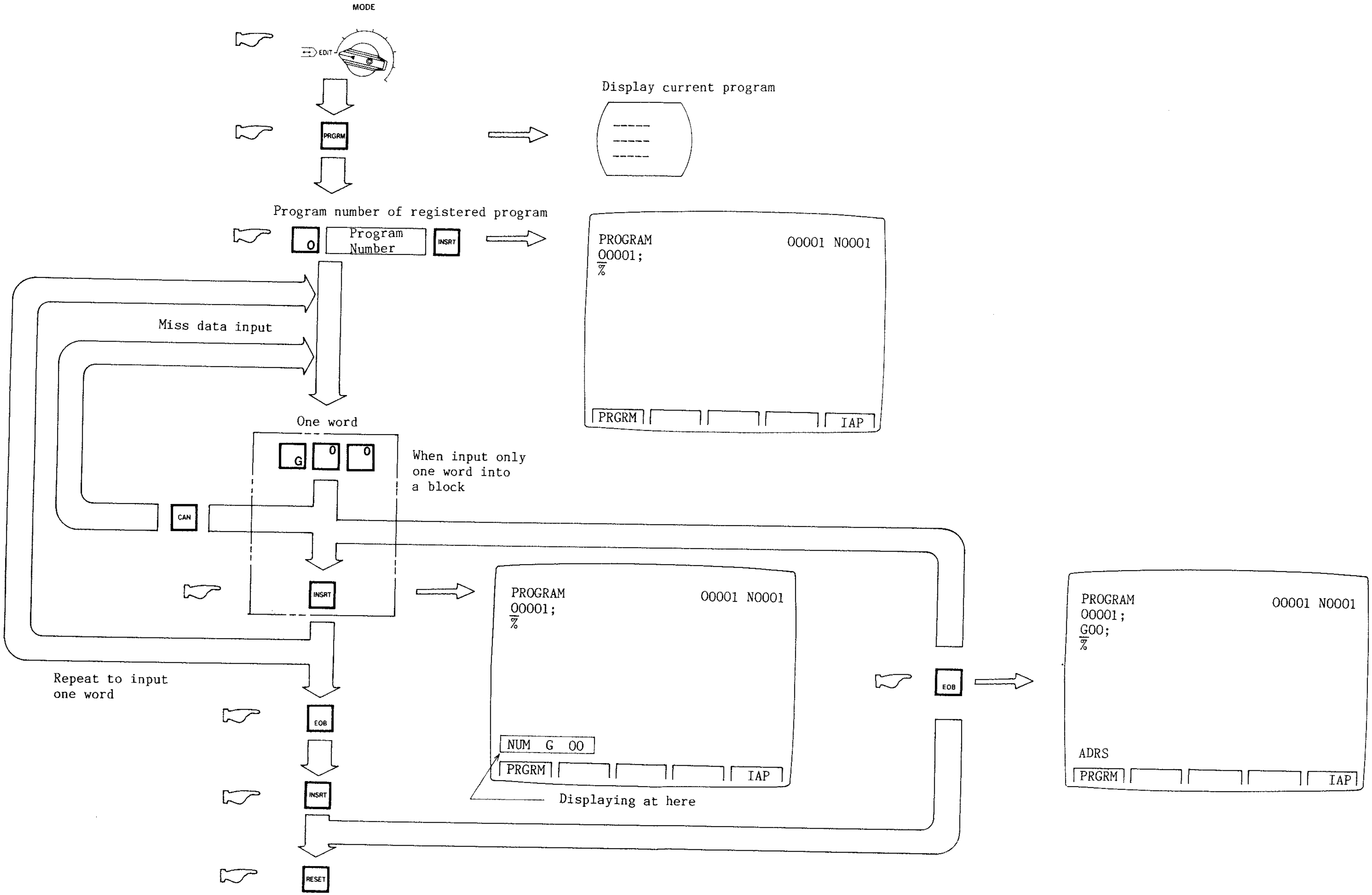
BND-S OTC OPE 90/7

4-8 Storing Data of NC PROCESSOR to Memory



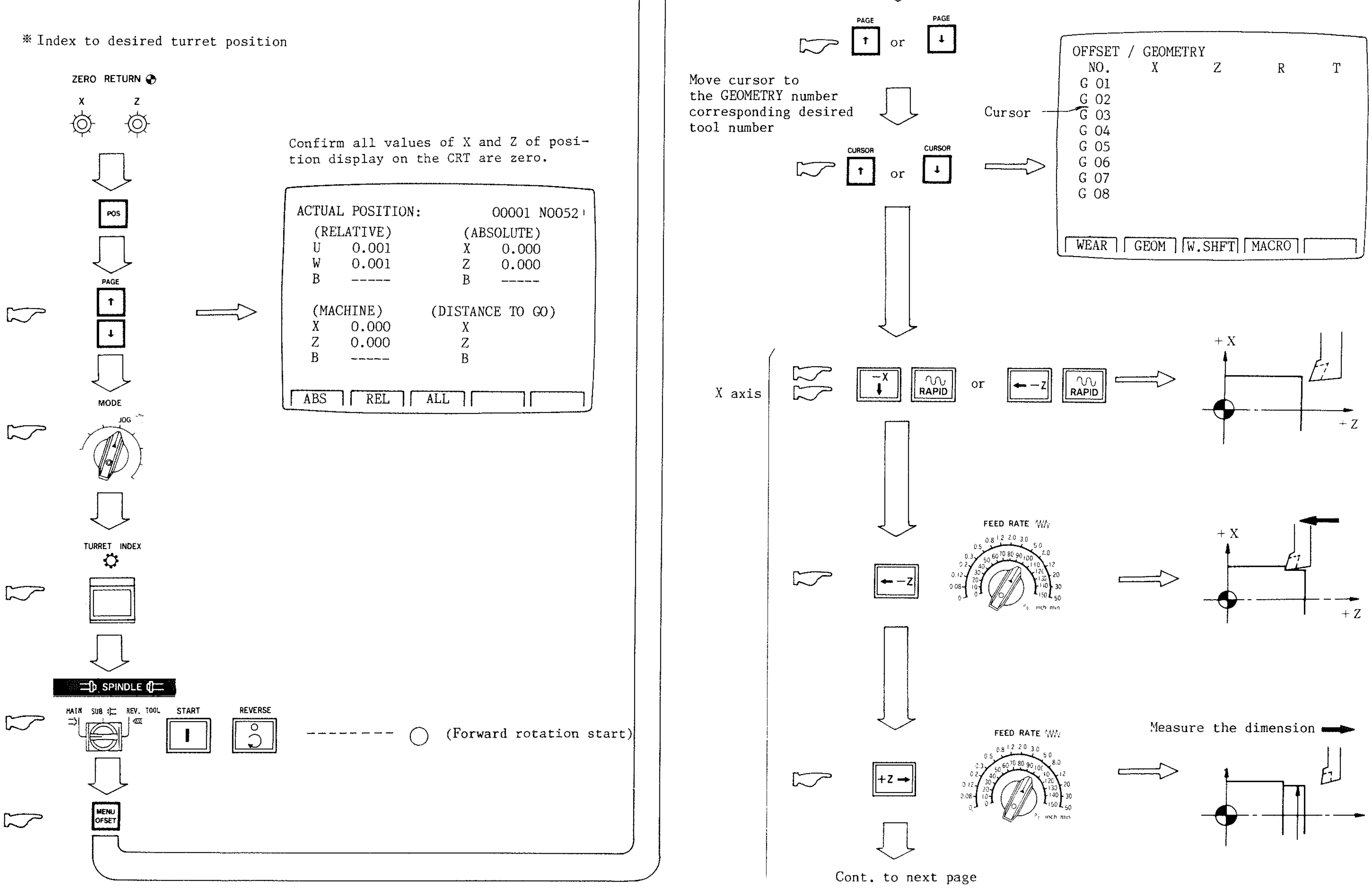
BND-S OTC OPE 90/7

4-9 Storing Part Program By MDI



4-10 Writing of the Work Coordinate System

[1] In case of turning and boring tools



SPINDLE

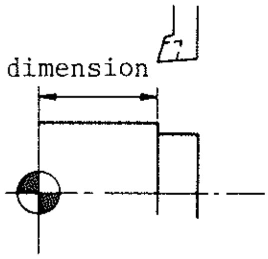
STOP

Input the measured value

M X 2 5 .

INPUT

Set for Z-axis as same procedure as X-axis



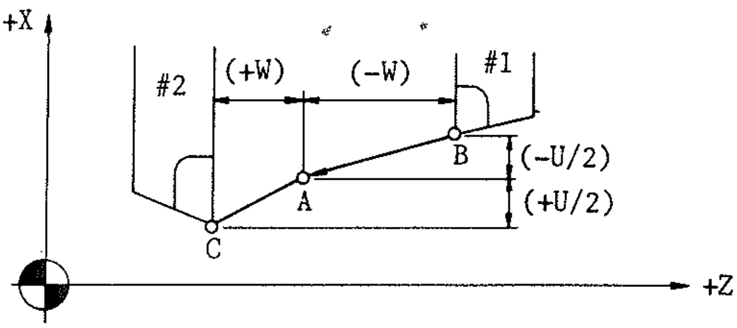
Set other turret position by the same procedure

BND-S OTC OPE 90/7

[3] Alteration of work coordinate system

When increasing or decreasing the coordinate value already set, the address U or W can be used.
By inputting the value with the address U or W, this value will be added to the coordinate value already set. And then, this calculated value becomes new work coordinates.
When the work coordinate system is changed completely, follow the procedure of item [1] of [2].

When setting tool in +X area



MENU OFSET

PAGE ↑ or PAGE ↓

```
OFFSET / GEOMETRY
  NO.      X        Z        R       T
 G 01
 G 02
 G 03
 G 04
 G 05
 G 06
 G 07
 G 08

[WEAR] [GEOM] [W.SHFT] [MACRO] [    ]
```

Move cursor to the GEOMETRY number corresponding desired tool number.

CURSOR ↑ or CURSOR ↓

```
OFFSET / GEOMETRY
  NO.      X        Z        R       T
 G 01
 G 02
Cursor  G 03
 G 04
 G 05
 G 06
 G 07
 G 08

[WEAR] [GEOM] [W.SHFT] [MACRO] [    ]
```

For incremental offset value in X-axis

U

For incremental offset value in Z-axis

W

Numeral

###.###

INPUT

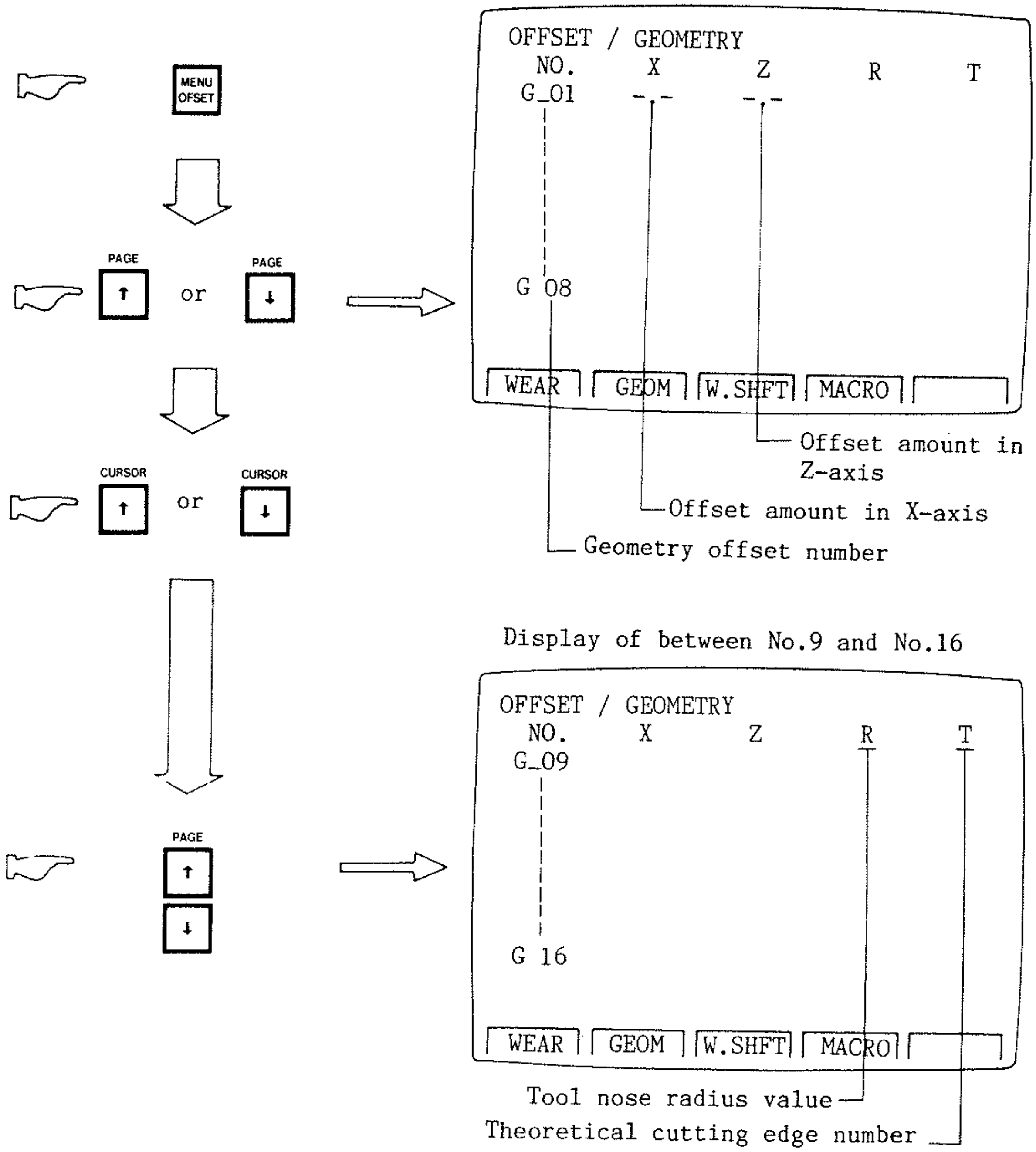
4-11 Display and Setting of Tool Nose-R Compensation

In case of taper cutting or arc cutting, there is a case where a work piece is not finished as programmed dimension due to tool nose R.
Tool nose-R compensation compensates this error automatically.
Tool nose-R compensation is executed by G code (G40 to G42), designation of assumed cutting edge number and tool nose R value.
For the value of tool nose-R, R values of insert is set without a sign.

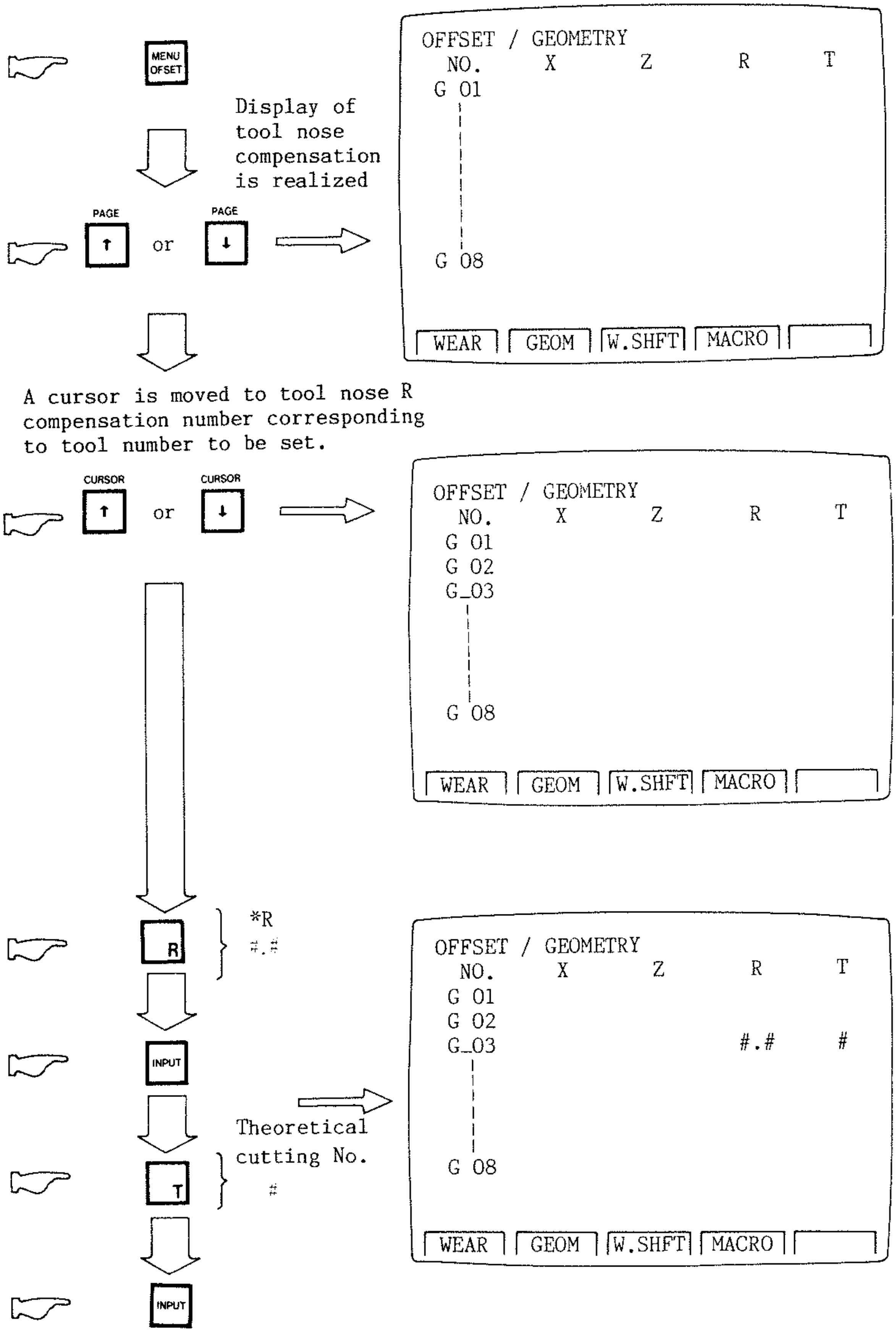
Setting unit and maximum setting value is as follows:

Setting unit .... .0001"(0.001mm)
Maximum setting value of tool nose R .. 99.9999"(999.999mm)

[1] Display of tool nose R compensation



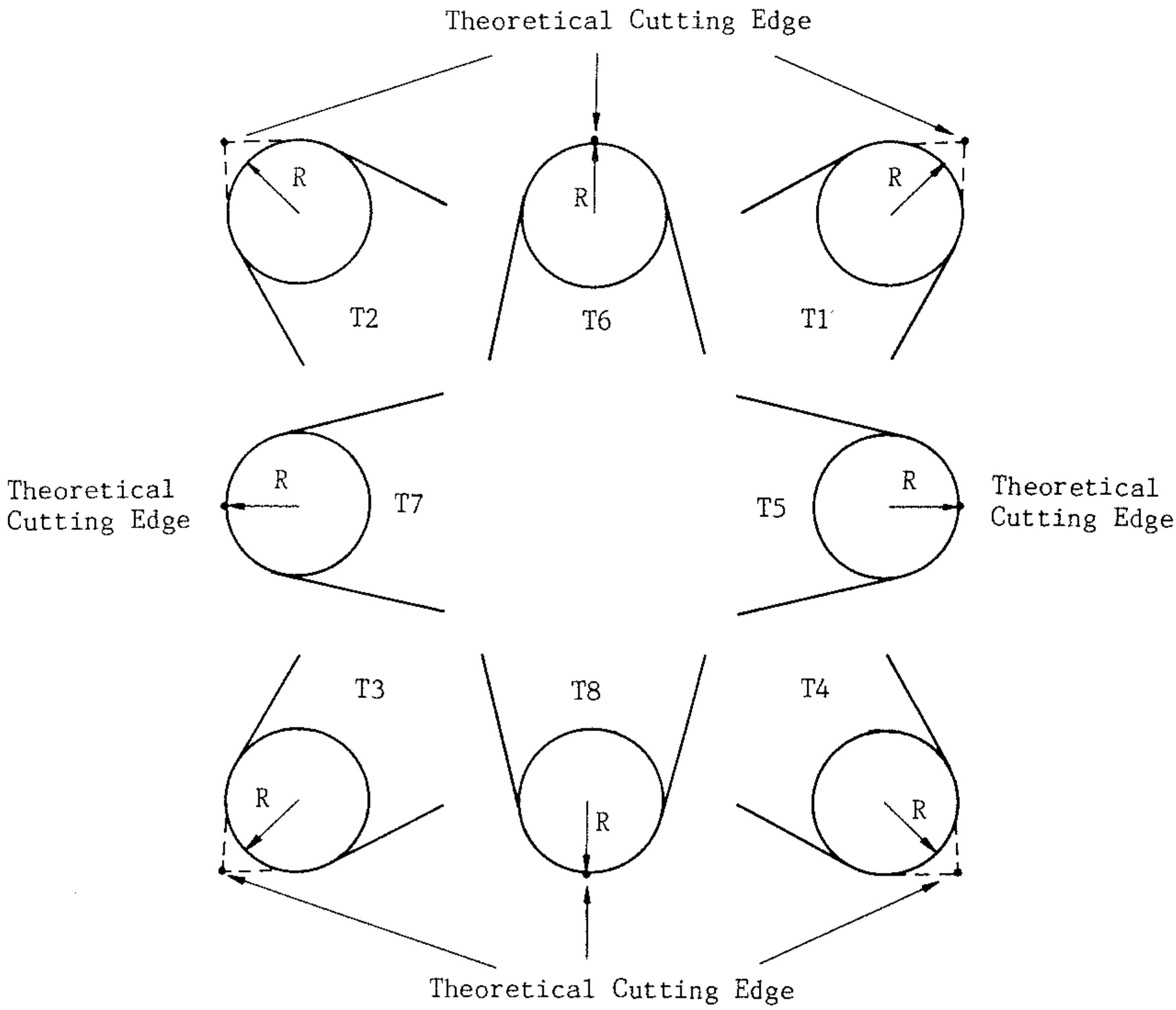
[2] Setting of tool nose R compensation



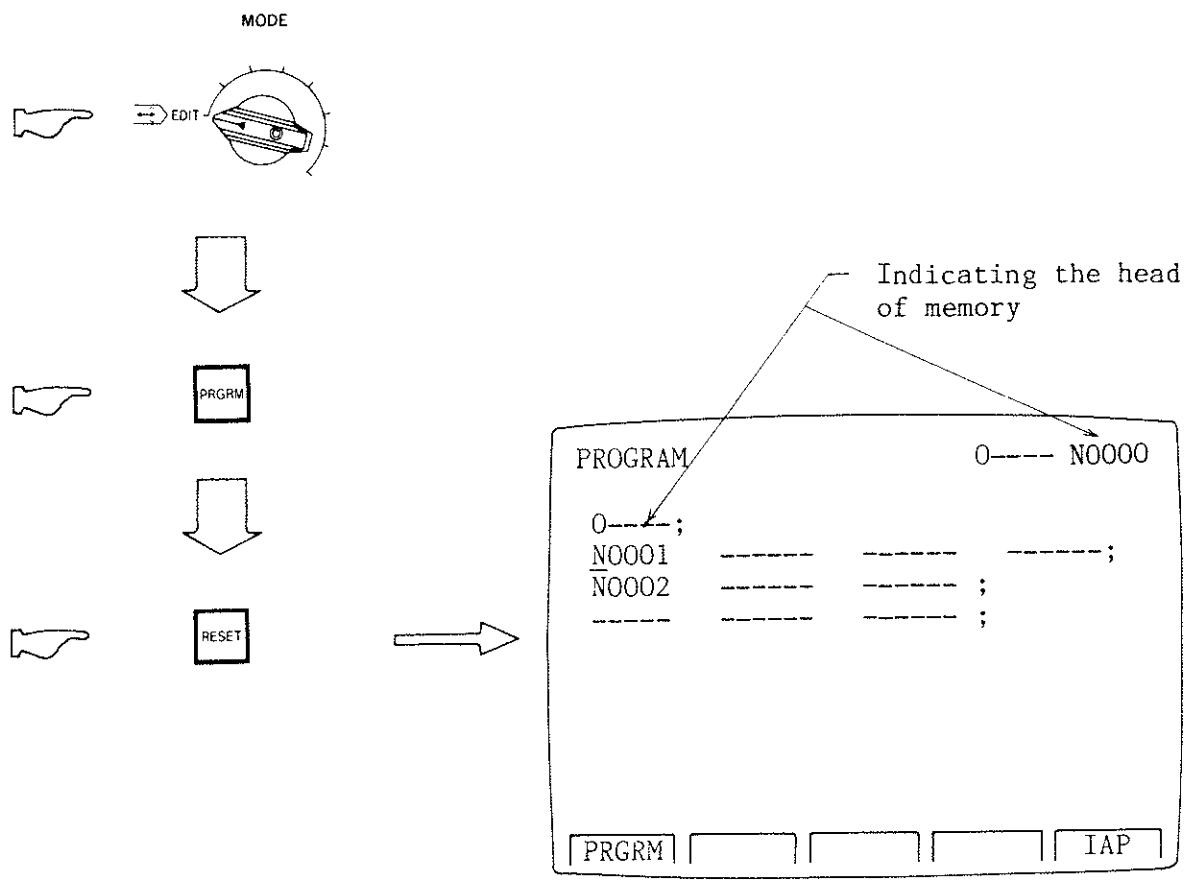
BND-S OTC OPE 90/7

[3] Command of the direction of the theoretical cutting edge

The direction of the theoretical cutting edge is selected from 8 types (T1 to T8) shown in below, depending on the direction of the tool nose center viewed from the theoretical cutting edge, on the programming. This selected number (Standard tool nose number) must be set corresponding to offset number by MDI operation.

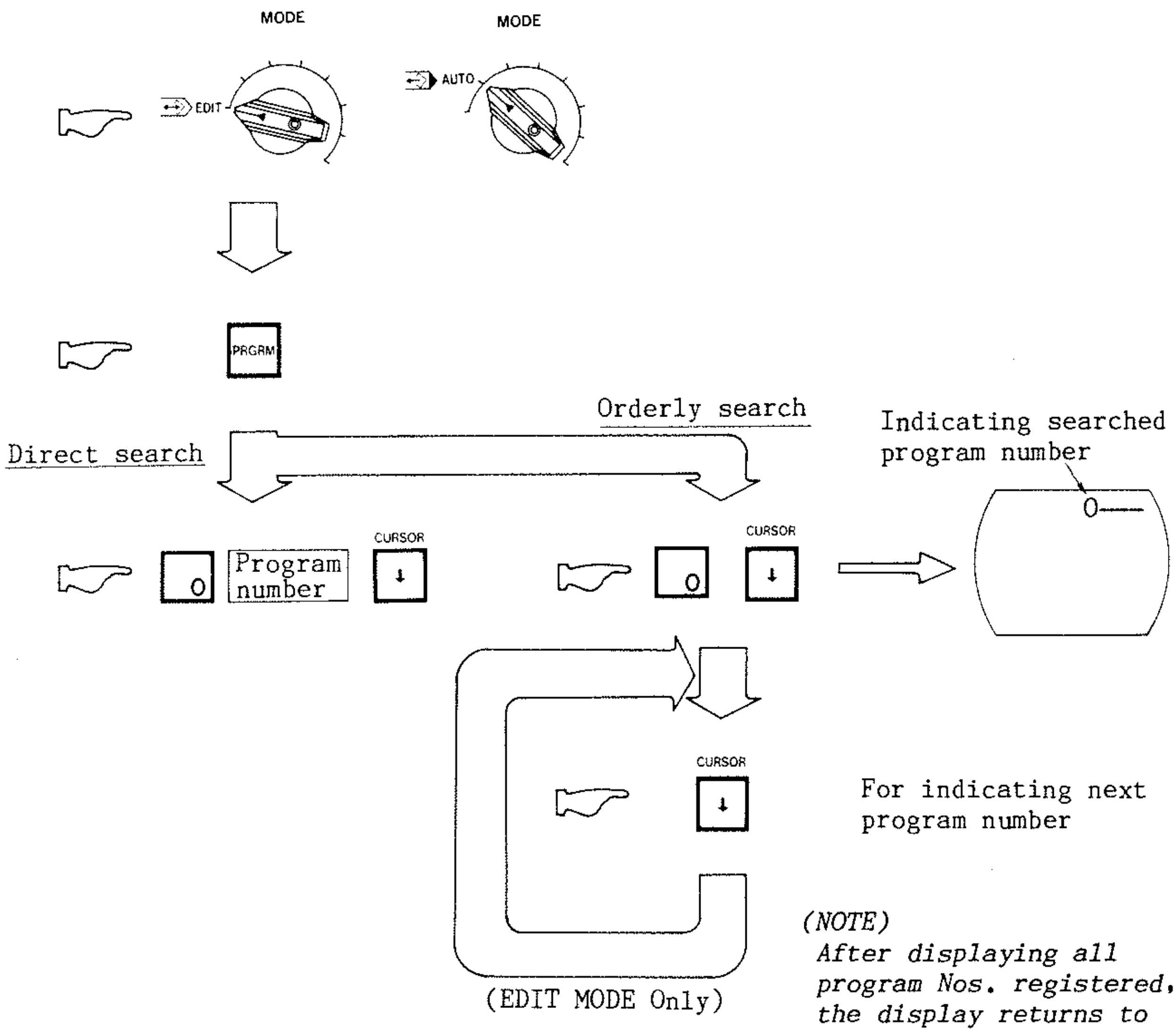


4-12 Returning to the Head of Program



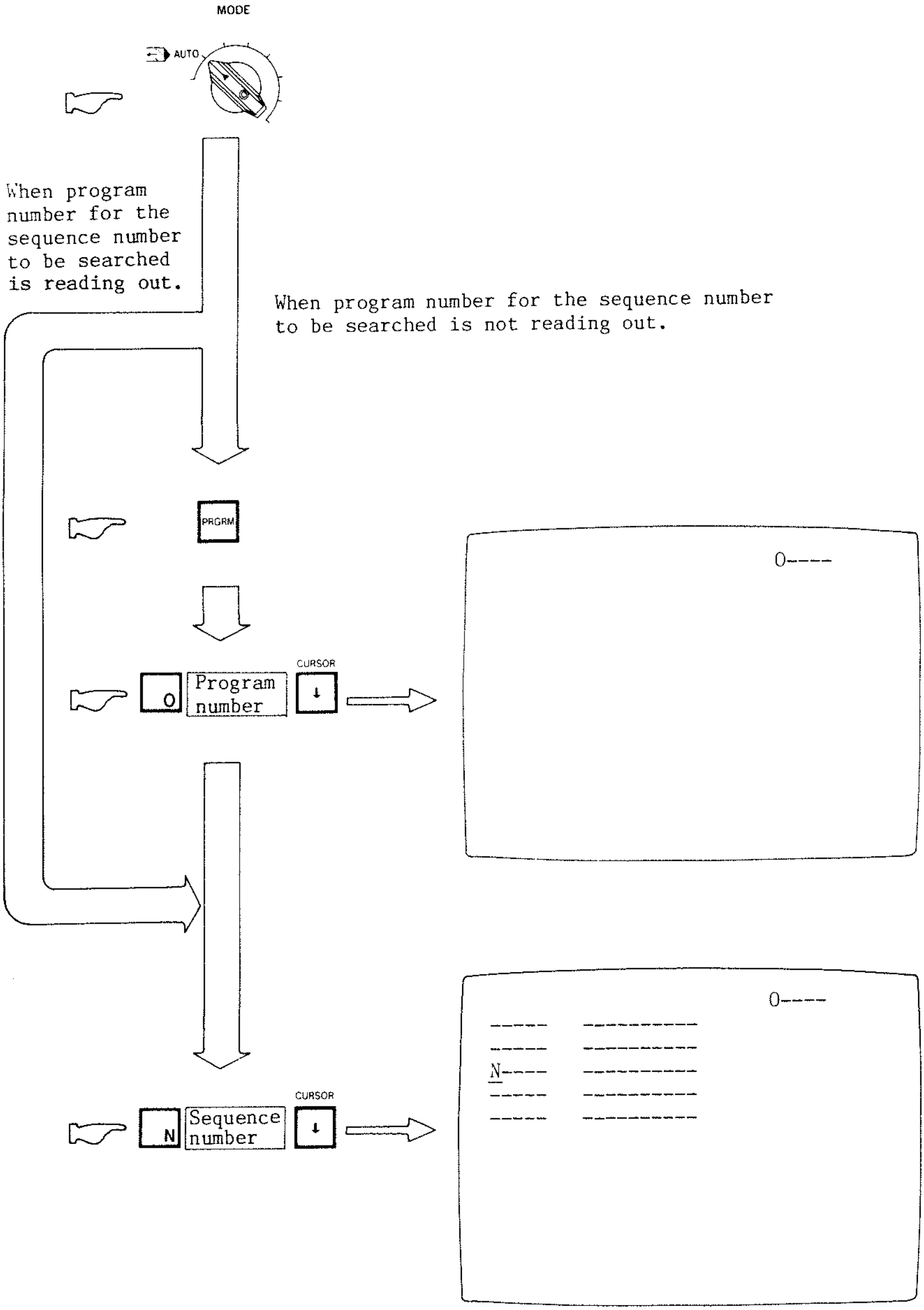
4-13 Search Operation

[1] Program number search

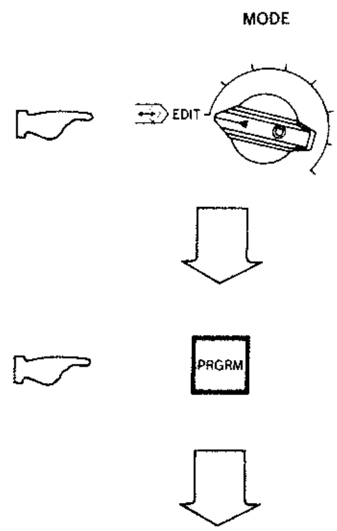


(NOTE)
After displaying all program Nos. registered, the display returns to the first one.

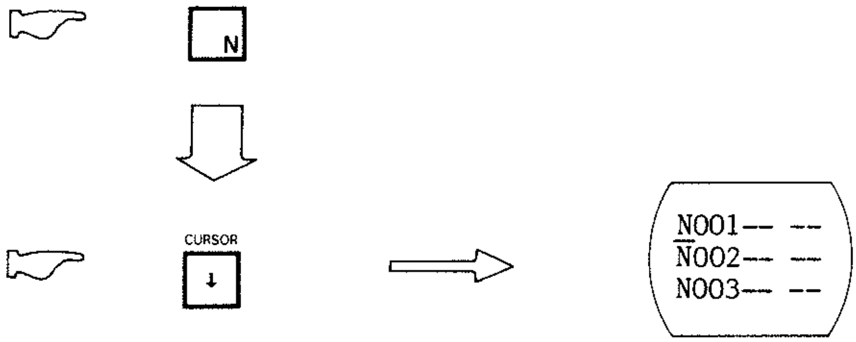
[2] Sequence number search in AUTO mode



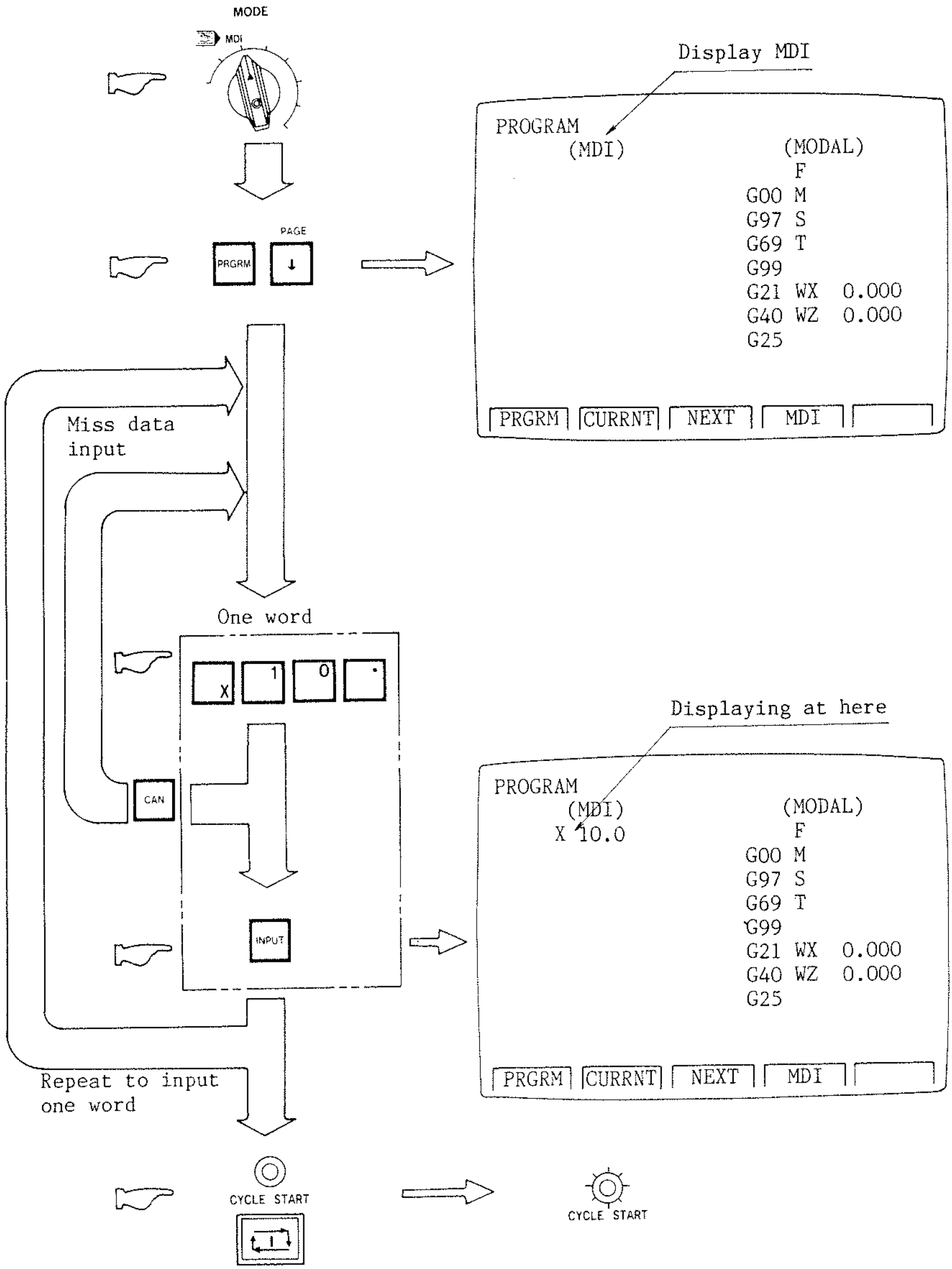
[3] Method of program search by address



In case of searching for address 'N'



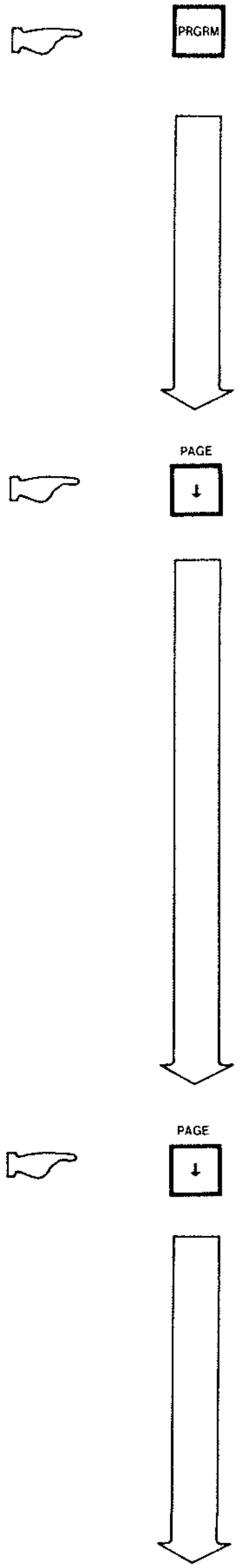
4-14 MDI Operation



[NOTE]

Modal G codes can not be cancelled by using CAN *button.*
Therefore, a correct data should be input again.

4-15 Display of Command Value



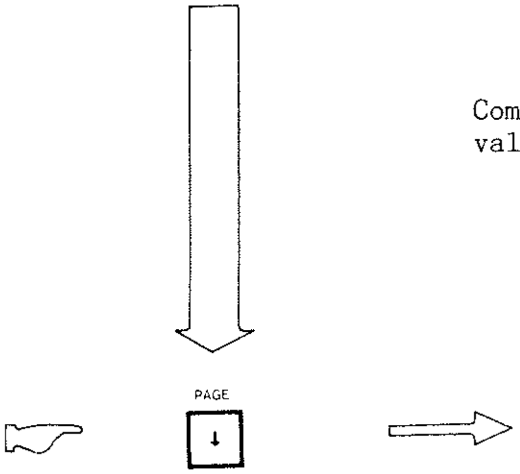
Command values being executed, and modal values previously specified are displayed.

```
PROGRAM
    (CURRENT)                (MODAL)
     X    10.000               F 11.0000
 G01 F   11.0000          G01 M
     T      0303          G97 S
                          G69 T    0303
                          G99
                          G21 WX  0.000
                          G40 WZ  0.000
                          G25

[PRGRM] [CURRNT] [ NEXT ] [ MDI ] [      ]
```

Command values being executed or to be executed next are displayed.

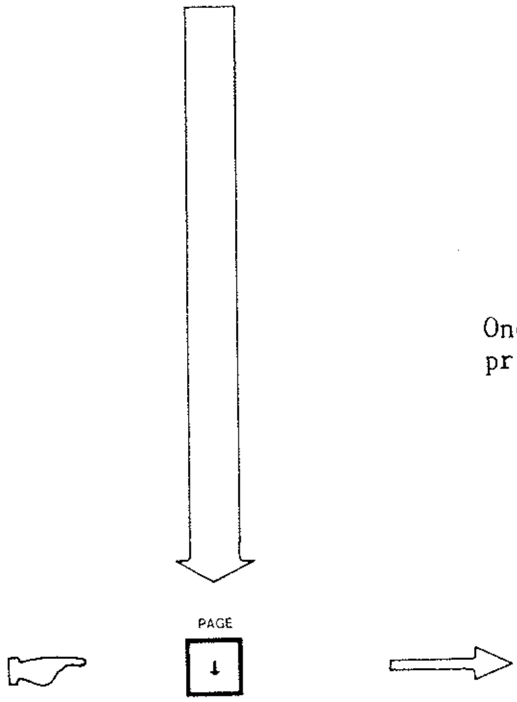
```
PROGRAM
    (CURRENT)                (NEXT)
     X    10.000              Z 100.000
 G01 F   11.0000          G00
     T      0303

[PRGRM] [CURRNT] [ NEXT ] [ MDI ] [      ]
```

BND-S OTC OPE 90/7



Command values entered from the MDI or modal values previously specified are displayed.

```
PROGRAM
      (MDI)                  (MODAL)
     Z 230.000                F 11.0000
 G00                      G01 M
                          G97 S
                          G69 T    0303
                          G99
                          G21 WX  0.000
                          G40 WZ  0.000
                          G25

[PRGRM] [CURRNT] [ NEXT ] [ MDI ] [     ]
```



One page including a block executed in the program is displayed.

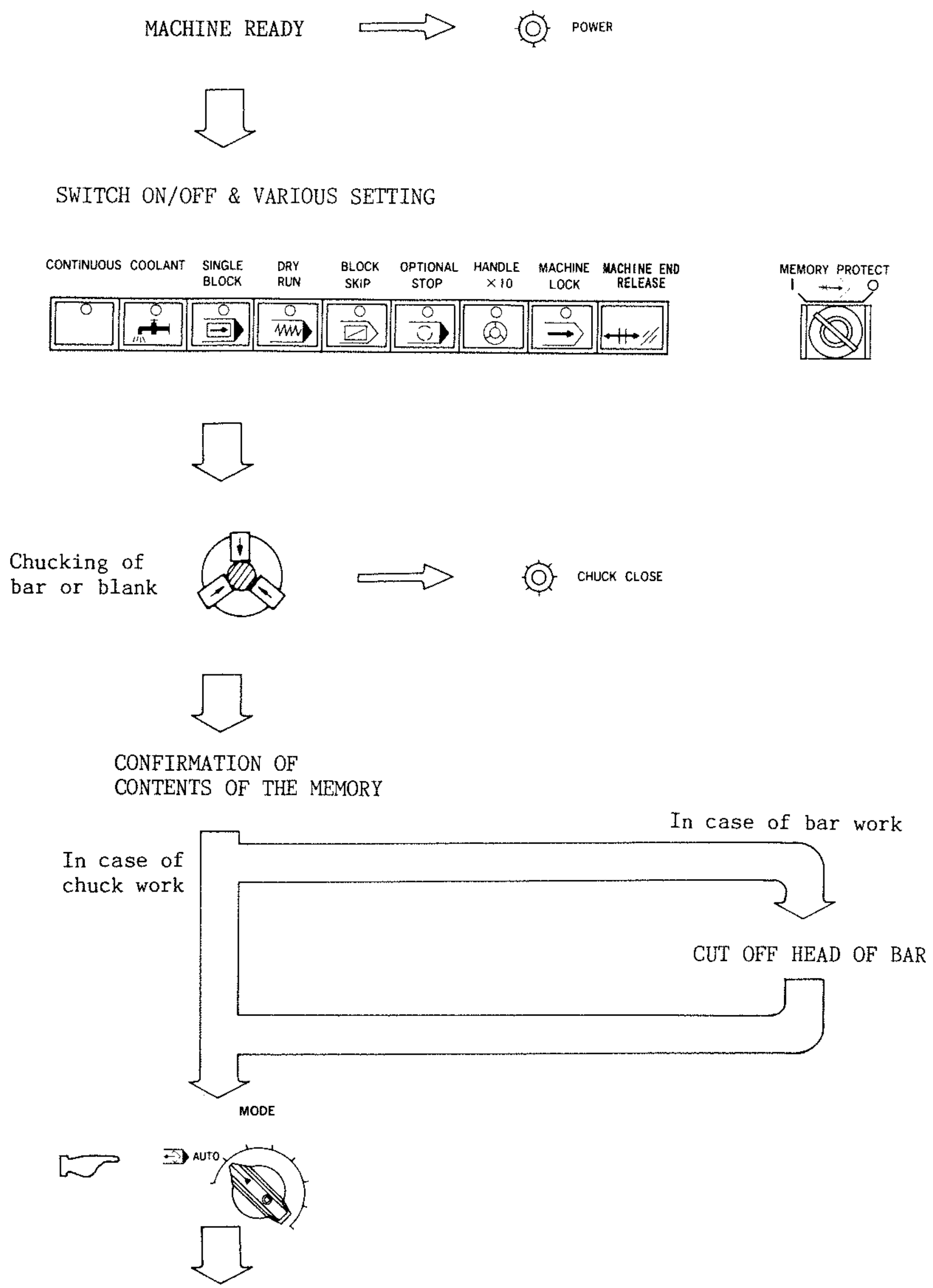
```
PROGRAM
 O1000 ;
 N100 S27 ;
 N101 G00 X100.0 Z50. ;
 N102 G01 X230.0 Z56. ;
 N103 W-1.0 ;
 N104 U-20. ;
 %

[PRGRM] [CURRNT] [ NEXT ] [     ] [     ]
```

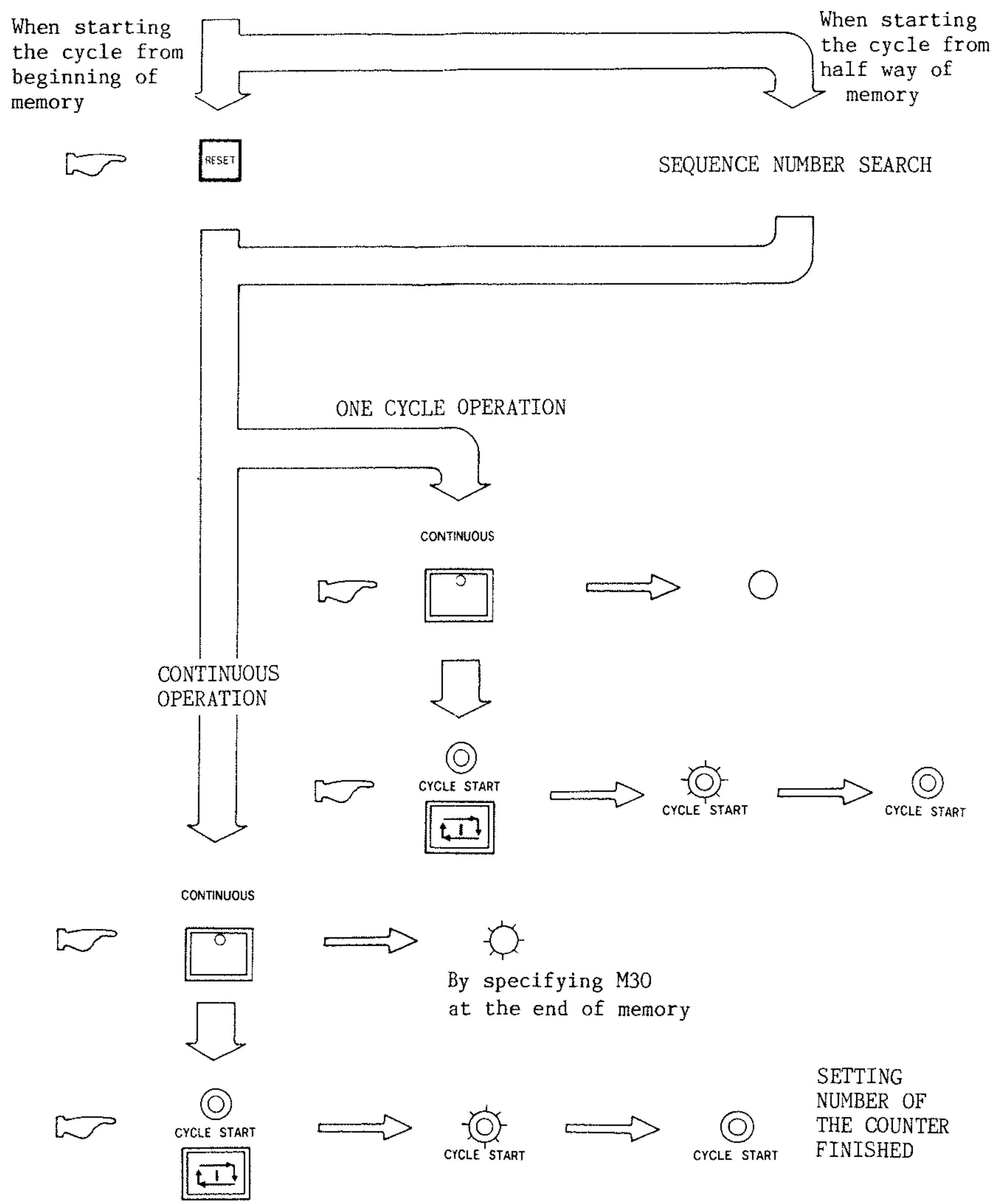
A cursor is indicated at the beginning of a block being executed.

BND-S OTC OPE 90/7

4-16 Memory Operation

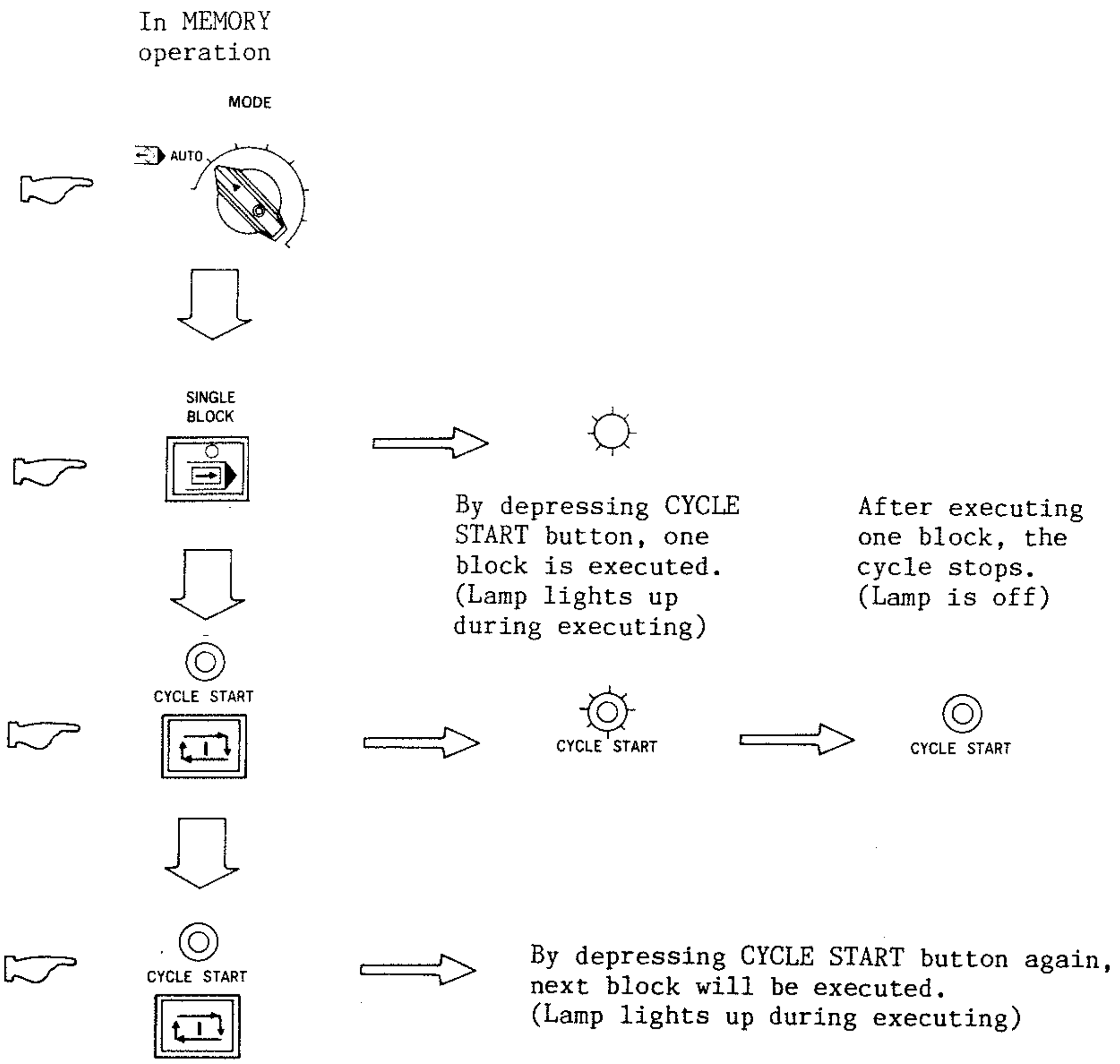
[1] Memory Operating procedure





(Note)
M02 is applied for chucking work. When M02 is commanded, the CONTINUOUS switch is inoperable.

[2] SINGLE BLOCK operation

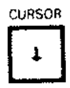


BND-S OTC OPE 90//

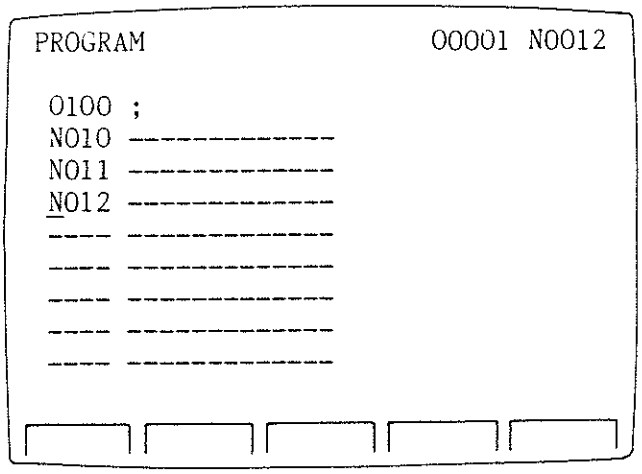
[3] Starting cycle from half way of program

Execution of necessary
G, M, S and T functions
by MDI operation

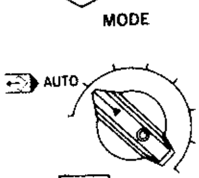
Program search

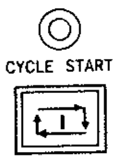


Select cursor to the program to be executed



Confirmation of current position





Executing from N12

4-17 Setting and Display of Tool Wear Offset Value

Each tool can be compensated by tool position offset and tool nose radius compensation.

The tool position offset is a T-function which compensates the difference between 'programmed cutting tool edge position' and 'actual setting tool edge position'. When a tool specified by T □□□□ command is moved, the offset value corresponding to the tool wear offset number is added to the commanded value in the program and the tool is moved to the offset position.

(Note 1)

Tool offset number '00' is meaning the cancellation of tool offset.

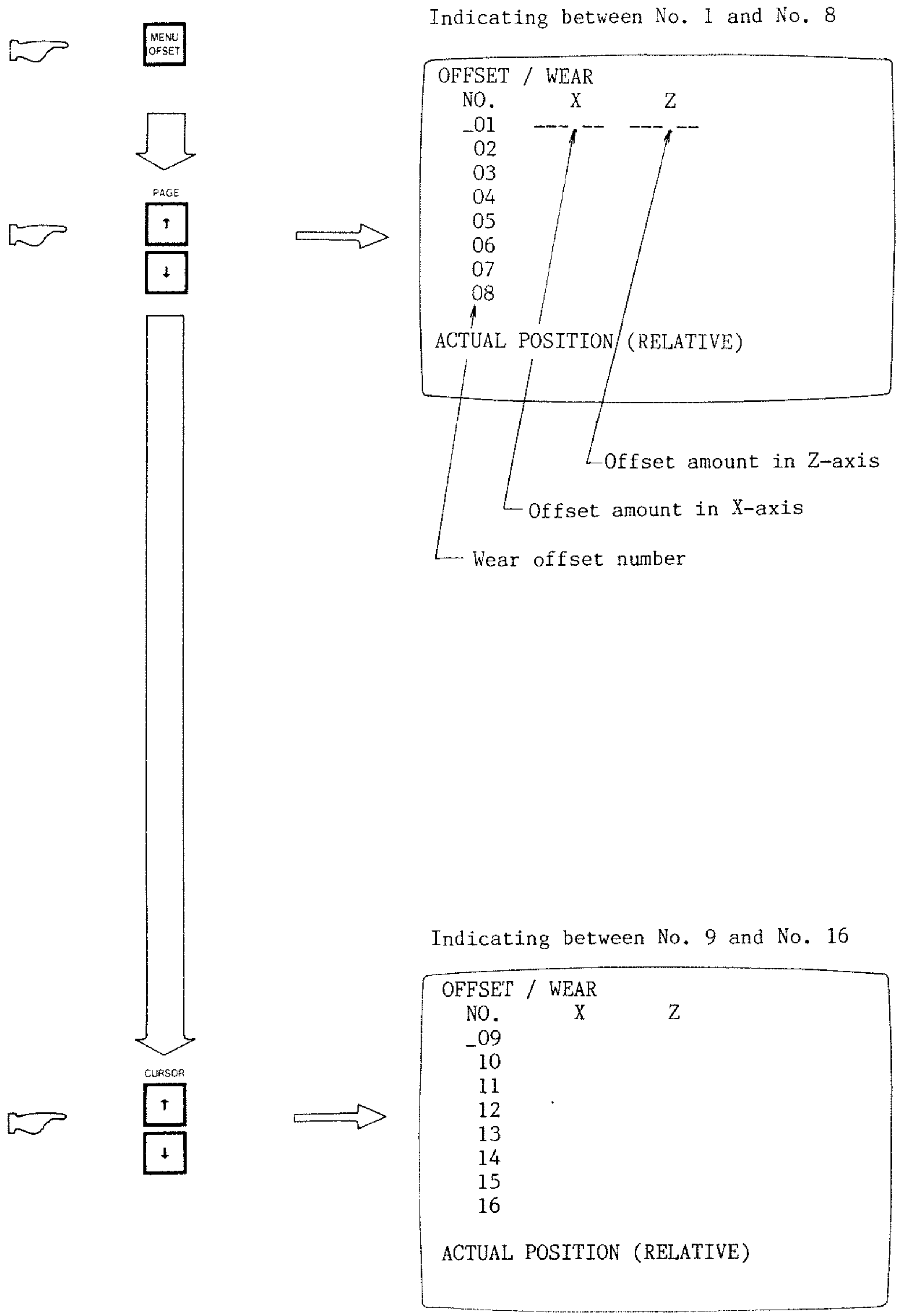
(Note 2)

Tool offset values stored in the memory will not be erased by turning off the power.

(Note 3)

When new work coordinate system is set by tool wear compenssation, corresponding tool wear compensation amount is cleared and becomes zero.

[1] Display of tool wear offset value

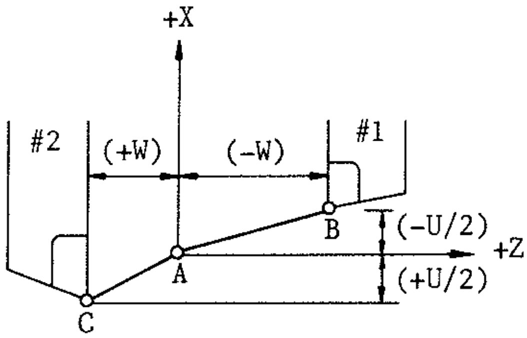


BND-S OTC OPE 90/7

[2] Setting of tool wear offset value

Tool wear offset value can be set newly or altered in any mode, even during automatic running.

Setting value of the tool position offset can be known by checking the difference between the coordinate value on the program and the actual cutting dimension, and it is set with the sign (+ or -).
The value of the X-axis is the value of diameter.



Setting ranges are as follows;

Setting unit; .0001" (0.001 mm)
Max. setting value of tool position offset; ±99.9999" (±999.999 mm)
Max. setting value of tool nose radius; 99.9999" (999.999 mm)

(1) Setting procedure of tool wear offset value

MENU OFSET

PAGE ↑ ↓ and CURSOR ↑ ↓

```
OFFSET / WEAR
  NO.      X        Z
   01
  _02    --.--    --.--
   03
   04
   05
   06
   07
   08
```

For absolute offset value in X-axis

X

For absolute offset value in Z-axis

Z

For incremental offset value in X-axis

U

For incremental offset value in Z-axis

W

Numeral

##.###

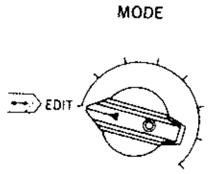
INPUT

```
OFFSET / WEAR
  NO.      X        Z
   01
  _02   ##.###    --.--
   03
   04
   05
   06
   07
   08
```

BND-S OTC OPE 90/7

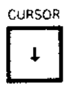
4-18 Editing Operation

[1] Designation of editing portion



Confirmation
of program
number

Direct sequence
number search



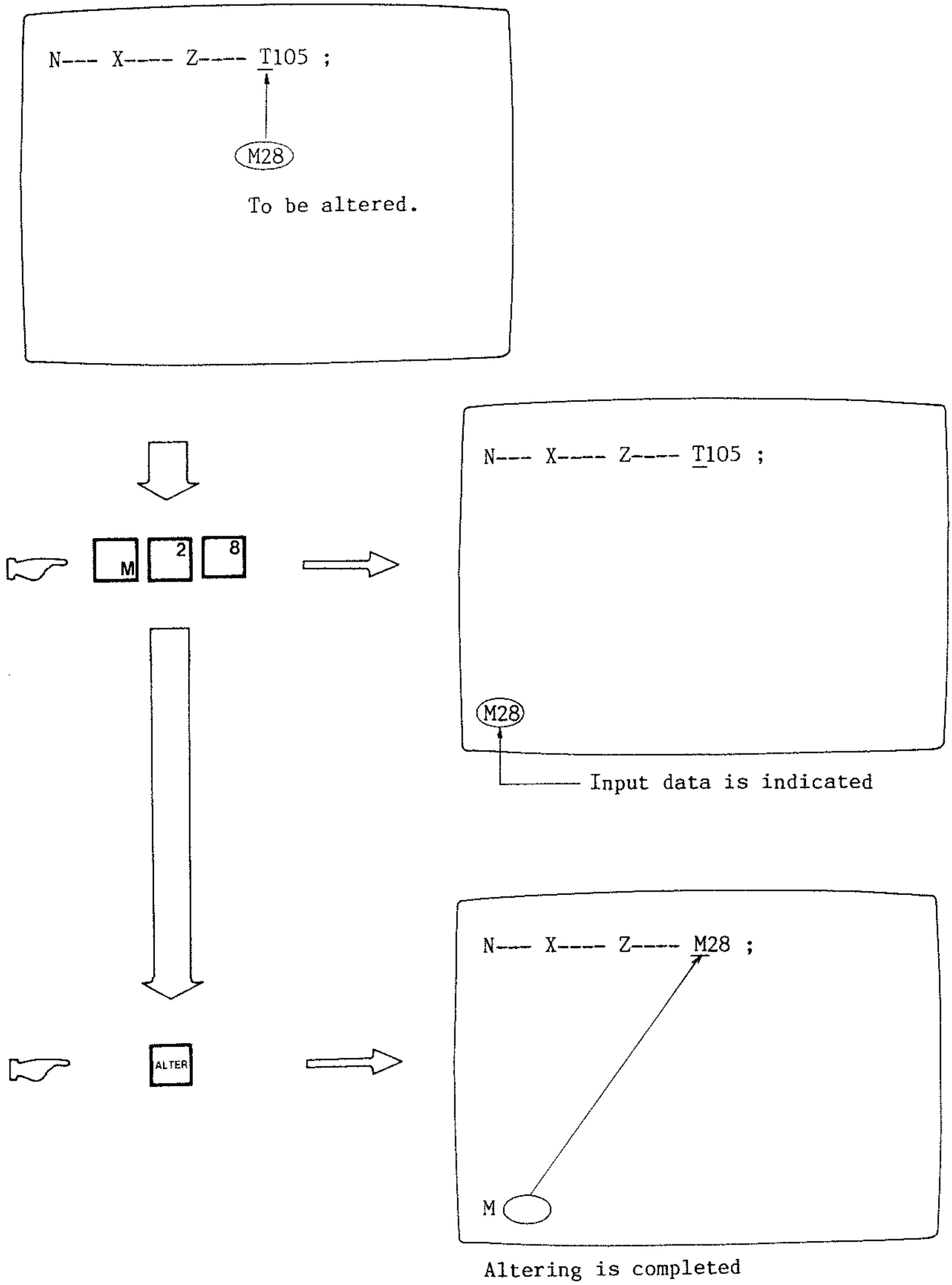
Search word orderly

```
N010 G00 X----- Z----- ;
```

Refer to sections of
'Inserting a word',
'Altering a word',
'Deleting a word',
'Delcting a block' and
'Deleting blocks'.

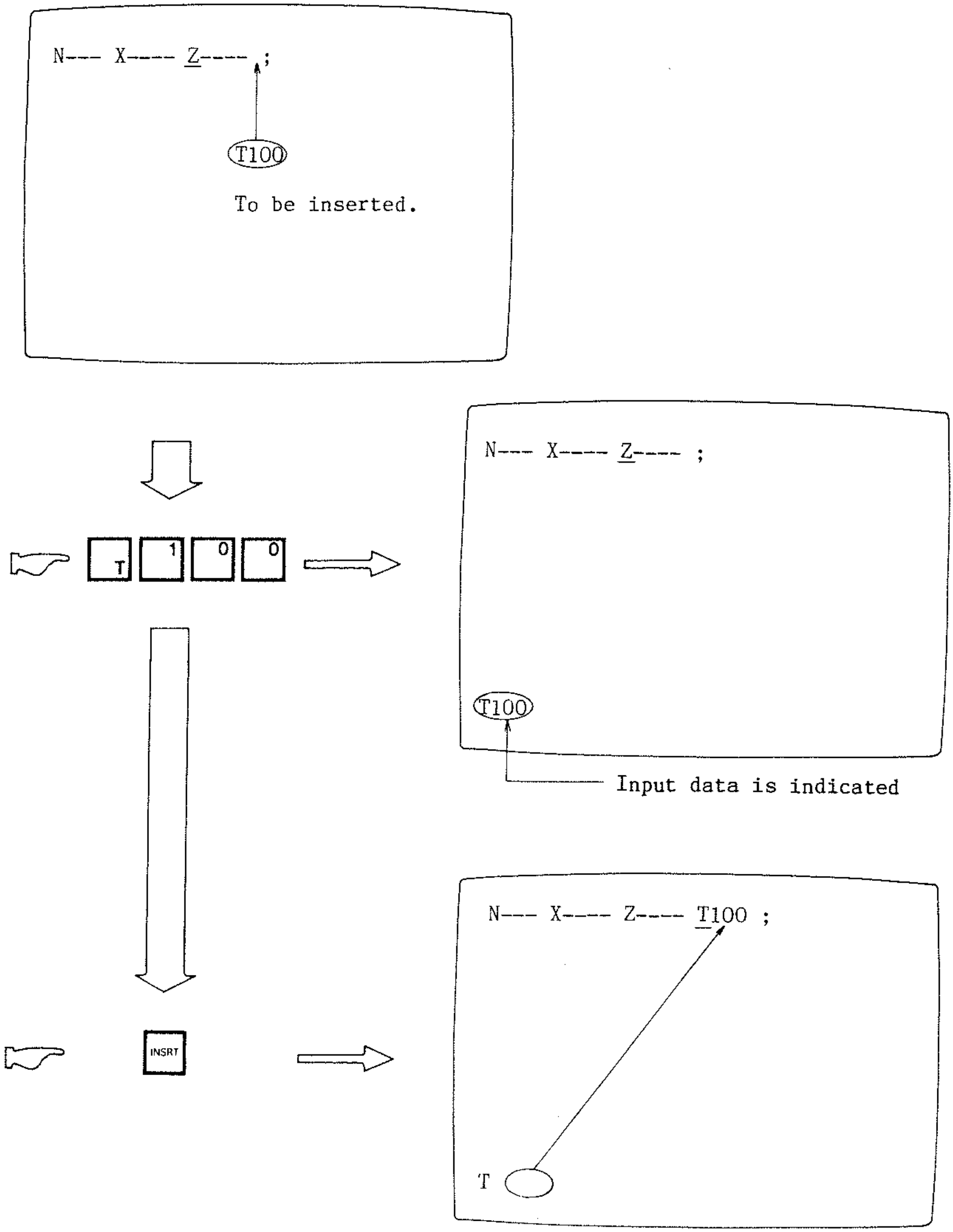
[2] Altering a word

Search out a word to be altered.



[3] Inserting a word

Search out a preceding word to be inserted.

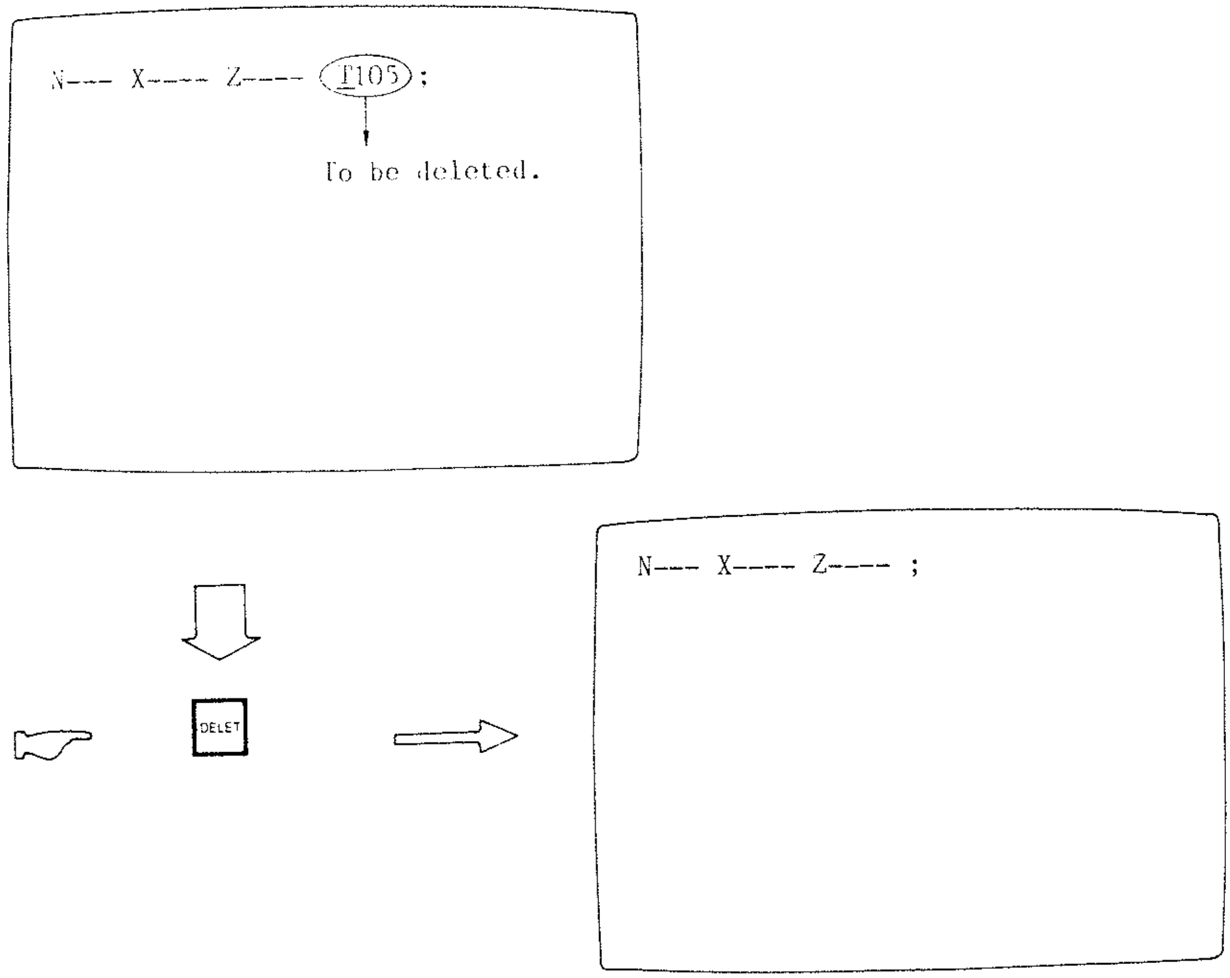


Inserting is completed

BND-S OTC OPE 90/7

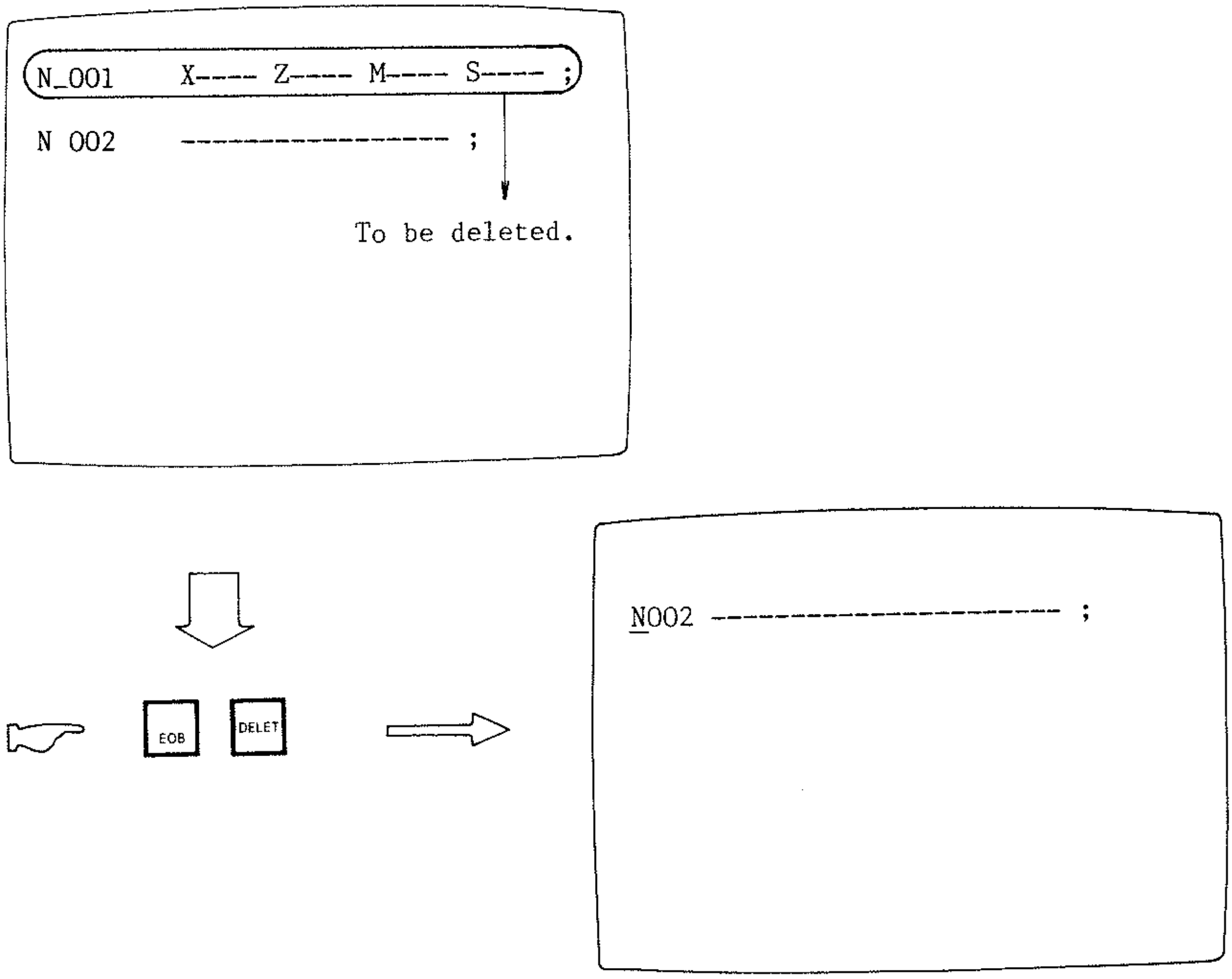
[4] Deleting a word

Search out a word to be deleted.



[5] Deleting a block

Search out the first word of block to be deleted.



[6] Deleting for more than one block

Search out the first word to be deleted.

```
N001 X---- Z---- ----- ;
N002 Z---- ----- ;
N003 ----- ----- ;
N004 ---- ;
N005 ---- ;
N--- ---------- ;
N--- ------- ;
```

To be deleted.

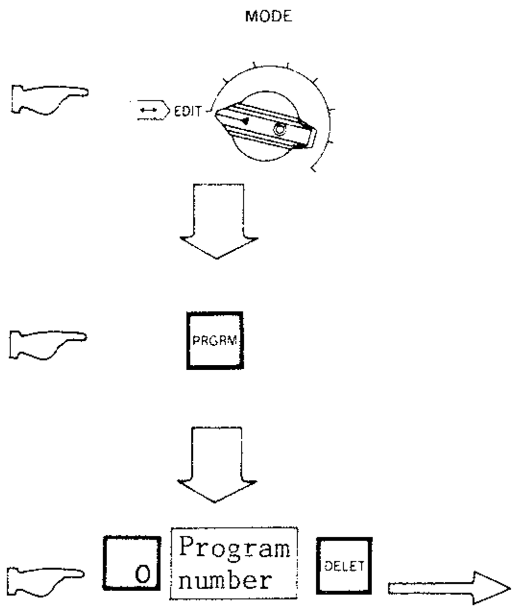
Input sequence number of
the last block to be deleted.

N 0 0 3

DELET

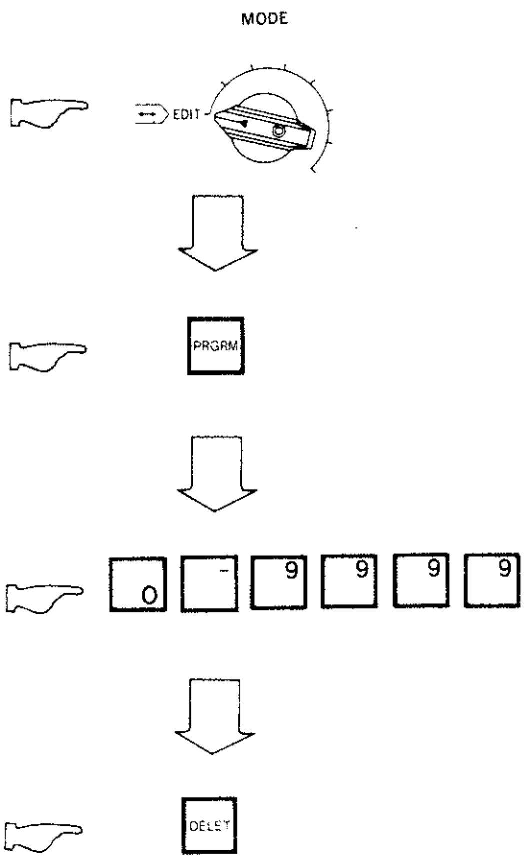
```
N004 ---- ;
N005 ---- ;
N--- ---------- ;
N--- -------- ;
```

[7] Deleting a program

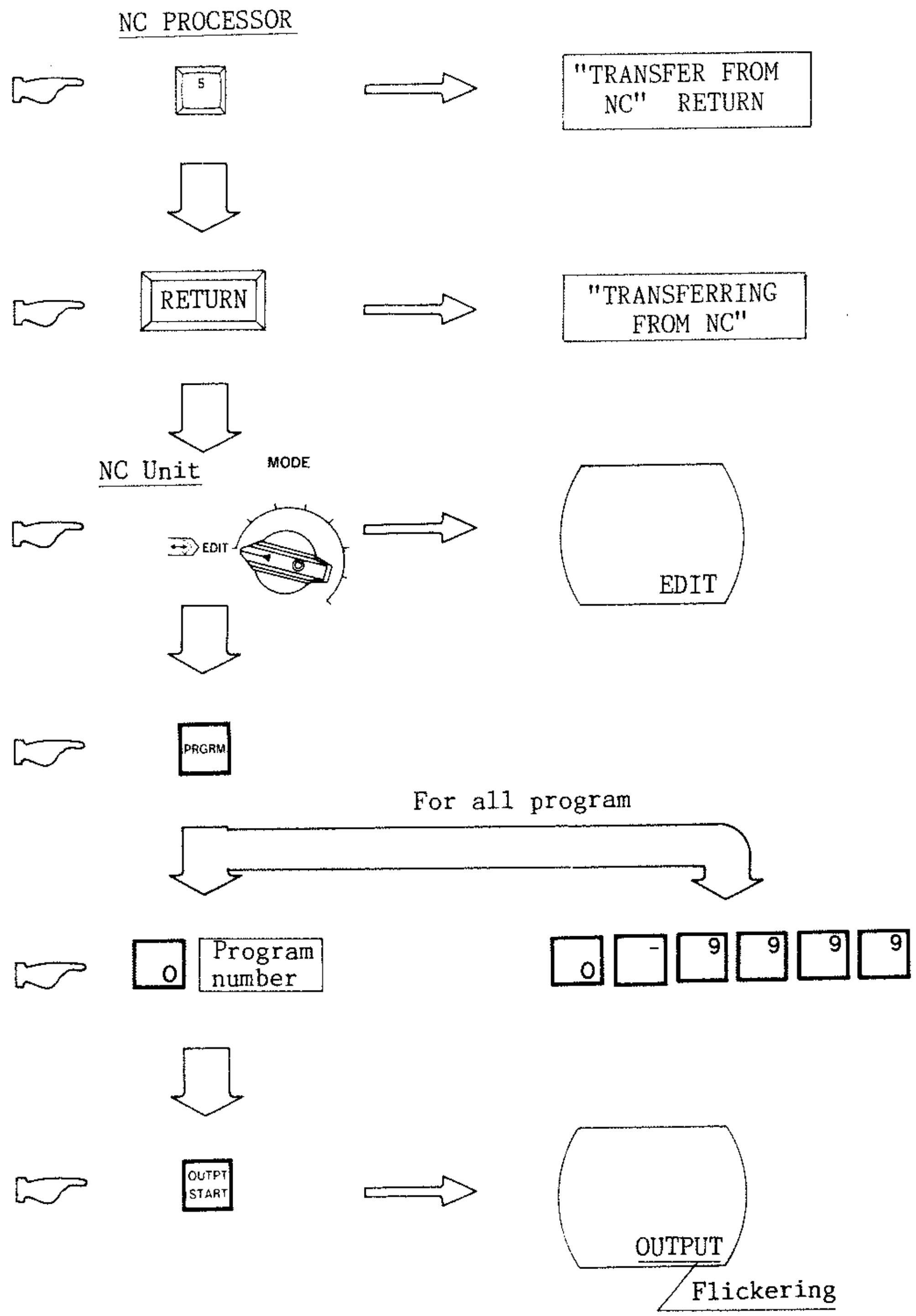


Program of input number is deleted.

[8] Deleting all program

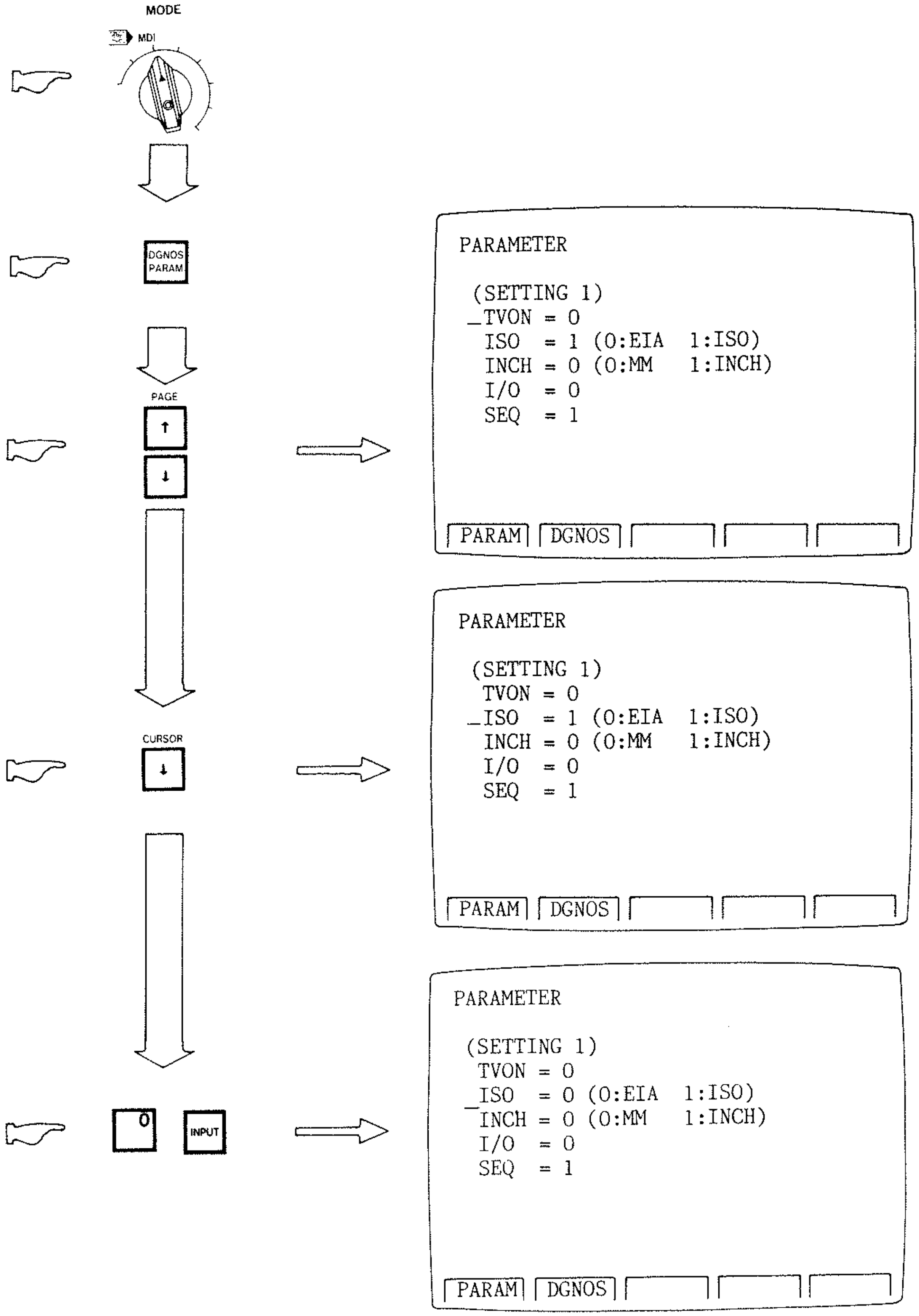


4-19 Transferring Stored Program to NC PROCESSOR



4-20 Display and Setting of Input/Output Parameter (SETTING PARAMETER)

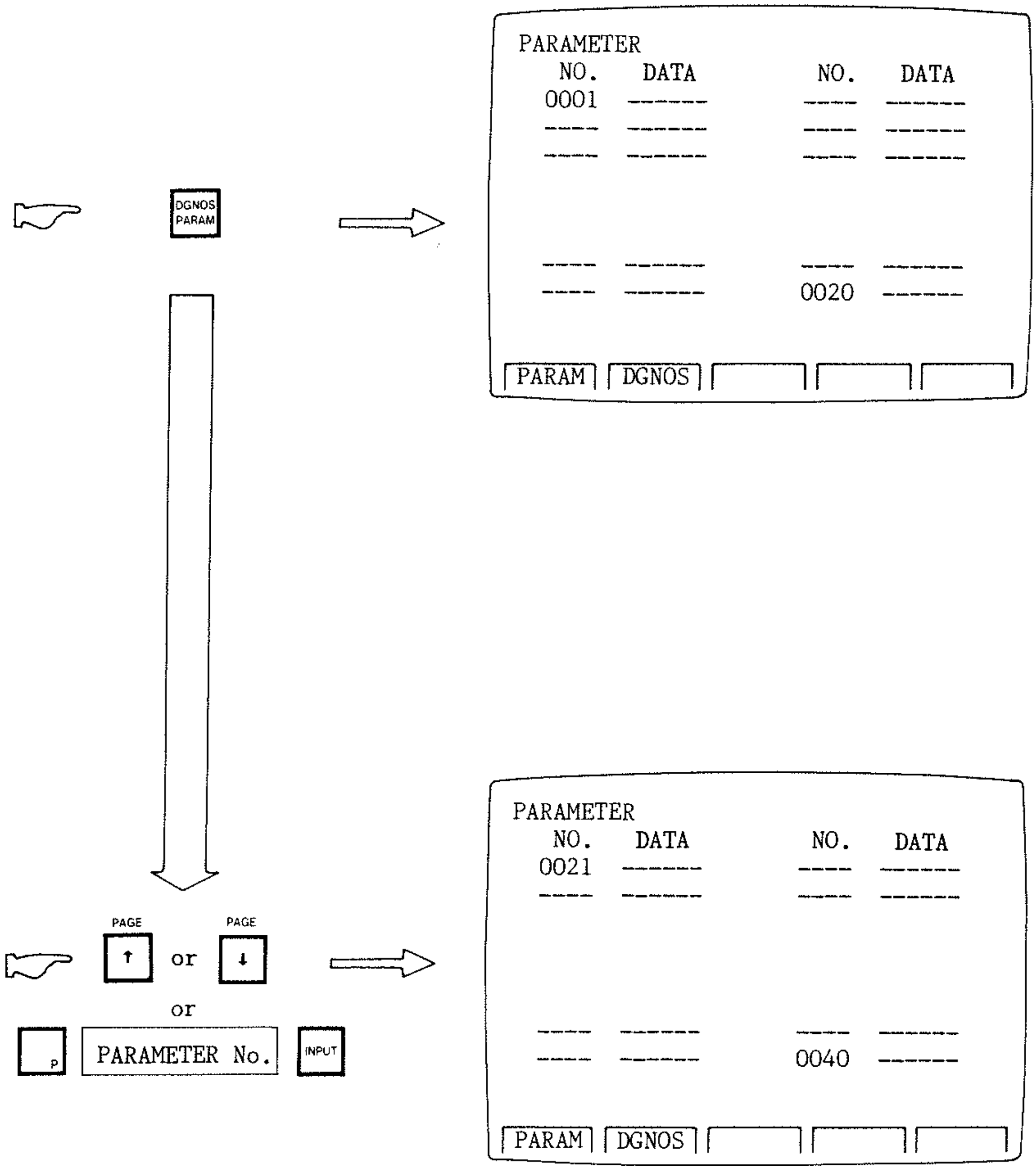
Various conditions, which are necessary on machine operation, can be changed easily by SETTING PARAMETER display.



4-21 Display and Change of Parameter

Various parameters are stored in the memory and various conditions such as rapid traverse rate etc. shown in parameter list are determined by setting them.

[1] Display of parameter

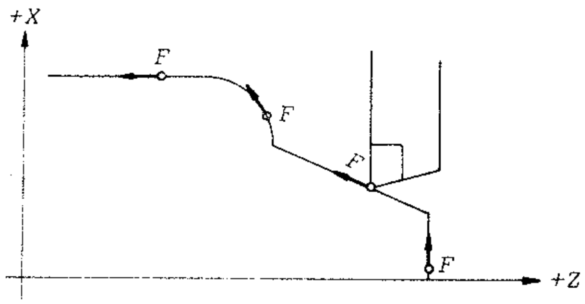


[NOTE 1]

G98 and G99 commands are modal G codes.
Therefore, once one of them is specified, it is effective until other one is specified.

[NOTE 2]

The feed rate for taper cutting or circular cutting represents the tangential feed rate. Accuracy of the tangential feed rate is ±2%.



[NOTE 3]

An override in increments of 10% from 0 to 150% can be provided for the amount of the feed rate specified by the F function by using the FEED RATE OVERRIDE dial on the operating panel.

[NOTE 4]

When programming by using the decimal point, G98 and G99 must be specified at first.

G98 F1.0 ------- Correct
F1.0 G98 ------- Incorrect

2-6 S - Function (Spindle Speed Function)

The spindle speed directly designates the amount of RPM by using 2 to 4 digits numbers or the amount of cutting speed by using 1 to 4 digits numbers following the address 'S'. When the S code is commanded in G96 (Constant cutting speed control) mode, the number following the address S specifies the cutting speed (Ft/min or m/min), and when the S code is commanded in G97 (Constant cutting speed control cancel) mode, the number following the address S specifies the RPM.
However, the spindle RPM can be specified without specifying G97 just after the power is ON, because the initial state of these G code is G97.
The S-code is modal code. Therefore, the commanded S code is effective until the other S code is commanded. And when changing from G96 to G97 or vice versa, the S code must be designated again.

G96	Constant Cutting Speed Control
G97	Constant Cutting Speed Control Cancel

96

[1] Spindle RPM direct command (G9~~7~~ mode)

The spindle revolutions per minute (RPM) can be specified directly by using 3 to 4 digits numbers following the address 'S'.

96

| G9~~7~~ S ;

(1) Example commands of spindle speed

S1234 ⟶ 1234 rpm
S123 ⟶ 123 rpm

(2) Range of spindle speed

BND-20S
220 to 6500 rpm

BND-34S
170 to 5000 rpm

(Note 1)

The S command becomes effective instantaneously when it is commanded. When the S command and the move command are specified in a block, the both commands will be executed at the same time.
However, if M03 (Spindle forward) or M04 (Spindle reverse) is not commanded previously, the spindle will not start until it is commanded.
When a new S command is specifited during the spindle is rotating, the spindle speed will change to new commanded speed instantaneously.

(Note 2)

When the S code is commanded in a block together with M03 (Spindle forward) or M04 (Spindle reverse), the spindle will start instantaneously, but the block will not proceed to next one until the spindle speed reaches the commanded speed.

(Note 3)

The S code is modal. Although the spindle is actually stopped by M05 (Spindle stop) command, the S command will remain effective until the other S code is commanded.
Therefore, if M03 (Spindle forward) or M04 (Spindle reverse) is commanded again, the spindle will start by the speed previously commanded.

[2] Cutting speed command (G96 mode) .... Constant cutting speed control

The cutting speed (Ft/min or m/min) can be specified directly by using 1 to 4 digits numbers following the address 'S'. By this command, the CNC system calculates the spindle speed at the specified speed corresponding to the move of current position of the tool on X-axis. In the following blocks, the cutting speed can be changed by S command.

(1) Example command of cutting speed

S125 ⟶ 125 ft/min or 125 m/min

(2) Clip command of max. spindle speed

When commanding the cutting speed, the spindle speed may go up too fast if the tool is close to the center line of the spindle. The maximum spindle speed is clipped at 5000 rpm on the machine by setting parameter No.556. However, the maximum spindle speed can be programmed as follows;

G50 S1500 ; .... Max. spindle speed is 1500 rpm

[NOTE 1]

In G96 mode, the coordinate system should be set by G50 so that the center line of the spindle meets with zero in X-axis. By this setting, the X coordinate will be equal to the diameter of workpiece at the current cutting point.

[NOTE 2]

The calculation for constant cutting speed is normally made with the current value of X coordinate as follows;

X-coordinate value = Position value in X-axis - Tool position offset value

Therefore, the tool position offset value does not affect to the spindle speed calculated by constant cutting speed control.

[NOTE 3]

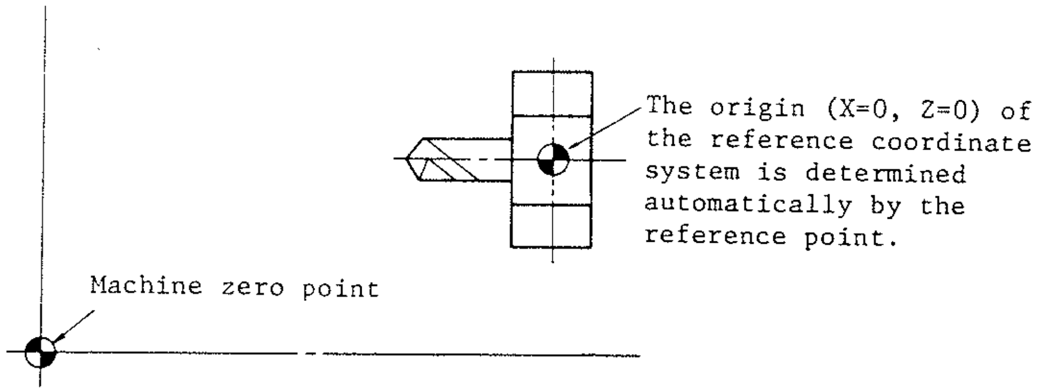
In case of drilling operation, G97 mode should be used. If G96 mode is used, the X-coordinate will be zero and the spindle speed may not be calculated correctly.

2-7. Coordinate System Setting

The machine uses two coordinate systems, i.e. the reference and work coordinate systems.

(1) Reference coordinate system setting

In order to operate the machine in the automatic mode (MEMORY), it is necessary to set the reference coordinate system. To do so, return the machine to the reference point manually so that the reference point is written automatically to the slide position registor as the origin (X0,Z0) of the reference coordinate system.
As this machine has the move command executed on the work coordinate system (which is described below), no move command is executed by this reference coordinate system. It should be noted, however, that the reference coordinate system setting by means of the manual zero return function is essential to the correct execution of the program because it provides the reference point for the automatic calculation of the current position on the work coordinate system of the tool selected in the program. In order to return the turret slide to the reference point by programming, cancel the work coordinate system to reset the reference coordinate system and specify the reference point coordinates X0, Z0.



(NOTE 1)
Always be sure to return the machine to the zero point manually before its operation after the power switch is turned on. When this return operation is omitted, the start interlock works, the alarm is displayed and the turret slide can not be moved by program.

(NOTE 2)
Do not program G50 X(U)___ Z(W)___;. When this is programmed, the reference coordinate system is altered and the subsequent program is not executed correctly.

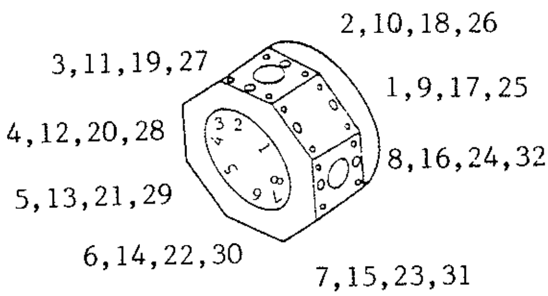
(2) Work coordinate system setting

The machine accepts two tools on the turret tool station, provided with 16 tool position memories associated with tool position numbers 1 to 16. The programmed position to which each tool moves, is written to the memory in the form of coordinates on the work coordinate system. (Refer to operation description for further details.)
This tool position memory is called when the T□□ 00 ; is commanded in the program at the same time as the tool number is selected on the tool station.
The current position of the selected tool automatically translates to the work coordinate system coordinates, reflecting the slide position registor and tool position memory contents, regardless of the slide position.
Therefore, the part program is executed using the selected tool on the work coordinate system.

Commanded method

T □□ 0 0 ;
Tool number (on the turret tool mounting face)

The tool numbers are defined as follows.



- Example -

(1) When ① and ② are commanded, the turret index and coordinate value are altered.

T0100 ; ----------- ①
.
.
.   Work command
.
T0200 ; ----------- ②
.
.
.   Work command
.

(2) When ② is commanded, only the coordinate value is altered without performing turret indexing.

```
T0100 ; --------- ①
  .  )
  .  |
  .  }  Work command
  .  |
     )
T0900 ; --------- ② Only the coordinate system is altered without
                     performing turret indexing.
  .  )
  .  |
  .  }  Work command
  .  |
     )
```

2-8. T-Function (Tool Function)

When programming the T-Function, the first two digits (00 - 32) following the address T specify the turret tool number and geometry offset number, while the last two digits (00 - 32) specify the wear offset number.
The "00" coding for the wear offset number cancels tool position compensation, i.e. the amount of tool position compensation is zero and no nose radius compensation takes place.
The machine accepts 32 sets of specifications each for the tool position compensation, nose radius compensation and geometry offset number functions.

```
T □□□□ ;
  │ └──── Wear offset number (00 - 32)
  └────── Tool number/geometry offset number (01 -32)
```

[1] Tool number/geometry offset number specification

When the tool number and geometry offset number are specified by coding T □□0 0 ;, the specified tool is selected on the turret and its position is assumed as that of the machine.

```
T □□0 0 ;
  └──── Tool number (on the turret tool mounting face)
```

(NOTE)
Do not use the wear offset number other than '00' when the turret station is selected.

[2] Tool offset (General)

There are two kinds of tool offsets, that are tool position offset and tool nose radius compensation. Each offset value must be stored into the memory by using MDI (Manual Data Input).
Both tool position and tool nose radius compensation are specified by using the same offset number.
The work coordinate multi-shift function of the machine, however, permits the tool position memory contents to be altered so easily that no tool position compensation for the purpose of correcting a tool setting error is required.
In other words it is necessary for the operator only to correct the wear in the tool while using the machine. This facilitates tool management.
The tool nose radius compensation is effective only when one of G41 to G42 (Direction of the compensation) is specified. After specifying G40 (Compensation cancell), the tool nose radius compensation will not be effective and the tool position offset will remain effective. When tool offset number '00' is specified, the both offsets will be cancelled, but the command of G41 to G42 will remain. Therefore, G40 must be specified after this block.

	Offset Method	G code	Storage of Offset Values	T code
Tool Position Offsets	Offset Value Commanded	Unneces-sary	X Z (W)No.1 (W)No.2 (W)No.3 . . . (W)No.32	T□□03
Tool Nose Radius Compen-sation	Full Automatic	G40 G41 G42	R .... No.1 .... No.2 .... No.3 . . . . No.32	

[3] Tool position offset (Wear offset)

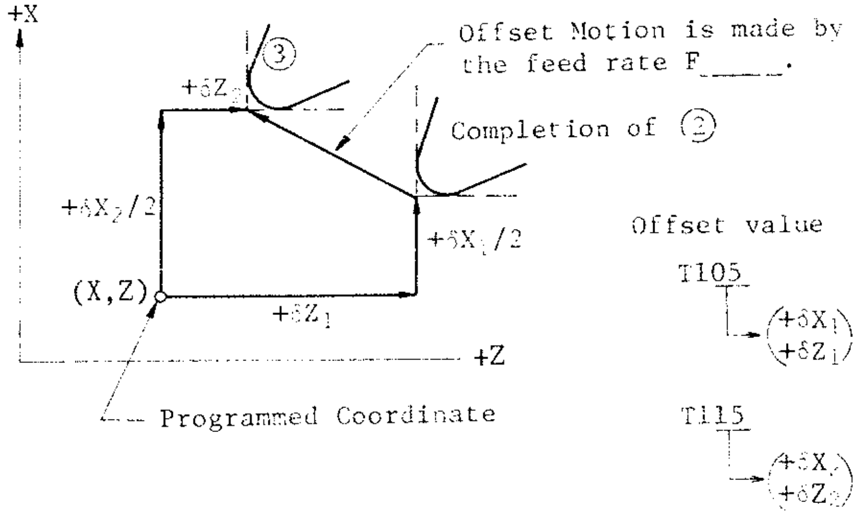
When the wear offset number is specified,the offset value corresponding to the wear offset number is added to the commanded value in the program and the tool is moved to the offset position. Therefore, the difference between the coordinate values on the programming and the actual cutting dimension must be stored into the memory. When the tool wears and the dimension changes, the amount of the change can be corrected without changing the program by resetting the tool offset value.

(1) Tool offset motion

As mentioned above, when the tool specified by T is moved, the offset value corresponding to the wear offset number is added to the commanded value in the program and the tool is moved to the offset position. When the tool offset is programmed in the block without movement command, the tool moves only for offset value. Once tool offset is programmed, the tool offsets for the corresponding tool keep in effect until another offset number is specified. If other number is specified or the offset value is corrected, the offset value is compensated for the amount of its difference. Therefore, when the tool offset is cancelled by T□□00, the tool moves to the programmed position with subtracting the tool offset value, or the tool moves only for offset value if there is no movement command.

-Example -

```
Program  G00 X ____ Z ____ T101 ; ----- ① ..... Setting of
         .                                      coordinate
         .                                      system.
         .
         G01 X ____ Z ____ F ____; --- ②
         T115 ;          --------------- ③ .... Block of the
                                                offset motion
```



BND-S PRO 0-TC 90/8

(2) Movement speed of the tool position offset

The movement speed of the tool position offset is determined by the feedrate command that is effective in the block. Therefore, the feedrate command (G00 or G01___ F___) should be specified before or in the block containing the tool offset number.

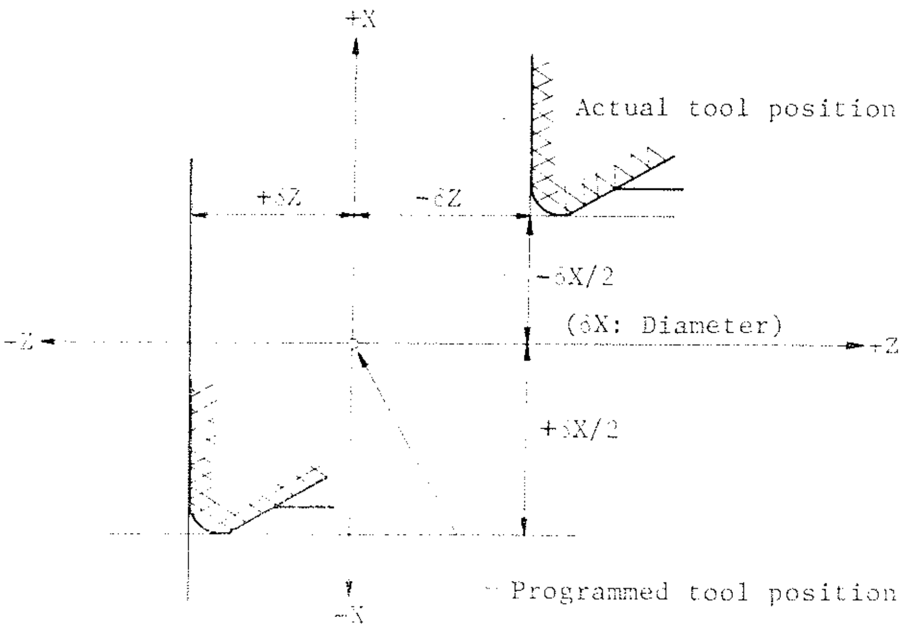
(3) Setting of the tool offset value

(a) The tool offset value is stored into the offset memory corresponding to the wear offset number by using MDI operation. Setting range of the tool offset value is as follows;

	INCH	METRIC
Setting range	0 to ±99.9999 in	0 to ±999.999 mm
Step	.0001 in	0.001 mm

(b) The tool offset value of the X-axis is the value of diameter.

(c) The tool position offset value is a difference between the coordinate value on the program and the actual cutting dimension, and the sign (+ or −) of the offset value specifies the direction of the offset motion.



BND-S PRO 0-TC 90/8

(4) Method of the tool position offset command

The tool offset starts at the block in which the T code (T ____) is commanded, and the tool starts to move by the offset value corresponding to the wear offset number.
Since T code is modal, it is retained until the other T code is designated.

\- Example -

```
G00 X____ Z____ T0202 ;
G01 X____ Z____ F____ ;
            .
            .
G00 X 0   Z 0   T0200 ; ———— Tool offset is cancelled
```

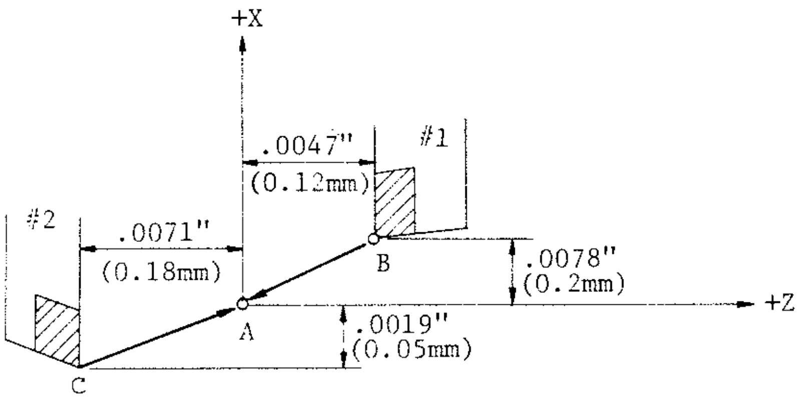
When the tool offset value is required to change, the T code whose wear offset number is changed should be commanded.

\- Example -

```
G00 X____ Z____ T0202 ;
G01 X____ Z____ F____ ;
            .
            .
G01 T0213 ; ——————————— Wear offset number 13 is replaced
                        with 02.
                        Tool offset moticn is made at the
                        feed rate F____.
```

(5) Example to use tool position offset

(a) Normal tool position offset



Point A: Tool point set in the program
Point B: Actual tool point of the tool #1
U : .0078"(0.2mm), W : .0047"(0.12mm) - off position
Point C: Actual tool point of the tool #2
U : -.0019"(-0.05mm), W:-.0071"(-0.18mm)-off position

In this example, the tool offset values are as follows;

	Inch Input		Metric Input	
	Tool No.		Tool No.	
	01	02	01	02
U	-0.0078	0.0019	-0.2	0.05
W	-0.0047	0.0071	-0.12	0.18

By setting these offset values corresponding to the offset number into the offset memory, the positions of tool #1 and #2 become the same as they locate at the point A.

- Example of programming -

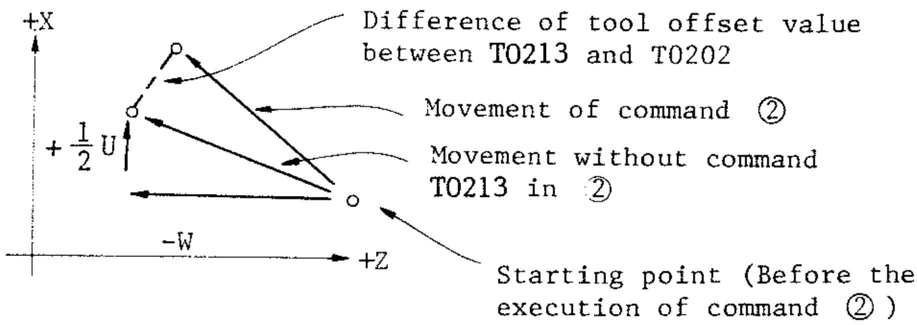
```
    T100 ;
G00 X4.0 Z5.0 T101 ; ..... Using the tool #1
G01 Z2.0 F.012 ;
    X4.5 ;
G00 X9.0 Z8.0 ;
```

(b) Changing of taper angle
(Example to use two pairs of offset for a tool)

When the T command and the move command are specified in the same block, the tool moves to the offset position. Therefore, this function can be used for correcting the taper angle in the taper cutting.

- Example of programming -

```
G00 X____ Z ____T0202 ; -----------①
G01 X____ Z ____F______ ;

G01 U_____W_____F _____T0213 ;------②
```



In the above case, the taper angle is corrected by the difference of the offset value between T0202 and T0213.

[4] Notes of T-function

(1) The tool offset must be cancelled before M02 or M30 is commanded.

(2) The tool offset must be cancelled when 'Reference point return check' (G27) is commanded. If the G27 is commanded at the state that the tool offset is effective, the control will be the state of reference point return check error, because the tool offset value is added to the programmed position.

[5] Tool nose radius compensation

Most cutting tools must have the nose radius on the cutting edge. The tool nose radius will make a different dimension than what programmed especially in taper and circular cutting. This difference can be compenstated by storing the value of the nose radius and using G41 and G42 codes. For further detail, refer to item 4 "Tool Nose Radius Compensation".

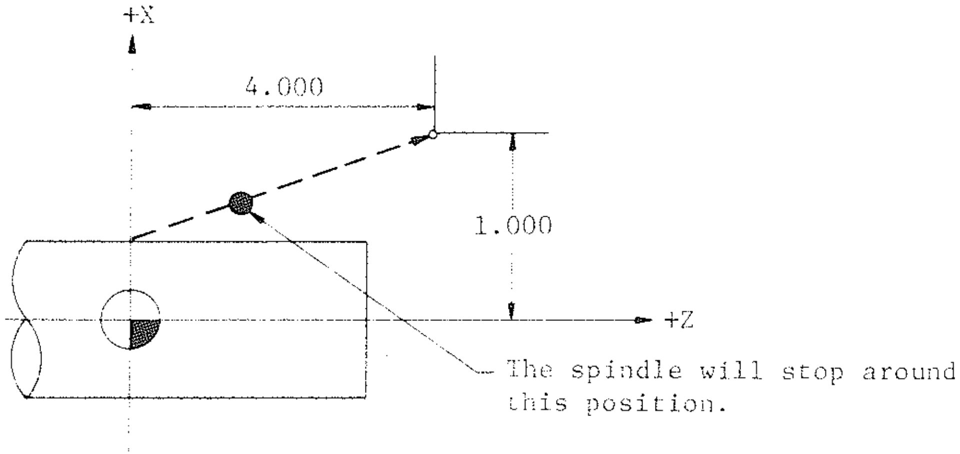
2-9. M-Function (Miscellaneous Function)

The M-Function is designated by using a 2 digits number (00 - 99) following the address M for specifying an auxiliary function.
When a move command and M-Function are specified in the same block, M function will execute after completion of the movement in case of M00, M01, M02, M05, M09, M25, M29 and M30, and both will start simultaneously in case of other M codes.

\- Example -

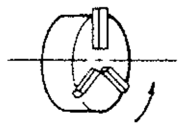
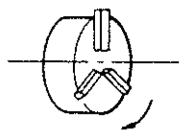
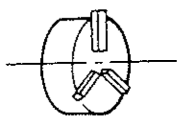
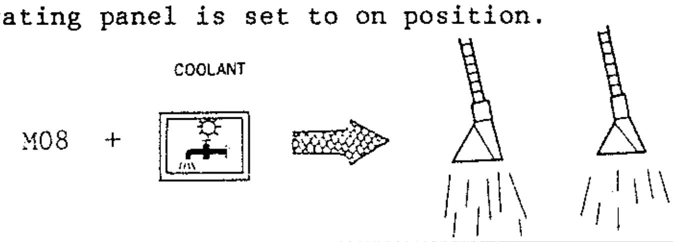
G00 X2.0 Z4.0 M05 ;

Spindle stop

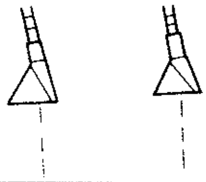
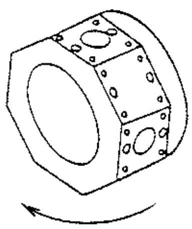
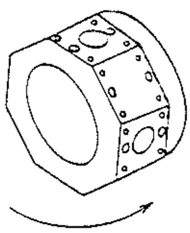


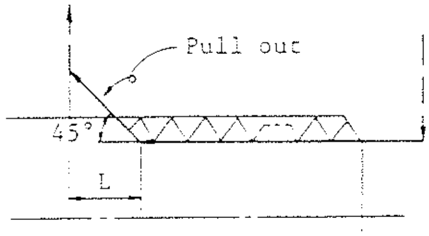
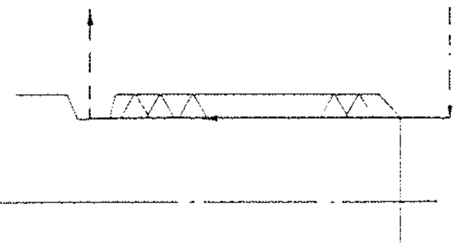
More than one M code can not be specified in a block.

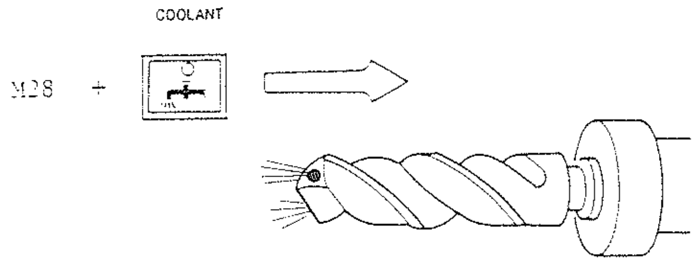
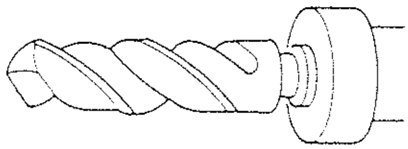
M code	Function	Explanation
M00	Program Stop	When M00 is commanded in automatic operation mode (MDI or MEM mode), the automatic operation will stop after completion of the commands in the block containing M00. When the machine is stopped by M00 code, manual operation can be done if the MODE SELECTOR Switch is turned to JOG position. To restart cycle, select the MODE SELECTOR Switch to previous automatic operation mode and then depress the CYCLE START button. *(NOTE 1)* *If the slide moves manually while the machine is stopped by M00 code, the absolute zero point is changed to the amount of its movement.* *(NOTE 2)* *Spindle also stops after completion of M00, then Chuck Open-Close can be done by manually without changing the mode.*

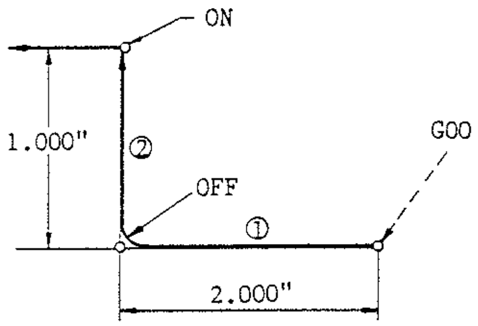
M code	Function	Explanation
M01	Optional Stop	This is usually used to stop the machine and check workpieces at the end of each tool operations. OPTIONAL STOP Switch is used to select this code. ON : M01 is effective. The machine stops on the block. OFF: M01 is not effective. The machine runs continuously.
M02	End of Program (Rewind reset)	This code is used in the last block of chucking work part program to end the program. When this code occurs during the automatic operation of the machine, the program returns to the head after performing the other command in the block, the control is reset, the automatic mode ends and the machine stops.
M03	Spindle Forward (Counter-clockwise)	Specifies to start the main spindle rotation in counterclockwise direction. S code should be specified in the same block or previously. If M03 code is specified when the chuck is open, the sequence error will occur. 
M04	Spindle Reverse (Clockwise)	Specifies to start the main spindle rotation in clockwise direction. S code should be specified in the same block or previously. If M04 code is specified when the chuck is open, the sequence error will occur. 
M05	Spindle Stop	Specifies to stop the main spindle rotation. Even M05 is specified, the commanded spindle speed remains effective. Therefore, if M03 or M04 is specified again, the spindle will rotate by the same speed as the previous speed. 
M08	Coolant ON	Specifies to start the coolant pump. The coolant pump will start when the COOLANT switch on the operating panel is set to on position. 

BND-S PRO 0-TC 90/8

M code	Function	Explanation
M09	Coolant OFF	Specifies to stop the coolant pump. 
M10	Splash guard Open (Optional)	The splash guard is opened with this command.
M11	Splash guard Close (Optional)	The splash guard is closed with this command.
M12	Bar Feeding	When M12 is commanded in automatic mode (MDI or MEM mode), a series of operations from step 1 to 3 are automatically performed. Step 1. Chuck open Step 2. Bar feed out Step 3. Chuck close
M13	Main spindle forward orientation start (Optional)	This code stops main spindle at the fixed angle, at the time of working by revolving tools attached with main spindle. When spindle positioning is employed, main spindle is locked by 7.5° index.
M15	Turret Indexing, Counter- clockwise	This code is used to switch the direction of turret indexing to CW when it is set in the automatic selection mode. As this code is a non-modal code, it should be used in the same block as the T code.  CW
M16	Turret Indexing, Clockwise	The turret indexes in clockwise by specifying M16 in the same block of T code. This M16 is a non-modal code.  CCW

M code	Function	Explanation
M17	Main Chuck Open	Specifies to open the chuck automatically such as bar work.
M18	Main Chuck Close	Specifies to close the chuck automatically such as bar work.
M19	Main Spindle Reverse Rotation Orientation Start (Optional)	This is just the reverse rotation orientation of M13.
M23	Pull out Threading ON	When M23 is specified before the command of thread cutting cycle G76 or G92, the threading tool will pull out at the terminating thread portion.  The width of thread pull out (L) is set by parameter No.0109 (THDCH).
M24	Pull out Threading OFF	Cancel the command of pull out threading function which as specified by M24 code. M24 code is the modal code. 
M25	Total Counter (Optional)	Two kinds of counter are installed to this machine. Each time M25 in commanded, a piece counter counts. * However, under condition where a counter switch is made ON, a counter counts but under condition where a switch is made OFF, a counter does not count.
	Preset Counter (Optional)	Each time M25 is instructed, a piece counter counts and when acculated value reaches the set valve, a lamp at upper section of machines is turned on and a machine stops. * However, under condition where a counter switch is made ON, a counter counts but under condition where a switch is made OFF, a counter does not count.

M code	Function	Explanation
M28	High Pressure Coolant ON (Optional)	Specifies to start the high pressure coolant pump. The pump will start when the COOLANT switch on the operating panel is set to ON position. 
M29	High Pressure Coolant OFF (Optional)	Specifies to stop the high pressure coolant pump. 
M30	Program Rewinding and Stop (Continuous running)	Specifies at the last block of the program. The function of M30 command differs, depending on whether the CONTINUOUS switch on the operating panel is set to ON or OFF position. <u>In case of continuous switch ON.</u> Return to the head of the memory by M30 command and restart automatically. (Continuous running) <u>In case of continuous switch OFF.</u> Return to the head of the memory by M30 command, reset and stop. (One Cycle running) M30 will be used at end of the bar work program
M33	Main spindle forward & high pressure coolant ON (Optional)	One command can start main spindle forward and high pressure coolant pump motor simultaneously. But the coolant is not discharged when "Coolant" switch on the NC operation panel is turned "OFF".
M34	Main spindle reverse & high pressure coolant ON (Optional)	One command can start main spindle reverse and high pressure coolant pump motor simultaneously. But the coolant is not discharged when "Coolant" switch on the NC operation panel is turned "OFF".

M code	Function	Explanation
M35	Part Catcher Forward (Optional)	This command moves the part catcher forward.
M36	Part Catcher Backward (Optional)	This command moves the part catcher backward.
M43	Revolving Tool Normal Rotation Start (Optional)	Revolving tool starts normal rotation.
M44	Revolving Tool Reverse Rotation Start (Optional)	Revolving tool starts reverse rotation.
M45	Revolving Tool Stop	Revolving tool stops.
M51	Error Detection OFF	Specifies to release the state of error detection ON. When the power is turned on, M51 will be in effect, and it will remain effective until M52 is commanded.
M52	Error Detection ON	When M51 is in effect, the control proceeds to the next block regardless of the pulse lag of servo between blocks for linear and circular interpolation except positioning (G00). This permits the machine to move smoothly between blocks. However, the corner of the workpiece may not be quite sharp. If M52 is commanded, the control proceeds to the next block after the pulse lag of servo comes within the allowable range, to avoid rounding of a corner. M52 command is modal, and it will remain effective until M51 is commanded. G00 (positioning) is always executed at the state of error detection ON regardless of the command of M51 or M52. -- Example -- G00 X---Z---; G01 W-2.0 F.012 M51 ;----- ① G01 U 1.0 M52 ; ---------- ② Z-----; 

BND-S PRO 0-TC 90/8

M code	Function	Explanation
M76	Main Air Blow No.1 ON (Optional)	Air blows when M76 and M77 is commanded.
M77	Main Air Blow No.2 ON (Optional)	
M79	Main Air Blow Stop (Optional)	Air blowing stops.
M83	Material Feeder Start (Optional)	When an automatic bar feeder is attached, feed of material is performed. A program when top cut is performed becomes as follows, at this time, a switch of "1" code on the NC operation panel is made ON and a switch of block delete at feeder side to "with". / N1 T ____ 00; (Stopper number) / N2 G00 X0 Z ____ ; (Top cut amount) / N3 M17 ; / N4 M83 ; / N5 M18 ; / N6 G00 Z ____ ; (Stopper Z relief amount) / N7 G27 X0 T0000 ; (Stopper X relief amount) / N8 T ____ ; (Number of cut off tool) / N9 G96(G97) S ____ M3 ; / N10 G00 X ___ Z ___ T ___ M8 ; (Cut off tool position) / N11 G01 X ___ F ___ ; / N12 G00 X ___ ; (Cut off tool X relief amount) / N13 Z ___ ; (Cut off tool Z relief amount) / N14 M84 ; (Block delete reset) N15 T ___ ; (Stopper number) . . Subsequently, normal program
M84	Block Delete OFF Reset (Optional)	When an automatic bar feeder is attached, this function is used in the program of top cut. Top cut is performed only when new material is feed and its signal is transmitted from feeder side as block delete OFF signal. When this signal is transmitted, "/" code switch on the operating panel becomes invalid. After top cut has been completed, when M84 is programmed, "/" becomes valid again.
M91	Main encoder ON	This main encoder is used when machining is performed at the main spindle side. Rotation display on the CRT changes to rotation speed of main spindle and a slide, X and Z axis are synchronized with rotation of main spindle.

M code	Function	Explanation
M93	Simultaneous start of spindle forward rotation and coolant	Spindle forward and coolant are performed by one command. Coolant comes out only when panel switch is "ON".
M94	Simultaneous start of spindle reverse and coolant	Spindle reverse rotation and coolant are performed by one command. Coolant comes out only when panel switch is "ON".
M95	Spindle, coolant stop	Stop of spindle rotation, coolant is stopped by one command.
M98	Subprogram call	This code is used to enter a subprogram. When a program contains a fixed sequence or a recurring pattern, the fixed sequence can be registered in the memory as the sub-program. The sub-program can be called in AUTO mode. Further, the called sub-program can also call another sub-program. The sub-program call can be nested two times. The sub-program can be called up to 999 times with one call command. Use following program format to call the sub-program. M98 P○○○ ○○○○ └ Sub-program number (○○○○) └ Number of call repetitions (○○○) * When the number of call repetitions is omitted, it is regarded as one. Sub-program Sub-program number will be made by using 4 digit number following to O. To return to the main program, write M99 at end of the sub-program. O○○○○ ; ——— Sub-program number N1 .... ; N2 .... ; . . } Sub-program . Nn .... ; Nn + 1 M99 ; ← End of sub-program
M99	End of sub-program	This code shows the end of a sub-program. Executing M99 take the control back to the main program.
M103	Subspindle Forward Rotation	Subspindle rotation in counterclockwise direction. It is synchronized with rotation of main spindle.

M code	Function	Explanation
M104	Subspindle Reverse Rotation	Subspindle rotation in clockwise direction.
M105	Subspindle stop	Rotation of subspindle stops.
M113	Sub spindle normal rotation orientation start (Optional)	This code stops sub spindle at the fixed angle, at the time of working by revolving tools attached with sub spindle. When spindle positioning is employed, sub spindle is locked by 7.5° index.
M117	Subspindle chuck, open	Subspindle chuck is open. * Time for completion of chuck opening is determined by setting of PC parameter 340, 341.
M118	Subspindle chuck, close	Subspindle chuck is close. * Time for completion of chuck close is determined by setting PC parameter 345, 346.
M119	Sub spindle Reverse Rotation Orientation Start (Optional)	This is just the reverse rotation orientation of M113.
M140 M141 M142	B axis Operation Start Command 1, 2 and 3 (Optional)	By this command, a program to operate B axis (sub spindle) is started.
M157	Sub work ejector forward (Air cylinder type)	This command moves the work ejector forward.
M158	Sub work ejector backward(Air cylinder type)	This command moves the work ejector backward.
M159	Sub work ejector automatic operation mode(Air cylinder type)	When M159 is commanded in automatic mode, a series of operations from step 1 to step 4 are automatically performed. Step 1. Sub spindle chuck open Step 2. Sub ejector forward Step 3. Forward end signal confirmation Step 4. Sub ejector backward *M159=M117 → M157 → FIN → M158
M176	Sub air blow No.4 ON (For sub spindle) (Optional)	Air blow when M176 and M177 is commanded.

M code	Function	Explanation
M177	Sub air blow No.5 ON (Optional)	Air blow when M176 and M177 is commanded.
M179	Sub air blow stop (Optional)	Air blowing stops for sub spindle
M190	Waiting Command	This is waiting M code to synchronize operation of sub spindle (B axis) and operation of main turret (X, Z axis).
M191	Sub encoder ON	This sub encoder is used when machining is performed at the sub spindle side. Rotation display on the CRT changes to rotation speed of sub spindle and a slide, X and Z axis are synchronized with rotation of sub spindle.
M193	Simultaneous start of sub spindle forward rotation and coolant	Sub spindle forward rotation and coolant are performed by one command. Coolant comes out only when panel switch is "ON".
M194	Simultaneous start of sub spindle reverse rotation and coolant	Sub spindle reverse rotation and coolant are performed by one command. Coolant comes out only when panel switch is "ON".
M195	Sub spindle coolant stop	Stop of sub spindle rotation, coolant is stopped by one command.
M203	Main and sub spindle forward synchronous rotation	Main and subspindle start simultaneously for forward rotation.
M204	Main and sub spindle reverse synchronous rotation	Main and subspindle start simultaneously for reverse rotation.
M205	Main and subspindle synchronous stop	Rotation of main and subspindles stop.
M206	Synchronous operation release	Release main & sub spindle from synchronous operation. During synchronous operation by M203 and M204, main and sub spindles can not stop by M05 or M105. Stop main and sub spindles by M205, or stop each spindle after releasing them from synchronous operation by M206.

M code	Function	Explanation
M213	Part Conveyor ON (Optional)	When a toggle switch of part conveyor is turned "ON" in "MDI" or "Automatic" operation, the part conveyor starts only when "M213" is commanded.
M215	Part Conveyor OFF (Optional)	When a toggle switch of part conveyor is turned "ON" in "MDI" or "Automatic" operation, the part conveyor stops only when "M215" is commanded.
M240	Drill failure detection (Optional)	This is a device to confirm existence of a drill. With drill (ON), programs are executed, and without drill (OFF), operation stops instantly.
M272	Cutting off confirmation forward movement (Optional)	Cutting off confirmation moves forward.
M273	Cutting off confirmation backward movement (Optional)	Cutting off confirmation moves backward.
M274	Parts separator open (Optional)	Parts separator opens and drops products into chute.
M275	Parts separator close (Optional)	Parts separator remains close.

2-10. Other Function

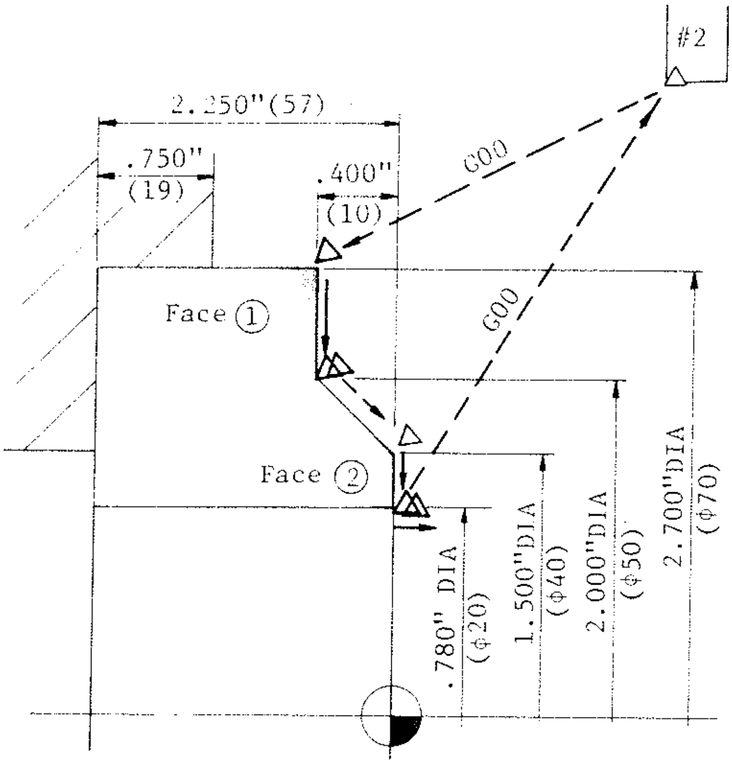
[1] "/" Block Delete (Optional Block Skip)

When the BLOCK SKIP switch on the operating panel is set to ON position, a block containing '/' (Slash) in front of the sequence number will be skipped and will not be executed.

(NOTE)
The "/" function may be disabled by using an external signal even when the BLOCK SKIP switch is set to ON.
M84 is used to reset the "/" function in programming in this case.

3. EXAMPLE OF PROGRAMMING OF VARIOUS OPERATIONS

3-1. Facing Operation

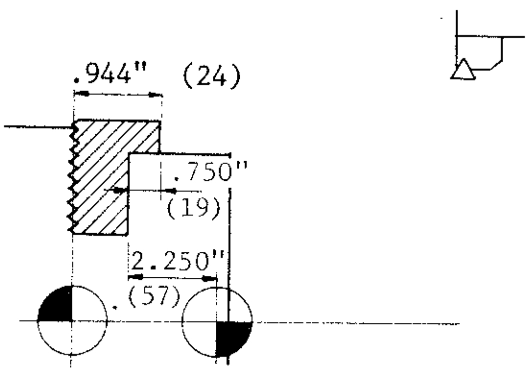


Cutting speed;
550 ft/min
(150 m/min)
Feed rate;
.012 in/rev
(0.3 mm/rev)

BND-S PRO 0-TC 90/8

(1) A tool selection is specified.

T 0 2 0 0 ;



(2) Specify constant cutting speed control and cutting speed. And start the spindle.

G96 S550 M03 ; (G96 S150 M03 ;)

(3) Positioning the tool for face ① , and specify the tool position offset and high pressure coolant on.

G00 X2.75 Z-.4 M28 T0202 ; (G00 X73.0 Z-10.0 M28 T0202 ;)

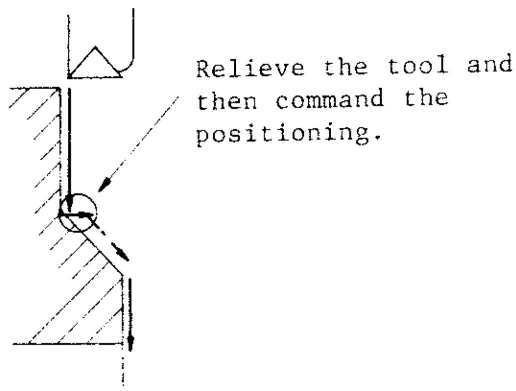
(4) Face down to 2.000" dia. (φ50) by using .012 in/rev (0.3 mm/rev) feedrate.

G01 X2.0 F.012 ; (G01 X50.0 F.3 ;)

(5) Relieve the tool by .050" (1.000 mm) and positioning the tool for face ② .

Z-.35 ; (Z-9.0 ;)

G00 X1.55 Z0 ; (G00 X42.0 Z0 ;)



(6) Face down to .71 dia (φ19) by using .012 in/rev (0.3 mm/rev) feedrate.

G01 X0.71 F.012 ; (G01 X19.0 F.3 ;)

- Program -

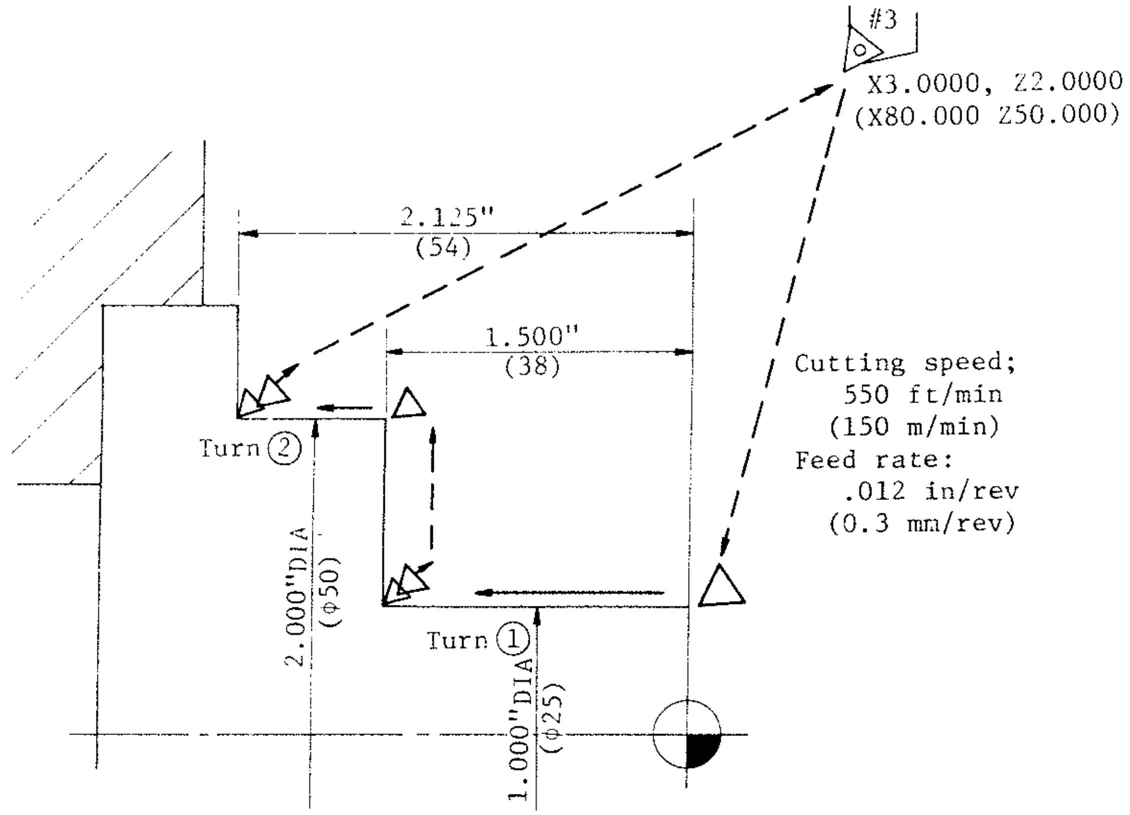
(INCH)

```
    T0200 ;
G96 S550 M03 ;
G00 X2.75 Z-.4 M28 T0202 ;
G01 X2.0 F.012 ;
    Z-.35 ;
G00 X1.55 Z0 ;
G01 X.71 F.012 ;
G00 Z1.0 ;
    X3.5 Z4.0 M29 ;
     F120 may be omitted.
```

(MM)

```
    T0200 ;
G96 S150 M03 ;
G00 X73.0 Z-10.0 M28 T0202 ;
G01 X50.0 F.3 ;
    Z-9.0 ;
G00 X42.0 Z0 ;
G01 X19.0 F.3 ;
G00 Z25.0 ;
    X80.0 Z100.0 M29 ;
     F.3 may be omitted.
```

3-2. Turning Operation



- Program -

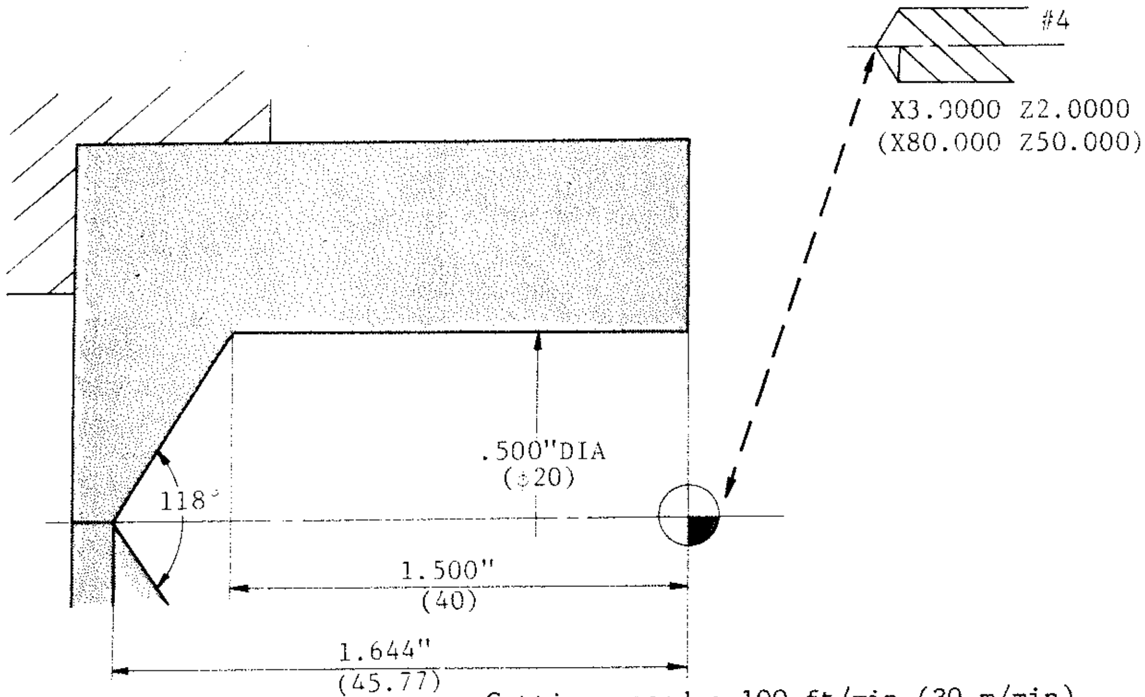
(INCH)

```
   T0300 ;    ----------------------- Select turret station
G50 S1500 ;   ----------------------- Clipped spindle speed
G96 S550 M03 ;  -------------------- Specify cutting speed spindle start
G00 X1.0 Z0.05 M28 T0303 ;  -------- Positioning, High pressure coolant
                                     ON, Wear offset No.3
G01 Z-1.5 F.012 ;  ----------------- Turn ① by .012 in/rev feedrate
    X1.15 Z-1.45 ;  ---------------- Relief
G00 X2.0 ;  ------------------------ Positioning for turn ②
G01 Z-2.125 F120 ;  ---------------- Turn ②
    X2.05 Z-2.07 ;  ---------------- Relief
G00 X3.0 Z2.0 M29 ;  --------------- The position without tool interference.
                                     Coolant OFF.
```

(MM)

```
   T0300 ;
G50 S1500 ;
G96 S150 M03 ;
G00 X25.0 Z.5 M28 T0303 ;
G01 Z-38.0 F.3 ;  ------------------ Turn ① by 0.3 mm/rev feedrate
    X26.0 Z-37.5 ;
G00 X50.0 ;
G01 Z-54.0 F.3 ;
    X51.0 Z53.5 ;
G00 X80.0 Z50.0 M29 ;
```

3-3. Drilling Operation



Cutting speed ; 100 ft/min (30 m/min)
Feed rate ; .015 in/rev (0.38 mm/rev)

In case of the drilling, constant cutting speed control (G96) can not be used. Therefore, a spindle speed must be figured out and it is specified with G97.

$$\text{Spindle speed} = \frac{12 \times \text{Cutting speed}}{\pi \times D} = \frac{12 \times 100}{\pi \times 0.5} = 764 \text{ (rpm)} \rightarrow \text{In Inch system}$$

$$\text{Spindle speed} = \frac{1000 \times \text{Cutting speed}}{\pi \times D} = \frac{1000 \times 30}{\pi \times 20} = 477 \text{ (rpm)} \rightarrow \text{In Metric system}$$

- Program -

(INCH)

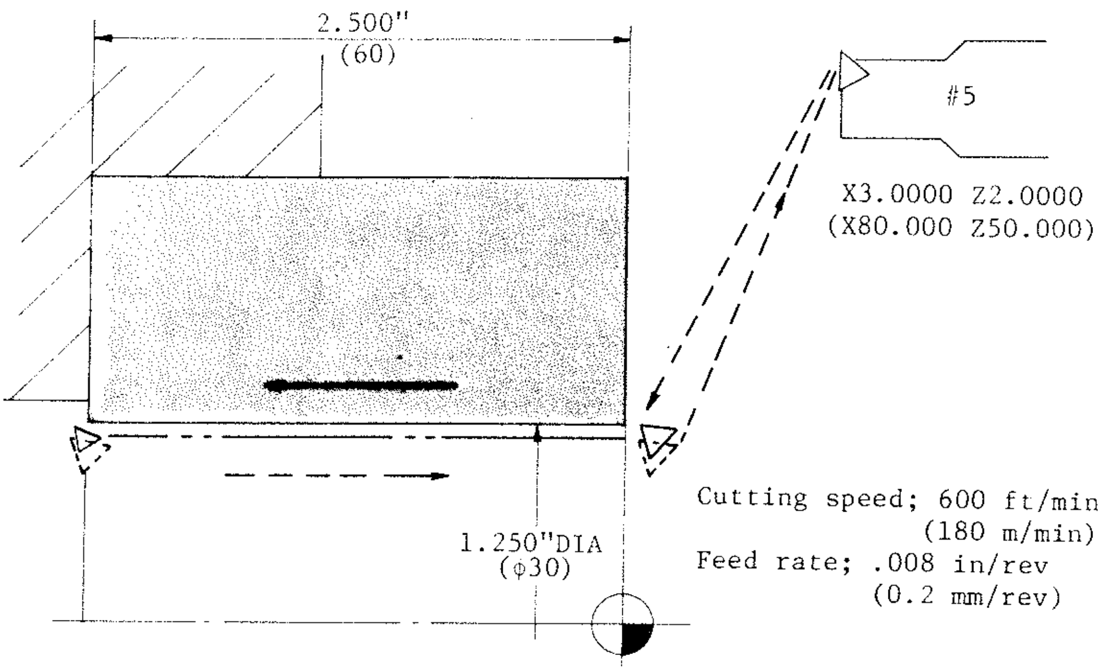
```
    T0400 ;        -------------------------- Select turret station
G97 S764 M03 ;     -------------------------- Spindle starts by 764 rpm
G00 X0 Z.1 T0404 M08 ;   ------------------- Positioning, coolant ON,
                                             Wear offset No.4
G01 Z-1.644 F.015 ;  ----------------------- Drilling by .015 in/rev feed rate
G00 Z.1 ;          -------------------------- Pull out drill
    X3.0 Z2.0 M09 ;  ----------------------- The position without tool
                                             interference.  Coolant OFF.
```

(MM)

```
    T0400 ;
G97 S477 M03 ;
G00 X0 Z.5 T0404 M08 ;
G01 Z-45.77 F.38 ;  ------------------------ Drilling by 0.38 mm/rev feed rate
G00 Z.5 ;
    X80.0 Z50.0 M09 ;
```

BND-S PRO 0-TC 90/8

3-4. Boring Operation



Cutting speed ; 550 ft/min (170 m/min)
Feed rate ; .012 in/rev (0.3 mm/rev)

\- Program -

(INCH)

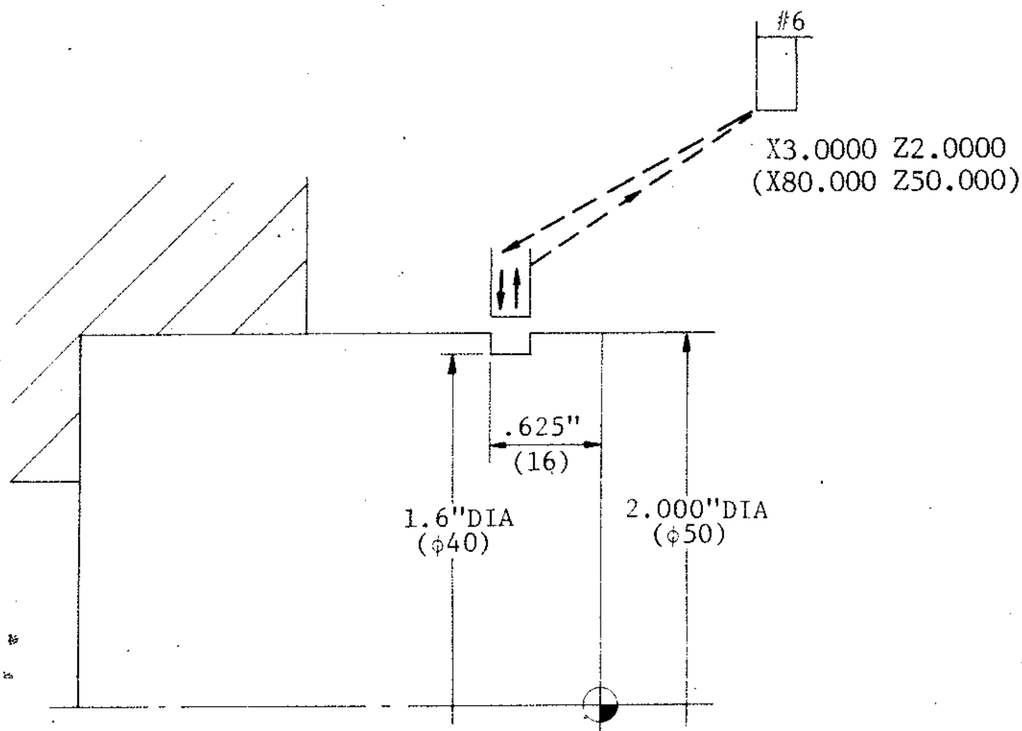
```
    T0500 ;
G50 S1500 ;
G96 S600 M03 ;
G00 X1.25 Z.05 M08 T0505 ;
G01 Z-2.51 F.008 ;
G00 X1.15 ;
    Z.05 ;
    X3.0 Z2.0 M09 ;
```

(MM)

```
    T0500 ;
G50 S1500 ;
G96 S180 M03 ;
G00 X30.0 Z.5 M08 T0505 ;
G01 Z-61.0 F.2 ;
G00 X29.0 ;
    Z.5 ;
    X80.0 Z50.0 M09 ;
```

BND-S PRO 0-TC 90/8

3-5. Grooving Operation



Cutting speed ; 400 ft/min (120 m/min)
Feed rate ; .005 in/rev (0.13 mm/rev)

\- Program -

(INCH)

```
    T0600 ;
G96 S400 M03 ;
G00 X2.1 Z-.625 M28 T0606 ;
G01 X1.6 F.005 M52 ;
G00 X2.1 M29 ;
    X3.0 Z2.0 M51 ;
```

(MM)

```
    T0600 ;
G96 S120 M03 ;
G00 X51.0 Z-16.0 M28 T0606 ;
G01 X40.0 F.13 M52 ;
G00 X51.0 M29 ;
    X80.0 Z50.0 M51 ;
```

M51 → Error detection OFF

M52 → Error detection ON

[NOTE]

If the grooved dimension is variant because the grooving tool or the workpiece is deflected, the dwell command (G04) may help. The dwell can be programmed as follows;

```
 .
G01 X1.6 F.005 M52 ;
G04 U0.5 ;  or G04 P500 ........... 0.5 sec. dwell time
G00 X2.1 M29 ;
    .
    .
    .
```


BND-S PRO 0-TC 90/8

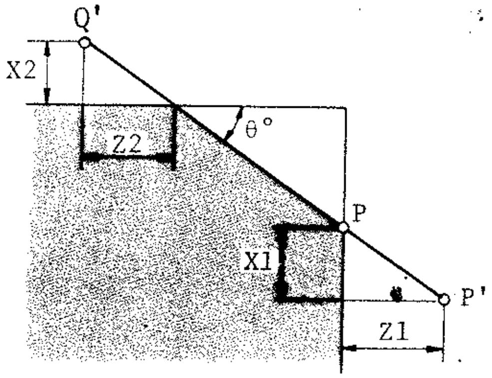
3-6. Taper Cutting and Chamfering Operation

Taper cutting and chamfering either inside or outside are programmed by using the same method when single point tool is used.
In this case, the dimension is affected by the tool nose radius.
Refer to item 4 for detailed explanation of the 'Tool Nose Radius Compensation'.
Usually, these operations are done by combining a turning, facing and boring application. However, 45° or 60° chamfers can also be done by using a form chamfering tool separately.

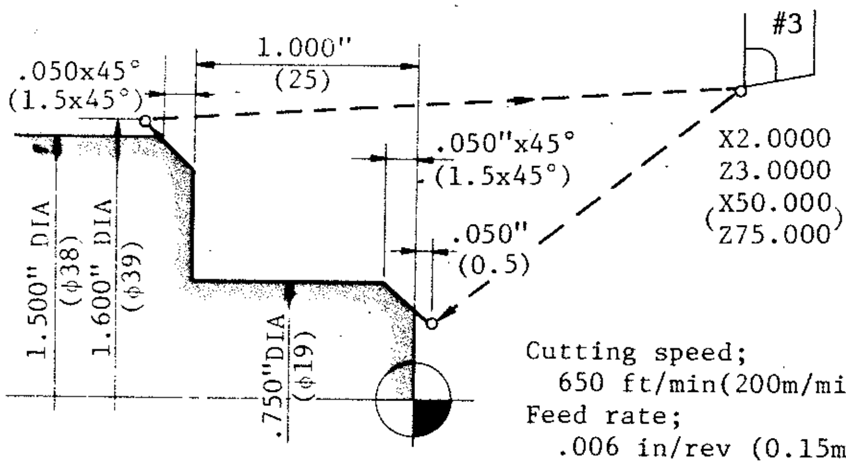
When the angle ($\theta°$) and the point (P) are given on the part drawing, the unknown value X1 and Z2 are calculated as follows;

$$X1 = Z1 \tan \theta°$$

$$Z2 = \frac{X2}{\tan \theta°}$$



- Example Program -



Cutting speed;
650 ft/min(200m/min)
Feed rate;
.006 in/rev (0.15mm/rev)

(INCH)

```
    T0300 ;
G50 S2500 ;
G96 S650 M03 ;
G00 X.55 Z.05 M28 T0303 ;
G01 X.75 Z-.05 F.006 ;
    Z-1.0 ;
    X1.6 C-.1 ;
G00 X2.0 Z3.0 M29 ;
```

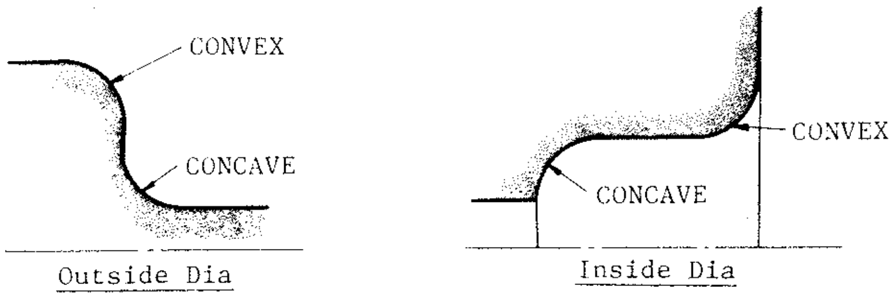
(MM)

```
    T0300 ;
G50 S2500 ;
G96 S200 M03 ;
G00 X15.0 Z.5 M28 T0303 ;
G01 X19.0 Z-1.5 F.15 ;
    Z-25.0 ;
    X39.0 C-2.0 ;
G00 X50.0 Z75.0 M29 ;
```

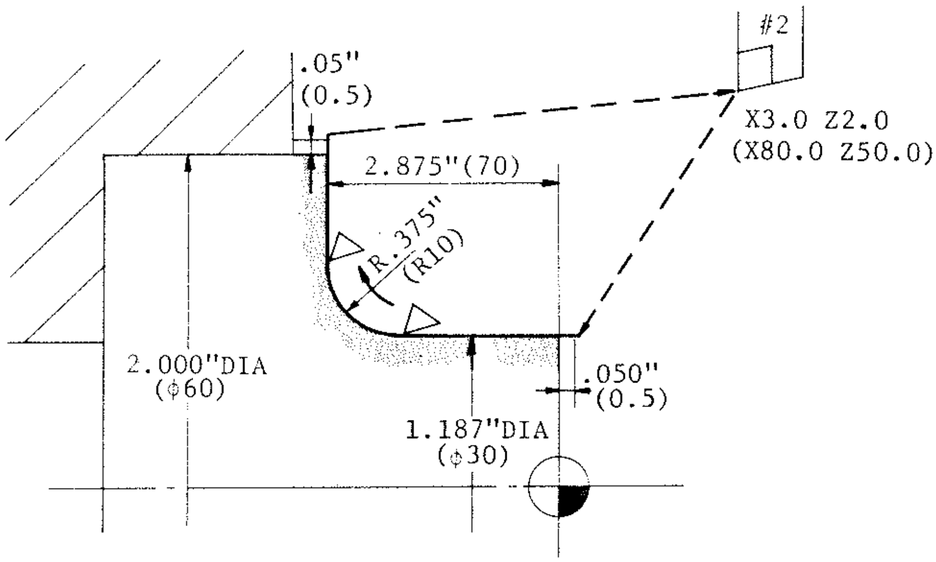
[NOTE]
This program does not consider tool nose radius compensation.

3-7. Radius Cutting Operation

Either concave or convex arcs are machined by specifying the code G02, G03 or G01.
The dimension of the radius is affected by the tool nose radius.
Refer to item 4 for detailed explanation of 'Tool Nose Radius Compensation'.



There are two kinds of methods to program the following circular cutting.



Cutting speed ; 550 ft/min (170 m/min)
Feed rate ; .012 in/rev (0.3 mm/rev)

(1) By using G02

(INCH)

```
    T0200 ;
G50 S2000 ;
G96 S550 M03 ;
G00 X1.187 Z.05 M28 T0202 ;
G01 Z-2.5 F.012 ;
G02 X1.937 Z-2.875 I.375 ;
G01 X2.1 ;
G00 X3.0 Z2.0 M29 ;
```

(MM)

```
    T0200 ;
G50 S2000 ;
G96 S170 M03 ;
G00 X30.0 Z0.5 M28 T0202 ;
G01 Z60.0 F.3 ;
G02 X50.0 Z70.0 I10.0 ;
G01 X61.0 ;
G00 X80.0 Z50.0 M29 ;
```

(2) By using G01

(INCH)

```
    T0200 ;
G50 S2000 ;
G96 S550 M03 ;
G00 X1.187 Z.05 M28 T0202 ;
G01 Z-2.875 R.375 F.012 ;
    X2.1 ;
G00 X3.0 Z2.0 M29 ;
```

(MM)

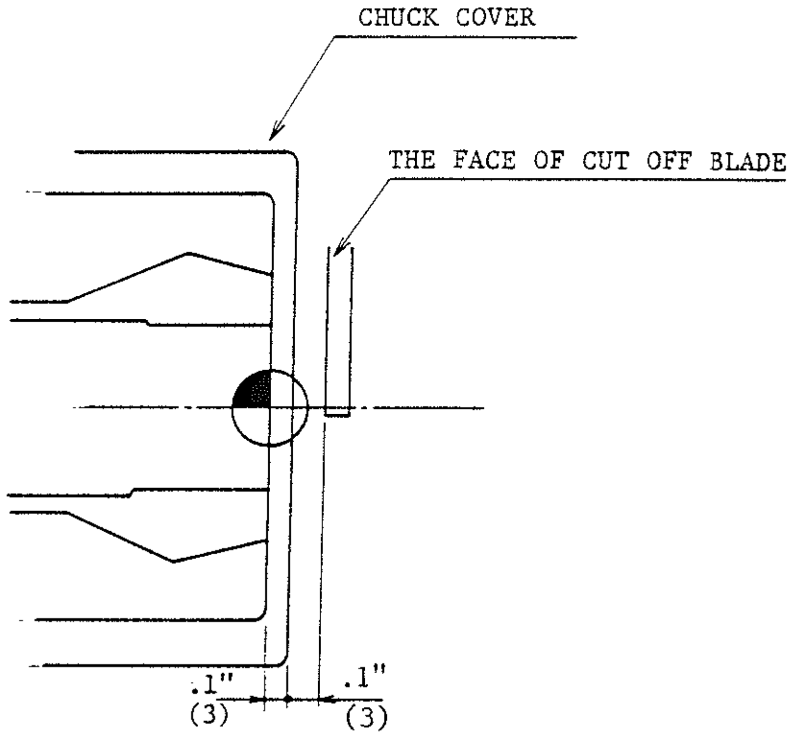
```
    T0200 ;
G50 S2000 ;
G96 S170 M03 ;
G00 X30.0 Z0.5 M28 T0202 ;
G01 Z-70.0 R10.0 F.3 ;
    X61.0 ;
G00 X80.0 Z50.0 M29 ;
```

[NOTE]

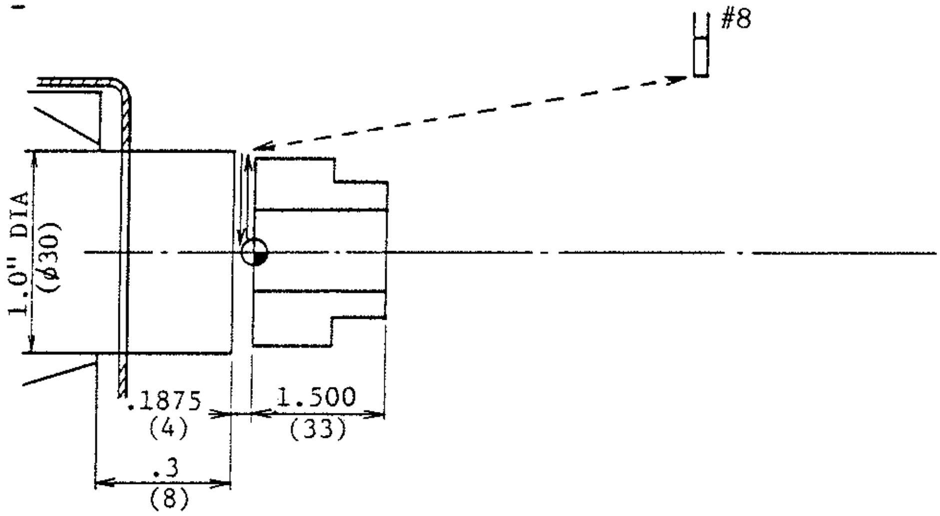
These programs do not consider tool nose radius compensation.

3-8. Cutting off Operation

Cut-off position on bar work should be .1"(3mm) for BND-20S, BND-34S away from the face of the chuck sleeve as shown an example in below.



- Example -



- Program -

```
  (INCH)                          (MM)
     T0800 ;                         T0800 ;
G50 S2600 ;                     G50 S2600 ;
G96 S400 M03 ;                  G96 S120 M03 ;
G00 X1.05 Z0 T0808 ;            G00 X31.0 Z0 T0808 ;
G01 X0 F30000 ;                 G01 X0 F500 ;
G00 X1.05 ;                     G00 X31.0 ;
    X2.0 Z3.0 ;                     X40.0 Z50.0 ;
```

[NOTE 1]

Constant cutting speed control is useful for cutting off operation. In this case, a clipped spindle speed (rpm) should be programmed in the block containing G50.
In the above example, 2600 rpm is clipped spindle speed.

G50 S2600 ;

→ Clipped spindle speed

[NOTE 2]

When the cut off tool is bloken or worn out, the part may not be cut off completely. Therefore, when the cut off tool returns to home position, it must pull out of the bar diameter and return to home position.

3-9. Bar Feedout Operation

The machine accepts the following two bar feeding methods.

① By standard bar feeding equipment

② By automatic bar feeder

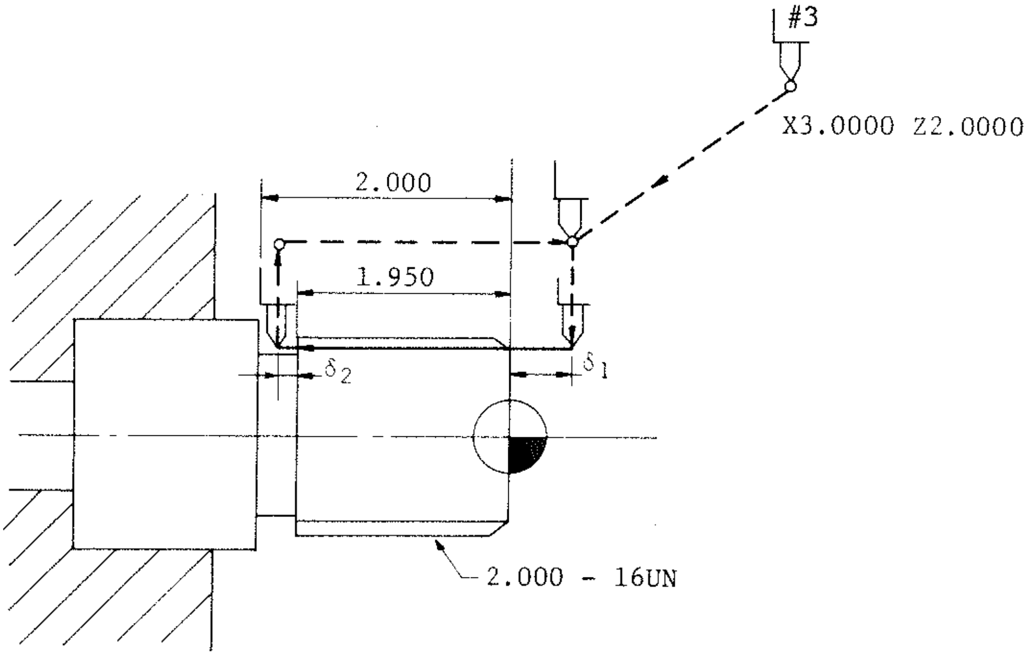
In the case of ①, a series of sequence the chuck open ⟶ bar feed out ⟶ chuck close is performed by specifying M12.

However, M12 is not used for ②. This makes it necessary to program each process. Refer to the M84 description for the programming method.

(NOTE)
Double barfeed is possible by specifying the each motion individually for barfeeding.

3-10. Threading Operation

In Inch system

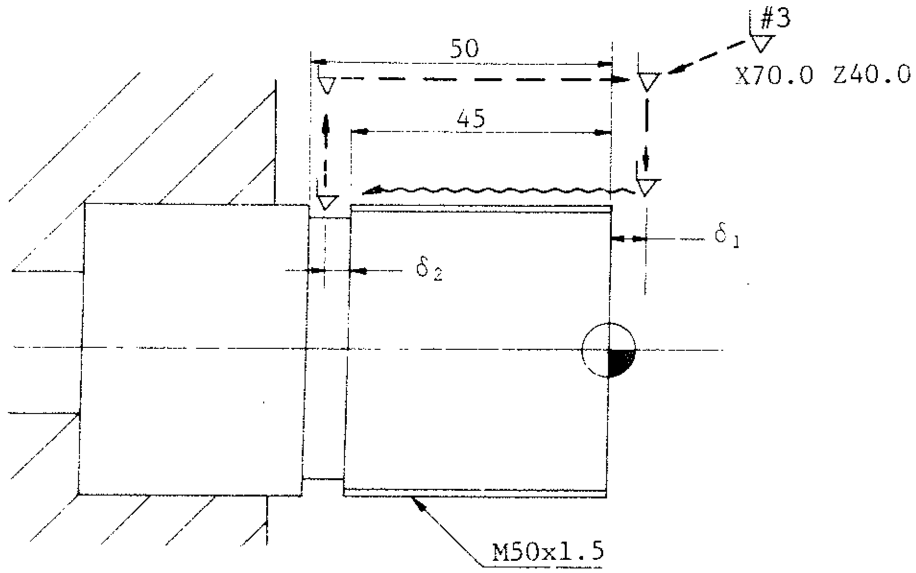


Spindle speed ; 670 rpm
δ_1 ; .100 in
δ_2 ; .025 in

\- Program -

```
      (INCH)

     T0300 ;
 G97 S670 M03 ;
 G00 X3.0 Z.1 M28 T0303 ; .......... Rapid approach
     X1.96 ; ....................... Infeed for 1st pass
 G32 Z-1.98 F.0625 ; ............... 1st pass
 G00 X3.0 ; ........................ Relief
     Z.1 ;
     X1.94 ; ....................... Infeed for 2nd pass
 G32 Z-1.98 ; ...................... 2nd pass
 G00 X3.0 ;
     Z.1 ;
     X1.93 ; ....................... Infeed for 3rd pass
 G32 Z-1.98 ; ...................... 3rd pass
 G00 X3.0 ;
     Z.1 ;
     X1.9250 ; ..................... Infeed for 4th pass
 G32 Z-1.98 ; ...................... 4th pass
 G00 X3.0 ;
     Z.1 ;
     X1.9215 ; ..................... Infeed for final pass
 G32 Z-1.98 ; ...................... Final pass
 G00 X3.0 ;
     X3.0 Z2.0 M29 ;
```

In Metric system



Cutting speed; 90 m/min

-Program-

```
        (MM)

      T0300 ;
  G97 S573 M03 ;
  G00 X60.0 Z20.0 M28 T0303 ; ....... Rapid approach
      X49.582 ; ..................... Infeed for 1st pass
  G32 Z-47.0 F1.5 ; ................. 1st pass
  G00 X60.0 ; ....................... Relief
      Z20.0 ;
      X49.214 ; ..................... Infeed for 2nd pass
  G32 Z-47.0 ; ...................... 2nd pass
  G00 X60.0 ;
      Z20.0 ;
      X48.845 ; ..................... Infeed for 3rd pass
  G32 Z-47.0 ; ...................... 3rd pass
  G00 X60.0 ;
      Z20.0 ;
      X48.478 ; ..................... Infeed for 4th pass
  G32 Z-47.0 ; ...................... 4th pass
  G00 X60.0 ;
      Z20.0 ;
      X48.109 ; ..................... Infeed for final pass
  G32 Z-47.0 ; ...................... Final pass
  G00 X60.0 ;
      X70.0 Z40.0 M29 ;
```

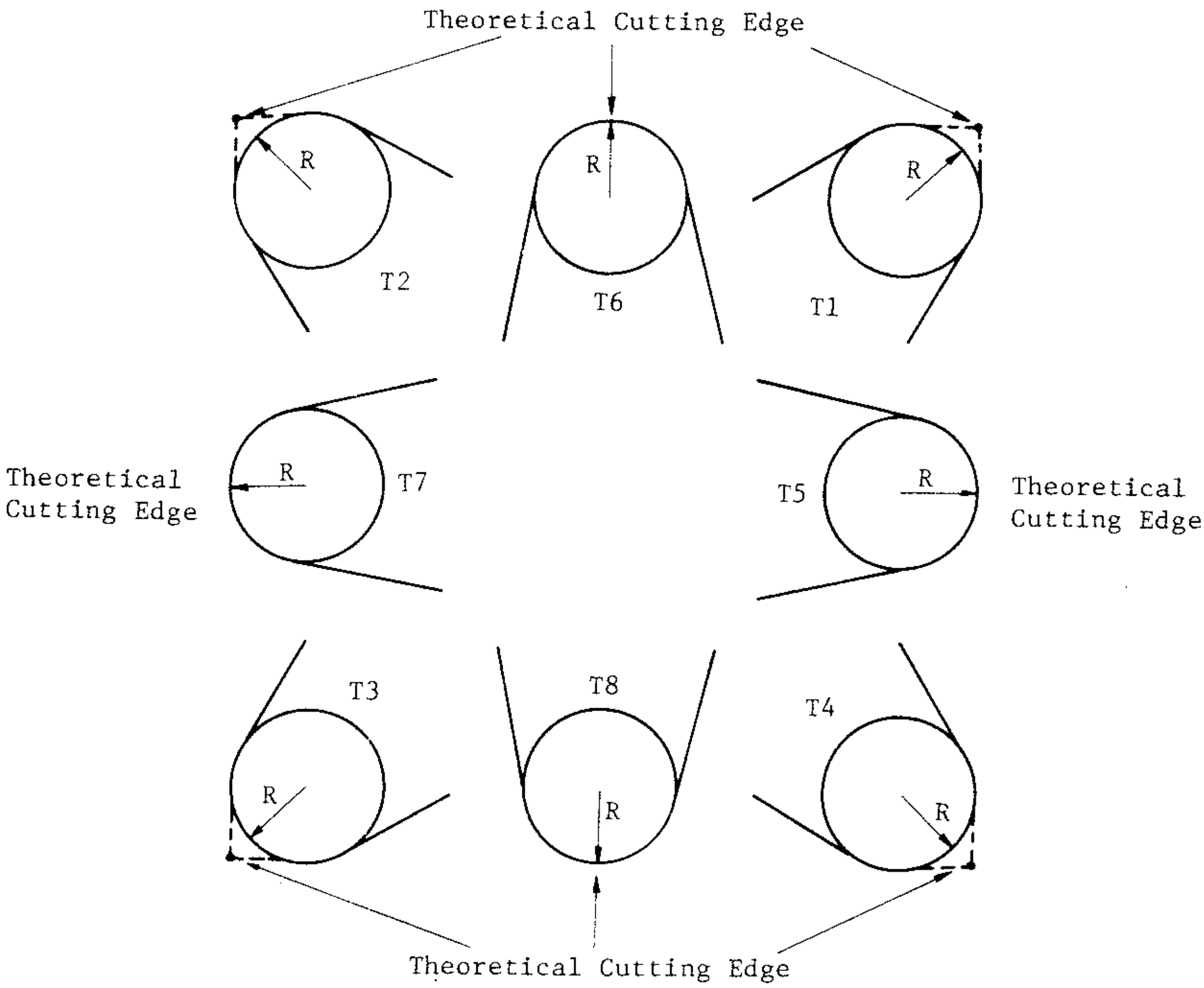
(1) G40 to G42 are modal G codes and they are retained until the other one is commanded.
The tool nose radius compensation is cancelled in the following cases.
- o When the power is turned on.
- o When G40 is specified.
- o When the compensating number specify as '00'.

(2) The compensation mode (G40 to G42) can be specified in the same block with other G code.

[2] Command of the direction of the theoretical cutting edge

The direction of the theoretical cutting edge is selected from 8 types (T1 to T8) shown in below, depending on the direction of the tool nose center viewed from the theoretical cutting edge, on the programming.
This selected number (Standard tool nose number) must be set corresponding to offset number by MDI operation.

BND-S PRO 0-TC 90/8



[3] Setting of nose radius value

The value of tool nose radius is set by MDI operation, corresponding to the offset number.

[4] Setting contents by MDI operation

The contents to be set by MDI operation are as follows;

- o Wear offset number
- o Offset value in X-axis
- o Offset value in Z-axis
- o Nose radius value
- o Direction of theoretical cutting edge (Standard tool nose number)

[5] Command of the compensation

Tool nose radius compensation starts when both G code of the compensation mode (G41 or 42) and offset number by T-code are specified, and stop when cancellation of the compensation (G40) is specified.

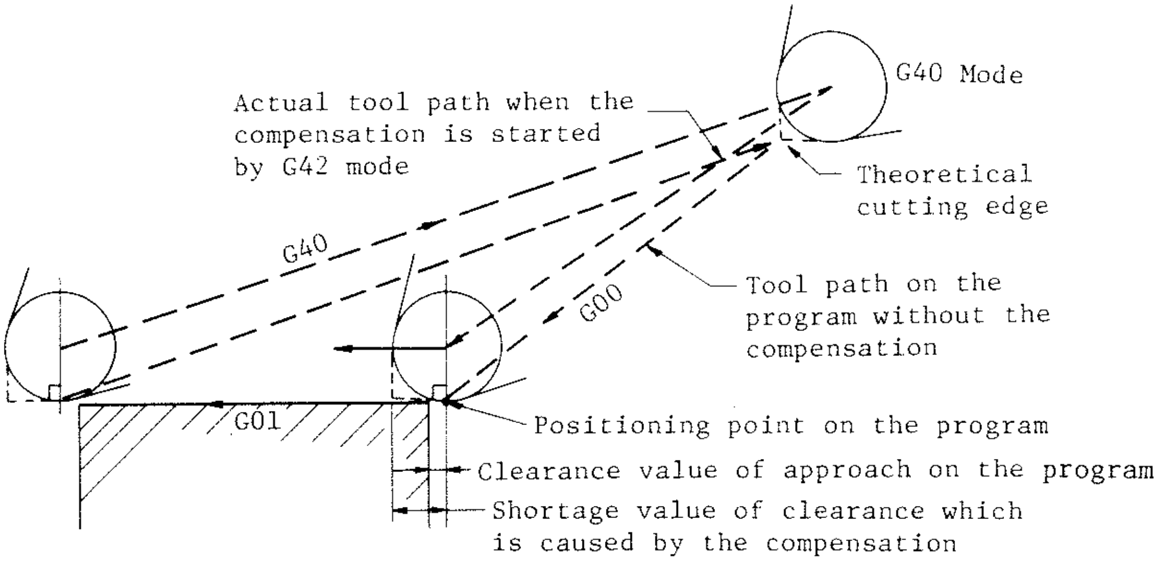
General form of programming with tool nose radius compensation is as follows;

```
            Previous operation
                    .
                    .
      ---------------------------------------------------
      |    T04 00 ;                                     |
      | G(41 or 42) G00 X____  Z_____ T04 04 ;          |
      |             .                                   |
      |             .                                   |
      |             .                                   |
      | G40 X____  Z_____      ;                        |
      ---------------------------------------------------
           T    00 ;
                    .
                    .
              Next operation
```

(1) While the tool nose radius compensation is effective, the control reads two to three blocks ahead of data to calculate the next movement position. Therefore, the data with movement command in two blocks ahead is necessary to perform the compensation. If two blocks do not include movement command continuously, the compensation will be temporarily cancelled. It means that two continuous blocks without movement command such as M or S code, only G code, G04 (Dwell) or setting of stored stroke limit etc. should not be programmed.

(2) If G41 is commanded while in the G41 mode, or G42 is commanded while in the G42 mode, the compensation will not be performed correctly.

(3) When the compensation is started, the tool will start to move from the accordant point consisting of the programmed coordinate position and the theoretical cutting edge position as the following sketch. Therefore, the clearance of approach to the workpiece should be bigger than normal when the compensation is used.

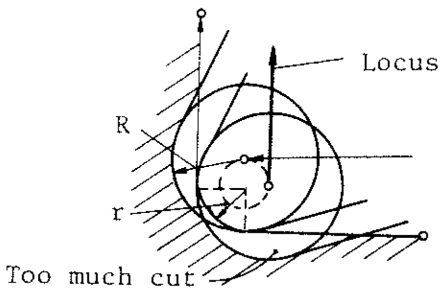


The nose radius center is moved to the position which is perpendicularly away by nose radius value from specified position in the first block after specifying the compensation.

(4) The command in the next block of specifying the compensation and the command in the preceding block of cancelling the compensation must be the block included G00, G01, G90, G94 or G70.
If a circular is specified in there, the alarm (No.34) will occur because the junction of the locus of the tool nose radius center can not be calculated.

4-4 Prohibited command on the compensation

(1) Cutting of inside corner which the specifying radius (r) is smaller than the tool nose radius (R), can not be commanded.
If commanded, the alarm (No.41) will occur.

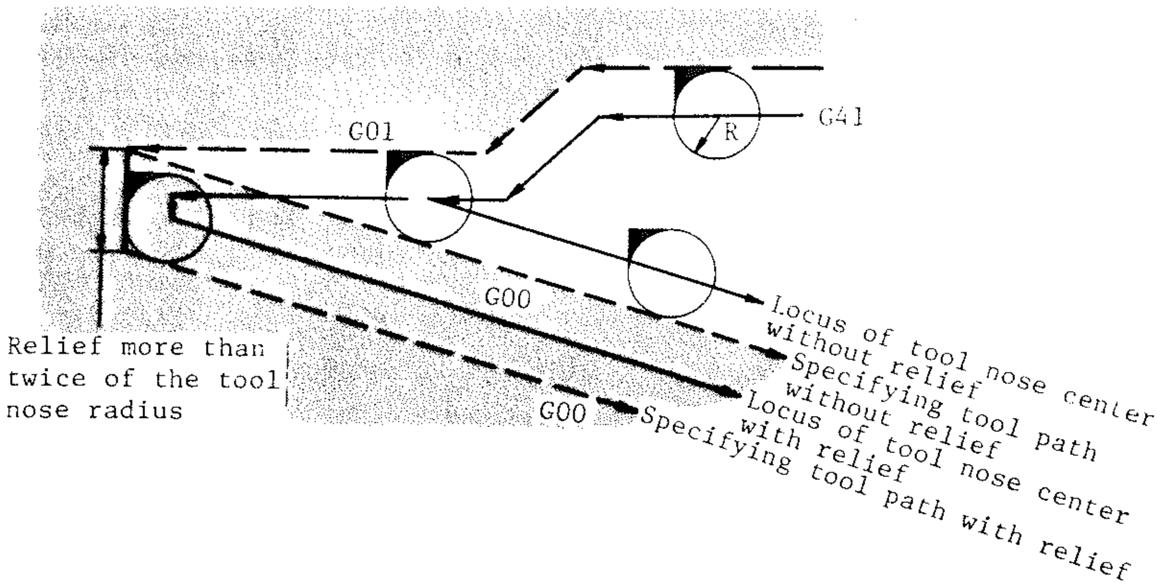


(2) Cutting of a groove smaller than the tool nose diameter can not be commanded.
If commanded, the alarm (No.41) will occur.

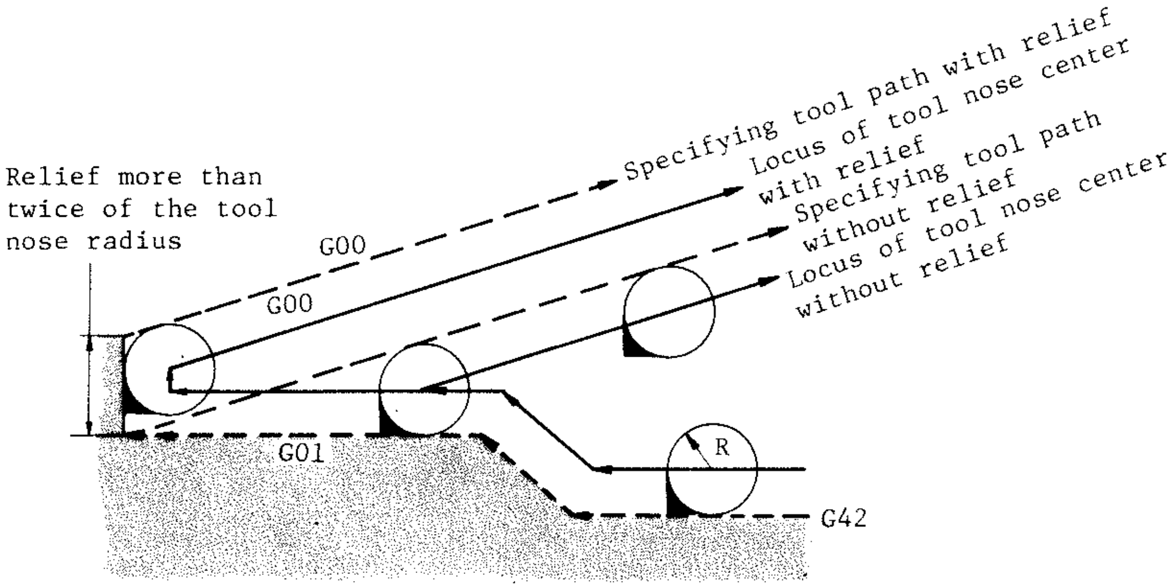
(3) Cutting without junction on the locus of tool nose radius center can not be commanded.
If commanded, the alarm (No.33) will occur.

(4) When a sharp angle motion is specified during the compensation as shown in below, the tool may not reach to specifying point or may come back to the same locus as it goes forward.
In such case, program the relief more than twice of the tool nose radius or program without using the compensation.

BND-S PRO 0-TC 90/8



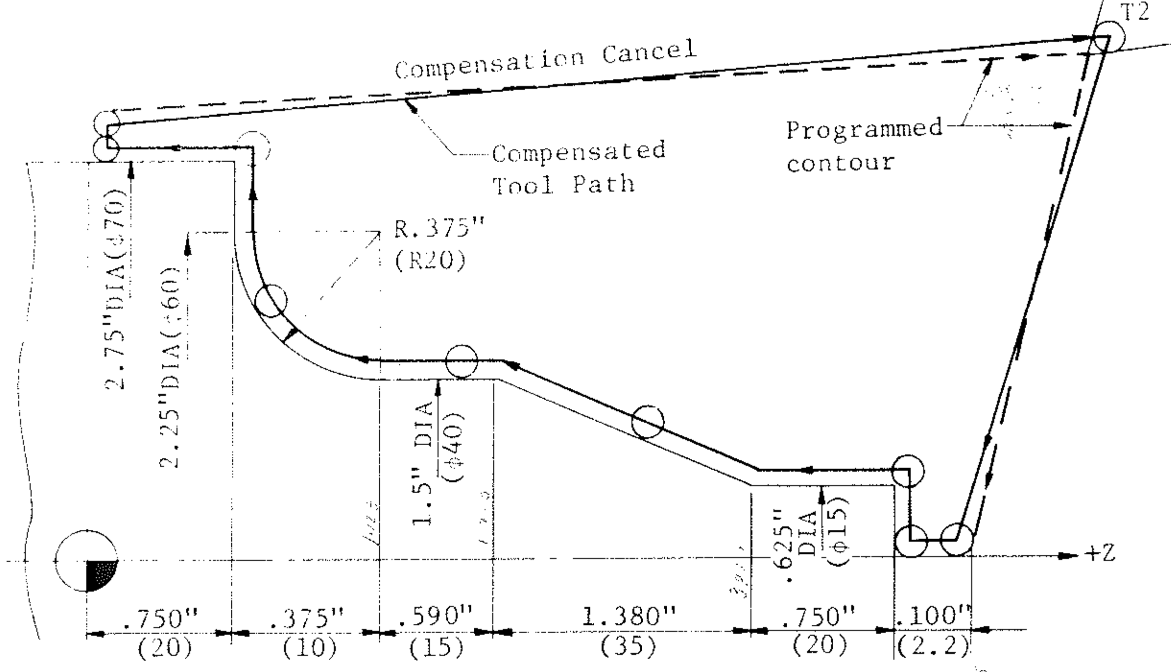
G01
G41
R
G00
G00
Relief more than
twice of the tool
nose radius
Locus of tool nose center
without relief
Specifying tool path
without relief
Locus of tool nose center
with relief
Specifying tool path with relief



Specifying tool path with relief
Locus of tool nose center
with relief
Specifying tool path
without relief
Locus of tool nose center
without relief
Relief more than
twice of the tool
nose radius
G00
G00
G01
R
G42

Sample program for Using Tool Nose Radius Compensation

[1] Sample program - 1



In Inch system

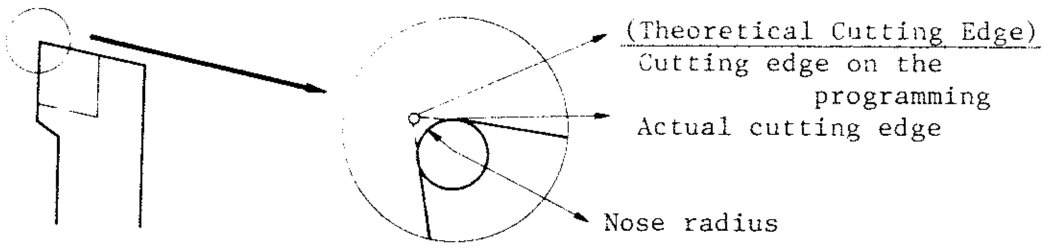
```
G20       ;
          T200 ;
G50       S4500 ;
G96       S400 M03 ;
G42  G00  X0 Z3.945 M28 T0202 ; ---- The compensation start
G01       Z3.845 F.01 ;
          X.625 F.012 ;
          Z3.095 F.03 M52 ;
          X1.5 Z1.715 ;
          Z.75 R.375 F.012 M51 ;
          X2.75 ;
          Z0 F.015 ;
          X2.95 ;
G00       X4.0 Z5.0 M29 ;
G40       ; ------------------------ The compensation finish
```

In Metric system

```
G21       ;
          T200 ;
G50       S4500 ;
G96       S120 M03 ;
G42  G00  X0 Z102.2 M28 T0202 ; ---- The compensation start
G01       Z100.0 F.15 ;
          X15.0 F.2 ;
          Z80.0 F.3 M52 ;
          X40.0 Z45.0 ;
          Z20.0 R10.0 F.25 M51 ;
          X70.0 ;
          Z0 F.3 ;
          X80.0 ;
G00       X100.0 Z170.0 M29 ;
G40       ;  ----------------------- The compensation finish
```

4. TOOL NOSE RADIUS COMPENSATION (G40 ∿ G42)

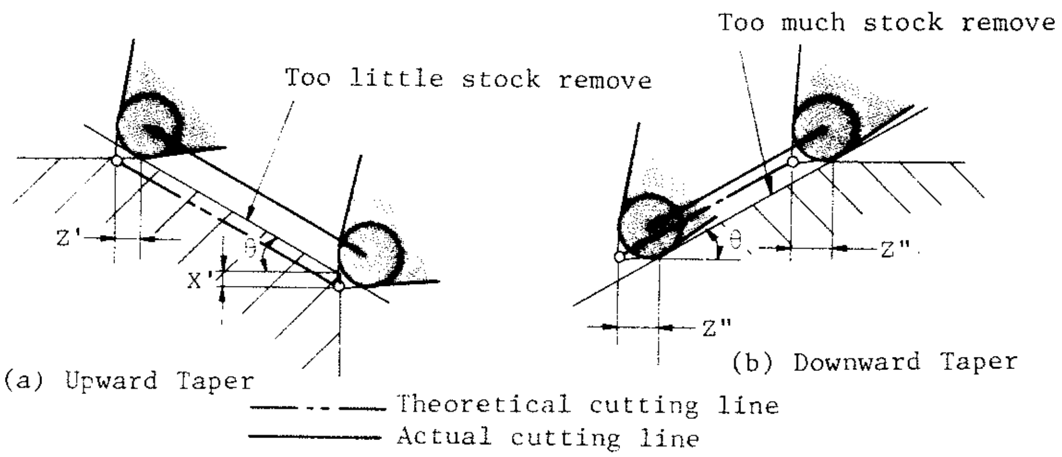
Most cutting tools must have the nose radius on the cutting edge. The tool nose radius will make a different dimension than what programmed especially in taper and circular cutting. This difference should be compensated on the programming.
There are two kinds of method to compensate the tool nose radius, by manual calculation or automatically. However, BND-20S, BND-34S use the automatic tool nose radius compensation function (G40 to G42).



4-1. Different Dimension by Tool Nose Radius

(1) In case of taper cutting

When a taper is cutting, different dimensions between theoretical cutting dia. and actual cutting dia. will occur as shown in sketch below.
This difference will cut less on the upward taper and cut more on the downward taper.

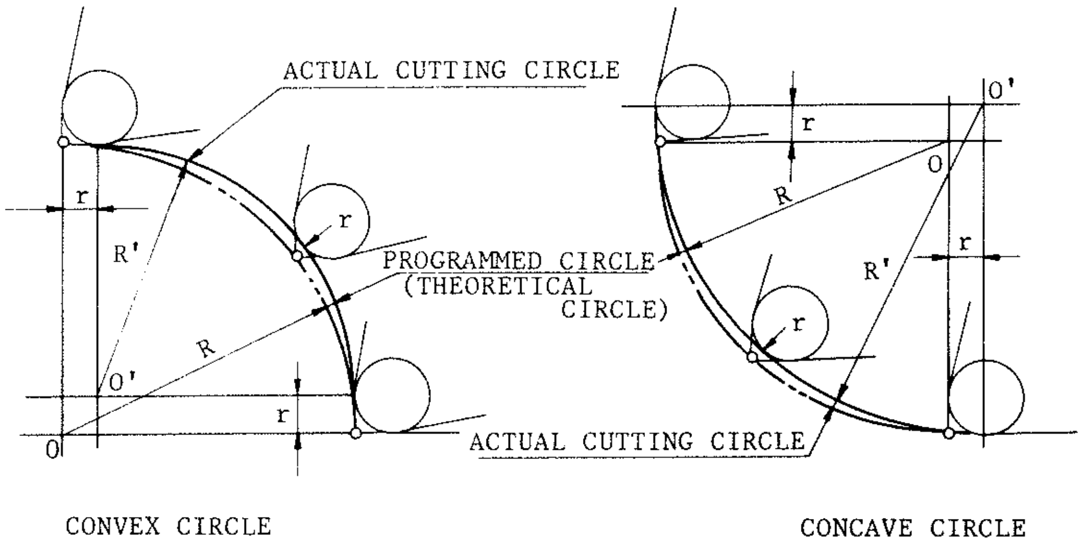


(2) In case of circular cutting

In case of circular cutting, the actual cutting radius differs from the theoretical radius as follows;

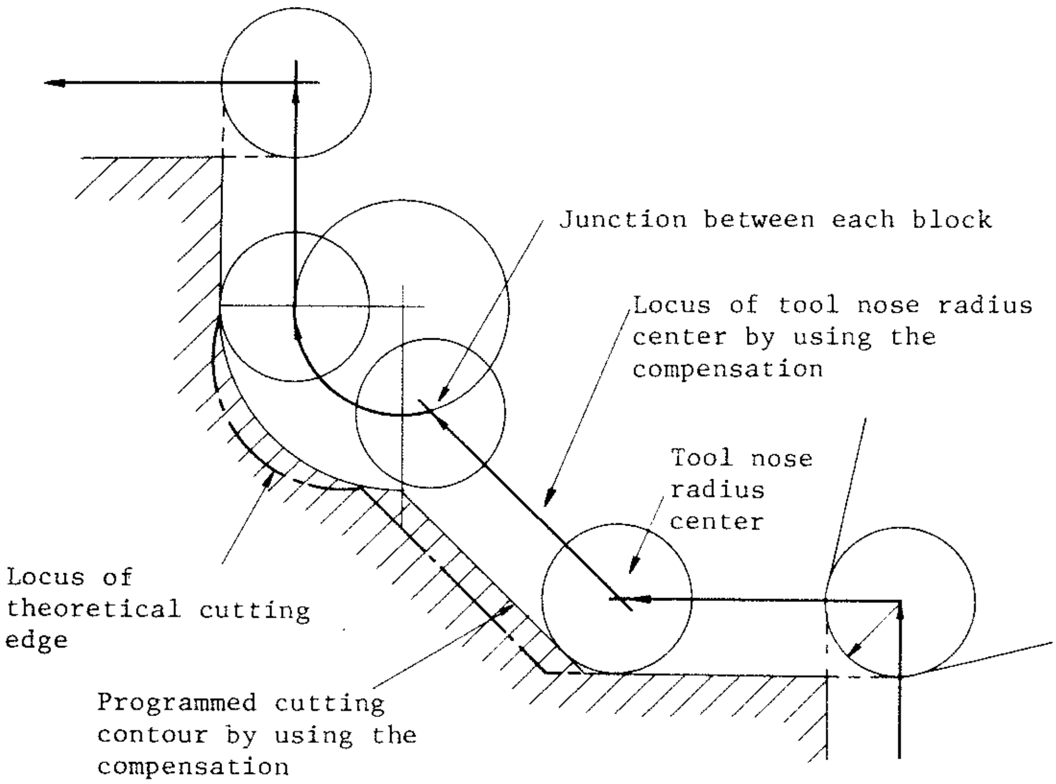
CONVEX CIRCLE
The actual cutting radius will be smaller than the programmed radius.
This is the same as the center of the circle moved to point 0 to 0' on the sketch below.

CONCAVE CIRCLE
The actual cutting radius will be larger than the programmed radius.
This is the same as the center of the circle moved to point 0 to 0' on the sketch below.



4-2. The motion of tool nose radius compensation

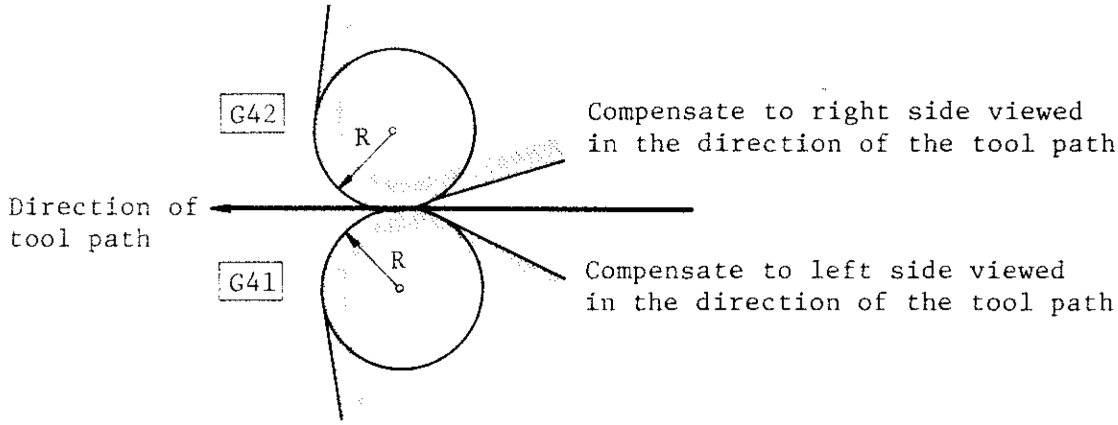
The tool nose radius compensation controls the movement command on the programming which is made by using the theoretical cutting edge. By programming the compensation, this movement command is exchanged to the movement of the tool nose radius center.
The center position of tool nose radius is figured by compensated direction (G41, G42) and the data of tool nose radius, according to the programming path.
While the compensation is effective, the control reads two to three blocks ahead of data to calculate the next movement position.
Thus, the locus of the tool nose radius center is calculated automatically in the control system and the programming path is replaced to the path of the tool nose radius center.



4-3 Command of Tool Nose Radius Compensation

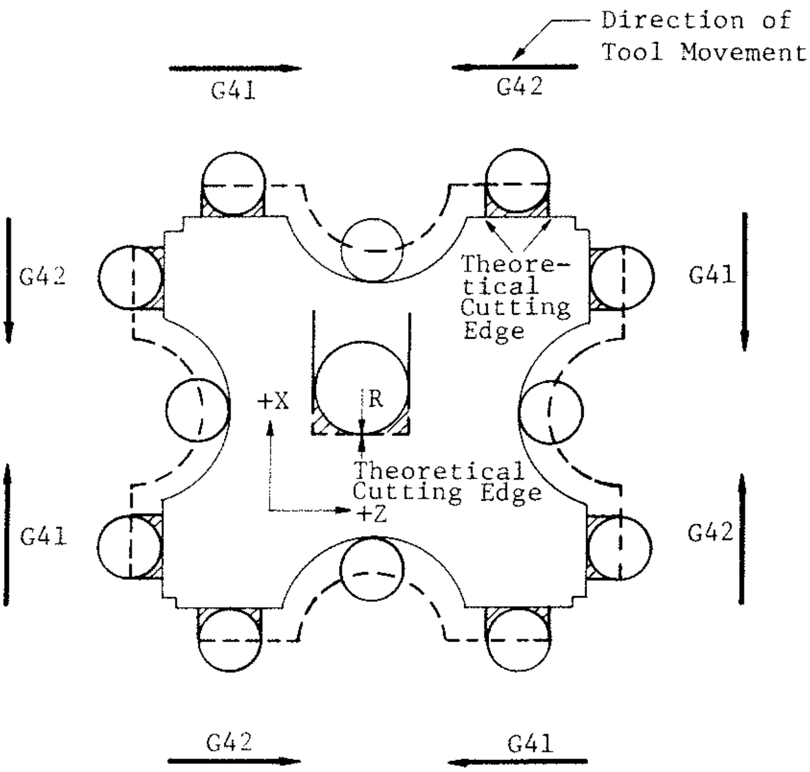
[1] G code designation

There are three kinds of G codes (G40 to G42) for specifying the compensation mode, depending on the compensated direction.

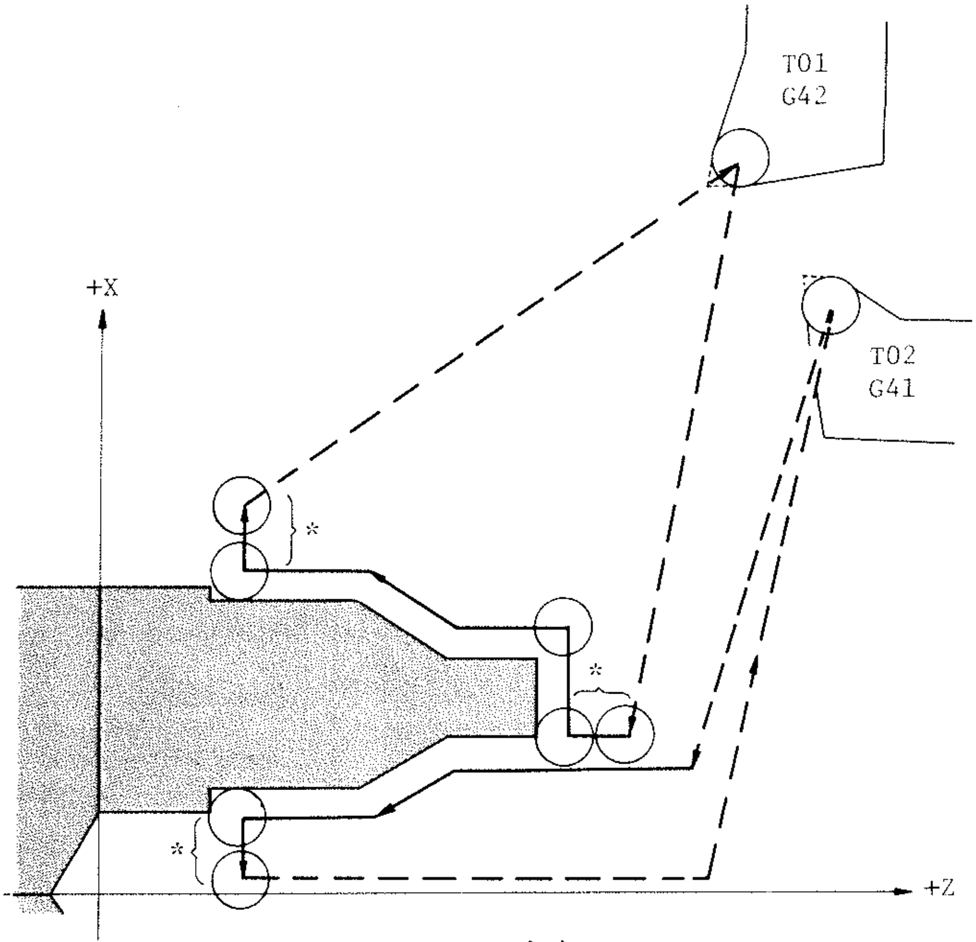


G40 Cancellation of the compensation

Generally, G42 is used for cutting outer diameter and G41 is used for cutting inner diameter.



[2] Sample program - 2



Make relief at marked '*'

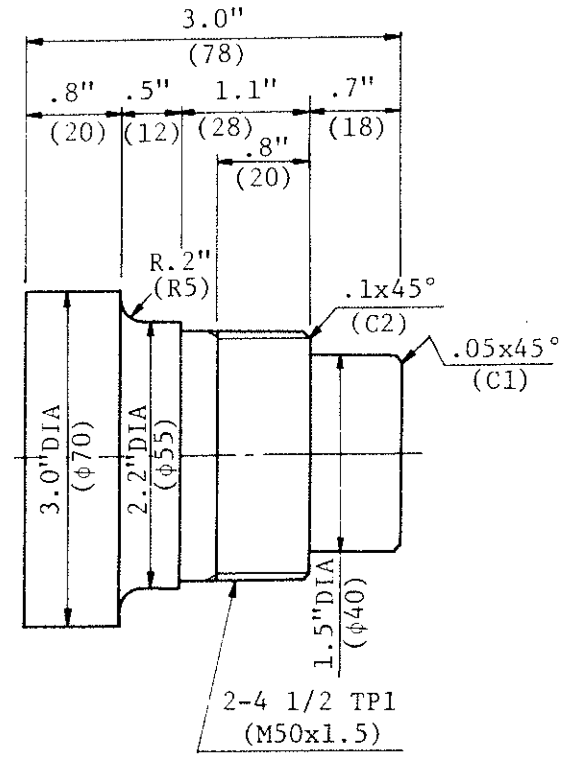

```
G20 or G21 ;
           T0100 ; --------- Selection of turret station No.1
G96        S500 M03 ;
G42 G00    X____ Z____ M28 T0101 ;
G01        Z____ F____ ;
           X____ ;
           Z____ ;
           X____ Z____ ;
           Z____ ;
           X____ ;
G40        X____ Z____ M29 ;
           M01 ;
           T0200 ; --------- Selection of turret station No.2
G96        S120 M03 ;
G41 G00    X____ Z____ M28 T0202 ;
G01        Z____ F____ ;
           Z____ ;
           X____ Z____ ;
           Z____ ;
           X____ ;
G00        Z____ ;
G40        X____ Z____ M29 ;
           M01 ;
```

5. EXAMPLE OF PROGRAMMING

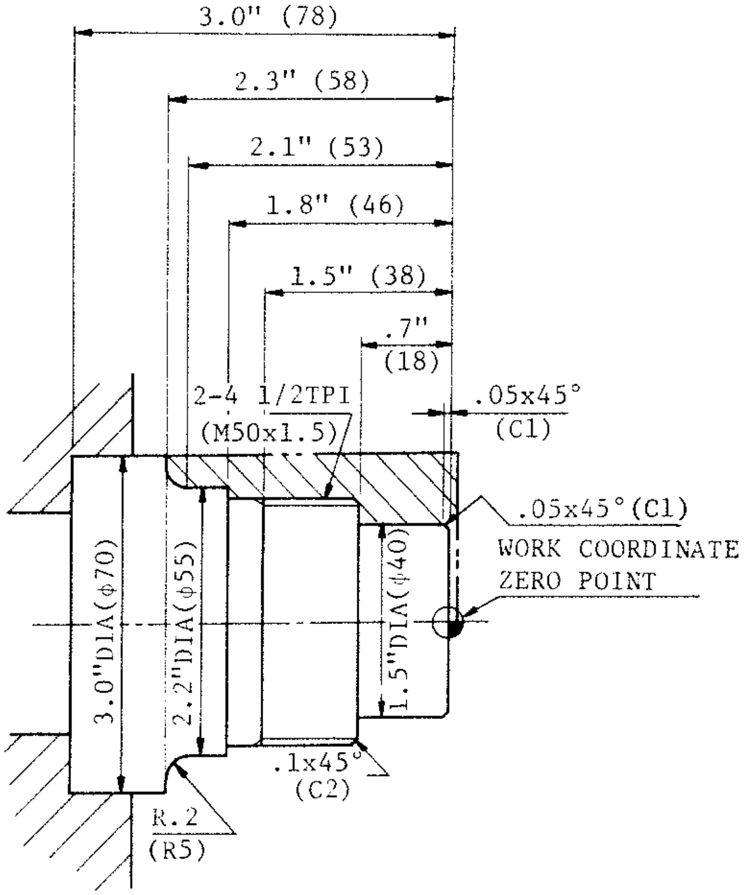
5-1. Part Dimensioning

Change style of part dimensioning for easier programming

- ORIGINAL DRAWING -



- DRAWING OF THE PART DIMENSIONING -



5-2. Part Layout Sheet

To make a layout of the machining, determine the cutting process, cutting conditions, cutting tools and tooling location etc.

TOOL POST NO.	DESCRIPTION	DIAMETER in (mm)	SPINDLE SPEED RPM	SURFACE SPEED FT/MIN (m/min)	FEED IN/rev (mm/rev)	LENGTH in (mm)	TIME(SEC)	
							CUT	MOVE
1	FACING	3.05 -0 (71 -0)	840 -4500 (909 -4500)	660 (200)	0.008 (0.2)	1.565 (36.5)	4.4 (4)	
2	ROUGH TURNING	2.6 (60)	970 (1061)	660 (200)	0.012 (0.3)	2.58 (62.5)	13.3 (11.8)	
2	"	2.8 -2.12 (71 -54)	900 -1189 (897 -1179)	660 (200)	0.012 (0.3)	2.47 (58.5)	11.8 (11.3)	
2	"	2.2 -1.89 (57 -48)	1146 -1334 (1117 -1326)	660 (200)	0.012 (0.3)	2.1 (52.2)	8.5 (8.5)	
2	"	2 -1.6 (52 -41)	1260 -1576 (1224 -1553	660 (200)	0.012 (0.3)	.91 (25.2)	3.2 (3.6)	
3	FINISH TURNING	3 -1.5 (70 -40)	840 -1680 (909 -1591)	660 (200)	0.006 (0.15)	3 (71.85)	24 (23)	
4	THREADING	2 (50)	630 (637)	330 (100)	0.222220 (1.5)	1×6 =6 (26×6 =156)	2.6 (10)	

5-3. Work Coordinate system Setting and Tool Selecting Position

(1) Work coordinate system setting

By specifying T□□00 ;, the current tool position translates to the coordinate value on the work coordinate system, at the same time as the tool is selected, regardless of the turret slide position. This eliminates the need of setting the work coordinate system in programming.

The work coordinate zero point is set by writing the position of each tool on the work coordinate system to the tool position memory when setting the tools.

5-4. Process Sheet

The process sheet can be made based by the part layout sheet as shown in item 5-2, with considering the toolings, the tool nose radius compensation and constant cutting speed control etc.
The following shows the program of this sample part.

[1] BND PROCESS SHEET

In inch system

```
O0001 ;
N201 G50 S3000 ;
N202     T0100 ;
N203 G96 S660 M03 ;
N204 G00 X3.04 Z0 T0101 ;
N205 G01 X-.08 F.008 ;
N206 G00 Z.8 ;
     G00 X0 Z0 T0000 ;
N207     M01 ;


N301     T0200 ;
N302 G96 S660 M03 ;
N303 G00 X2.6 Z.08 T0202 ;
N304 G01 Z-2.28 F.012 ;
N305     X3.04 ;
N306 G00 Z.08 ;
N307     X2.12 ;
N308 G01 Z-1.78 ;
N309     X2.4 ;
N310     Z-2.1 ;
N311     X2.7 ;
N312 G00 Z.08 ;
N313     X1.89 ;
N314 G01 Z-.68 ;
N315     X2.1 ;
N316     Z-1.78 ;
N317     X2.5 ;
N318 G00 Z.08 ;
N319     X1.6 ;
N320 G01 Z-.68 ;
N321     X1.9 ;
N322 G00 Z.8 ;
     G00 X0 Z0 T0000 ;
N323     M01 ;
```

```
N701           T0300 ;
N702 G96       S660 M03 ;
N703 G41 G00 X1.24 Z.08 T0303 ;
N704 G01       X1.5 Z-.05 F.006 ;
N705           Z-.7 ;
N706           X1.8 ;
N707           X2.0 Z-.8 ;
N708           Z-1.8 ;
N709           X2.16 ;
N710           X2.2 Z-1.82 ;
N711           Z-2.3 R.2 ;
N712           X3.5 ;
N713 G40 G00 Z.8 ;
     G00       X0 Z0 T0000 ;
N714           M01 ;


N801           T0400 ;
N802 G97       S630 M03 ;
N803 G00       X3.0 Z-.5 T0404 M23 ;
N804           M28 ;
N805 G92       X1.96 Z-1.54 F.22222 ;
N806           X1.91 ;
N807           X1.87 ;
N808           X1.83 ;
N809           X1.79 ;
N810           X1.7594 ;
N811 G00       M29 ;
N812           M24 ;
N813           M05 ;
N814 G27       X0 Z0 T0000 ;
N815           M02 ;
```

In Metric system

```
O0001 ;
N201 G50 S3000 ;
N202     T0100 ;
N203 G96 S200 M03 ;
N204 G00 X72.0 Z0 T0101 ;
N205 G01 X-2.0 F.2 ;
N206 G00 Z10.0 ;
     G00 X0 Z0 T0000 ;
N207     M01 ;


N301     T0200 ;
N302 G96 S200 M03 ;
N303 G00 X60.0 Z2.0 T0202 ;
N304 G01 Z-55.0 F.3 ;
N305     X71.0 ;
N306 G00 Z2.0 ;
N307     X54.0 ;
N308 G01 Z-45.7 ;
N309     X56.0 ;
N310     Z-53.0 ;
N311     X61.0 ;
N312 G00 Z2.0 ;
N313     X41.0 ;
N314 G01 Z-17.7 ;
N315     X51.0 ;
N316     Z-45.7 ;
N317     X57.0 ;
N318 G00 Z2.0 ;
N319     X41.0 ;
N320 G01 Z-17.7 ;
N321     X52.0 ;
N322 G00 Z10.0 ;
     G00 X0 Z0 T0000 ;
N323     M01 ;
```

```
N701            T0300 ;
N702 G96        S200 M03 ;
N703 G41 G00 X34.0 Z2.0 T0303 ;
N704 G01        X40.0 Z-1.0 F.15 ;
N705            Z-18.0 ;
N706            X46.0 ;
N707            X50.0 Z-20.0 ;
N708            Z-46.0 ;
N709            X54.0 ;
N710            X55.0 Z-46.5 ;
N711            Z-58.0 R5.0 ;
N712            X72.0 ;
N713 G40 G00 Z10.0 ;
     G00        X0 Z0 T0000 ;
N714            M01 ;


N801            T0400 ;
N802 G97        S637 M03 ;
N803 G00        X70.0 Z-13.0 T0404 M23 ;
N804            M28 ;
N805 G92        X49.34 Z-39.12 F1.5 ;
N806            X49.06 ;
N807            X48.85 ;
N808            X48.67 ;
N809            X48.51 ;
N810            X48.37 ;
N811 G00        M29 ;
N812            M24 ;
N813            M05 ;
N814 G27        X0 Z0 T0000 ;
N815            M02 ;
```

6. CANNED CYCLE

By using the canned cycles, the program can be made simple such as specifying a series of paths (infeed → cutting (or threading) → relief → return) in one block instead of using three or four blocks or only the value to be changed needs to be specified for repetition of operations. There are three kinds of Canned Cycles, G90, G92 and G94.

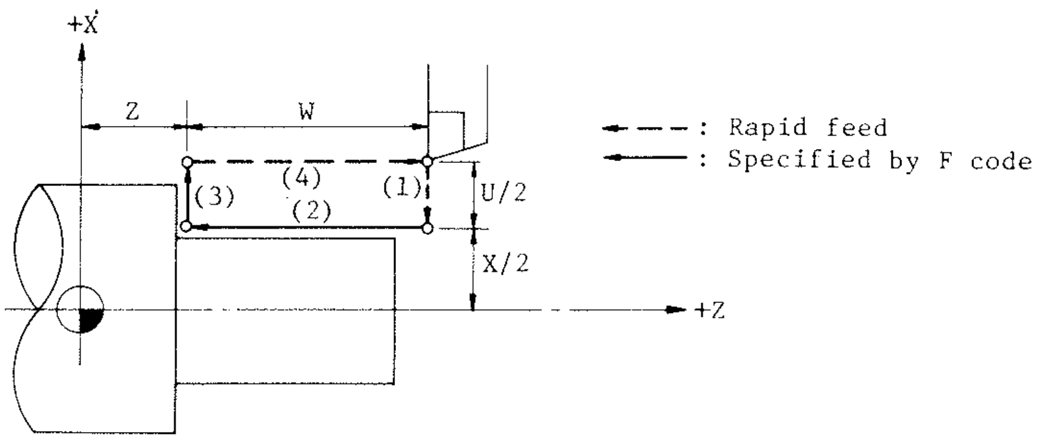
6-1. G90 (Turning Cycle)

[1] Straight turning cycle by using G90

The tool paths (1) to (4) in sketch below are executed by specifying as the following command.
Because G90 is modal, the cycle operation will be continued by specifying the diameter to be cut in the following blocks of G90 command.

```
G90 X(U)_____ Z(W)_____ F_____ ;
    X(U)_____ ;          } Diameter
      ¦                    to be cut
      ¦
```

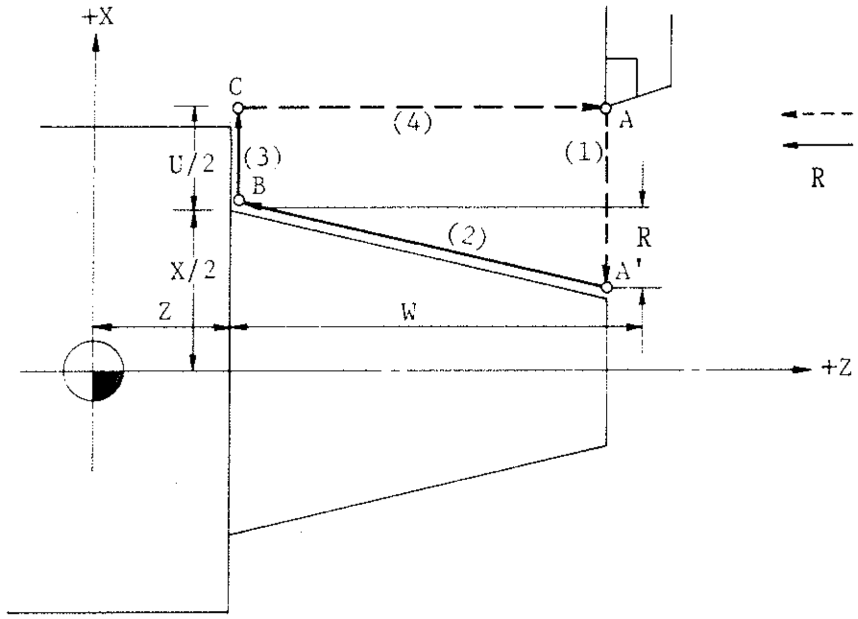
G90 X(U) ----- Z(W) ----- F ---- ;



[2] Taper turning cycle by using G90

The tool paths (1) to (4) in sketch are executed by specifying as the following command and the cycle operation will be continued by specifying the diameter to be cut in the following blocks of G90 command.
The value of the address R specifies the different radius value between position B and A' and the sign (+ or -) is determined by the direction of position A' viewed from position B.

```
G90 X(U)_____ Z(W)_____ R____ F_____ ;
    X(U)_____ ;
        |
        |
```

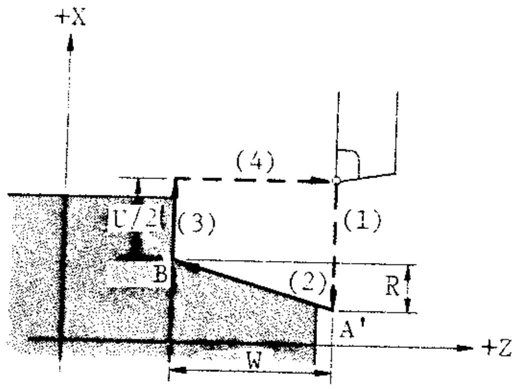


←----: Rapid feed
←——: Specified by F code
R : Radius value

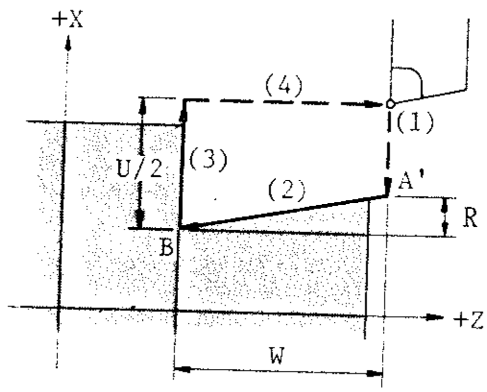
BND-S PRO 0-TC 90/8

(1) When the incremental programming is used, the relation between the sign (+ or -) of numbers following the address U, W or R and the tool path is as follows;
The sign of R is determined by the direction of position A' viewed from position B.

In case of U(-), W(-) and R(-)

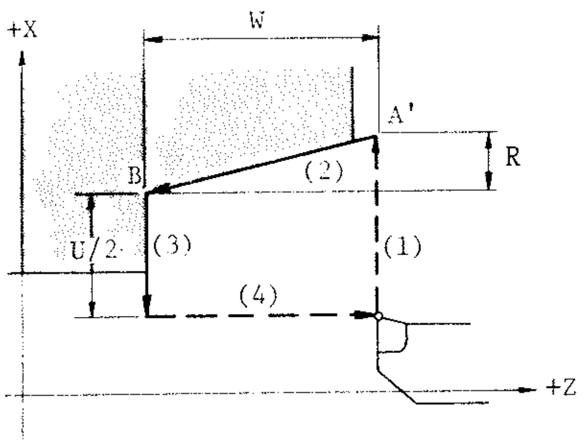


In case of U(-), W(-) and R(+)

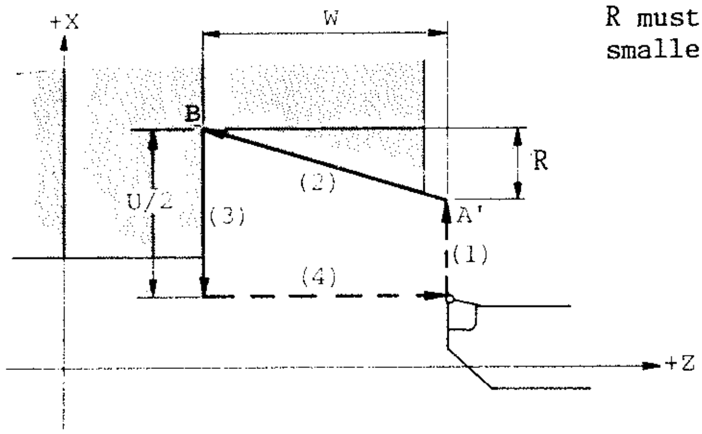


R must be the same or smaller than U/2.

In case of U(+), W(-) and R(+)



In case of U(+), W(-) and R(-)



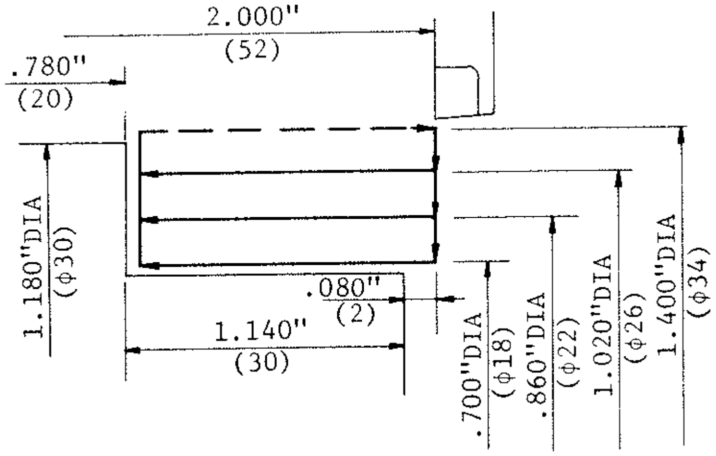
R must be the same or the smaller than U/2.

[3] Notes on programming of G90

(1) Because data values of X(U), Z(W) and R are modal during a canned cycle, these datas are effective until a new data is commanded.
However, these datas are cleared when G00, G01, G02, G03 or G32 is specified.

[4] Example of programming by using G90

(1) Straight turning



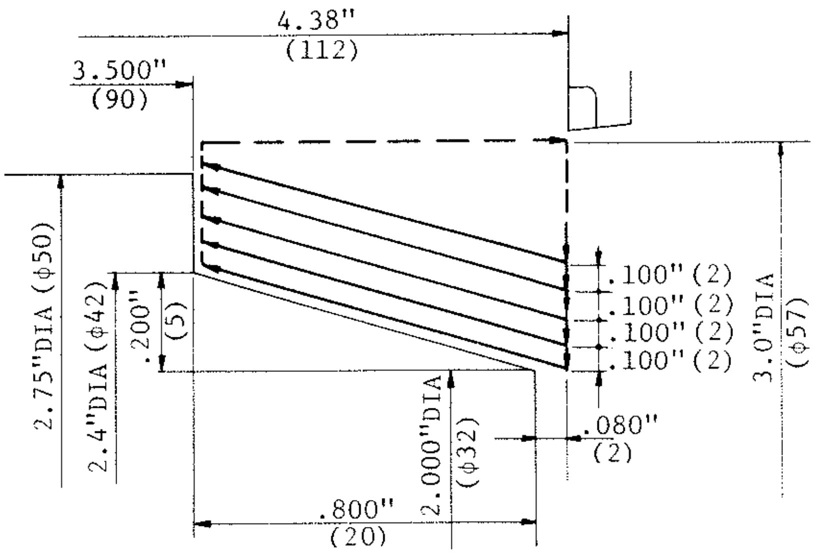
In Inch system

```
G00 X1.4 Z2.0 ;
G90 X1.02 Z.78 F.008 ;
    X.86 ;
    X.7 ;
G00 X___Z___;
```

In Metric system

```
G00 X34.0 Z52.0 ; -----Positioning to starting position
G90 X26.0 Z20.0 F.2 ;--Starting the G90 cycle cutting
    X22.0 ; }  ----------- Diameter is changed.
    X18.0 ; }
G00 X___Z___ ; -------Cancel the G90 cycle
```

(2) Taper turning



In Inch system

```
G00 X3.0 Z4.38 ;
G90 X2.8 Z3.5 R-.22 F.008 ;
    X2.7 ;
    X2.6 ;
    X2.5 ;
    X2.4 ;
G00 X___Z___;
```

In Metric system

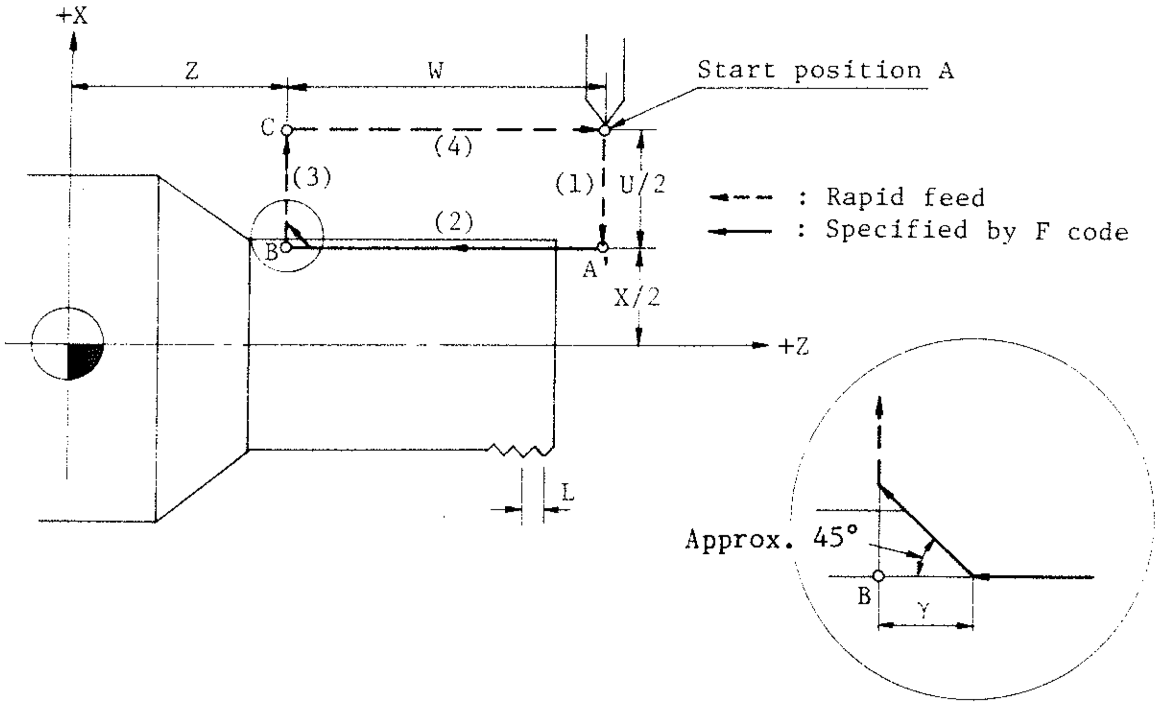
```
G00 X57.0 Z112.0 ;
G90 X52.0 Z90.0 R-5.5 F.35 ;
    X50.0 ;
    X48.0 ;
    X46.0 ;
    X44.0 ;
    X42.0 ;
G00 X___Z___ ;
```

6-2. G92 (Thread Cutting Cycle)

[1] Straight thread cutting cycle by using G92

The tool paths (1) to (4) in sketch shown below are executed by specifying as the following command.
Because G92 is modal, the threading cycle will be continued by specifying the diameter to be cut in the following blocks of G92 command.

```
G92 X(U)____ Z(W)____ F____;
                        └── Specify the thread lead (L)
    X(U)____ ; }
    X(U)____ ; } Specify the diameter to be cut
       :       }
```



Detailed thread pull out

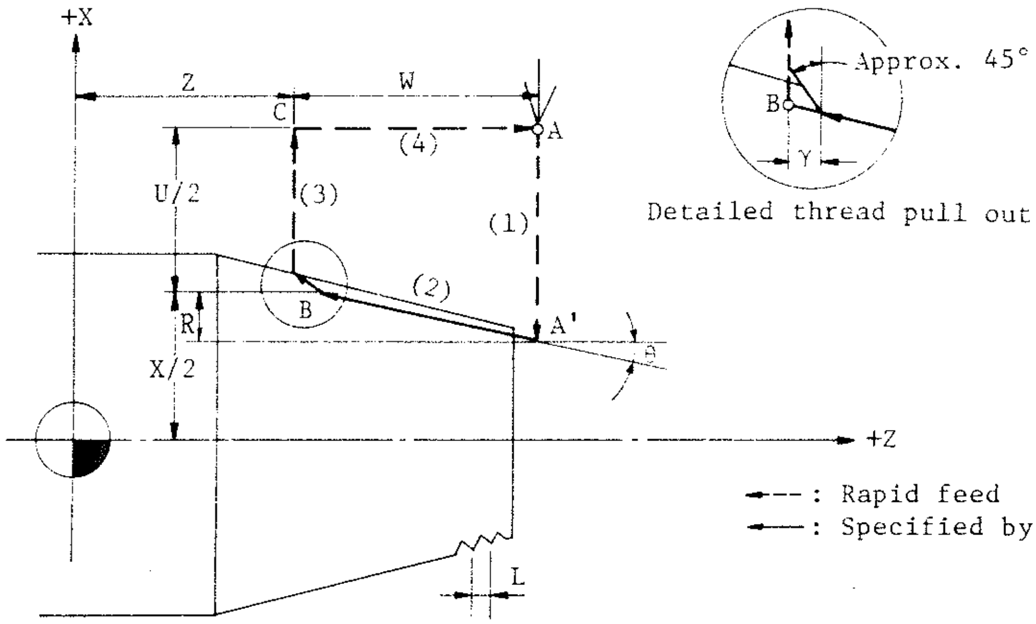
(1) Thread pull out

When M23 (Thread pull out ON) is specified in the previous block of G92, thread pull out will be performed as 'Detailed thread pull out' shown in above sketch.
The pull out width (r) is set by the parameter No. 0109 in the range of 0L to 12.7 L(L is specified thread lead by F).
Normally, it is set to 1.0L.
If M24 (Thread pull out OFF) is specified in the previous block of G92, the thread pull out will not be performed.

[2] Taper thread cutting cycle by using G92

The tool paths (1) to (4) in sketch below are executed by specifying as the following command and the threading cycle will be continued by specifying the diameter to be cut in the following blocks of G92 command.
The value of the address R specifies the different radius value between position B and A' and the sign (+ or -) is determined by the direction of position A' viewed from position B.

```
G92 X(U)_____ Z(W)_____ R_____ F_____;
    X(W)_____;                   └── Specify thread lead (L)
    X(U)_____;
        |      } Specify the diameter to be cut
        |
```



Detailed thread pull out

←--- : Rapid feed
←—— : Specified by F code

(1) Thread pull out

Thread pull out function can be used by specifying M23 (Thread pull out ON) or M24 (Thread pull out OFF) as the same as in straight thread cutting cycle.

[3] Notes on programming of G92

(1) M, S and T functions for G92 cycle must be specified in the preceding blocks.

(2) When the SINGLE BLOCK switch is set to ON position during the thread cutting cycle by G92, the cycle will stop after completion of the tool path (1) to (4).

(3) If the FEED HOLD Switch is depressed during the thread cutting cycle by G92, the cycle will stop after completion of the tool path (4).

(4) The incremental value R is specified by a radius value.

(5) The lead is specified in the Z-axis when the angle is smaller than 45° and in the X-axis by using radius valus when the angle is bigger than 45°.

(6) The FEED OVERRIDE is not effective during the thread cutting cycle.
----It is regarded as 100%.

(7) The same spindle speeds must be used during the thread cutting cycle. If the spindle speed is not constant, the lead become incorrect due to the servo lag. Therefore, the constant cutting speed control must be cancelled by using command of G97 S____ ;.

(8) The allowance of incorrect lead δl should be as big as possible for better chips removal.

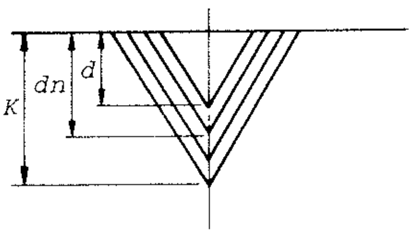
(9) The spindle speed has a limitation during the thread cutting cycle, depending on the pitch of the thread.

Thread lead x Spindle speed ≦ 472 in/min (12000 mm/min)
In(mm) rpm

The following table shows the maximum pitch (lead) of the thread according to the spindle speed.

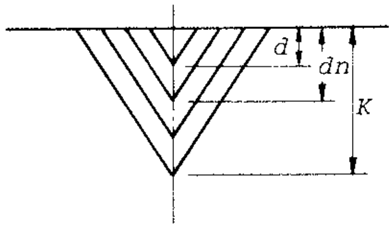
RPM	Maximum Thread Lead		TPI	RPM	Maximum Thread Lead		TPI
	MM	INCH			MM	INCH	
200	50	1.965		6500	1.5	.06	16
500	20	.786	1	7000	1.4	.056	18
1000	10	.393	2	7500	1.3	.05	19
1500	6	.262	4	8000	1.2	.049	21
2000	5	.196	5	8500	1.1	.046	23
2500	4	.157	6	9000	1.1	.043	23
3000	3.3	.131	7	9500	1	.04	25
3500	2.8	.112	9	10000	1	.039	25
4000	2.5	.098	10	10500	0.95	.037	26
4500	2.2	.087	11	11000	0.9	.035	28
5000	2	.078	12	11500	0.86	.034	29
5500	1.8	.07	14	12000	0.83	.032	30
6000	1.6	.065	15				

(10) *There are two methods to determine a depth of thread in each pass of the thread.*



Constant cutting area

$dn = d\sqrt{n}$



Constant cutting depth

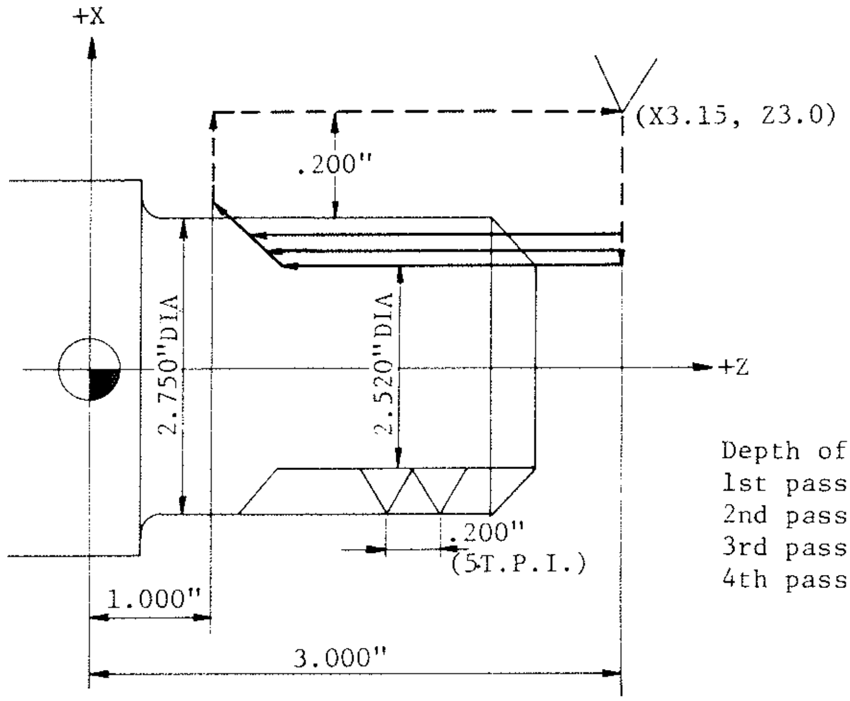
$dn = \dfrac{n \times K}{N}$

d; Cutting depth of the first pass
dn; Cutting depth of No. n pass
N; Total number of passes
K; Total depth of thread

[4] Example of programming by using G92

(1) Straight thread cutting

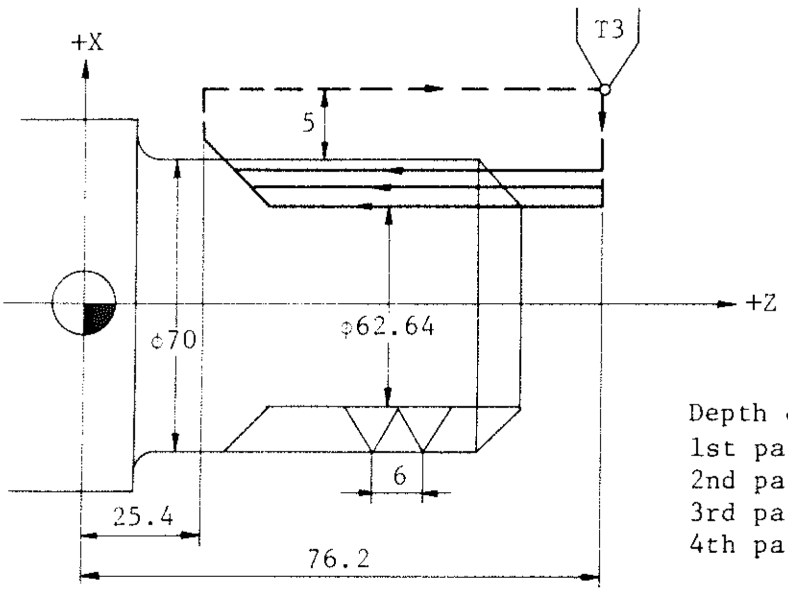
<u>In Inch system</u>



Depth of thread
1st pass .060"
2nd pass .020"
3rd pass .020"
4th pass .015"

```
G00 X3.15 Z3.0 M23 T303 ; ........ Thread pull out ON
G92 X2.63 Z1.0 F0.2 ; )
    X2.59 ;            |
    X2.55 ;            }   ........ Thread cutting cycle
    x2.52 ;            )               (4 passes)
G00 X___ Z___ M24 ; ............ Thread pull out OFF
```

In Metric system

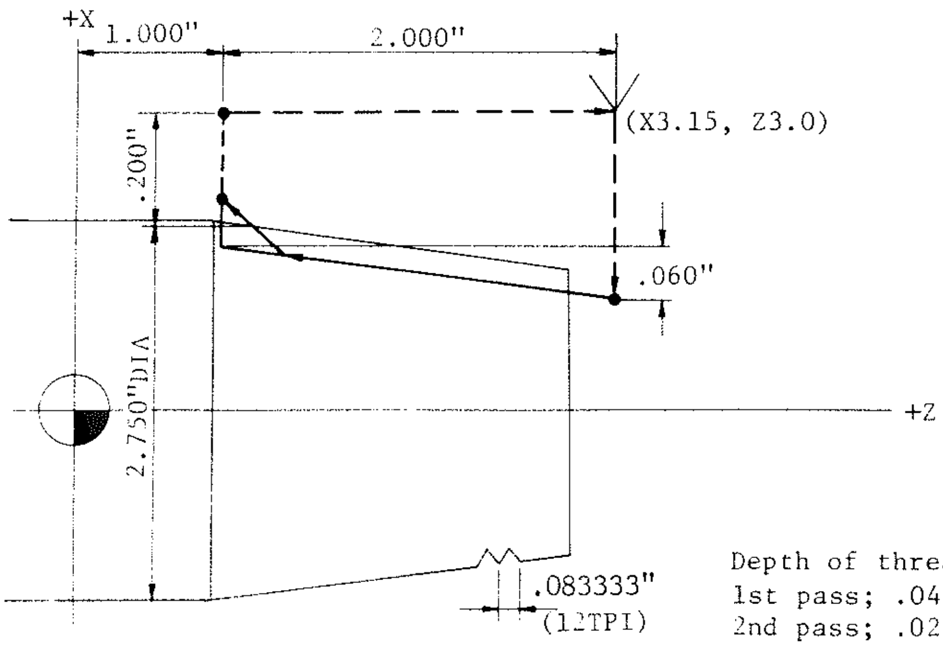


Depth of thread
1st pass; 1.8mm
2nd pass; 0.7mm
3rd pass; 0.6mm
4th pass; 0.58mm

```
G00 X80.0 Z76.2 M23 T303 ; ....... Thread pull out ON
G92 X66.4 Z25.4 F6.0 ;
    X65.0 ;          ]
    X63.8 ;          }  ....... Thread cutting cycle
    X62.64 ;         ]          (4 passes)
G00 X____ Z____M24 ; ............ Thread pull out OFF
```

(2) Taper thread cutting

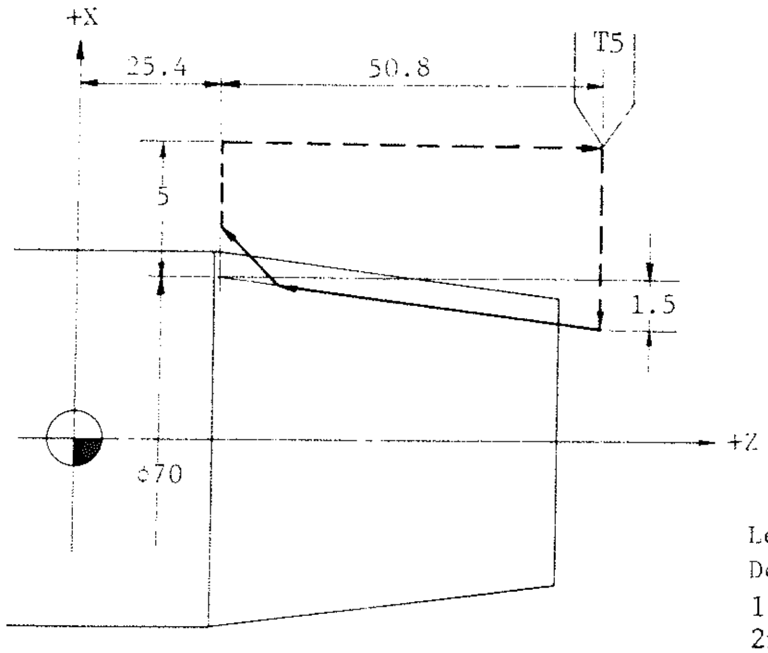
In Inch system



Depth of thread
1st pass; .040"
2nd pass; .020"
3rd pass; .015"

```
G00 X3.15 Z3.0 M23 T0505 ;
G92 X2.67 Z1.0 R-.06 F83333 ;
    X2.63 ;
    X2.6 ;
G00 X____Z____M24 ;
```

In Metric system



Lead ; 2.0
Depth of thread
1st pass; 0.7mm
2nd pass; 0.6mm
3rd pass; 0.5mm

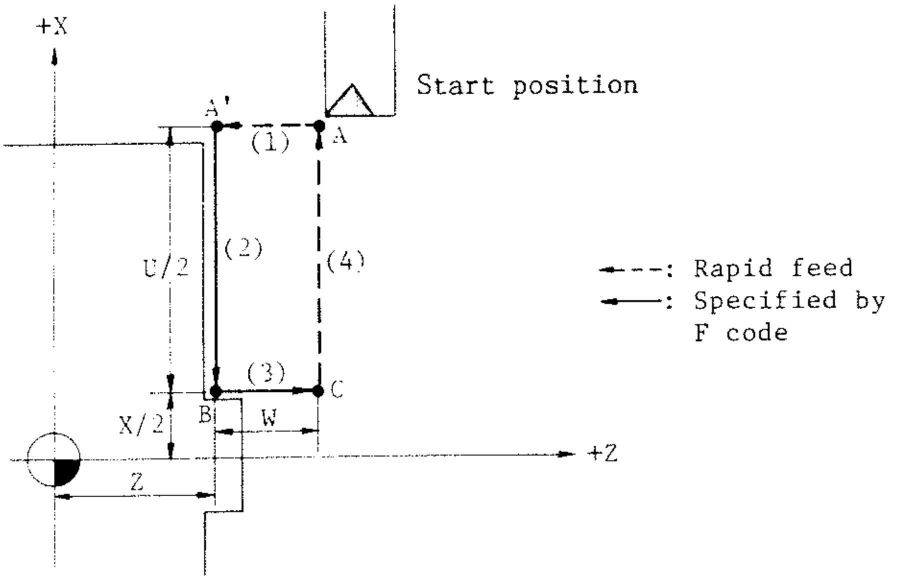
```
G00 X80.0 Z76.2 M23 T0505 ;
G92 X70.0 Z25.4 R-1.5 F2.0 ;
    X68.8 ;
    X67.8 ;
G00 X____ Z____ M24 ;
```

6-3. G94 (Facing Cycle)

[1] Straight facing cycle by using G94

The tool paths (1) to (4) in sketch below are executed by specifying as the following command.
Because G94 is modal, the cycle operation will be continued by specifying the depth to be cut in the following blocks of G94 command.

```
G94 X(U)____ Z(W)____ F____;
             Z(W)____ ;  } Depth to be cut
             Z(W)____ ;  } in Z-axis
                 |
                 |
```



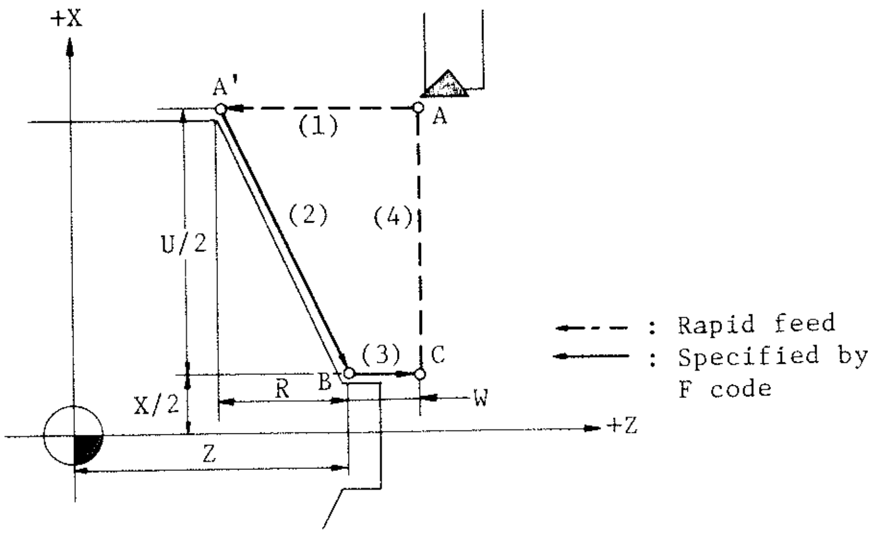
When the incremental programming is used, the sign (+ or -) of numbers following the address U or W is determined by the direction of the tool paths (1) and (2) viewed from the start position.

BND-S PRO 0-TC 90/8

[2] Taper facing cycle by using G94

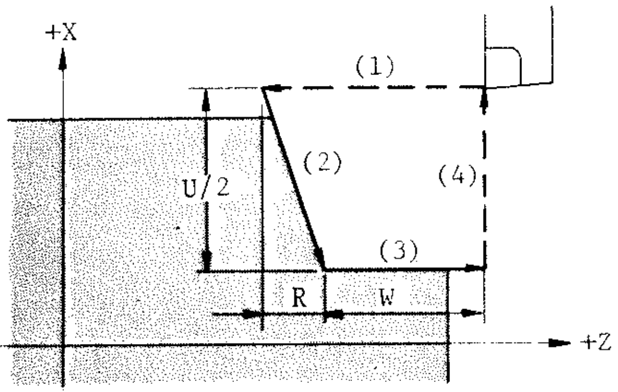
The tool paths (1) to (4) in sketch are executed by specifying as the following command and the cycle operation will be continued by specifying the depth to be cut in the following blocks of G94 command.

```
G94 X(U)____ Z(W) ____ R ____ F ____ ;
             Z(W) ____ ;  └── Value of taper
             Z(W) ____ ;
               |          Depth to be cut
               |
```

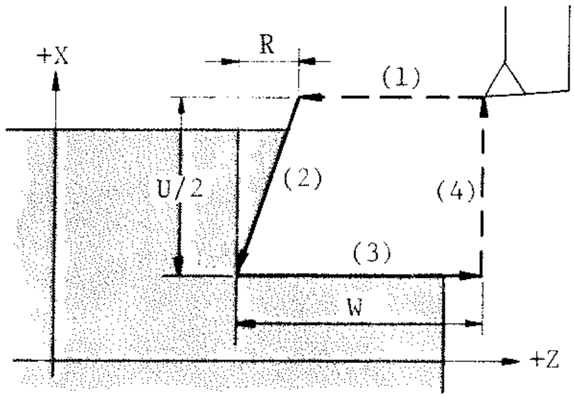


(1) When the incremental programming is used, the relation between the sign (+ or -) of numbers following the address U, W or R and the tool path is as follows;
The sign of R is determined by the direction from the command position B and the sign of U and W is determined by the direction from the start position.

In case of U(-), W(-) and R(-),

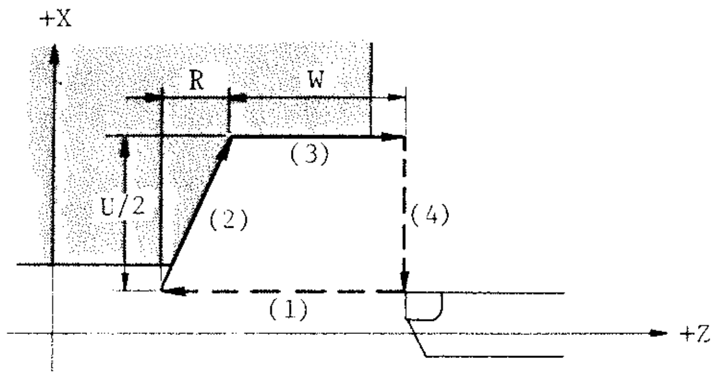


In case of U(-), W(-) and R(+),

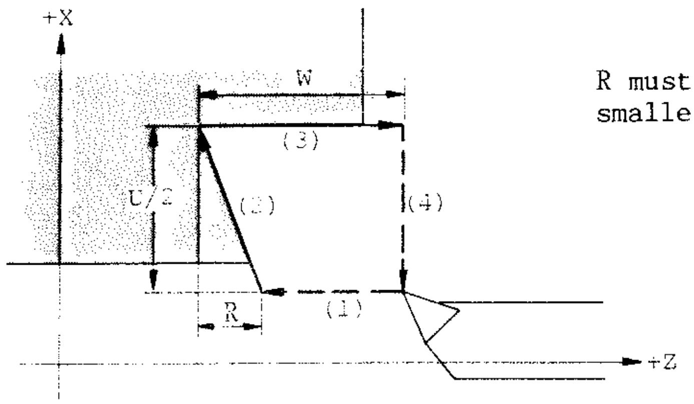


R must be the same or smaller than W.

In case of U(+), W(-) and R(-),



In case of U(+), W(-) and R(+),



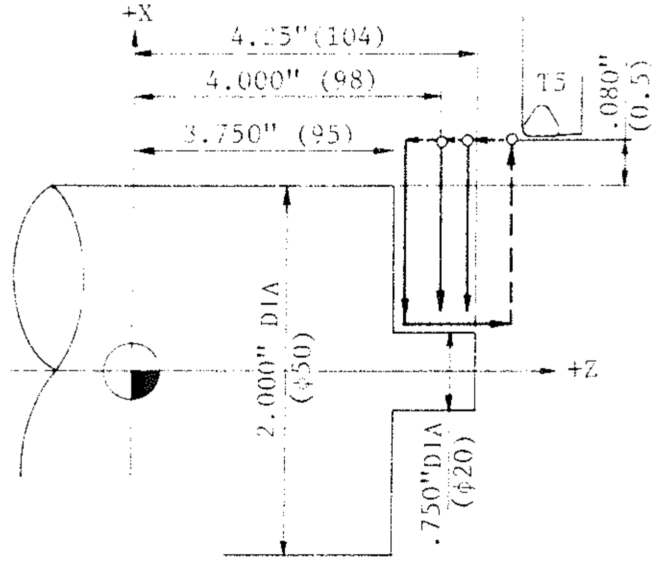
R must be the same or smaller than W.

[3] Notes on programming of G94

(1) Because data values of X(U), Z(W) and R are modal during a canned cycle, these datas are effective until a new data is commanded.
However, these datas are cleared when G00, G01, G02, G03, or G32 is specified.

[4] Example of programming by using G94

(1) Straight facing



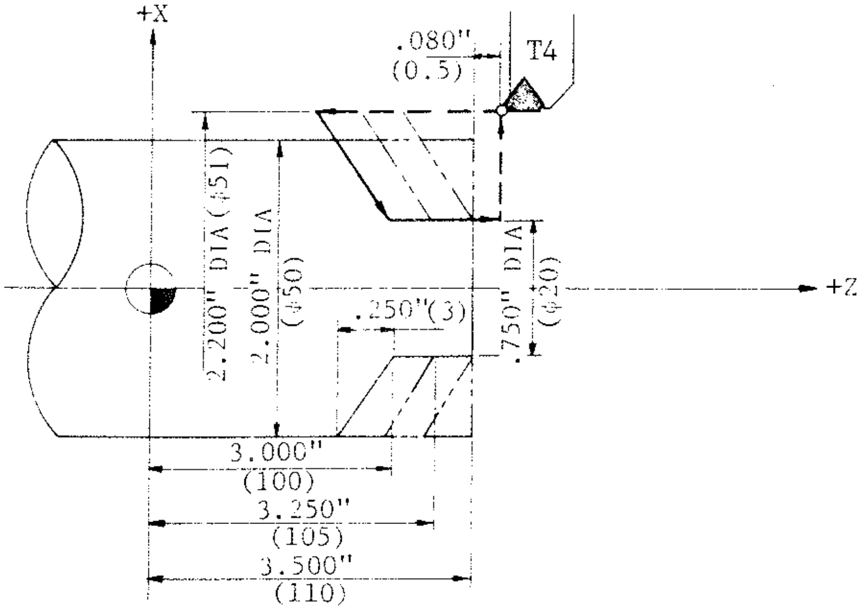
In Inch system

```
G00 X2.16 Z4.33 T505 ;
G94 X.75 Z4.15 F.012 ;
    Z4.0 ;
    Z3.75 ;
G00 X___ Z___ ;
```

In Metric system

```
G00 X51.0 Z104.5 T505 ;
G94 X20.0 Z101.0 F.35 ;
    Z98.0 ;
    Z95.0 ;
G00 X___ Z___ ;
```

(2) Taper facing



In Inch system

```
G00 X2.2 Z3.58 T404 ;
G94 X.75 Z3.5 R-.29 ;
    Z3.25 ;
    Z3.0 ;
G00 X___ Z___ ;
```

In Metric system

```
G00 X51.0 Z110.5 T404 ;
G94 X20.0 Z110.0 R-3.1 ;
    Z105.0 ;
    Z100.0 ;
G00 X___ Z___ ;
```

7. MULTIPLE REPETITIVE CUTTING CYCLES

By using canned cycles G90, G92 or G94, a series of continuous commands can be performed to eliminate the need of three or four blocks on the standard G codes. But, when the program of a complicate shaped part is made, the canned cycle may not be simple enough.
The multiple repetitive cutting cycle will help for making simpler program, such as rough and finish cutting by specifying the shape of a part.
There are seven kinds of multiple repetitive cutting cycle G70, G71, G72, G73, G74, G75, and G76.
In case of G71, G72 or G73, a series of rough cutting cycle along the finished shape of a part is executed by specifying the sequence numbers of the program for the finished shape following a block of these G codes.
And, the program for the finished shape is executed by specifying G70 and the sequence number of the program for the finished shape, in any time after rough cutting operation.

In case of G74, G75 or G76, a series of cutting operation is executed by the information in the block of these G codes.

<table>
<tr><th>G CODE</th><th>FUNCTION</th><th colspan="2">REMARKS</th></tr>
<tr><td>G70</td><td>Finish Cutting Cycle</td><td></td><td rowspan="4">Possible to use the compen-sation</td></tr>
<tr><td>G71</td><td>Stock Removal Cutting Cycle in Turning</td><td rowspan="3">Finish Cutting is available by G70</td></tr>
<tr><td>G72</td><td>Stock Removal Cutting Cycle in Facing</td></tr>
<tr><td>G73</td><td>Pattern Repeated Cutting Cycle</td></tr>
<tr><td>G74</td><td>Longitudinal Pecking Cycle</td><td colspan="2" rowspan="3">Impossible to use the compensation</td></tr>
<tr><td>G75</td><td>Cross Pecking Cycle</td></tr>
<tr><td>G76</td><td>Thread Cutting Cycle</td></tr>
</table>

7-1. G71 (Stock Removal Cutting Cycle in Turning)

When the finish cutting tool paths are specified following the block contained G71, rough turning with specified cutting depth will be executed.
At first, longitudinal turning with cutting depth U is repeated until no more cutting depth U is left, and then finally cut along the finished shape with finishing allowance U and W left.
Finish cutting can be done by using G70.

[1] Method of program

This is a form to program the rough turning cycle by using G71.

```
N___G71 U Δd R e ;
```

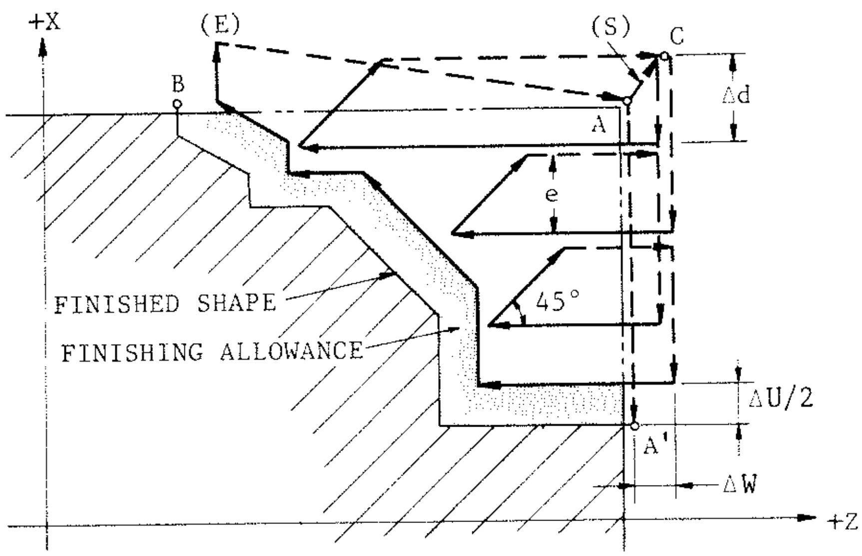
- Δd — Depth of cut (Specified by radius value)
- e — Relieving amount

```
N___G71 P ns Q nf U ±___W ±___F___S___ ;
N ns -------------- ;
N __ -----------F__ ;
N __ -----------S__ ;
N __ -----------T__ ;
N __ -------------- ;
N nf -------------- ;
```

- ns — Sequence number of starting finish cutting cycle
- nf — Sequence number of the end of finish cutting cycle
- U — Depth of finishing allowance in X-axis (Specified by diameter)
- W — Depth of finishing allowance in Z-axis

F,S : Specify the feed rate and the S-function during rough turning cycle.
If these are not specified, previous F and S function will be effective.

The finish cutting tool paths A→A'→B in sketch below are specified in the block between the sequence number Nns and Nnf.
T, F, S and M functions in there are effective only in finish cutting cycle by G70.



Depth of cut(Δd): Designate without sign. The cutting direction depends on the direction AA'.
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No.0717, and the parameter is changed by the program command.

Relieving amount (e) : This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter N.0718, and the parameter is changed by the program command.

[2] Notes on the programming of G71

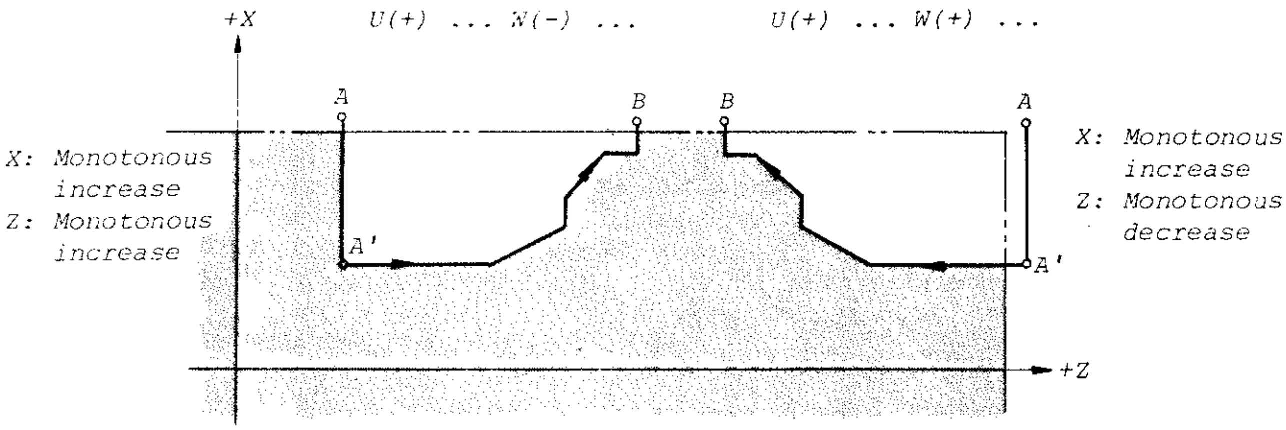
(1) G71 cycle start from the position A and finishes at the position A. Therefore, the tool must be approached to the position A before starting G71.

(2) T, F and S codes specified in between the sequence number Nns and Nnf are ignored during the G71 cycle. They are effective only in G70 (Finish Cutting Cycle).

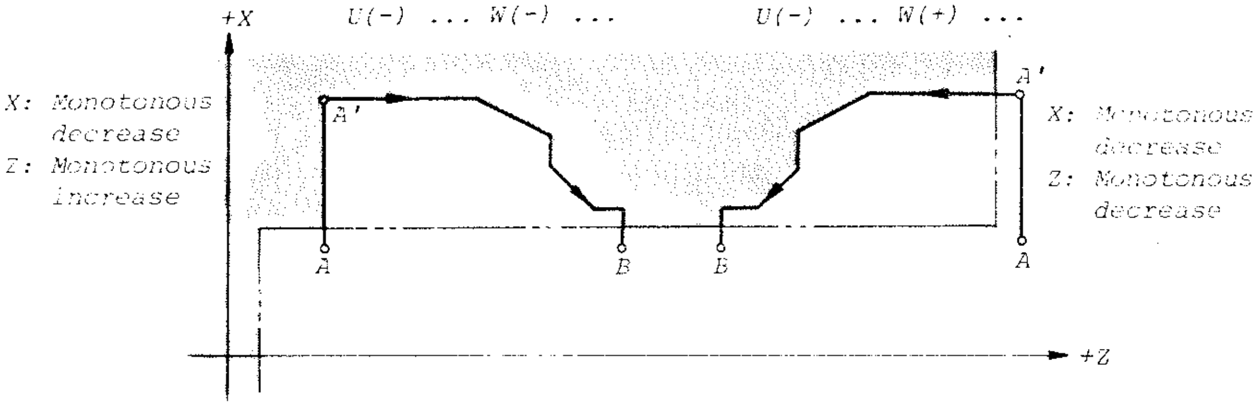
(3) The address U and W must be programmed with signs. If a wrong sign is programmed, the workpiece may be cut too much.

(4) The following sketch shows four patterns to use G71. All of these cutting cycles are made for rough longitudinal cutting in parallel to Z-axis.
The signs (+ or -) of U and W are shown in sketch.

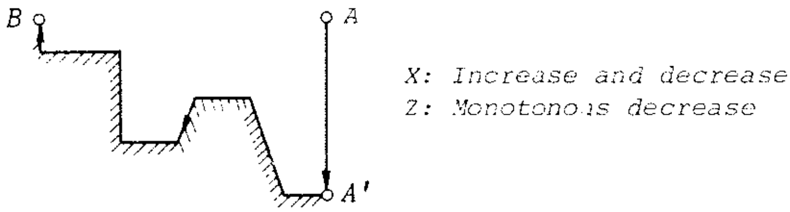
(a) Turning



(b) Boring



(5) The tool paths between A' and B should be programmed to be monotonous increase or decrease in X and Z coordinates.
The following tool path in sketch is not monotonous increase or decrease. Therefore, it can not be programmed.

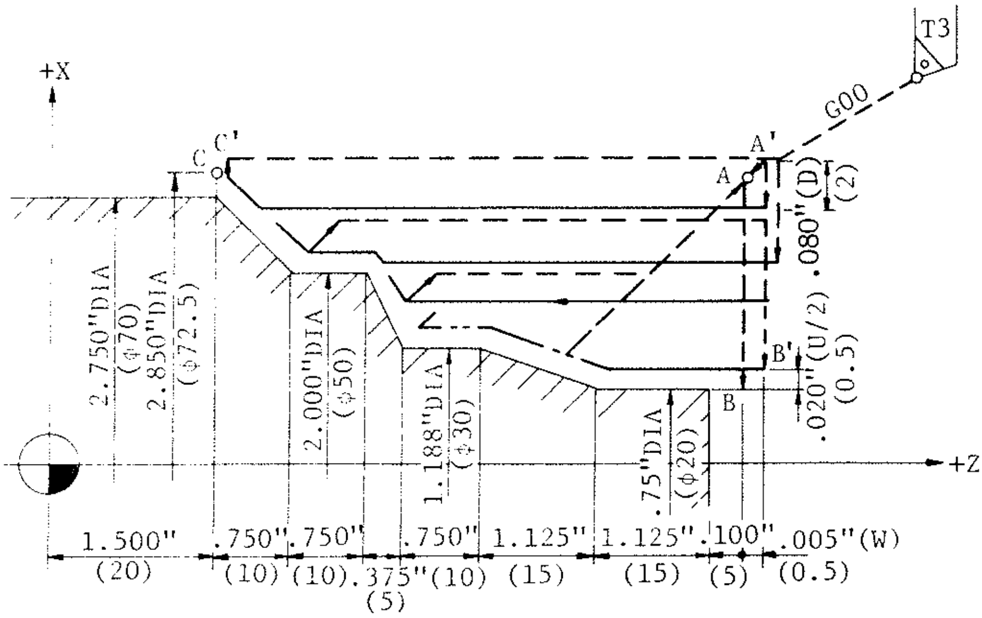


(6) The subprogram cannot be called from the block between sequence number "ns" and "nf".

(7) In MDI mode, G71 cannot be commanded. If it is commanded, P/S alarm No.67 is generated.

(8) Do not program so that the final movement command of the finishing shape block group designated with P and Q for G71 finish with corner rounding.
If it is specified, P/S alarm (No.69) is generated.

[3] Example of programming by using G71



In Inch system

```
N101      T0300 ;
N102 G96 S500 M03 ;
N103 G00 X2.85 Z6.475 T0303 M28 ;
N104 G71 U.08 R.04 ;
N105 G71 P106 Q113 U.04 W.005 F.006 ;
N106 G00 X.75 S600 ;
N107 G01 W-1.225 F.005 ;
N108      X1.188 W-1.125 M52 ;   Finished shape
N109      W-.75 ;                 (8 Blocks)
N110      X2.0 W-.375 ;
N111      W-.75 ;
N112      X2.75 W-.75 ;
N113      X2.85 M51 F.012 ;
N114 G00 X3.5 Z7.0 M29 ;
     G00 X0 Z0 T0000 ;
N115      M01 ;
```

In Metric system

```
N101      T0300 ;
N102 G96 S120 M03 ;
N103 G00 X72.5 Z85.5 T0303 M28 ;
N104 G71 U2.0 R1.0 ;
N105 G71 P106 Q113 U1.0 W.5 F.15 ;
N106 G00 X20.0 S150 ;
N107 G01 W-20.0 F.1 ;
N108      X30.0 W-15.0 M52 ;   Finished shape
N109      W-10.0 ;              (8 Blocks)
N110      X50.0 W-5.0 ;
N111      W-10.0 ;
N112      X70.0 W-10.0 ;
N113      X72.5 M51 ;
N114 G00 X90.0 Z100.0 M29 ;
     G00 X0 Z0 T0000 ;
N115      M01 ;
```

7-2. G72 (Stock Removal Cutting Cycle in Facing)

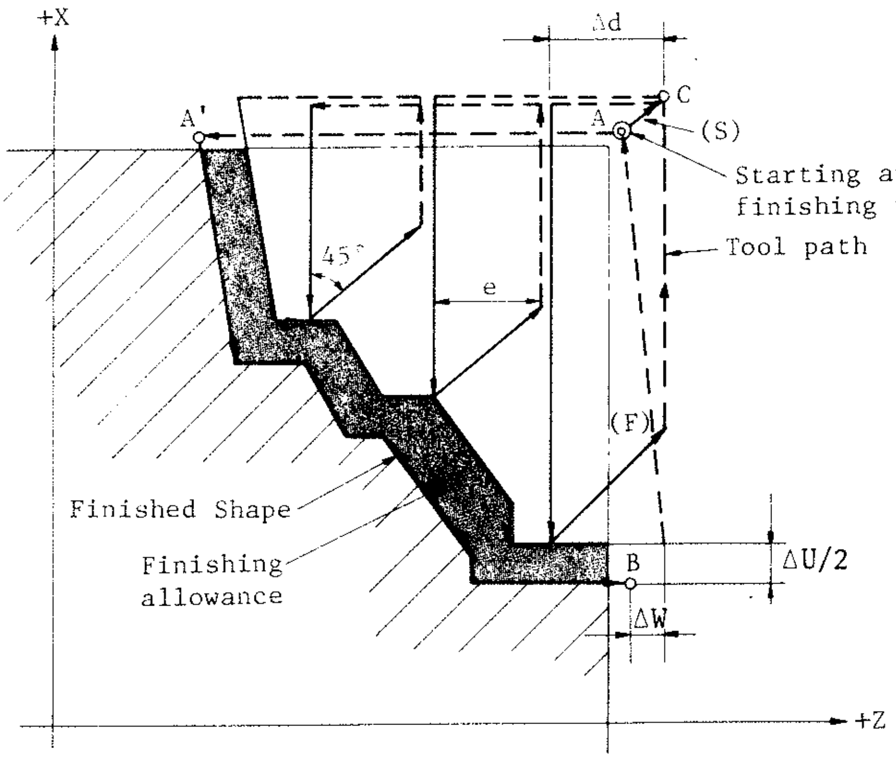
When the finish cutting tool paths are specified following the block contained G72, rough facing with specified cutting depth will be executed.
At first, rough facing in parallel to X-axis with cutting depth W is repeated until no more cutting depth W is left, and then finally cut along the finished shape with finishing allowance U and W left.
Finish cutting can be done by using G70.
The restriction on the programming and notes of G72 are the same as G71.

[1] Method of program

This is a form to program the rough facing cycle by using G72.

```
              Depth of cut in Z-axis
                  Relieving amount
N____G72 WΔd R e ;
```

```
                Sequence number of starting finish cutting cycle
                   Sequence number of the end of finish cutting cycle
                        Depth of finishing allowance in X-axis
                        (Specified by diameter)
N___ G72 P ns Q nf U ±___ W±___ F___S ___ ;
N ns ----------- ;
N___ ----------- ;            Depth of finishing allowance in Z-axis
N___ --- F___--- ;
N___ --- S___--- ;       F,S : Specify the feed rate and the S-function
N___ --- M___--- ;             during rough facing cycle.
N___ --- T___--- ;             If these are not specified, previous
N___ ----------- ;             F and S function will be effective.
N nf ----------- ;
                     The finish cutting tool paths A→A'→B in sketch
                     are specified in the block between the sequence
                     number Nns and Nnf.
                     T, F, S and M functions in there are effective
                     only in finish cutting cycle by G70.
```



Depth of cut(Δd): Designate without sign. The cutting direction depends on the direction AA'.
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 0717, and the parameter is changed by the program command.

Relieving amount(e): This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 0718, and the parameter is changed by the program command.

[2] Notes on the programming of G72

(1) G72 cycle starts from the position A and finishes at the position A. Therefore, the tool must be approached to the position A before starting G72.

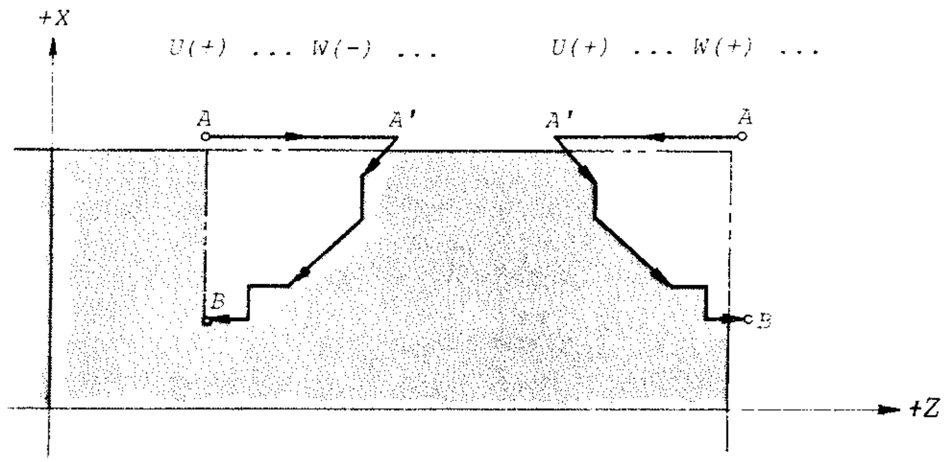
(2) T, F and S codes specified in between the sequence number Nns and Nnf are ignored during the G72 cycle. They are effective only in G70 (Finish Cutting Cycle).

(3) The address U and W must be programmed with sign. If a wrong sign is specified, the workpiece may be cut too much.

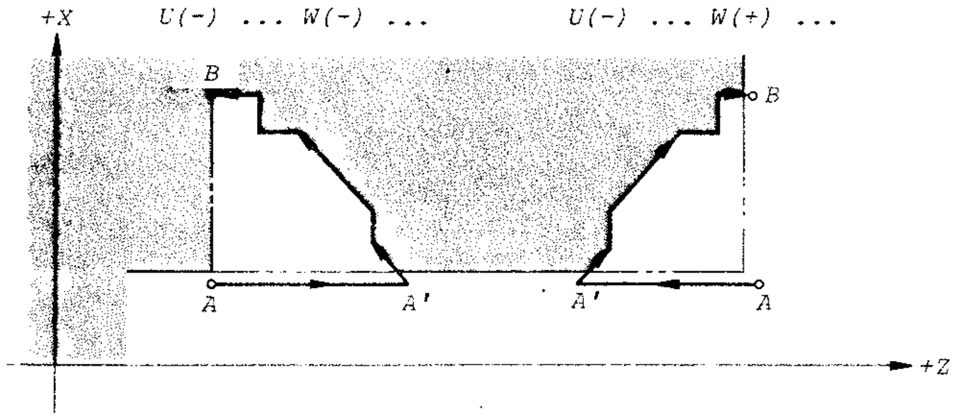
(4) The subprogram cannot be called from the block between sequence number "ns" and "nf".

(5) The following sketch shows four patterns to use G72. All of these cutting cycle are made for rough facing in parallel to X-axis. The signs (+ or -) of U and W are shown in sketch.

(a) O.D. Cutting

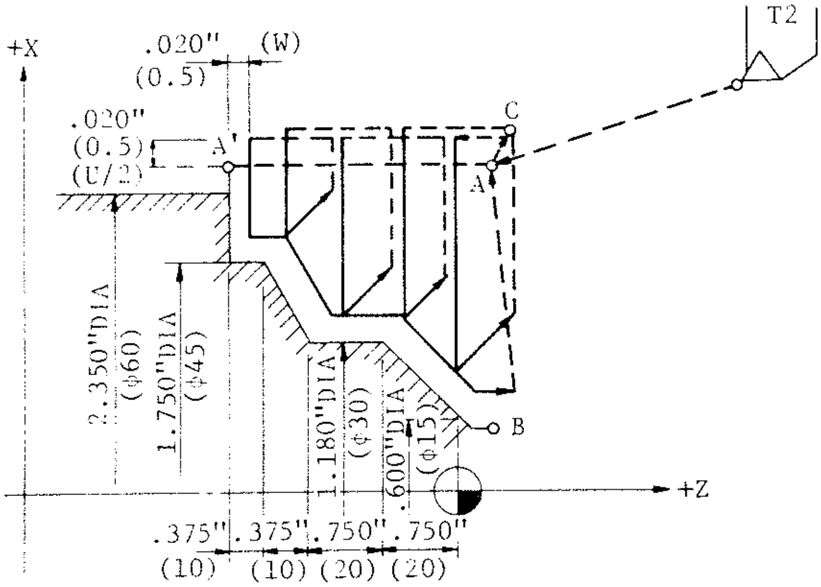


(b) I.D. Cutting



(6) In MDI mode, G72 cannot be commanded. If it is commanded, P/S alarm No.67 is generated.

[3] Example of programming by using G72



In Inch system

```
N200      T0200 ;
N201 G96 S500 M03 ;
N202 G00 X2.5 Z.2 T0202 M28 ;
N203 G72 W.08 R.04 ;
N204 G72 P205 Q211 U.04 W.02 F.006 ;
N205 G00 Z-2.25 ;
N206 G01 X1.75 F.003 ;
N207      Z-1.875 ;
N208      X1.18 Z-1.5 ;
N209      Z-.75 ;
N210      X.5226 Z.1 ;
N211      Z.2 F.012 ;
N212 G00 X3.5 Z2.0 M29 ;
     G00 X0 Z0 T0000 ;
N213      M01 ;
```

Finished shape (7 Blocks) (N205–N211)

.
.
.
.

```
N400          T0400 ;
N401 G96      S560 M03 ;
N402 G41 G00  X2.5 Z.2 T0404 M28 ;
N403 G70      P205 Q211 ;
N404 G40 G00  X3.5 Z2.0 M29 ;
     G00      X0 Z0 T0000 ;
N405          M01 ;
```

Finish cutting cycle by using G70 and T4

In Metric system

```
N200           T0200 ;
N201 G96       S120 M03 ;
N202 G00       X62.0 Z2.7 T0202 M28 ;
N203 G72       W2.0 R1.0 ;
N204 G72       P205 Q211 U1.0 W.5 F.15 ;
N205 G00       Z-60.0 ;
N206 G01       X45.0 F.1 ;
N207           Z-50.0 ;
N208           X30.0 Z-40.0 ;          Finished shape
N209           Z-20.0 ;                (7 Blocks)
N210           X13.725 Z1.7 ;
N211           Z2.7 F2.0 ;
N212 G00       X65.0 Z50.0 M29 ;
     G00       X0 Z0 T0000 ;
N213           M01 ;

                .
                .
                .
                .
                .
N400           T0400 ;
N401 G96       S150 M03 ;
N402 G41 G00 X62.0 Z2.7 T0404 ;    Finish cutting cycle by using
N403 G70       P205 Q211 ;         G70 and T4
N404 G40 G00 X65.0 Z50.0 M29 ;
     G00       X0 Z0 T0000 ;
N405           M01 ;
```

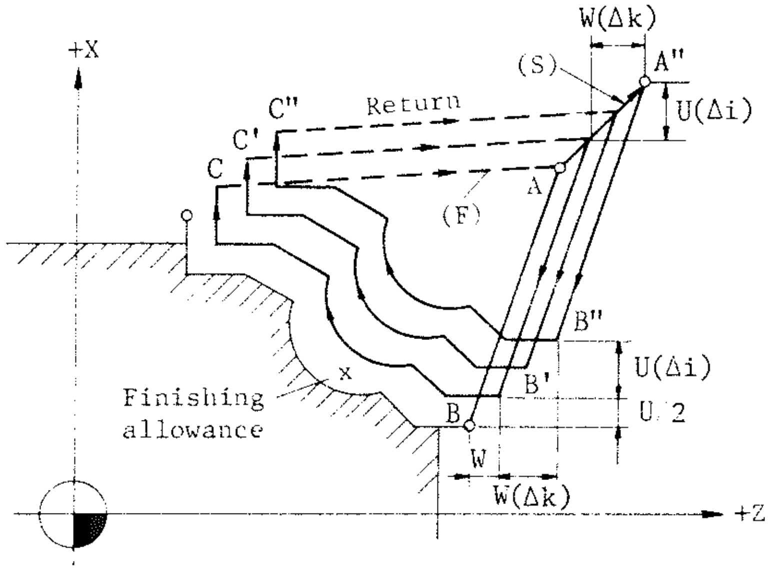
7-3 G73 (Pattern Repeated Cutting Cycle)

When the finish cutting tool paths are specified following the block contained G73, rough cutting will be executed with finishing allowance U/2 and W left. This rough cutting is repeatedly made along the shape of the finish cut. And total of rough cutting depth which is specified by using the address U(Δi) and W(Δk) is divided by number of passes which are specified by using the address R. Therefore, this cycle is useful for the workpiece whose rough shape is already made such as a forging or casting.
The finish cutting can be done by using G70, the same as G71 or G72.

[1] Method of program

This is a form to program the pattern repeated cutting cycle by using G73.

```
               ┌── Total depth of rough cut in X-axis
               │   (Specified by radius value)
               │
               │      ┌──Total depth of rough cut in Z-axis
               │      │
               │      │  ┌──Number of passes for rough cutting
               ↓      ↓  ↓
N___G73 U ± Δi W ± Δk R __ ;

             ┌── Sequence number of starting finish cutting cycle
             │
             │   ┌── Sequence number of the end of finish cutting cycle
             ↓   ↓
N___G73 Pns Qnf U ±__ W ±__ F__S__ ;
Nns ---------- ;      ↑      └──Depth of finishing allowance in Z-axis
N__ ---------- ;      │
N__ ---F__---- ;      │
N__ ---S__---- ;      └─ Depth of finishing allowance in X-axis
N__ ---T__---- ;          (Specified by diameter)
N__ ---------- ;
N__ ---------- ;  ── The finish cutting tool paths A B C in
Nnf ---------- ;     sketch below are specified in the block
                     between the sequence number Nns and Nnf.
                     T, F, S and M function in there are
                     effective only in finish cutting cycle
                     by G70.
```



[2] Notes on the programming of G73

(1) G73 cycle starts from the position A and finishes at the position A. Therefore, the tool must be approached to the position A before starting G73.

(2) T, M, F and S codes specified in between the sequence number Nns and Nnf are ignored during the G73 cycle. They are effective only in G70 (Finish Cutting Cycle).

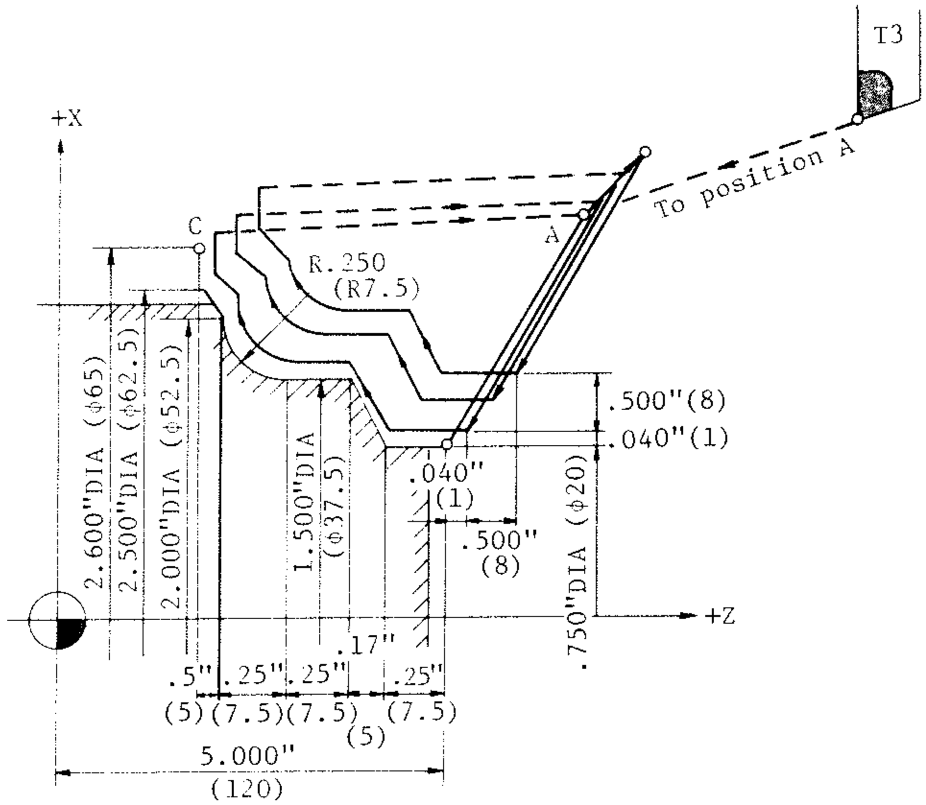
(3) The addresses U(Δi),W(Δk), U and W must be programmed with the sign.

(4) The returned movement C'→A is executed by rapid feed rate (G00). However, the movement of infeeding motion A"→B" and A'→B' will be executed according to the program (G00 or G01).

(5) The program of finish cutting tool paths in between the sequence number Nns and Nnf must be specified at the following block of G73.

(6) In MDI mode, G73 cannot be commanded. If it is commanded, P/S alarm No.67 is generated.

[3] Example of programming by using G73



In Inch system

```
N300          T0300 ;
N301 G96      S550 M03 ;
N302 G42 G00  X2.9 Z5.8 T0303 M28 ;
N303 G73      U.5 W.5 D3 ;
N304 G73      P305 Q310 U.08 W.04 F.006 ;
N305 G00      X.75 Z5.0 ;
N306 G01      W-.25 F.004 ;
N307          X1.5 Z-.17 ;              Finished shape
N308          W-.5 R.25 ;
N309          X5.0 C-.5 ;
N310 G00      X2.6 ;
N311 G40 G00  X4.0 Z6.0 M29 ;
     G00      X0 Z0 T0000 ;
N312          M01 ;
```

In Metric system

```
N300          T0300 ;
N301 G96      S180 M03 ;
N302 G42 G00 X75.0 Z130.0 T0303 M28 ;
N303 G73      U8.0 W8.0 D4 ;
N304 G73      P305 Q310 U2.0 W1.0 F.15 ;
N305 G00      X20.0 Z120.0 ; ┐
N306 G01      Z112.5 F.1 ;   │
N307          X37.5 Z107.5 ; │
N308          Z92.5 R7.5 ;   │   Finished shape
N309          X62.5 C5.0 ;   │
N310 G00      X65.0 ;        ┘
N311 G40 G00 X80.0 Z180.0 M29 ;
     G00      X0 Z0 T0000 ;
N312          M01 ;
```

7-4 G70 (Finish Cutting Cycle)

After rough cutting by G71, G72, or G73, following block containing G70 specifies the finish cutting cycle which is programmed the finished shape following blocks of G71, G72, or G73 previously.

[1] Method of program

This is a form to program the finish cutting cycle by using G70.

N ____ G70 Pns Qnf ;

- Qnf: Sequence number of the end of the finished shape program
- Pns: Sequence number of starting the finished shape program

[2] Notes on the programming of G70

(1) F and S functions specified in the finished shape program are effective in the finish cutting cycle (G70). However, F and S functions for rough cutting cycle specified in the block containing G71, G72, or G73 are ignored in the finish cutting cycle.

(2) In blocks between "ns" and "nf" referred in G70 through G73, the subprogram cannot be called.

[3] Example programming by using G70

Refer to example programmings of G71 and G72.

BND-S PRO 0-TC 90/8

7-5 G74 (Longitudinal Pecking Cycle)

The G74 cycle is the longitudinal cutting cycle with pecking motion and it repeats with specified cutting depth in each infeed until the tool reaches the specified diameter. Therefore, chip control is possible by using this cycle and the cycle also be used for the drill pecking operation.

[1] Method of program

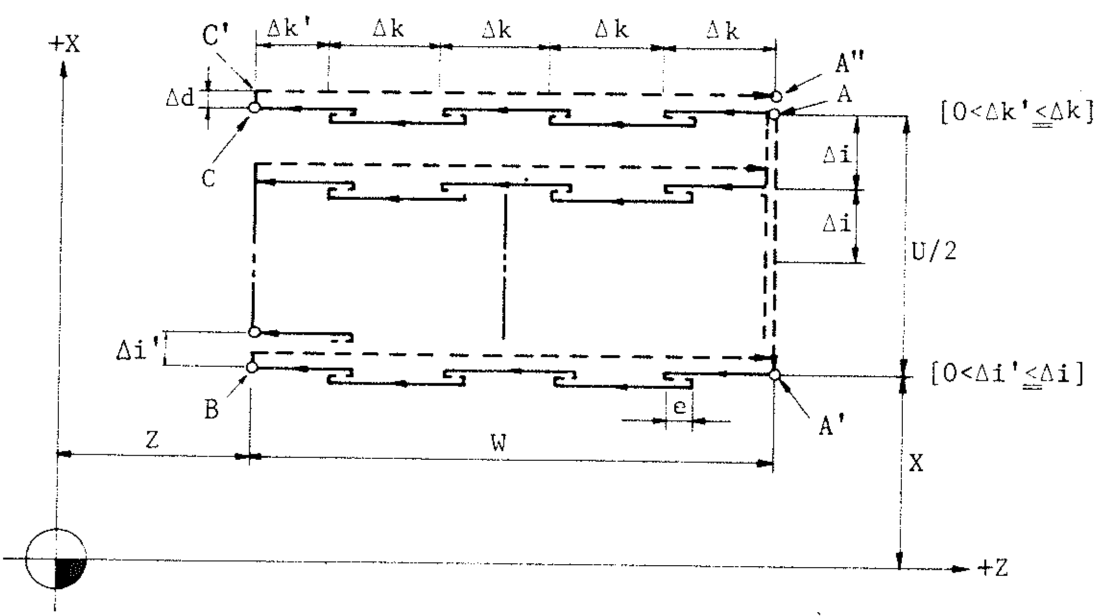
N___G74 R e ;

- e: Relieving value

N___G74 X(U) ±___Z(W) ±__ PΔi QΔk R±Δd F__ ;

- X(U): Absolute position (B) in X-axis or the distance between position A and B
- Z(W): Absolute position (B) in Z-axis or the distance between position A and B
- PΔi: Movement amount in X-axis for a pass (Specified without the sign and by diameter)
- QΔk: Movement amount in Z-axis for a pass (Specified without the sign)
- R±Δd: The amount of relief
- F: Feed rate

←---- : Rapid feed
←—— : Specified by F code



Relieving value (e) is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No.0722, and the parameter is changed by the program command.

By specifing the G74 cycle, a tool moves from the position A to C with repeating pecking motion and returns from the position C to A at rapid feed rate. These motion will repeat with cutting depth (Δi) in each infeed until the tool reaches the position B.

[2] Notes on programming of G74

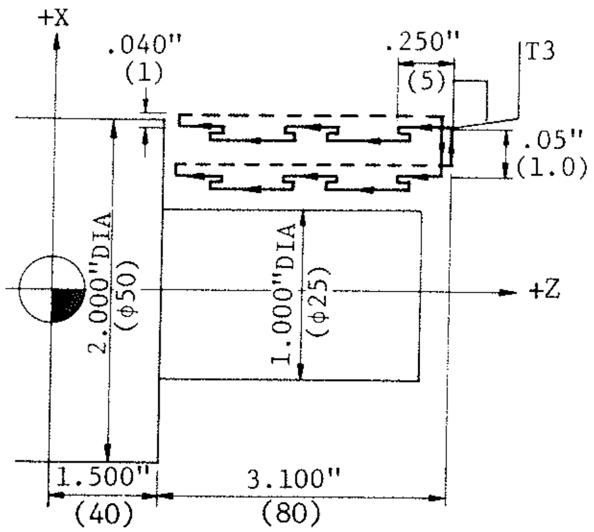
(1) G74 cycle starts from the position A and finishes at the position A. Therefore, the tool must be approached to the position A before starting G74.

(2) When the value of R(e) is specified by zero or R(e) is not specified, the relief motion is not executed.

(3) When X(U) and P are not specified, only one operation is made in the Z direction. This motion can be used for deep hole drilling operation.

(4) The tool nose radius compensation is ineffective for G74 cycle.

[3] Example of programming by using G74



In Inch system

```
N301      T0300 ;
N302 G96 S450 M03 ;
N303 G00 X1.9 Z4.6 T0303 M28 ;
N304 G74 R.05 ;
N305 G74 X1.0 Z1.5 P.05 Q.25 D.04 F.006 ;
N306 G00 X4.0 Z4.8 M29 ;
     G00 X0 Z0 T0000 ;
N307      M01 ;
```

In Metric system

```
N301      T0300 ;
N302 G96 S120 M03 ;
N303 G00 X48.0 Z120.0 T0303 M28 ;
N304 G74 R1.0 ;
N305 G74 X25.0 Z40.0 P1.0 Q5.0 D1.0 F.2 ;
N306 G00 X70.0 Z125.0 M29 ;
     G00 X0 Z0 T0000 ;
N307      M01 ;
```

7-6 G75 (Cross Pecking Cycle)

The G75 cycle is the cross cutting cycle with pecking motion and it repeats with specified cutting depth in each infeed until the tool reaches the specified position in Z-axis. Therefore, chip control can be programmed easily.

[1] Method of program

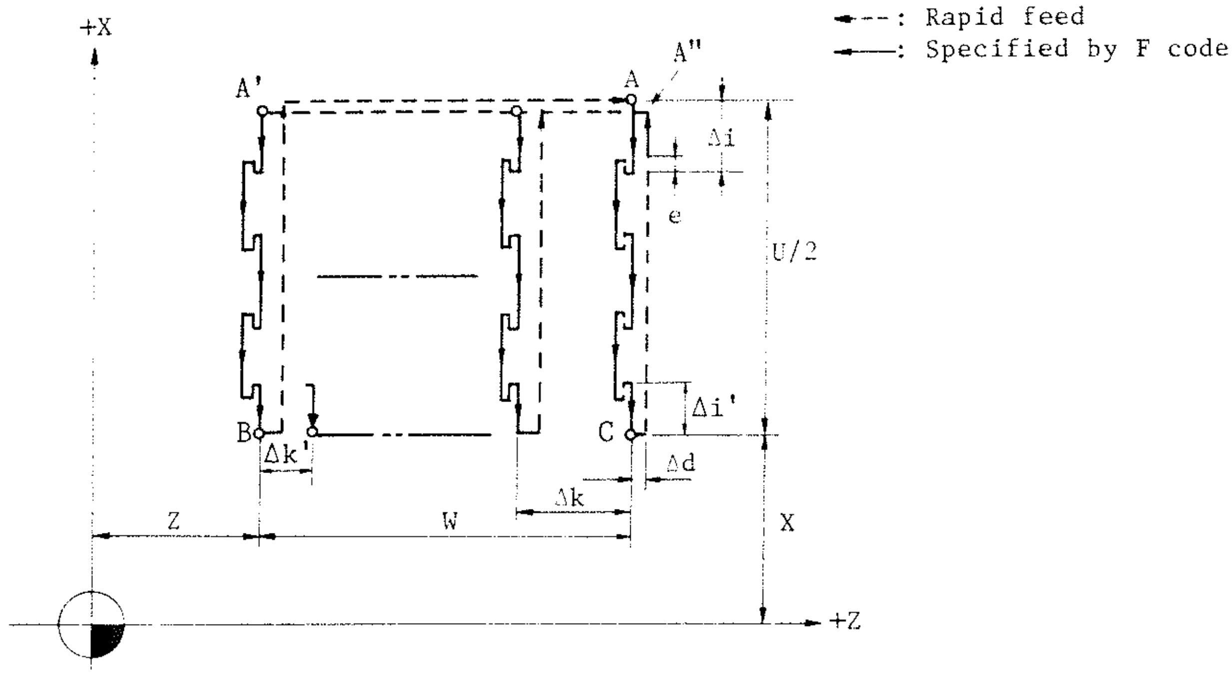
N ____ G75 R(e) ;

- R(e): Relieving value

N ____ G75 X(U) ± ___ Z(W) ± ___ P(Δi) Q(Δk) R(Δd) F ___ ;

- X(U): Absolute position (B) in X-axis or the distance between A' and B
- Z(W): Absolute position (B) in Z-axis or the distance between the position C and B
- P(Δi): Movement amount in X-axis for a pass (Specified without the sign and by diameter)
- Q(Δk): Movement amount in Z-axis for a pass (Specified without the sign)
- R(Δd): The amount of relief (Specified without the sign)
- F: Feed rate

By specifying the G75 cycle, a tool moves from the position A to C with repeating pecking motion and returns from the position C to A at rapid feed rate. These motion will repeat with cutting depth Q(Δk) in each infeed until the tool reaches the position B.



Relieving value (e) is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 0722, and the parameter is changed by the program command.

[2] Notes on the programming of G75

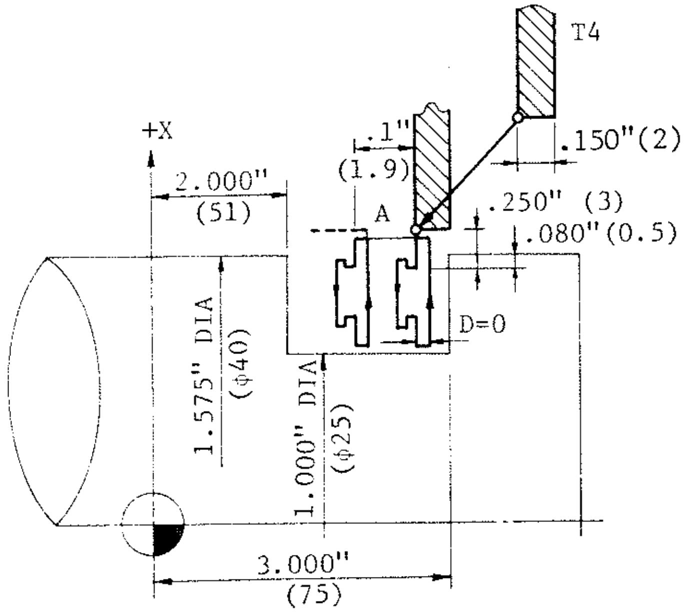
(1) G75 cycle starts from the position A and finishes at the position A. Therefore, the tool must be approached to the position A before starting G75.

(2) When the value of R(e) is specified by zero or R(e) is not specified, the relief motion is not executed.

(3) When Z(W) and P are not specified, only one cycle operation is made in the X direction. This motion may be useful for deep grooving.

(4) The tool nose radius compensation is ineffective for G75 cycle.

[3] Example of programming by using G75



In Inch system

```
N401      T0400 ;
N402 G96 S400 M03 ;
N403 G00 X1.675 Z2.9 T0404 M28 ;
N404 G75 R.05 ;
N405 G75 X1.0 Z2.0 P.25 Q.1 F.006 ;
N406 G00 X4.0 Z4.0 M29 ;
     G00 X0 Z0 T0000 ;
N407      M01 ;
```

In Metric system

```
N401      T0400 ;
N402 G96 S120 M03 ;
N403 G00 X41.0 Z73.1 T0404 ;
N404 G75 R1.0 ;
N405 G75 X25.0 Z51.0 P3.0 Q1.9 F.07 ;
N406 G00 X50.0 Z85.0 M29 ;
     G00 X0 Z0 T0000 ;
N407      M01 ;
```

BND-S PRO 0-TC 90/8

7-7 G76 (Thread Cutting Cycle)

This cycle provides automatic thread cutting for straight or taper thread.
The infeed for each pass of threading in this cycle is made along the angle of the thread.

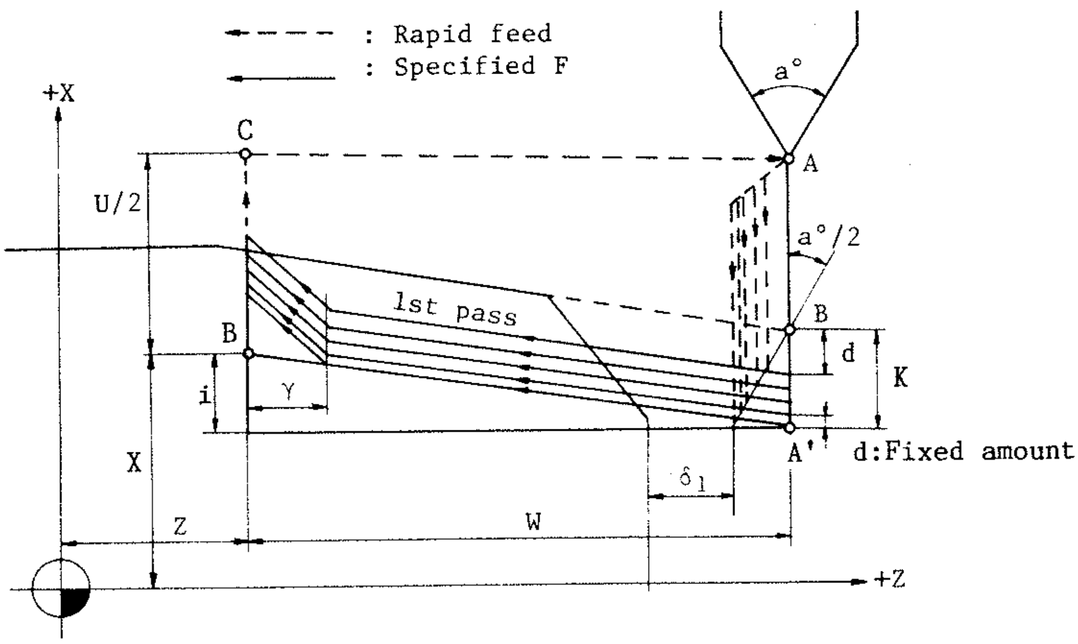
[1] Method of program

```
N___G76 P(m)(r)(a) Q(Δdmin) R(d) ;
```

- m: Repetitive times in finishing cut (1 to 99)
- r: Width of thread pull out
- a: Angle of the thread
- Δdmin: Minimum depth of cut
- d: Fixed amount

```
N___G76 X(U) ±___Z(W) ±___ R±(i) P(k) Q(Δd) F___;
```

- X(U): Absolute position (B) in X-axis or distance between the position A and B
- Z(W): Absolute position (B) in Z-axis or distance between the position A and B
- i: Difference of radius value between start and end position of the thread at the final pass
- k: Height of the thread (Specified without the sign and by radius value)
- Δd: Depth of the 1st pass
- F: Lead of the thread



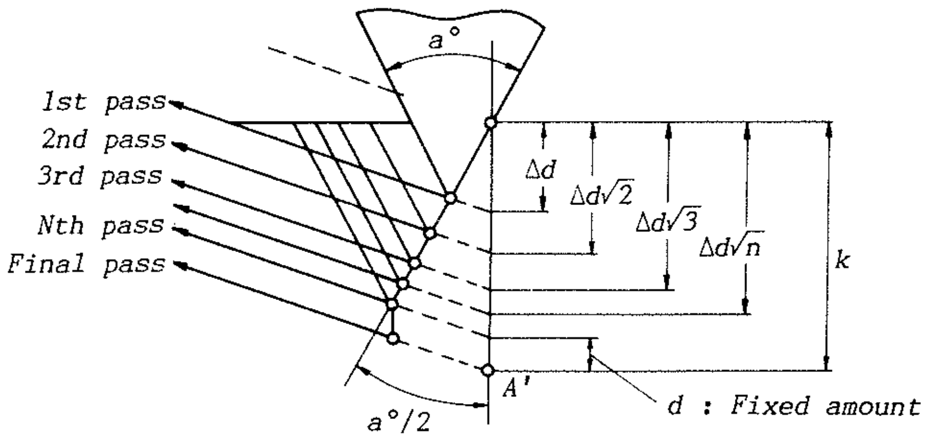
[2] Notes on programming of G76

(1) (m), (r), and (a) are specified by address P at the same time.

Example
When m=2, r=1.2ℓ, and a=60°, specify as shown below.

```
P 02 12 60 ;
  ↑  ↑  ↑
  |  |  └── a
  |  └───── r
  └──────── m
```

(2) When a thread angle is specified, the infeed for each pass is as follows ; 'd' can be clamped by the parameter No.0725, and the parameter is changed by the program command.



The depth of thread (d) in Nth pass is $d=\Delta d\sqrt{n}$.
The final pass is made with the depth of fixed amount (d) by infeeding perpendicularly. This fixed amount (d) is set by the parameter No.0726, and the parameter is changed by the program command.

(3) Straight thread

When the address R(i) is not specified, the straight thread is made. The infeed for each pass is the same as the taper threading.

(4) The following angles can be used as the command of thread angle.

$\boxed{A = 0°,\ 29°,\ 30°,\ 55°,\ 60°,\ 80°}$

This thread angle (a) is set by the parameter No.0724, and the parameter No. 0724, and the parameter is changed by the program command.

(5) The sign (+ or -) of the address R(i) is determined by the direction of the position A' viewed from the position B.

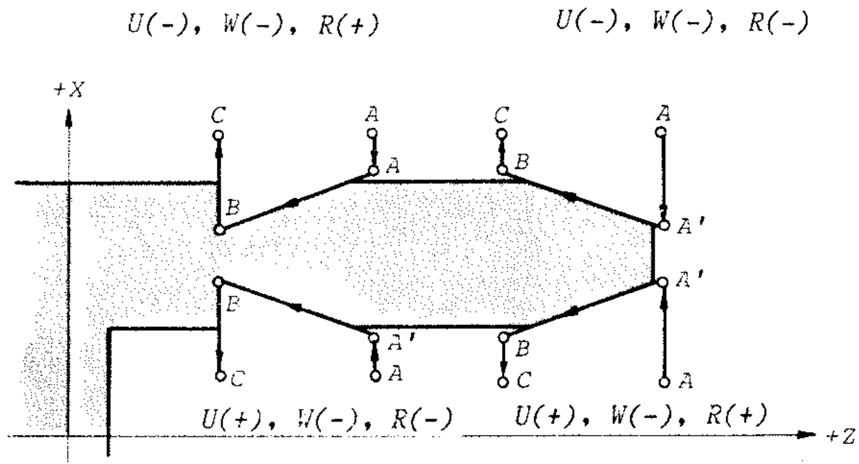
(6) The address P(k) and Q(Δd) are specified without the sign.

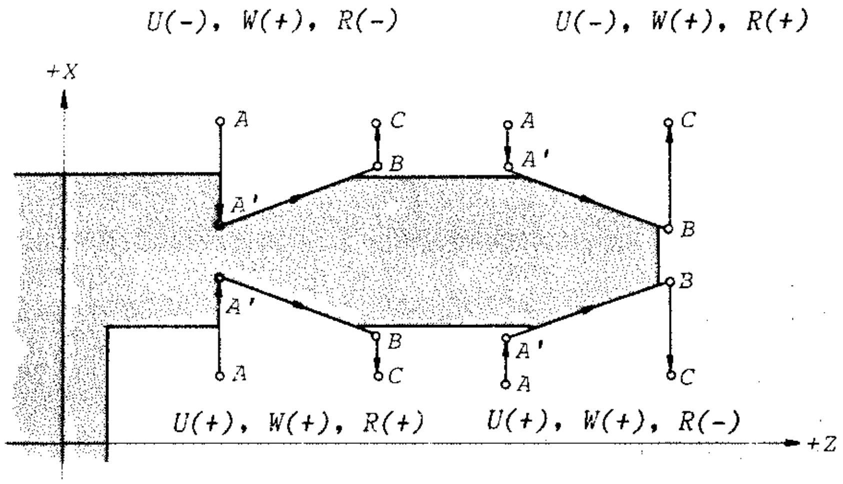
(7) If M23 is commanded in the preceding block, of G76, the thread will be pulled out, and if M24 is commanded, the thread will not be pulled out.
When the thread lead is expressed by ℓ, the value of ℓ can be set from 0.0ℓ to 9.9ℓ in 0.1ℓ increment (2-digit number from 00 to 99). This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No.0022, and the parameter is changed by the program command.

(8) Refer to the table in explanation of G32 to find out the value of $\delta 1$ and $\delta 2$ (Length of incorrect lead).

(9) The tool nose radius compensation is ineffective in G76 cycle.

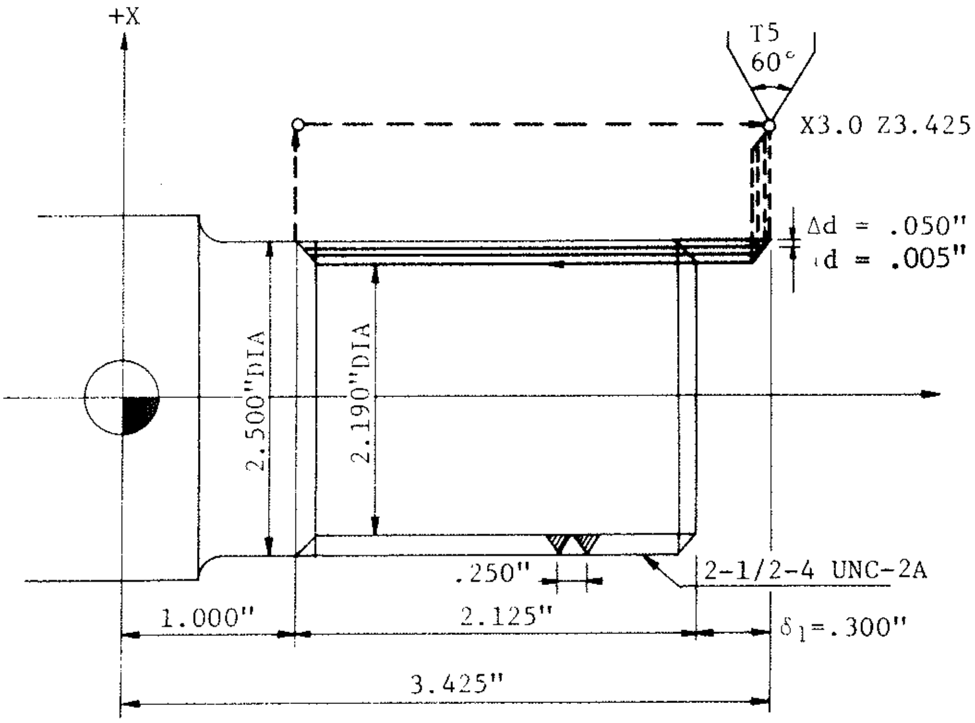
(10) There are eight patterns to use G76 by the sign (+ or -) of U, W and R(i). Their signs are determined by the direction of the position B viewed from the position A.





[3] Example of programming by using G76

In Inch system



In case of Q(Δd)= .050 and the fixed amount R(d)= .005, the number of passes is calculated as follows;

$$P(k) = \frac{2.500 - 2.190}{2} = .155$$

The number of passes is,

$$\Delta d\sqrt{n} \geqq P(k) - d = .155 - .005 = .150$$

$$n \geqq \left(\frac{.150}{.050}\right)^2 = 9.0 \qquad \underline{9 \text{ passes}}$$

Therefore, the depth of thread in each pass is as follows;

1st pass ——→ .050
2nd pass $.050 \times \sqrt{2} = .071$
3rd pass $.050 \times \sqrt{3} = .087$
4th pass $.050 \times \sqrt{4} = .100$
5th pass $.050 \times \sqrt{5} = .112$
6th pass $.050 \times \sqrt{6} = .122$
7th pass $.050 \times \sqrt{7} = .132$
8th pass $.050 \times \sqrt{8} = .141$
9th pass $.050 \times \sqrt{9} = .150$
The final pass = 9th pass + d = .155

BND-S PRO 0-TC 90/8

The length of incorrect lead can be found from the table in explanation of G32.

$\delta 1 \geq .300$

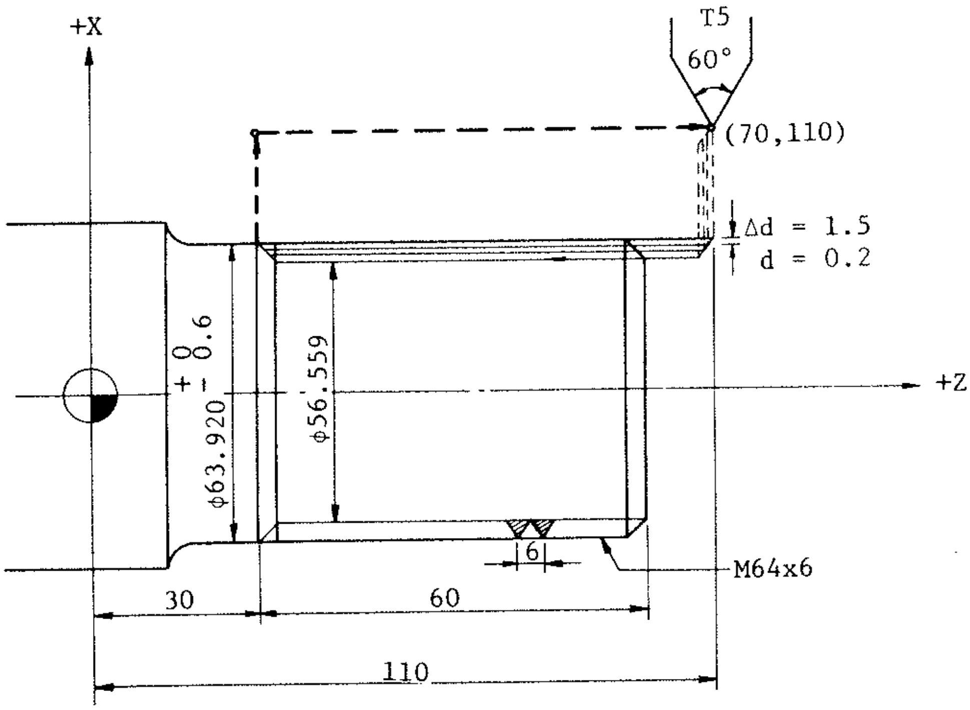
If the cutting speed is 400 ft/min, the spindle speed is 610 rpm.

L x R = .250 x 610 = 152.5 < 472 in/min within the restriction

The program is as follows ;

```
N501      T0500 ;
N502 G97 S610 M03 ;
N503 G00 X3.0 Z3.425 T0505 M23 ;
N504 G76 P010560 Q.005 R.005 ;
N505 G76 X2.19 Z1.0 P.155 Q.05 F.25 ;
N506 G00 X3.5 Z4.0 M24 ;
     G00 X0 Z0 T0000 ;
N507      M01 ;
```

In Metric system



In case of Q(Δd)= 1.5 and the fixed amount R(d)=0.2, the number of passes is calculated as follows;

$$P(k) = \frac{63.920 - 56.559}{2} = 3.68$$

The number of passes is,

$$P(k) \geqq 3.68-0.2+(1.5 - 0.75) = 4.23$$

$$n \geqq \left(\frac{4.23}{1.5}\right)^2 = 7.95 \quad \underline{8 \text{ passes}}$$

Therefore, the depth of thread in each pass is as follows ;

1st pass $1.5 \longrightarrow 0.75$
2nd pass $\sqrt{2} \times 1.5 - 0.75 = 1.37$
3rd pass $\sqrt{3} \times 1.5 - 0.75 = 1.85$
4th pass $\sqrt{4} \times 1.5 - 0.75 = 2.25$
5th pass $\sqrt{5} \times 1.5 - 0.75 = 2.60$
6th pass $\sqrt{6} \times 1.5 - 0.75 = 2.92$
7th pass $\sqrt{7} \times 1.5 - 0.75 = 3.22$
8th pass $\sqrt{8} \times 1.5 - 0.75 = 3.492$
The final pass = 8th + d = 3.692

The length of incorrect lead can be found from the table in explanation of G32.

$$\delta 1 \geqq 7.5 \text{ mm}$$

If the cutting speed is 120 m/min, the spindle speed is 597 rpm.

$$L \times R = 6.0 \times 597 = \underline{3582 < 12000 \text{ mm/min}} \text{ within the restriction}$$

The program is as follows ;

```
N501      T0500 ;
N502 G97 S597 M03 ;
N503 G00 X70.0 Z110.0 M23 T0505 ;
N504 G76 P010560 Q.2 R.2 ;
N505 G76 X56.559 Z30.0 P4.443 Q1.5 F6.0 ;
N506 G00 X80.0 Z120.0 M24 ;
     G00 X0 Z0 T0000 ;
N507      M01 ;
```

(Note)
Notes on thread cutting are the same as those on G32 thread cutting and G92 thread cutting cycle.

BND-S PRO 0-TC 90/8

7-8 Notes on Multiple Repetitive Cycles (G70 to G76)

(1) In the blocks where the multiple repetitive cycle are commanded, the addresses P, Q, X, Z, U, W, and R should be specified correctly for each block.

(2) In the block which is specified by address P of G71, G72, or G73, G00 or G01 of 01 group should be commanded. If it is not commanded, P/S alarm No. 65 is generated.

(3) In MDI mode, G71, G72, or G73 cannot be commanded. If it is commanded, P/S alarm No. 67 is generated. G74, G75, and G76 can be commanded in MDI mode.

(4) In the blocks in which G70, G71, G72 or G73 are commanded and between the sequence number specified by P and Q, M98 (subprogram call) and M99 (subprogram end) cannot be commanded.

(5) In the blocks between the sequence number specified by P and Q, the following commands cannot be specified.
- One shot G code except for G04 (dwell)
- 01 group G code except for G00, G01, G02 and G03
- 06 group G code
- M98/M99

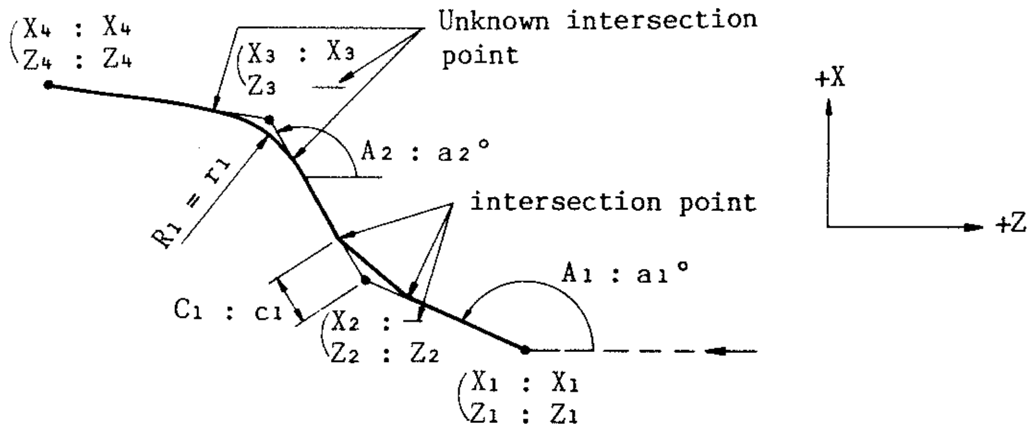
(6) While a multiple repetitive cycle is being executed, it is possible to stop the cycle and to perform manual operation. But, when the cycle operation is restarted, the tool should be returned to the position where the cycle operation is stopped.
If the cycle operation is restarted without returning to the stop position, the movement in manual operation is added to the absolute value, and the tool path is shifted by the movement amount in manual operation.

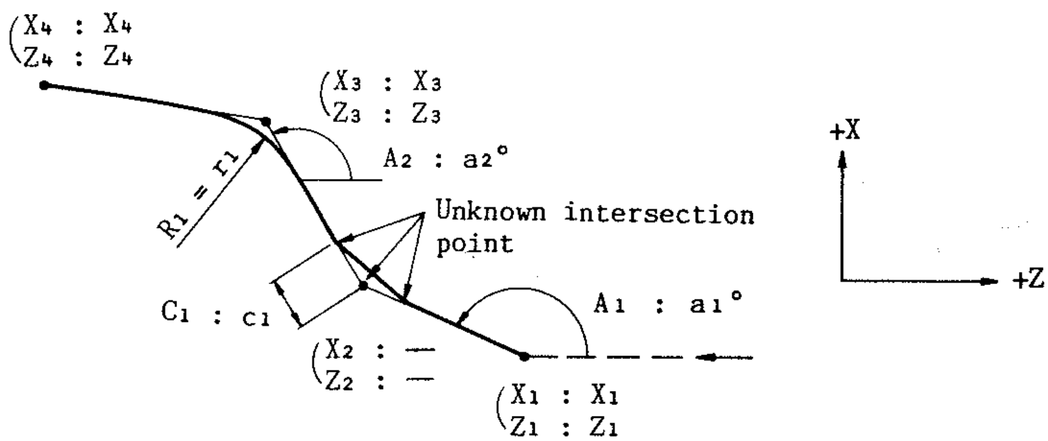
(7) When G70, G71, G72, or G73 is executed, the sequence number specified by address P and Q should not be specified twice or more in the same program.

(8) **Do not program so that final movement command of the finishing shape block group designated with P and Q for G70, G71, G72 and G73 finishes with pull out or corner rounding. If it is specified, P/S alarm No. 69 is generated.**

8. DRAWING DIMENSION DIRECT INPUT (OPTION)

This function automatically computes an intersection point and programs when only angle is given, taper intersection point is unknown for chamfering and corner rounding, or when only X coordinates or Z coordinates are given.

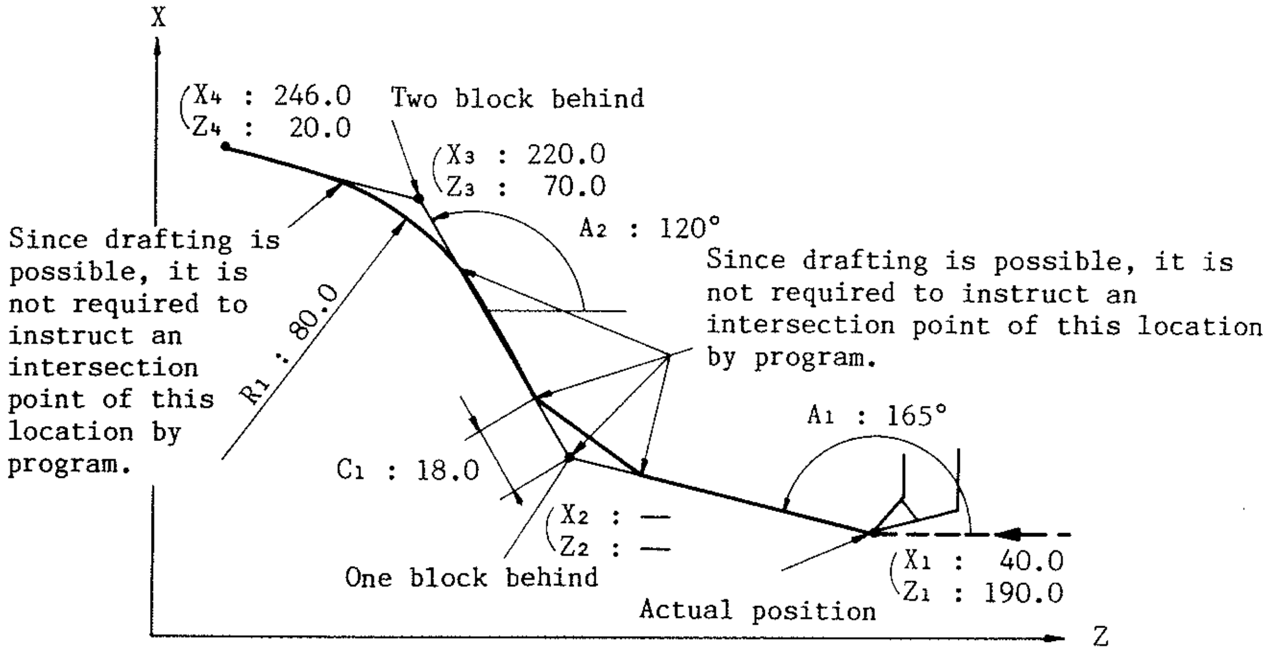




BND-S PRO 0-TC 90/8

[1] Instruction method

Programming is possible by using the values given on the machining drawing which are two blocks behind actual position, it is not required to compute each intersection point and a program can be made by using angle of taper, value of chamfer and value of corner as is.



Program

```
G01 A165.0 C18.0 F0.2 ;
    X220.0 Z70.0 A120.0 R80.0 ;
    X246.0 Z20.0 ;
           .
           .
           .
           .
           .
```

[2] Commanding of angle, arc and chamfer

(1) Commanding of angle

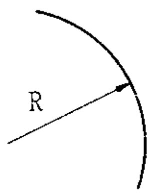
Angle is commanded subsequent to address A and from reference line in counter-clockwise.

When only angle is commanded in one block, both coordinates value (absolute command) and angle must be designated in the following block.

N1 A____;
N2 X____Z ____A ____; ←-------- It is required to designate both coordinates and angle in N2 block.

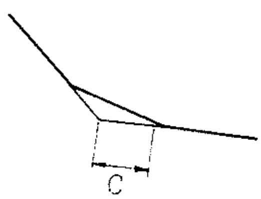
(2) Commanding of arc

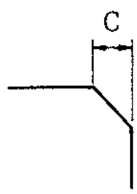
Arc is commanded by a radius value following the address R.



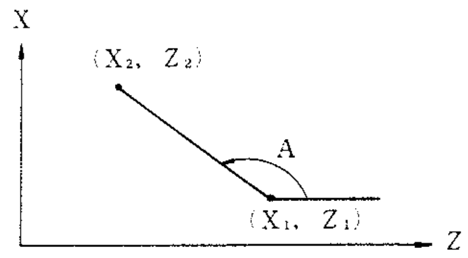
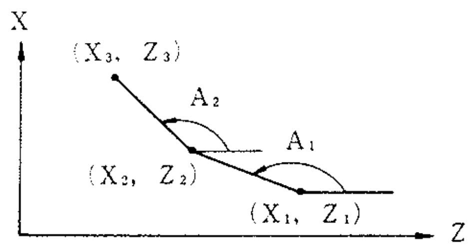
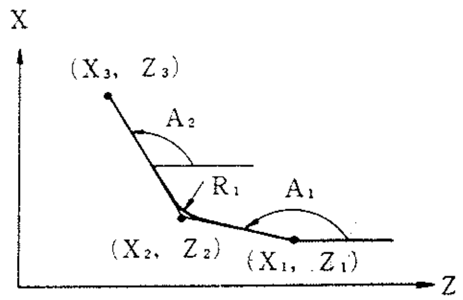
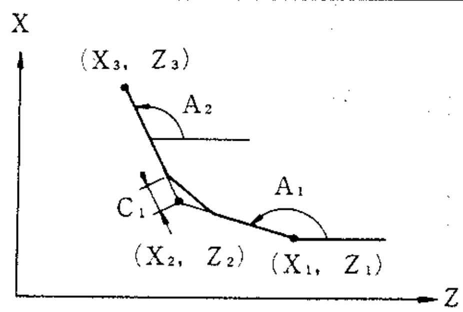
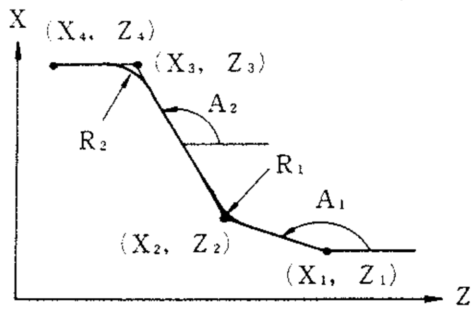
(3) Commanding of chamfer

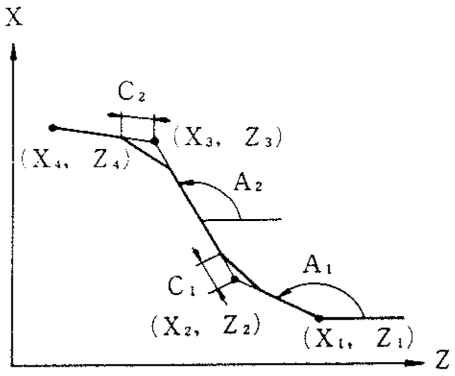
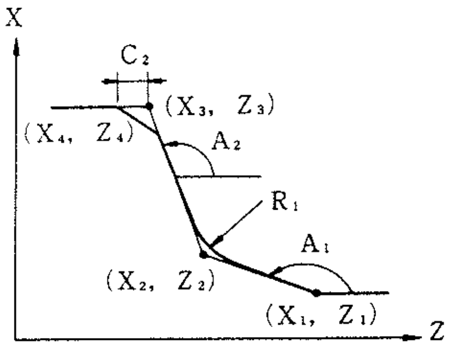
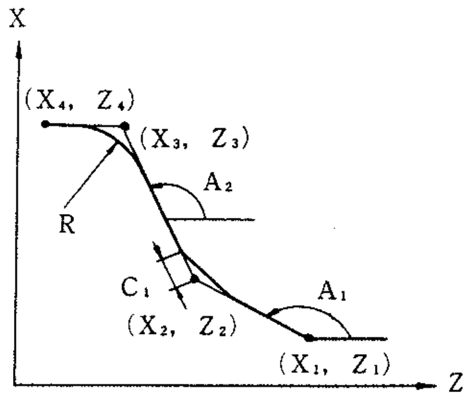
Chamfer is commanded by a chamfer value following the address C.



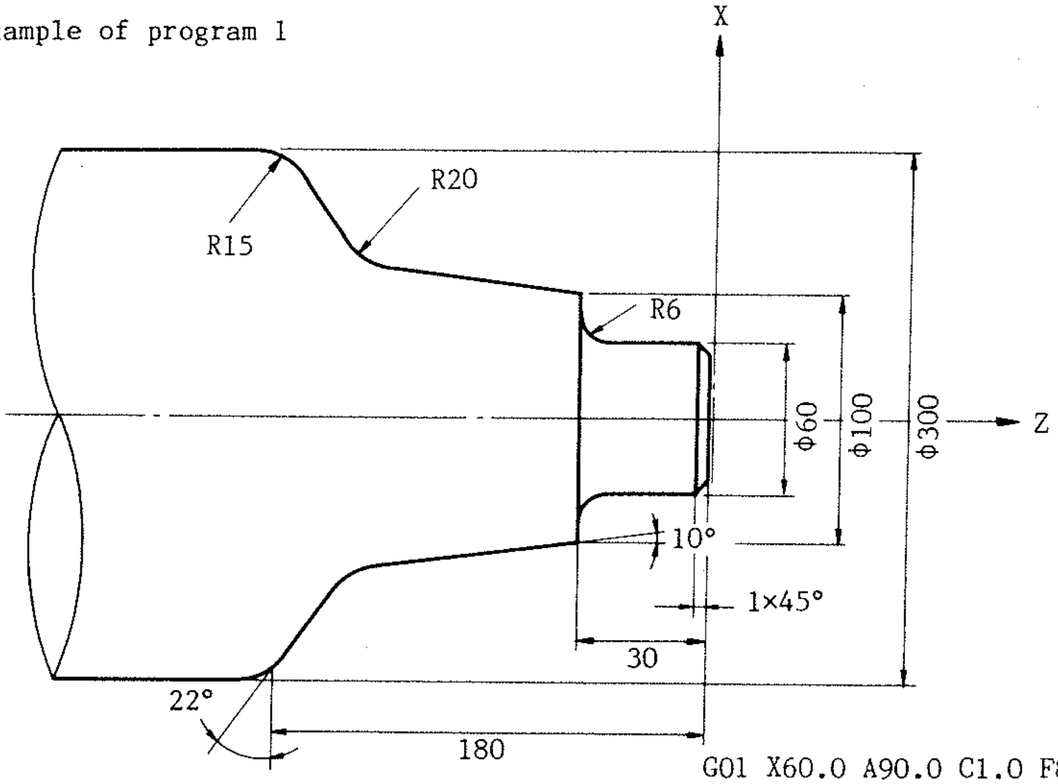


[3] Commands table

	Commands	Movement of tool
1	X_2 ___ (Z_2 ___) A ___ ;	
2	A_1 ___ ; X_3 ___ Z_3 ___ A_2 ___ ;	
3	X_2 ___ Z_2 ___ R_1 ___ ; X_3 ___ Z_3 ___ ; or A_1 ___ R_1 ___ ; X_3 ___ Z_3 ___ A_2 ___ ;	
4	X_2 ___ Z_2 ___ C_1 ___ ; X_3 ___ Z_3 ___ ; or A_1 ___ C_1 ___ ; X_3 ___ Z_3 ___ A_2 ___ ;	
5	X_2 ___ Z_2 ___ R_1 ___ ; X_3 ___ Z_3 ___ R_2 ___ ; X_4 ___ Z_4 ___ ; or A_1 ___ R_1 ___ ; X_3 ___ Z_3 ___ A_2 ___ R_2 ___ ; X_4 ___ Z_4 ___ ;	

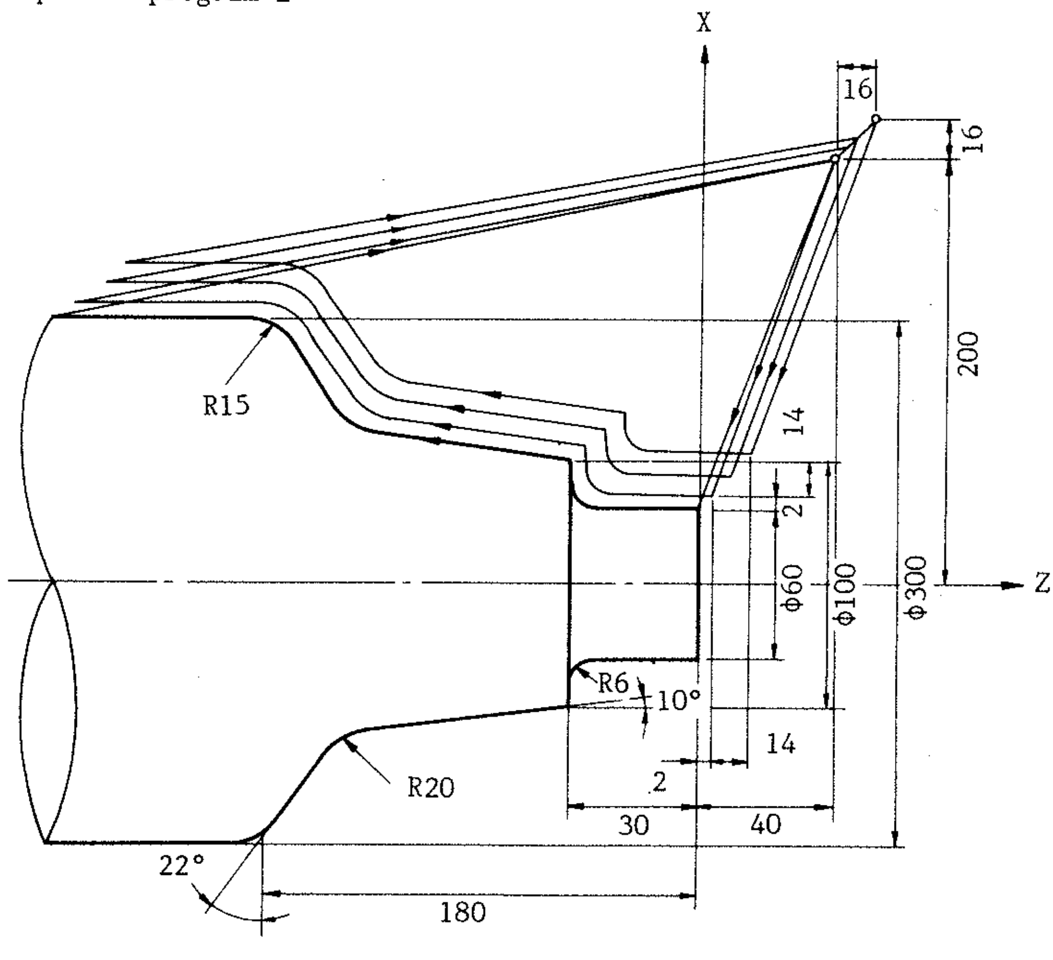
	Commands	Movement of tool
6	X_2 __ Z_2 __ C_1 __ ; X_3 __ Z_3 __ C_2 __ ; X_4 __ Z_4 __ ; or A_1 __ C_1 __ ; X_3 __ Z_3 __ A_2 __ C_2 __ ; X_4 __ Z_4 __ ;	
7	X_2 __ Z_2 __ R_1 __ ; X_3 __ Z_3 __ C_2 __ ; X_4 __ Z_4 __ ; or A_1 __ R_1 __ ; X_3 __ Z_3 __ A_2 __ C_2 __ ; X_4 __ Z_4 __ ;	
8	X_2 __ Z_2 __ C_1 __ ; X_3 __ Z_3 __ R_2 __ ; X_4 __ Z_4 __ ; or A_1 __ C_1 __ ; X_3 __ Z_3 __ A_2 __ R_2 __ ; X_4 __ Z_4 __ ;	

Example of program 1




```
G01 X60.0 A90.0 C1.0 F80 ;
    Z-30.0 A180.0 R6.0 ;
    X100.0 A90.0 ;
    A170.0 R20.0 ;
    X300.0 Z-180.0 A112.0 R15.0 ;
    Z-230.0 A180.0 ;
```

Example of program 2



```
N001 G00 X400.0 Z40.0 ;
N002 G73 U14.0 W14.0 R3 ;
N003 G73 P004 Q009 U4.0 W2.0 F0.2 S120 ;
N004 G00 X60.0 Z0 ;                              ┐
N005 G01 Z-30.0 A180.0 R6.0 F0.15 ;              │
N006     X100.0 A90.0 ;                          │  Program with direct drawing
N007     A170.0 R20.0 ;                          ├  demension input.
N008     X300.0 Z-180.0 A112.0 R15.0 ;           │
N009     Z-230.0 A180.0 ;                        ┘
N010 G70 P004 Q009 ;
```

(Note 1)
The following G code can not be used in the same block as command of drawing dimension direct input or between blocks of drawing dimension direct input which define sequential figure.

(1) G codes in group 00 (Other than G04)

(2) G02, G03, G90, G92 and G94 in group 01

(Note 2)
Corner rounding cannot be inserted into a threading block .

(Note 3)
Commands of chamfering and corner rounding specified in 14.3 and chamfering and corner rounding of drawing dimension direct input can not be used.

(Note 4)
When the end point of previous block is determined in the next block according to sequential commands of drawing dimension direct input, the single block stop is not done, but the feed hold stop is done at the end point of the previous block.

(Note 5)
A limit angle to calculate intersection point in the following program is ±1°.
(This is because the travel distance to be obtained in this calculation is too large.)

(a) X A ; (When angle command designates a value within ±1° at 0° or 180° , an alarm occurs.)

(b) Z A ; (When angle command designates a value within ±1° at 90° or 270° , an alarm occurs.)

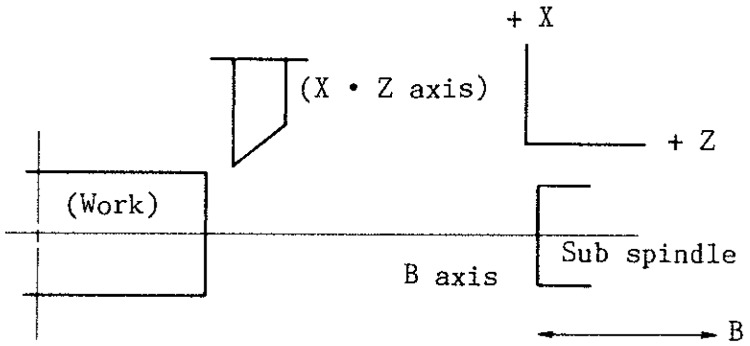
(Note 6)
An alarm occurs if the angle made by the 2 lines is within ±1° when calculating the point of intersection.

(Note 7)
Chamfer or corner R is ignored if angle made by the two lines within ±1°.

9. CONTROL OF B AXIS (CONTROL OF SUB SPINDLE)

9-1. Outline of Function

This function aims to install the fundamental control axis (X, Z axis) of "FANUC Series 0-TC" and independent one axis (B axis) parallel with Z axis and make it possible to operate B axis together with fundamental axis and perform composite machining of (X, Z axis) and (B axis).



B axis uses a control axis from PMC and by commanding operation command group and operation start command against B axis during NC program, it becomes possible to make B axis perform a series of operation in parallel with normal NC operation.

```
Example O-1234 ;
            :
          Normal program
            :
     G101 ;  ────────────── ① Start of operation command registration
     G00  B-80.0 ;  ─┐
     G01  B-100.0  F0.1 ;├── ② B axis operation command group
     G00  B-50.0 ;  ─┘
     G100 ;  ────────────── ③ End of registration of operation
                               command
     G00 X20.0 Z50.0 ;
     G01 X0 F0.1 ;
            :
     G00 X10.0 ;
     M140;  ─────────────── ④ Operation start command
     G01 Z30.0 F0.2 ;
            :
     M30 ;
```

① to ③ : Operation command of B axis is commanded between blocks of "G101" ("G102" "G103") and "G100".

④ : Operation of B axis registered by ① to ③ is started. Command is made by the auxiliary function (M140, M141 M142) and subsequently operation of B-axis in executed in parallel with fundamental NC.

9-2. Programming

A program of command against B axis is as mentioned in the previous paragraph and performed by registration of operation command and operation start command.

[1] Registration of operation command

This is a function to register data memory of operation command of B-axis 3 kinds of data memory can be registered for operation of B-axis. To differentiate an instruction from normal NC program, progammming is made between the blocks of "G101" (or "G102" "G103") and "G100" instructed operation registered executes the operation when the following M code for start of operation start command.

"G101" ∿ "G100"	Registration of No.1 program	→ Instructed by M140
"G102" ∿ "G100"	Registration of No.2 program	→ Instructed by M141
"G103" ∿ "G100"	Registration of No.3 program	→ Instructed by M142

```
O-1234
  |
  |  Normal NC program
  |
G101 ; ——————————————— Start of registration of B axis
                        operation command
  |
  |  B axis operation command
  |
G100 ; ——————————————— End of registration of B axis
                        operation command
  |
  |  Normal NC program
  |       (B axis operation is performed by M140 on the way.)
  |
M30 ;
```

The code which can be used for B axis operation commands are G code, M code and T code. (Refer to the paragraph of 9-3 detail of operation command for detail.)

B axis operation command code

Command code	Outline	
G00	Positioning (rapid feed)	
G01	Linear interpolation (cutting feed)	
G04	Dwell	
G28	Reference point return	
G80	Fixed cycle cancel	T type only (G80–G86)
G81	Spot drilling cycle	
G82	Counter boring cycle	
G83	Deep Drilling cycle	
G84	Tapping cycle	
G85	Boring cycle	
G86	Boring cycle	
G98	Feed per minute	
G99	Feed per revolution	
T__	Tool position offset	
M__	Auxiliary function	

(Note 1) Instruct a block of "G101" "G102" "G103" "G100" only be independent block.

(Note 2) When operation is registered in the data memory, operation command instructed before is canceled and new operation instruction is registered. In addition, when there is an alarm in the operative instruction desired to be registered, the data memory is initialized and a condition where there is no registration appears.

[2] Operation start command

Execution start of operation command of B axis registered in the previous paragraph is commanded. When start command is made, commanded operation is executed independently from normal NC operation (X, Z axis).
Execution is made by auxiliary function (M140)

```
0-1234 ;
  |
  |
  |
 M140 ; -------- B axis operation start command
  |               * By M140 command, B axis operation program and X and
  |                 Z axis in next block of M140 are concurrently
  |                 executed.
 M30 ;            * M140 (M141 or M142)
```

9-3 Detail of Operation Instruction

The contents of G code, M code, and T code becomes as follows so that they can be used between "G101"("G102" or "G103") ∿ "G100".

1. G00 .... Positioning (rapid feed)

   G00 B99.9 ;

   Movement is made from current position to the position commanded by B or by transfer amount with rapid feed.
   In case of mm input : -9999.999mm ∿ +9999.999mm
   In case of INCH input : -999.9999inch ∿ +999.9999inch

2. G01 .... Linear interporation (cutting feed)

   G01 B99.9 F9999 ; { Feed per minute (G98) 0 to 393.33 in/min (1 to 10000 mm/min)
   Feed per revolution (G99) .000001 to 9.999999 in/rev (0.0001 to 50.0000 mm/rev) }

   Movement is made from current position to the position or by transfer amount commanded by B with commanded speed.

3. G04 .... Dwell

   G04 U9999.999 (P9999)

   Transfer to nest block is delayed by time commanded by U or P.
   (For U, decimal input is possible and for P, minimum setting unit is input.)

4. G28 .... Reference point return

   G28 ;

   Automatic return to the reference point is made.
   Since there is no processing of intermediate point as normal G28, pay attention.
   The following instruction can not be made.

   G28 B ____; (Example of no instruction)

5. G80 ∿ G86 .... Drilling fixed cycle (Refer to the separate sheet for detail.) (Mainly used for T type)

   G 8□ B 9 9 9 9 R 9 9 9 9 Q 9 9 9 9 P 9 9 9 9 F 9 9 9 9

   Among drilling fixed cycle which can be performed by FANUC OM-MODEL B, cycle operations corresponding G80 ∿ G86 can be made.
   Instruction methods of each data are the same as OM-MODEL B except for the following.

   . All operates initial level return mode.
   . Number of repetition (K) cannot be designated.
   . Since decimal point input of address cannot be made, instruction is made by the minimum setting unit.
     (0.001 mm during millimeter input and 0.0001 inch during inch input.)
   . During fixed cycle mode, reference point return is always necessary.
     (When there is no setting, an alarm of P/S 505 occurs.)
   . Setting (d) of cutting start point at G83 (deep drilling cycle) is set at No. 9005 of parameter.

6. G98 .... Feed per minute
   G99 .... Feed per revolution

In this device model initial value of start of operation instruction registration during (G101, G102, G103) and G110 is set at G99 (feed per revolution).
In the case of G99 (feed per revolution) mode, feed speed E is obtained by calculating from actual spindle rotation speed at the time of start of each cutting feed block.
Pay attention since during movement of cutting feed block feed speed does not change by changing spindle rotation speed.

(Note)
Spindle rotation speed cannot be changed between G101 (G102, G103) to G100.

Example

G110 G01 B....F0.2 ; ....0.2 mm/rev
Since a model initial value is G99, operation ia made by feed per revolution.

G110 G98 G01 B....F200 ...... 200 mm/MIN
Since a model initial value is G99, G98 must be always designated after G110 (G101, G102, G103) when G98 (feed per minute) is used.
The relationship of feed per revolution and feed per minute is as follows :

Inch Feed per minute (inch/MIN)= Spindle rotation speed (RPM) x feed per revolution (inch/rev).

Metric Feed per minute (mm/MIN)= Spindle rotation speed (RPM) x feed per revolution (mm/rev).

BND-S PRO 0-TC 90/8

7. T Code

Refer to tool position offset of the following paragraph.

8. Auxiliary Function (M Function)

* For S type, the following M codes can be used between "G101" (G102, G103) ∿ "G100".

M03 -- Spindle Forward (Counter-clockwise)
M04 -- Spindle Reverse (Clockwise)
M05 -- Spindle Stop
M12 -- Bar Feeding
M17 -- Main Chuck Open
M18 -- Main Chuck Close
M28 -- High Pressure Coolant ON (Optional)
M29 -- High Pressure Coolant OFF (Optional)
M35 -- Part Catcher Forward (Optional)
M36 -- Part Catcher Backward (Optional)
M83 -- Material Feeder Start (Optional)
M84 -- Block Delete OFF Reset (Optional)
M103 -- Subspindle Forward Rotation
M104 -- Subspindle Reverse Rotation
M105 -- Subspindle stop
M117 -- Subspindle chuck, open
M118 -- Subspindle chuck, close
M176 -- Sub air blow No.4 ON (For sub spindle)(Optional)
M177 -- Sub air blow No.5 ON (Optional)
M179 -- Sub air blow stop (Optional)
M190 -- Waiting Command
M191 -- Sub encoder ON
M203 -- Main and sub spindle forward synchronous rotation
M204 -- Main and sub spindle reverse synchronous rotation
M205 -- Main and sub spindle synchronous stop
M206 -- Synchronous operation release

9. Synchrnous Operation (Waiting Command) M190

Normally, although fundamental NC program and operation of B axis are quite independent, by combining auxiliary function for B axis control and auxiliary function with fundamental NC program, synchronization can be obtained.

(Fundamental NC operation)	(B axis operation)
① M140 :	① G00 B-50.0 ;
② G01 X20.0 F0.1 ;	② G01 B-60.0 F0.15 ;
③ G00 Z50.0 ;	③ G00 B-50.0 ;
④ M190 ;	④ M190 ;
⑤ G00 X10.0 ;	⑤ G00 B-59.0 ;

When command of fundamental "M190" with ladder of PMC and "M190" with B axis operation is received, both FIN signals are output.
⑤ of fundamental NC operation and ⑤ of B axis operation are started at the same time.

(Note)
In B axis operation program, M code and T code cannot be commanded in the same block.

BND-S PRO 0-TC 90/8

9-4 Offset (Mainly used for T type)

1. Instruction Code

With automatic operation mode, they are effective between "G101 (G102, G103) ∿ "G100".
However, instruct as an independent block.
By instruction of

T70 ∿ T79 ;

a termination position of movement instruction of B-axis can be shifted by a value set is the AUX screen to "+" or "-" side.
By utilizing this function, compensation can be made without changing the program by setting shift between a value of tool position assumed when programming is made and a tool position of tool which is used for actual working.

2. Command of offset amount

B axis moves by offset amount set in offset number by T code command.
In addition, this auxiliary function number is displayed on "AUX screen" and setting of offset amount is made here.

A range of value which can be set as offset amount is as follows :

	mm input	inch input
Offset amount	0±9999.999	0±999.9999

(Unit is minimum setting unit.)

In this device, the offset amount can be set by separating into "WEAR" and "GEOMETRY".
In addition, the offset amount moves by the total of both "WEAR" and "GEOMETRY" to be set. Incidentally, since the offset input of this function in only absolute input both for "WEAR" and "GEOMETRY".

3. Cancel of tool position offset

T70 : Tool position offset cancel number
T71 to T79 : Tool position offset number (9 kinds)

4. Movement of offset amount

For this machine, in the block of T code, offset amount is added to axis movement amount in next axis movement block only by output of MF and M codes.

5. AUX screen

OFFSET (B-AXIS)

No.	(WEAR)	(GEOMETRY)
71	0.000	0.000
72	0.000	0.000
73	0.000	0.000
74	0.000	0.000
75	0.000	0.000
76	0.000	0.000
77	0.000	0.000
78	0.000	0.000
79	0.000	0.000

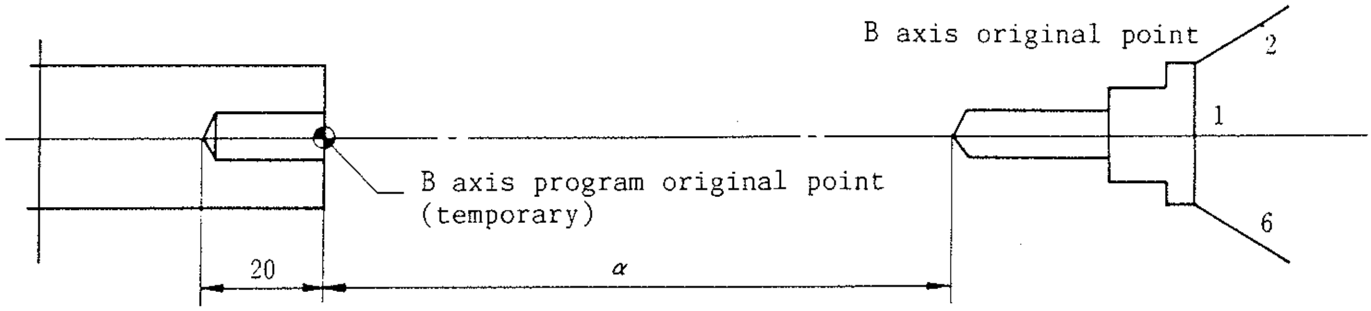
Cursor

GEOM=

S1000 T0100

AUTO

Example) B axis shall start from the original point and returns to the original point after completion of machining.



A value of "α" is input to the tool number and corresponding offset (shape).

* In above figure of example a "cursor" is moved to the shape of T71 by pressing [↑] [↓] (cursor).

* $-\alpha$ [INPUT] is input.

* A value of a is the distance from B axis original point to B axis program original point.

Above program of operation:

```
   G101 ;
(1) T71 ;
(2) G00 B1.0 ;
(3) G01 B -20.0 F___ ;
(4) T70 ;
(5) G00 B0 ; (OR, G28;)
   G100 ;
```

Explanation of operation:

(1) T71 is read but B axis does not move.
(2) Moved to the position of T71 + 1.0 by rapid feed.
    (T71 + 1.0 = -100.0 + 1.0 =-99.0)
(3) Moved to the position of T71 -20.0 by F feed.
    (T71 - 20.0 = -100.0 -20.0 = -120.0)
(4) T70 offset cancel.
(5) Returns to the original point by rapid feed.
    ("B0" under the offset cancel condition is the original point.)

Normally, the following subturrets and used:

Subturret	No.1 --- T71	Subturret	No.4 --- T74
"	No.2 --- T72	"	No.5 --- T75
"	No.3 --- T73	"	No.6 --- T76

9-5 Independent Movement and JOG Operation of B axis

1. Independent movement of B axis

A) By registerating movement command in the data memory and commanding operation start, independent movement of B axis can be performed.

```
          |
          |  Normal program
          |
N10 G101 ; (Or G102, G103)
N11 G00 B-50.0 ;
N12 G81 B-100.0 R-60.0 F0.2 ;
N13 M190 ;
N14 G100 ;
N15 M140 ; (M141 or M142)
N16 M190 ;
          |
          |  Normal program
          |
```

As shown in above-mentioned program, by utilizing M190 (Waiting command), independent operation of B axis is possible.

Explanation : (1) NC operation is executed to N9 along program.
(2) N10 to N14 are registered in the data memory but not executed.
(3) Operation command of B axis operation registered in the data memory is performed by N15, M140 command.
(4) Although main side becomes stop condition by N16 M190 command, at subside, after operation of B axis has been completed, main side proceeds to next block by N13 M190 command.
(5) Subsequently, NC is executed along program of fundamental control axis (X, Z axis).

B) Different from the method where operation of B-axis is started by registering before mentioned No.1 ∿ No.3 program (G101 ∿ G103), the operation of one motion can be instructed and executed.

G110 .... ; (Operation instruction)

By instructing G110 block, the operation of one motion of B-axis can be instructed and executed.

G110 G01 B ... F ... ; Independent movement of B-axis block.

BND-S PRO 0-TC 90/8

Caution)

1. *The operation instructed by G110 must be the operation of one motion.*

   G110 G00 B ..... ;
 G110 G28 ; *Instruction possible*

   Although the operation by G110 is one motion, by instructing continuously G110 as mentioned above, continuous operation is possible.

2. *Under MDI mode, instruction of B-axis by G110 can not be made.*

3. *Operation instruction of plural motions such as instructions of fixed cycle with G110 (G81 ∿ G86) cannot be made.*
   (P/S alarm 506 occurs.)

   G110 G81 B___ R____ F___ ; An alarm 506 occurs.

4. *When G110 is used, tool offset cannot be used.*

5. ***During the tool nose radius compensation mode, two or more G110 blocks can not be commanded continuously.***

2. JOG operation and reference point return of B axis

JOG operation and manual reference point return can be performed in the same manner as fundamental control axis (X, Z axis). Movement by manual pulse generator (handle) is also possible.

9-6 Data Memory

The data memory in which operation command is registered counts movement, dwell, auxiliary function, etc. as one block and has memory of total of 200 blocks. Even in case of drilling fixed cycle of G81, etc., registration is made by dividing into movement, dwell, etc.
In addition, since all data memory are backed up by battery, operation command is not cleared when power source of the system is made OFF.

Example :

```
   ¦
   ¦
G101 ;
G00 B-10.0 ; ------------------------------- One block
G04 P1500 ; -------------------------------- One block
G81 B-50.0 R-30.0 F0.15 ; ------------------ Three blocks
G00 B0 ; ----------------------------------- One block
G100 ;
   ¦                                    Total six blocks
   ¦
```

9-7 Alarm

Since a check method of each command data is the same as normal NC program, normal P/S alarm such as excess of digit number, address which cannot be used, etc. is output. Here, an alarm peculiar to this machine is explained,. An alarm with this function is output as P/S alarm from 500 to 599.

Alarm with B axis control function

Alarm Number	Explanation
P/S501	Registration start command (G101, G102, G103) is not made and end command (G100) was commanded.
P/S502	During registration command, registration start command (G101, G102, G103) was commanded again.
P/S503	During execution of B axis operation, new operation command was tried to be registered.
P/S505	During fixed cycle mode, there is no setting of address R.
P/S506	Number of registerated block exceeded 200 blocks.
P/S507	For cutting feed, feed speed is not commanded.
P/S508	For G83 (Deep drilling cycle), there is no designation of cutting amount (Q). Or, 0 is designated.
P/S581	More than 6 of operation start command M codes are commanded.
P/S582	Not registered program was tried to start.
P/S584	During synchronous control, command was tried to make for B-axis. During operation of B-axis, synchronous control was tried to make.

9-8 Caution and Others

1. Addresses which are usable for operation command are nine kinds of G, B, R, Q, P, F, M, T and U. In addition, for address G, command of G code of ten kinds other than G00 is possible.
   When other addresses (X, Z, ...) and G code are commanded, an error does not occur and they are neglected.

2. At the time of registration start of operation command (G101), modal data is as follows ;

   G function .... G00 (Positioning)
   G80 (Fixed cycle cancel)
   G99 (Feed per revolution)

   F • P • Q 0

3. Single block can be used for B axis too,

a. Display of "cycle start lamp" of B axis is only turned on during its operation and turned off after completion of operation. (B axis, etc. operate.)
b. Display of "cycle start lamp" when both main (X, Z axes) and B axis operate is turned on when either axis operates and only turned off when all axes stop.
c. Single block cannot be released during operration of B axis.
   When single block is used and single block is released (OFF) during operation of B axis, B axis stops at the position after completing the operating program.
   Since B axis stops at the position of completed block and only X axis and Z axis implement continuously when this action is taken when B axis operates together with X- and Z-axes, there is a possibility of interference.
   Never make "single block" OFF during operation of B axis.

4. Pay attention since the following operation can not be performed by this function, (Only B-axis)

   a. MDI operation

   Incidentally, above-mentioned matter applies only the B-axis, and it is possible for the fundamental axis (X-, Z-axis)

5. The following item are slightly different from the fundamental axis (X-, Z-axis) which can be operated by this function.

   a. Feed per revolution

      Feed per revolution of this function differs from fundamental axes (X-, Z-axis) feed speed F is obtained by calculating from spindle rotation at the time of start of each cutting block.
      For this reason, if spindle rotation speed is changed during movement of cutting feed block, feed speed of the block does not change.
      When spindle rotation is changed or a spindle is started from zero speed.
      Sometime require for the timer (G04) before operation instruction of B-axis is made.

Example) When the spindle is started from spindle speed zero.

```
G97 S1000 M3 ;
G101 ;
G00 B___ ;
G01 B___ ;
  :
G100 ;
```

Since spindle rotation is zero when operation instruction of B-axis is registered, proper feed speed is not given.

⇓

```
G97 S1000 M3 ;
G101 ;
G04 U0.5 ;
G00 B___ ;
G01 B___ ;
  :
G100 ;
```

Feed speed of B-axis moves at a proper value.

Refer to "ACTF" of the program check screen of B-axis for properness of feed speed F of B-axis.

10. STANDARD PROGRAM STYLE FOR BND-20S, BND-34S

10-1. Programming of Bar Work

In case of M12 (Bar Push out)

```
O___          ;
N□01 G20 (G21) ;
N□02 G50  S___ ;
N□03      T□□00;
N□04 G00  X0 Z___  T□□□□ ; <---------- Bar stopper
N□05      M12 ;  <------------------ Bar feed operation
N□06      Z___ ;


N◇◇1      T◇◇00 M91 ; <------------- Main encoder ON
N◇◇2      S___  M03 ; <---- Main spindle normal rotation start
N◇◇3 G00  X___  Z___  T◇◇◇◇ M8 ; <---- Coolant ON
          .
          .
          .
          .
          .
NΔ01      TΔΔ00 M91 ; <------------- Main encoder ON
NΔ02 G00  X___ Z___ S___ TΔΔΔΔ M87 ; <--Main and sub spindle normal
NΔ03 G101 ; <-------------------┐         rotation start
NΔ04 G00 B-___ ;                |
NΔ05      M190 ;                | B axis operation command
NΔ06 G100 ; <-------------------┘
NΔ07      M140 ; <---B axis operation start command } Independent
NΔ08      M190 ; <---Waiting command                } operation
NΔ09      M118 ; <---Sub spindle chuck close          command of B axis
NΔ10 G01  X0 F___ ; <------------- Cutting off
NΔ11 G00 X___ ; <----------------- Cut off tool retract
NΔ12 G101 ; <--------------------┐
NΔ13 G00 B-___ ;                 |
NΔ14      M190 ;                 | B axis rearward
NΔ15 G100 ; <--------------------┘
NΔ16      M140 ; <--- B axis operation start command } Independent
NΔ17      M190 ; <---Waiting command                 } operation
                                                       command of B axis

N∇∇01     T∇∇00 ; <--------------- Subspindle side operation
N∇∇02 G00 X___ Z___ S___ T∇∇∇∇ ;
          |
          |
N∇∇15     X___ M09 ; <------------- Coolant OFF
N∇∇16 G101 ; <----┐
N∇∇17 G00 B0 ;    | B axis zero point return
N∇∇18     M190 ;  |
N∇∇19 G100 ; <----┘
N∇∇20     M140 ; <--B axis operation start command } Independent operation
N∇∇21     M190 ; <--Waiting command                } command of B axis
N∇∇22     M205 ; <------------------- Main and subspindle stop
N∇∇23     M30 ; <-------------------- End of program
```

BND-S PRO 0-TC 90/8

10-2. Programming without Nose R-comp

```
        :         }
        :         }
        :         } See 10-1. Programming of Bar Work
        :         }
        :         }

N□15 G00  X____ Z____ M09 ;
N□16      M01 ;

NΔ01      TΔΔ00 ;
NΔ02 G97  S____ M03 ;  <---------- When constant cutting speed control
                                   is not used
NΔ03 G00  X____ Z____ TΔΔΔΔ M08 ;
NΔ04 G01  X____ Z____ F____ ;
        |         }
        |         }
        |         } See 10-1 Programming of Bar Work
        |         }
        |         }
```

BND-S PRO 0-TC 90/8

10-3. Programming with Nose R-Comp

```
        :
        :                          } See 10-1. Programming of Bar Work
        :

N□06 G41 or G42   G00 X____ Z____T□□□□;
N□07 G01              X____ Z____F____;
        :
        :
N□17 G40   G00  X____ Z____M09 ;
N□18            M01;
        :
        :                          } See 10-1. Programming of Bar Work
        :
```

BND-S PRO 0-TC 90/8

11. LIST OF TAPE FORMAT FOR BND-20S, BND-34S

11-1. Preparatory Function (G Function)

G code	Function	Group
G00	Positioning	01 *
G01	Linear Interpolation and Cornering	01
G02	Circular Interpolation, Clockwise	01
G03	Circular Interpolation, Counterclockwise	01
G04	Dwell	00
G10	Program Input of Offset Value (Optional)	00
G20	Inch Input	06
G21	Metric Input	06
G27	Reference point Return Check	00
G28	Automatic return to reference point	00
G32	Thread Cutting	01
G40	Tool Nose Radius Compensation Cancel	07 *
G41	Tool Nose Radius Compensation Left	07
G42	Tool Nose Radius Compensation Right	07
G50	Maximum Spindle Speed Setting (RPM)	00

B axis operation command code

G code	Function
G100	Registration end of B axis operation command
G101	Registration start of B axis operation command

Multiple Repetitive Cutting Cycle

G70	Finish Cutting Cycle	00
G71	Stock Removal Cutting Cycle in Turning	00
G72	Stock Removal Cutting Cycle in Facing	00
G73	Pattern Repeated Cutting Cycle	00
G74	Longitudinal Pecking Cycle	00
G75	Cross Pecking Cycle	00
G76	Thread Cutting Cycle	00

Canned Cycle

G90	Turning Cycle	01
G92	Thread Cutting	01
G94	Facing Cycle	01

G96	Constant Surface Speed Control	02
G97	Constant Surface Speed Control Cancel	02 *
G98	Feed Rate Per Minute	05
G99	Feed Rate Per Reyolution	05 *

(NOTE 1)
The make '' in the list of G-codes shows the initial state when the power is turned on.*

(NOTE 2)
The G codes in the group 00 are not modal. They are effective only in the block in which they are specified.

(NOTE 3)
An alarm occurs when a G code not listed in the above table is specified (No. 010).

(NOTE 4)
A number of G codes can be specified in a block even if they do not belong to the same group. When a number of G codes of the same group are specified, the G code specified last is effective.

11-2. Miscellaneous Function (M Function)

M code	Function
M00	Program Stop
M01	Optional Stop
M02	End of Program (Rewind reset)
M03	Spindle Forward (Counter-clockwise)
M04	Spindle Reverse (Clockwise)
M05	Spindle Stop
M08	Coolant ON
M09	Coolant OFF
M10	Splash Guard Open (Optional)
M11	Splash Guard Close (Optional)
M12	Bar Feeding
M13	Main Spindle Forward Orientation Start (Optional)
M15	Turret Indexing, Clockwise
M16	Turret Indexing, Counter-clockwise
M17	Main Chuck Open
M18	Main Chuck Close
M19	Main Spindle Reverse Rotation Orientation Start (Optional)
M23	Pull out Threading ON
M24	Pull out Threading OFF
M25	Counter (Optional)
M28	High Pressure Coolant ON (Optional)
M29	High Pressure Coolant OFF (Optional)
M30	Program Rewinding and Stop (Continuous running)
M33	Main Spindle Forward & High Pressure Coolant ON (Optional)
M34	Main Spindle Reverse & High Pressure Coolant ON (Optional)
M35	Part Catcher Forward (Optional)
M36	Part Catcher Backward (Optional)
M43	Revolving Tool Normal Rotation Start (Optional)
M44	Revolving Tool Reverse Rotation Start (Optional)
M45	Revolving Tool Stop
M51	Error Detection OFF
M52	Error Detection ON
M76	Main Air Blow No.1 ON (Optional)
M77	Main Air Blow No.2 ON (Optional)
M79	Main Air Blow Stop (Optional)
M83	Material Feeder Start (Optional)
M84	Block Delete OFF Reset (Optional)
M91	Main encoder ON
M93	Simultaneous start of spindle forward rotation and coolant
M94	Simultaneous start of spindle reverse and coolant
M95	Spindle, coolant stop
M98	Subprogram call
M99	End of sub-program
M103	Subspindle Forward Rotation
M104	Subspindle Reverse Rotation
M105	Subspindle stop
M113	Sub spindle normal rotation orientation start (Optional)
M117	Subspindle chuck, open
M118	Subspindle chuck, close
M119	Sub spindle Reverse Rotation Orientation Start (Optional)
M140	B axis Operation Start Command 1 (Optional)
M141	B axis Operation Start Command 2 (Optional)
M142	B axis Operation Start Command 3 (Optional)
M157	Sub work ejector forward (Air cylinder type)
M158	Sub work ejector backward (Air cylinder type)
M159	Sub work ejector automatic operation mode (Air cylinder type)
M176	Sub air blow No.4 ON (For sub spindle)(Optional)
M177	Sub air blow No.5 ON (Optional)

M code	Function
M179	Sub air blow stop (Optional)
M190	Waiting Command
M191	Sub encoder ON
M193	Simultaneous start of sub spindle forward rotation and coolant
M194	Simultaneous start of sub spindle reverse rotation and coolant
M195	Sub spindle coolant stop
M203	Main and Sub Spindle Forward Synchronous Rotation
M204	Main and Sub Spindle Reverse Synchronous Rotation
M205	Main and Sub Spindle Synchronous Stop
M206	Synchronous operation release
M213	Part Conveyor ON (Optional)
M215	Part Conveyor OFF (Optional)
M240	Drill failure detection (Optional)
M272	Cutting off confirmation forward movement (Optional)
M273	Cutting off confirmation backward movement (Optional)
M274	Parts separator open (Optional)
M275	Parts separator close (Optional)

11-3. Tool Function (T Function)

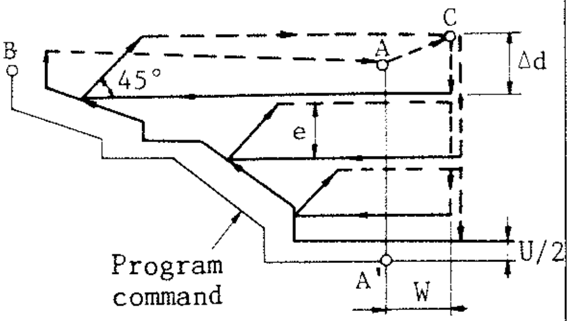
T □□□□

- Wear Offset Number (01 - 32, 00: Offset Cancel)
- Tool number/geometry offset number (01 - 32)
- Specifying direction of nose radius compensation

11-4. Spindle Function (S Function)

S 3-Digits in G96 Mode ----- Specify the Cutting Speed
S 4-Digits in G97 Mode ----- Specify the Spindle Speed

11-5. Multiple Repetitive Cycles

<table>
<tr><th>Code</th><th>Cutting Cycle</th><th>Method of Programming</th></tr>
<tr><td>G70</td><td>Finish Cutting of N(ns) to N(nf)</td><td>G70 P ns Q nf ;</td></tr>
<tr><td>G71</td><td>

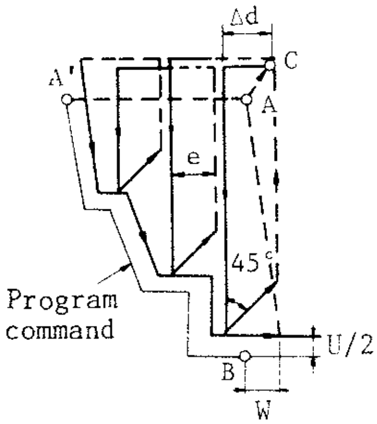
N(ns) to N(nf)</td><td>G71 U Δd R e ;

G71 P(ns) Q(nf) U±___W±___
F___S___ ;

N(ns) ______ ;
F___ ;
S___ ;
M___ ;

N(nf) ______ ;

Program of finished shape
(A → A' → B)

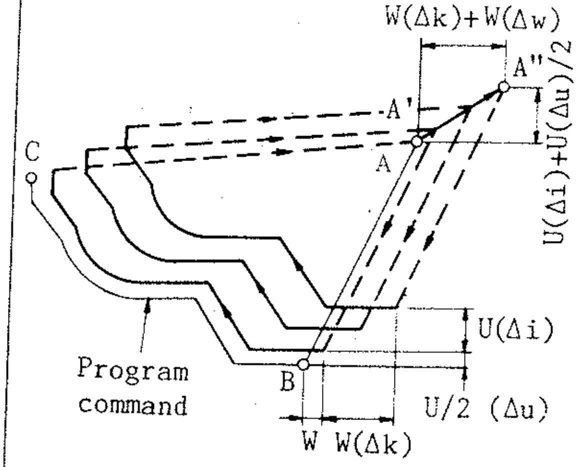
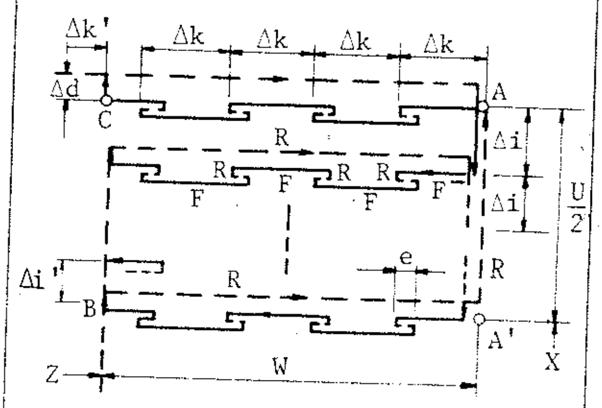
U : Depth of cut
R : Relieving amount
P,Q : Sequence number for cycle start and finish
U,W : Finishing allowance in X and Z direction
(Specified by diameter)</td></tr>
<tr><td>G72</td><td>
</td><td>G72 W Δd R e ;

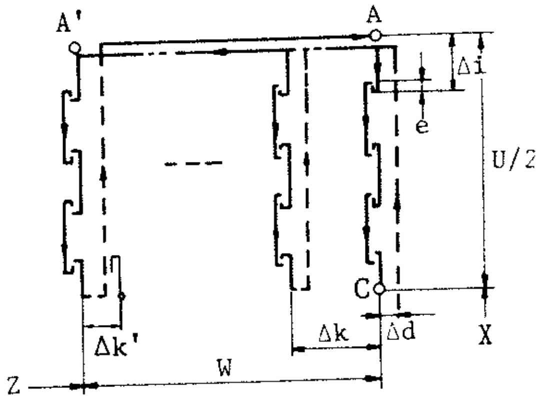
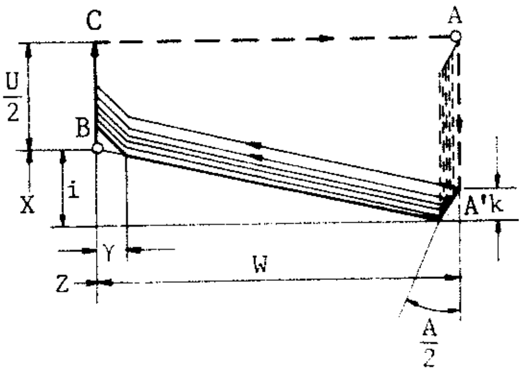
G72 P(ns) Q(nf) U±___W±___
F___S___ ;

N(ns) ______ ;
F___ ;
S___ ;
M___ ;
N(nf) ______ ;

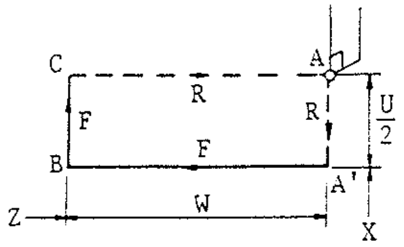
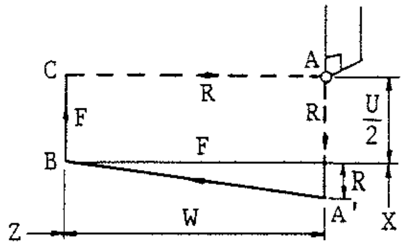
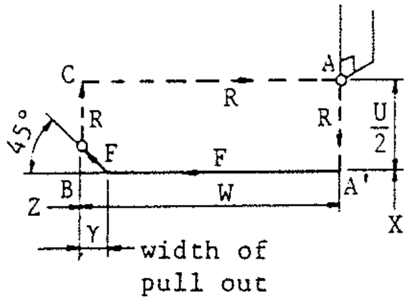
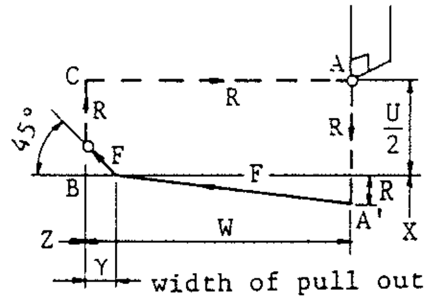
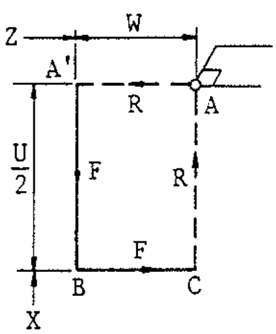
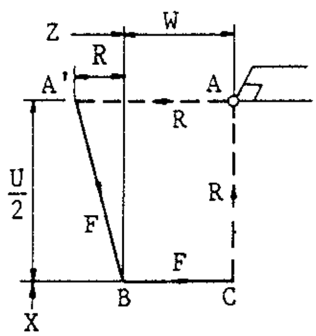
Program of finished shape
(A → A' → B)

W : Depth of cut
R : Relieving amount
P,Q : Sequence number for cycle start and finish
U,W : Finishing allowance in X and Z direction
(Specified by diameter)</td></tr>
</table>

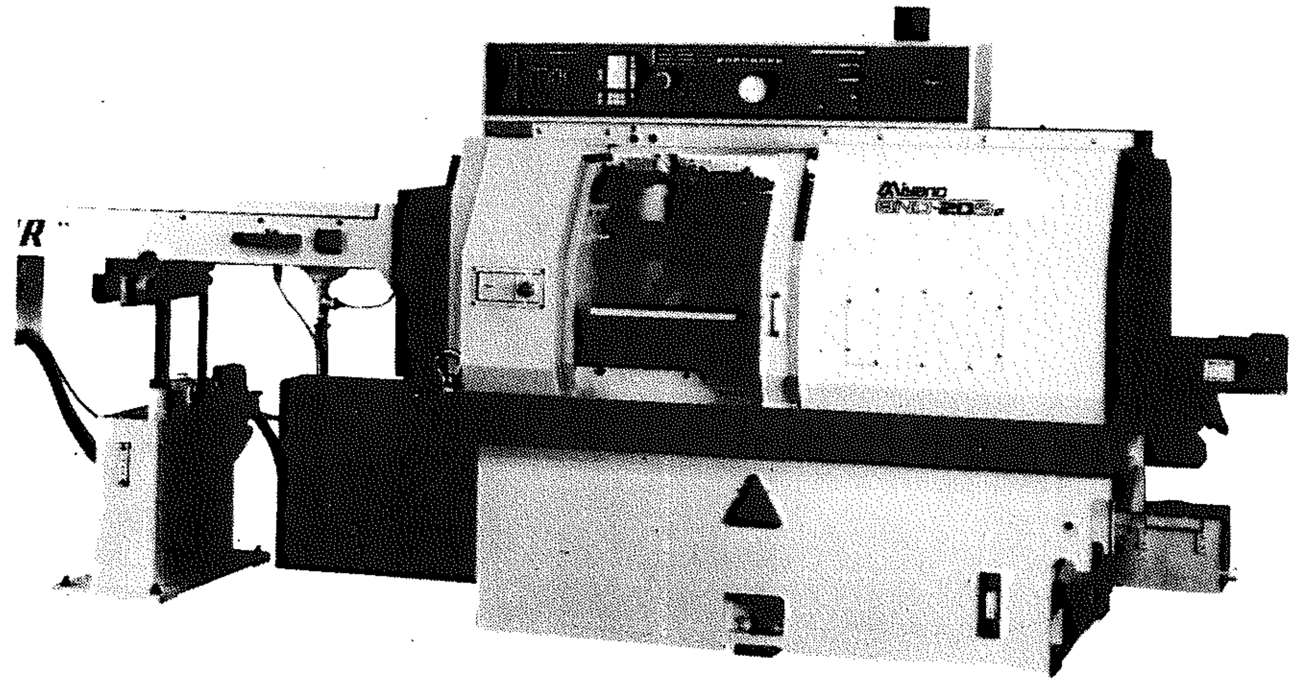
Code	Cutting Cycle	Method of Programming
G73		G73 U±$\underline{\Delta i}$ W±$\underline{\Delta k}$ R ___ ; G73 P$\underline{ns}$ Q$\underline{nf}$ U± ___ W± ___ F ___ S ___ ; N(ns) ______ ; F______ ; S______ ; M______ ; N(nf) ______ ; Program of finished shape (A → B → C) U(Δi) : Total depth of rough cut in X-axis W(Δk) : Total depth of rough cut in Z-axis R : Number of passes for rough cutting P,Q : Sequence number for cycle start and finish U,W : Finishing allowance in X and Z direction (Specified by diameter)
G74		G74 R $\underline{e}$; G74 X(U)± ___ Z(W)± ___ P$\underline{\Delta i}$ Q$\underline{\Delta k}$ R±$\underline{\Delta d}$ F ___ ; R : Relieving value X(U) : Absolute position (B) in X-axis or the distance between position A and B Z(W) : Absolute position (B) in Z-axis or the distance between position A and B PΔi : Movement amount in X-axis for a pass QΔk : Movement amount in Z-axis for a pass RΔd : The amount of relief

Code	Cutting Cycle	Method of Programming
G75		G75 R e ; G75 X(U)± __ Z(W)± __ PΔi QΔk RΔd F __ ; R : Relieving value X(U) : Absolute position (B) in X-axis or the distance between A' and B Z(W) : Absolute position (B) in Z-axis or the distance between the position C and B PΔi : Movement amount in X-axis for a pass QΔk : Movement amount in Z-axis for a pass RΔd : The amount of relief
G76		G76 Pmra QΔdmin Rd ; G76 X(U)± __ Z(W)± __ R±i Pk QΔd F __ ; Pm : Repetitive times in finishing cut Pr : Width of thread pull out Pa : Angle of the thread QΔdmin : Minimum depth of cut Rd : Fixed amount X(U): Absolute position (B) in X-axis or distance between the position A and B Z(W): Absolute position (B) in Z-axis or distance between the position A and B Ri : Difference of radius value between start and end position of the thread at the final pass Pk : Height of the thread QΔd : Depth of the 1st pass

11-6. Canned Cycle

<table>
<tr><th>Code</th><th>Straight Cutting Cycle</th><th>Taper Cutting Cycle</th></tr>
<tr><td>G90</td><td>G90 X(U)------ Z(W)------ F------ ;

</td><td>G90 X(U)-----Z(W)-----R-----F----- ;

</td></tr>
<tr><td>G92</td><td>G92X(U)-----Z(W)-----F----- ;

</td><td>G92 X(U)-----Z(W)-----R-----F----- ;

</td></tr>
<tr><td>G94</td><td>G94 X(U)-----Z(W)-----F----- ;

</td><td>G94 X(U)-----Z(W)-----R-----F----- ;

</td></tr>
</table>

BND – 20S$_2$
BND – 34S$_2$



Miyano

PROGRAMMING

OCT. 1989
EDITION 3-0

CONTENTS

1. THE OUTLINE OF THE PROGRAMMING 1
1-1. Regular Procedure for Making Program 1
1-2. The Motion of the Turret Slide 2
1-3. The Cutting Area 2
1-4. Part Dimensioning 5
1-5. Part Layout Sheet 6
1-6. Process Sheet 7

2. TAPE INFORMATION 8
2-1. Move Command for the Cutting Tools 10
2-2. The Absolute and the Incremental Input 11
2-3. The Program Format 13
2-4. G-Function (Preparatory Function) 17
[1] G50 (Maximum Spindle Speed Setting) 17
[2] G00 (Positioning) 18
[3] G01 (Linear Interpolation, 45° Chamfering, Rounding) 19
[4] G02, G03 (Circular Interpolation, Radius Programming for Circular Interpolation) ... 25
[5] G04 (Dwell) 30
[6] G10 (Changing of tool offset amount) 32
[7] G20,G21 (Inch/Metric Input) 33
[8] G27 (Reference point Return Check) 34
[9] G28 (Automatic return to reference point) 35
[10] G32 (Thread Cutting) 36
[11] Other G-codes 49
[12] List of G-codes and Their Group 50
2-5. F-Function (Feed Function) 51
[1] Feed rate per revolution of the spindle (G99) 51
[2] Feed rate per minute (G98) 52
2-6. S-Function (Spindle Speed Function) 54
[1] Spindle RPM direct command (G97 mode) 54
[2] Cutting speed command (G96 mode) 56
2-7. Coordinate System Setting 57
2-8. T-Function (Tool Function) 60
[1] Tool number/geometry offset number specification 60
[2] Tool offset (General) 61
[3] Tool position offset (Wear offset) 62
[4] Notes of T-function 66
[5] Tool nose radius compensation 66
2-9. M-Function (Miscellaneous Function) 67
2-10. Other Function 78

3. EXAMPLE OF PROGRAMMING OF VARIOUS OPERATIONS 79
3-1. Facing Operation 79
3-2. Turning Operation 81
3-3. Drilling Operation 82
3-4. Boring Operation 83
3-5. Grooving Operation 84
3-6. Taper Cutting and Chamfering Operation 85
3-7. Radius Cutting Operation 86
3-8. Cutting off Operation 88
3-9. Bar Feedout Operation 90
3-10. Threading Operation 91

4. TOOL NOSE RADIUS COMPENSATION (G40 to G42) 93
4-1. Different Dimension by Tool Nose Radius 93
4-2. The motion of tool nose radius compensation 95
4-3. Command of Tool Nose Radius Compensation 96
4-4. Prohibited command on the compensation 100

5. EXAMPLE OF PROGRAMMING 104
5-1. Part Dimensioning 104
5-2. Part Layout Sheet 105
5-3. Work Coordinate System Setting and Tool Selecting Position 106
5-4. Process Sheet 107

6. CANNED CYCLE 111
6-1. G90 (Turning Cycle) 111
6-2. G92 (Thread Cutting Cycle) 116
6-3. G94 (Facing Cycle) 122

7. MULTIPLE REPETITIVE CUTTING CYCLES 126
7-1. G71 (Stock Removal Cutting Cycle in Turning) 126
7-2. G72 (Stock Removal Cutting Cycle in Facing) 131
7-3. G73 (Pattern Repeated Cutting Cycle) 136
7-4. G70 (Finish Cutting Cycle) 140
7-5. G74 (Longitudinal Pecking Cycle) 141
7-6. G75 (Cross Pecking Cycle) 144
7-7. G76 (Thread Cutting Cycle) 147
7-8. Notes on Multiple Repetitive Cycles (G70 to G76) 153

8. DRAWING DIMENSION DIRECT INPUT (OPTION) 154
[1] Instruction method 155
[2] Commanding of angle, arc and chamfer 156
[3] Commands table 157

9. CONTROL OF B AXIS (CONTROL OF SUB SPINDLE) 161
9-1. Outline of Function 161
9-2. Programming 162
9-3. Detail of Operation Instruction 165
9-4. Offset 167
9-5. Independent Movement and JOG Operation of B axis 171
9-6. Data Memory 173
9-7. Alarm 174
9-8. Caution and Others 175

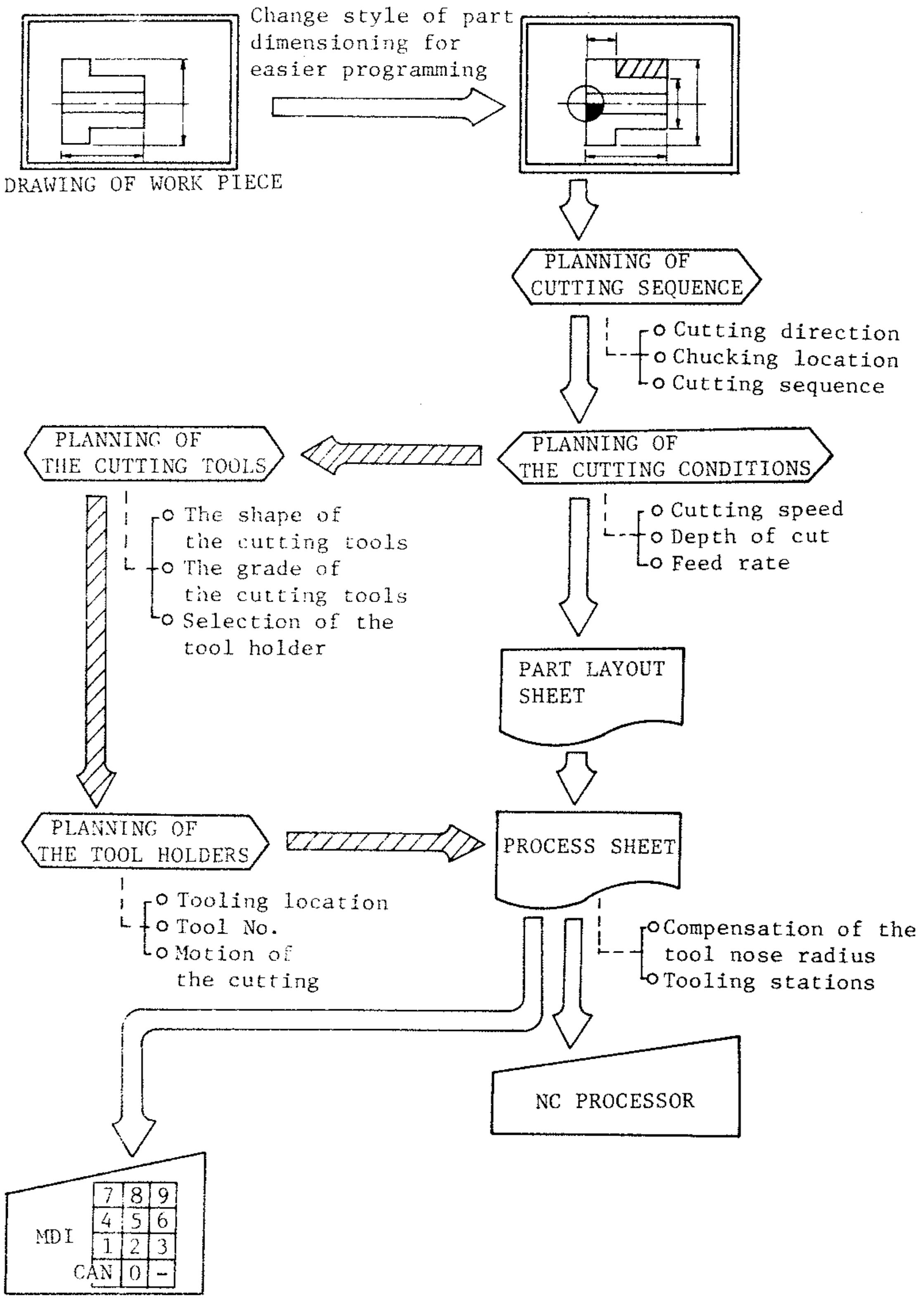
10. STANDARD PROGRAM STYLE FOR BND-20S, BND-34S 177
10-1. Programming of Bar Work 177
10-2. Programming without Nose R-comp 178
10-3. Programming with Nose R-comp 179

11. LIST OF TAPE FORMAT FOR BND-20S, BND-34S 180
11-1. Preparatory Function (G Function) 180
11-2. Miscellaneous Function (M Function) 182
11-3. Tool Function (T Function) 183
11-4. Spindle Function (S Function) 183
11-5. Multiple Repetitive Cycles 184
11-6. Canned Cycle 187

1. THE OUTLINE OF THE PROGRAMMING

BND-20S, BND-34S are designed for bar and chucking works. The following instructions are the guide for making correct programming sheets and NC program for various types of jobs.

1-1. Regular Procedure for Making Program



BND-S PRO 0-TC 90/8

1-2. The Motion of the Turret Slide

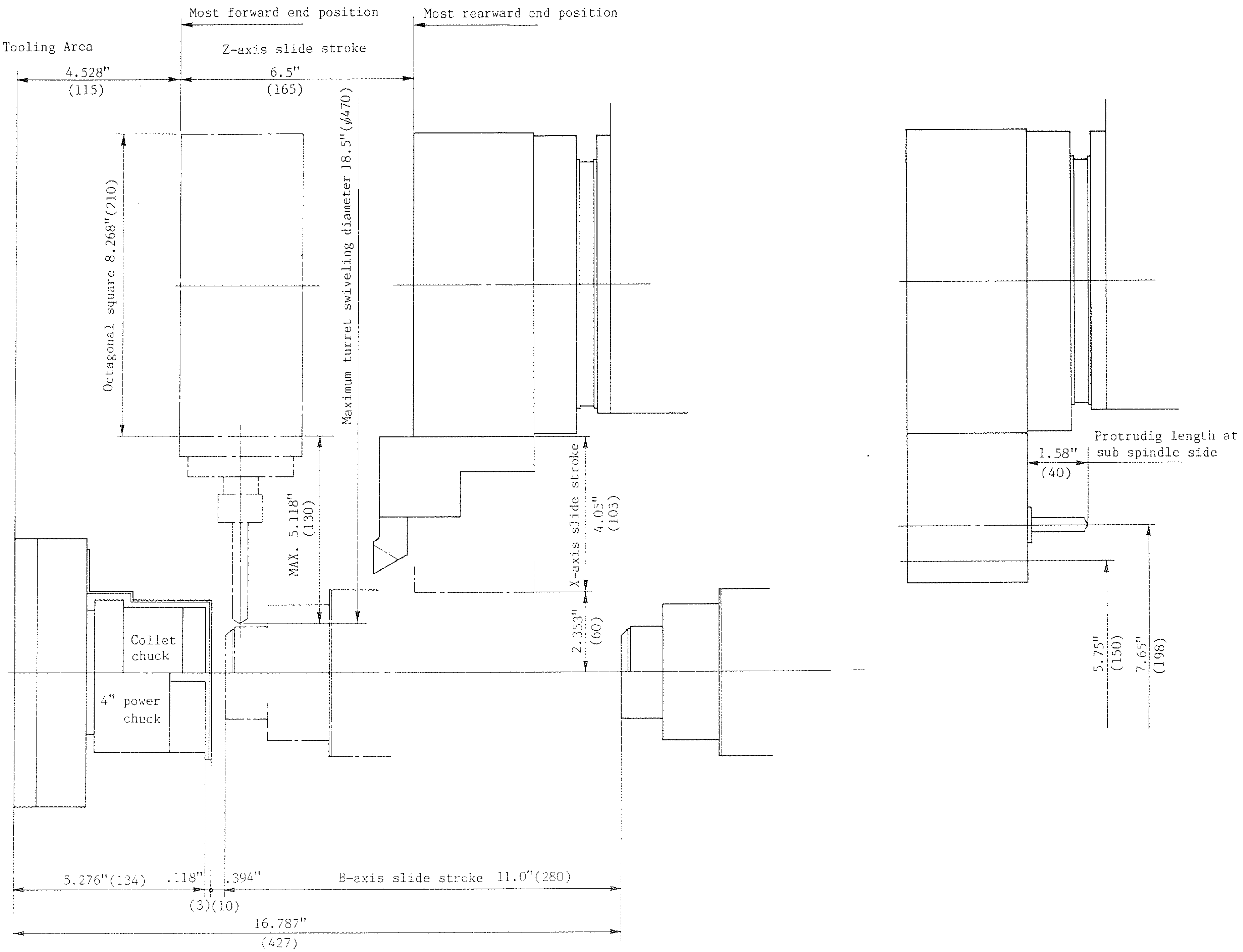
The machine determines the X-axis of the cross slide and the Z-axis of the longitudinal slide, and the direction of the movement of each axis is determined by the sign "+" (Plus) or "-" (Minus) as follows:

- o +X ------- The main turret slide moves away from the center line of the spindle.
- o -X ------- The main turret slide moves towards the center line of the spindle.
- o +Z ------- The main turret slide moves away from the headstock.
- o -Z ------- The main turret slide moves toward the headstock.
- o +B ------- The sub spindle slide moves away from the headstock.
- o -B ------- The sub spindle slide moves toward the headstock.

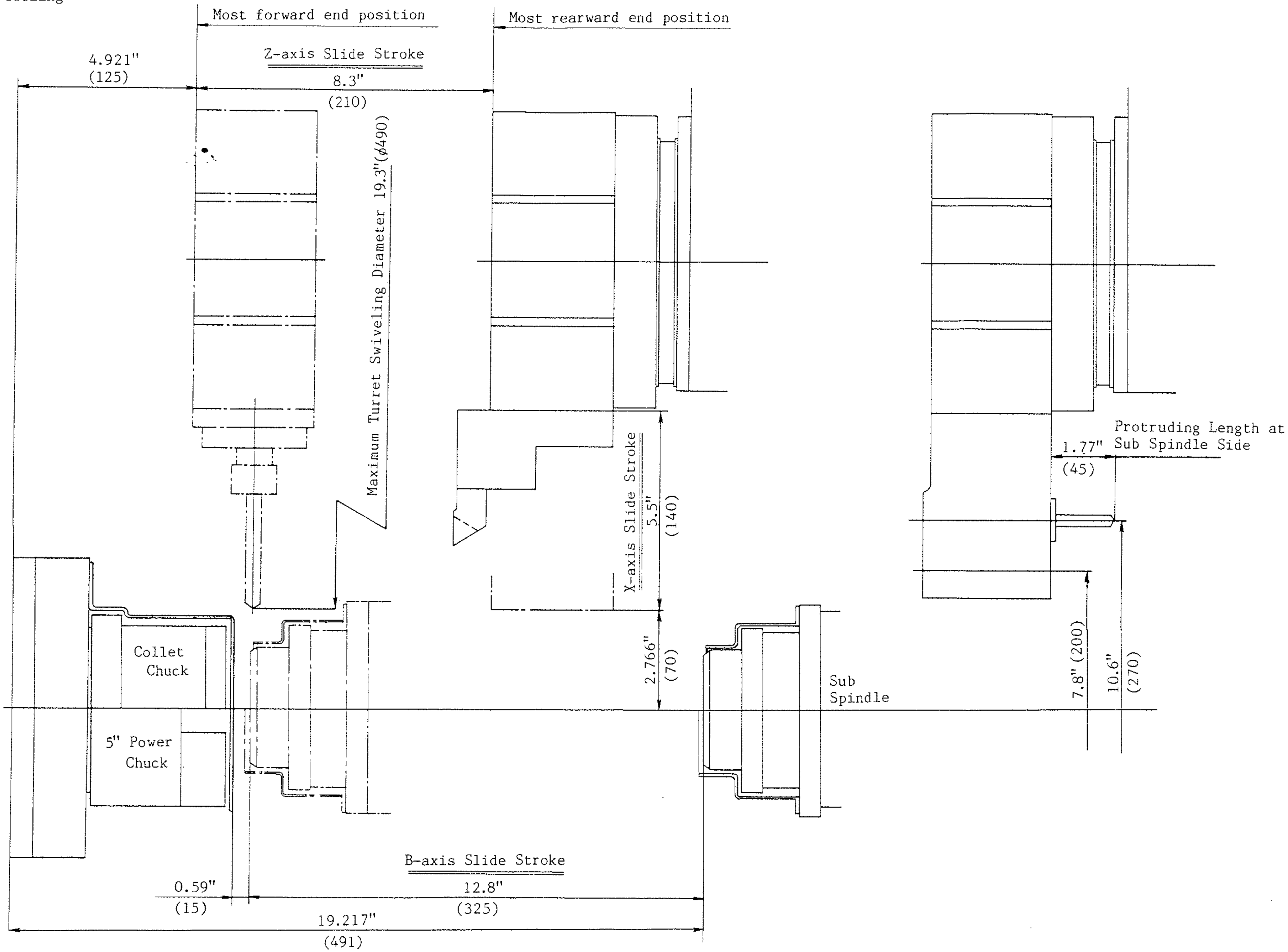
1-3. The Cutting Area

The following schematic sketch shows the cutting area which is programmable on the machine.

BND-20S Tooling Area



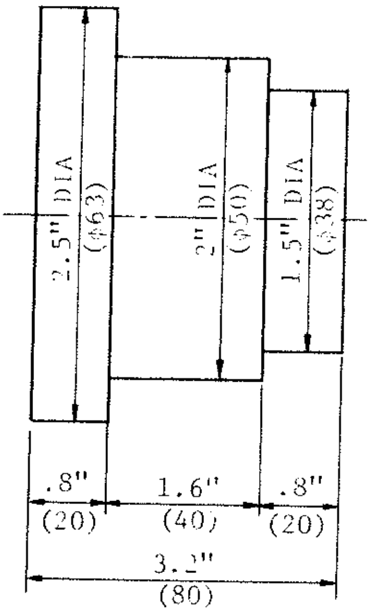
• BND-34S Tooling Area



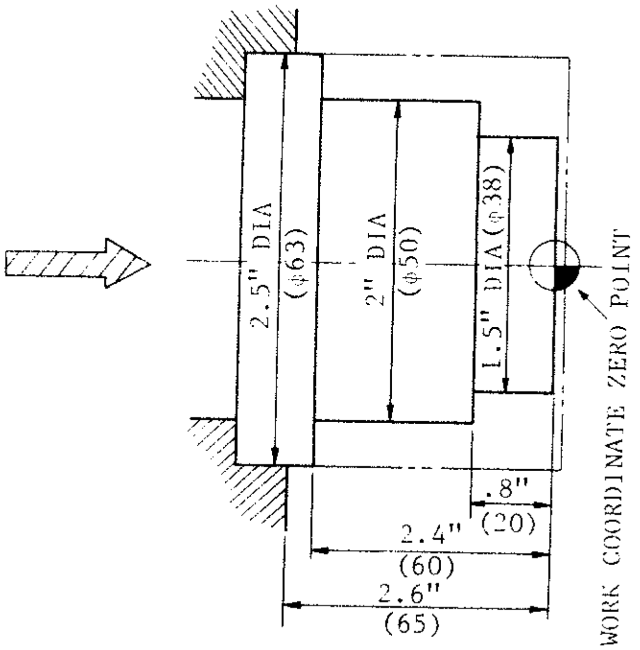
1-4. Part Dimensioning

The NC machine requires a drawing of a part which has dimensions showing the work coordinate zero point, for easier programming.

<u>Example of part dimensioning</u>



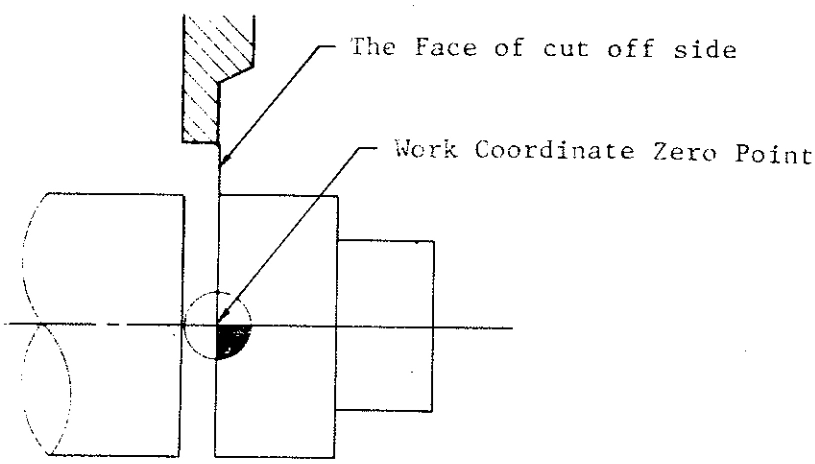
Original Drawing



Drawing of the part dimensioning showing the work coordinate zero point

(NOTE)

The coordinate zero point (Work coordinate zero point) can be determined to be any place such as the face of the chuck, the face of the part etc. However, the face of the workpiece on chucking work and the cut-off face on bar work are convenient.



1-5. Part Layout Sheet

The part planning sheet is used for planning the sequence of the cutting cycle, the cutting conditions, the cutting tools, the tooling set up and the cycle time etc.

(1) Planning of cutting sequence

- o The direction of the machining cycle
  Consider the next operation and the chucking location.
- o The sequence of the cutting
  Consider faster cycle time and the direction of burr etc.
- o The chucking location
  The chucking location should be as wide as possible to get better cutting results.

(2) The cutting conditions

- o The proper cutting speed
- o The proper depth of cut
- o The proper feed rate

Refer them to machining data books etc.

(3) The cutting tools

- o The shape of the tools
- o The grade of the tools
- o Select the cutting tool holder

(Note)
Our standard tool holder is designed to use BND-20S 5/8" x 5/8" (16mm x 16mm) or 3/4" x 3/4" (20mm x 20mm), BND-34S 3/4" x 3/4" (20mm x 20mm) cutting tools.

(4) In order to get better chip removal

Program the pecking motion or use as oil hole drill with the high pressure coolant system on the drilling operation.

(5) Determine the setting tool length

1-6. Process Sheet

The process sheet is written to control the slide motion and cutting conditions by using the NC data as applied to the part layout sheet. If this data is transferred to NC unit through RS232C interface or MDI unit, the machine will be operated in this sequence

Command of the sequence number
Command of the slide motion
Command the position of the cross slide (X-axis)
Command the position of the longitudinal slide (Z-axis)
Command the arc center distance when circular interpolation is used.
Command the feed rate
Command the spindle speed or cutting speed
Command the selection of the tool station
Command the Miscellaneous function such as the spindle ON-OFF etc.
Command the end of block (EOB)
This is called <u>one block</u>

PROCESS SHEET

N	G	X(U)	Z(W)	I	K	F	S	T	M			
N 1	G21									;		
N 2	G50						S3500			;		
N 3								T100		;		
N 4	G97						S1200		M3	;		
N 5	G00	X 0	Z100.					T101		;		
N 6			Z 64.							;		
N 7	G01		Z 58.5			F1.5				;		
N 8	G00		Z120.							;		
N 9								T200		;		
N10	G00	X 0	Z100.				S2000	T202		;		
N11			Z 62.							;		
N12	G01		Z 52.			F1.5				;		

Showing the slide motion in this block

(NOTE)
The Make ';' shows for CR(End of Block).
This mark also displays on the CRT display unit.

2. TAPE INFORMATION

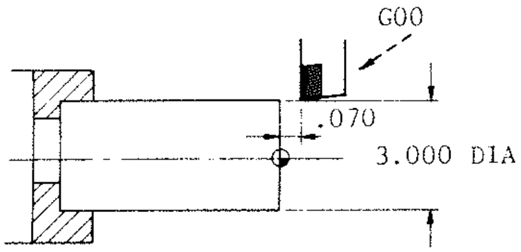
Decimal Point Programming

The machine can be used the decimal point for the following addresses.

X, Z, U, W, R, C, A, B, I, K, and F

The position of the decimal point means inch (mm) or second.

[1] Dimension



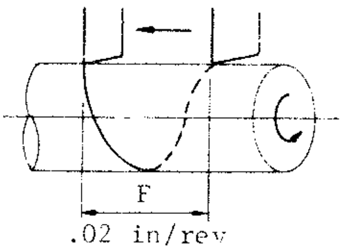
Decimal point programming	G00 X3.0 Z0.07;
Integral number programming	G00 X30000 Z700;
Mixed programming	G00 X3.0 Z700;

(Note 1)
Decimal number and integral number can be mixed in one block.
(Note 2)
The decimal point makes a big different meaning of dimension as follows ;

X2.0 (or X2.) ⟶ 2 inches
X2 ⟶ .0002 inch

(Note 3)
When a value beyond the minimum set unit is disregarded, a value beyond the minimum set unit is disregarded. In addition, the digit number more than the maximum digit cannot be disregarded.

[2] Feed



Decimal point programming
G01 X____ Z____ F0.02;

Integral number programming
G01 X____ Z____ F20000;

(Note 1)
When a value beyond the minimum set unit is disregarded, a value beyond the minimum set unit is disregarded. In addition, the digit number more than the maximum digit cannot be disregarded.

[3] Time

2.5 seconds of dwell time

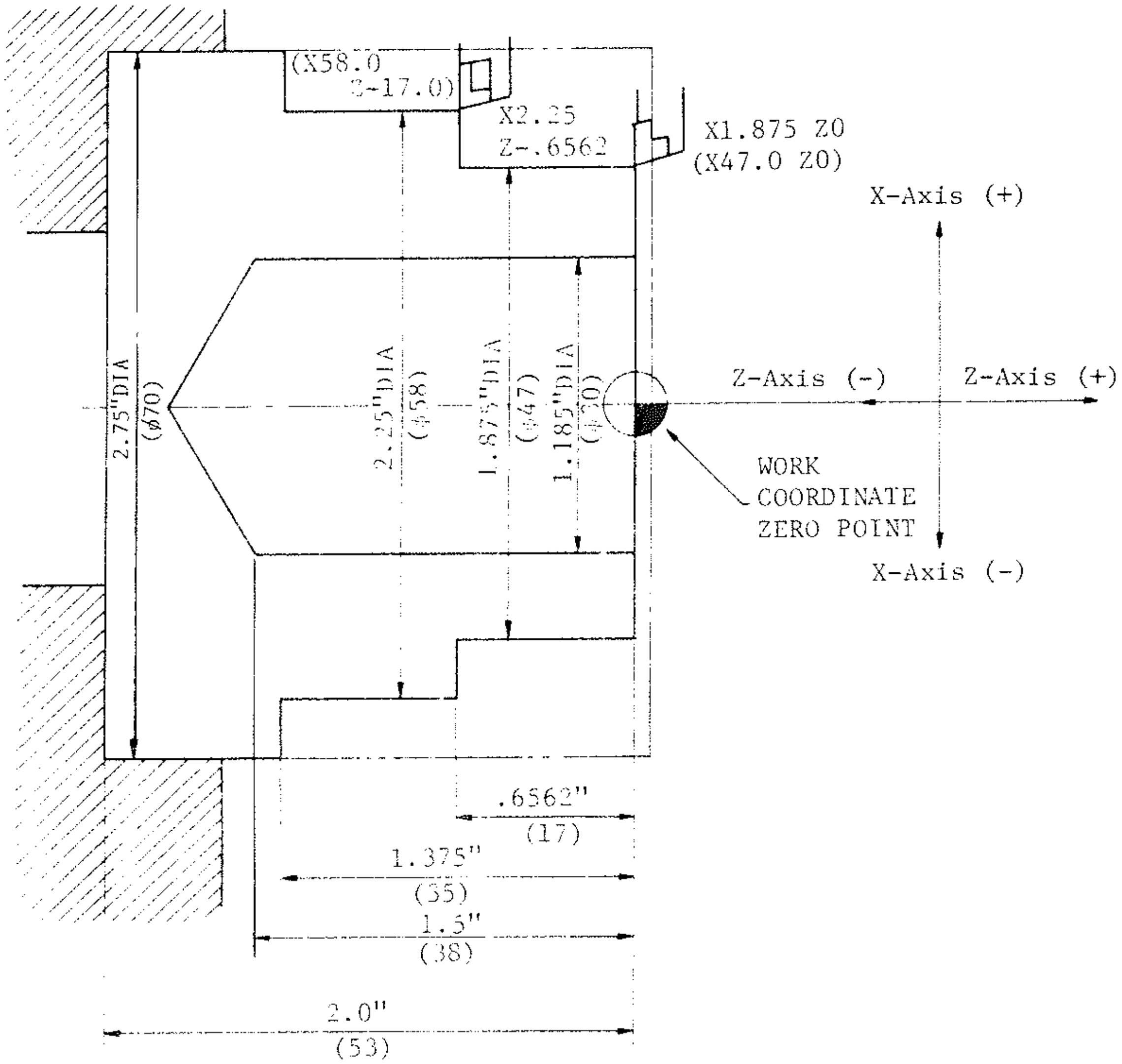
Decimal point programming	G04 X2.5;
	G04 U2.5;
Integral number programming	G04 P2500;

(Note 1)
Decimal point programming can not be used for the address 'P'.

(Note 2)
When a value beyond the minimum set unit is disregarded, a value beyond the minimum set unit is disregarded. In addition, the digit number more than the maximum digit cannot be disregarded.

-5 700.
.57
3000
2.0

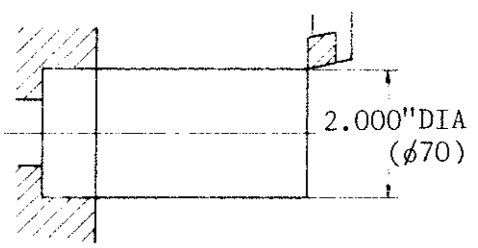
2-1. Move Command for the Cutting Tools



X (Cross Slide Direction)

X .... Command using the dimension of the diameter.

- EXAMPLE -



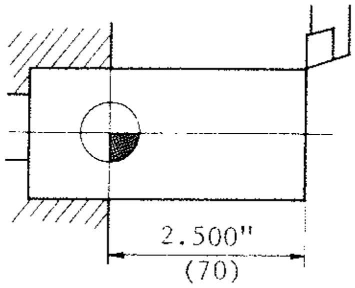
In this example, X command will be shown as X20000 (or X2.0) —— Inch
X70000 (or X70.0) —— Metric

BND-S PRO 0-TC 90/8

Z (Longitudinal Slide Direction) ←--→

Z □□□□□□□ ..... Command the distance from the work coordinate zero point.

- EXAMPLE -



In this example, Z command will be shown as

Z25000 (or Z2.5) —— Inch
Z70000 (or Z70.0) —— Metric

2-2. The Absolute and the Incremental Input

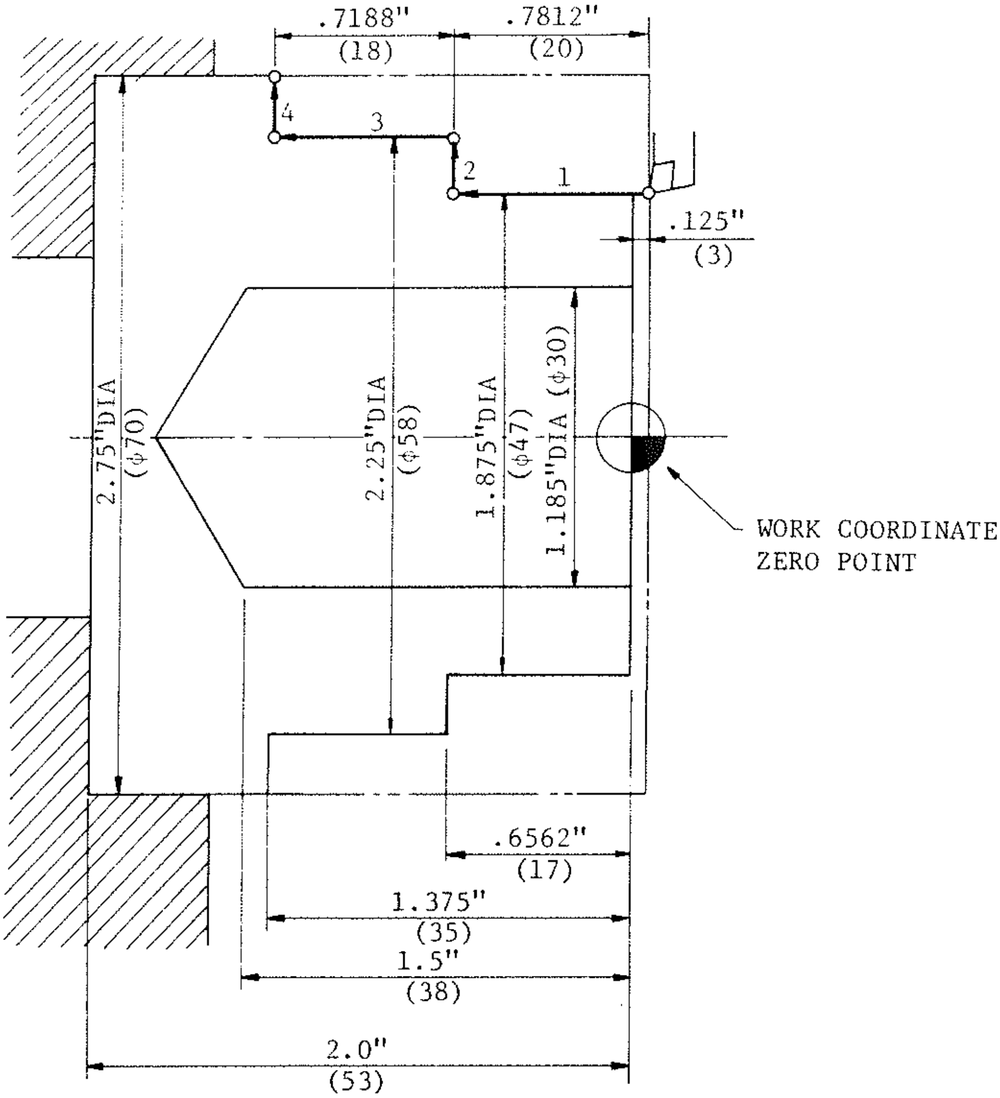
The machine can be used with either the absolute input or the incremental input, and both systems can be mixed in a block. For example, it is possible to specify the absolute input in the X-axis and the incremental input in the Z-axis.
The difference of the absolute and the incremental input is specified by the address. If the address X or Z is used, the absolute value will be shown, and if the address U or W is used, the incremental value will be shown.

(Note)
In order to specify an arc center, the incremental value must always be used in radius designation by using the I, K or R code.

Address	Incremental /Absolute	Dia./Rad. Designation	Meaning
X	Absolute	Diameter	The position of the slide along the X-axis.
Z			The position of the slide along the Z-axis.
B			The position of the slide along the B-axis.
U	Incremental	Diameter	The amount of slide movement along the X-axis.
W			The amount of slide movement along the Z-axis.
I	Incremental	Radius	The distance from the arc starting point to the center of the arc in the X-axis direction.
K			The distance from the arc starting point to the center of the arc in the Z-axis direction.
R	Incremental	Radius	The amount of the arc radius. The amount of cutting depth for X-axis in G90 to G94.

(NOTE)
If the address X and U or Z and W are programmed in the same block, the latter address will be effective.

- Example of Programming with Incremental and Absolute System -



An example of programming using the incremental and the absolute input for the tool path of the above drawing are shown as follows:

	Tool Path	Incremental	Absolute	Incremental and Absolute
Inch		G00 X1.875 Z.125	G00 X1.875 Z.125	G00 X1.875 Z.125
	1	G01 W-.7812	G01 Z-.6562	G01 W-.7812
	2	U.375	X2.25	X2.25
	3	W-.7188	Z-1.375	W-.7188
	4	U.5	X2.75	X2.75
Metric		G00 X47.0 Z3.0	G00 X47.0 Z3.0	G00 X47.0 Z3.0
	1	G01 W-20.0	G01 Z-17.0	G01 W-20.0
	2	U11.0	X58.0	X58.0
	3	W-18.0	Z-35.0	W-18.0
	4	U12.0	X70.0	X70.0

2-3. The Program Format

The following explanations are for the addresses and the instructions for their use.

Address	Function	Format	The Instructions for their Use
O	Program Number Program Name	O4	Designated by a 4-digits number following the address O. This control can store several programs in its memory. The program number can be used to differentiate one program from another. Designated by a 4-digits number following the address O. This control can store several programs in its memory. The program number can be used to differentiate one program from another. In addition, it is possible to add the program name after the program number. The program name can be used for any code which can be used in the NC. O □□□□(△△△------△) Program name (Up to 31 characters are possible) Program number Address Incidentally, although it is possible to register a message from the reader puncher interface, it is not possible to register and edit from the keyboard of MDI and CRT.
N	Block Number (Sequence Number)	N4	Designated by a maximum 4-digit number (Usually 3-digits numbers are used) following the address N. Any numbers can be used. However,sequencial numbers are recommended. The machine can be operated without programming the N-Code, but, the sequence number search can not be done. Sequence Number N | G | X(U) | Z(W) N1 N2 | G00 | X30.0 | Z25.0 N3 | G01

Address	Function	Format	The Instructions for their Use
G	Preparatory Function	G2	Designated by a 2-digits number following the address G. The motions of the slides, etc. can be programmed by using the G codes such as G00, G01 etc. Refer to item 2-4 for explanation of the various G-codes.
F	Feed Function	F1.6 (F3.4)	The feed function directly designates the amount of feed per revolutions of the spindle,using 7-digits numbers following the address F. Refer to item 2-5 for fúrther details.
S	Spindle-speed Function	S4/5	Designated by a 4-digits number following the address S for specifying the amount of spindle speed in r.p.m or surface speed in feet (m) per a minute. Refer to item 2-6 for further details.
T	Tool Function	T4	Designated by a 4 digits number following the address T. The first 2 digits following the address T specify the selection of the turret station, then the following 2 digits specify either the tool offset number or the cancellation of the offset. T 0 4 0 4 Specify tool offset No.4 Specify the turret station No.4 T 0 4 0 0 Specify the cancellation of the offset Refer to item 2-8 for further detail.
M	Miscellaneous Function	M3	Designated by a 3-digits number following the address M for specifying auxiliary function, such as spindle start (M03) or spindle stop (M05) etc. Refer to item 2-9 for explanation of the various M codes.

Address	Function	Format	The Instructions for their Use
P U X	Dwell	X5.3 U5.3 P53	Designated by a 8-digits number following the address P, used with code G04, for specifying the number of seconds of dwell time. (The address U or X can be used instead of the address P for specifying the dwell time.) The address P is also used for specifying the block number when the multiple repetitive cycle or the subroutine function is programmed. Refer to item 2-4 for further details.
I K	The positioning of the Center in Machining of the Arcs.	I4.4 K4.4 (I5.3) (K5.3)	Designated by a 8-digit number following the address I or K for specifying the position of the center of the Arc when the circular interpolation is programmed by using G02 or G03. The address I is used for the positioning of the X-axis and the address K is used for the positioning of the Z-axis. Refer to item 2-4 for further detail.
R	Radius of Arc	R4.4 (R5.3)	Designated by a 8 digit number following the address R for specifying the value of the radius when using G01, G02 or G03.
/	Block Skip	/	When the BLOCK DELETE switch located on operating panel is set to the ON position, the commands of a block containing "/" (Slash) will not be executed. Refer to item 2-10 for further details.
; (CR)	End of Block	;	Designates the end of a block. The address ';' is also used for specifying the end of the label skip when the label skip function is used.
X	The points of movement in the X-axis Dwell	X4.4 (X5.3)	Designated by a 8 digit number following the address X for specifying an absolute coordinate point in the X-axis when using the diameters of the part. The address X is also used for specifying the dwell time.
U	The points of movement in the X-axis Dwell	U4.4 (U5.3)	Designated by a 8 digit number following the address U for specifying an incremental coordinate point in the X-axis when using the diameters of the part. The address U is also used in the multiple repetitive cycle. Refer to the explanation of the multiple repetitive cycle.

Address	Function	Format	The Instructions for their Use
Z	The points of movement in the Z-axis	Z4.4 (Z5.3)	Designated by a 8 digit number following the address Z for specifying an absolute coordinate point in the Z-axis.
W	The points of movement in the Z-axis	W4.4 (W5.3)	Designated by a 8 digit number following the address W for specifying an incremental coordinate point in the Z-axis. The address W is also used in the multiple repetitive cycle. Refer to the explanation of the multiple repetitive cycle.
B	Command of movement of sub-spindle in B-axis direction	B4.4 (B5.3)	Coordinate value for B-axis is commanded with 8 digits following to address B.

2-4. G-Function (Preparatory Function)

The G-code indicates a 2 digit number following the address G for specifying the motion of the slides such as linear or circular. Generally, a command of the G code continues until other G code is specified (Modal state).

[1] G50 (Maximum Spindle Speed Setting)

In G96 (Constant Cutting Speed Control) mode, the spindle speed is speciefied by using the cutting speed. In this case, the spindle speed may go up too fast when the tool is close to the center line of the spindle.
Therefore, the upper limit of the spindle speed must be programmed when G96 is used.

The form of this command is,

```
G50 S____ ;
       └─→ Specify the upper limit of the
           spindle speed in r.p.m.
```

-Example-

G50 S2500 ;
The maximum spindle speed is held at 2500 r.p.m.

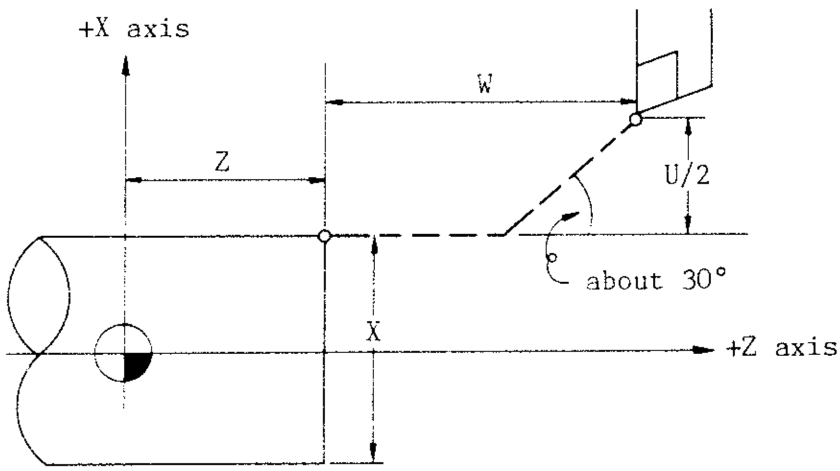
[NOTE]
The maximum spindle speed specified by G50 command is not cleared by reset operation.

[2] G00 (Positioning)

A tool is moved at a rapid traverse rate for each slide direction to the position (X,Z) in the work coordinate system set by the tool offset command (T □□□□), or away by (U, W) from the start position of the tool.

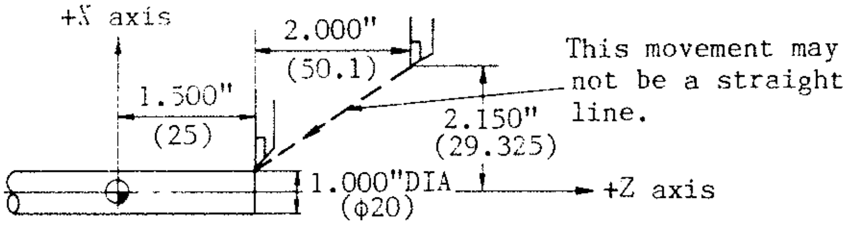
The forms of those commands are as follows;

Absolute system	G00 X______ Z______ ;
Incremental system	G00 U______W______ ;



The rapid traverse rate 472 IPM (12m/min) in the X-axis and 472 IPM (12m/min) in the Z-axis will always be applied for this command. Therefore, the tool path may not be a straight line when both X and Z axis are programmed simultaneously.
And, the acceleration is automatically applied when the slides start to move and the deceleration is automatically applied when the slides are positioning.

- Example -



G00 X1.0 Z1.5 ;   or G00 U-3.3 W-2.0 ; ---- Inch

G00 X20.0 Z25.0 ; or G00 U-38.65 W-50.1 ; --- Metric

[NOTE 1.]
Designation of a feed rate is not necessary for the positioning command G00.

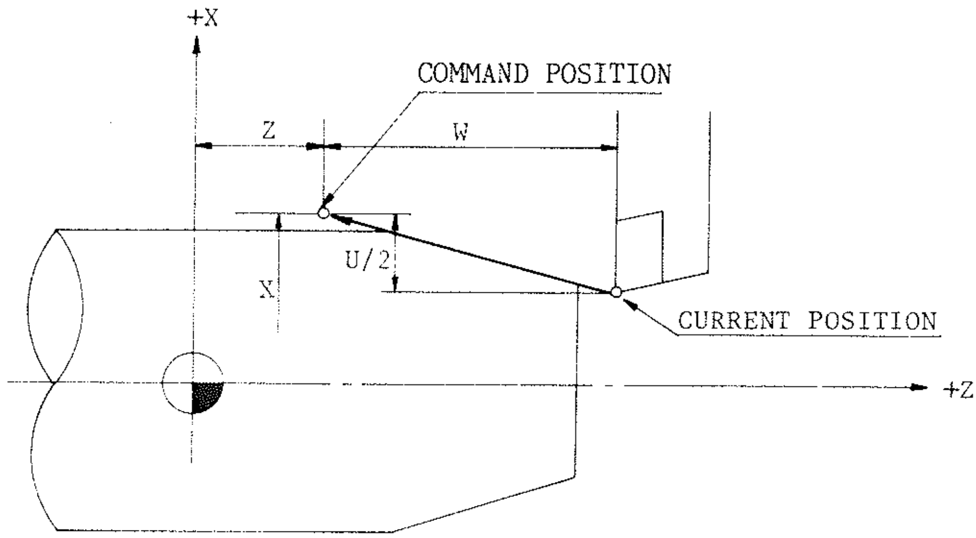
[NOTE 2.]
The absolute and incremental systems can be mixed in the positioning command.
G00 X______ W______ ; or G00 U______ Z______ ;

[3] G01

(1) Linear Interpolation

A tool is moved on a straight line at a feed rate specified by the F code to the position (X, Z) in the coordinate system set by the tool offset command (T □□□□), or the distance (U, W) from the current position.
The forms of those commands are as follows;

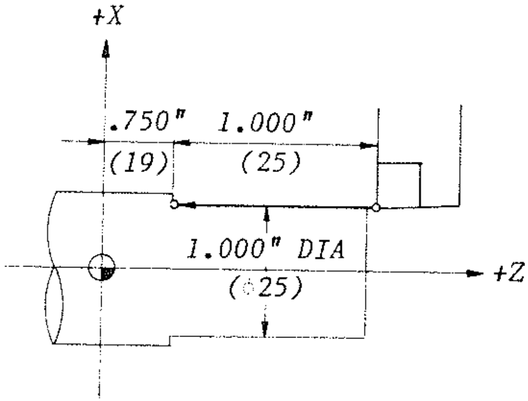
Absolute system	G01 X_____ Z______ F______ ;
Incremental system	G01 U_____ W______ F______ ;



[NOTE]

The feed rate specified by F code represents the tangential feed amount. The accuracy of the feed rate will be ±6%.

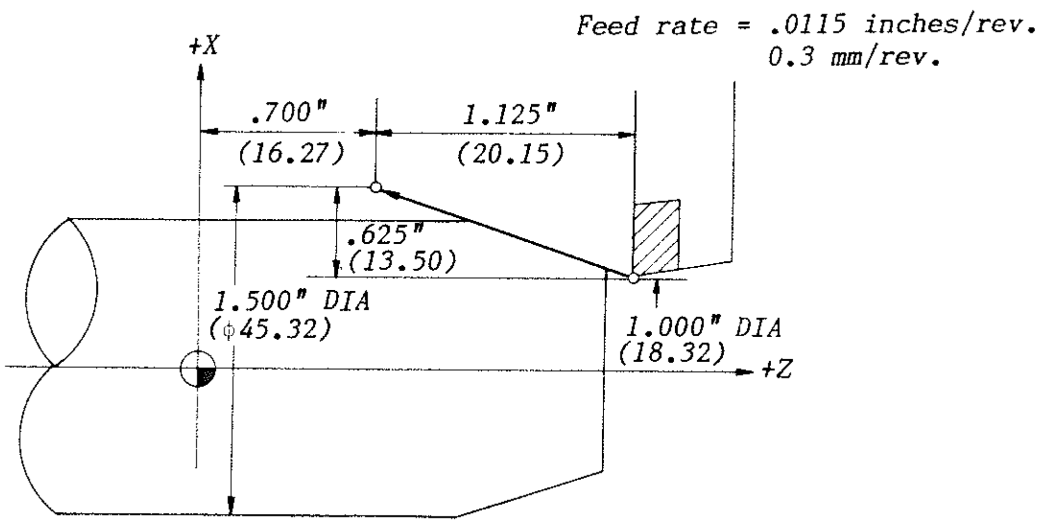
- Example 1 -



Feedrate = .012 inches/rev.
0.3 mm/rev

G01 Z.750 F.012 ;	*or*	*G01 W-1.0 F12000 ;*	*Inch*
G01 Z19.0 F0.3 ;	*or*	*G01 W-25.0 F3000 ;*	*Metric*

- Example 2 -



G01 X1.5 Z.700 F.0115 ; or G01 U1.25 W-1.125 F11500 ; ——Inch

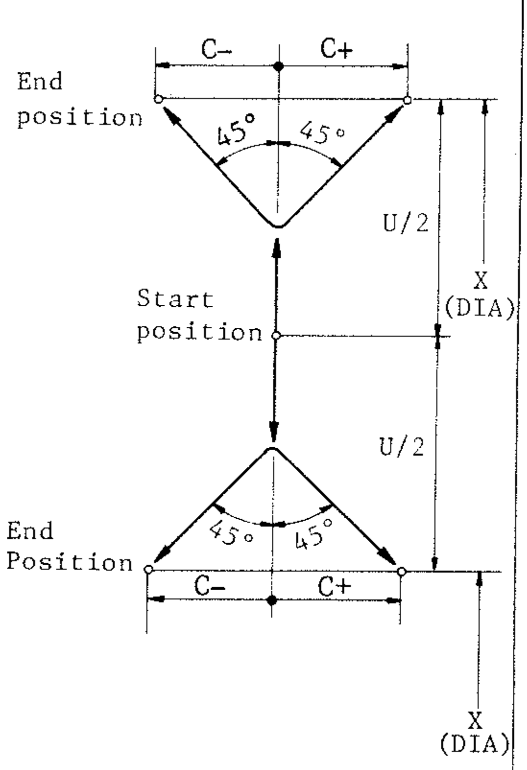
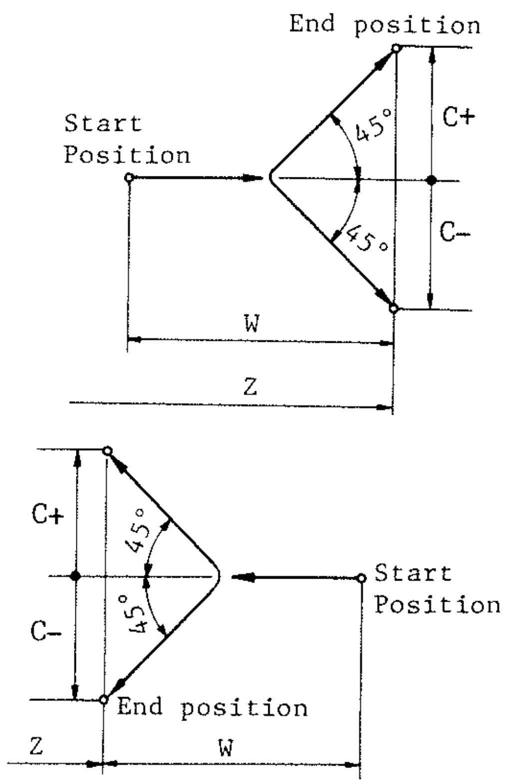
G01 X45.32 Z16.27 F0.3 ; or G01 U27.0 W-20.15 F3000 ; ——Metric

BND-S PRO 0-TC 90/8

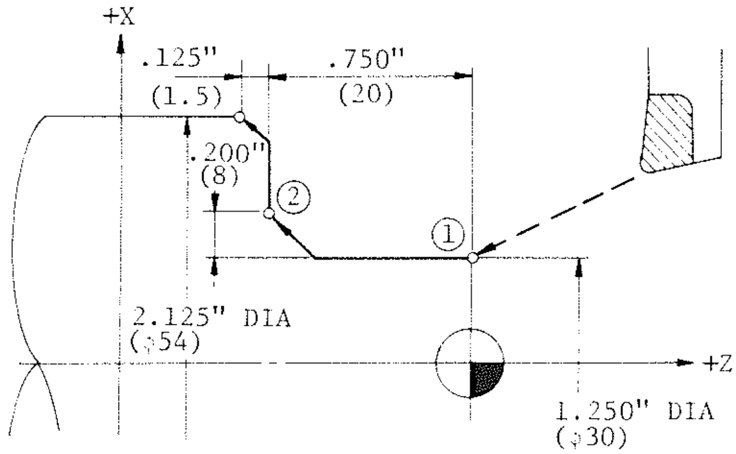
(2) 45° Chamfering

The form of 45° chamfering command is as follows;

```
G01 X(U)-------- C------- F-------- ; ———for X-axis
or
G01 Z(W)-------- C------- F-------- ; ———for Z-axis
```

Chamfering for X-axis	Chamfering for Z-axis
G01 X(U) ____ C____ F____;	G01 Z(W)____ C____ F____ ;
	
C ± —— Chamfering value Chamfering direction	C ± —— Chamfering value Chamfering direction

- Sample programming of 45° chamfering -



Inch

```
G00 X1.25 Z0 ;
G01 Z-.75 C.2 F.006 ;.............. Tool path ①
    X2.125 C-.125 ; ............... Tool path ②
```

Metric

```
G00 X30.0 Z0 ;
G01 Z-20.0 C8.0 F.2 ;
    X54.0 C-1.5 ;
```

(NOTE 1)
The movement for chamfering must be a single movement along the X or Z axis in G01 mode.
The next block must be a single movement along the X or Z axis perpendicular to the former block.

(NOTE 2)
The following conditions generate alarms.

- *Both the X and Z axes were specified and either C was specified in G01 mode. (Alarm No. 054)*
- *The move distance along the X or Z axis is smaller than the chamfering value in the block in which the chamfering was specified. (Alarm No.055)*
- *The motion specified in block after the block specifying chamfering is not perpendicular to motion specified in the former block in G01 mode. (Alarm No. 051, 052)*

(NOTE 3)
When C is specified for one block by G01, the latter will be valid.

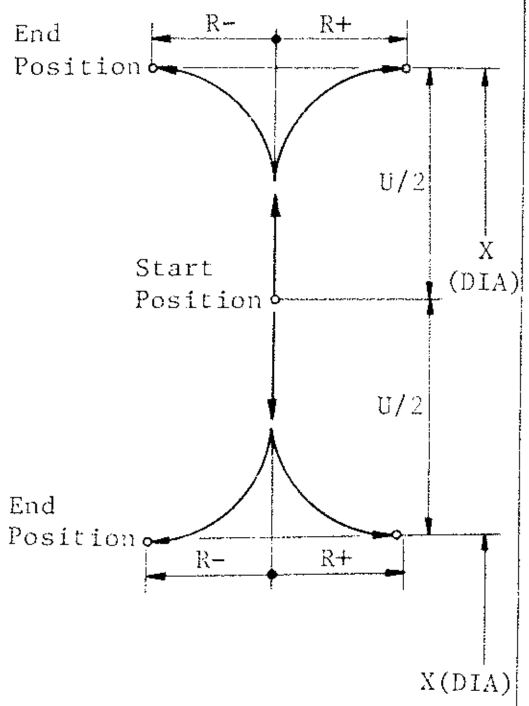
(NOTE 4)
Chamfering cannot be used in a thread cutting block.

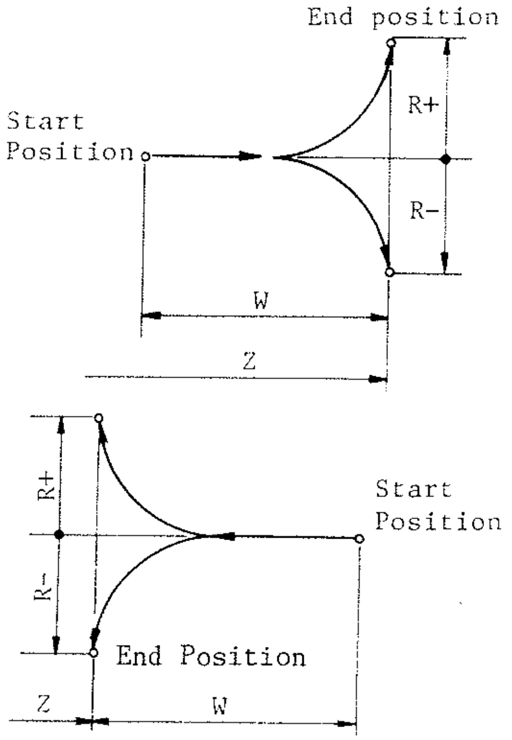
(3) Rounding

The form of rounding command is as follows;

```
G01 X(U)--------R--------F------ ; ———— for X-axis
or
G01 Z(W)--------R-------F------ ; ———— for Z-axis
```

Rounding for X-axis	Rounding for Z-axis
G01 X(U) ___ R___ F___ ;	G01 Z(W) ___ R___ F___ ;



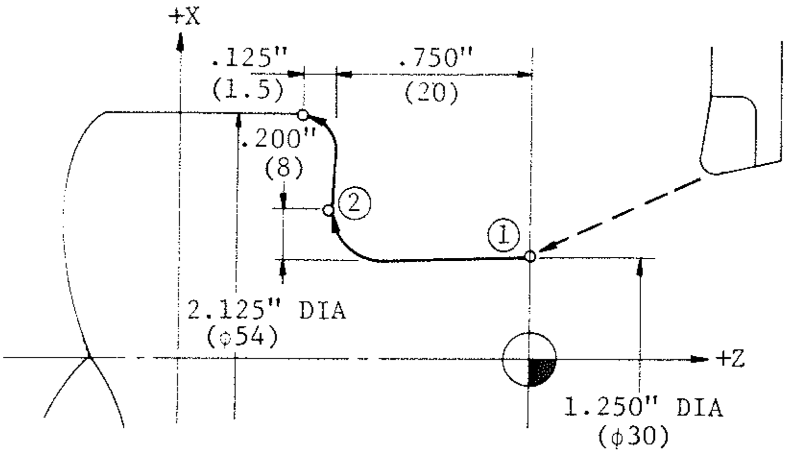


R±

Rounding value (r)

Rounding direction

- Sample programming of rounding -



Inch

```
    G00 X1.25 Z0 ;
    G01 Z-.750 R.200 F.006 ; ........... Tool path ①
   (G01)X2.125 R-.125 ; ................ Tool path ②
```

Metric

```
    G00 X30.0 Z0 ;
    G01 Z-20.0 R8.0 F.3 ;
   (G01)X54.0 R-1.5 ;
```

(NOTE 1)

The movement for corner R must be a single movement along the X or Z axis in G01 mode.
The next block must be a single movement along the X or Z axis perpendicular to the former block.

(NOTE 2)

The following conditions generate alarms.

- *Both the X and Z axes were specified and either R was specified in G01 mode. (Alarm No. 054)*
- *The move distance along the X or Z axis is smaller than the corner R value in the block in which the corner R was specified. (Alarm No. 055)*
- *The motion specified in block after the block specifying corner R is not perpendicular to motion specified in the former block in G01 mode. (Alarm No. 051, 052)*

(NOTE 3)

When R is specified for one block by G01, the latter will be valid.

(NOTE 4)

Rounding cannot be used in a thread cutting block.

BND-S PRO 0-TC 90/8

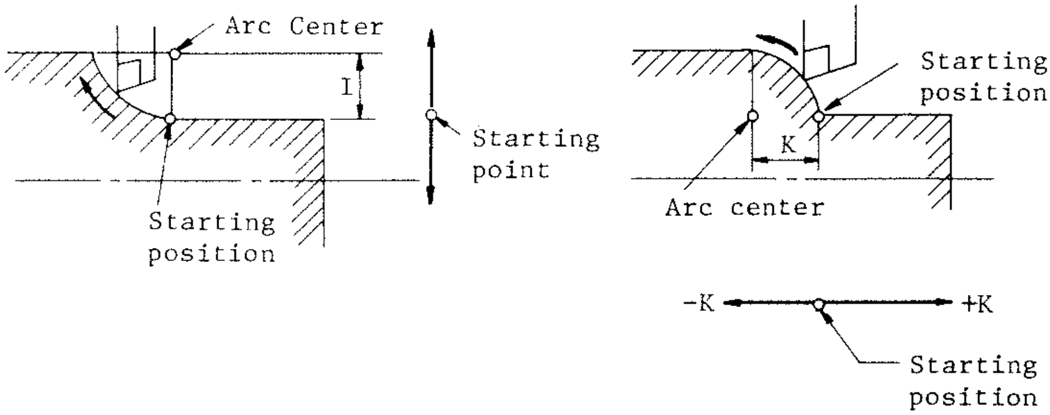
[4] G02, G03

(1) Circular Interpolation

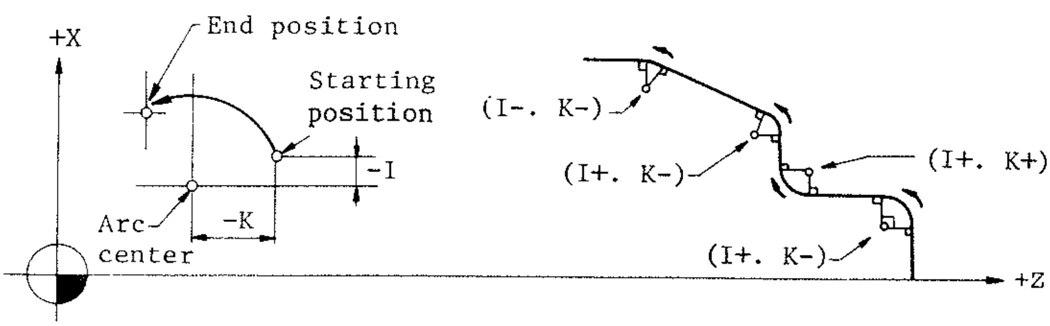
When G02 or G03 is commanded, a tool is moved along an arc at a feed rate specified by the F-function, to the absolute position (X, Z) or the distance (U, W) from the present position of the tool.

G02 (Clockwise) G03 (Counter Clockwise)

The position of the arc center is specified by using the address I in the X-axis (with radius value) and the address K in the ~~X~~Z-axis.

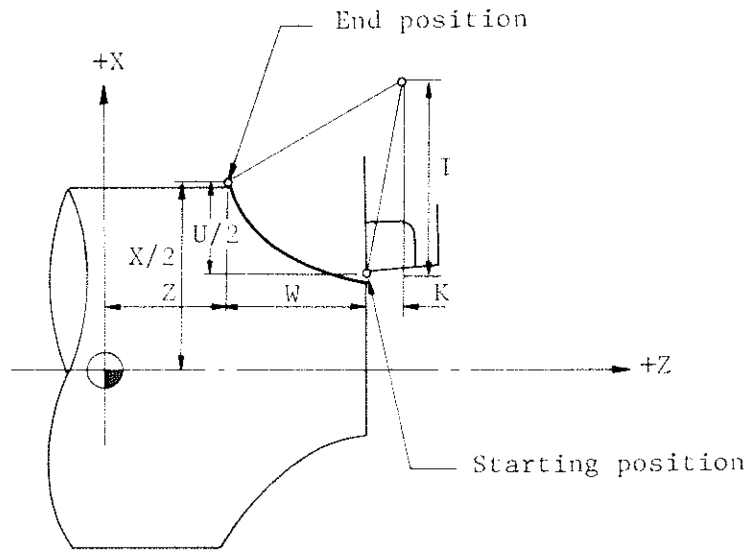


The following sketch shows examples to put the signs '+' (plus) or '-' (Minus) with the addresses I and K.

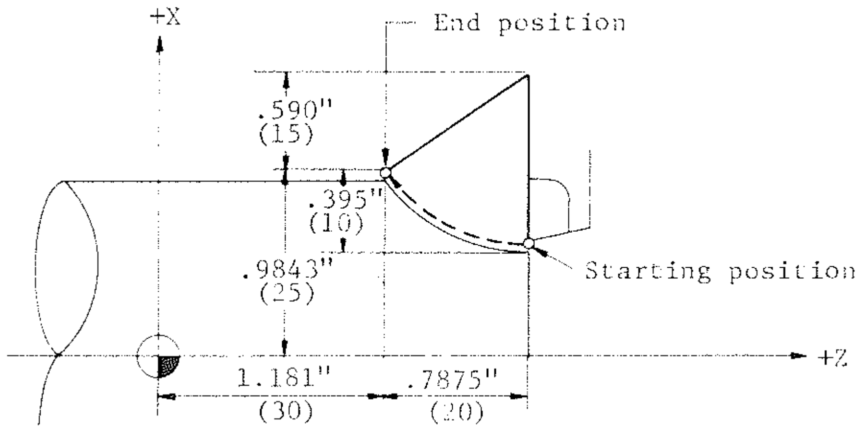


a) The form of G02 (Clockwise) command

G02 X(U)_______ Z(W)_______ I______ K_____ F______ ;



- Example of G02 command -



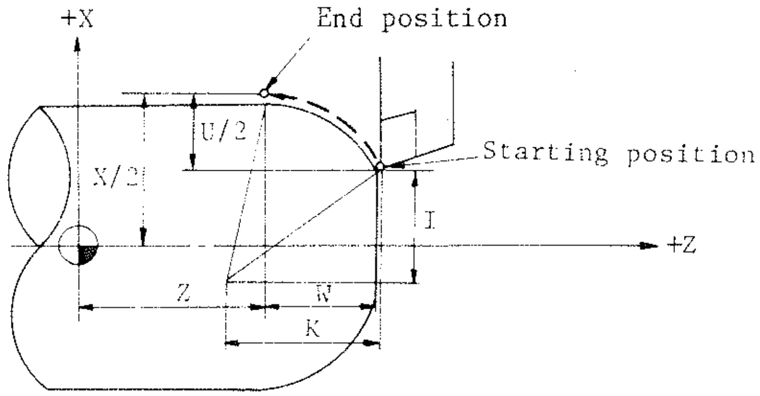
Feed rate = .012 inches/rev.
0.3 mm/rev

Inch
G02 X1.9686 Z1.181 I.985 F.012 ;
or G02 U.790 W-.7875 I.985 F.012 ;

Metric
G02 X50.0 Z30.0 I25.0 F.3 ;
or G02 U20.0 W-20.0 I25.0 F.3 ;

b) The form of G03 (Counter clockwise) command

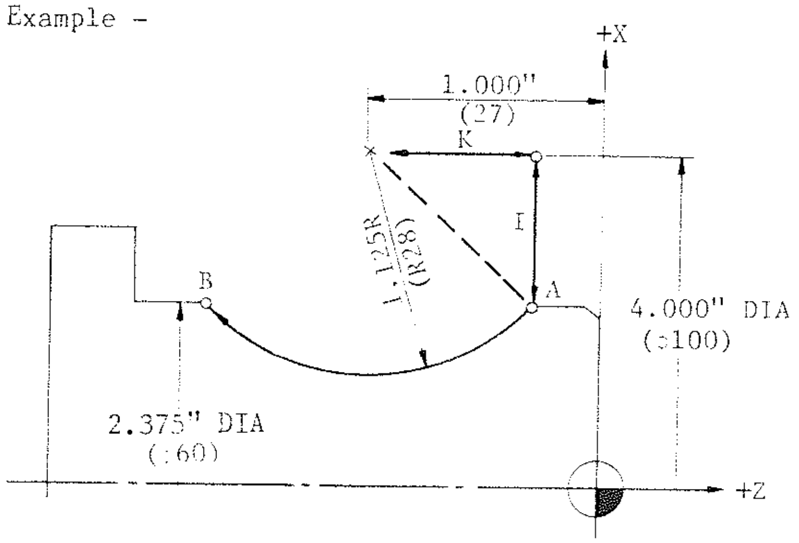
G03 X(U)______ Z(W)______ I______K______F______ ;



c) Multiquadrant circular command

Circular interpolation of an arc on multiquadrant can be programmed in a single block as the following example. However, a closed circle can not be programmed.

- Example -



	(Inch)	(Metric)
*Position of arc center	X4.0, Z-1.0	X100.0, Z-27.0
*Dimension of I	$\frac{4.000 - 2.375}{2} = .8125$	$\frac{100 - 60}{2} = 20$
*Dimension of K	$\sqrt{1.125^2 - .8125^2} = .7781$	$\sqrt{28^2 - 20^2} = 19.596$
*Position of point A	X2.375, Z-.2219	X60.0, Z-7.4

*The program is as follows;

```
G01 Z-.2219;                                    G01 Z-7.4 ;
G02 X2.375 Z-1.7781 I.8125 K-.7781 F__;         G02 X60.0 Z-46.596 I20.0 K-19.596 F__ ;
```

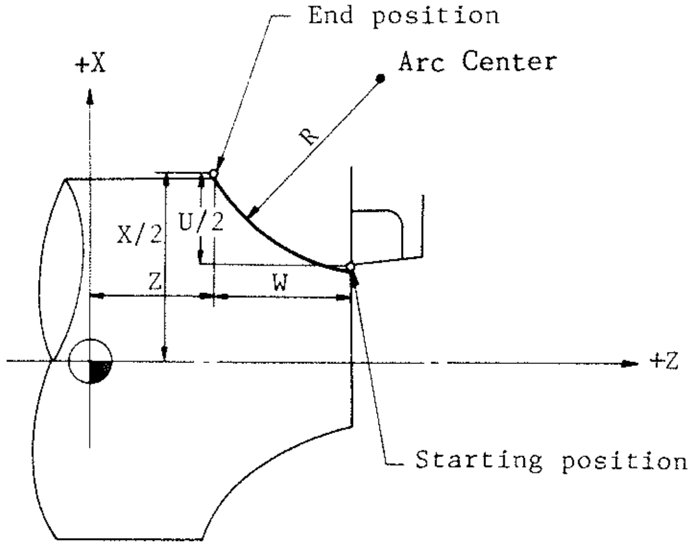
These are point B command

BND-S PRO 0-TC 90/8

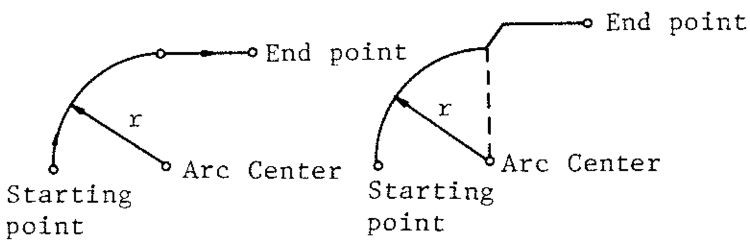
(2) Radius Programming for Circular Interpolation

The forms of radius programming for circular interpolation is as follows ;

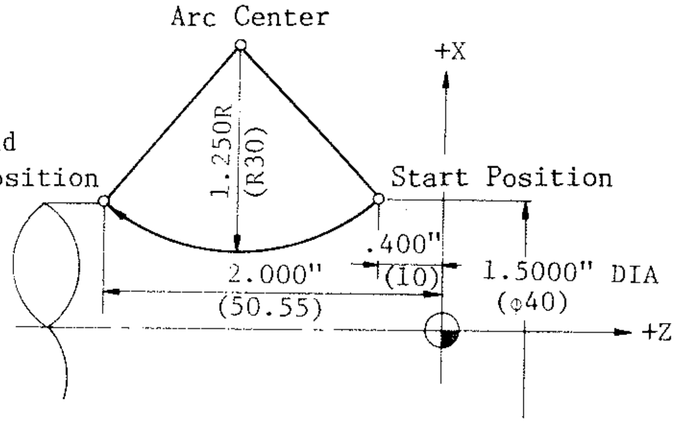
G02 X(U)----- Z(W)----- R----- F -----	; Clockwise
G03 X(U)----- Z(W)----- R----- F -----	; Counter Clockwise



When the end point of arc is not designated on the circumference specified by the radius, an axis will move to the end point after the other axis meet with its coordinate, as shown in sketch below.



-Sample programming of G02 -



G02 Z-2.0 R1.250 F.008 ; or G02 W-1.6 R1.250 F.008 ; —Inch

G02 X-50.55 R30.0 F.2 ; or G02 W-40.55 R30.0 F.2 ; —Metric

(NOTE 1)

An arc more than 180 degrees cannot be command.
Accordingly : G02 R___; and G03 R___;
are not full circle but a circle of zero degree is assumed. So the tool does not move.

(NOTE 2)

When address R specifies a value less than half of the distance between the start point and the end point, a half circle will be generated.

(NOTE 3)

Feedrate in circular interpolation is specified with an address F.

(NOTE 4)

If address R is omitted, the tool moves to the end point linearly.

(NOTE 5)

When either I or K is zero, the word can be omitted.

[5] G04

(1) Dwell

The feed can be stopped for a designated period of time by programming the following command.

```
G04 P----- ;

or

G04 U-----;

or

G04 X----- ;
```

The maximum command value of the dwell time is 99999.999 seconds.

For example, when 2.5 seconds of dwell time is needed, the command is as follows;

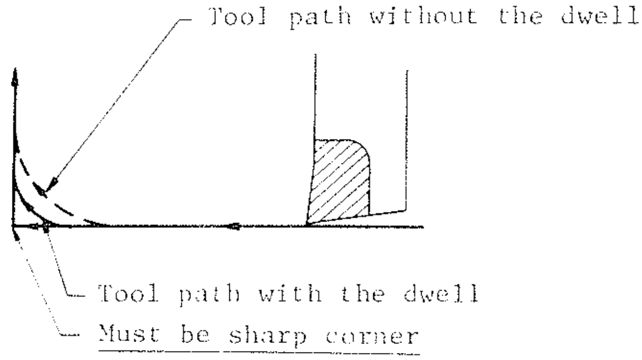
```
G04 P2500 ;
G04 U2.5 ;
G04 X2.5 ;
```

[NOTE 1]
The decimal point can not be used when the address P is specified.

[NOTE 2]
Dwell begins after the commanded feedrate of the previous block attains zero.

- Application examples of G04 command -

(1) When a sharp corner is required as shown in below, a radius may occur if the dwell is not used. This is caused by the delay of the servo system.



(2) In the operation of grooving or drilling etc., the chip can be broken by using G04 command.

```
.
.
.
.
.
G01 X2.5 F.012 ;
G04 P200 ;        The feed will stop at 0.2 sec.
G01 X2.0 ;        This will usually break the
G04 P200 ;        chip in most of materials.
.
.
.
.
.
```

BND-S PRO 0-TC 90/8

[6] G10 Changing of tool offset amount (Programmable data input)

Offset values can be input by program.
This command can be used to input offset values at a time, from a program in which they are punched by specifying this command succesively, instead of inputting them one at a time from the MDI unit.

The command is as follows;

```
G10 P___ X___ Z___ R___ Q___ ;

or

G10 P___ U___ W___ C___ Q___ ;
```

P : Offset number
For wear offset amount : P = wear offset number
For geometry offset amount : P = 10000 + geometry offset number

X : Offset value in X-axis (absolute)

Z : Offset value in Z-axis (absolute)

U : Offset value in X-axis (incremental)

W : Offset value in Z-axis (incremental)

R : Tool nose radius offset value (absolute)

C : Tool nose radius offset value (incremental)

Q : Standard tool nose number

In absolute command, the values specified by addresses X and Z are set as an offset value corresponding to the offset number specified by address P.
In incremental command, the values specified by addresses U and W are added to the stored offset value corresponding to the offset number specified by address P.

[7] G20, G21 (Inch/Metric Input)

Either inch or metric input can be selected by these G-codes.

Unit system	G code	Least input increment
Inch	G20	0.0001 inch
mm	G21	0.001 mm

This G-code must be specified at the head of program by using an individual block.
If the different one from the unit setting is used, the following parameters must be changed. Otherwise, the machine can not be operated properly.

Parameter No. 717,718,719,720,721,722,723,724,725,726.
The parameter is changed by the program command.

The following unitary value are changed by specifying these G codes.

(1) Commanded and displayed value of feed speed by F code.

(2) Commanded value and current value.

(3) Offset value.

(4) The unit of a division of the manual pulse generator.

BND-S PRO 0-TC 90/8

[NOTE 1]

When the power is turned on, the state of G20/G21 will be the same as it was effective when the power is shut off.

[NOTE 2]

G20/G21 must not be switched in the middle of program.

[NOTE 3]

The input unit can be changed with the setting parameter.
The setting parameter automatically changes with G20, G21.

[8] G27 (Reference point Return Check)

This function checks if the turret slide returns to the home position. When the program is made so that the turret slide starts from and returns to the home position.

The form of G27 command is as follows;

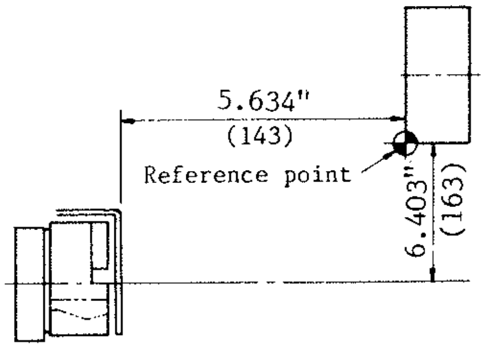
```
G27 X(U) 0  Z(W) 0  T0000 ;
```

After positioning at the absolute coordinates (X,Z) or the incremental coordinates (U,W), the position is checked if it is the home position or not.
When the position meets with the home position, the ZERO RETURN lamp will light up and the automatic operation will continue.
When either or both of the axis does not meet with the home position, the alarm code '092' (Reference point Return error) will be displayed and the automatic operation will stop. (The CYCLE START lamp is turned off.)
When G27 is commanded in tool offset mode, the position may not meet with the home position. Therefore, tool offset mode should be cancelled when G27 is commanded as follows;

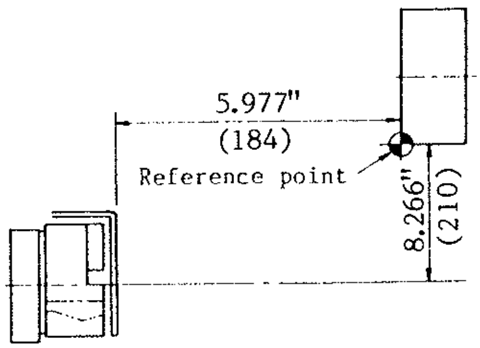
[NOTE 1]

The home position means the definite position on the machine where the turret slide is returned by the manual zero return operation.

BND-20S



BND-34S



BND-S PRO 0-TC 90/8

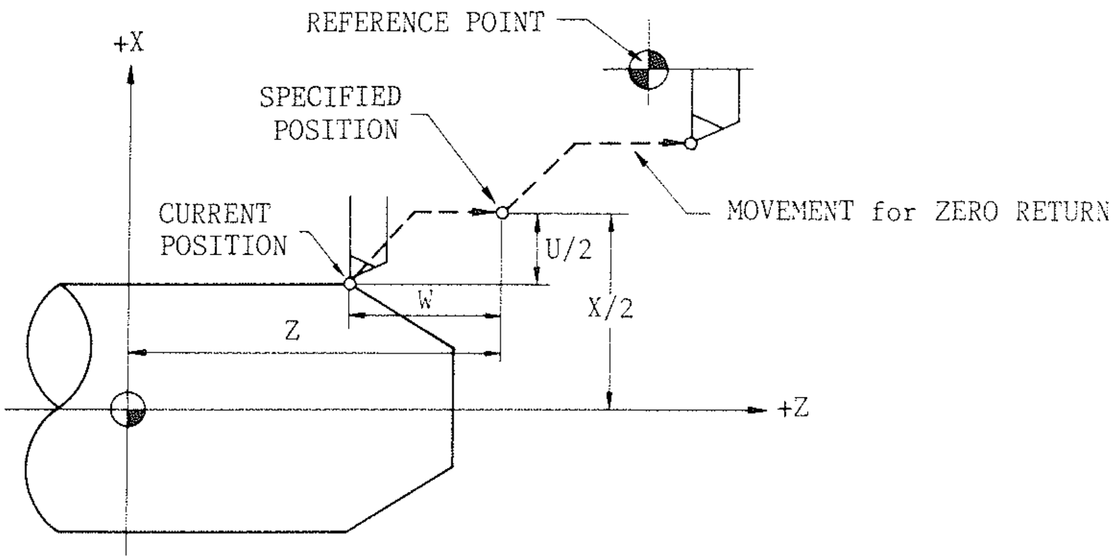
[9] G28 (Automatic return to reference point)

The turret slide is moved at a rapid traverse rate for each direction simultaneously to the position (X,Z) or away from the current position by (U,W) and then automatically returned to the reference point.

The form of this command is as follows;

G28 X(U) ---- Z(W) ---- ;

When the turret slide is returned to the home position, the ZERO RETURN lamp will light up and the automatic operation will be continued.

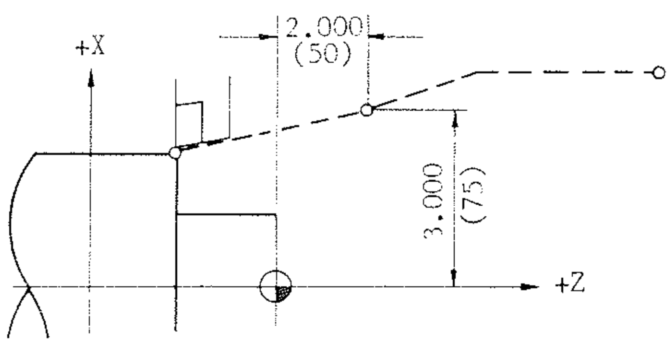


[NOTE 1]

If X(U) and Z(W) are not commanded in the block of G28 as G28 ;, the zero return will start from the current position.

- Sample Programming of G28 -

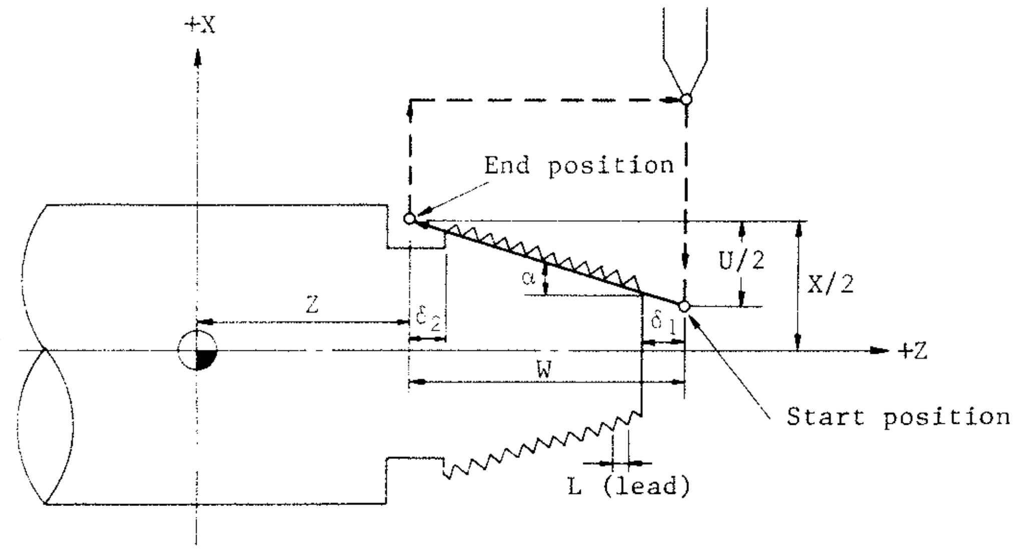
G28 X6.0 Z2.0 ; ——— Inch
G28 X150.0 Z50.0 ; —— Metric



[10] G32 (Thread Cutting)

Straight, taper and face thread cutting are effected by using G32 command. The position of the thread end point is specified by (X, Z) in the absolute system or the distance by (U, W) from the present position in the incremental system.
The lead of the thread is directly specified by the numerical value following the address F.
The form of the G32 command is as follows;

G32 X(U)_____ Z(W)_____ F_____ ;



(1) Designation of thread lead and programmable lead range

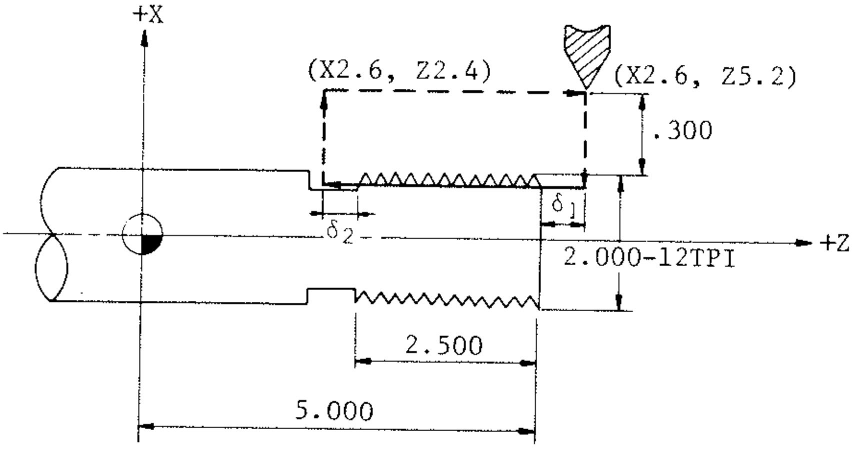
The thread lead is specified by using F code for odd-lead thread.

Programmable Lead Range

	Format	Programmable Range
Inch	F16	.000001 to 9.999999 in
Metric	F34	0.0001 to 500.0000 mm

a) Example of programming of straight thread cutting

Inch



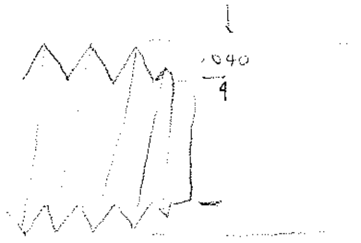

```
Thread lead       : .0833 in (12 TPI)
      δ1          : .200  in
      δ2          : .100  in
Depth of thread   : .0400  in
Number of passes  : 4 passes
Spindle speed     : 955 rpm
```

ea side = .050 [illegible]

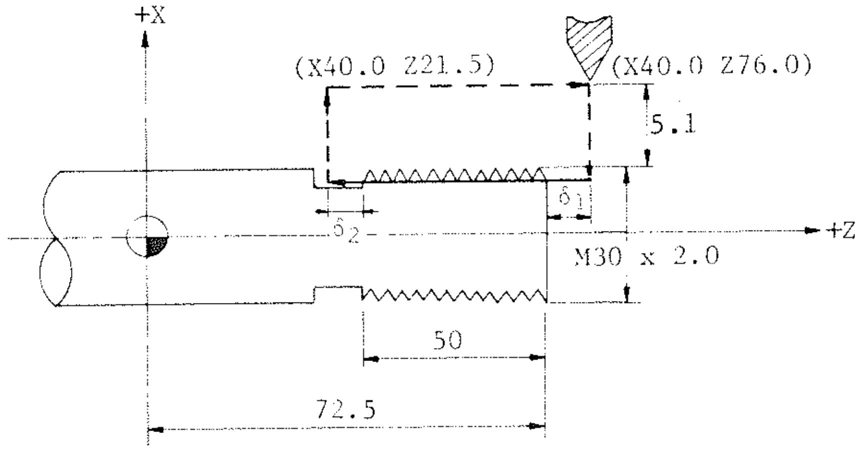
- Program -

```
      .
      .
      .
 G97 S955 M03 ;
 G00 X2.6 Z5.2 ;
     X1.96 ;
 G32 Z2.4 F83333 ;
 G00 X2.6 ;
     Z5.2 ;
     X1.94 ;
 G32 Z2.4 ;
 G00 X2.6 ;
     Z5.2 ;
     X1.93 ;
 G32 Z2.4 ;
 G00 X2.6 ;
     Z5.2 ;
     X1.92 ;
 G32 Z2.4 ;
 G00 X2.6 ;
     Z5.2 ;
      .
      .
      .
```

Δ THD DEPTH IS .040 ea side. Depth [illegible] side
Cut #1 to 2.000 - 1.960 = .040 dia
Cut #2 to 1.96 - 1.940 = .020 dia
#3 1.94 - 1.93 = .010 dia
#4 1.93 - 1.92 = .010 dia
.080 Depth



Metric



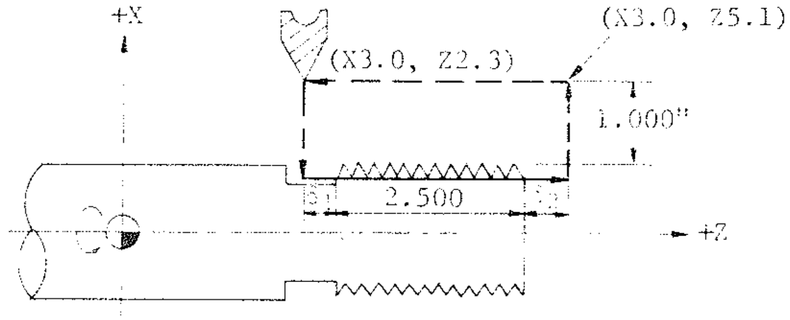
Thread lead	:	M30 × 2.0
δ_1	:	3.5 mm
δ_2	:	1.0 mm
Depth of thread	:	1.2 mm
Number of passes	:	4 passes
Spindle speed	:	955 rpm

- Program -

```
      .
      .
      .
G97 S955 M03 ;
G00 X40.0 Z76.0 ;
    X29.2 ;
G32 Z21.5 F2.0 ;
G00 X40.0 ;
    Z76.0 ;
    X28.6 ;
G32 Z21.5 ;
G00 X40.0 ;
    Z76.0 ;
    X28.0 ;
G32 Z21.5 ;
G00 X40.0 ;
    Z76.0 ;
    X27.4 ;
G32 Z21.5 ;
G00 X40.0 ;
    Z76.0 ;
      .
      .
      .
```

b) Cut to the direction of '+Z' with spindle forward.

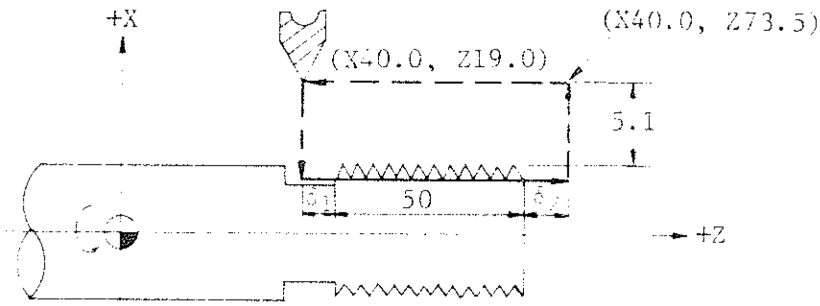
Inch



Program for sample part in a) is as follows;

```
      .
      .
G97 S955 M03 ;
G00 X3.0 Z2.3 ;
    U2.04 ;
G32 W2.8 F83333 ;
G00 X3.0 ;
    Z2.3 ;
      .
      .
      .
```

Metric



– Program –

```
      .
      .
G97 S935 M03 ;
G00 X40.0 Z19.0 ;
    X29.2 ;
G32 Z73.5 F2.0 ;
G00 X40.0 ;
    Z19.0 ;
      .
      .
      .
```

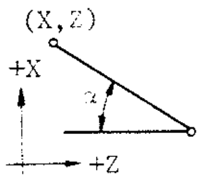
(2) Command format of thread cutting

In case of straight thread cutting, the word X(U) is omitted, and in case of face thread cutting, the word Z(W) is omitted.

Type	Command format
Straight thread	G32 Z(W) ____ F ____ ;
Taper thread	G32 X(U) ____ Z(W) ____ F ____ ;
Face thread	G32 X(U) ____ F ____ ;

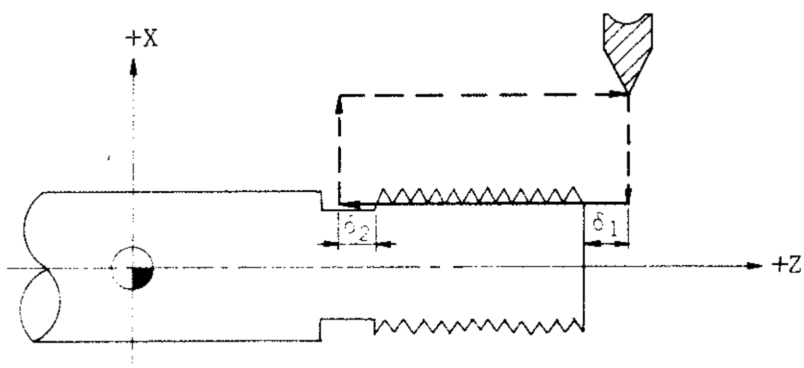
(3) Direction of lead for taper thread

The lead of the taper thread is determind by angle (α) of the taper in Z-axis as follows.

Limitation of taper Angle		Direction of Lead
	$\alpha \leqq 45°$	Lead in the direction of Z-axis
	$\alpha > 45°$	Lead in the direction of X-axis

(4) Allowance of incorrect lead (δ_1, δ_2)

The allowance δ_1 and δ_2 (Shown in the sketch below) must be made to eliminate incorrect lead for the thread near the start and the end positions which is due to delay of the servo system.



The amount of δ_1 and δ_2 can be found by using the calculation formula table.
However, they can be calculated approximately by using the following equation.

Equation	Meanings
$\delta_1 > \frac{\ell \cdot S \cdot T}{60} - (\ell n \frac{1}{a} - 1)$	ℓ (in) : Lead of thread S (rpm): Spindle speed T : Constant (Normal value 1/33) a : Accuracy of thread $= \frac{\Delta \ell}{\ell}$ ----- Lead error ℓn : Natural logarithm(log e)
$\delta_2 > \frac{\ell \cdot S \cdot T}{60}$	

a	0.005	0.01	0.015	0.02	0.025
$\ell_n \frac{1}{a} - 1$	4.30	3.61	3.19	2.91	2.68

- Example -

Inch

Thread : 6TPI ℓ = .1667 in.
Spindle speed : S = 560 rpm
Thread accuracy : a = 0.01

$$\delta_1 = \frac{\ell \cdot S \cdot T}{60} (\ell_n \frac{1}{a} - 1) = \frac{.1667 \times 560 \times 1/33}{60} \times 3.61$$

$$= .170 \text{ (in.)}$$

$$\delta_2 = \frac{\ell \cdot S \cdot T}{60} - = \frac{.1667 \times 560 \times 1/33}{60}$$

$$= .047 \text{ (in.)}$$

Metric

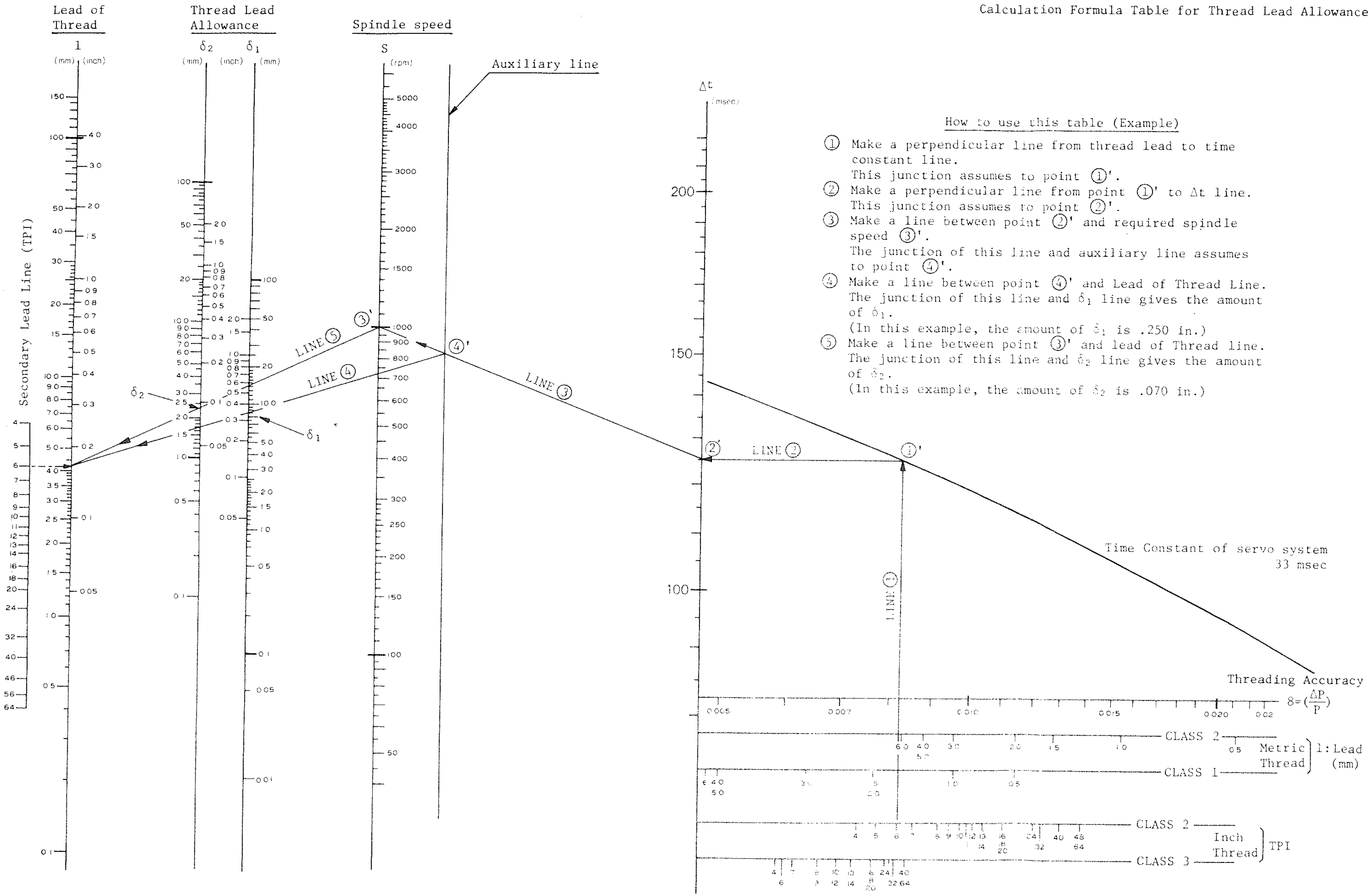
Thread : ℓ = 4 mm
Spindle speed : S = 560 rpm
Thread accuracy : a = 0.009

$$\delta_1 = \frac{\ell \cdot S \cdot T}{60} - (\ell_n \frac{1}{a} - 1) = \frac{4 \times 560 \times 1/33}{60} \times 3.75$$

$$= 4.19 \text{ mm}$$

$$\delta_2 = \frac{\ell \cdot S \cdot T}{60} = \frac{4 \times 560 \times 1/33}{60}$$

$$= 1.13 \text{ mm}$$



How to use this table (Example)

① Make a perpendicular line from thread lead to time constant line.
This junction assumes to point ①'.

② Make a perpendicular line from point ①' to Δt line.
This junction assumes to point ②'.

③ Make a line between point ②' and required spindle speed ③'.
The junction of this line and auxiliary line assumes to point ④'.

④ Make a line between point ④' and Lead of Thread Line.
The junction of this line and δ_1 line gives the amount of δ_1.
(In this example, the amount of δ_1 is .250 in.)

⑤ Make a line between point ③' and lead of Thread line.
The junction of this line and δ_2 line gives the amount of δ_2.
(In this example, the amount of δ_2 is .070 in.)

(5) Notes of thread cutting

a) Feed Override is not effective during a thread cutting cycle.
... It is regarded as 100%.

b) Spindle speed must be the same in all passes of the threading.
If spindle speed changes during the thread cutting cycle, the leads will become incorrect due to the servo lag. Therefore, the constant cutting speed control can not be used for thread cutting.

c) Thread pull-out (M23) is not effective with G32 command.
If necessary, G76 or G92 should be commanded.

d) If the SINGLE BLOCK switch is set to ON position or the FEED HOLD button is depressed during the thread cutting operation, these function will be effective after completion of the next block.

e) If the DRY RUN switch is set to on position during the thread cutting operation, the feed rate will be regarded as setting jog feed rate of the FEED RATE dial.

f) Thread cutting command (G32, G76, G92) should not be commanded in G98 (Feed amount per minute) mode.

g) The allowance of incorrect lead δ_1 should be as big as possible for better chips removal.

h) The spindle speed has a limitation during thread cutting operation, depending on the lead of the thread.

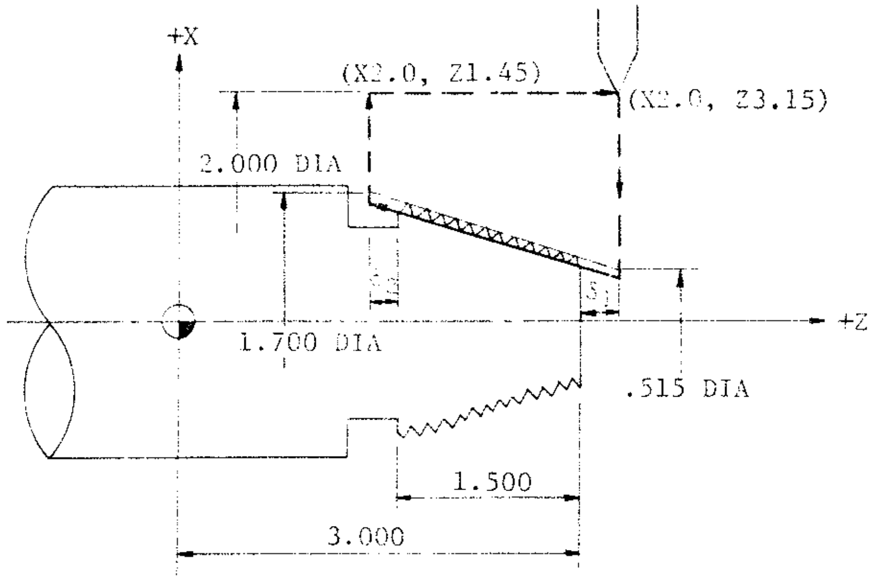
Lead In(mm)	x	Spindle Speed rpm	≦	472 in/min (12000 mm/min)

The following table shows the maximum lead of the thread according to the spindle speed.

RPM	Maximum Thread Lead		TPI	RPM	Maximum Thread Lead		TPI
	MM	INCH			MM	INCH	
200	50	1.965		6500	1.5	.06	16
500	20	.786	1	7000	1.4	.056	18
1000	10	.393	2	7500	1.3	.05	19
1500	6	.262	4	8000	1.2	.049	21
2000	5	.196	5	8500	1.1	.046	23
2500	4	.157	6	9000	1.1	.043	23
3000	3.3	.131	7	9500	1	.04	25
3500	2.8	.112	9	10000	1	.039	25
4000	2.5	.098	10	10500	0.95	.037	26
4500	2.2	.087	11	11000	0.9	.035	28
5000	2	.078	12	11500	0.86	.034	29
5500	1.8	.07	14	12000	0.83	.032	30
6000	1.6	.065	15				

(6) Example of programing of taper thread cutting

Inch

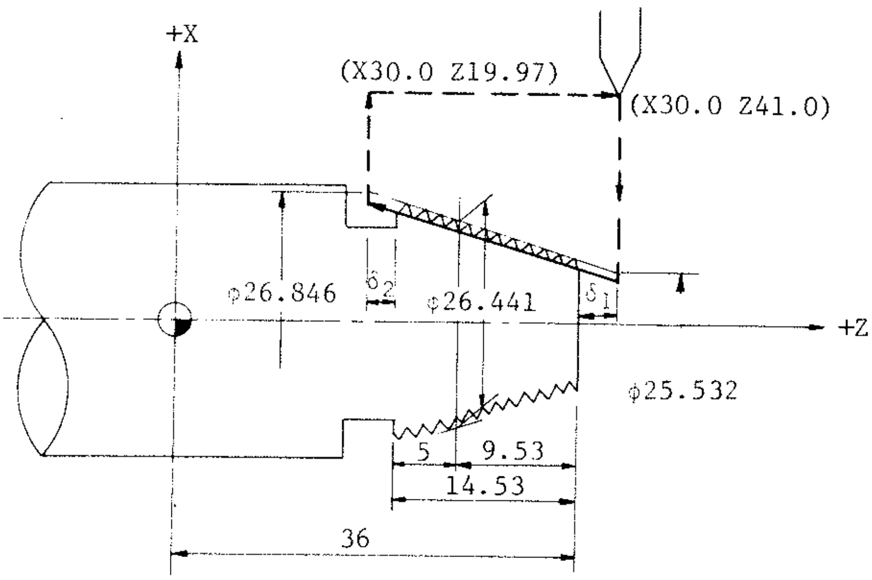


Thread lead	:	.055556 in. (18 TPI)
δ_1	:	.150 in
δ_2	:	.050 in.
Depth of thread	:	.045 in.
Number of passes	:	4 Passes
Spindle Speed	:	1200 rpm

- Program -

```
     .
     .
     .
G97 S1200 M03 ;
G00 X2.0 Z3.15 ;
    X.520 ;
G32 X1.67 Z1.45 F55556 ;
G00 X2.0 ;
    Z3.15 ;
    X.480 ;
G32 X1.63 Z1.45 ;
G00 X2.0 ;
    Z3.15 ;
    X.470 ;
G32 X1.62 Z1.45 ;
G00 X2.0 ;
    Z3.15 ;
    X.460 ;
G32 X1.61 Z1.45 ;
G00 X2.0 ;
    Z3.15 ;
     .
     .
     .
```

Metric



Thread lead	:	L=1.8143 (PT3/4,ISO R7)
δ 1	:	5 mm
δ 2	:	1.5 mm
Depth of thread	:	1.162 mm
Number of passes	:	4 Passes
Spindle Speed	:	1200 rpm

- Program -

```
    .
    .
G97 S1200 M03 ;
G00 X30.0 Z41.0 ;
    X24.951 ;
G32 X26.265 Z19.97 F1.8143 ;
G00 X30.0 ;
    X41.0 ;
    X24.37 ;
G32 X25.684 Z19.97 ;
G00 X30.0 ;
    X41.0 ;
    X23.789 ;
G32 X25.103 Z19.970 ;
G00 X30.0 ;
    Z41.0 ;
    X23.209 ;
G32 X24.523 Z19.97 ;
G00 X30.0 ;
    Z41.0 ;
    .
    .
    .
```

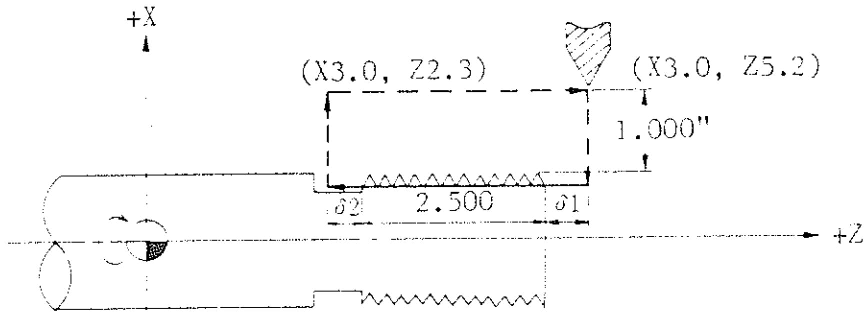
BND-S PRO 0-TC 90/8

(7) Example of programming of left hand thread cutting.

There are two ways to cut left hand thread on the machine.

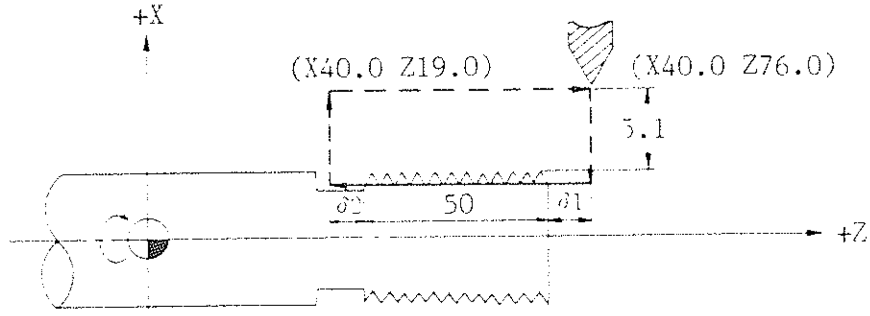
a) Reverse the spindle and cut the thread to the direction of '-Z' by using the tool holder and cutting tool for reverse spindle.

Inch



Thread lead : .0833 in (12 TPI)
δ 1 : .200 in
δ 2 : .100 in
Spindle speed : 955 rpm

Metric



Thread lead : M30 × 2.0
δ 1 : 3.5 mm
δ 2 : 1.0 mm
Spindle speed : 955 rpm

- Program -

Inch

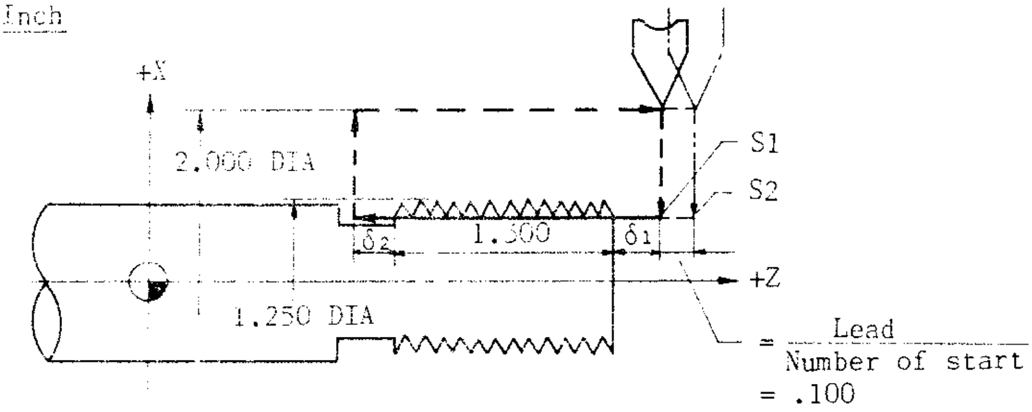
```
.
.
G97 S955 M04 ;   <---- Spindle reverse
G00 X3.0 Z5.2 ;
    U2.04 ;
G32 W-2.8 F83333 ;
G00 X3.0 ;
    Z5.2 ;
.
.
.
```

Metric

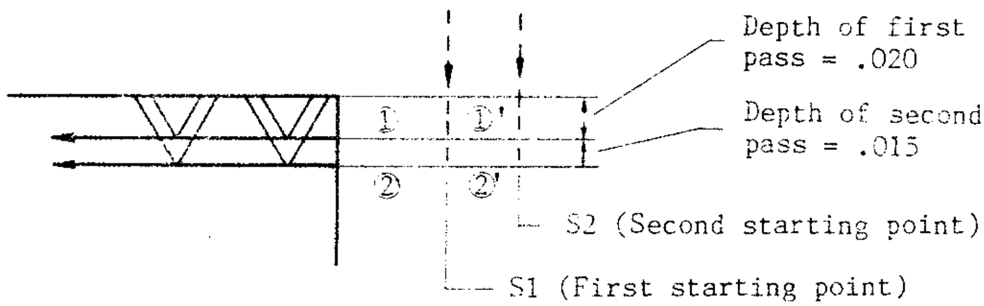
```
.
.
G97 S955 M04 ;   ----> Spindle reverse
G00 X40.0 Z76.0 ;
    X29.2 ;
G32 Z21.5 F2.0 ;
G00 X40.0 ;
    Z76.0 ;
.
.
.
```

(8) Example of programming of multi - start thread cutting

Multi-start thread cutting can be done by starting each thread with calculated distance from the previous thread start.
In order to find the starting point for each thread, simply divide the thread lead by the number of starts. Then, program the start of each additional thread with this distance from the previous starting point. The end position should be the same.



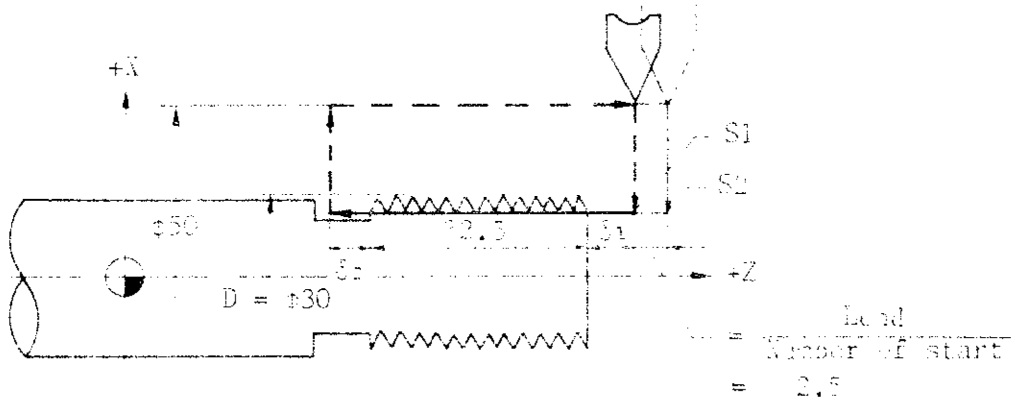
Thread lead : .200 in .....2 starts thread
δ1 : .400 in
δ2 : .200 in
Spindle speed : 900 rpm



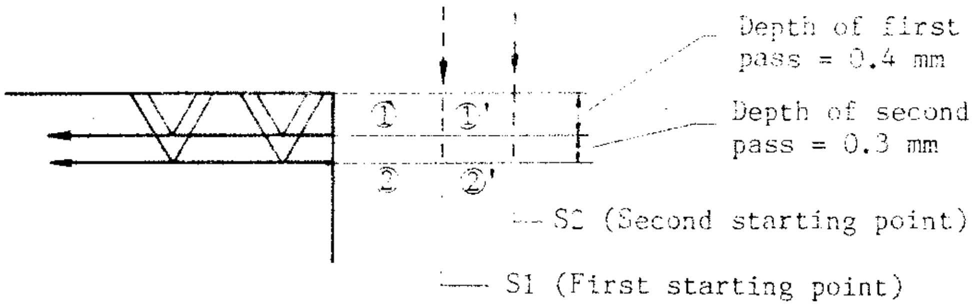
-Program -

```
      .
      .
G97 S900 M03 ;
G00 X1.21 ;
G32 W-2.1 F.2 ;   ........................ Tool path ①
G00 X2.0 ;
    W2.2 ;
    X1.21 ;
G32 W-2.2 ;  ............................. Tool path ①'
G00 X2.0 ;
    W2.1 ;
    X1.18 ;
G32 W-2.1 ;  ............................. Tool path ②
G00 X2.0 ;
    W2.2 ;
    X1.18 ;
G32 W-2.2 ;  ............................. Tool path ②'
    .
    .
```

Metric



Thread lead : 5 mm ..... 2 starts thread
δ 1 : 9.5 mm
δ 2 : 2.7 mm
Spindle speed : 900 rpm



- Program -

```
      .
      .
G97 S900 M03 ;
G00 U-20.8 ;
G32 W-44.7 F5.0 ; .................... Tool path ①
G00 X50.0 ;
    W47.2 ;
    U-20.8 ;
G32 W-47.2 ; ......................... Tool path ①'
G00 X50.0 ;
    W44.7 ;
    U-21.4 ;
G32 W-44.7 ; ......................... Tool path ②
G00 X50.0 ;
    W47.2 ;
    U-21.4 ;
G32 W-47.2 ; ......................... Tool path ②'
G00 X50.0 ;
    W44.7 ;
      .
      .
      .
      .
```

[11] Other G-codes

Other G-codes shown below are standard on the BND-20S, BND-34S.

(1) Multiple repetitive cycles

For repetitive machining peculiar to turning, such as metal removing in rough cutting, a series of paths that is specified usually in three or four blocks, can be specified in one block by using multiple repetitive cycle.
The following list shows various multiple repetitive cycles.

G-code	Function
G70	Finish Cutting Cycle
G71	Stock Removal Cutting Cycle in Turning
G72	Stock Removal Cutting Cycle in Facing
G73	Pattern Repeated Cutting Cycle
G74	Longitudinal Pecking Cycle
G75	Cross Pecking Cycle
G76	Thread Cutting Cycle

(2) Canned Cycle

There are three kinds of canned cycles as follows ;

G-code	Function
G90	Turning Cycle
G92	Thread Cutting Cycle
G94	Facing Cycle

(3) Tool nose radius compensation

The compensation mode can be programmed by using the following G-codes.

G-code	Function
G40	Tool Nose Radius Compensation Cancel
G41	Tool Nose Radius Compensation Left
G42	" Right

(4) Constant surface speed control and feed rate per minute or per revolution.

G-code	Function
G96	Constant Surface Speed Control
G97	Constant Surface Speed Control Cancel
G98	Feed Rate Per Minute
G99	Feed Rate Per Revolution

[12] List of G-codes and Their Group

G-code	Function	Group
G00	Positioning	01 *
G01	Linear Interpolation and Cornering	01
G02	Circular Interpolation, Clockwise	01
G03	Circular Interpolation, Counterclockwise	01
G04	Dwell	00
G10	Program Input of Offset Value (Optional)	00
G20	Inch Input	06
G21	Metric Input	06
G27	Reference point Return Check	00
G28	Automatic return to reference point	00
G32	Thread Cutting	01
G40	Tool Nose Radius Compensation Cancel	07 *
G41	Tool Nose Radius Compensation Left	07
G42	Tool Nose Radius Compensation Right	07
G50	Maximum Spindle Speed Setting (RPM)	00
G70	Finish Cutting Cycle	00
G71	Stock Removal Cutting Cycle in Turning	00
G72	Stock Removal Cutting Cycle in Facing	00
G73	Pattern Repeated Cutting Cycle	00
G74	Longitudinal Pecking Cycle	00
G75	Cross Pecking Cycle	00
G76	Thread Cutting Cycle	00
G90	Turning Cycle	01
G92	Thread Cutting	01
G94	Facing Cycle	01
G96	Constant Surface Speed Control	02
G97	Constant Surface Speed Control Cancel	02 *
G98	Feed Rate Per Minute	05
G99	Feed Rate Per Revolution	05 *

(NOTE 1)

The make '' in the list of G-codes shows the initial state when the power is turned on.*

(NOTE 2)

The G codes in the group 00 are not modal. They are effective only in the block in which they are specified.

(NOTE 3)

An alarm occurs when a G code not listed in the above table is specified (No. 010).

(NOTE 4)

A number of G codes can be specified in a block even if they do not belong to the same group. When a number of G codes of the same group are specified, the G code specified last is effective.

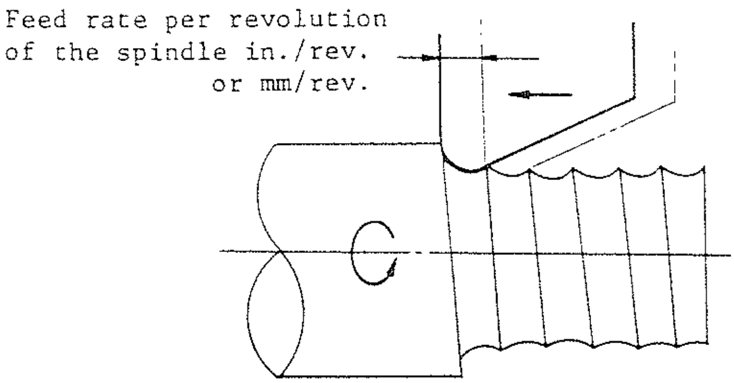
BND-S PRO 0-TC 90/8

2-5 F - Function (Feed Function)

The feed function directly designates the amount of feed rate per revolution of the spindle or per minute by using 7-digits number following the address F.
When the power is on, the initial state is the same as specifying feed rate per revolution of the spindle (G99).
Therefore, when specifying feed rate per minute, G98 must be specified before F code is designated.
Because the F code is modal, the F code is effective until the other F code is commanded. Therefore, when changing from G98 to G99 or vice versa, F-code must be designated again.

[1] Feed rate per revolution of the spindle (G99)

While G99 is selected, the feed rate per revolution of the spindle is designated by 6-digits (Metric) or 7-digits (Inch) number following the address F.



(1) Example commands of feed rate

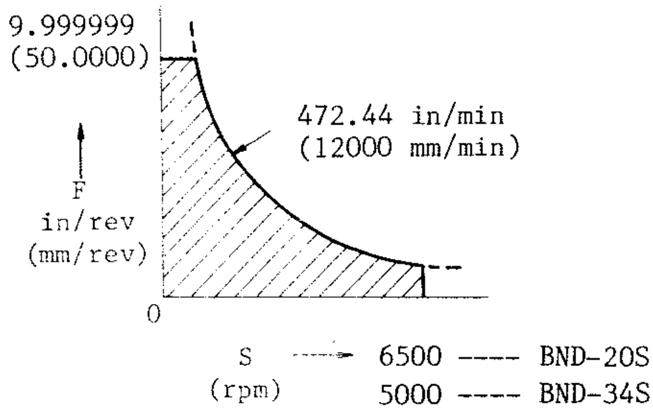
Command	Inch	Metric
F1234	.001234 in/rev.	0.1234 mm/rev.
F34	.000034 in/rev.	0.0034 mm/rev.

(2) Range of feed rate

.000001 to 9.999999 in/rev. (0.0001 to 50.0000 mm/rev.)
Range of feed rate is limited by selection of spindle speed as follows;

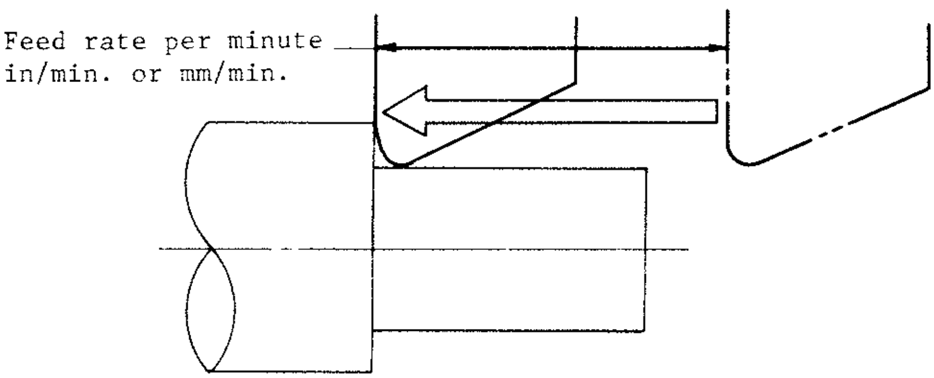
$$\underset{\substack{\text{in/rev.}\\ \text{(mm/rev.)}}}{F} \times \underset{\text{rpm}}{S} \leq 472.44 \text{ in/min. (12000 mm/min).}$$

Limitation of Feed rate (F) and Spindle speed (S)



[2] Feed rate per minute (G98)

While G98 is selected, the feed rate per minute is designated by 4-digits (Metric) or 5-digits (Inch) number following the address F.



(1) Example commands of feed rate

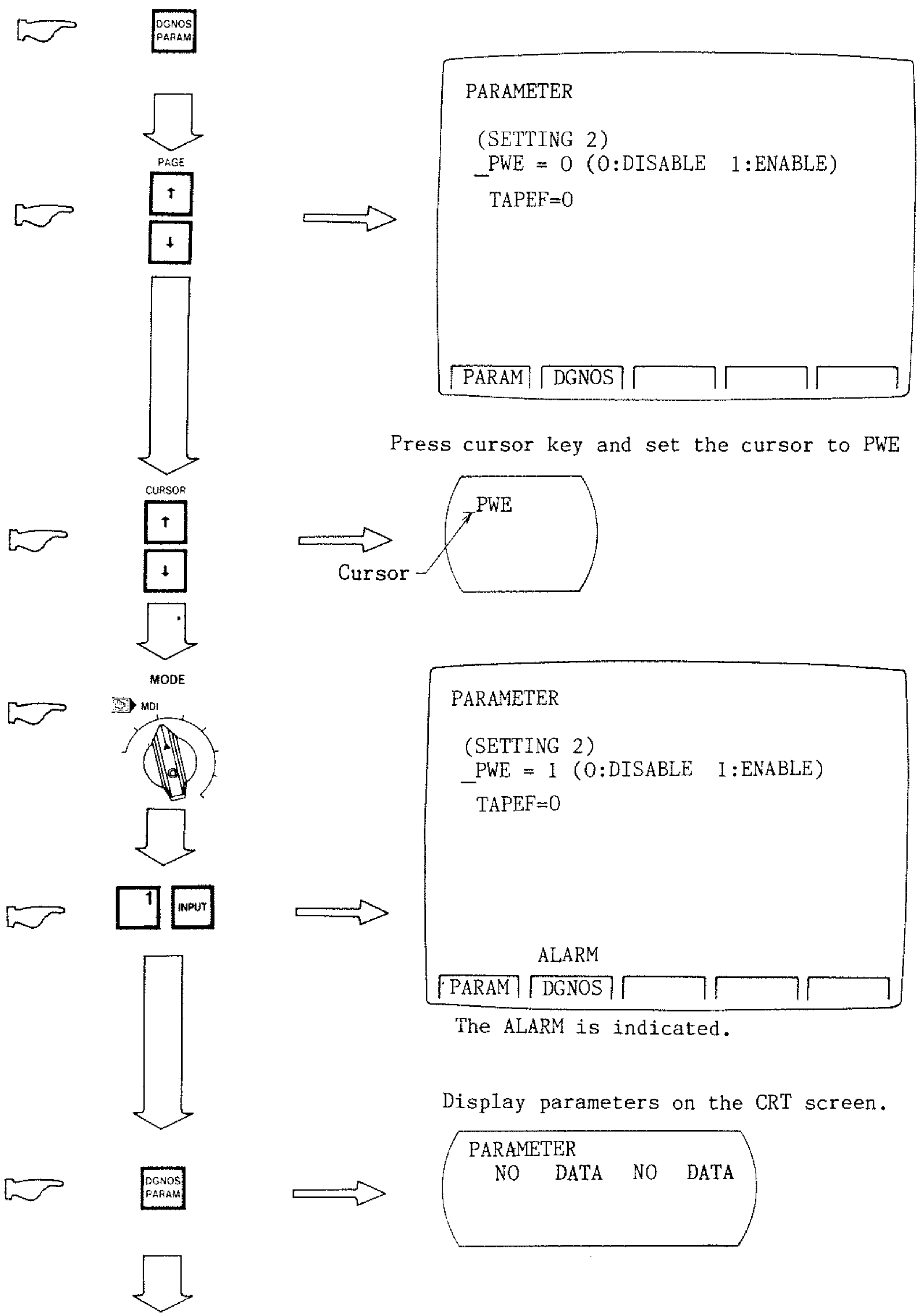
Command	Inch	Metric
F1234	12.34 in/min.	1234 mm/min.
F34	.34 in/min.	34 mm/min.

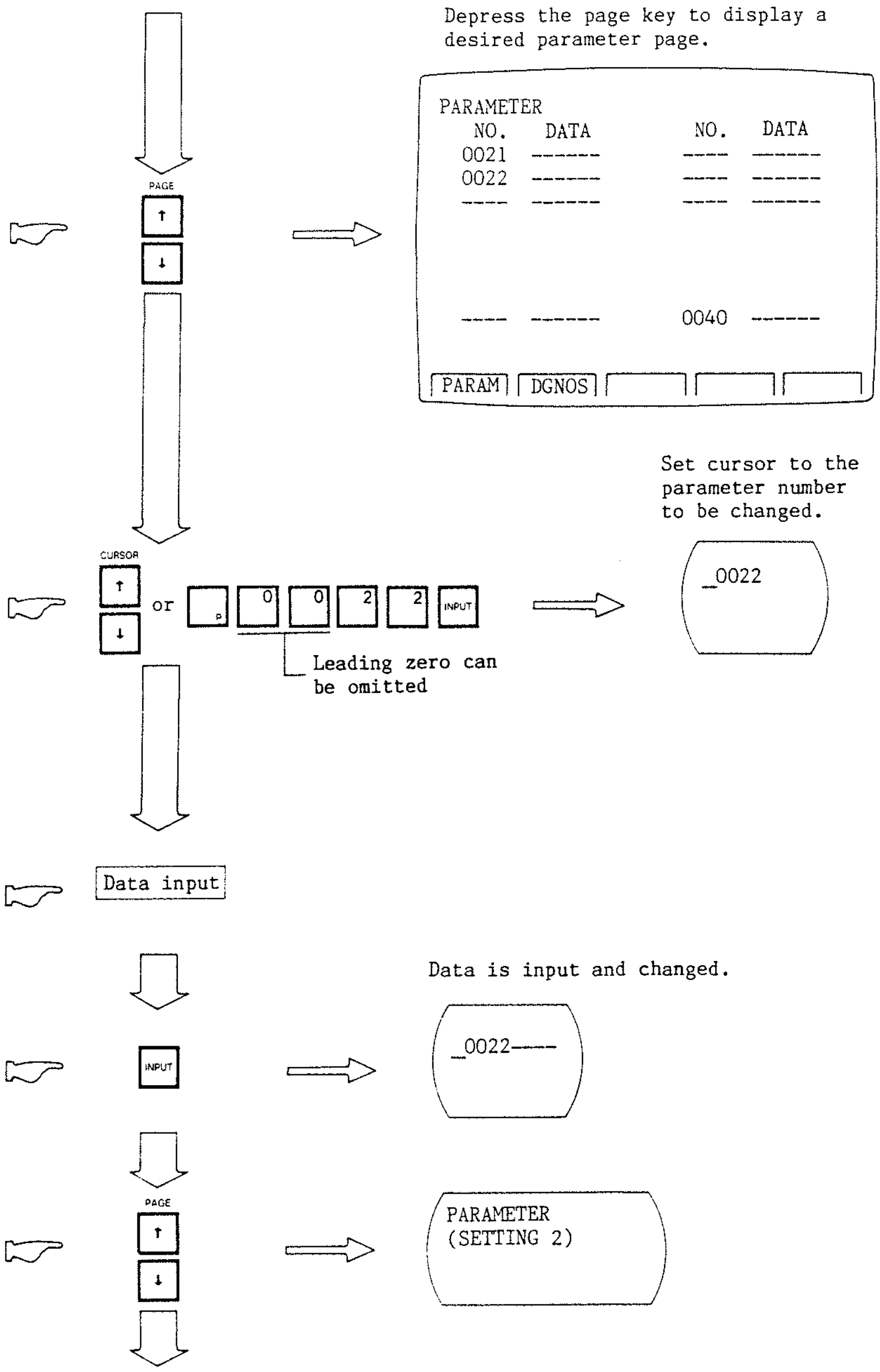
(2) Range of feed rate

0 to 472.44 in/min
(1 to 12000 mm/min)

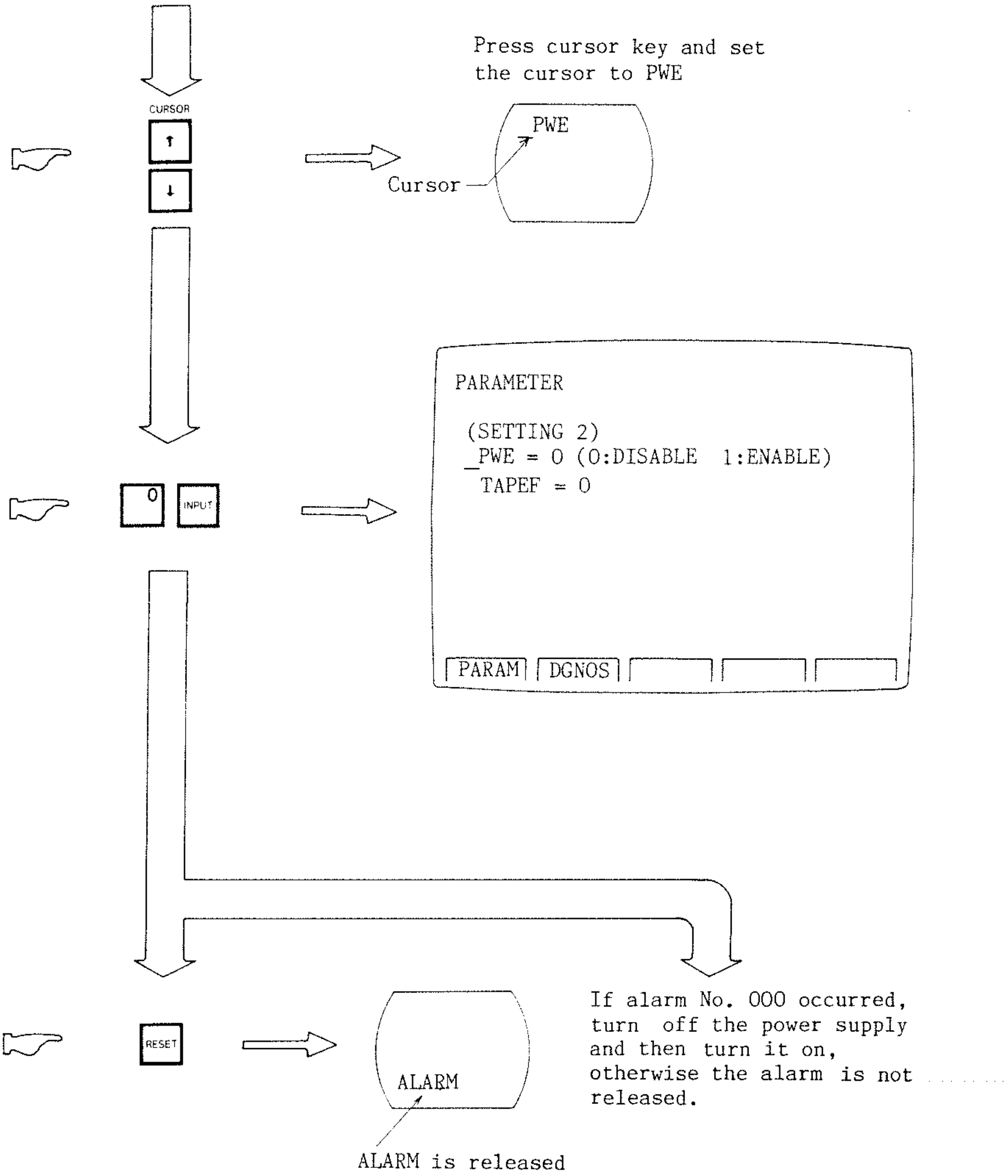
BND-S PRO 0-TC 90/8

[2] Setting and changing parameter

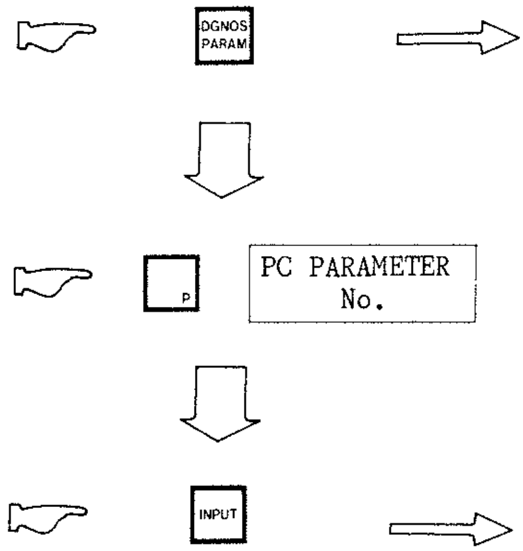




BND-S OTC OPE 90/7


CURSOR
↑
↓
Press cursor key and set
the cursor to PWE
PWE
Cursor
0
INPUT
PARAMETER
(SETTING 2)
_PWE = 0 (0:DISABLE 1:ENABLE)
TAPEF = 0
PARAM
DGNOS
RESET
ALARM
ALARM is released
If alarm No. 000 occurred,
turn off the power supply
and then turn it on,
otherwise the alarm is not
released.

[3] Display of PC parameter and PC timer



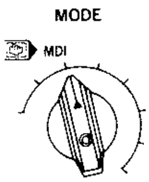
A screen of diagnosis is displayed.

DIAGNOSTIC			00001 N1005
NO.	DATA	NO.	DATA
300	00000000	310	00000000
301	00000001	311	00100000
302	00000000	312	00110000
303	00000000	313	00000000
304	00000010	314	00000000
305	00000000	315	00000010
306	00000000	316	00000011
307	00000000	317	00000000
308	00000010	318	00110000
309	00001000	319	00100000

[PARAM] [DGNOS] [ ] [ ] [ ]

[4] Setting and changing PC parameter

Call out PC parameter
number to be altered



Data input

INPUT

DIAGNOSTIC   O0001 N1005

NO.	DATA	NO.	DATA
0000	00000000	0010	00000000
_0001	00000000	0011	00100000
0002	00000000	0012	00110000
0003	00000000	0013	00000000
0004	00000010	0014	00000000
0005	00000000	0015	00000100
0006	00000000	0016	00000011
0007	00000000	0017	00000000
0008	00000010	0018	00110000
0009	00001000	0019	00100000

[PARAM] [DGNOS] [ ] [ ] [ ]

4-22 Graphic Function

On the CRT display, the programmed tool path can be drawn while being cut. Therefore, the progress condition of cutting can be checked by observing the path on the CRT display. Besides, to scale the screen is also possible.

[1] Setting of graphic parameter

AUX GRAPH

G.PRM

G.PRM

```
GRAPHIC PARAMETER

  WORK LENGTH       W =
  WORK DIAMETER     D =

  PROGRAM STOP      N =
  AUTO ERASE        A =
  LIMIT             L =

  GRAPHIC CENTER    X =
                    Z =
  SCALE             S =

[      ][      ][ ZOOM ][      ][      ]
```

WORK LENGTH : Set the length of the workpiece at .0001" (0.001 mm)

WORK DIAMETER : Set the diameter of the workpiece at .0001" (0.001 mm)

PROGRAM STOP : When desiring partial display of the program, the end of sequence number should be set. After being allowed to draw once this value is canceled automatically.

AUTO ERASE : In the case of setting 1, first, when starting to operate automatically under the reset status, an old drawing is erased automatically.

GRAPHIC CENTER & SCALE : No need to always set because they are calculated automatically.

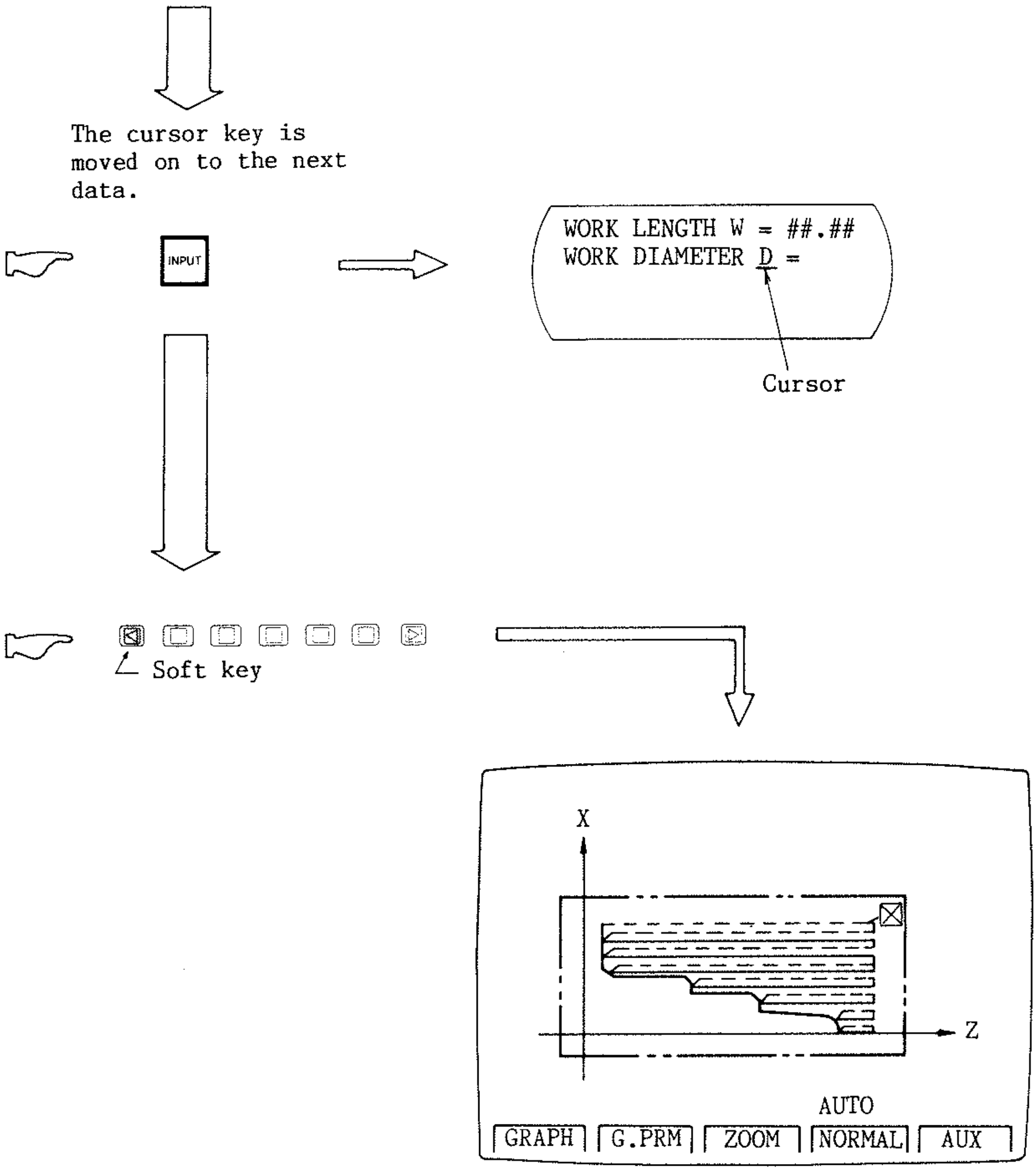
CURSOR ↑ ↓

WORK LENGTH W=

Cursor

Data input

Cont. to next page



(Note 1)

It is necessary to start the program by automatic operation. Because drawings are done due to renewal of coordinate values via automatic operation.
When executing drawing only, select MACHINE LOCK.

(Note 2)

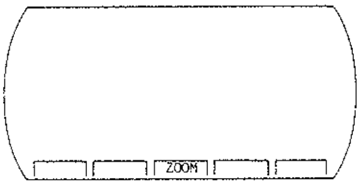
When the feeding speed is too high, in some cases, it is impossible to draw correctly. In these cases, reduce the speed by dry-run, etc., and then allow drawing to be done again.

(Note 3)

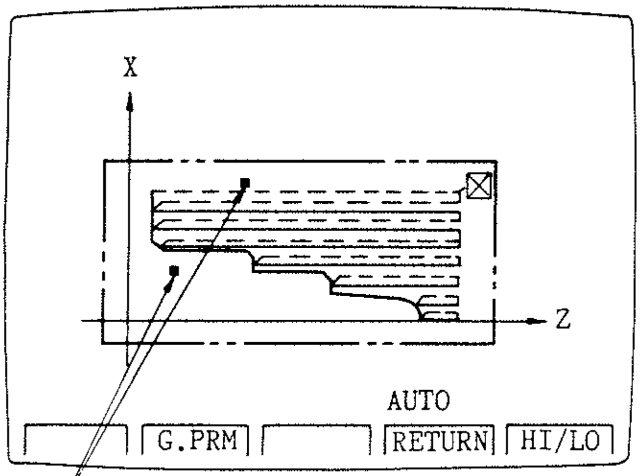
Coordinate zero point is marked with ⊠ .

[2] Magnification of screen

The drawn screen can be partially magnified.



ZOOM



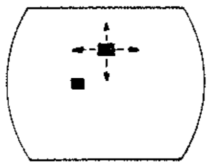
ZOOM Cursor

Shifting between upper and lower zoom cursor

HI/LO

↑8 N ←4 X →6 F ↓2 W

Movement of zoom cursor

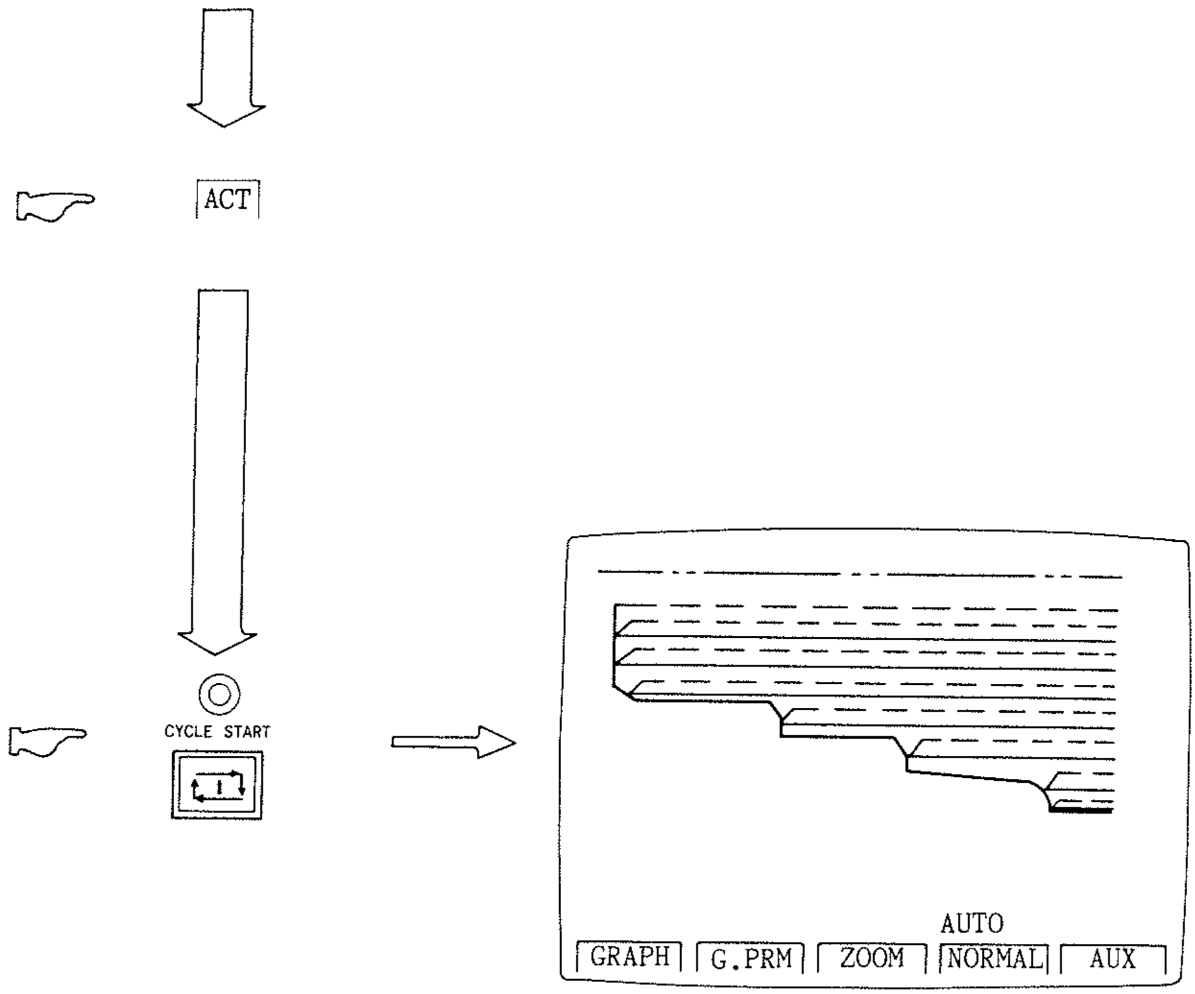


Move to the position surrounding a section of cursor to be enlarged

Cont. to next page

BND-S OTC OPE 90/7

From previous page



* When desired to return to the original scale, start the automatic operation after depressing the soft key [NORMAL].

4-23 Conversational Programming with Graphic Function (Option)

[1] Programming

PRGRM

In case where other programs are not registered

```
PROGRAM
 _

[PRGRM] [     ] [     ] [     ] [ CAP ]
```

Currently selected program is displayed where other programs are already registered

```
PROGRAM
O0001 ;
N100 G00 T0100 ;
N200 G00 X100. Z50. ;
        |
        |

[PRGRM] [     ] [     ] [     ] [ CAP ]
```

Input new program number

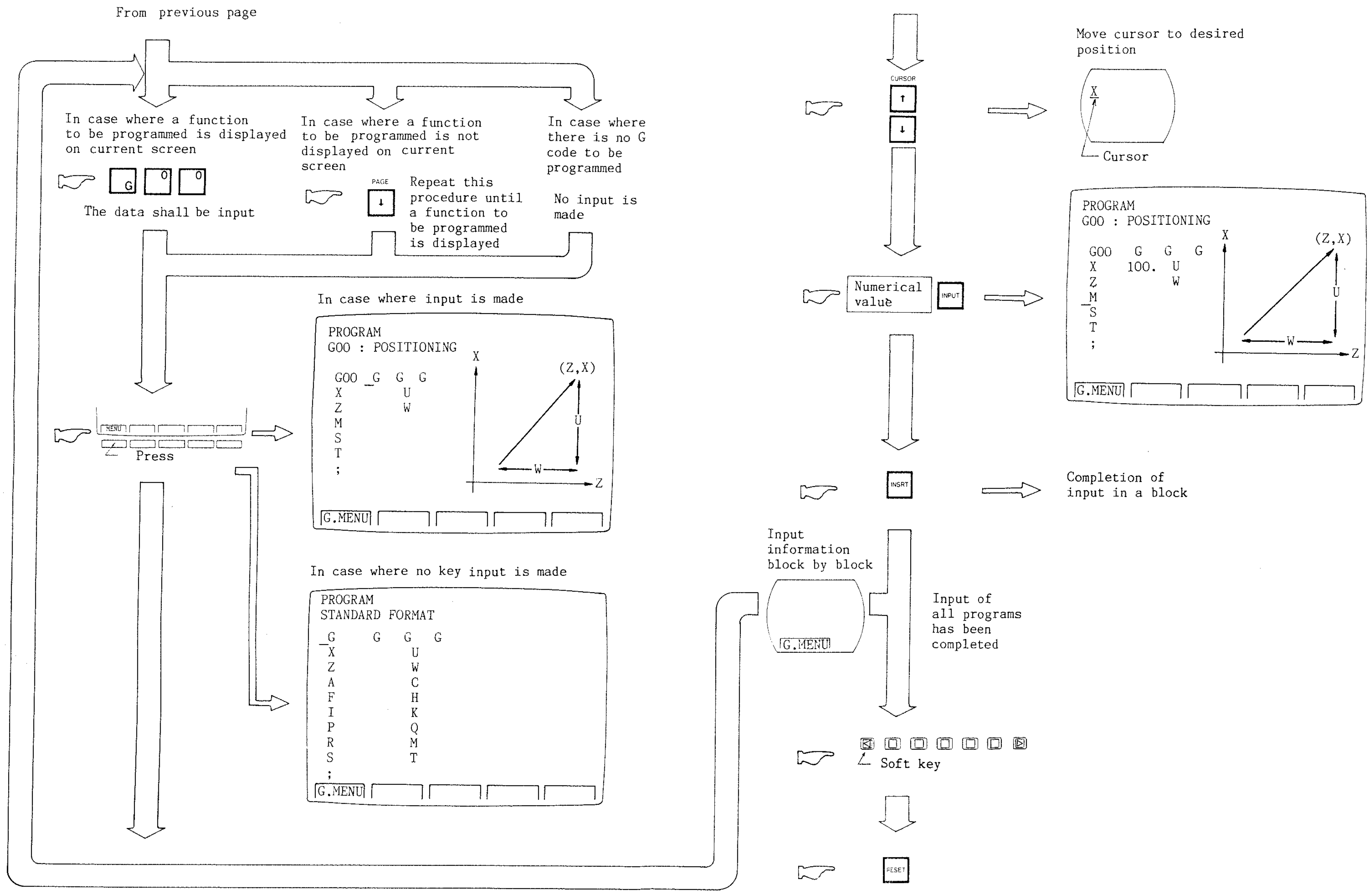
O Numeral EOB

TAP

Press

```
PROGRAM
G00 : POSITIONING
G01 : LINEAR IPL.
G02 : CIRCULAR IPL. CW
G03 : CIRCULAR IPL. CCW
G04 : DWELL
G10 : OFFSET VALUE SETTING <O>
G20 : INCH
G21 : METRIC
G25 : SPINDLE SPEED DETECT OFF
G26 : SPINDLE SPEED DETECT ON
G27 : REF. POINT RETURN CHECK
G28 : REF. POINT RETURN

[ MENU ] [     ] [     ] [     ] [     ]
```

Cont. to next page

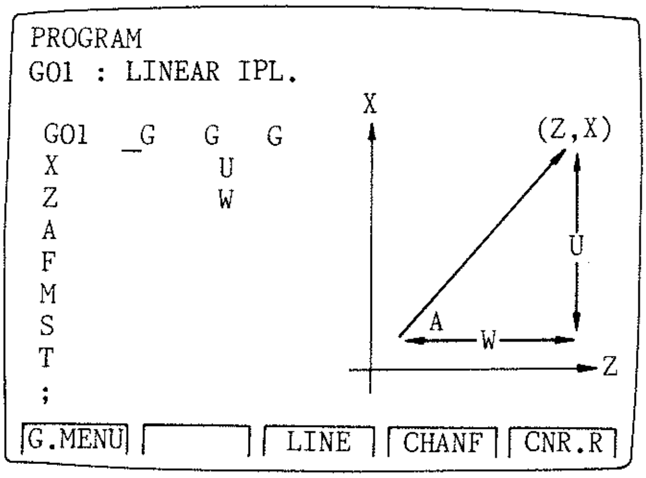


From previous page
In case where a function to be programmed is displayed on current screen
G 0 0
The data shall be input
In case where a function to be programmed is not displayed on current screen
PAGE
Repeat this procedure until a function to be programmed is displayed
In case where there is no G code to be programmed
No input is made
Press
In case where input is made
PROGRAM
G00 : POSITIONING
G00 G G G
X U
Z W
M
S
T
;
X
(Z,X)
U
W
Z
G.MENU
In case where no key input is made
PROGRAM
STANDARD FORMAT
G G G G
X U
Z W
A C
F H
I K
P Q
R M
S T
;
G.MENU
CURSOR
Move cursor to desired position
X
Cursor
Numerical value
INPUT
PROGRAM
G00 : POSITIONING
G00 G G G
X 100. U
Z W
M
S
T
;
X
(Z,X)
U
W
Z
G.MENU
INSRT
Completion of input in a block
Input information block by block
G.MENU
Input of all programs has been completed
Soft key
RESET

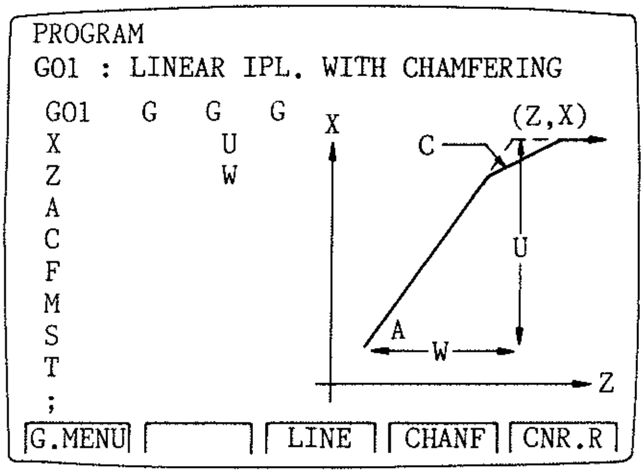
[2] Program for chamfering and corner radius.

When linear interpolation is programmed, block of chamfering and corner radius can be programmed at the same time.

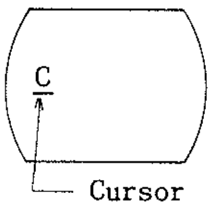
Screen display of linear interpolation



☞ CHANF or CNR.R ⇒



☞ CURSOR ↑ ↓ ⇒



* Move a cursor to "R" in case of corner radius

☞ Numerical value INPUT

☞ INSRT

[3] Method of using multiple repetitive cycle (G70 to G73)

Set sequence of setting data to 1

⇩

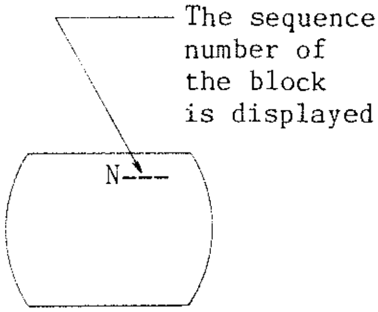
Set 1 in parameter No. 550

⇩

All blocks of finished shape program shall be programmed on the conversational screen

⇩

Last block of finished shape program shall be displayed on the conversational screen ⇨



⇩

Make a memo of the sequence number of last block

⇩

Search the first block of finished shape program

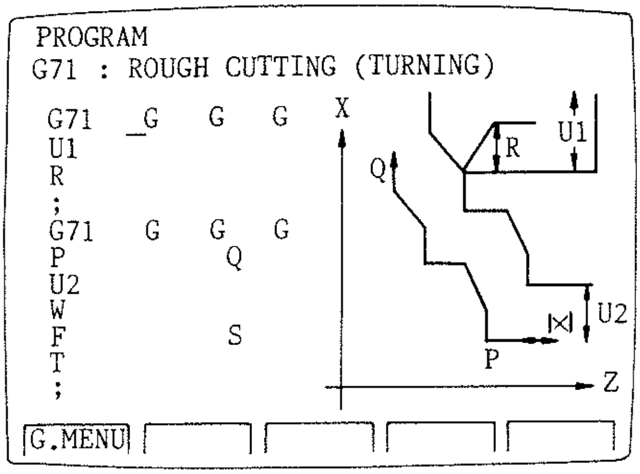
PAGE ↑

⇩

Make a memo of the sequence number of first block

⇩

Display the blocks of G70 to G73 containing information already programmed

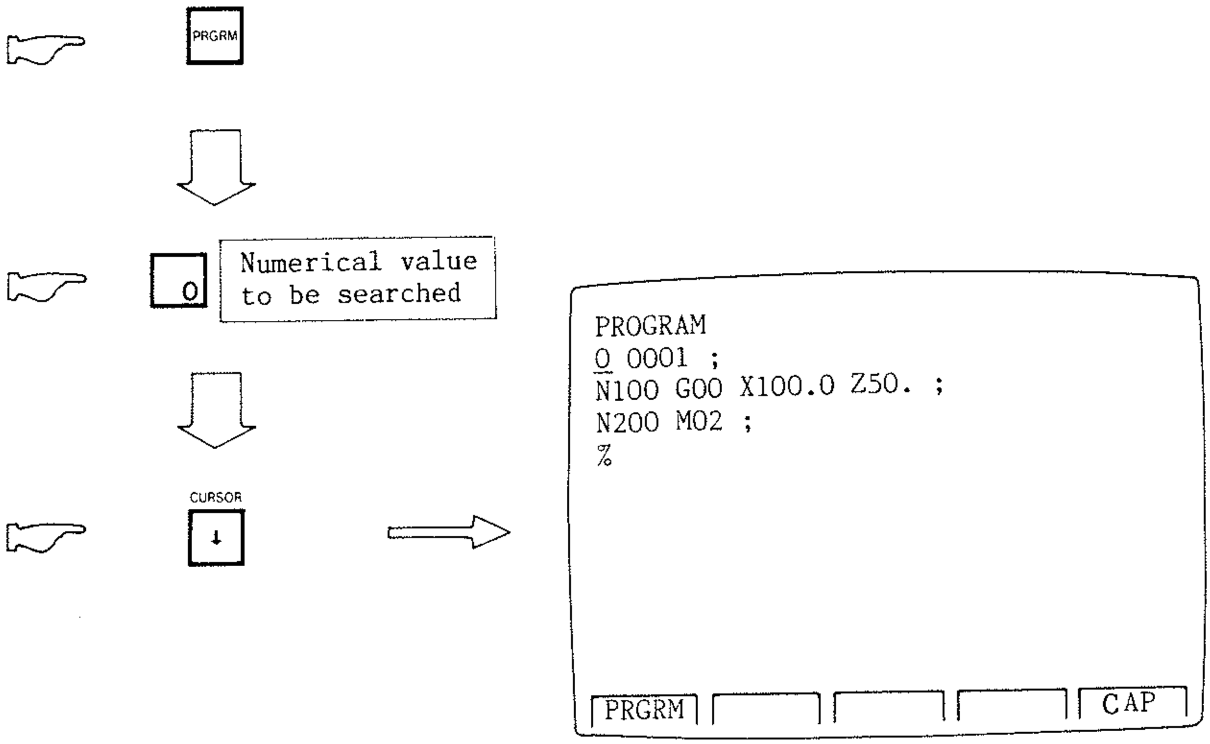


Input values of P and Q, into the sequence number which previously made

ALTER

[4] Confirmation of program

(1) In case where display is made by conventional format



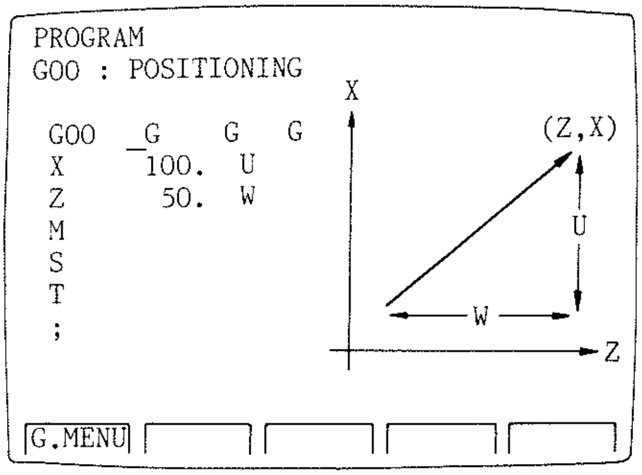
(2) In case where display is made on conversational screen

Move cursor to
the head of block
to be confirmed

↓

Soft key

[CAP] ⟹



↓

PAGE [↑] ⟹ Prior block is displayed

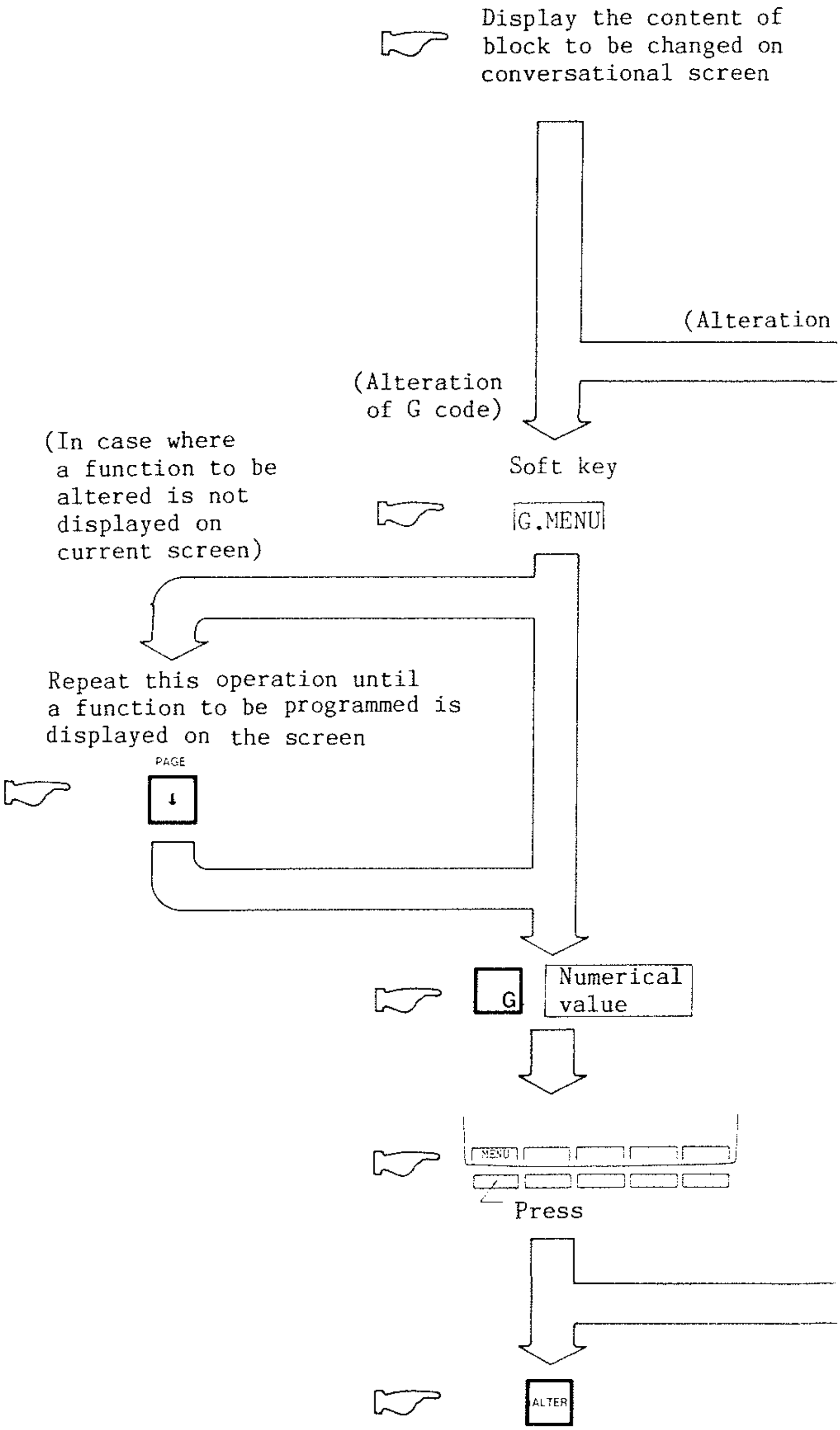
↓

PAGE [↓] ⟹ Subsequent block is displayed

[5] Editing of program

All editing on conversational screen shall be one block unit

(1) Alteration of block



(2) Inserting a block

IAP

Display the block previous to the place desired insertion

PAGE ↑ ↓

N005

N005---- ; N006---- ;

Insert location

Soft key "G.MENU"

```
PROGRAM
G00 : POSITIONING
G01 : LINEAR IPL.
G02 : CIRCULAR IPL. CW
G03 : CIRCULAR IPL. CCW
G04 : DWELL
G10 : OFFSET VALUE SETTING <O>
G20 : INCH
G21 : METRIC
G25 : SPINDLE SPEED DETECT OFF
G26 : SPINDLE SPEED DETECT ON
G27 : REF. POINT RETURN CHECK
G28 : REF. POINT RETURN

[MENU] [ ] [ ] [ ] [ ]
```

(In case where function to be programmed is displayed on current screen)

G 0 0

Input that data

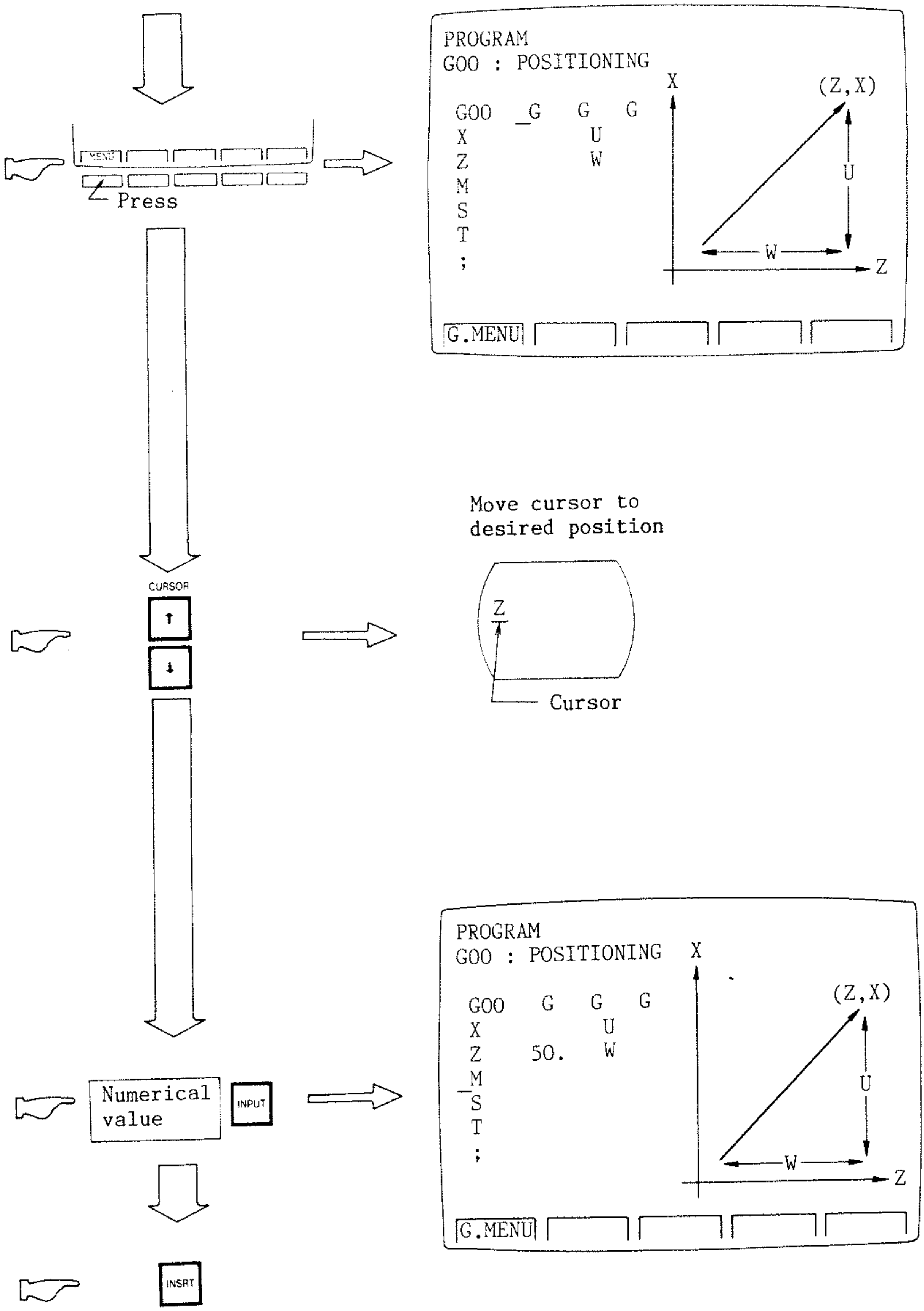
(In case where function to be programmed is not displayed on current screen.)

PAGE ↓

Repeat this operation until function to be programmed is displayed.

(In case where there is no G code to be programmed)

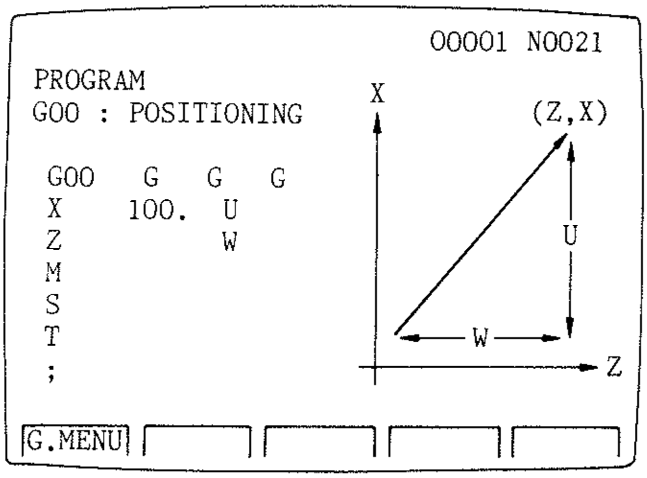
No key input is made.



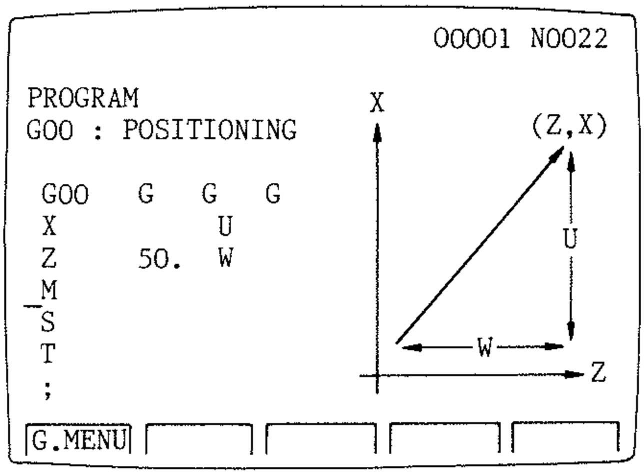
BND-S OTC OPE 90/7

(3) Deletion of block

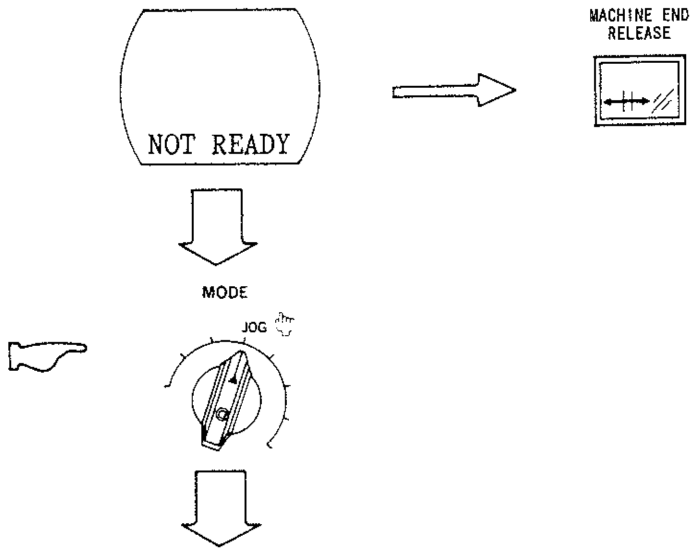
Display the content of block to be deleted shall be displayed on conversational screen.



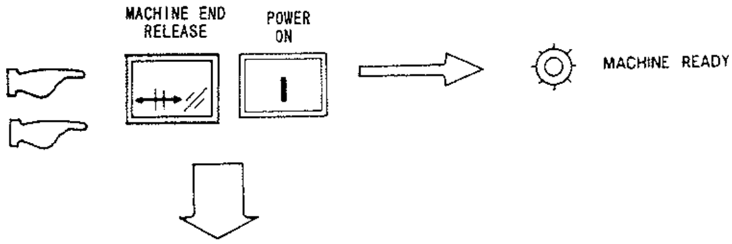
DELET



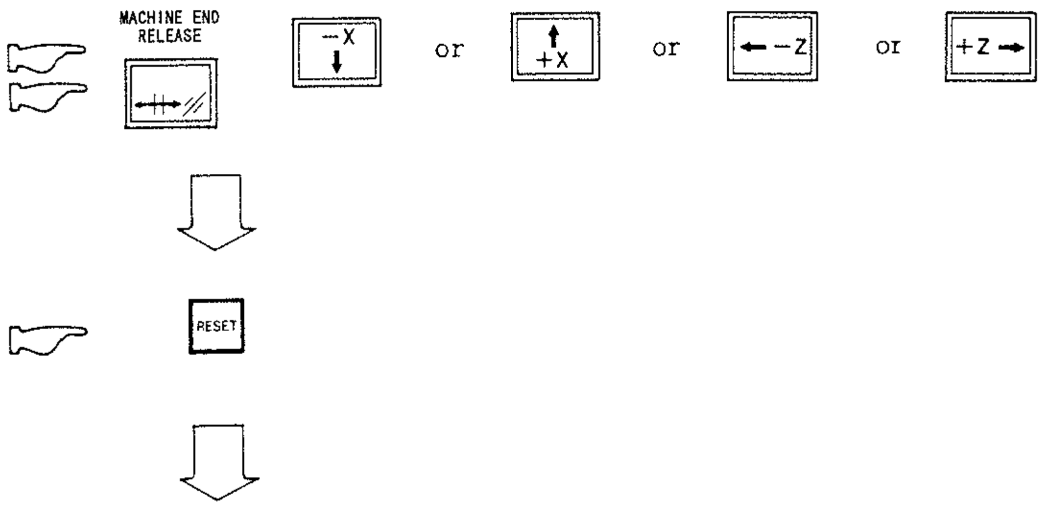
4-24 When Slide Activated Machine End Micro Switch



Depress POWER ON button while
depressing MACHINE END RELEASE button.



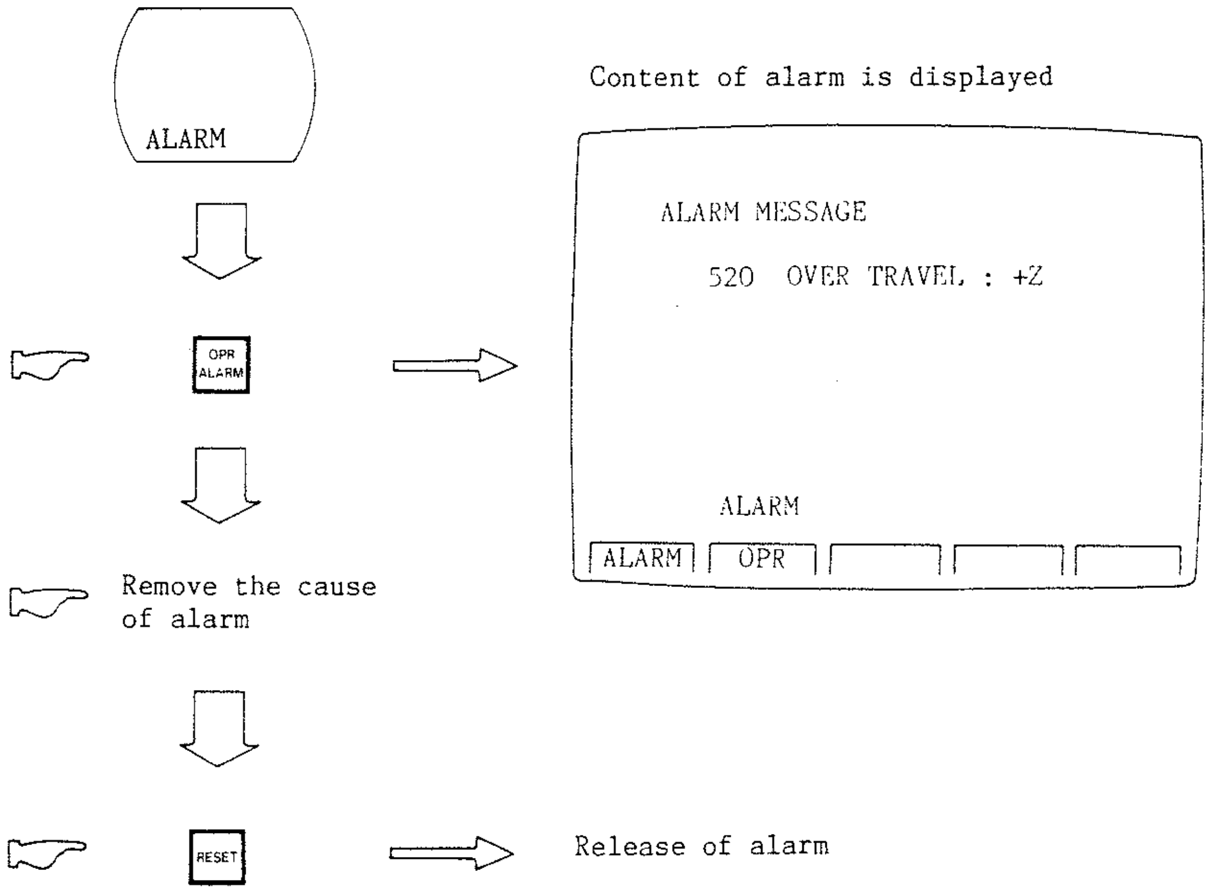
Move slide away from its overtravelled
position to safer position.



Perform manual
zero return.

4-25 Alarm Display

The machine is always diagnosing by itself. When an error is detected, 'ALARM' will be flickered in the CRT display unit and the machine will stop automatically.



(NOTE)

When an alarm occurs, the alarm message is usually automatically displayed.

[1] Alarm list

(1) Program errors (P/S alarm)

Number	Content
000	A parameter which requires the power off was input, turn off power.
001	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective. Correct the tape.
003	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	A numeral or the sign "-" was input without an address at the beginning of a block.
005	The address was not followed by the appropriate data but was followed by another address or EOB code.
006	Sign "-" input error (Sign "-" was input after an address with which it cannot be used. Or two or more "-" signs were input).
007	Decimal point "." input error (A decimal point was input after an address with which it cannot be used. Or two decimal points were input.)
009	Unusable character was input in significant area.
010	An unusable G code was commanded.
011	Feedrate was not commanded to a cutting feed or the feedrate was inadequate.
014	In variable lead threading, lead increasing or decreasing values commanded by address K exceed the maximum command value. Or, the command is made so that the lead becomes a negative value.
029	The offset value specified by T code is too large.
030	The offset number in T function specified for tool offset is too large.
031	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified.
032	In setting an offset amount by G10, the offset amount was excessive.

BND-S OTC OPE 90/7

033	A point of intersection cannot be determined for tool nose radius compensation.
034	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation.
035	Skip cutting (G31) was specified in tool nose radius compensation mode.
038	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center.
039	Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R.
040	Overcutting will occur in tool nose radius compensation in a canned cycle G90 or G94.
041	Overcutting will occur in tool nose radius compensation.
050	The chamfering or a corner R was specified in a block which includes a thread cutting command.
051	The block after a block containing a chamfering or a corner R specification was not a G01 command.
052	The move direction or the move amount in a block following chamfering or a corner R command was not adequate.
053	In the command of chamfer and corner R, more than two of I, K and R are commanded. Or, in the direct input of drawing dimensions, C or R is not located after a comma(,).
054	A block in which the chamfering or the corner R was specified includes a taper command.
055	The move distance in the block which includes the chamfering or the corner R specification is smaller than the chamfering amount or the corner R.
056	In the block command next to the block of angle designation only (A-), both of the angle designation of a terminal point and the designation of an angle are not placed. I(K) is commanded to X-axis(Z-axis) by the command for chamfer.
057	In the direct input drawing dimensions, a terminal point was not caluculated correctly.
058	In the direct input of drawing dimensions, a terminal point of the block was not found.
059	The program with the selected number cannot searched, in external program number search.
060	Commanded sequence number was not found in the sequence number search.

061	Address P or Q is not specified in G70, G71, G72, or G73 command.
062	o The depth of cut in G71 or G72 is zero or negative value. o The repetitive count in G73 is zero or negative value. o The negative value is specified to Δi or Δk in G74 or G75. o A value other than zero is specified to address U or W, though Δi or Δk is zero in G74 or G75. o A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. o Zero or a negative value is specified to the height of thread or depth of cut of 1st time in G76. o The specified minimum depth of cut in G76 is greater than the height of thread. o An unusable angle of tool tip is specified in G76.
063	The sequence number specified by address P in G70, G71, G72 or G73 command cannot be searched.
065	o G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. o Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively.
066	An unallowable G code was commanded between two blocks specified by address P and Q in G71, G72 or G73.
067	G70, G71, G72, or G73 command with address P and Q was specified in MDI mode.
069	The final move command in the blocks specified by P and Q of G70, G71, G72 and G73 ended with chamfering of corner R.
070	The memory area is insufficient.
071	The address to be searched was not found. Or the program with specified program number was not found in program number search.
072	The number of programs to be stored exceeded 125.
073	The commanded program number has already been used.
074	The program number is other than 1 to 9999.
076	Address P was not commanded in the block which includes an M98 command or a G65 command.
077	The subprogram was called in three or five folds.
078	A program number or a sequence number which was specified by address P in the block which includes an M98, M99 or G66 was not found.
079	The contents of the program stored in the memory did not agree with that in tape in collation.
080	In the area specified by parameter ε, the measuring position reach signal does not come on. (Automatic tool compensation function)

081	Automatic tool compensation was specified without a T code. (Automatic tool compensation function)
082	T code and automatic tool compensation were specified in the same block. (Automatic tool compensation function)
083	In automatic tool compensation, an invalid axis was specified or the command is incremental. (Automatic tool compensation function)
085	When entering data in the memory by using ASR or Reader/Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate is incorrect.
086	When entering data in the memory by using Reader/Puncher interface, the ready signal (DR) of Reader/Puncher was turned off.
087	When entering data in the memory by using Reader/Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read.
090	The reference point return cannot be performed normally because the reference point return start point is too close to the reference point or the speed is too slow.
092	The commanded axis by G27 (Reference point return check) did not return to the reference point.
100	Setting data PWE is set to 1. Turn it to 0 and reset the system.
101	The power was turned off while rewriting the contents of the memory in the part program storage & editing operation. When this alarm is generated, set the setting data PWE to 1 and turn on the power while pushing the DELET to clear the memory.
110	An absolute value of the data of fixed decimal point indication exceeds an allowable range.
111	The calculation result of macro instruction exceeds the allowable range (-2^{32} to $2^{32}-1$).
112	Division by Zero was specified. (including tan 90°)
113	The function which cannot be used in the custom MACRO is commanded.
114	An undefined H code is designated in G65 block.
115	A value not defined as a variable number is designated.
116	The variable number designated with P is forbidden for assignment.
118	The multiple degree of parenthesis exceeds the upper limit (quintaple).
119	The argument of SQRT or BCD is negative.
123	In the DNC operation, the MACRO control command is used.

BND-S OTC OPE 90/7

124	DO-END does not correspond to each other.
125	In the block with G65, the unusable address was specified.
126	In the DOn, $1 \leqq n \leqq 3$ is not realized.
127	The NC command and the MACRO command exist at the same time.
128	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched.
130	In B-axis control, an axis control command by PMC was made during of control. Or, to the coutrary, cf control was desired to be made for an axis control from PMC.

(2) Servo alarms

Number	Content
400	X, Z axis overload signal turns on.
401	X, Z axis velocity control READY signal (VRDY) turns off.
404	Position control READY signal (PRDY) turns off, while velocity control READY signal (VRDY) does not turn off, or velocity control READY signal (VRDY) turns on, although READY signal (PRDY) is not turned on yet when turning on the power supply.
405	A position control system error. A reference point return failure due to a trouble in CNC or servo system. Start operation with the manual reference point return again.
410	In X-axis, a value of positional deviation amount during stop exceeds the set value.
411	The position deviation amount is larger than the set value during move or stop in X-axis.
413	The contents of the X-axis error register exceed ±32767, or the velocity command value of DA converter is outside a range of -8192 - +8191. This error is usually caused by various setting failure.
414	There is abnormality, in the digital servo series of X-axis. Detail of content is output in No.720 of Diagnosis.
415	An attempt was made to specify a velocity of exceeding 511875 detection unit/sec in X-axis. This error is usually caused by a CMR setting failure.
416	An error in the position detection system of X-axis pulse coder (break alarm).

417	This alarm occurs when any of the following condition of X-axis is realized. i) A value other than specified is set in the motor type of parameter 8120. ii) A correct value (111 or -111) is not set in the motor rotation direction of parameter 8122. iii) Incorrect value such as below zero is set in the speed feedback pulse number per motor revolution of parameter 8123. iv) Incorrect value such as below zero is set in the position feedback pulse number per revolution of motor of parameter 8124.
420	In Z-axis, a value of positional deviation amount during stop exceeds the set value.
421	The position deviation amount is larger than the set value during move or stop in Z-axis.
423	The position deviation amount exceeds ±32767 in Z-axis, or the velocity command value of DA converter is outside a range of -8192 - +8191. This error is usually caused by various setting failure.
424	There is abnormality of digital servo system of Z-axis. Detail of content is output in No.721 of Diagnosis.
425	An attempt was made to specify a velocity of exceeding 511875 detection unit/sec in Z-axis. This error is caused by a CMR setting failure.
426	An error of the position detection system of Z-axis pulse coder (break alarm).
427	This alarm occurs when any of the following condition of Z-axis is realized. i) A value other than specified is set in the motor type of parameter 8220. ii) A correct value (111 or -111) is not set in the motor rotation direction of parameter 8222. iii) An incorrect data such as below zero is set in the speed feedback pulse number per rotation of motor of parameter 8223. iv) An incorrect data such as below zero is set in the position feedback pulse number per rotation of motor of parameter 8224.
430	In B-axis, a value of positional deviation amount during stop exceeds the set value.

431	The position deviation amount is larger than the set value during move or stop in B-axis.
433	The position deviation amount exceeds ±32767 in B-axis, or the velocity command value of DA converter is outside a range of -8192 - +8191. This error is usually caused by various setting failure.
434	There is abnormality of digital servo system of B-axis. Detail of content is output in No.722 of Diagnosis.
435	An attempt was made to specify a velocity of exceeding 511875 detection unit/sec in B-axis. This error is caused by a CMR setting failure.
436	An error of the position detection system of B-axis pulse coder (break alarm).
437	This alarm occurs when any of the following condition of B-axis is realized. i) A value other than specified is set in the motor type of parameter 8320. ii) A correct value (111 or -111) is not set in the motor rotation direction of parameter 8322. iii) An incorrect data such as below zero is set in the speed feedback pulse number per rotation of motor of parameter 8323. iv) An incorrect data such as below zero is set in the position feedback pulse number per rotation of motor of parameter 8324.

BND-S OTC OPE 90/7

(3) Overtravel alarms

Number	Content
510	Overtravel to exceed the (+) stroke limit of X-axis
511	Overtravel to exceed the (-) stroke limit of X-axis
520	Overtravel to exceed the (+) stroke limit of Z-axis
521	Overtravel to exceed the (-) stroke limit of Z-axis
530	Overtravel to exceed the (+) stroke limit of B-axis
531	Overtravel to exceed the (-) stroke limit of B-axis

(4) Overheat alarm

Number	Content
700	Master PCB is overheated.
704	This is overheat of the spindle due to the detection of spindle variation.

(5) System alarm

Number	Content
910	RAM parity error (low byte). Replace master PCB.
911	RAM parity error (high byte). Replace master PCB.
912	Common RAM with digital servo parity error. (LOW)
913	Common RAM with digital servo parity error. (LIGHT)
914	Local RAM of digital servo parity error.
920	Watch dog alarm. Replace master PCB.
930	CPU error (abnormal interrupt). Replace master PCB.
940	This alarm occurs when either of following conditions is realized. i) A printed board of digital servo system is defective. ii) When there is more than three control axes, B-axis control printed board is not provided. iii) A master printed board for analogue servo is used.
950	This is broken fuse alarm. Replace the fuse. (+24E; FX14)
998	ROM parity error.

BND-S OTC OPE 90/7

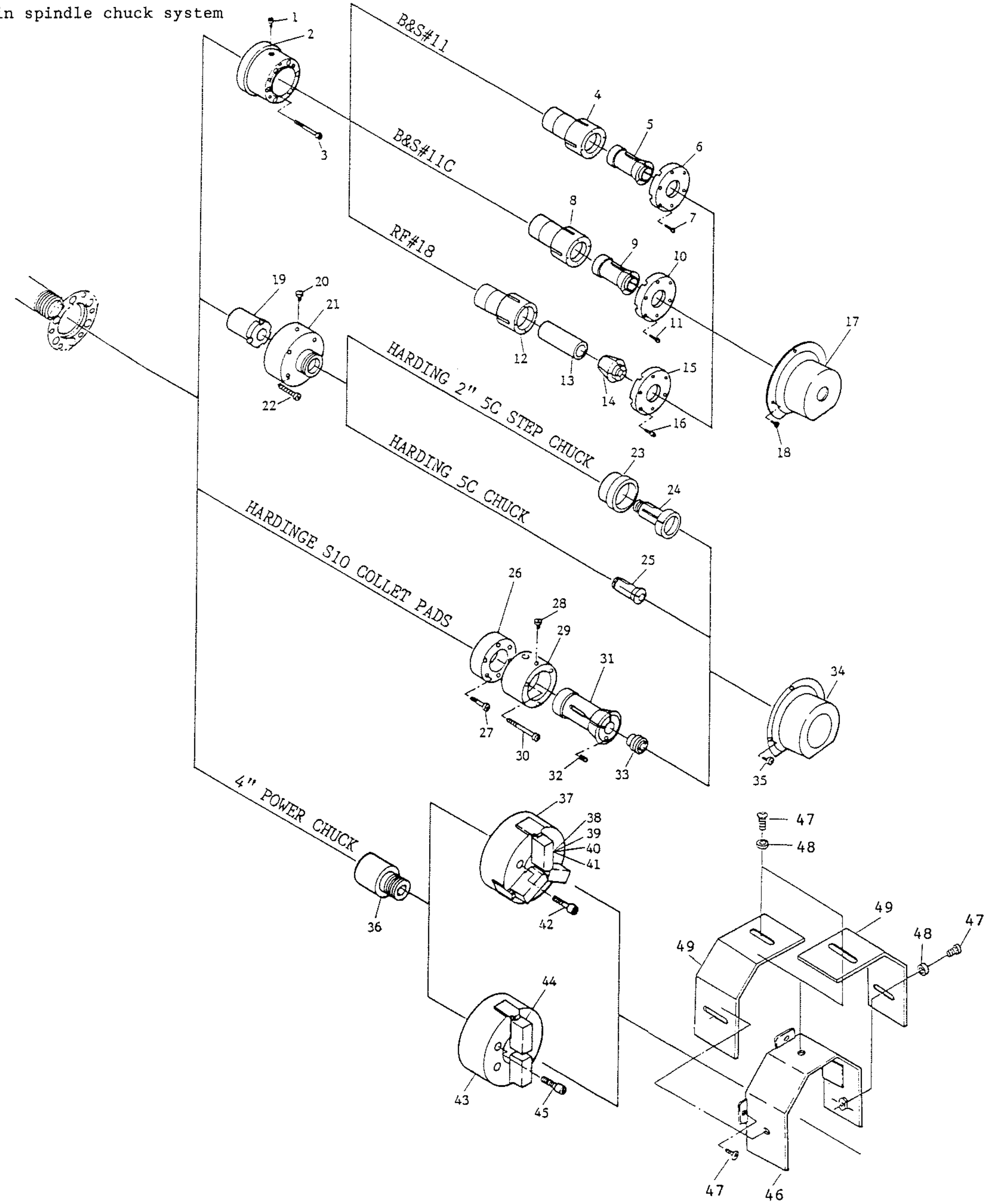
(6) Background edit alarm (BP/S)

Number	Content
???	BP/S alarm occurs in the same number as P/S alarm which occurs in the normal program edit.
140	The program which is selected in the foreground is desired to be selected or deleted.

Note) An alarm in the background edit is not a screen of normal alarm and displayed in the key input line of the background edit screen and can be reset by MDI key operations described in P555 to 557 of FANUC OPERATOR'S MANUAL.

5. MAIN SPINDLE CHUCK SYSTEM

[1] BND-20S Main spindle chuck system
(for USA)

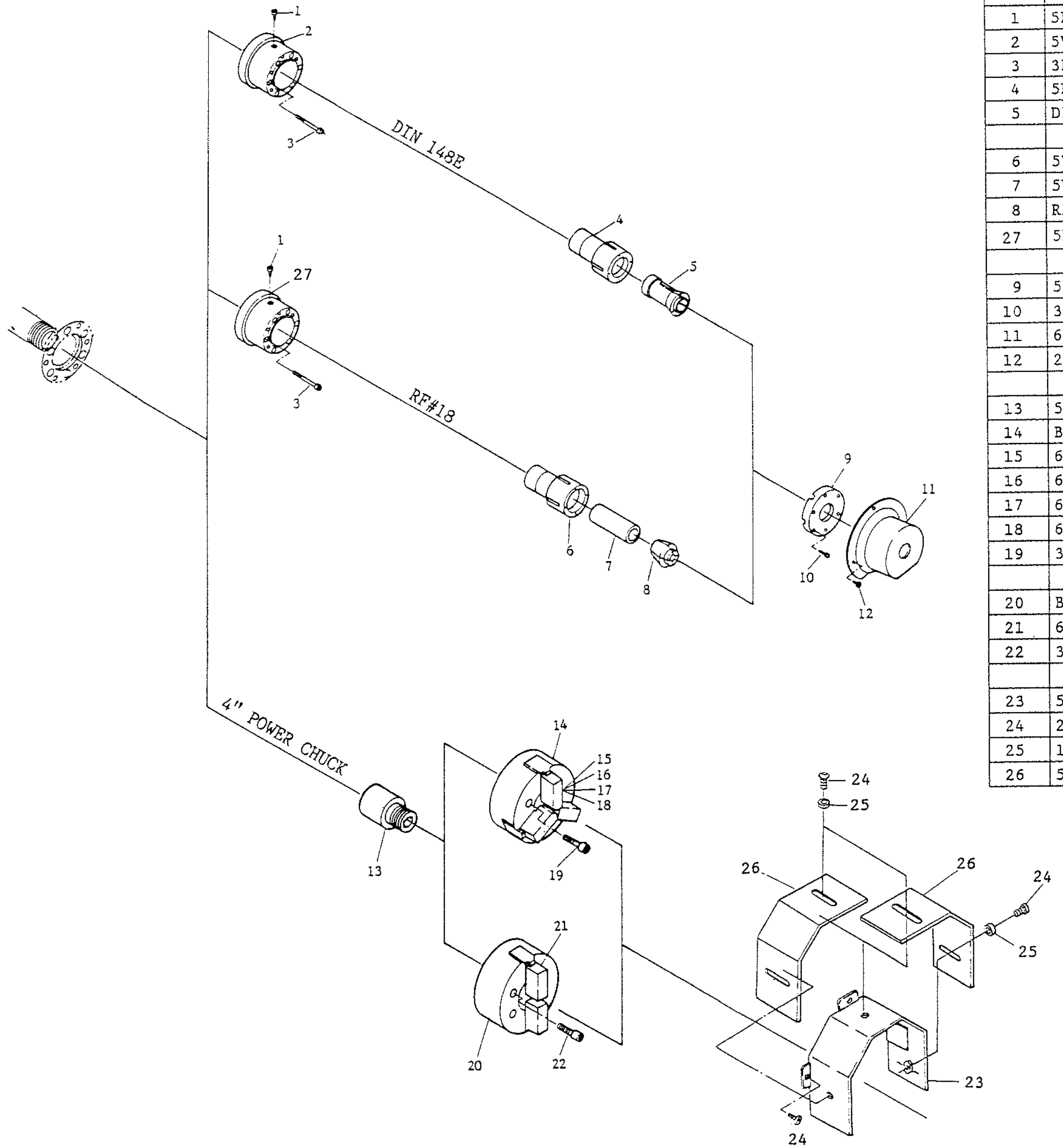


Parts List

* Mark must be changed for each bar size.

Item	Parts Number	Parts Name	Quantity
1	5E01 2050	PIN	1
2	5D01 2010	ADAPTER	1
3	3B 8 65	SOCKET SCREW M8x1.25 L=65	1
4	5D01 2050	CHUCK SLEEVE	1
5	B&S#11	COLLET CHUCK	*
6	5D01 2030	CHUCK CAP	1
7	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
8	5D01 5010	CHUCK SLEEVE	1
9	B&S#11C	COLLET CHUCK	*
10	5D01 2030	CHUCK CAP	1
11	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
12	5V01 3010	CHUCK SLEEVE	1
13	5V01 3020	SPACER	*
14	RF#18	RUBBERFLEX COLLET	*
15	5D01 4020	CHUCK CAP	1
16	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
17	6L01 0010	OIL GUARD	*
18	26B 6 12	MACHINE SCREW M6x1.0 L=12	3
19	5D02 0020	DRAW BAR	1
20	5E02 0030	PIN	1
21	5D02 0010	ADAPTER	1
22	3B 8 50	SOCKET SCREW M8x1.25 L=50	3
23		HARDINGE 2" 5C STEP CHUCK CLOSER	1
24		HARDINGE 2" 5C STEP CHUCK	*
25		HARDINGE 5C CHUCK	*
26	5D02 1030	ADAPTER	1
27	3B 8 30	SOCKET SCREW M8x1.25 L=30	3
28	5E01 2050	PIN	1
29	5D02 1010	CHUCK SLEEVE	1
30	3B 8 50	SOCKET SCREW M8x1.25 L=50	3
31	5D02 1020	MASTER COLLET	
32		S10 CLAMP ASSEMBLY (HARDINGE)	3set
33		S10 COLLET PADS (HARDINGE)	*
34	6L02 0010	OIL GUARD	1
35	26B 6 12	MACHINE SCREW M6x1.0 L=12	3
36	5D06 5010	ADAPTER	1
37	B-04-28	3 JAWS HOLE THROUGH HYDRAULIC CHUCK	1
38	61P-40-9796	SOFT JAWS (H=23)	*
39	61P-41-0161	SOFT JAWS (H=36)	* OPT
40	61P-41-0162	SOFT JAWS (H=48)	* OPT
41	61P-31-2351	HARD JAWS	1set OPT
42	3B 10 55	SOCKET SCREW M10x1.5 L=55	3
43	BT05-28	2 JAWS HOLE THROUGH HYDRAULIC CHUCK	1
44	61P-40-9913	SOFT JAWS	*
45	3B 10 55	SOCKET SCREW M10x1.5 L=55	4
46	5V06 0010	OIL GUARD A	1
47	26B 6 12	MACHINE SCREW M6x1.0 L=12	6
48	1W 6	WASHER	6
49	5V06 0020	OIL GUARD B	2

[2] BND-20S Main Spindle Chuck System (Excluding USA)

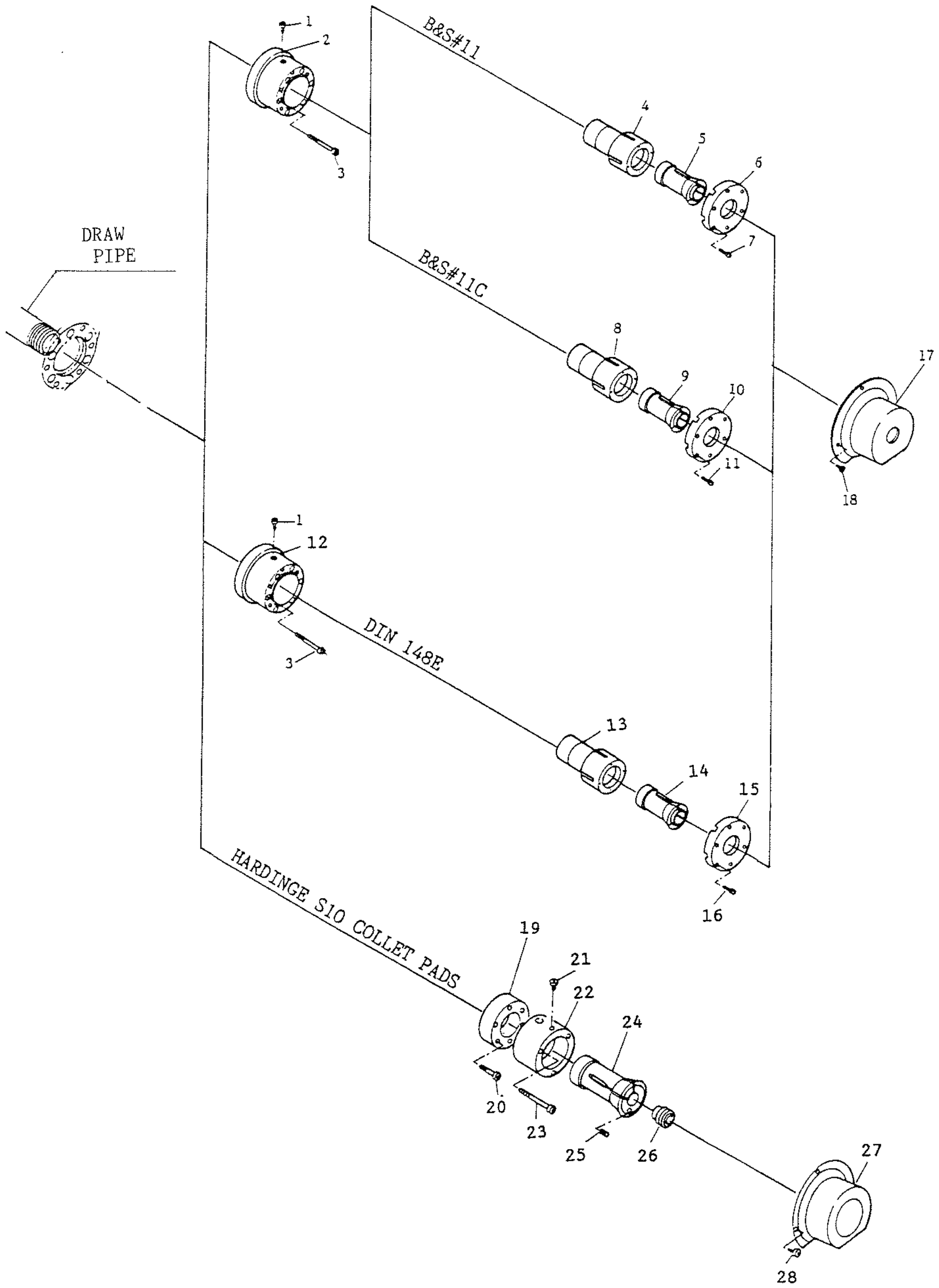


Parts List

Item	Parts Number	Parts Name	Quantity
1	5E01 2050	PIN	1
2	5V01 4010	ADAPTER	1
3	3B 8 65	SOCKET SCREW	6
4	5D01 4010	CHUCK SLEEVE	1
5	DIN 148E	COLLET CHUCK	*
6	5V01 3010	CHUCK SLEEVE	1
7	5V01 3020	SPACER	*
8	RF#18	RUBBERFLEX COLLET	*
27	5D01 2010	ADAPTER	1
9	5D01 4020	CHUCK CAP	1
10	3B 6 20	SOCKET SCREW	6
11	6L01 0010	OIL GUARD	*
12	26B 6 12	MACHINE SCREW M6x1.0 L=12	3
13	5D06 5010	ADAPTER	1
14	B-04-28	3 JAWS HOLE THROUGH HYDRAULIC CHUCK	1
15	61P-40-9796 ⚠	SOFT JAWS (H=23)	*
16	61P-41-0161 ⚠	SOFT JAWS (H=36)	* OPT
17	61P-41-0162 ⚠	SOFT JAWS (H=48)	* OPT
18	61P-31-2351 ⚠	HARD JAWS	1set OPT
19	3B 10 55	SOCKET SCREW M10x1.5 L=55	3
20	BT04-28	2 JAWS HOLE THROUGH HYDRAULIC CHUCK	1
21	61P-40-9913 ⚠	SOFT JAWS	*
22	3B 10 55	SOCKET SCREW M10x1.5 L=55	4
23	5V06 0010	OIL GUARD A	1
24	26B 6 12	MACHINE SCREW M6x1.0 L=12	6
25	1W 6	WASHER	6
26	5V06 0020	OIL GUARD B	2

* Mark must be changed for each bar size.

[3] BND-20S Main Spindle Orientation Collet Chuck System

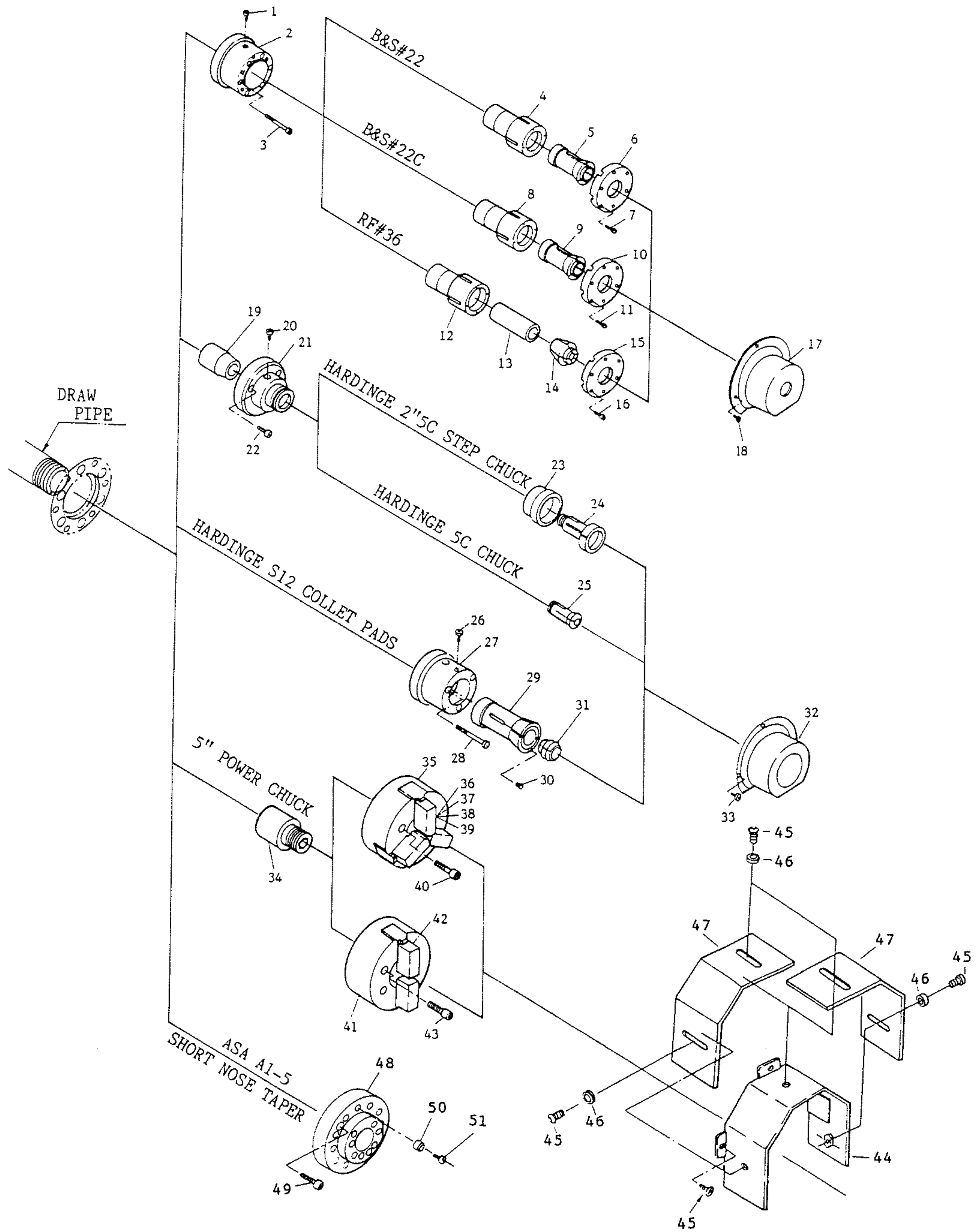


Parts List

Item	Parts Number	Parts Name	Quantity
1	5W60 8620	PIN	1
2	5D01 2010	ADAPTER	1
3	3B 8 65	SOCKET SCREW M8x1.25 L=65	6
4	5V01 1510	CHUCK SLEEVE	1
5	5V01 1520	B&S#11 COLLET CHUCK	*
6	5D01 2030	CHUCK CAP	1
7	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
8	5V01 0510	CHUCK SLEEVE	1
9	5V01 0520	B&S#11C COLLET CHUCK	*
10	5D01 2030	CHUCK CAP	1
11	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
12	5V01 4010	ADAPTER	1
13	5V01 4510	CHUCK SLEEVE	1
14	5V01 4520	DIN 148E COLLET CHUCK	*
15	5D01 4020	CHUCK CAP	1
16	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
17	6L01 0010	OIL GUARD	*
18	26B 6 12	MACHINE SCREW M6x1.0 L=12	3
19	5D02 1030	ADAPTER	1
20	3B 8 30	SOCKET SCREW M8x1.25 L=30	3
21	5E01 2050	PIN	1
22	5D02 1010	CHUCK SLEEVE	1
23	3B 8 50	SOCKET SCREW M8x1.25 L=50	3
24	5D02 1020	MASTER COLLET	1
25		S10 CLAMP ASSEMBLY (HARDINGE)	3set
26		S10 COLLET PADs (HARDINGE)	*
27	6L02 0010	OIL GUARD	*
28	26B 6 12	MACHINE SCREW M6x1.0 L=12	3

* Mark must be changed for each bar size.

[4] BND-34S Main Spindle Chuck System (USA)

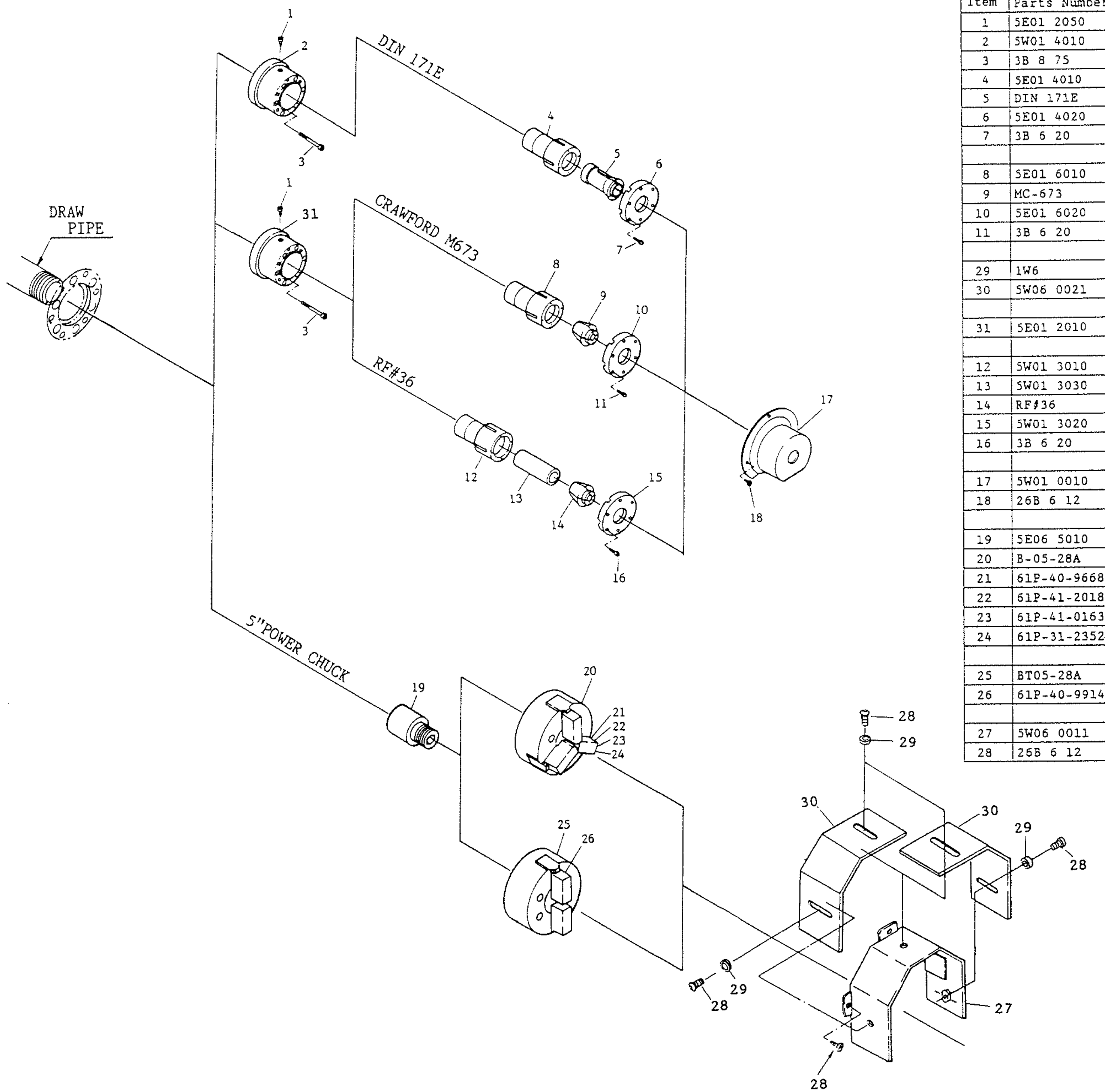


Parts List

* Mark must be changed for each bar size.

Item	Parts Number	Parts Name	Quantity
1	5E01 2050	PIN	1
2	5E01 2010	ADAPTER	1
3	3B 8 75	SOCKET SCREW M8x1.25 L=75	6
4	5E01 5010	CHUCK SLEEVE	1
5	B&S#22	COLLET CHUCK	*
6	5E01 5020	CHUCK CAP	1
7	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
8	5E01 2020	CHUCK SLEEVE	1
9	B&S#22C	COLLET CHUCK	*
10	5E01 2030	CHUCK CAP	1
11	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
12	5W01 3010	CHUCK SLEEVE	1
13	5W01 3030	SPACER	*
14	RF#36	RUBBERFLEX COLLET	*
15	5W01 3020	CHUCK CAP	1
16	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
17	5W01 0010	OIL GUARD	*
18	26B 6 12	MACHINE SCREW M6x1.0 L=12	3
19	5E02 0020	DRAW BAR	1
20	5E02 0030	PIN	1
21	5E02 0010	ADAPTER	1
22	3B 8 20	SOCKET SCREW M8x1.25 L=20	3
23		HARDINGE 2" 5C STEP CHUCK CLOSER	1
24		HARDINGE 2" 5C STEP CHUCK	*
25		HARDINGE 5C CHUCK	*
26	5E01 2050	PIN	1
27	5E02 1010	CHUCK SLEEVE	1
28	3B 8 90	SOCKET SCREW M8x1.25 L=90	3
29	5E02 2010	MASTER COLLET	1
30		S12 CLAMP ASSEMBLY (HARDINGE)	3set
31		S12 COLLET PADS (HARDINGE)	*
32	5W02 1010	OIL GUARD	1
33	26B 6 12	MACHINE SCREW M6x1.0 L=12	3
34	5E06 5010	ADAPTER	1
35	B-05-28A	3 JAWS HOLE THROUGH HYDRAULIC CHUCK	1
36	61P-40-9668	SOFT JAWS (H31)	*
37	61P-41-2018	SOFT JAWS (H38)	* OPT
38	61P-41-0163	SOFT JAWS (H53)	* OPT
39	61P-31-2352	HARD JAWS	1set OPT
40	3B 10 60	SOCKET SCREW M10x1.5 L=60	3
41	BT05-28A	2 JAWS HOLE THROUGH HYDRAULIC CHUCK	1
42	61P-40-9914	SOFT JAWS	*
43	3B 10 60	SOCKET SCREW M10x1.5 L=60	4
44	5W06 0011	OIL GUARD A	1
45	26B 6 12	MACHINE SCREW M6x1.0 L=12	6
46	1W 6	WASHER	6
47	5W06 0021	OIL GUARD B	1
48	5E06 6010	ADAPTER	1
49	3B 8 30	SOCKET SCREW M8x1.25 L=30	3
50	5E06 6020	DRIVING BUTTON	1
51	4B 6 16	FLAT SCREW M4x0.7 L=16	1

[5] BND-34S Main Spindle Chuck System (Excluding USA)

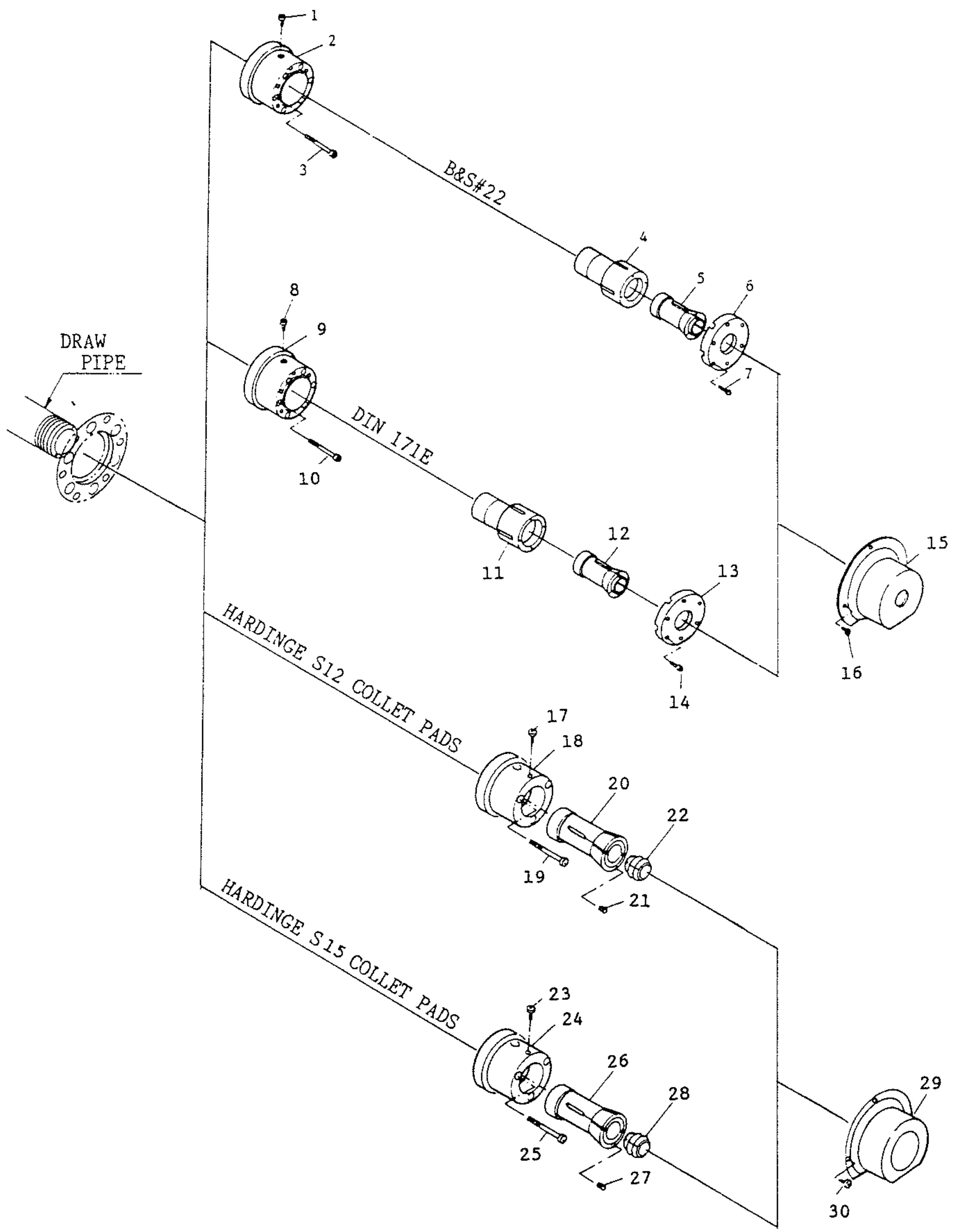


Parts List

Item	Parts Number	Parts Name	Quantity
1	5E01 2050	PIN	1
2	5W01 4010	ADAPTER	1
3	3B 8 75	SOCKET SCREW M8x1.25 L=75	6
4	5E01 4010	CHUCK SLEEVE	1
5	DIN 171E	COLLET CHUCK	*
6	5E01 4020	CHUCK CAP	1
7	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
8	5E01 6010	CHUCK SLEEVE	1
9	MC-673	CRAWFORD AUTO MALTIBORE COLLET	*
10	5E01 6020	CHUCK CAP	1
11	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
29	1W6	WASHER	6
30	5W06 0021	OIL GUARD B	1
31	5E01 2010	ADAPTER	1
12	5W01 3010	CHUCK SLEEVE	1
13	5W01 3030	SPACER	*
14	RF#36	RUBBERFLEX COLLET	*
15	5W01 3020	CHUCK CAP	1
16	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
17	5W01 0010	OIL GUARD	*
18	26B 6 12	SOCKET SCREW M6x1.0 L=12	3
19	5E06 5010	ADAPTER	1
20	B-05-28A	3 JAWS HOLE THROUGH HYDRAULIC CHUCK	1
21	61P-40-9668△	SOFT JAWS (H31)	*
22	61P-41-2018△	SOFT JAWS (H38)	* OPT
23	61P-41-0163△	SOFT JAWS (H53)	* OPT
24	61P-31-2352△	HARD JAWS	1set OPT
25	BT05-28A	2 JAWS HOLE THROUGH HYDRAULIC CHUCK	1
26	61P-40-9914△	SOFT JAWS	*
27	5W06 0011	OIL GUARD A	1
28	26B 6 12	MACHINE SCREW M6x1.0 L=10	6

* Mark must be changed for each bar size.

[6] BND-34S Main Spindle Orientation Collet Chuck System

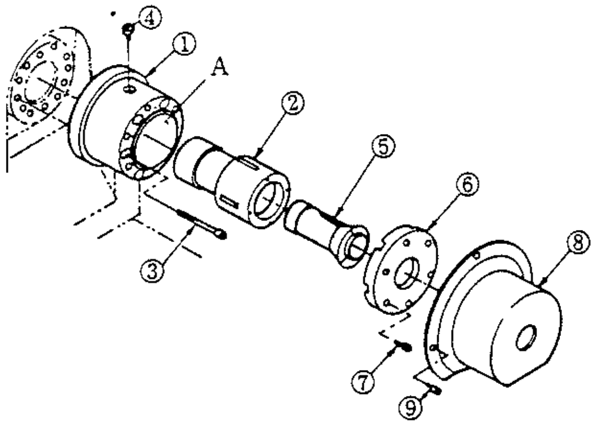


Parts List

Item	Parts Number	Parts Name	Quantity
1	5W01 5530	PIN	1
2	5E01 2010	ADAPTER	1
3	3B 8 75	SOCKET SCREW M8x1.25 L=75	6
4	5W01 5520	CHUCK SLEEVE	1
5	5W01 5510	COLLET CHUCK	*
6	5E01 5020	CHUCK CAP	1
7	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
8	5W01 6031	PIN	1
9	5W01 4010	ADAPTER	1
10	3B 8 75	SOCKET SCREW M8x1.25 L=75	6
11	5W01 4520	CHUCK SLEEVE	1
12	5W01 4510	COLLET CHUCK	*
13	5E01 4020	CHUCK CAP	1
14	3B 6 20	SOCKET SCREW M6x1.0 L=20	6
15	5W01 0010	OIL GUARD	*
16	26B 6 12	MACHINE SCREW	3
17	5E01 2050	PIN	1
18	5E02 1010	CHUCK SLEEVE	1
19	3B 8 90	SOCKET SCREW M8x1.25 L=90	3
20	5E02 2010	MASTER COLLET	1
21		S12 CLAMP ASSEMBLY (HARDINGE)	3set
22		S12 COLLET PADS	*
23	5E01 2050	PIN	1
24	5E02 1010	CHUCK SLEEVE	1
25	3B 8 90	SOCKET SCREW M8x1.25 L=90	3
26	5E02 1020	MASTER COLLET	1
27		S15 CLAMP ASSEMBLY (HARDINGE)	3set
28		S15 COLLET PADS	*
29	5W02 0010	OIL GUARD	*
30	26B 6 12	MACHINE SCREW	3

* Mark must be changed for each bar size.

6. INSTALLATION AND REMOVAL OF COLLET FOR MAIN SPINDLE OF BND-20S, BND-34S



[1] Installation

1) Install an adapter ① to spindle nose.
   Set a dial indicator on the inside diameter of chuck sleeve ② of adapter and adjust the runout of the adapter within .0002" (5μm) and tighten bolts ③.

2) Screw in the chuck sleeve ② into draw pipe (Refer to P84, P85, P87, P88).
   Screw the chuck sleeve in until the front face is flush with the adapter front face while the chuck is closed.

3) Retain the chuck sleeve ② by a pin ④ .
   There are three pin installation holes for adapter and there are four pin slots of chuck sleeve. Select the location where a pin installation hole and pin slot is matched.
   Screw in the pin into an adapter. Tighten the pin so that it may not be dropped off during use.

4) Insert a collet chuck ⑤ into chuck sleeve and install the chuck cap ⑥ by bolts ⑦ .
   Tighten the bolt tightly so that it may not be loosened during use.

5) Install an oil guard ⑧ by set screws ⑨ .
   A hole of oil guard must be bigger than the material diameter +.078" (+2mm).
   If this hole is too big, cutting chips will go into the spring collet, it may result in unexpected accident.

[2] Removal

Removal can be performed in the reverse order of installation.

[3] Replacement of spring collet

1) Remove an oil guard.

2) Remove a chuck cap.

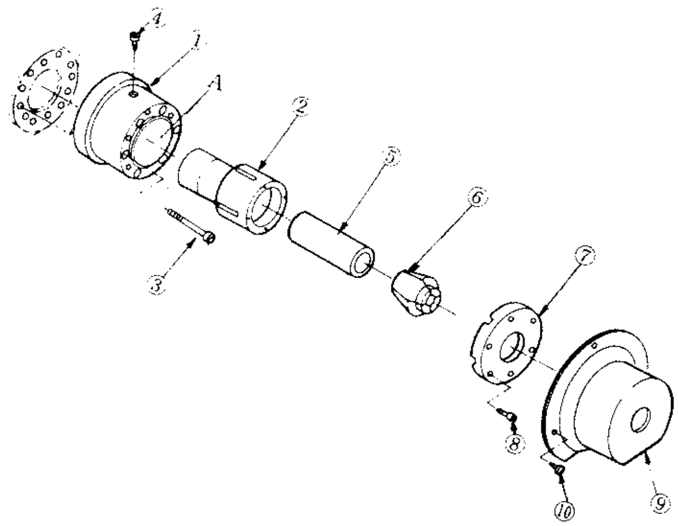
3) Remove a spring collet.

4) Install a spring collet for new set up to chuck sleeve.

5) Install a chuck cap.

6) Install the oil guard.

7. INSTALLATION AND REMOVAL OF RUBBER FLEX COLLET FOR BND-20S, BND-34S MAIN SPINDLE



[1] Installation

1) Install an adapter ① to spindle nose.
Set a dial indicator on the inside diameter of chuck sleeve ② of adapter and adjust the runout of the adapter within .0002"(5μm) and tighten bolts ③.

2) Screw in a chuck sleeve ② into draw pipe (Refer to P84, P85, P87, P88).
Under chuck close condition, screw in the chuck sleeve until top end of adapter and end surface of chuck sleeve coincide.

3) Retain the chuck sleeve ② by a pin ④ .
Since a pin installation hole of adapter is provided in three locations and a pin groove of chuck sleeve is provided in four locations.
Select the position where a pin installation hole and pin groove is matched. Tighten the pin enoughly so that it may not be dropped off during use.

4) Install a spacer ⑤ in the chuck cap. The inner diameter of this spacer should be .02"(0.5mm) bigger than the material diameter.

5) Install a chuck cap ⑦ to adapter by bolts ⑧ with a rubber flex collet ⑥ .
Tighten bolts ⑧ so that it may not be loosened during use.

6) Install an oil guard ⑨ by screws ⑩ . A hole of the oil guard must be bigger than the material diameter .078"(2mm). If it is too large compared with the material diameter, many cutting chips will go into the collet, it may result in unexpected accident.

[2] Removal

Removal can be performed in the reverse order of installation.

[3] Replacement of rubber flex collet

1) Remove an oil guard.

2) Remove a chuck cap. Since a rubber flex collet is removed together with the chuck cap, pay attention so that it may not be dropped.

3) Install new set up rubber flex collet to chuck cap.

4) Install new set up spacer.

5) Install a chuck cap.

6) Install new set up oil guard.

[4] Range of rubber flex collet size

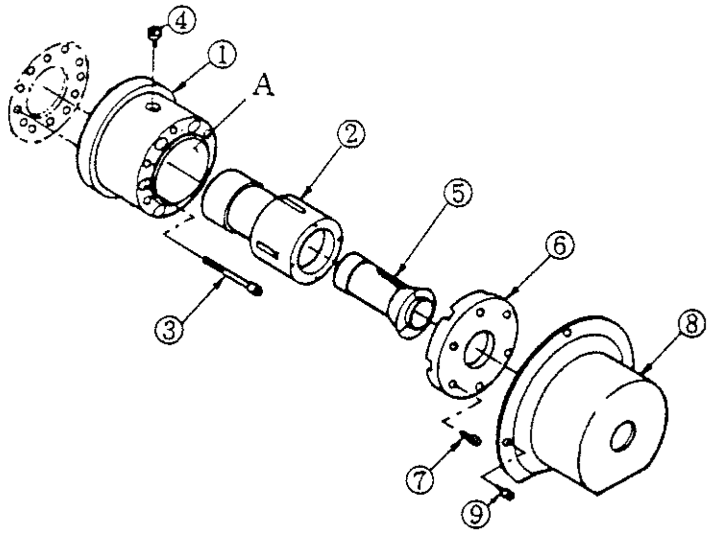
BND-20S2

Range of chucking diameter		Part number of rubber flex collet
±.01"	±0.3(mm)	
ϕ1/4" to ϕ5/16"	ϕ6 to ϕ8	1808
ϕ11/32" to ϕ3/8"	ϕ8 to ϕ10	1810
ϕ13/32" to ϕ15/32"	ϕ10 to ϕ12	1812
ϕ1/2" to ϕ17/32"	ϕ12 to ϕ14	1814
ϕ9/16" to ϕ5/8"	ϕ14 to ϕ16	1816
ϕ21/32" to ϕ11/16"	ϕ16 to ϕ18	1818
ϕ23/32" to ϕ25/32"	ϕ18 to ϕ20	1820

BND-34S2

Range of chucking diameter		Part number of rubber flex collet
± .01"	± 0.3(mm)	
ϕ 15/32" to ϕ 1/2"	ϕ 12 to ϕ 13	3612
ϕ 17/32" to ϕ 19/32"	ϕ 13 to ϕ 15	3614
ϕ 19/32" to ϕ 11/16"	ϕ 15 to ϕ 17	3616
ϕ 11/16" to ϕ 3/4"	ϕ 17 to ϕ 19	3618
ϕ 3/4" to ϕ 13/16"	ϕ 19 to ϕ 21	3620
ϕ 13/16" to ϕ 29/32"	ϕ 21 to ϕ 23	3622
ϕ 29/32" to ϕ 31/32"	ϕ 23 to ϕ 25	3624
ϕ 31/32" to ϕ 1-1/16"	ϕ 25 to ϕ 27	3626
ϕ 1-1/16" to ϕ 1-5/32"	ϕ 27 to ϕ 29	3628
ϕ 1-5/32" to ϕ 1-7/32"	ϕ 29 to ϕ 31	3630
ϕ 1-7/32" to ϕ 1-9/32"	ϕ 31 to ϕ 33	3632
ϕ 1-9/32" to ϕ 1-21/64"	ϕ 33 to ϕ 34	3634

8. INSTALLATION AND REMOVAL OF SPRING COLLET CHUCK FOR MAIN SPINDLE ORIENTATION OF BND-20S, BND-34S



[1] Installation

1) Install an adapter ① to spindle nose.
Set a dial gauge on the inside diameter of the chuck sleeve ② of adapter and align the runout within .0002"(5μm) and tighten bolts ③.

2) Screw in a chuck sleeve ② into a draw pipe (Refer to P86, P89).
Screw the chuck sleeve in until the front face is flush with the adapter front face while the chuck is closed.

3) Match installation hole of adapter pin with pin slot of chuck sleeve.
Adapter has three installation holes, and chuck sleeve has four pin slots. Try to find well matching position.

4) Install spring collet ⑤ to chuck cap ⑥ and put it into sleeve.
This time, put spring collet so that its pin hole may match with the holes as arranged in 3).

5) Fasten cap ⑥ with a bolt ⑦ and put in a pin ④ so that chuck sleeve and spring collet may not move around.

6) Install an oil guard ⑧ with set screws ⑨.
A hole of the oil guard must be bigger than the material diameter +.078" (+2.0mm). If it is too large compared with the material diameter, many cutting chips may go into the spring collet and it may result in unexpected accident.

[2] Removal

Removal can be done by reverse procedure of installation.

[3] Exchange of a spring collet

1) Remove oil guard.

2) Take off a pin for anti-rotating.

3) Take off a chuck cap and a used spring collet

4) Install a new spring collet to chuck cap, and put it into sleeve, so as to match a pin hole.

5) Install chuck cap and put pin for anti-rotating.

6) Install a new oil guard.

9. POWER CHUCK FOR BND-20S, BND-34S MAIN SPINDLE

[1] Specification

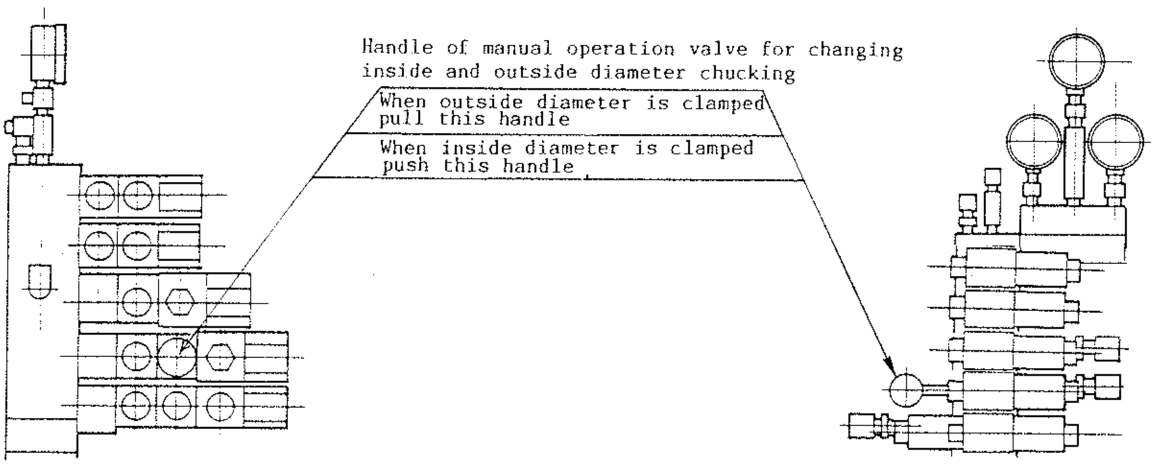
Item	BND-20S		BND-34S	
	3 jaw chuck	2 jaw chuck	3 jaw chuck	2 jaw chuck
Type	B- 04 - 28	BT04 - 28	B- 05 - 28A	BT05 - 28A
Maximum chucking capability	2" (ϕ 50)	2" (ϕ 50)	3"(ϕ 75)	3" (ϕ 75)
Jaw stroke	.16"(4.2mm) in diameter	.16"(4.2mm) in diameter	.20"(5.1mm) in diameter	.20"(5.1mm) in diameter
Maximum chucking pressure	240 PSI (18.5kgf/cm²)	180 PSI (13kgf/cm²)	180 PSI (13kgf/cm²)	124 PSI (9kgf/cm²)
Maximum spindle speed	6000rpm	6000rpm	5000rpm	5000rpm

BND-S OTC OPE 90/7

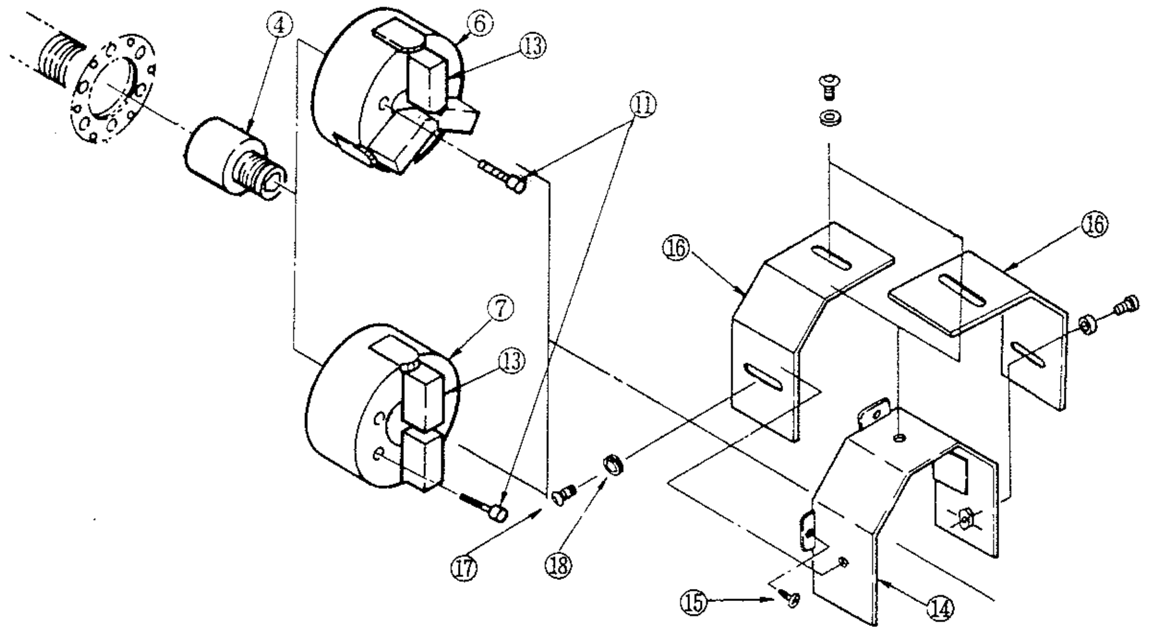
[2] Installation and removal of power chuck for main spindle

Although there are 3 jaw type and 2 jaw type for power chuck, the procedure of installation and removal is the same and, explanation is made by taking 3 jaw type as an example.

1) Change-over of outside diameter chucking and inside diameter chucking by power chuck.

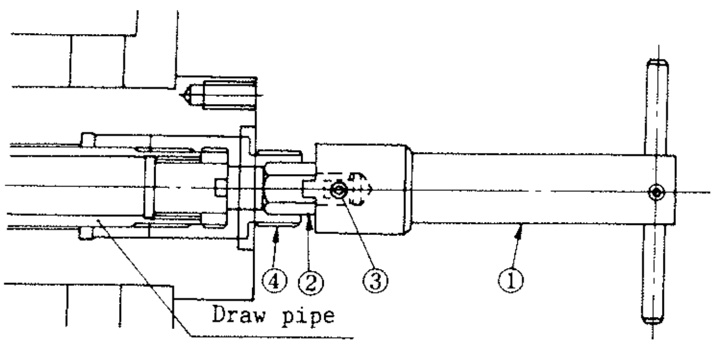


2) Installation

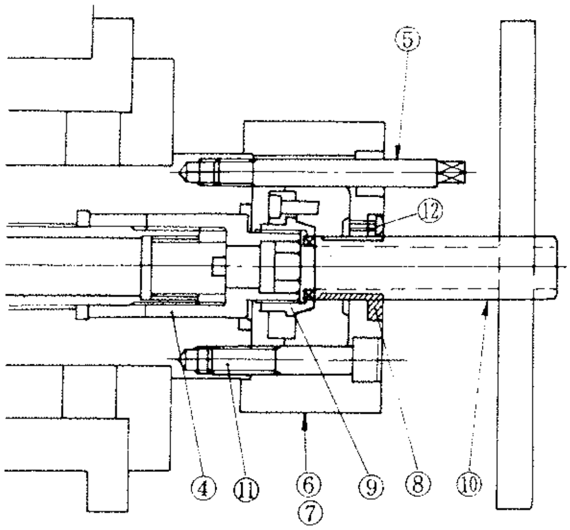


(1) Open thee chuck. (in case of inside diameter chucking, close the chuck.)

(2) As shown in the figure below, a hexagon spanner (2) is inserted in an attached wrench (1) and lock it by the screw (3) and insert into hexagon hole of an adapter (4) and screw in the dapter (4) in draw pipe.
Tighten firmly so that it may not be loosened during use.



(3) As shown in the figure below, install an attached tool guide bar (5) to power chuck installation tap hole (M10 x 1.5) at the end of spindle. Then, remove a cover (8) from a power chuck (6) or (7) and power chuck installation bolt hole is inserted into the guide bar (5) and power chuck main body is pushed to spindle side. An adapter (4) and screw of draw nut (9) contact and stop. Then, insert a handle (10) into the slot of draw nut (9) and screw in the draw nut (9) into adapter (4).
Tighten the draw nut (9) completely once and then turn back by about one and half turn. Then, remove the connecting handle (10) and guide bar (5) and install power chuck by bolts (11).
Align the chuck body, the runout of the power chuck should be within .0008"(0.02mm).



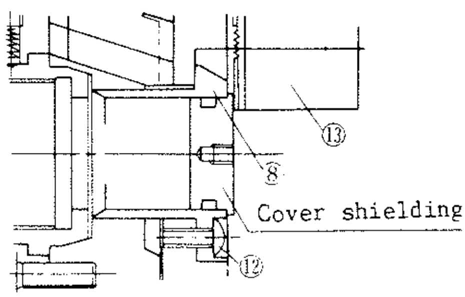
(4) Install the cover (8) by screws (12).

(5) Install a jaw (13).

(6) Install the oil guard A (14) by screws (15).

(7) Install oil guard B (16) with a small screw (17) and a washer (18).

(Caution) When a material of work is short, and it does not enter in the power chuck, a cover shielding as shown in the figure below should be installed, so that coolant may not enter into main spindle.

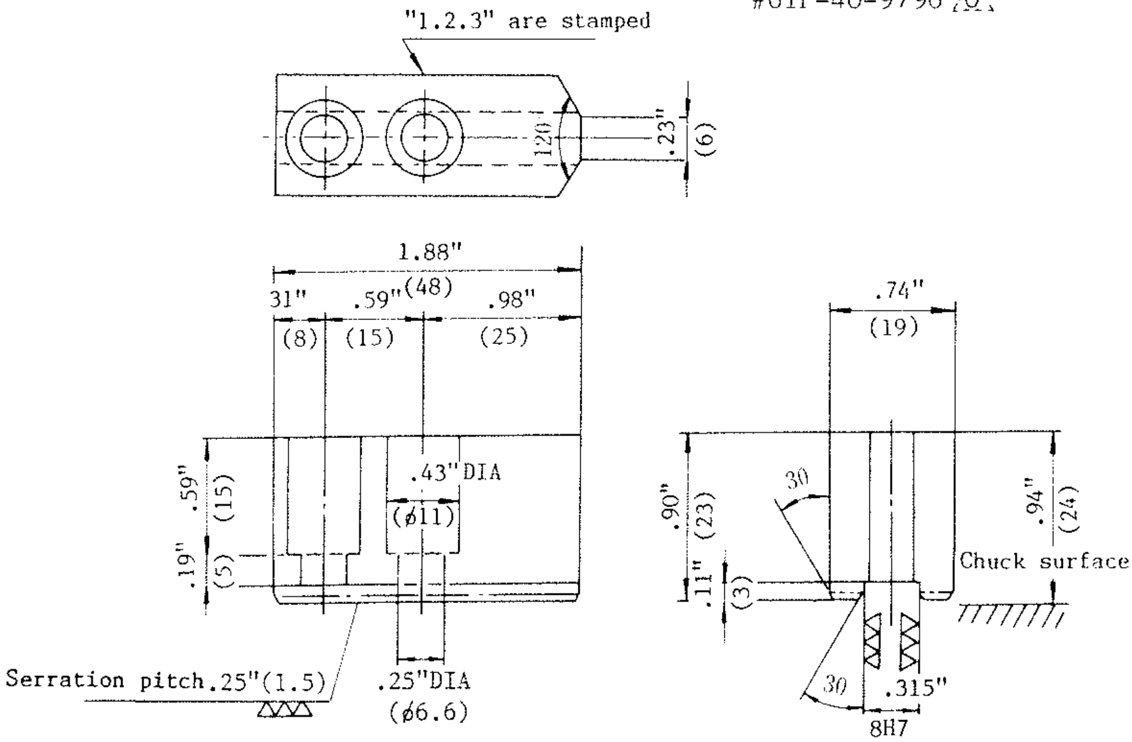


3) Removal
Removal is performed in the reverse order of installation.

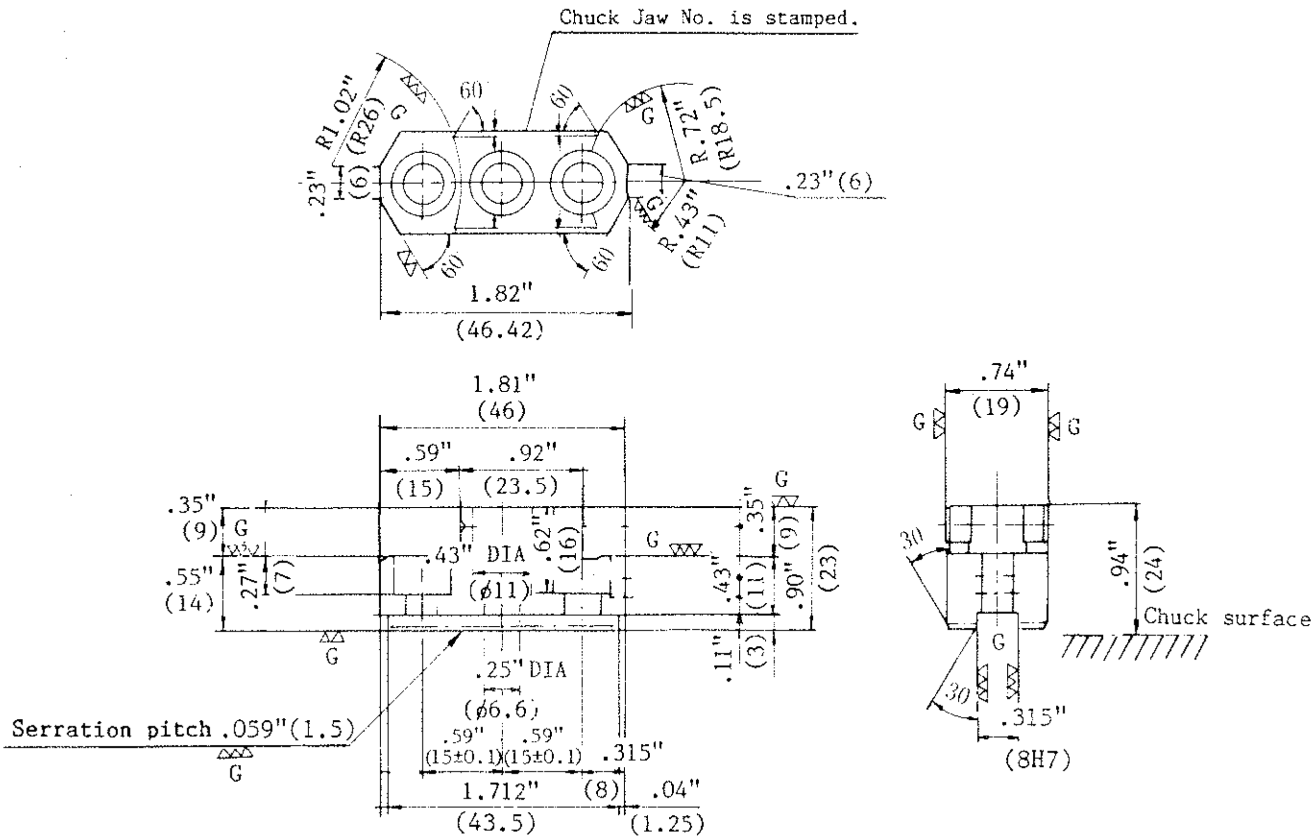
[3] Jaw dimension for BND-20S

1) Soft jaws for 3 jaw chuck (standard accessary)

#61P-40-9796



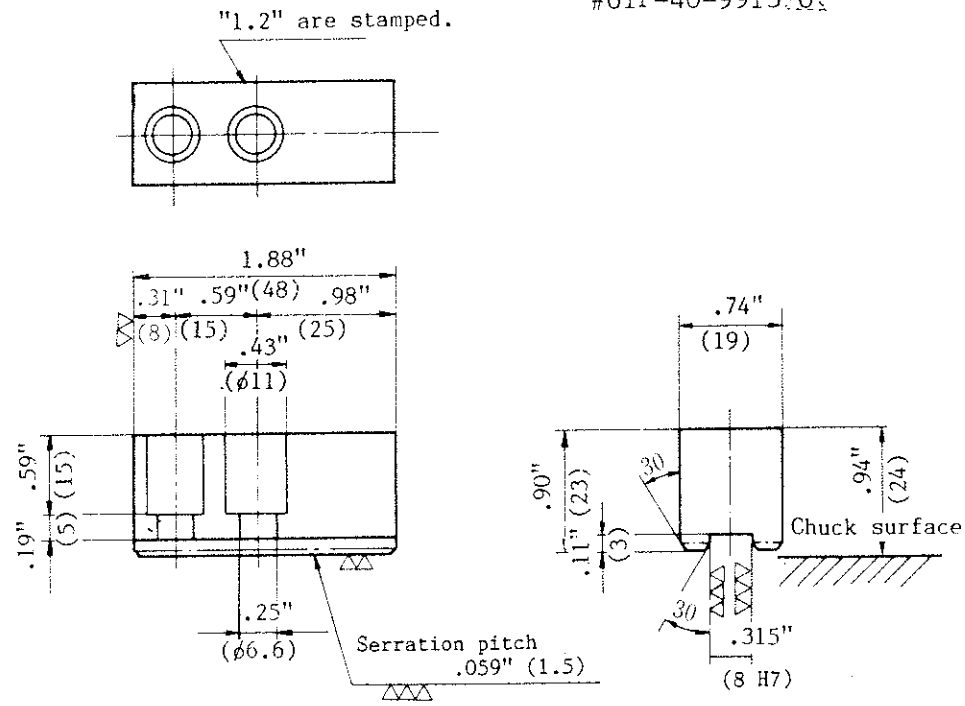
2) Hard jaws for 3 jaw chuck



BND-S OTC OPE 90/7

3) Soft jaws for 2 jaw chuck

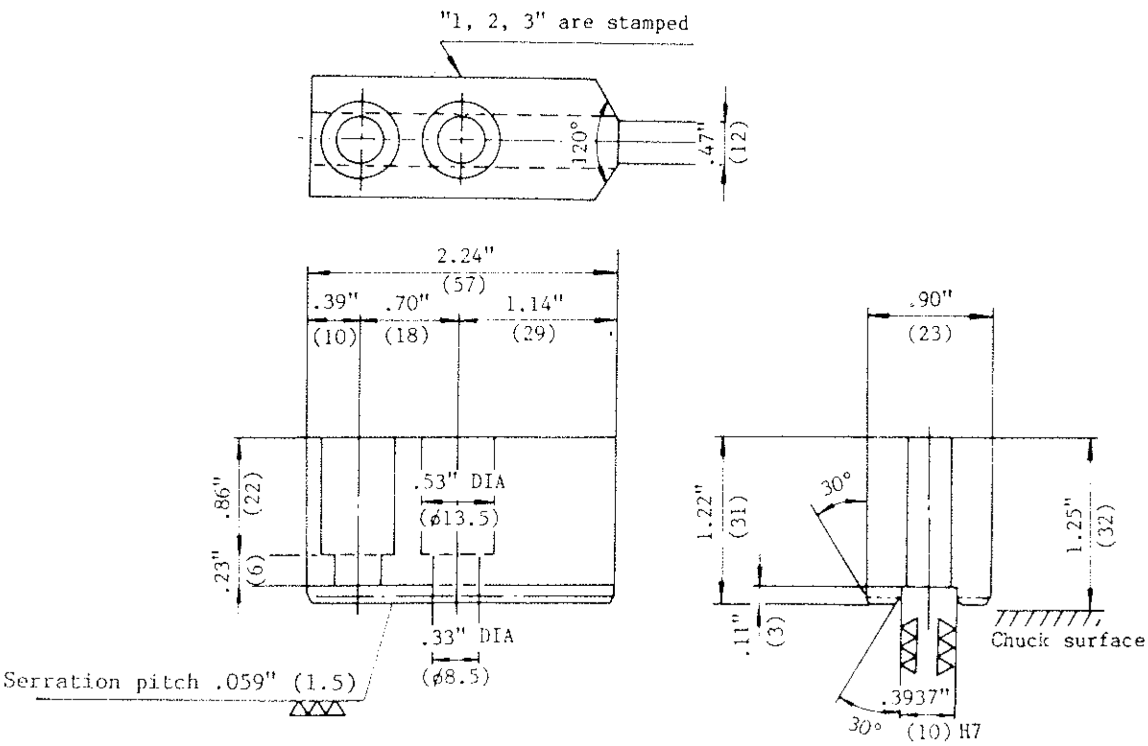
#61P-40-9913 $\hat{0}$



[4] Jaw dimension for BND-34S

1) Soft jaws for BND-34S 3 jaw chuck (Standard accessary)

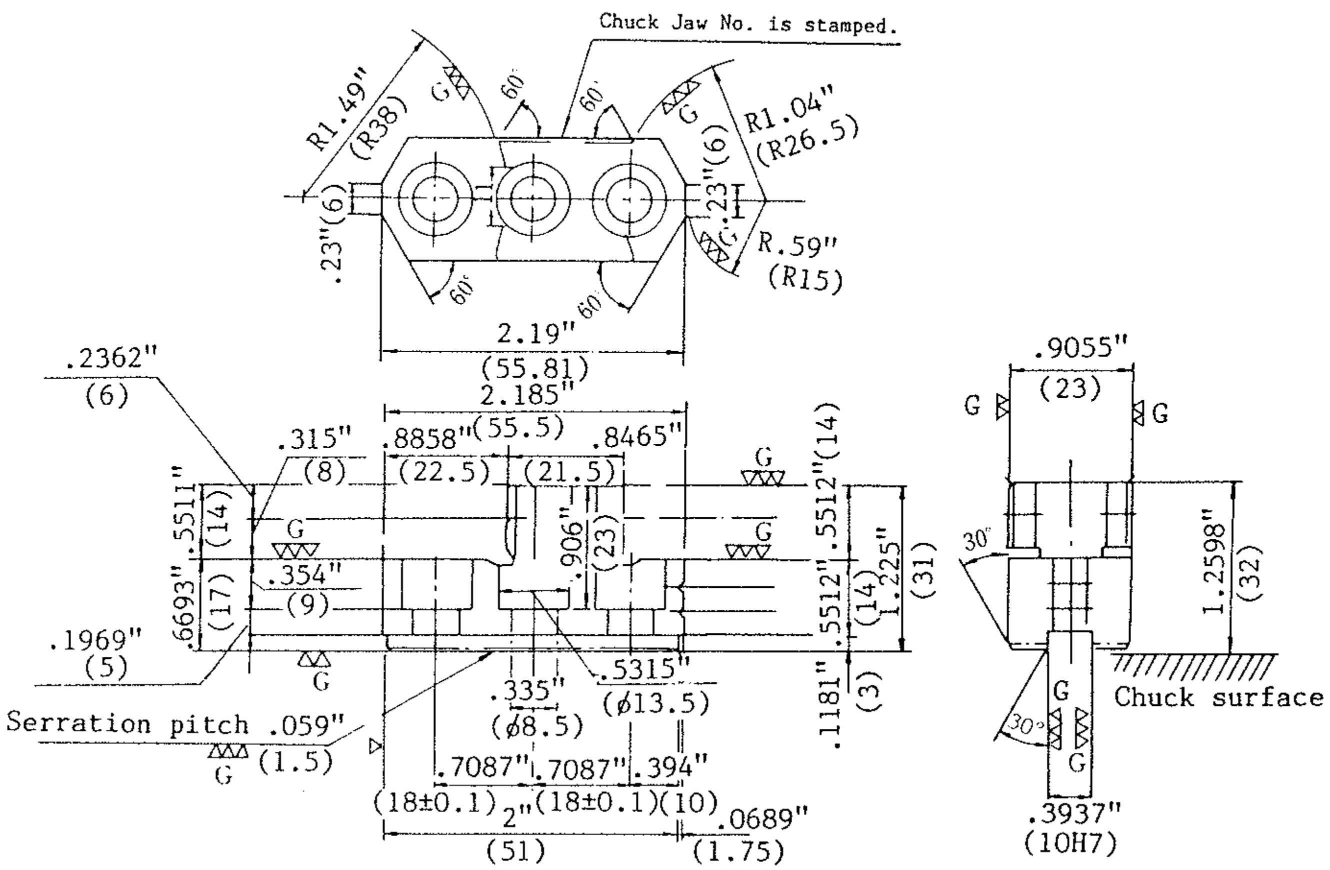
#61P-40-9668 $\hat{0}$ Soft jaw



BND-S OTC OPE 90/7

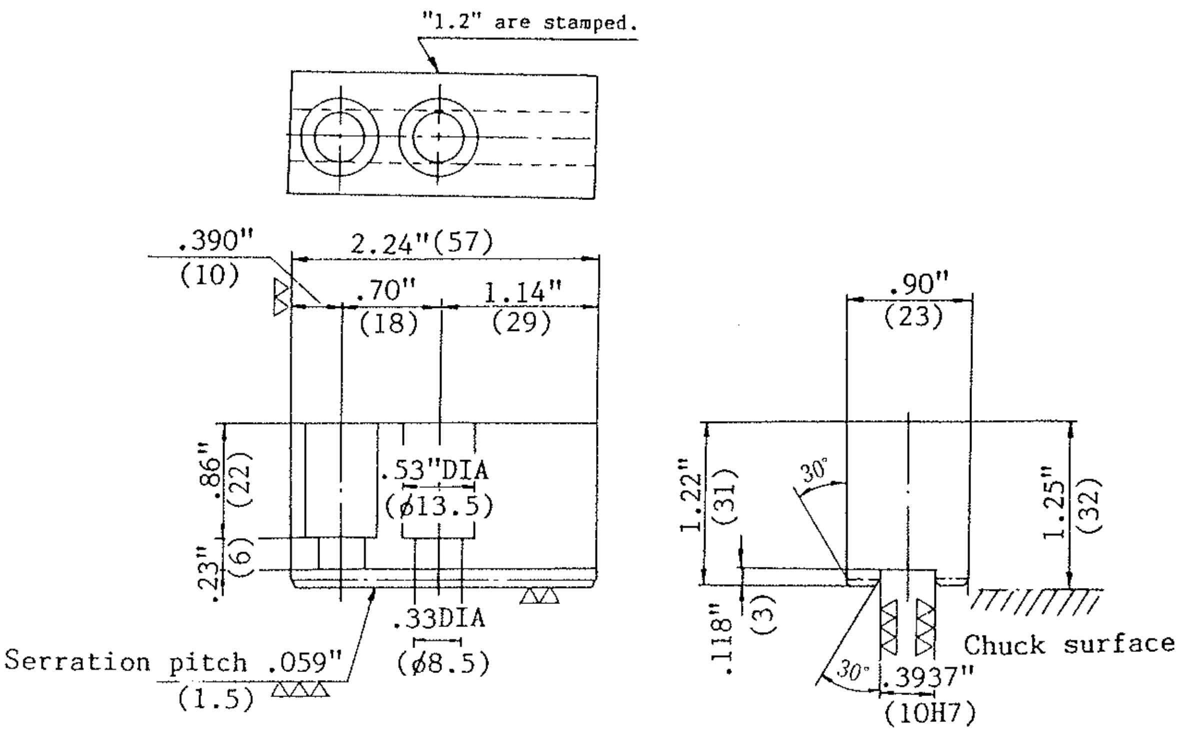
2) Hard jaws for BND-34S 3 jaw chuck

#61P-31-2352 1 Hard jaw



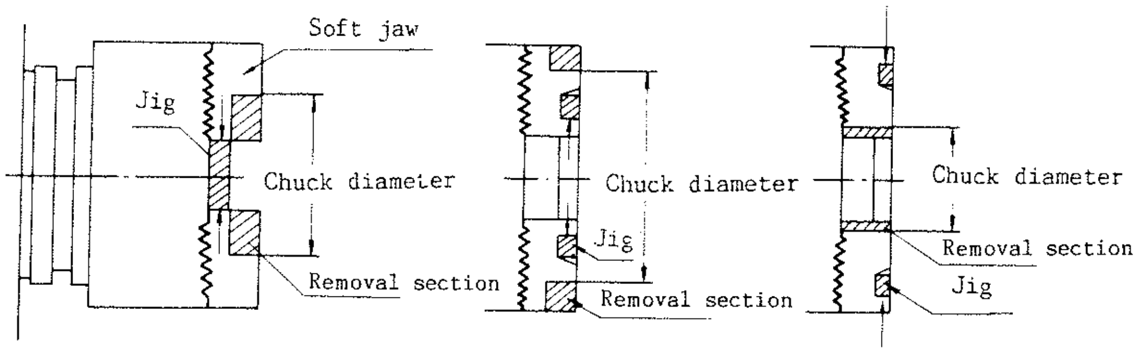
3) Soft jaws for BND-34S 2 jaw chuck

#61P-40-9914 0 Soft jaw



[5] Machining method of soft jaws

For machining of soft jaws, it is required to consider machining method depending on the shape of chuck as shown in the figure below.



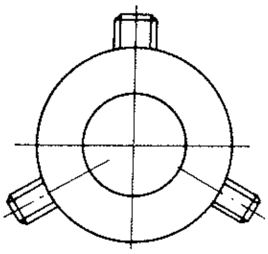
Machining procedure

1) Soft jaws and jig are determined depending on the shape of work.

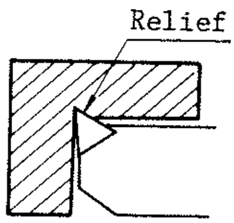
2) If removal section is large or the shape is special, milling operation is required.

3) A jig must be made but bearing inner race or outer race can be used. It is convenient when machining jig as shown in the figure on following page is made.

4) Relief machining is required in the stopper section as shown in the fugure on following page depending on the shape of work.



Example of soft jaw machining jig



Example of soft jaw relief machining

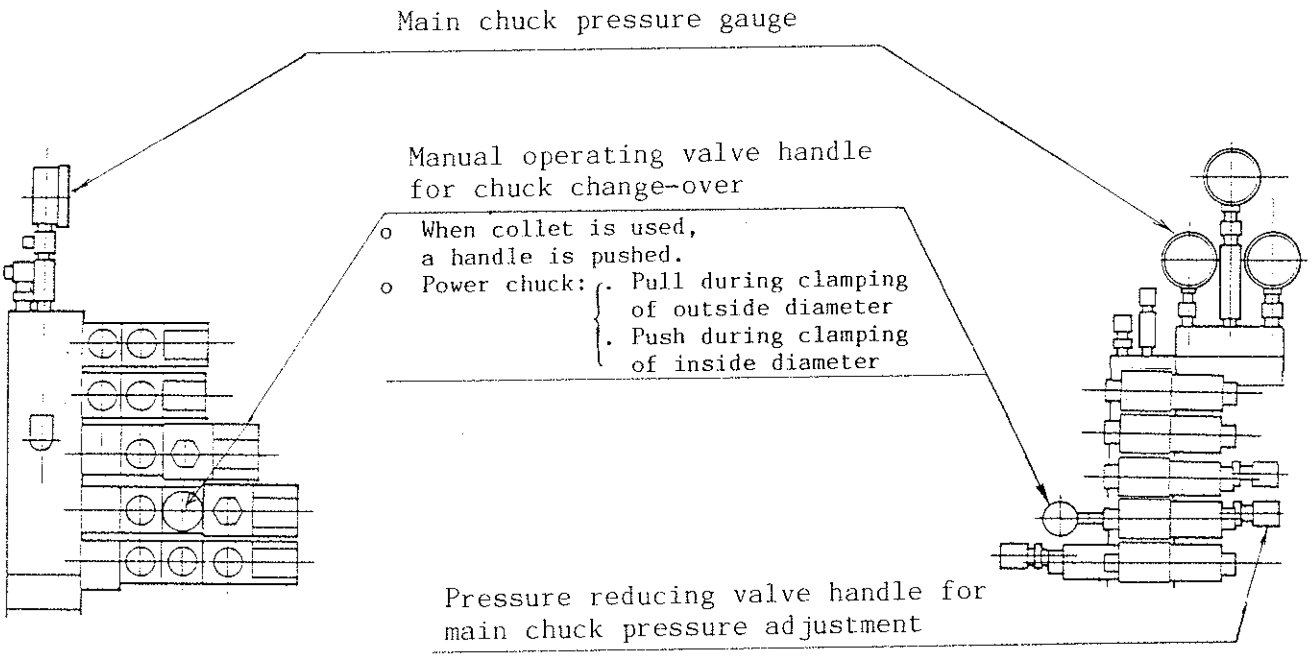
5) Various cautions for soft jaw machining
   - Material of soft jaw is normally used low carbon steel (1020 or 1018). Machinability is excellent but cutting condition is intermittent, low cutting condition should be selected.
   - Jig clamping force during soft jaw machining should be set as the same as the actual machining pressure.

10. ADJUSTMENT OF MAIN SPINDLE CHUCK CLAMPING FORCE

In case of collet (Spring collet and rubber flex collet) and power chuck, clamping force of chuck is adjusted by hydraulic pressure reducing valve. Clamping force of chuck is required to the adjusted depending on the thickness and shape of work piece and cutting condition. Also it depending on cutting resistance and centrifugal force of jaw due to spindle rotation. When it is too strong, machining accuracy is affected due to deformation of material. When it is too weak, there is danger such as drop-off and splash of work piece. Adjust it by paying enough attention.

o Adjustment of chuck clamping pressure

Adjustment of chuck clamping pressure is made by pressure reducing valve for main chuck shown in the figure below. It's setting pressure can be known by main chuck pressure gauge.



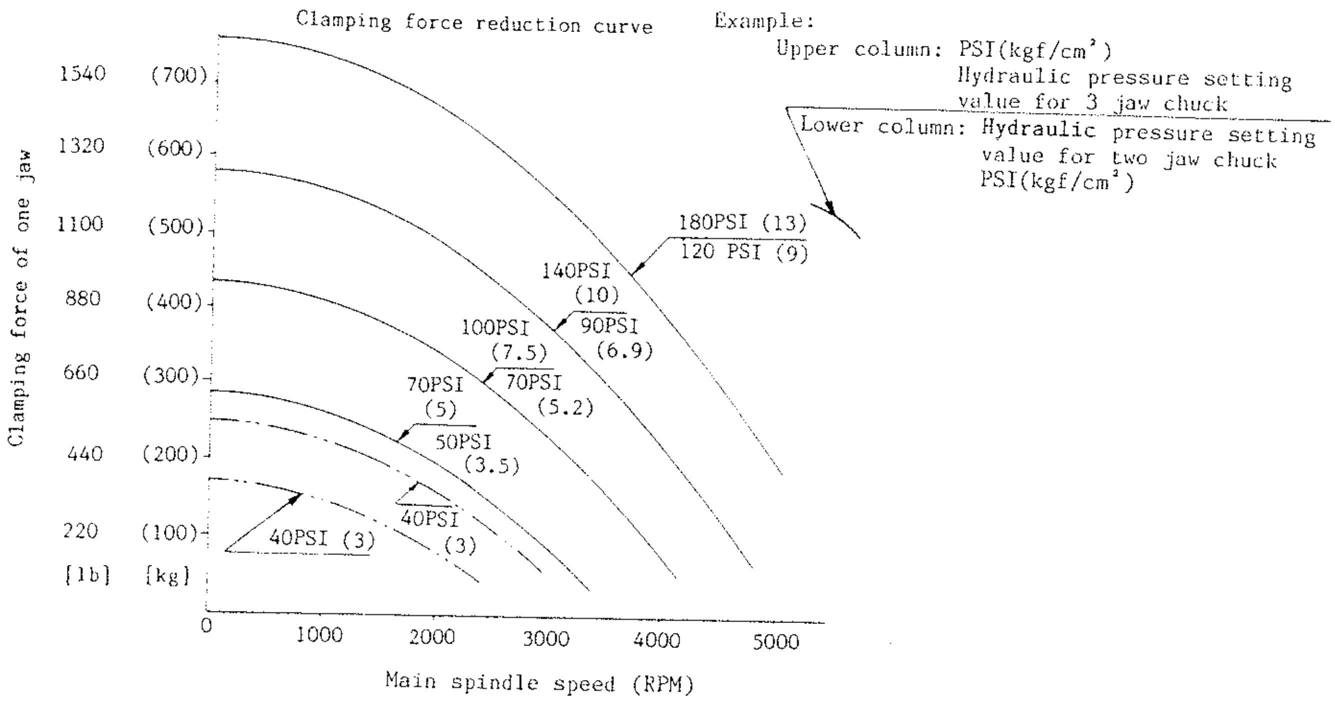
[1] Setting pressure for collet chuck (Bar work)

In the case of collet chuck, main chuck pressure is normally set at 500PSI (35kgf/cm^2) for BND-20S, BND-34S. (In case of thin wall pipe material, pressure reduction is required.)

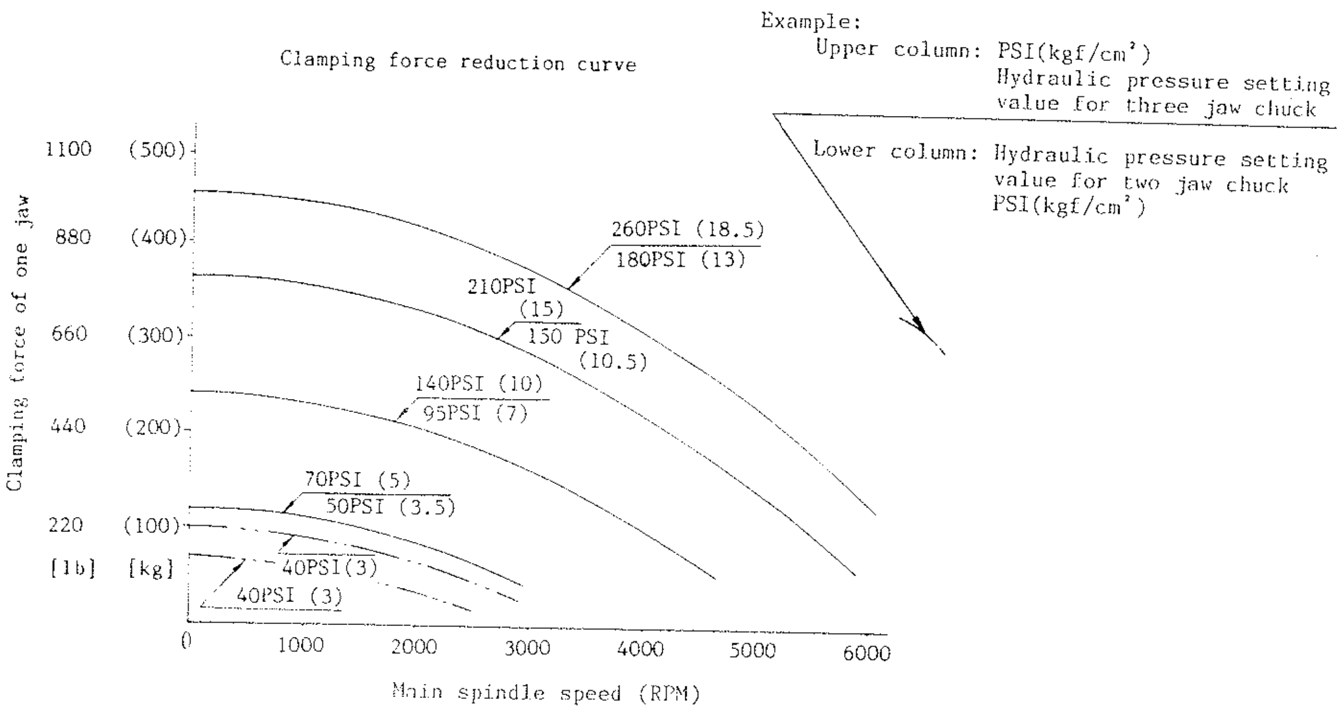
[2] Setting pressure for power chuck

For chucking pressure for power chuck is depending on type of power chuck, type of jaw, shape of work and rotation speed. Set by the graph of clamping force as shown is the figure below.

o B-05-28A (Three jaw chuck) and BT 05-28A (Two jaw chuck) for BND-34S

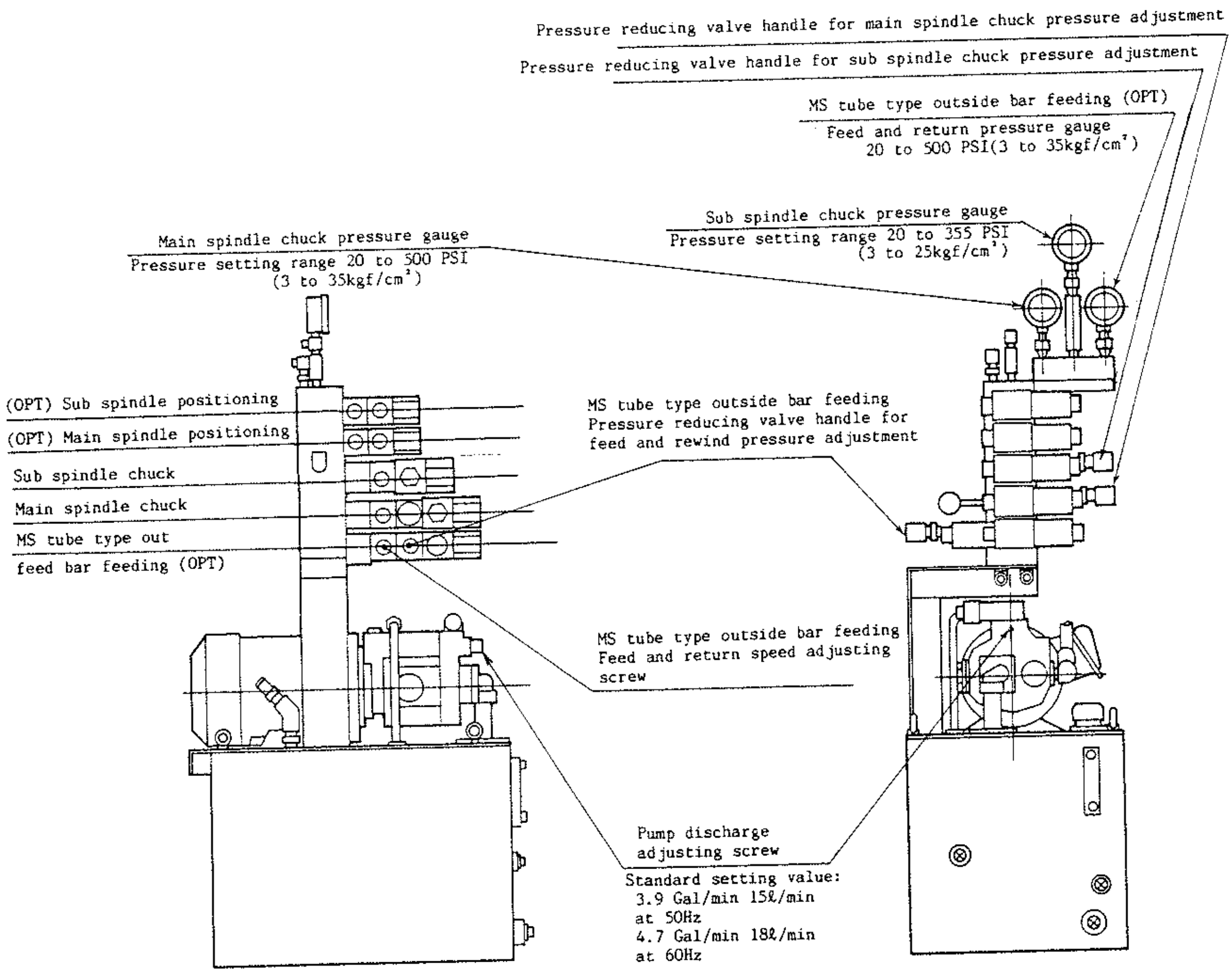


o B-04-28 (Three jaw chuck) and BT 04-28 (Two jaw chuck) for BND-20S

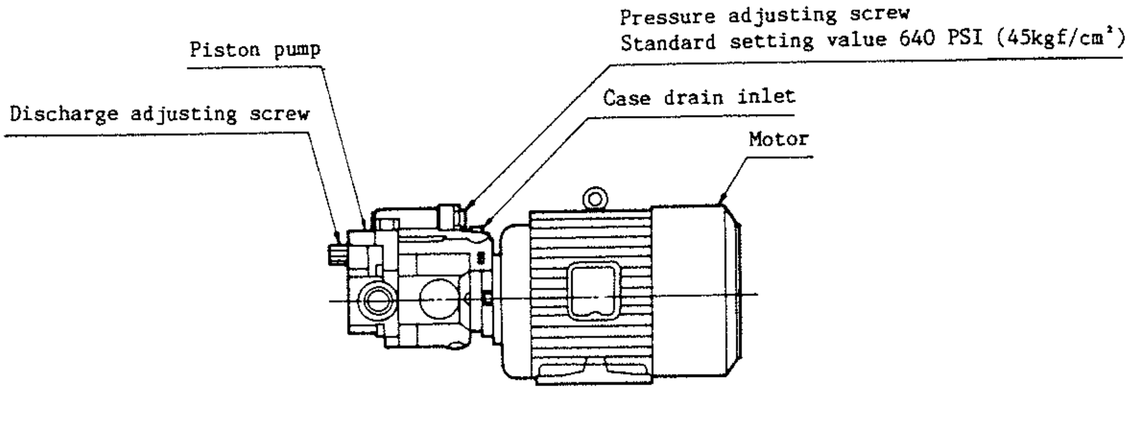


11. HYDRAULIC OPERATING

Hydraulic operating of main unit is necessary to adjust each section as shown in the figure below.

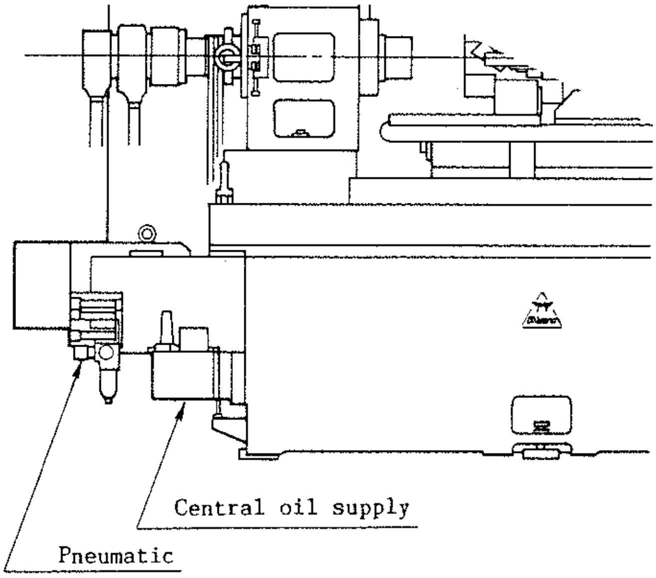


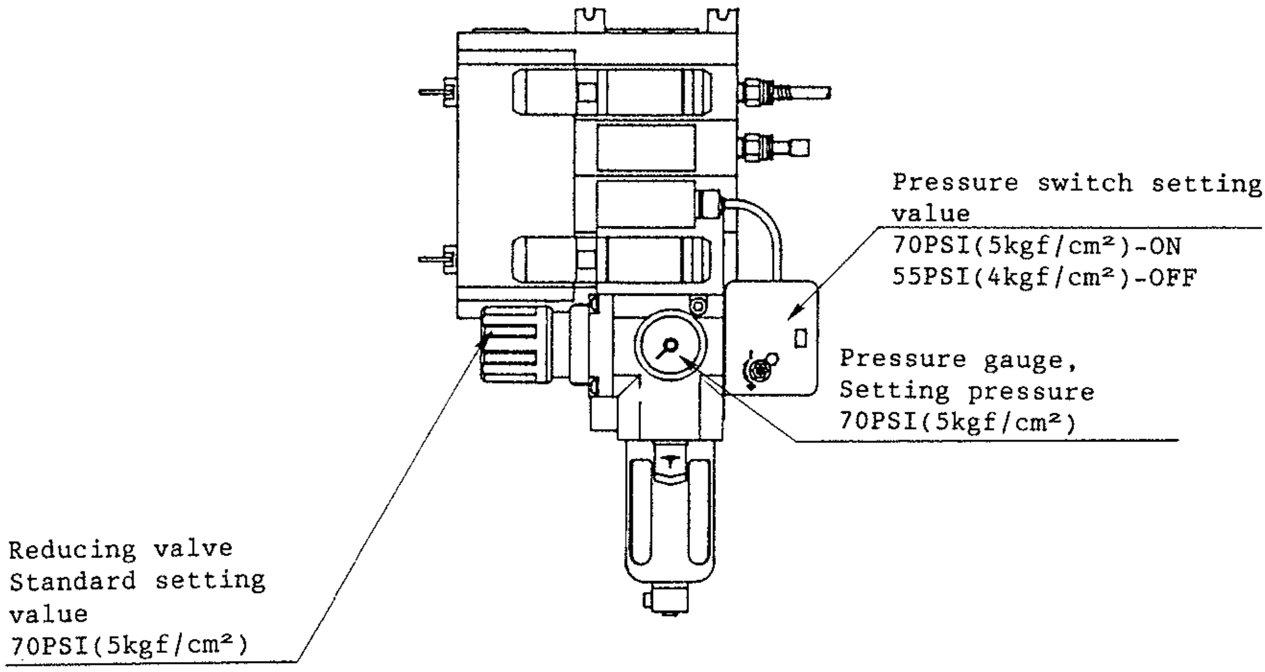
Detail of pump



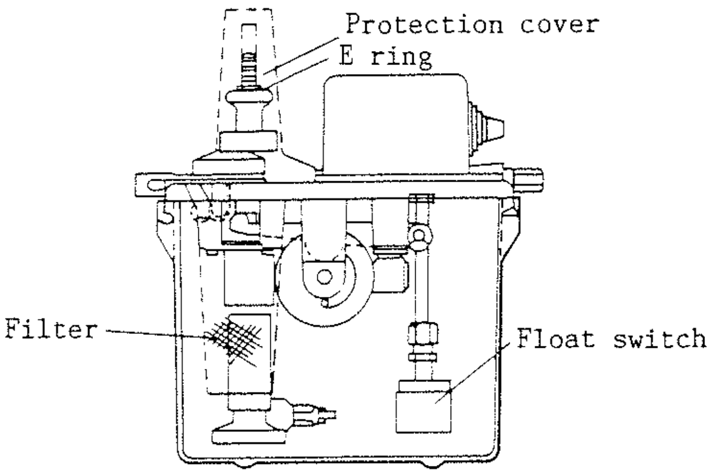
BND-S OTC OPE 90/7

12. PNEUMATIC

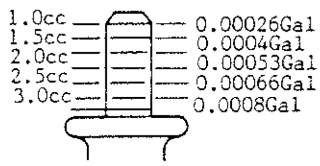




13. CENTRAL OIL SUPPLY



For adjustment of discharge, turn the instant button to the required graduation. After adjustment of discharge, set E ring.
Standard setting is 0.0005Gal/cy/15min (2cc/cy/15min).

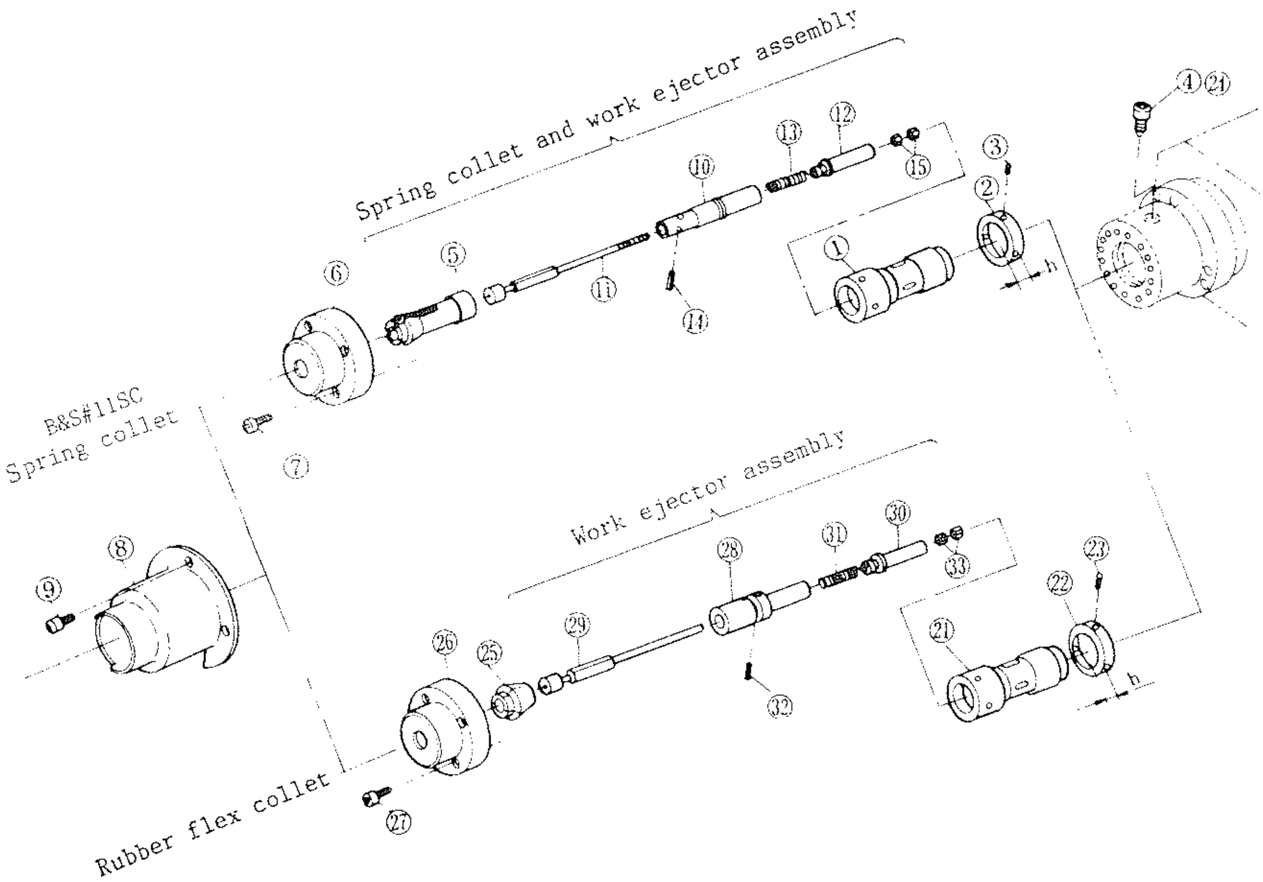


BND-S OTC OPE 90/7

14. BND-20S SUB SPINDLE CHUCK SYSTEM

BND-20S sub spindle work ejector system is available in two types described below. Change from spring type work ejector to air cylinder type work ejector cannot be made and both types cannot be used together.

[1] Spring type work ejector and chuck system (BND-20S) for USA



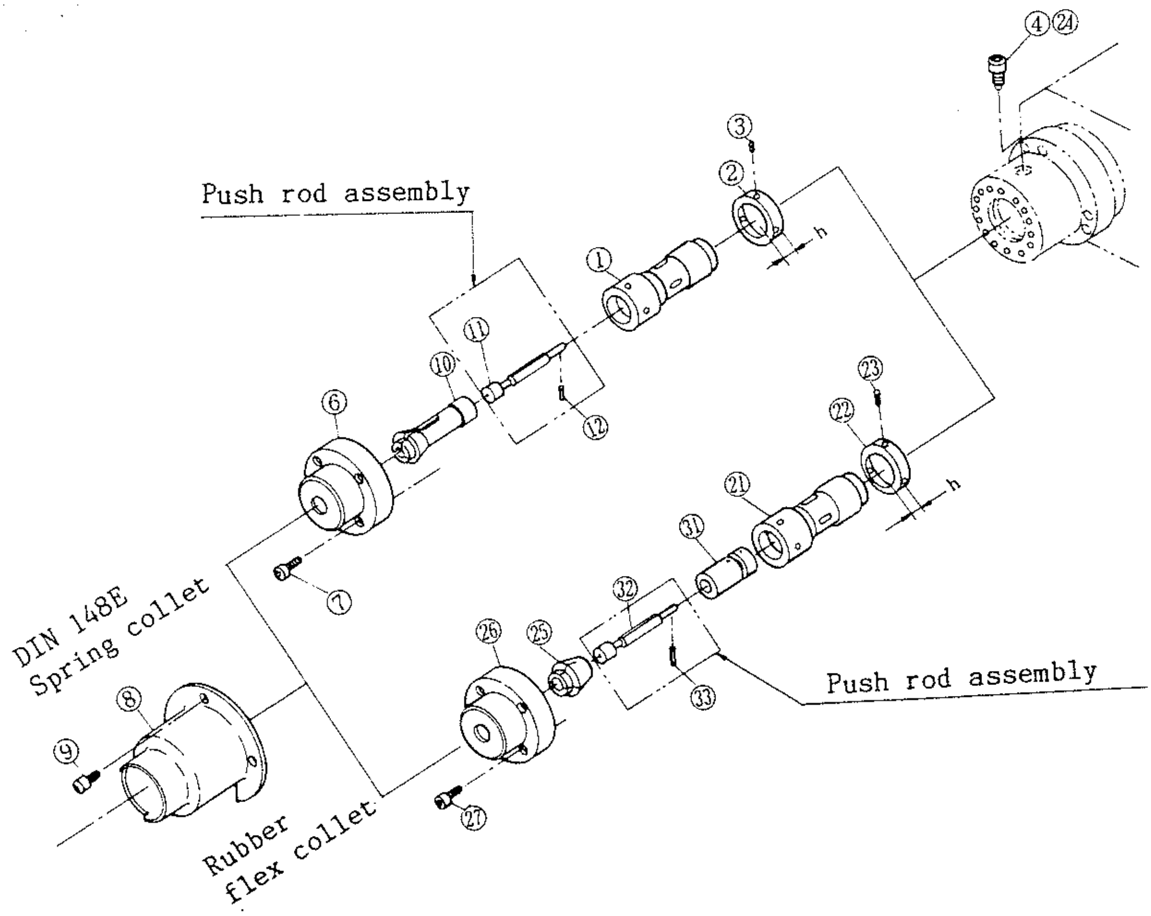
Part list of spring type work ejector and chuck system

Category	Code	Part number	Part name	Quantity	Remarks
Spring collet	1	5V602020	Chuck sleeve	1	
	2	5V603050	Spacer h=8.5	1	h=.3346"(8.5mm)
		5V603060	Spacer h=9.0	1	h=.3543"(9.0mm)
	3	4S44	Set screw	3	M4 x 0.7 L=4mm
	4	5V602030	Pin	2	
	5		Spring collet	1	Standard: B&S #11SC
	6	5V602010	Chuck cap		

Cate-gory	Code	Part number	Part name	Quantity	Remarks
Spring collet	7	3B616	Bolt	6	M6 x 1.0 L=16mm
	8	5V600430	Oil guard	1	
	9	3B48	Bolt	3	M4 x 0.7 L=8mm
	10	5V608510	Sleeve	1	
	11	5V602520	Push rod	1	Selected for each work piece
	12	5V602530	Plug	1	
	13	5V602540 5V602550 5V602560	Spring	1	Selected for each work piece
	14	9P26	Spring pin	1	ϕ2 L=6mm
	15	1N8	Hexagon nut	2	M8 x 1.25
Rubber flex collet	21	5V603020	Chuck sleeve	1	
	22	5V603030	Spacer h=.2756"(7.0mm)	1	Selected for each work piece
		5V603040	Spacer h=.315"(8.0mm)	1	
		5V603060	Spacer h=.3543"(9.0mm)	1	
		5V603070	Spacer h=.3937"(10.0mm)	1	
		5V603080	Spacer h=.4331"(11.0mm)	1	
	23	4S44	Set screw	3	M4 x 0.7 L=4mm
	24	5V603090	Pin	2	
	25	RF#18	Rubber flex collet	1	
	26	5V603010	Chuck cap	1	
	27	3B616	Bolt	6	M6 x 1.0 L=16mm
	28	5V603510	Sleeve	1	
	29	5V602520	Push rod	1	Selected for each work piece
	30	5V602530	Plug	1	
	31	5V602540 5V602550 5V602560	Spring	1	Selected for each work piece
	32	9P26	Spring pin	1	ϕ2 L=6mm
	33	1N8	Hexagon nut	2	M8 x 1.25
	8	5V600430	Oil guard	1	
	9	3B48	Bolt	3	M4 x 0.7 L=8mm

[2] Air cylinder type work ejector and chuck system (BND-20S) for excluding USA

Note : Maximum rotation speed for air cylinder type work ejector and chuck system is 3500RPM.



<u>Part list of air cylinder type work ejector and chuck system</u>

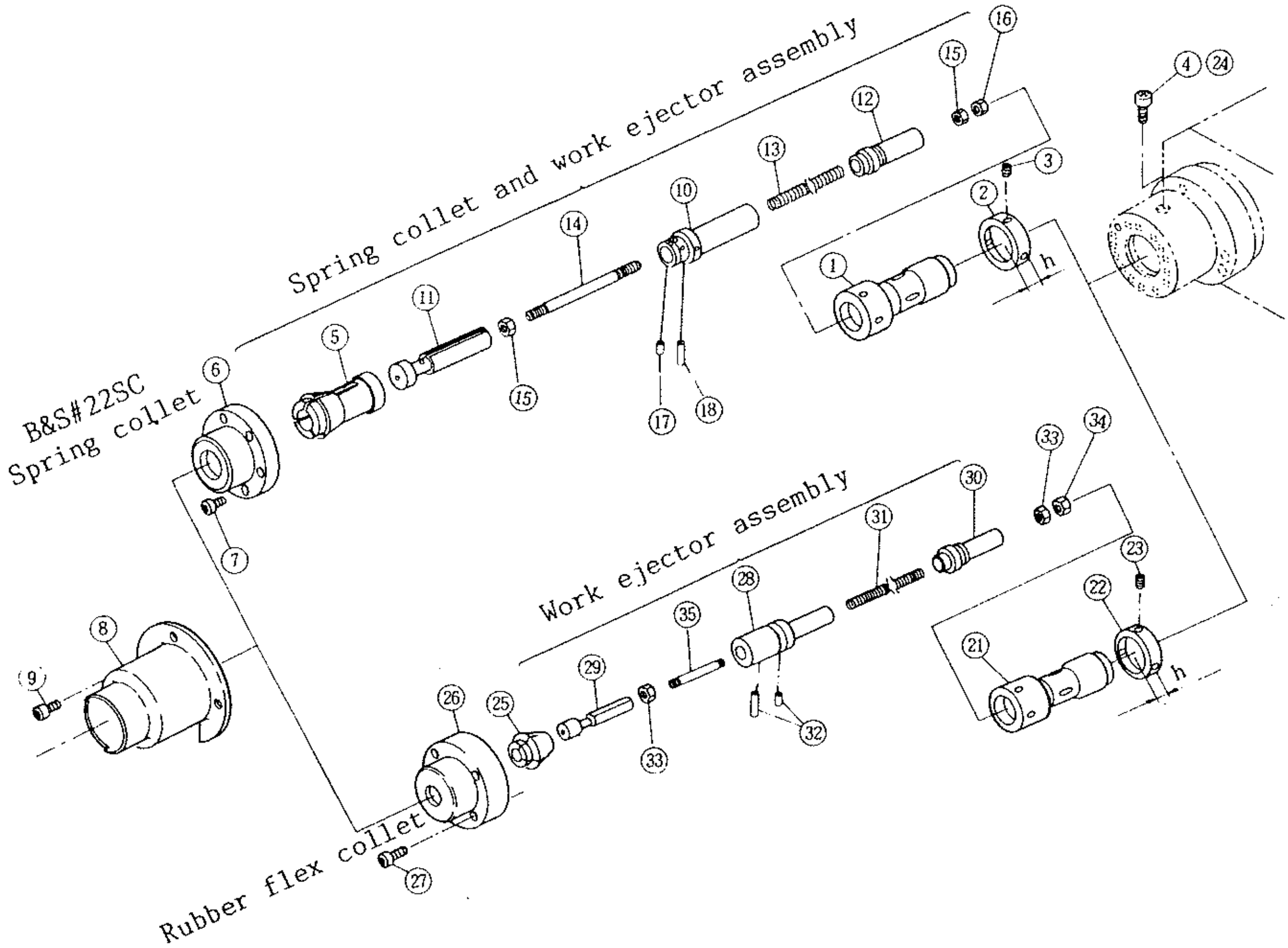
Cate-gory	Code	Part number	Part name	Quantity	Remarks
Spring collet	1	5V603020	Chuck sleeve	1	
	2	5V603050	Spacer h=8.5	1	h=8.5mm
		5V603060	Spacer h=9.0	1	h=9.0mm
	3	4S44	Set screw	3	M4 x 0.7 L=4mm
	4	5V602030	Pin	2	
	6	5V603010	Chuck cap	1	
	7	3B616	Bolt	6	M6 x 1.0 L=16mm
	8	5V600430	Oil guard	1	
	9	3B48	Bolt	3	M4 x 0.7 L=8mm
	10		Spring collet	1	Standard : DIN148E

Cate-gory	Code	Part number	Part name	Quantity	Remarks
Spring collet	11	5V602711	Push rod	1	Selected for each work piece
	12	9P316	Spring pin	1	
Rubber flex collet	21	5V603020	Chuck sleeve	1	
	22	5V603030	Spacer h=7.0mm	1	Selected for each work piece
		5V603040	Spacer h=8.0mm	1	
		5V603060	Spacer h=9.0mm	1	
		5V603070	Spacer h=10.0mm	1	
		5V603080	Spacer h=11.0mm	1	
	23	4S44	Set screw	3	M4 x 0.7 L=4mm
	24	5V603090	Pin	2	
	25	RF#18	Rubber flex collet	1	Selected for each work piece
	26	5V603010	Chuck cap	1	
	27	3B616	Bolt	6	M6 x 1.0 L=16mm
	8	5V600430	Oil guard	1	
	9	3B48	Bolt	3	M4 x 0.7 L=8mm
	31	5V603710	Sleeve	1	
	32	5V602711	Push rod	1	Selected for each work piece
	33	9P316	Spring pin	1	ϕ3 L=16mm

15. BND-34S SUB SPINDLE CHUCK SYSTEM

BND-34S sub spindle work ejector system is available in two types described below. Change from spring type work ejector to air cylinder type work ejector cannot be made and both types cannot be used together. Sub spindle collet system can be used for round shape work piece only.

[1] Spring type work ejector and chuck system (for USA)



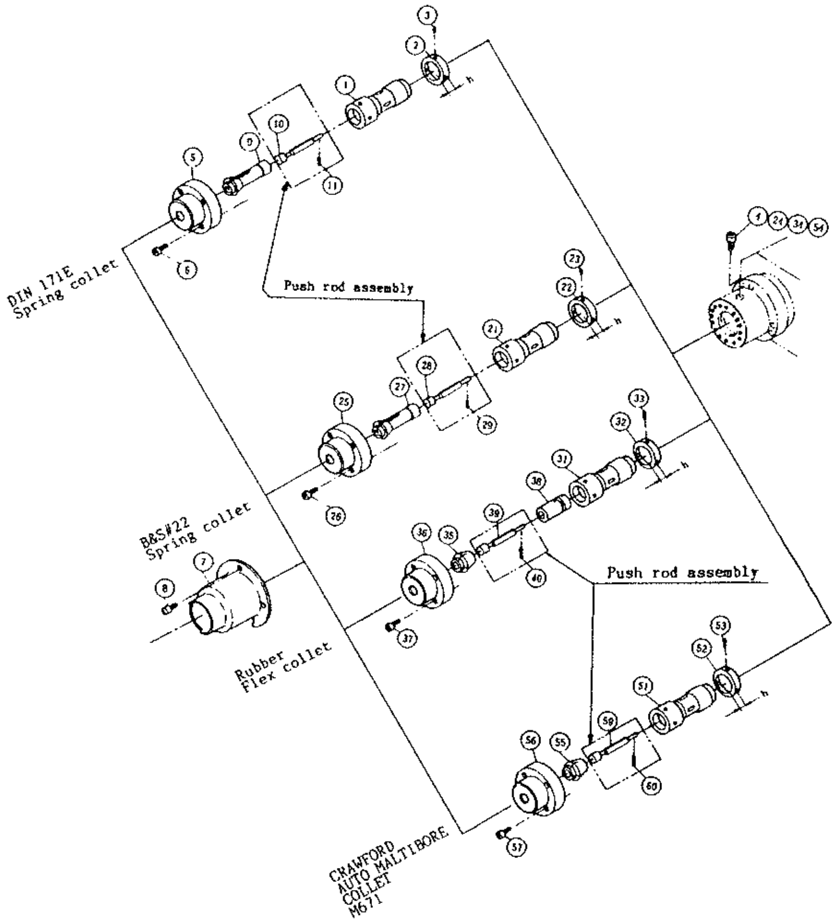
Part list of air cylinder type work ejector and chuck system

Cate-gory	Code	Part number	Part name	Quantity	Remarks
Spring collet	1	5W608020	Chuck sleeve	1	
	2	5W602030	Spacer h=10.5	1	h=.4134"(10.5mm)
		5W602040	Spacer h=11.0	1	h=.4331"(11.0mm)
	3	4S66	Set screw	3	M4 x 0.7 L=4mm
	4	5V603090	Pin	2	
	5		Spring collet	1	Standard: B&S #22SC
	6	5W608010	Chuck cap		

Cate-gory	Code	Part number	Part name	Quantity	Remarks
	7	3B616	Bolt	6	M6 x 1.0 L=.629"(16mm)
	8	5W600240	Oil guard	1	
	9	3B48	Bolt	3	M4 x 0.7 L=.314"(8mm)
	10	5W608510	Sleeve	1	
Spring collet	11		Push rod	1	Selected for each work piece
	12	5W602530	Plug	1	
	13	5W602550 5W602560 5W602570	Spring	1	Selected for each work piece
	14	5W602540	Spring guide	1	
	15	1N8	Hexagon nut	2	M8 x 1.25
	16	2N8	Nut	1	M8 x 1.25
	17	9P26	Spring pin	1	ϕ2 L=.236"(6mm)
	18	9P210	Spring pin	1	ϕ2 L=.393"(10mm)
	21	5W603020	Chuck sleeve	1	
	22	5W603040	Spacer h=.2559"(6.5mm)	1	Selected for each work piece
		5W603050	Spacer h=.2952"(7.5mm)	1	
		5W603060	Spacer h=.3346"(8.5mm)	1	
		5W603070	Spacer h=.374"(9.5mm)	1	
		5W603080	Spacer h=.4133"(10.5mm)	1	
Rubber flex collet	23	4S44	Set screw	3	M4 x 0.7 L=.157"(4mm)
	24	5W603030	Pin	2	
	25	RF#36	Rubber flex collet	1	
	26	5W603010	Chuck cap	1	
	27	3B616	Bolt	6	M6 x 1.0 L=.629"(16mm)
	28	5W603510	Sleeve	1	
	29		Push rod	1	Selected for each work piece
	30	5W602530	Plug	1	
	31	5W602550 5W602560 5W602570	Spring	1	Selected for each work piece
	32	9P212	Spring pin	2	ϕ2 L=.472"(12mm)
	8	5W600240	Oil guard	1	
	9	3B48	Bolt	3	M4 x 0.7 L=.314"(8mm)
	33	1N8	Hexagon nut	2	M8 x 1.25
	34	2N8	Nut	1	M8 x 1.25
	35	5W602540	Spring guide	1	

[2] Air cylinder type work ejector and chuck system (for excluding USA.)

Note : Maximum rotation speed for air cylinder type work ejector and chuck system is 3500RPM.



BND-S OTC OPE 90/7

Parts list of air cylinder type work ejector and chuck system

Cate-gory	Code	Part number	Part name	Quantity	Remarks
Spring collet DIN171E	1	5W604020	Chuck sleeve	1	
	2	5W602030	Spacer h=10.5	1	h=10.5mm
		5W602041	Spacer h=10.0	1	h=10.0mm
	3	4S66	Set screw	3	M6 x 1.0 L=6mm
	4	5V603090	Pin	2	
	5	5W604010	Chuck cap	1	
	6	3B616	Bolt	6	M6 x 1.0 L=16mm
	7	5W600240	Oil guard	1	
	8	3B48	Bolt	3	M4 x 0.7 L=8mm
	9		Spring collet	1	Standard : DIN171E

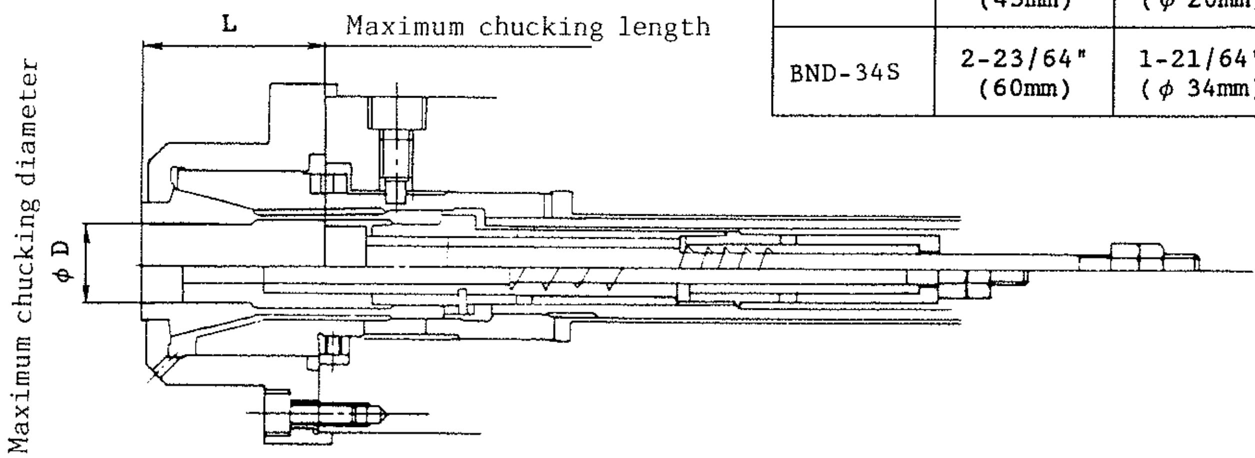
Category	Code	Part number	Part name	Quantity	Remarks
Spring collet DIN171E	10	5W602711 5W602721	Push rod	1	Selected for each work piece
	11	9P322	Spring pin	1	ϕ3 L=22mm
Spring collet B&S#22	21	5W608020	Chuck sleeve	1	
	22	5W602030	Spacer h=10.5	1	h=10.5mm
		5W602041	Spacer h=10.0	1	h=10.0mm
	23	4S66	Set screw	3	M6 x 1.0 L=6mm
	24	5V603090	Pin	2	
	25	5W608010	Chuck cap	1	
	26	3B616	Bolt	6	M6 x 1.0 L=16mm
	7	5W600240	Oil guard	1	
	8	3B48	Bolt	3	M4 x 0.7 L=8mm
	27		Spring collet	1	Standard : B&S#22
	28	5W602711 5W602721	Push rod	1	Selected for each work piece
	29	9P322	Spring pin	1	ϕ3 L=22mm
Rubber flex collet	31	5W603020	Chuck sleeve	1	
	32	5W603040	Spacer h=6.5mm	1	Selected for each work piece
		5W603050	Spacer h=7.5mm	1	
		5W603060	Spacer h=8.5mm	1	
		5W603070	Spacer h=9.5mm	1	
		5W603080	Spacer h=10.5mm	1	
	33	4S44	Set screw	3	M4 x 0.7 L=4mm
	34	5W60300	Pin	2	
	35	RF#36	Rubber flex collet	1	Selected for each work piece
	36	5W603010	Chuck cap	1	
	37	3B616	Bolt	6	M6 x 1.0 L=16mm
	7	5W600240	Oil guard	1	
	8	3B48	Bolt	3	M4 x 0.7 L=8mm
	38	5W603710	Sleeve	1	
	39	5W602711 5W602721	Push rod	1	Selected for each work piece
	40	9P322	Spring pin	1	ϕ3 L=22mm

Category	Code	Part number	Part name	Quantity	Remarks
Crawford auto maltibore collet M671	51	5W604020	Chuck sleeve	1	
	52	5W609010	Spacer h=6.5mm	1	Selected for each work piece
		5W609020	Spacer h=7.5mm	1	
		5W609030	Spacer h=9.5mm	1	
	53	4S66	Set screw	3	M6 x 1.0 L=6mm
	54	5W603090	Pin	2	
	55		Crawford auto maltibore collet M671	1	Selected for each work piece
	56	5W604010	Chuck cap	1	
	57	3B616	Bolt	6	M6 x 1.0 L=16mm
	7	5W600240	Oil guard	1	
	8	3B48	Bolt	3	M4 x 0.7 L=8mm
	59	5W602711 5W602721	Push rod	1	Selected for each work piece
	60	9P322	Spring pin	1	ϕ3 L=22mm

16. SPECIFICATION OF SUB SPINDLE SPRING TYPE WORK EJECTOR AND CHUCK SYSTEM

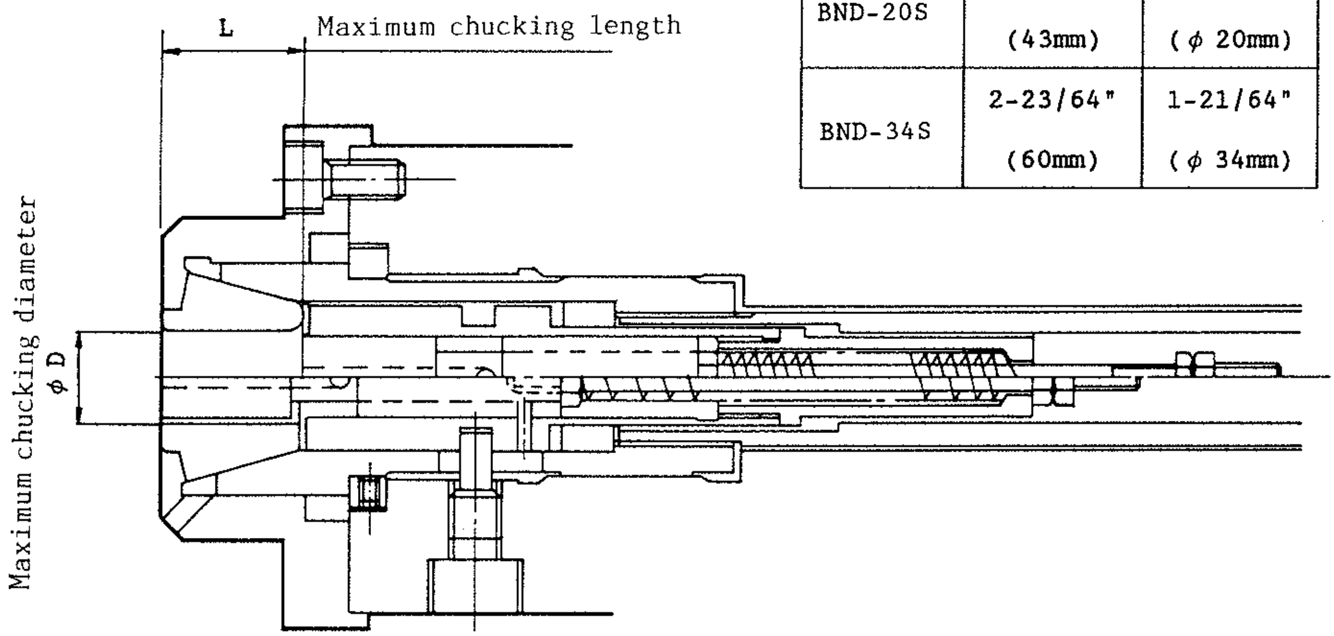
[1] In case of spring collet

	L	D
BND-20S	1-3/4" (45mm)	25/32" (ϕ 20mm)
BND-34S	2-23/64" (60mm)	1-21/64" (ϕ 34mm)



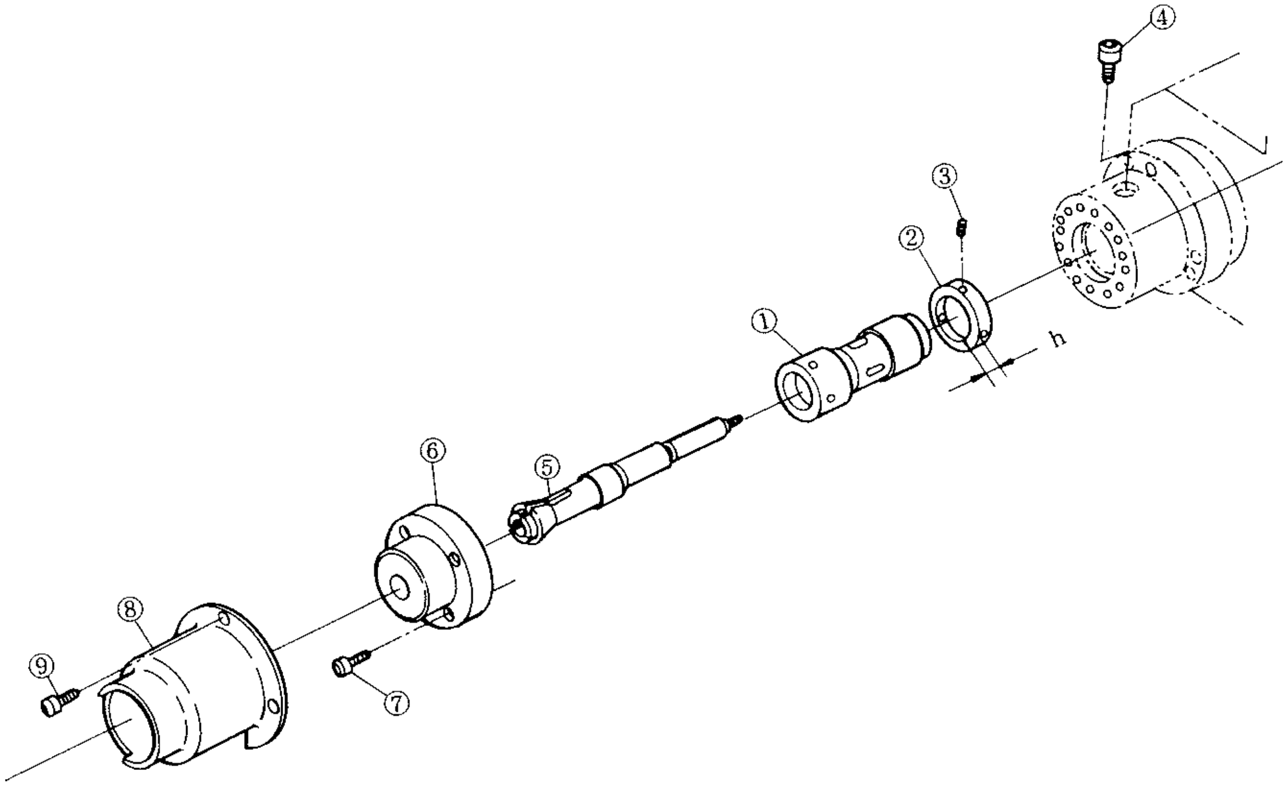
[2] In case of rubber flex collet

	L	D
BND-20S	1-11/16" (43mm)	25/32" (ϕ 20mm)
BND-34S	2-23/64" (60mm)	1-21/64" (ϕ 34mm)



17. INSTALLATION AND REMOVAL OF SPRING COLLET AND SPRING TYPE WORK EJECTOR

When replacement of chuck for the purpose of changing of set up and chuck cleaning is necessary, the following procedure is required.
(the same procedure as B & S #11SC and DIN-148E)

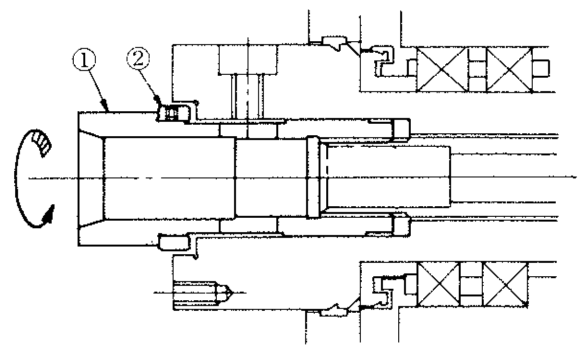


[1] Installation

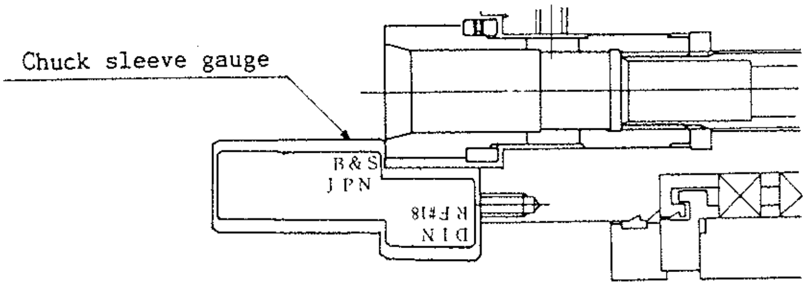
1) **Close the chuck. Confirm that the indicator lamp of sub spindle chuck close on the operating panel is turned on.**

2) Install a spacer h8.5(thickness) .3346"(8.5mm) for BND-20S, h10.5 (thickness) .4133"(10.5mm) for BND-34S2 ② to a chuck sleeve ① and lock it by set screws ③ . There are three set screws, tighten all of them.

3) Screw in the chuck sleeve ① as shown in the figure below. Chuck screw is right hand thread.

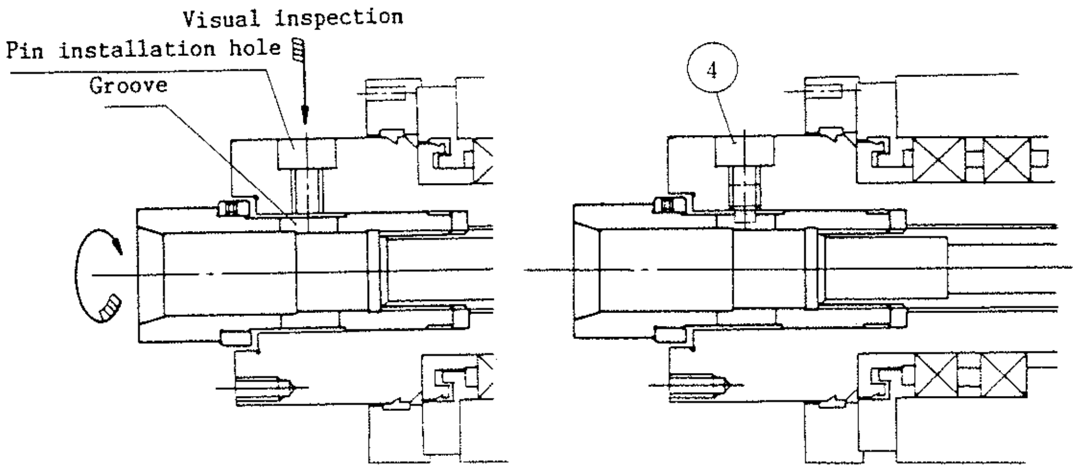


4) Mate the protrusion amount of chuck sleeve by a chuck sleeve gauge shipped with the machine as shown in the figure below. Incidentally, since the shape of chuck sleeve gauge differs depending on the model, match to "B & S or DIN" for BND-20S, BND-34S.



5) Install a pin 4

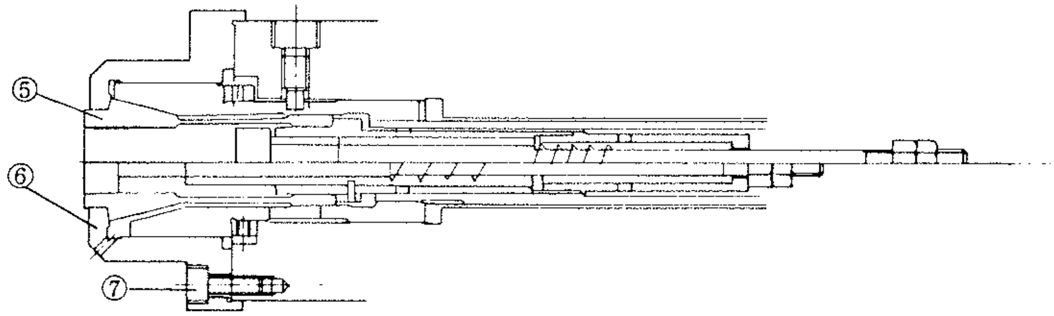
Rotate the chuck sleeve in the arrowed direction as shown in the figure below, and adjust the position of the chuck sleeve groove and pin hole. The groove of the chuck sleeve should be matched with the pin inserting hole within 1/4 revolution. When they are correctly matched, insert two pins surely. If only one pin is inserted, the unit may lose balance and vibration occurs during high speed operation.



6) Open the chuck. Confirm that the indicator lamp of sub spindle chuck close on the operating panel is turned off.

7) Insert a spring collet and work ejector assembly 5 into chuck sleeve.

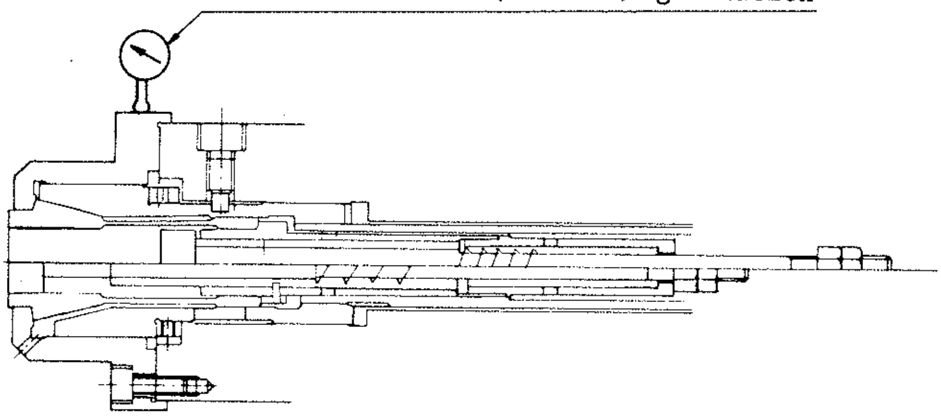
8) Install the chuck cap (6) temporarily by bolts (7).



9) Chuck the sample piece in the collet and check the amount of chuck opening and closing. Also check the work ejector motion. If the amount of chuck closing is not enough, rotate the chuck sleeve 1/4 turn in the arrowed direction. If the amount of chuck opening is not enough, the size of the chuck is not correct.

10) Check the runout of the spring collet.
Alignment of chuck cap is performed under chuck close condition. Set a dial indicator with .0001"(0.001mm) division of the scale on the outside of chuck cap as shown in the figure below. Rotate the sub spindle and adjust the runout of the chuck cap within .0004"(0.01mm). After adjustment, tighten the bolt (7) firmly and check the dial again. If this value is more than .0004"(0.01mm), adjust it again.

Dial indicator with .0001"(0.001mm) graduation



In order to minimize the runout of machined surface on the main spindle side and sub spindle side, it is necessary to use high accurate collet chuck and reduce the runout of the cap.

11) Install the oil guard (8) by bolts (9).

[2] Removal

Removal is performed in the reverse order of installation.

[3] Replacement of spring collet and work ejector.

When a spring collet and work ejector is replaced for setting change, the following procedure applies.

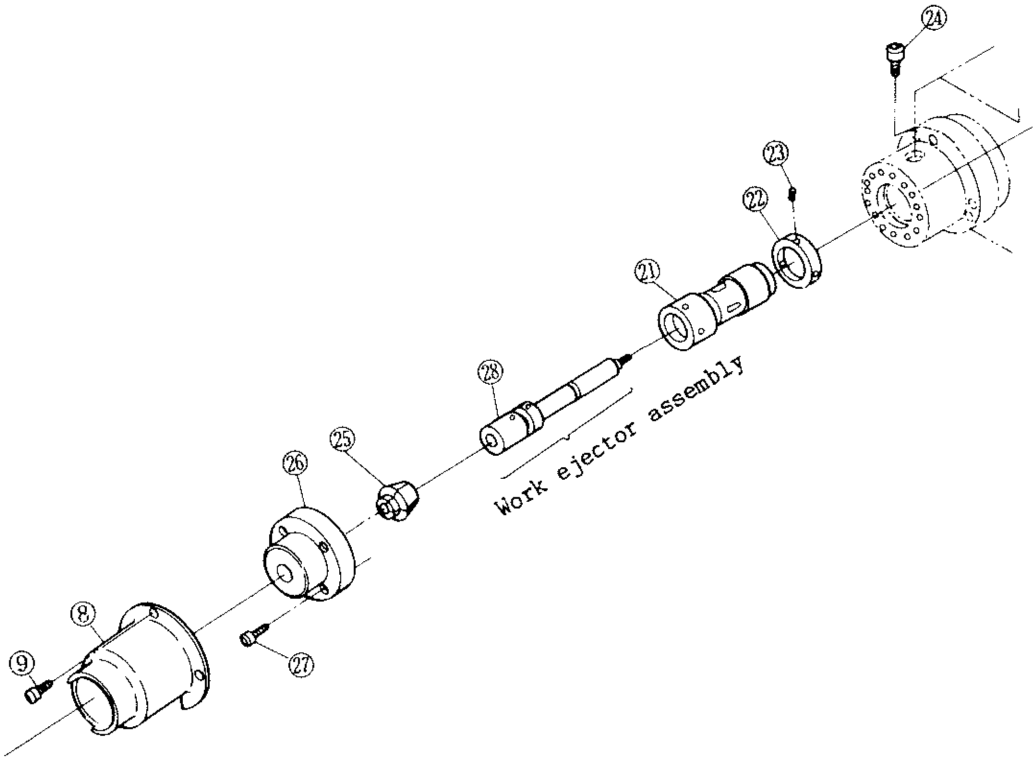
1) Remove the oil guard and chuck cap in the reverse order of installation.

2) Pull off old setting spring collet and work ejector under assembled condition.

3) Clean around chuck cap and sub spindle nose.

4) Install new setting spring collet and work ejector in the procedure subsequent to the item 6) of "Installation".

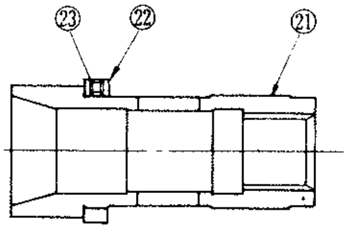
18. INSTALLATION AND REMOVAL OF RUBBER FLEX COLLET AND SPRING TYPE WORK EJECTOR



[1] Installation

1) Confirm that the indicator lamp of sub spindle chuck close on the operating panel is turned on.

2) **Fix a spacer h7(.2756") for BND-20S, h6.5(.2559") for BND-34S (a spacer is selected depending on the application) to a chuck sleeve 21 by set screw 23. Incidentally, since there are three set screws, tighten all of them.**



BND-20S

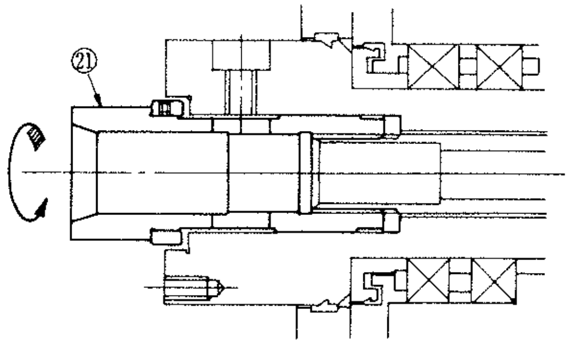
		Thickness	Work diameter
22	Spacer h.2756"(7) :	7mm	∿.0157"(0.4mm)
	Spacer h.315" (8) :	8mm	.0157"(0.4mm)∿.0315"(0.8mm)
	Spacer h.3543"(9) :	9mm	.0315"(0.8mm)∿.0472"(1.2mm)
	Spacer h.3937"(10) :	10mm	.0472"(1.2mm)∿.063" (1.6mm)
	Spacer h.4331"(11) :	11mm	.063" (1.6mm)∿.0787"(2.0mm)

BND-34S

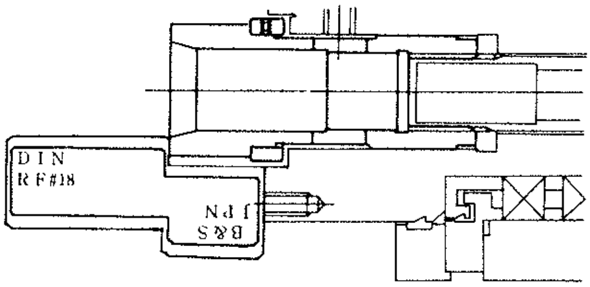
		Thickness	Work diameter
22	Spacer h.2559"(6.5):	6.5mm	~ .0157"(0.4mm)
	Spacer h.2952"(7.5):	7.5mm	.0157"(0.4mm)~ .0315"(0.8mm)
	Spacer h.3346"(8.5):	8.5mm	.0315"(0.8mm)~ .0472"(1.2mm)
	Spacer h.374" (9.5):	9.5mm	.0472"(1.2mm)~ .063" (1.6mm)
	Spacer h.4133"(10.5):	10.5mm	.063" (1.6mm)~ .0787"(2.0mm)

For selection of spacer, normally spacer h6.5(.2559") for BND-34S (h7.0(.2756") for BND-20S is used.
Other spacers are used in case of air cylinder type work ejector.

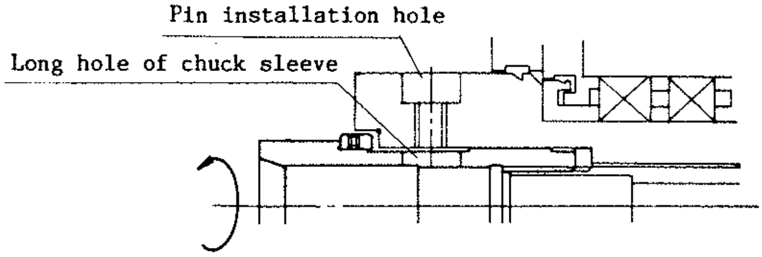
3) Screw in a chuck sleeve ㉑ . A chuck sleeve is right hand thread.



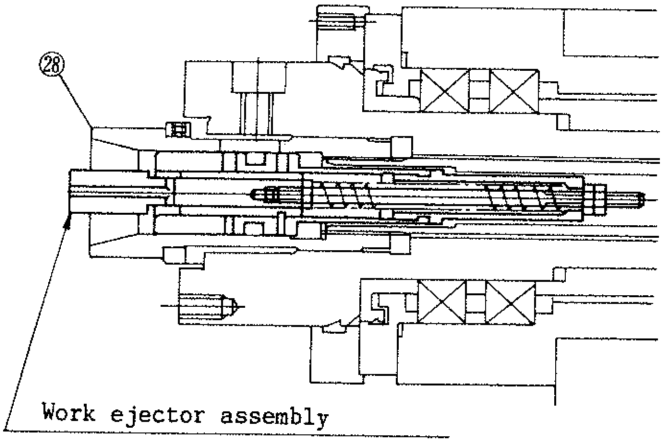
4) Mate protrusion amount of chuck sleeve to a chuck sleeve gauge delivered with the machine.



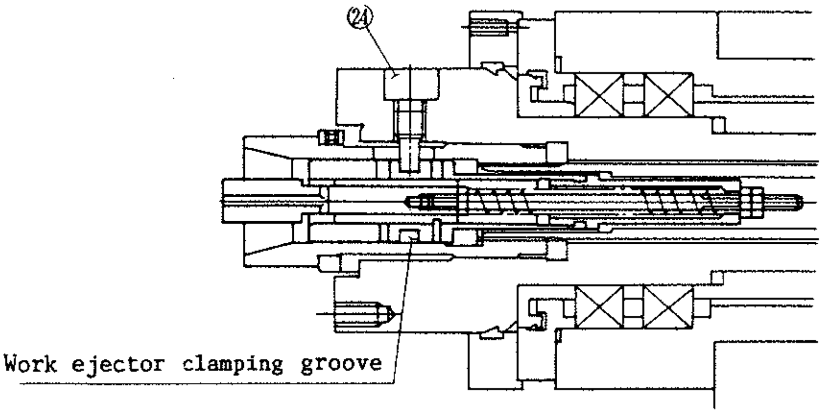
5) Adjust the pin inserting hole to match with the position of chuck sleeve long hole. Rotate the chuck sleeve in the arrowed direction as shown in the figure below. They should be matched within 1/4 revolution of the chuck sleeve.



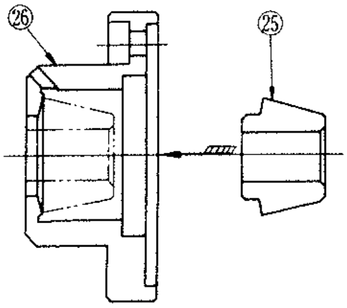
6) Open the chuck. Confirm that the indicator lamp of sub spindle chuck close on the operating panel is turned off.
Install the work ejector assembly (28) as shown in the figure below.



7) Install the pin (24) as shown in the figure below.
Confirm that these two pins are inserted in the groove of work ejector and tighten them firmly. These pins hold the work ejector assembly in longitudinal direction and prevent rotation of chuck sleeve screw.



8) Push in a rubber flex collet 25 in a chuck cap 26 in the direction of arrow in the figure below.



Range of rubber flex collet size for BND-20S

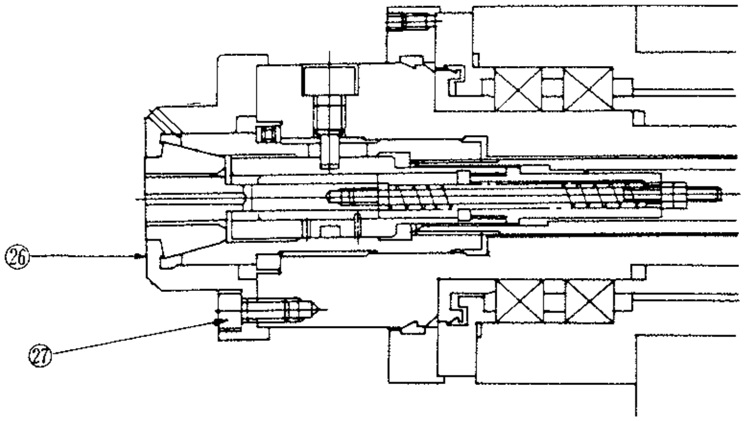
Part number	Range of material diameter	
	Unit mm	Unit inch
RF18 #1808 .315"	ϕ6 to ϕ8	.236 to .314
RF18 #1810 .3937"	ϕ8 to ϕ10	.315 to .393
RF18 #1812 .4724"	ϕ10 to ϕ12	.394 to .472
RF18 #1814 .5518"	ϕ12 to ϕ14	.473 to .551
RF18 #1816 .6299"	ϕ14 to ϕ16	.552 to .629
RF18 #1818 .7087"	ϕ16 to ϕ18	.63 to .708
RF18 #1820 .7874"	ϕ18 to ϕ20	.709 to .787

Collet size

Range of rubber flex collet size for BND-34S

Part number	Range of material diameter	
	Unit inch	Unit mm
RF36 #3612	.472 to .511	ϕ 12 to ϕ 13
RF36 #3614	.512 to .590	ϕ 13 to ϕ 15
RF36 #3616	.591 to .669	ϕ 15 to ϕ 17
RF36 #3618	.670 to .748	ϕ 17 to ϕ 19
RF36 #3620	.749 to .826	ϕ 19 to ϕ 21
RF36 #3622	.827 to .905	ϕ 21 to ϕ 23
RF36 #3624	.906 to .984	ϕ 23 to ϕ 25
RF36 #3626	.985 to 1.062	ϕ 25 to ϕ 27
RF36 #3628	1.063 to 1.141	ϕ 27 to ϕ 29
RF36 #3630	1.142 to 1.220	ϕ 29 to ϕ 31
RF36 #3632	1.221 to 1.299	ϕ 31 to ϕ 33
RF36 #3634	1.300 to 1.338	ϕ 33 to ϕ 34

9) Install the chuck cap by bolts 27 . At this time, tighten the bolts temporarily.



10) Adjustment of the amount of chuck opening and closing, alignment of chuck cap and installation of oil guard are performed in the same procedure as subsequent to the item 9) of installation of spring collet and spring type work ejector.

[2] Removal

Removal is performed in the reverse order of installation.

[3] Replacement of rubber flex collet and work ejector assembly

When a rubber flex collet and work ejector assembly are replaced for setting change, the following procedure applies.

1) Remove the oil guard and chuck cap in the reverse order of installation.

2) Remove the pin 24 . Since there are two pins, remove both two pins.

3) Remove the work ejector assembly for old setting under assembled condition.

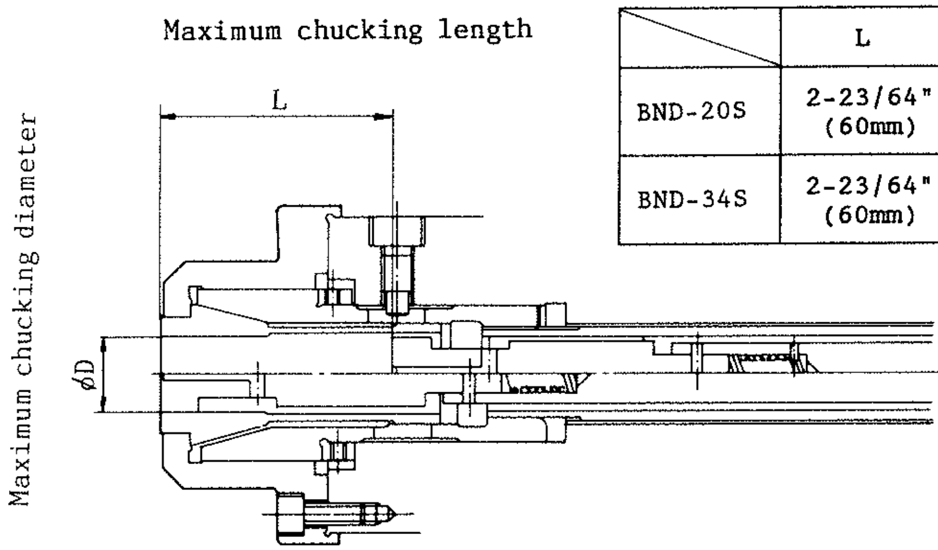
4) Clean around chuck cap and sub spindle nose.

5) Install the work ejector assembly for new setting, pin and chuck cap in the procedure subsequent to the item 6) of "Installation" after confirmation of closing and opening amount of collet and alignment of chuck cap, and install the oil guard.

19. SPECIFICATION OF AIR CYLINDER TYPE WORK EJECTOR AND CHUCK SYSTEM

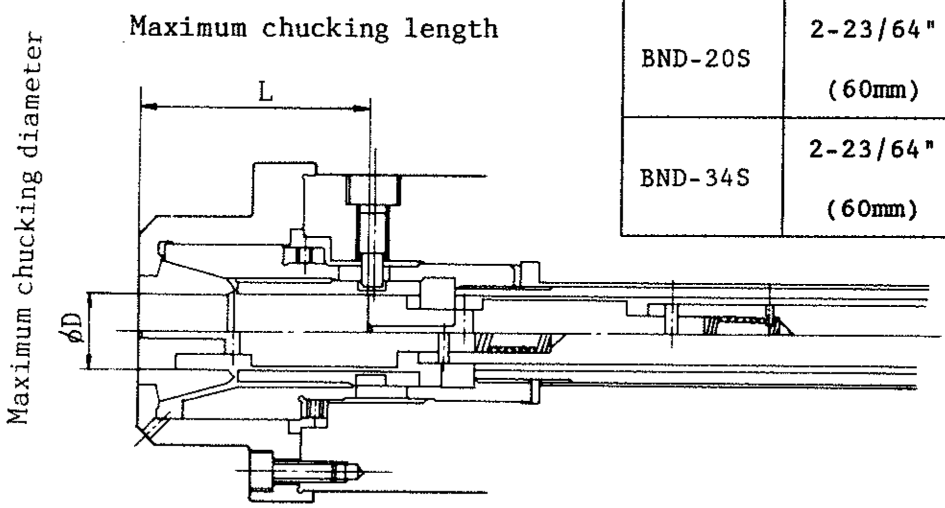
[1] In case of spring collet

	L	D
BND-20S	2-23/64" (60mm)	25/32" (ϕ 20mm)
BND-34S	2-23/64" (60mm)	1-21/64" (ϕ 34mm)



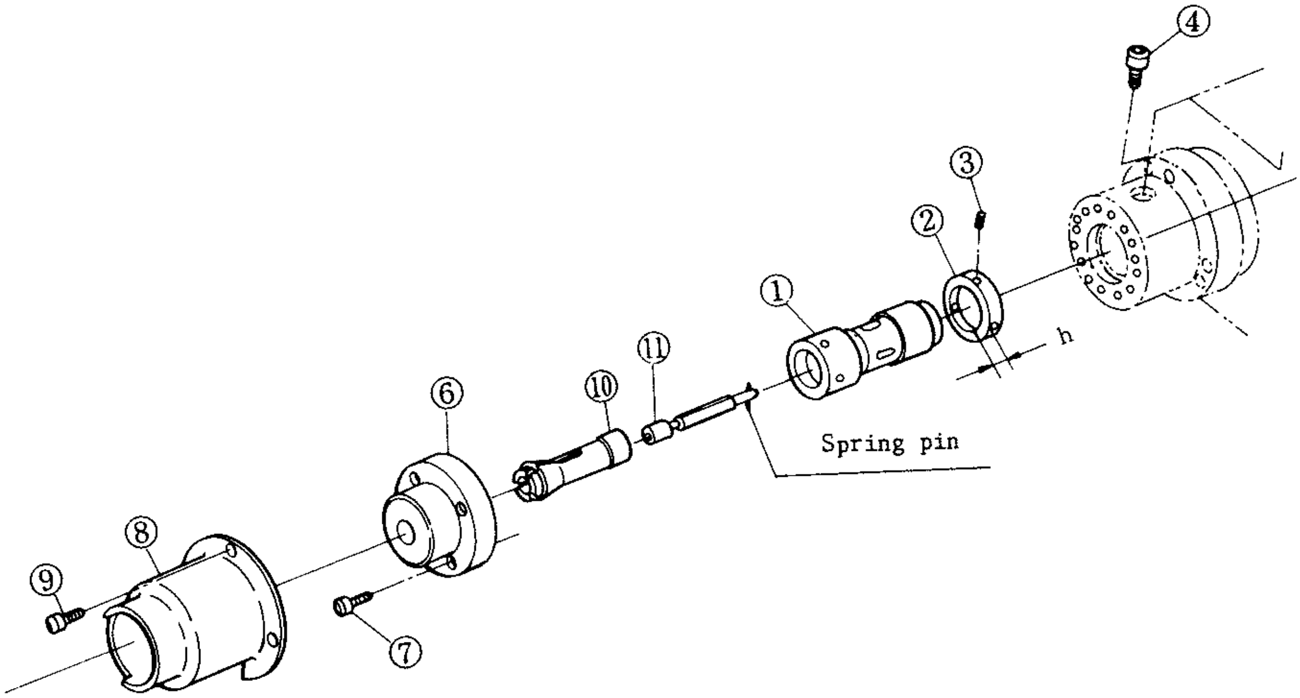
[2] In case of rubber flex collet

	L	D
BND-20S	2-23/64" (60mm)	25/32" (ϕ 20mm)
BND-34S	2-23/64" (60mm)	1-21/64" (ϕ 34mm)



In case of air cylinder type work ejector, maximum rotation speed of sub spindle is 3500RPM.
When the machine is used above 3500 RPM, vibration and noise may occur.

20. INSTALLATION AND REMOVAL OF SPRING COLLET AND AIR CYLINDER TYPE WORK EJECTOR FOR BND-20S, BND-34S

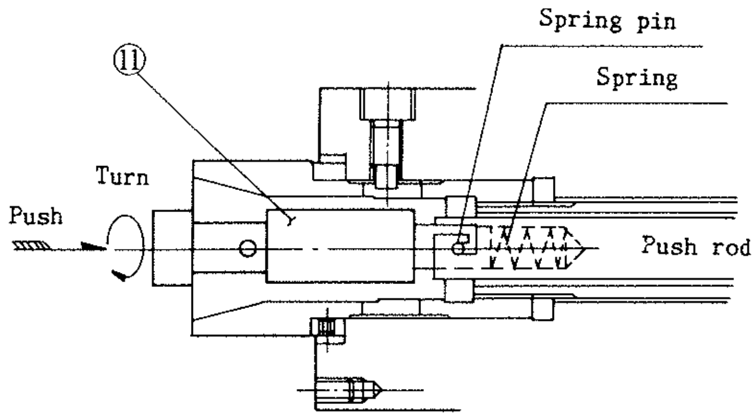


[1] Installation

Since installation of spring collet and air cylinder type work ejector is almost the same procedure as spring collet and spring type work ejector, so, only different items are explained.

a) For up to installation of chuck sleeve, refer to P118 to P119, 17[1] 1) to 6).

b) Install the work ejector assembly (11) to a push rod (figure below). On this push rod, a hook type groove and spring is provided to prevent coming off the work ejector. In order to install the work ejector, push it in and turn it as shown in the figure below. (when the work ejector is removed, it can be pulled off when it is pushed in and turned in the reverse direction of arrow.)



c) Insert a spring collet (11) into chuck sleeve.

d) Since subsequent installation of chuck cap, confirmation of closing and opening amount, alignment and instation of oil guard are the same procedure as spring collet of spring type work ejector specification, refer to P120 8) to 11).

[2] Removal

Removal is performed in the reverse procedure of installation.

[3] Replacement of spring collet and work ejector

When a spring collet and work ejector is replaced for setting change, the following procedure applies:

1) Remove the oil guard and chuck cap in the reverse procedure of "Installation".

2) Pull off the spring collet.

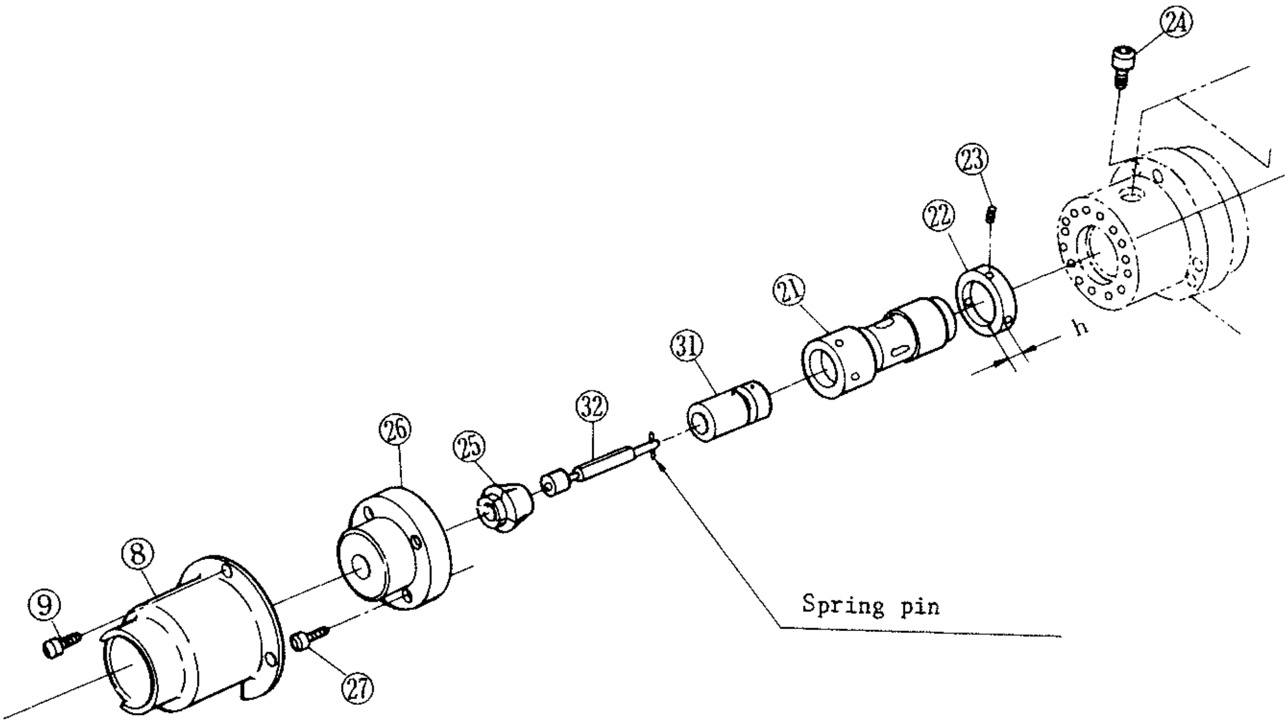
3) Remove the work ejector in the reverse procedure of "Installation". (Push and turn in opposite side of arrow direction.)

4) Clean around chuck cap and sub spindle nose.

5) By referring to b), c) and d) of "Installation", work edjector for new statting, spring collet and chuck cap are installed and alignment and installation of oil guard are performed.

BND-S OTC OPE 90/7

21. INSTALLATION AND REMOVAL OF RUBBER FLEX COLLET AND AIR CYLINDER TYPE WORK EJECTOR FOR BND-20S, BND-34S



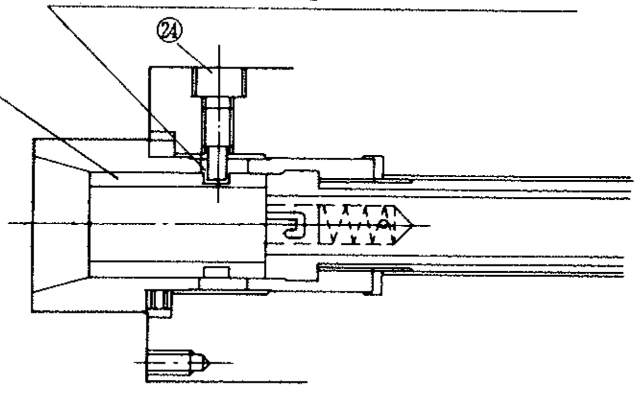
[1] Installation

Since installation of rubber flex collet and air cylinder type work ejector is almost the same procedure as rubber flex collet and spring type work ejector, so, only different items are explained.

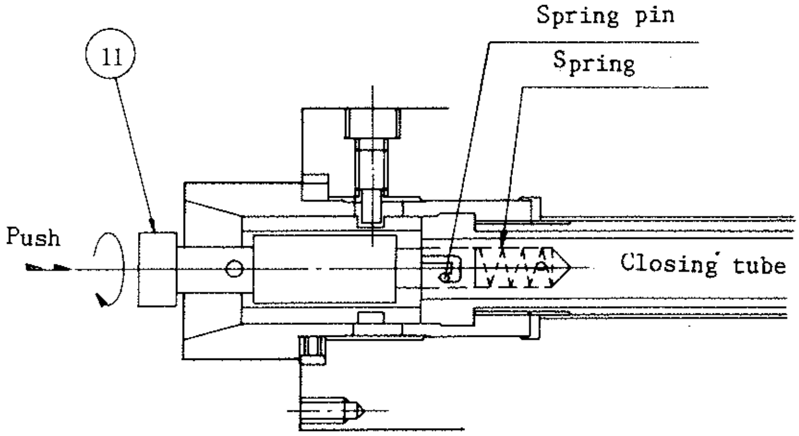
a) Refer to P122 to P123 18 1) to 5) for up to installation of chuck sleeve.

b) Insert the sleeve (31) into chuck sleeve.

c) Install the pin (24) as shown in the figure below. Confirm that these two pins are inserted in the groove of the sleeve and tighten them firmly. These pins hold the sleeve in longitudinal directhion and retaining the chuck sleeve.



d) Install the work ejector assembly (11) to closing tube.
On this closing tube, a hook type groove and spring is provided to prevent coming off the work ejector. In order to install the work ejector, push it in and turn it as shown in the figure below. (When removed, push the work ejector once and pull it off by turning in opposite direcion of arrow.)



e) Refer to P125 to P126 18 8) to 10) for alignment of chuck cap, rubber flex collet and chuck cap and installation of oil guard.

[2] Removal

Removal can be performed in the reverse procedure of installation.

[3] Replacement of rubber flex collet and work ejector

When a rubber flex collet and work ejector is replaced for setting, the following procedure applies:

1) Remove the oil guard and chuck cap in the reverse procedure of installation.

2) Remove the work ejector in the reverse procedure of "Installation". (Push and turn in opposite side of arrow direction.)

3) Clean around chuck cap and sub spindle nose.

4) Install the work ejector for new setting and rubber flex collet and chuck cap in the procedure of d) and e) of "Installation" and after alignment, install the oil guard.

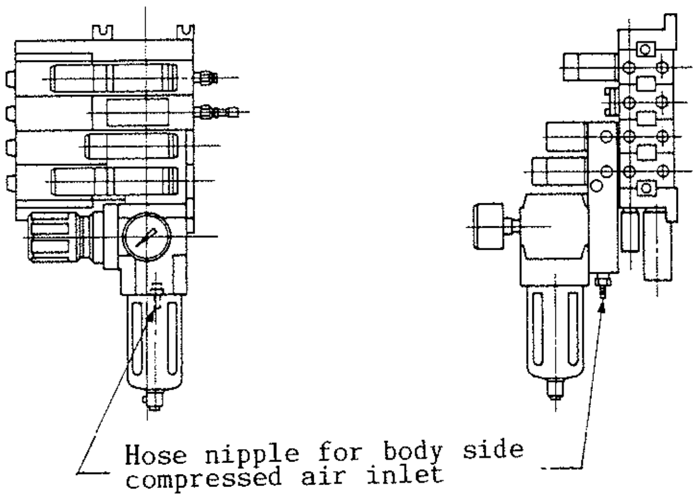
BND-S OTC OPE 90/7

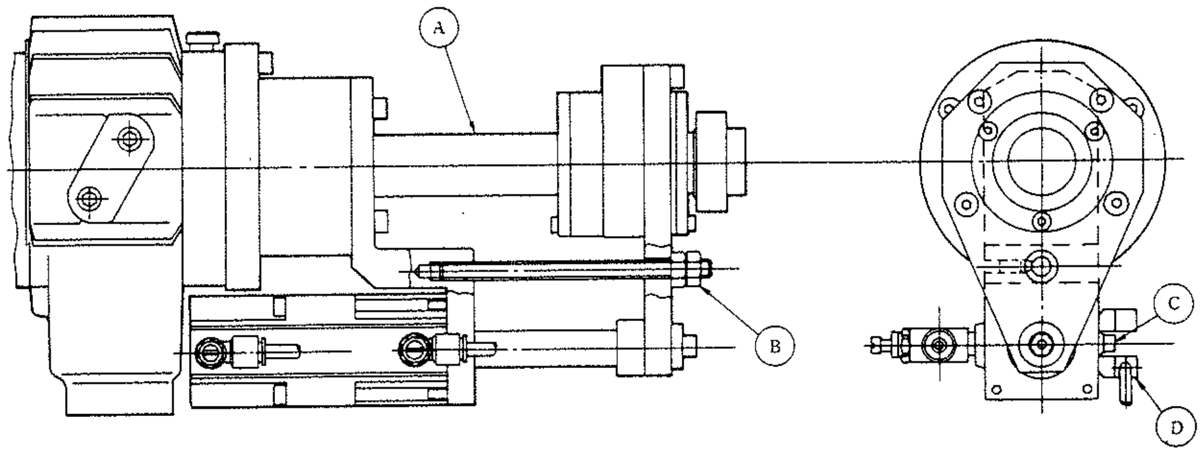
22. ADJUSTMENT OF CLAMP LENGTH OF THE AIR TYPE WORK EJECTOR

Adjust the work clamp length is the following procedure within the range of the maximum chucking length of the paragraph 18 "Specification for the air cylinder type work ejector and chuck system".

1) Open the sub chuck. Confirm that a neon lamp of "Chuck open" for the sub spindle is turned OFF on the separating panel.

2) Remove an air hose from a hose nipple of compressed air intake port at pneumatic main unit side as shown in the figure below.





3) Push the push rod Ⓐ to the spindle direction and move so that a distance from an end of collet and end of push rod may become work clamp length .039"(+1mm).

4) Loosen a hexagon nut Ⓑ and fix so that a stopper of cylinder original position may be located at E position by work clamp length.

5) Loosen a cross hold pan screw Ⓒ and move an original position auto switch Ⓓ to the position determined by work clamp length, at this time after it is confirmed that a red lamp of auto switch is turned on, tightenly by Ⓓ) to the position determined work clamp rength tighted by Ⓒ .

6) Install an air hose to the hose nipple.

BND-S OTC OPE 90/7

23. Bar Feeding System

The following bar feeding system is provided with this machine.

For BND-34S

Single bar feeder
- ASR F-34M
- #6M880000
  MS tube type
  outside bar feeding

Magazine bar feeder
- PF-V3M

BND-34S

For BND-20S

Single bar feeder
- ASR F-20M
- #6L880000
  MS tube type
  outside bar feeding

Magazine bar feeder
- PF-V2M

BND-20S

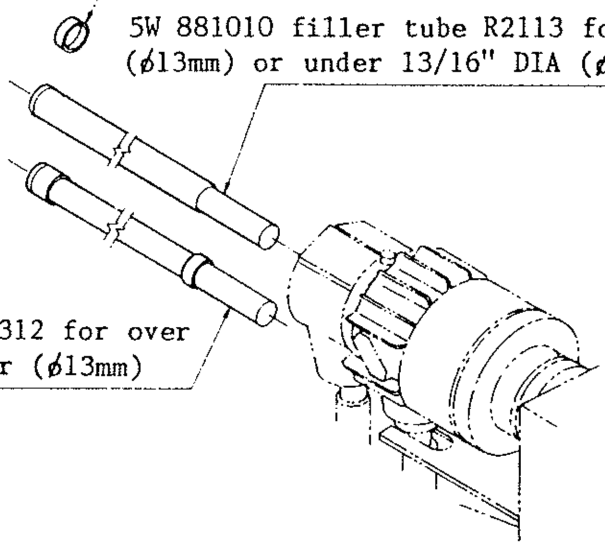
Refer to the instruction manual published by the maker (Alps Tool) for detail concering above-mentioned bar feeder other then MS tube type outside bar feeding. Here, only handling method of machine side concerning MS tube outside bar feeding, ASR F-34M/20M and PF-V3M/V2M is described.

[1] ASR F-34M/F-20M

1) Replacement part for ASR F-34M

5W 881050 bushing 3421 for over 27/32" DIA (ϕ21mm) or under 1-5/16" DIA (ϕ34mm)

5W 881010 filler tube R2113 for over 17/32" DIA (ϕ13mm) or under 13/16" DIA (ϕ21mm)

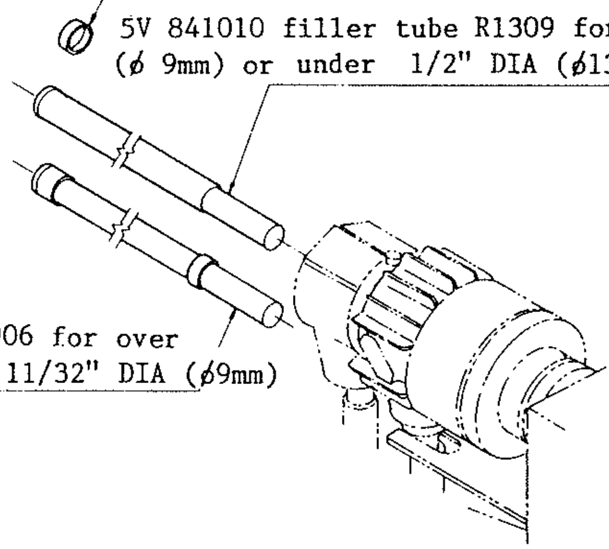


5W 881020 filler tube R1312 for over 1/2" DIA (ϕ12mm) or under (ϕ13mm)

2) Replacement part for ASR F-20M

5V 841050 bushing for over 17/32" DIA (ϕ13mm) or under 25/32" DIA (ϕ20mm)

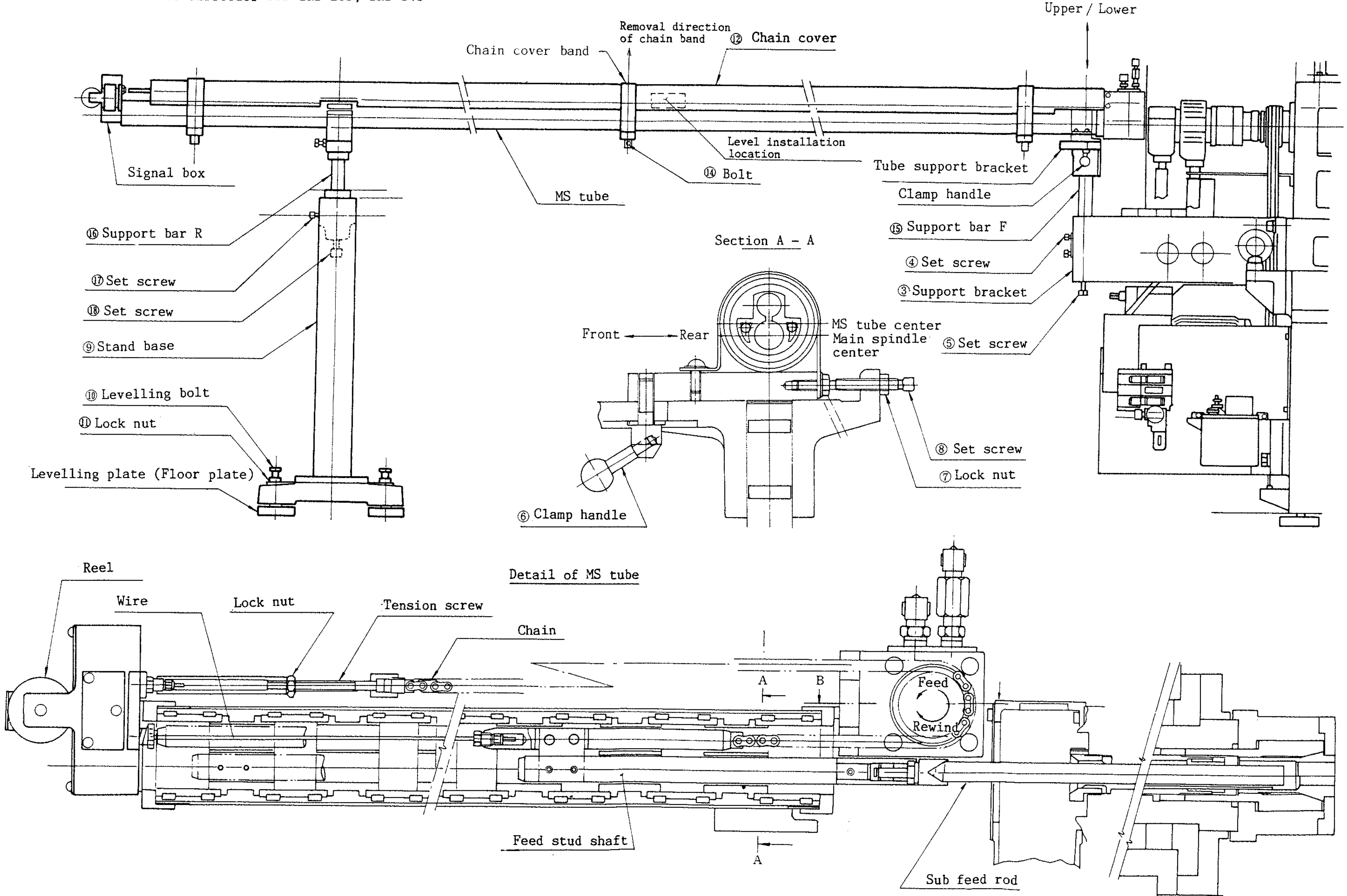
5V 841010 filler tube R1309 for over 3/8" DIA (ϕ 9mm) or under 1/2" DIA (ϕ13mm)



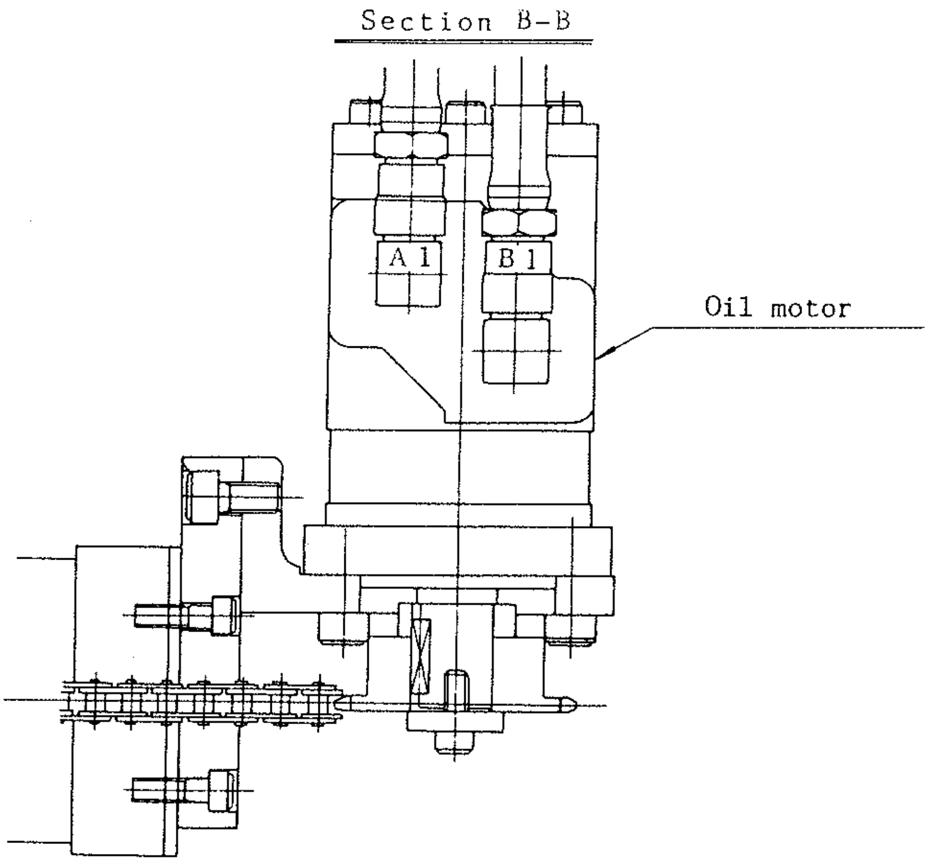
5V 841020 filler tube R0906 for over 1/4" DIA (ϕ6mm) or under 11/32" DIA (ϕ9mm)

BND-S OTC OPE 90/7

[2] Construction of MS tube barfeeder for BND-20S, BND-34S



BND-S OTC OPE 90/7



1)Specification of MS tube bar feeder

This device guides a bar material by MS tube and feeds by oil motor.

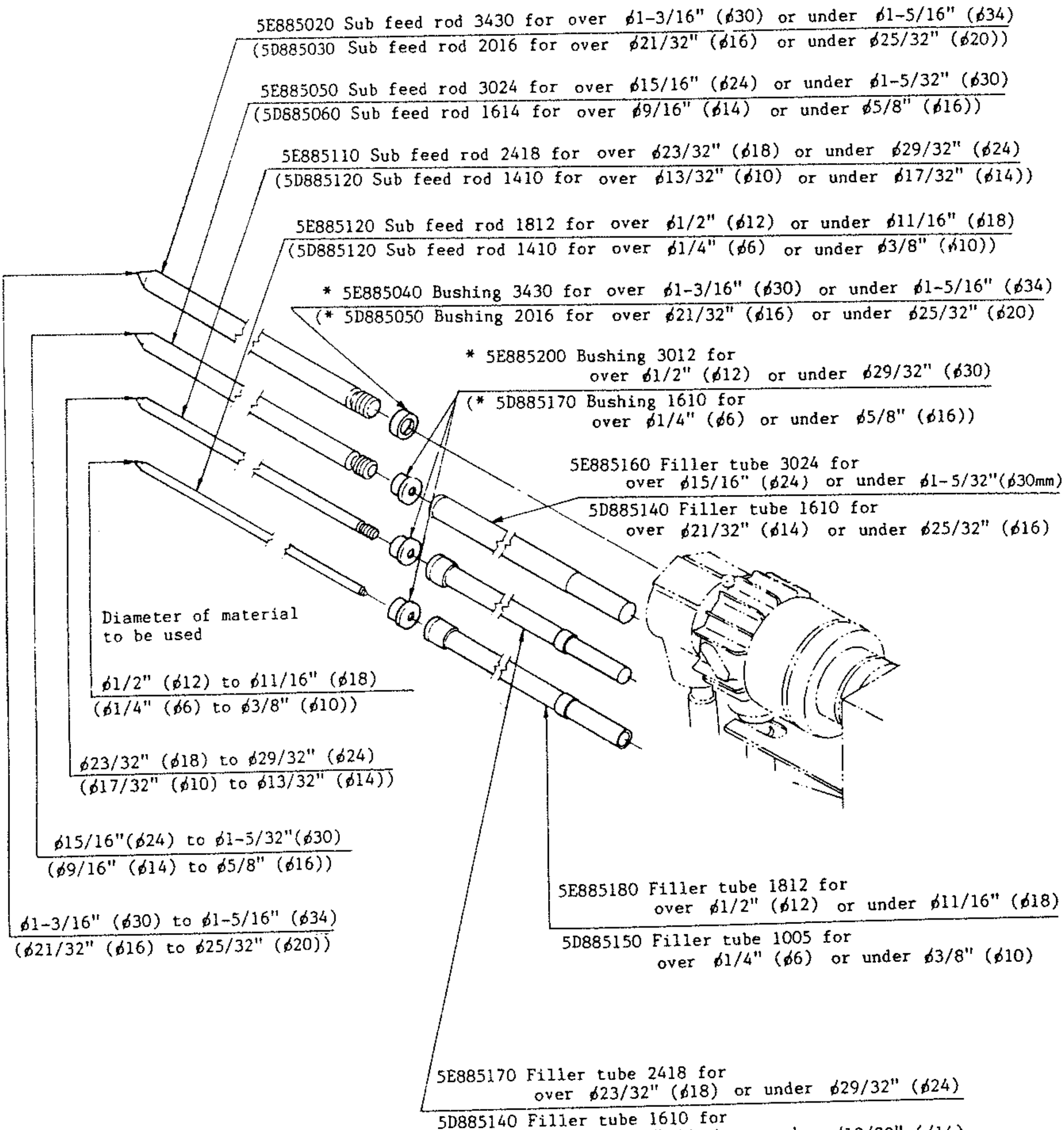
Item		BND-34S	BND-20S
Usable material diameter	maximum	1-5/16" DIA (ϕ34mm)	25/32" DIA (ϕ20mm)
	minimum	1/2" DIA (ϕ12mm)	1/4" DIA (ϕ6mm)
Maximum material length	12tt(4m)	12 ft (4m) Max MS tube number : 5E888500	12 ft (4m) Max MS tube number : 5D888500
Maximum rotation speed (Maximum surface speed : 492ft/min (150m/min))		4000RPM/1/2"DIA(ϕ12mm) 1400RPM/ in case of 1-5/16"DIA (ϕ34mm)	*5000RPM/1/4"DIA(ϕ6mm) 2400RPM/ in case of 25/32"DIA (ϕ20mm)
Feed speed (Standard value)		470 IPM(200mm/sec)	400 IPM(200mm/sec)
Return speed (Standard value)		700 IPM(300mm/sec)	700 IPM(300mm/sec)

2)Replacement part to be changed by material size

When MS tube bar feeder is equipped, it is required to replace a part depending on diameter of material to be used.

<u>Replacement part</u>

The part in the parenthesisss indicates for 20S
* mark must be changed for each material size.



List of replacement part.

Model	Diameter of material to be used		Part number of replacement part		
	inch	mm	Sub feed rod	Bushing	Filler tube
BND-34S	over ø1-3/16" or under ø1-5/16"	over (ø30mm) or under (ø34mm)	5E885020	5E885040	---------
	over ø15/16" or under ø1-5/32"	over (ø24mm) or under (ø30mm)	5E885050	5E885200	5E885160
	over ø23/32" or under ø29/32"	over (ø18mm) or under (ø24mm)	5E885110	Ditto	5E885170
	over ø1/2" or under ø11/16"	over (ø12mm) or under (ø18mm)	5E885120	Ditto	5E885180
BND-20S	over ø21/32" or under ø25/32"	over (ø16mm) or under (ø20mm)	5D885030	5D885050	--------
	over ø9/16" or under ø5/8"	over (ø14mm) or under (ø16mm)	5D885060	5D885170	5D885140
	over ø17/32" or under ø13/32"	over (ø10mm) or under (ø14mm)	5D885120	Ditto	Ditto
	over ø1/4" or under ø3/8"	over (ø 6mm) or under (ø10mm)	Ditto	Ditto	5D885150

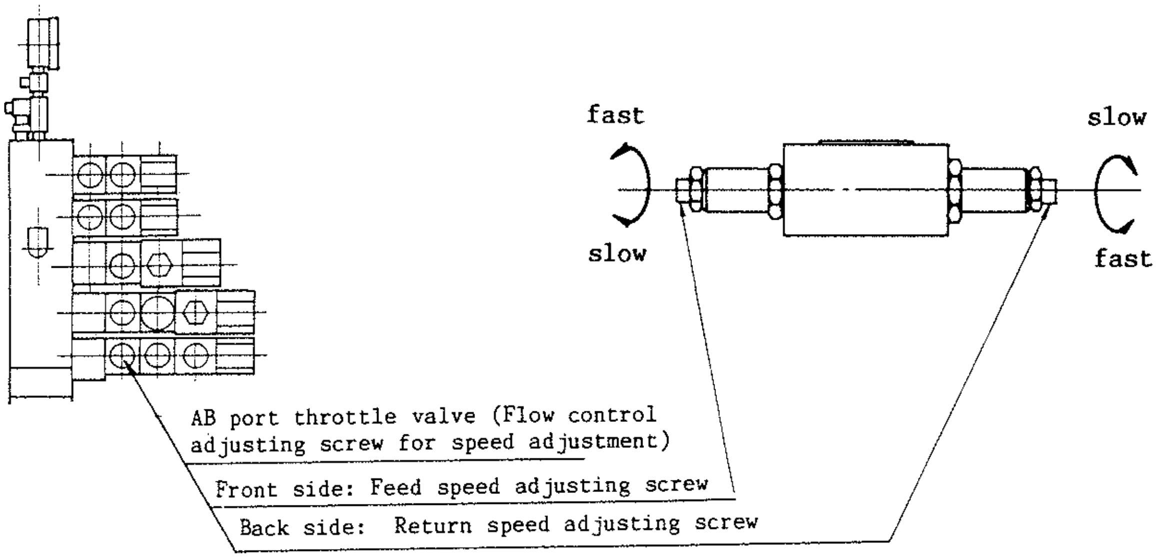
The bushing must be changed for each material size. It must be bigger than the material diameter .04"(1 mm).

3) Adjustment of MS tube bar feeder

After MS tube bar feeder installation has been completed, it requires following adjustment and inspection before actual operation.

(1) Adjustment of material feed and return speed

Adjustment of material feed and return speed is performed by hydraulic operating flow control valve (AB port throttle valve).



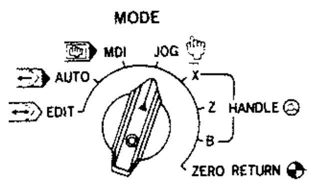
BND-S OTC OPE 90/7

Connect power supply. (Refer to the pragraph of connection of power supply of the operation section.)

Perform manual zero return and manual turret index. (Refer to the paragraph of manual zero return of the operating section).

Follwing operation is required for adjustment of feed speed.

A) Select JOG mode on the operation panel.

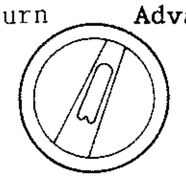


B) Select the selection switch for bar feed on the operation panel to advance side

A feed stud shaft of MS tube bar feed moves to advance side.
(At this time, if the shaft moves to return side, connection of hydraulic hose is reverse.)

Bar feed

Return Advance



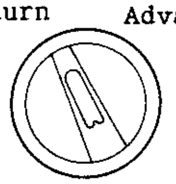
When the selector switch of bar feed is released, it returns to the neutral position and feed stud shaft stops immediately.

C)Select the selector switch on the operation panel to return side.

A feed stud shaft moves to return side.

Bar feed

Return Advance



While performing above-mentioned advance and return operation, measure the time required for whole stroke and adjust the adjusting screw of AB port throttle valve as shown in the following table.

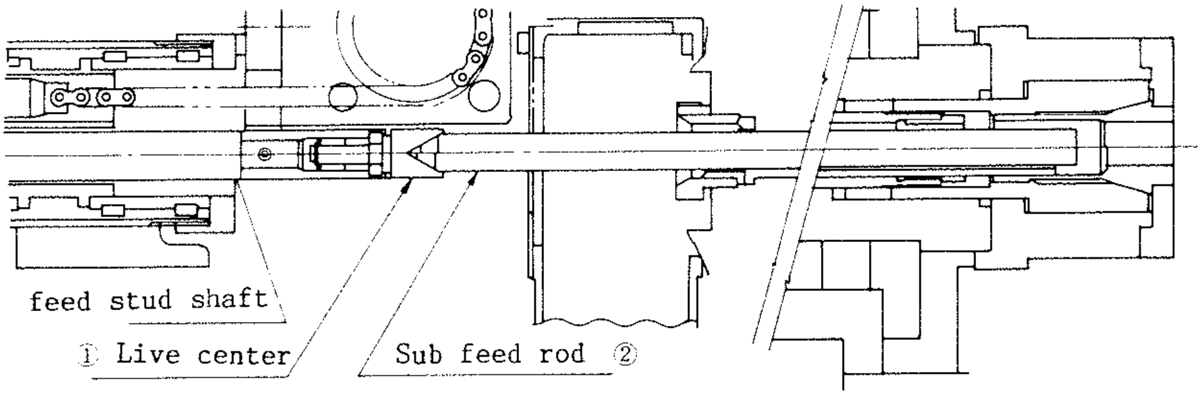
Item \ Model		Time required for whole stroke
		BND-20S, BND-34S
(3m) Specification	Feeding side Return side	15 second 10 second
12ft(4m) Specification	Feeding side Return side	20 second 13 second

(2) Alignment of MS tube bar feeder

To align the feed stud shaft and center of main spindle, perform following operation. Since vibration and noise occur if this alignment is not correct.

A) Insert a sub feed rod into main spindle.

B) Select "Advance" on the selector switch of bar feed on the operation panel and advance the feed stud shaft to the most advanced position as shown in the figure below.



C) Align the center of live center ① and sub feed rod ②. Adjustment of vertical direction, loosen the set screw ④ of support bracket ③ shown on P135 and vertical position is adjusted by a set screw⑤. For adjustment of horizontal direction, loosen the clamp handle ⑥ and lock nut ⑦ and adjust by the set screw ⑧ . After alignment has been completed, lock it by the set screw ④ and lock nut ⑦ tightly so that it may not be shifted.

D) Adjust the position of stand base ⑨ visually with the center of main spindle. Alignment of stand base is performed visually.

E) To adjust leveling bolts ⑩ of stand base, loosen the lock nut ⑪ and roughly align visually in vertical direction. (Accurate alignment in vertical direction is performed by using a level as shown below.) After rough alignment has been completed, lock it by the lock nut ⑪ so that it may not be loosened.

F) In order to set a leveling gauge on the MS tube, remove the chain cover ⑫. Since the chain cover is fixed by a bolt ⑭ of chain band ⑬ , remove the bolt ⑭ and pull up the chain band ⑬ . Place the leveling gauge at the center of the support bar F ⑮ and supprt bar R ⑯. (The leveling gauge is one used for leveling of machine.)

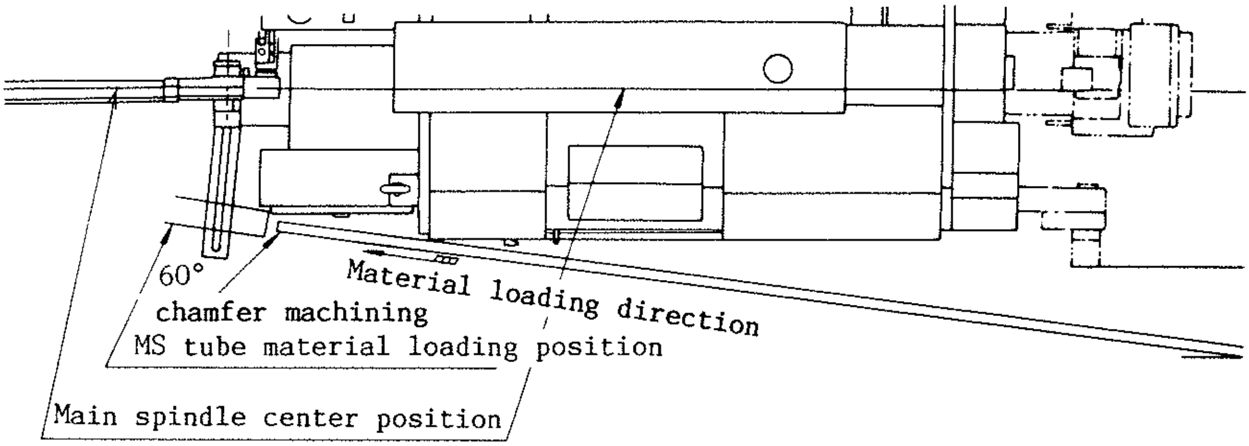
G) Loosen the set screw (17) and align MS tube vertically by the set screw (18).

H) After alignment has been completed, lock it by the square head bult (17) so that it may not be loosened. Install the chain cover (12) by chain cover band (13) .

I) Insert a bar meterial and make sure it the bar material moves smoothly. If the alignment is not correct, vibration may occur.

4) Loading of material

A) (Refer to the drawing on P135.) Loosen the clamp handle (6) and pull MS tube, to the material loading position as shown in the figure below. At this time, move the feed stud shaft to the rear end position.



B) Insert the meterial into MS tube from material loading direction as shown in the figure above.

C) Move MS tube to the center of main spindle. (MS tube is moved up to the position where the square head bolt (8) hits.)

D) Tighten the clamp handle (6) .

5) Bending allowance of bar material.

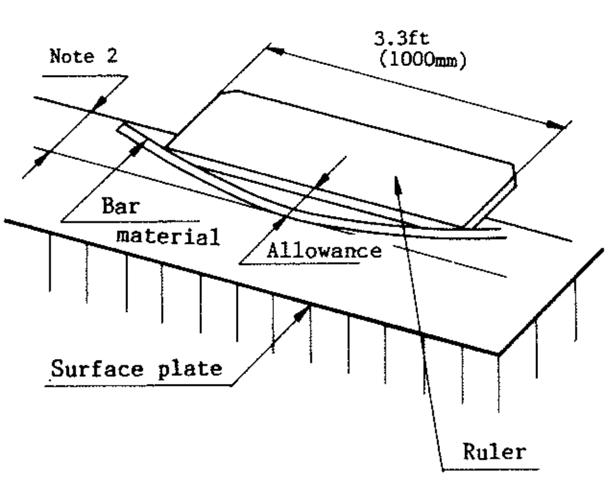
a) Bending of bar material should be less than 1/64" (0.5mm) per 3.3ft (1m).

Bending allowance of bar material

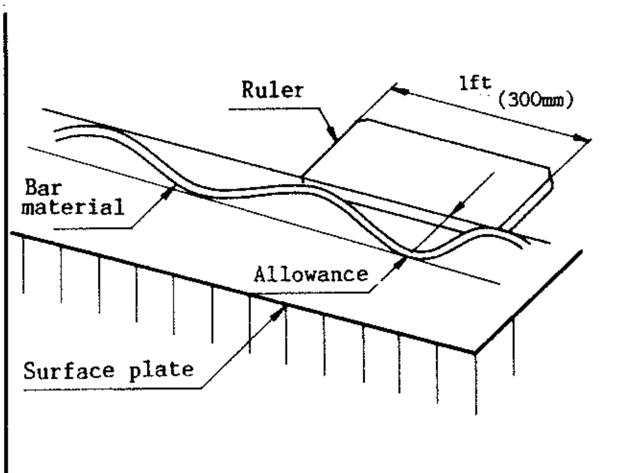
Diameter or lateral distance	Allowance					
	Major bending			Minor bending		
	Class A	Class B	Class C	Class a	Class b	Class c
Under 1/8"(3mm)				.001" (0.025mm) per 1ft (300mm)	.002" (0.05mm) per 1ft (300mm)	
Over 1/8"(3mm) under 3/8"(10mm)	.01" (0.25mm) per 3.3ft (1000mm)	1/64" (0.5mm) per 3.3ft (1000mm)	1/32" (1.0mm) per 3.3ft (1000mm)			0.1 per 1ft(300mm)
Over 3/8"(10mm) under 1-31/32"(50mm)				.0012" (0.03mm) per 1ft (300mm)	.0023" (0.06mm) per 1ft (300mm)	.005" (0.12mm) per 1ft (300mm)
Over 1-31/32"(50mm) under 4-23/32"(120mm)						

Note 1) There are two kinds of bending allowance.
Measurement of bending is performed by the following method.

Note 2) Major bending amount in full length of bar material must not exceed full length/3.3ft(1000) x allowance.



Measuring method of major bending

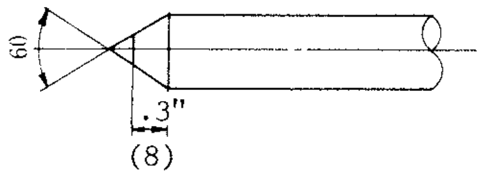


Measuring method of minor bending

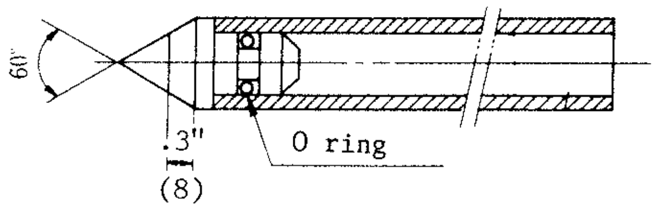
(from standard No. MAS 404-1973)

(b) Rear end chamfer of bar material

a) Bar material



b) Pipe material

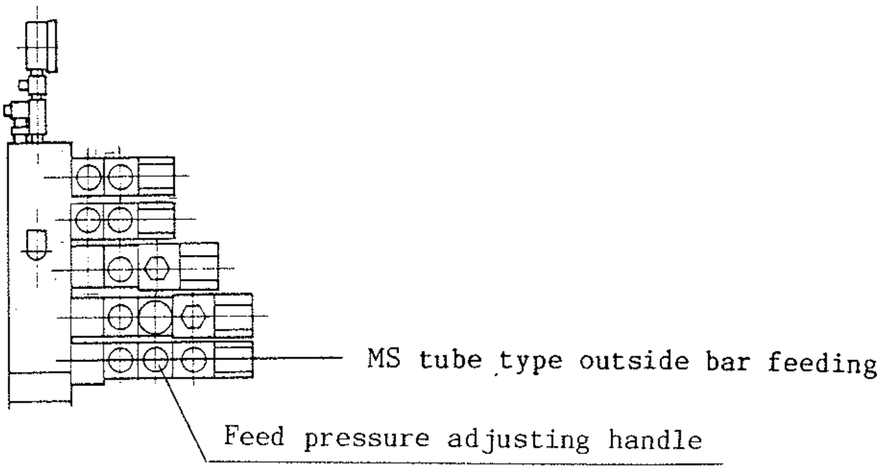


(c) Wipe off the bar material before loading.

6) Adjustment of material feed pressure

Required bar feed pressure is depending on the material size. It can be adjusted in the following procedure.

a) Material feed pressure is adjusted by hydraulic pressure adjusting handle (figure below). Although setting pressure differs depending on material weight, etc, normally, following setting value is used.



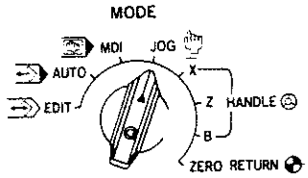
. Standard setting of material feed pressure

material diameter	Setting pressure
under 19/32" (ϕ15mm)	280PSI (20kgf/cm²)
over 5/8" (ϕ15mm) and under 31/32" (ϕ25mm)	350PSI (25kgf/cm²)
over 1" (ϕ25mm)	over 350PSI or under 500 PSI (25∿35kgf/cm²)

7) Manual feed operation for material loading

After material has been supplied to MS tube, material is feed to the position of chuck and preparation work for automatic operation is required. Material is feed to the position of chuck in the following procedure. After bar material is loaded in to MS tube, it is necessary to feed the material into the collet chuck.

a) Select JOG mode on the opration panel.



b)Open the main chuck. (Refer to the praragraph of manual operating switch of the operation section.) Confirm that the indicator lamp of chuck close is turned OFF.

c)Set the bar feed selector switch in advance position and feed the bar to the position of the collet chuck front end.

d)Close the main chuck.

8) Use of sub feed rod

After automatic cycle is started, and when the material becomes short, the machine stops by material used up signal. At this time, about 3.3 ft(1m) of material remains in the spindle.
This material can be feed by sub feed rod.

a) Set the selection switch on the operation panel to JOG.

b) Set the selection switch of bar feed to return and feed back the feed stud shaft about 3.3ft (1m).

c) Loosen the clamp handle and move MS tube to material loading position.

d) Insert a sub feed rod (The size should be matched with the material to be used) into MS tube from material loading direction.

e) Push the reset button on the operation panel and turn off the red indication lamp.

f) Move MS tube to the spindle center position and lock the clamp handle tightly.

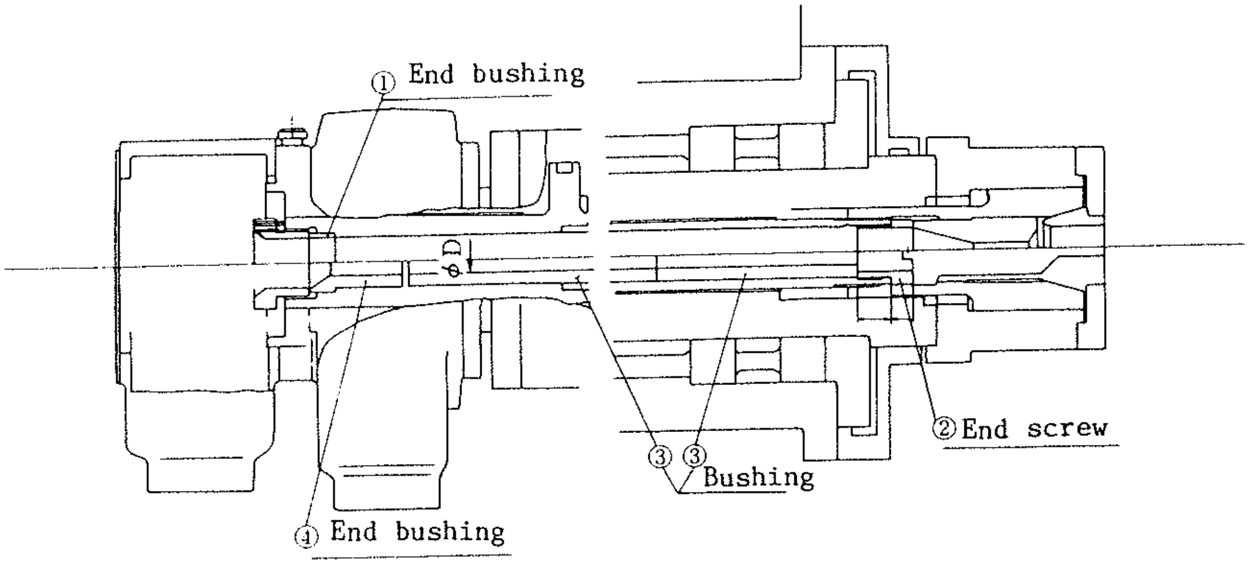
g) Set the selector switch of bar feed in advance position and advance the feed stud shaft until the sub feed rod contacts the material end.

When the machine stops by material used up signal again, some barstock will remain in the collet chuck (about 4"(100mm)). Pull out this barstock and load new barstock.

[3] Replacement part for PF-V2M/PF-V3M

When PF-V2M or PF-V3M is equipped, it is required to replace a part depending on the diameter of material to be used.

1) List of replacement part



List of replacement part for BND-20S

Code	Range of diameter of material used	Part number	Quantity	Remarks
1	21/32" DIA ∿ 25/32" DIA (ø16mm to ø20mm)	5D885050	1	Machined for each bar size
2	17/32" DIA ∿ 5/8" DIA (ø10mm to ø16mm)	5D885070	1	Selected for each bar size
3		5D885090	6	Machined for each bar size
4		5D885110	1	Machined for each bar size
2	1/4" DIA ∿ 3/8" DIA (6mm to ø10mm)	5D885070	1	Selected for each bar size
3		5D885080	6	Machine for each bar size
4		5D885100	1	Machine for each bar size

These bushing should be machined to the size of the material +.04"(+1 mm).

List of replacement part for BND-34S

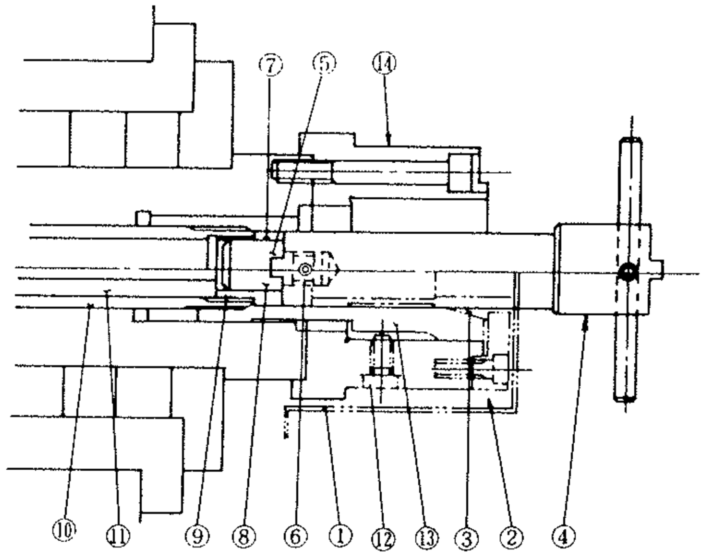
Code	Range of diameter of material used	Part number	Quantity	Remarks
1	1-3/16" DIA ∿ 1-5/16" DIA (ø30mm to ø34mm)	5E885040	1	Machined for each bar size
2	1/2" DIA ∿ 1-5/32" DIA (ø12mm to ø30mm)	5E885060	1	Selected for each bar size
3		5E885080	6	Machined for each bar size
4		5E885100	1	Machined for each bar size

These bushing should be machined to the size of material diameter +.04"(+1mm).

BND-S OTC OPE 90/7

2) Replacement of end screw

When an end screw B (7) is replaced due to change of material diameter, perform the following precedure.



a) Remove the oil guard (1), chuck cap (2), collet (3) pin (12) and chuck sleeve (13).
Insert a guide A (5) into end of a wrench (4) and lock by a set screw (6).

b) Remove an end screw B (7) under this condition.

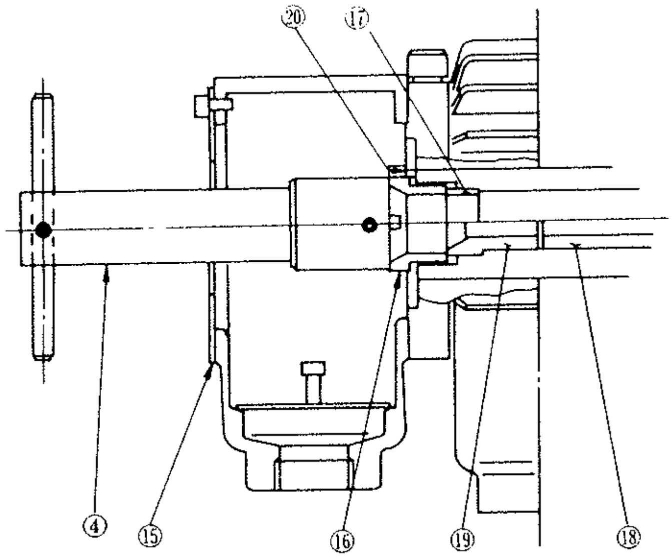
c) Then, replace a guide A (5) at end of wrench (4) with guide B (8).

d) Install an end screw (9) at end of guide B (8) and screw in an end screw (9) into draw pipe (10). At this time, tighten it firmly so that it may not be loosened during use.

e) Subsequently, install a collet, etc. is the same procedure as installation of collet.

3) Installation of bushing

When a bushing is installed or replaced for change of setting, perform the following procedure.



a) Remove the cover 15 .

b) Loosen the set screw 20 . Since set screw is doubled, a front screw is pulled out and a rear set screw is also loosened.

c) Remove the lock screw 16 by a wrench 4 (Guide A and B should be removed.)

d) Insert bushing 17 18 or 19 . These bushing should be machined for each size of barstock.

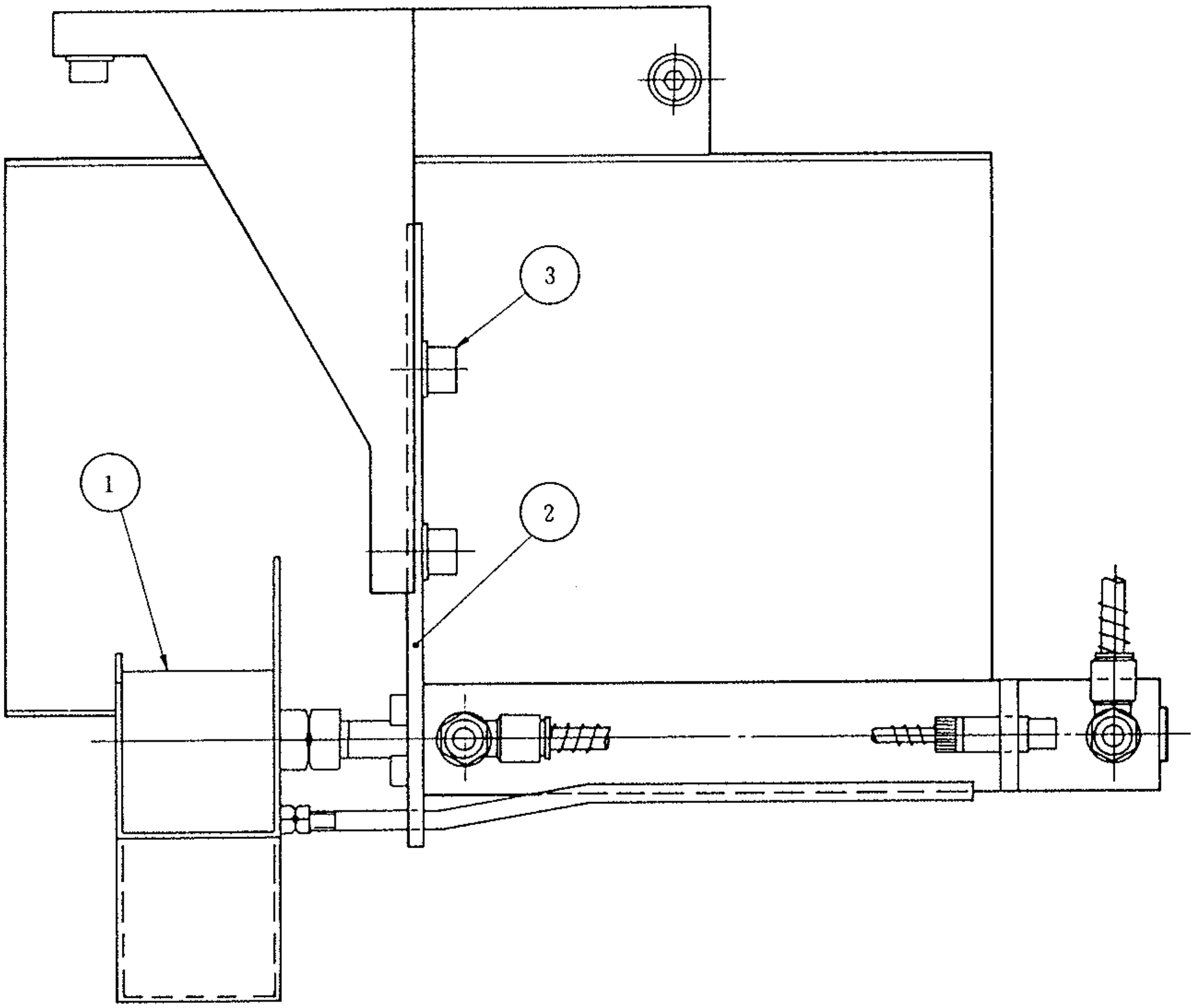
e) Install the lock screw 16 and lock by a set screw 20 so that it may not be loosended during use.

f) Install the cover 15 .

4) Removal of bushing
Removal can be perfromed in the reverse order of installation of bushing.
The bushing 18 is pushed out from collet side.

BND-S OTC OPE 90/7

24. ADJUSTING METHOD OF PART CATCHER



This equipment carries products finished from bar material out of tooling zone.

(1) Operation

Forward movement of products·reception operation is executed by M35 command, and backward movement by M36 command.

(2) Adjustment

1) According to length of products, put the bucket to the proper position by moving the guide plate ② back and forth after loosening the bolt ③.

2) A bucket for main spindle operation completion and one for sub spindle operation completion are provided.
Then, choose suitable one.

(3) Size of products that can be carried out

MAX. ϕ 20 × 100 (For BND-20S)
MAX. ϕ 34 × 125 (For BND-34S)

25. EXPLANATION OF ORIENTATION (Option)

1. Equipment : FRENIC 5000M3

2. M code :

   M13 ...Main spindle normal rotation orientation

   M19 ...Main spindle reverse rotation orientation

3. Rotation of orientation:

   Spindle automatically slows down to the number of revolutions previously set and stops.

4. Method of commanding the angle of rotation

   - When spindle is stopped, a given angle can be commanded.
     Input of M□□ S△△△ allows the spindle to be stopped at a given △△△°.

     (Direct commanding of 0 to 360°)

     (13 or 19)

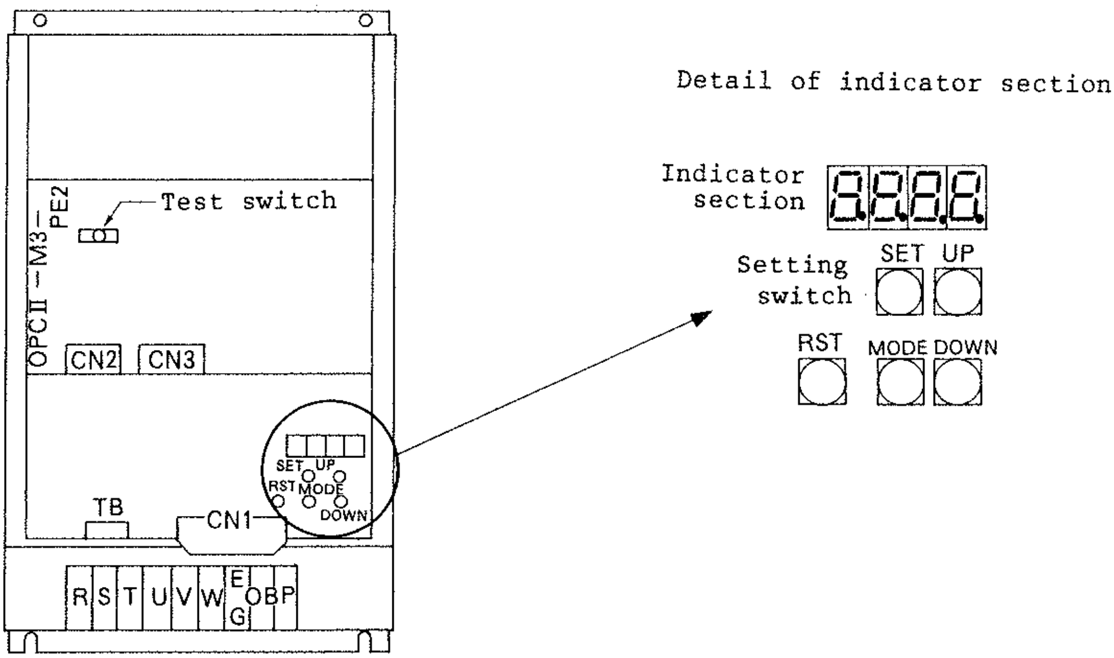
   - When giving an orientation command during spindle running, give the same command.

5. Method of "0° setting" of orientation

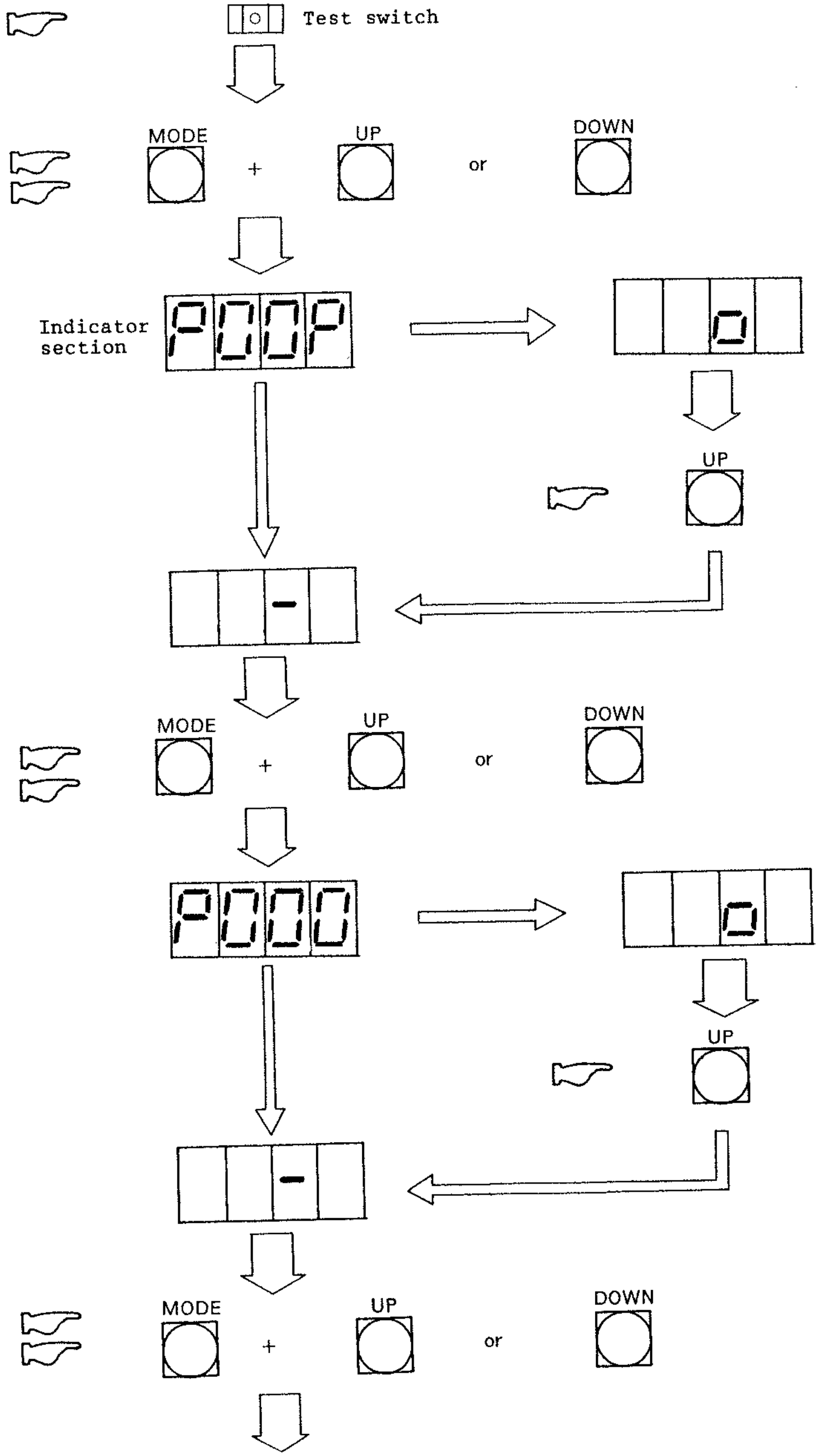
   A. Turn the spindle more than one rotation by "manual" or "auto" at your choice.
      * Be sure to perform it because of checking one rotation signal of position coder.

   B. Stop the spindle, and turn the spindle to "0° setting" manually or by M13.

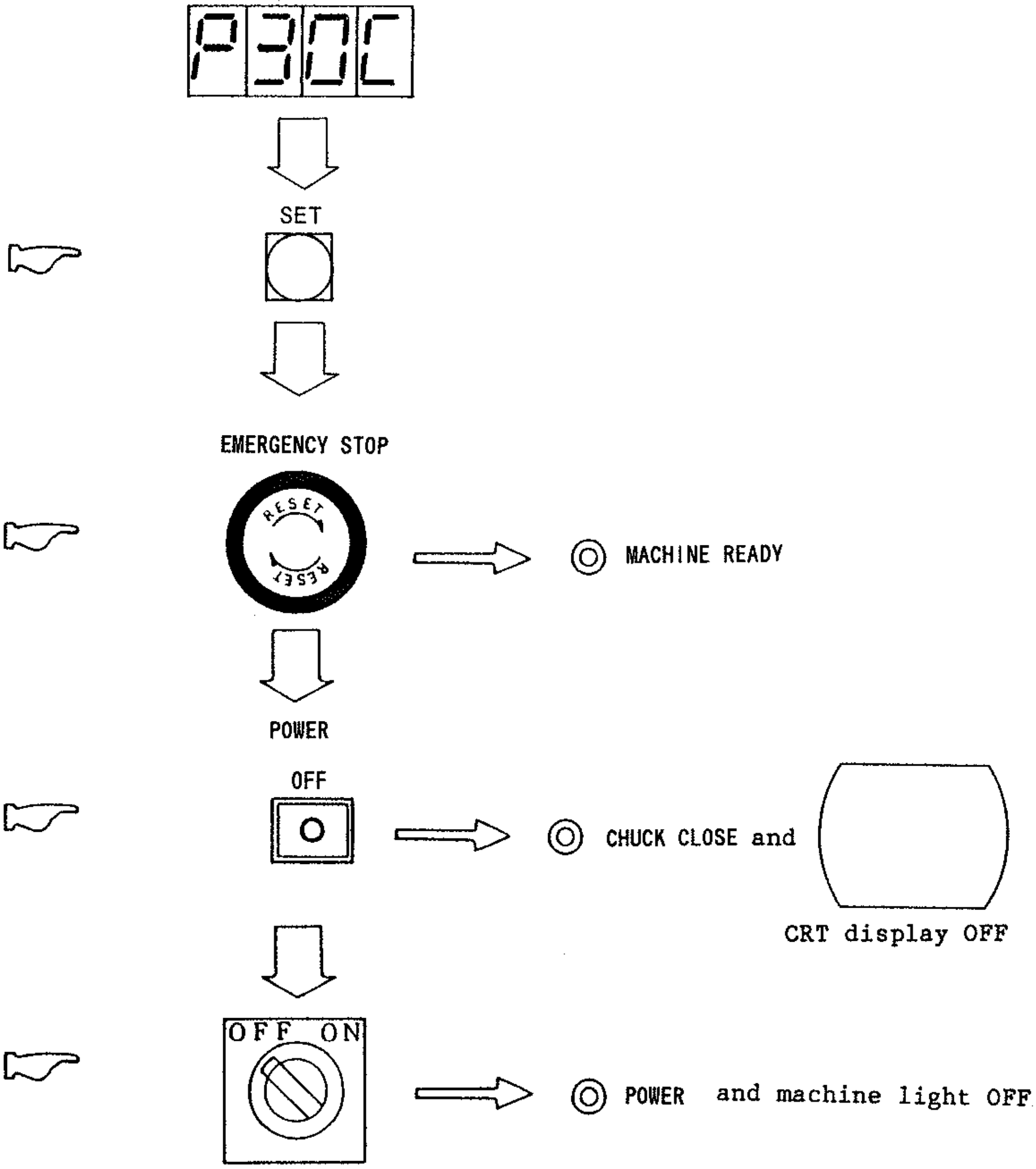
* Switch arrangement in inverter box



C.

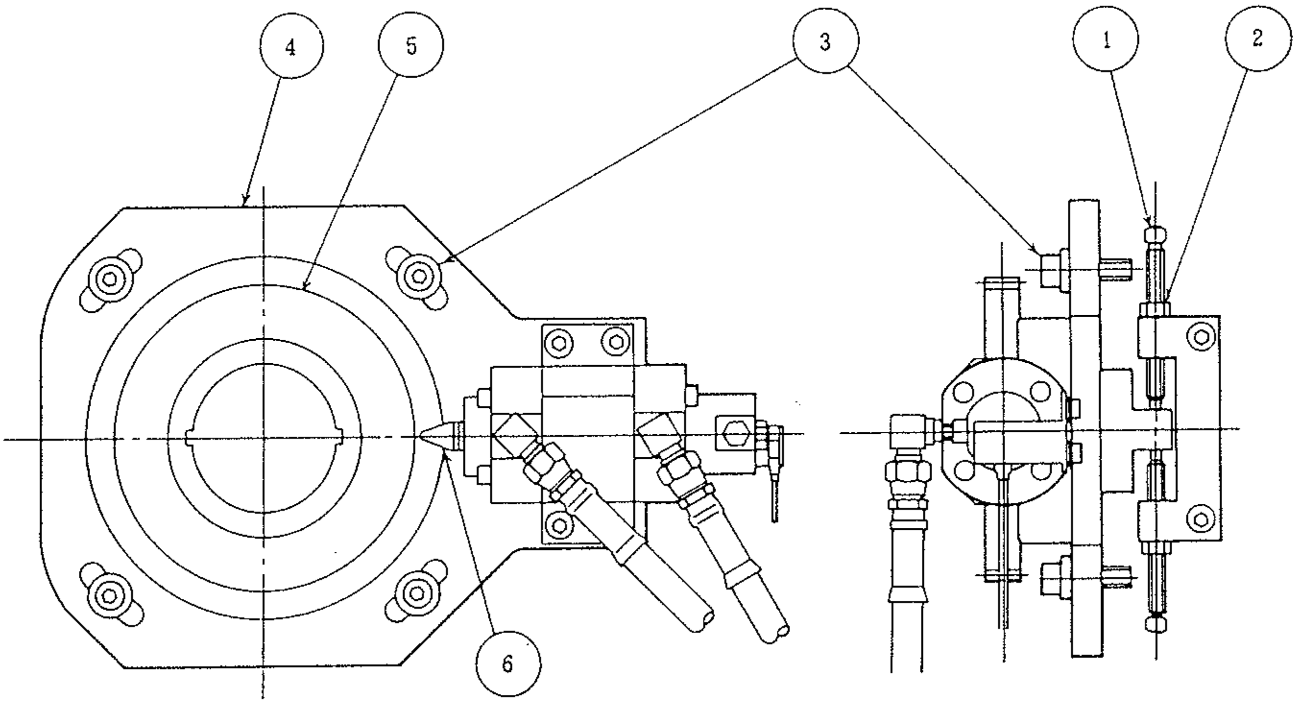


BND-S₂ OPE 0-TC 92/7



d. Turn on power again, and check the location.

26. MAIN SPINDLE POSITIONING



(1) Operation

Spindle orientation command can be executed by angle direct instruction (0 to 360°) of M code and S code.
M code is as follows.

M13 Main spindle normal rotation orientation
M19 Main spindle reverse rotation orientation

When actually instructing, be sure to take the following procedure because accurate orientation is not made without constant surface speed control cancellation.

G97 M□□ S□□□

M□□ → (M13,M19)

S□□□ → (Instruct 0 to 360° directly)

In case with positioning function, however, 1 positioning indexes 7.5°.

Note: When angle of positioning is commanded by S code, a decimal cannot be commanded. When 7.5° multiplied by odd number is commanded, round up or down a decimal.

Example When indexing 7.5°
Index 7.5° by commanding 7 or 8.

[2] Method of adjusting the main spindle positioning

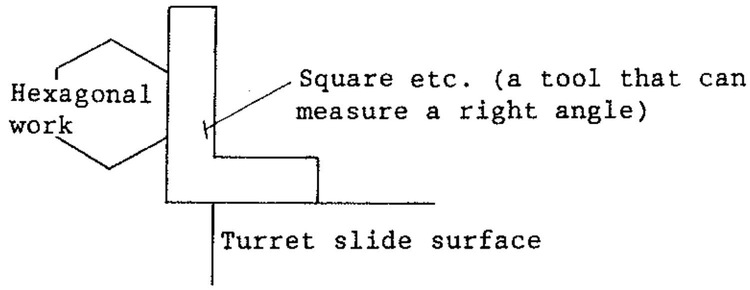
Create a state of mounting the spindle assembly for orientation and spring collet.

A. Turn the spindle more than one rotation manually or MDI operation to check one rotation signal of position coder.

B. Stop spindle to create a state of adding a material (hexagonal material) to collet chuck.

C. Throttle the throttle valve for main spindle positioning on the hydraulic tank to the full extent. (It is only for initial setting. It is unnecessary except at replacement of valve.)

D.



One face of the hexagonal material shall be square with turret slide as shown above.

E. Perform "0° setting" of printed circuit board for main orientation in this state.

* Refer to 25. EXPLANATION OF ORIENTATION (P.151).
  Start from item C of 5. Method of making "0° setting" of orientation.

F. Loosen ② nut and ③ bolt.

G. Adjusting ① bolt, adjust ⑥ lock pin so that it is centered in the bottom of ⑤ index gear.

H. Tighten ③ bolt softly.

M13 S0 [INPUT] → [START] ... After the spindle automatically slows down and stops, a knock pin gets in.

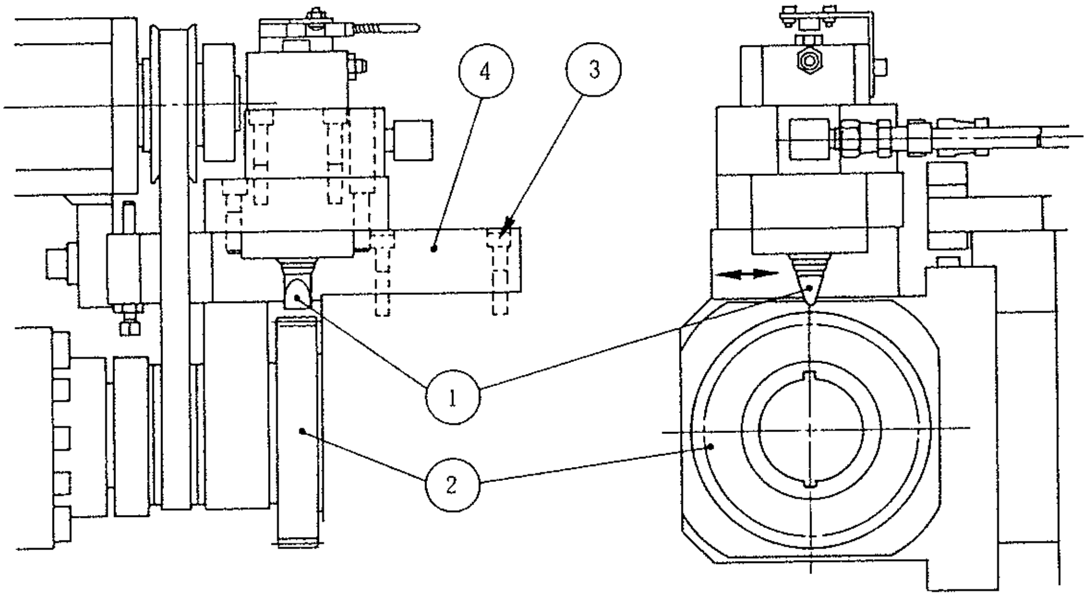
I. Check for right angle with a square or the like in this state. If readjustment is needed, repeat items F to I.

J. Adjustment is completed with tightening ③ bolt and ② nut.

K. Execute M13 S0 [INPUT] → [START] for checking.

Make sure that ⑥ lock pin gets smoothly in ⑤ index gear.
Adjust the throttle valve for positioning so that knock pin may get in slowly and knock pin out quickly.

27. SUB SPINDLE POSITIONING



[1] Operation

Spindle orientation command can be executed by angle direct instruction (0 to 360°) of M code and S code.
M code is as follows.

M113 Sub spindle normal rotation orientation
M119 Sub spindle reverse rotation orientation

Perform this instruction under the condition of constant surface speed control cancellation.

G97 M 1 1 3 S□□□
or
M 1 1 9

(Instruct 0 to 360° directly)

In case with positioning function, however, 1 positioning indexes 7.5°.

Note: When angle of positioning is commanded by S code, a decimal cannot be commanded. When 7.5° multiplied by odd number is commanded, round up or down a decimal.

Example When indexing 7.5°
Index 7.5° by commanding 7 or 8.

[2] Method of adjusting the of sub-spindle positioning

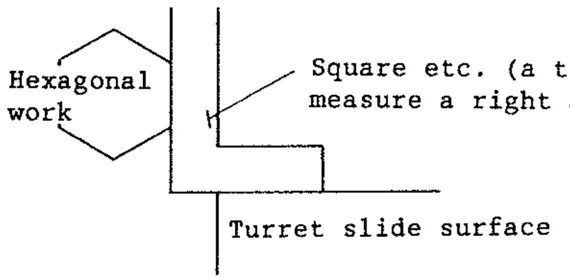
Create a state of mounting the sub spindle assembly for orientation and spring collet.

A. Turn the spindle more than one rotation manually or MDI operation to check one rotation signal of position coder.

B. Stop spindle turn to create a state of adding a material (hexagonal material) to collet chuck.

C. Throttle the throttle valve for sub-spindle positioning on the hydraulic tank to the full extent.
(It is only for initial setting. It is unnecessary except at replacement of valve.)

D.



One face of the hexagonal material shall be square with turret slide as shown above.

E. Perform "0° setting" of printed circuit board for sub-orientation in this state.
* Refer to 25. EXPLANATION OF ORIENTATION (P.151).
Start from item C of 5. Method of making "0° setting" of orientation.

F. Loosen ③ bolt.

G. Moving ④ encoder laterally, adjust ① lock pin so that it is centered in the bottom of ② index gear.

H. Tighten ③ bolt softly.
M113 (M119) S0 [INPUT] → [START] ...After the spindle automatically slows down and stops, a knock pin gets in.

I. Check for right angle with a square or the like in this state. If readjustment is needed, repeat items F to I.

J. Tighten ③ bolt.

K. For confirmation
M113 (M119) S0 [INPUT] → [START] for check.
Make sure that ① lock pin gets smoothly in ② index gear.
Adjust the throttle valve for positioning so that knock pin may get in slowly and knock pin out quickly.

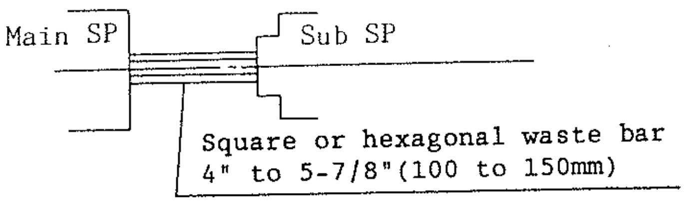
BND-S2 OPE 0-TC 92/7

28. METHOD FOR CORRECTING PHASE BETWEEN THE MAIN SPINDLE AND SUB SPINDLE

In the adjustment methods of the main spindle positioning mentioned in 26 and the sub spindle positioning in 27, it is not necessary to remove the shear of the phases if both spindle positions are accurate.
But, since it is quite possible to have some difference, be sure to confirm the removal in accordance with the following procedures, when a square bar is chucked and operated with W orientation.

A) Adjust or confirm "0° setting" position of the sub spindle by the adjusting method for the sub spindle mentioned in 27.

B) As illustrated below, complete chucking of the main spindle and the sub spindle, using a waste bar.



C) Under the above condition,

M113 S0 $\boxed{\text{INPUT}}$ → $\boxed{\text{START}}$ .... Sub positioning pin enters after slowing down and stopping automatically.
(M119)

D) Make readjustment according to the procedures in items E) to I) of [2] Method of adjusting the positioning of main spindle on page 156.

E) For confirmation

M13 (M113) S0 $\boxed{\text{INPUT}}$ → $\boxed{\text{START}}$ to confirm that main spindle is correctly positioned.

BND-S2 OPE 0-TC 92/7

29. METHOD OF MAKING "0° SETTING" OF SYNCHRONIZED OPERATION (Option)

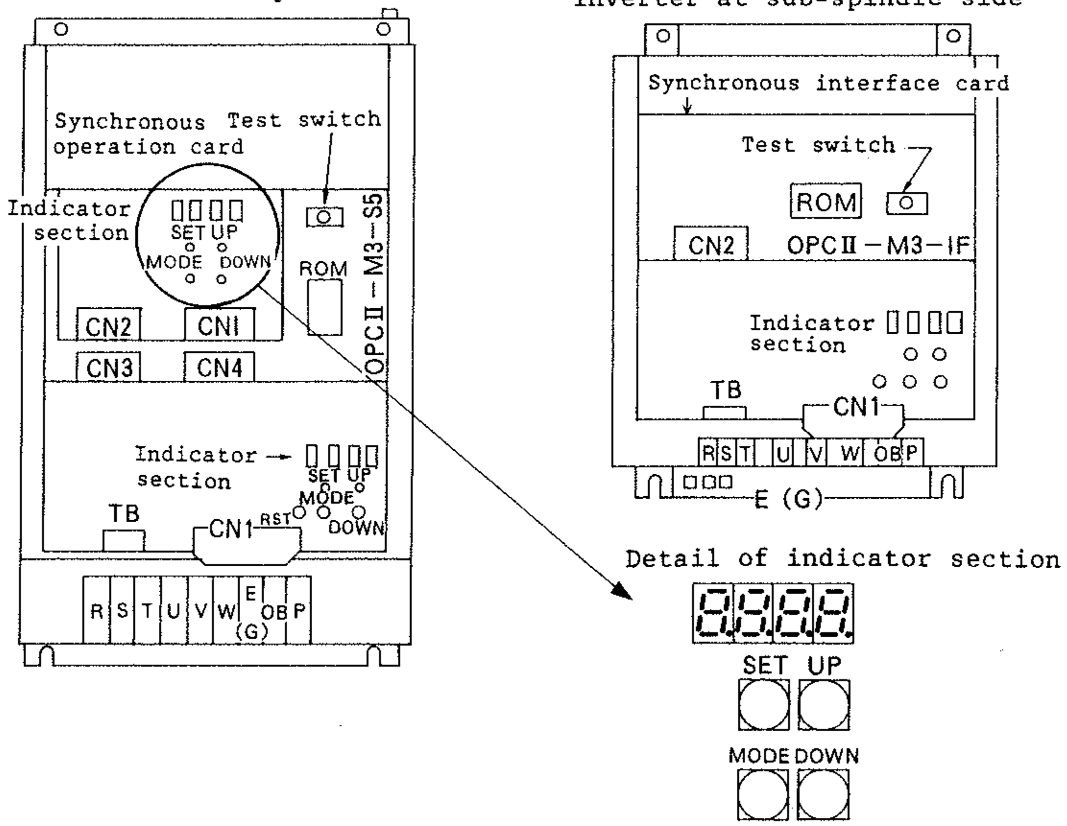
A. Turn both main spindle and sub-spindle more than one rotation by "manual" or "auto" at your choice.
   * Be sure to perform it because of checking one rotation signal of position coder.

B. Stop spindle (for both main spindle and sub-spindle), and turn the spindle to "0° setting" manually or M13, M113, etc.

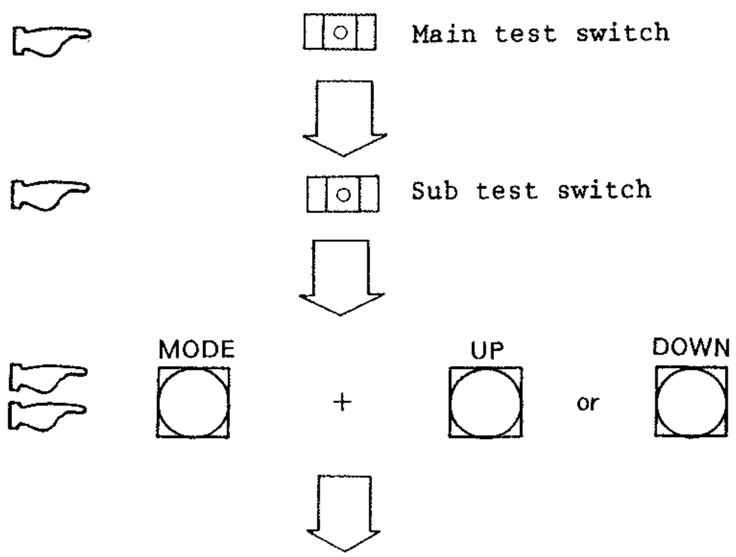
   * Synchronous card arrangement in inverter box

Inverter at main spindle side

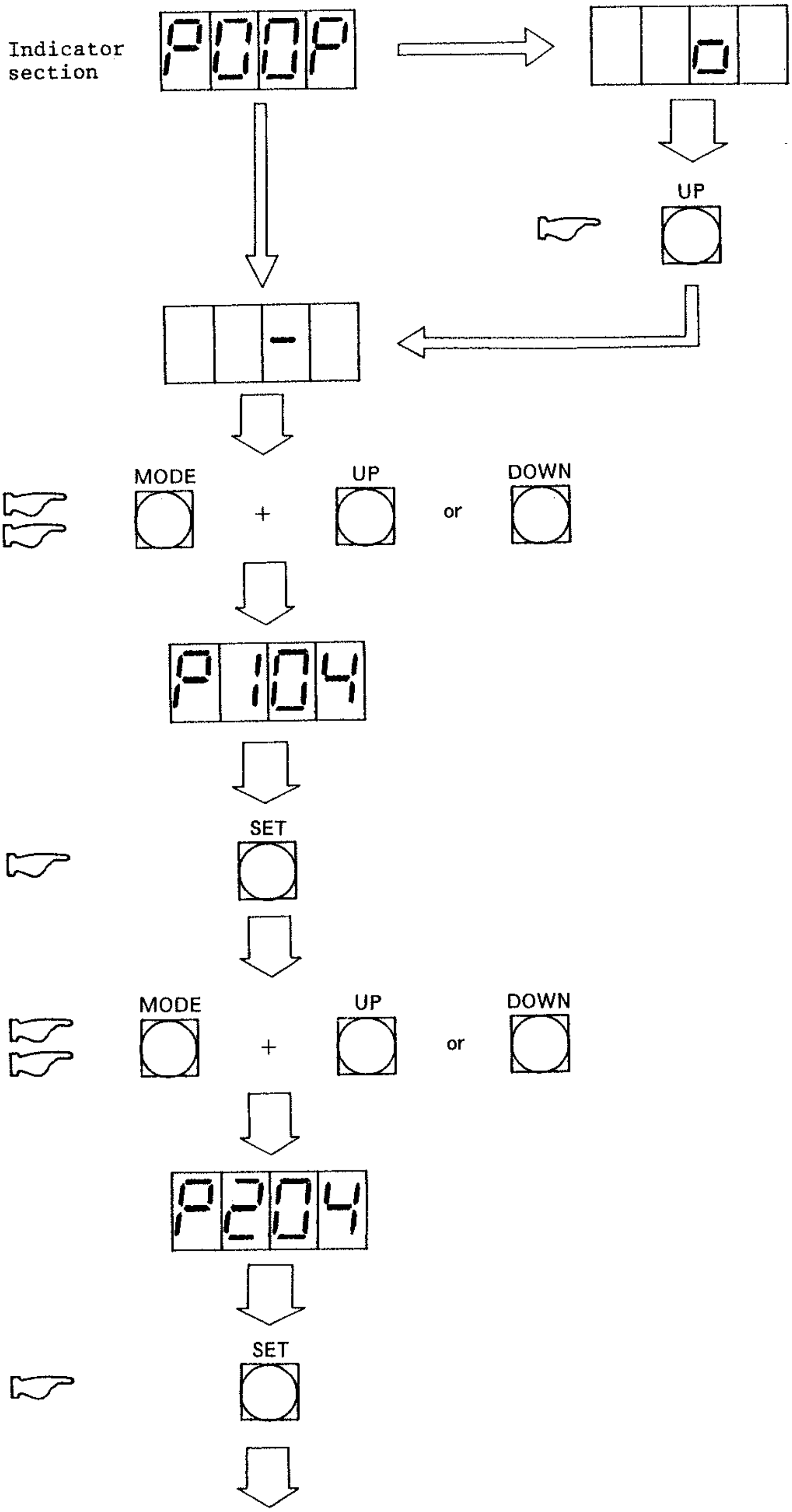
Inverter at sub-spindle side

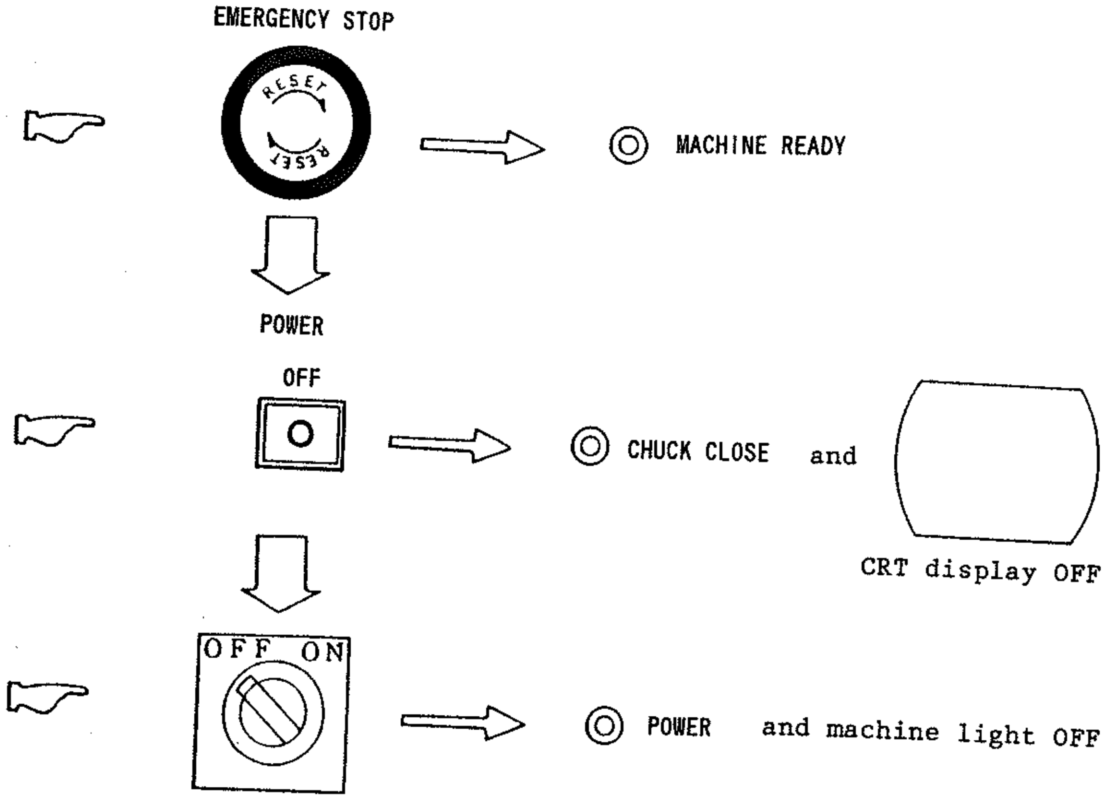


C)



BND-S2 OPE 0-TC 92/7


Indicator section
P00P
UP
MODE
+
UP
or
DOWN
P104
SET
MODE
+
UP
or
DOWN
P204
SET



d. Turn on power again, and check the location.