

# CNC LATHE INSTRUCTION MANUAL PROGRAMMING

SEIKI - SEICOS  $\Sigma$ 10L/21L  
45 Edition 1.01 11-2000





## Introduction

Thank you for your having purchased the machine, favoring our product lines for your use.

This manual contains fundamental information on the programming. Please read and fully understand the contents for your safe machine operation.

In particular, the contents of the items concerning safety in this manual and the descriptions on the “caution plates” attached to the machine are important. Please follow the instructions contained and keep them always in mind to ensure safe operation.

The reference record papers on adjusting setting values such as a parameter list are attached to the machine unit and enclosed in the packing. These are necessary for maintenance and adjustment of the machine later on. Please keep them safely not to be mislaid.

The design and specifications of this machine may be changed to meet any future improvement. As the result, there may arise some cases where explanations in this manual could become partly inconsistent with the actual machine. Please note this point in advance.

In this manual, items on the standard and optional specifications are handled indiscriminately. Please refer to the “delivery note” for the detailed specification of your machine confirmation.



# CONTENTS

|  |              |
|--|--------------|
| <b>1. PREPARATION FOR TOOL LAYOUT .....</b>                                  | <b>1 - 1</b> |
| 1-1 Tool Set .....   | 1 - 2        |
| 1-2 Tool Layout .....  | 1 - 6        |
| 1-3 NC Address and Range of Command Value .....                              | 1 - 7        |
| <b>2. PROGRAMMING .....</b>  | <b>2 - 1</b> |
| 2-1 Basis for Programming .....  | 2 - 1        |
| 2-1-1 Program Reference Point and Coordinate Values .....                    | 2 - 1        |
| 2-1-2 Regarding Machine Zero Point .....                                     | 2 - 2        |
| 2-1-3 Program Example .....  | 2 - 3        |
| 2-2 Details of F, S, T and M Functions .....                                 | 2 - 4        |
| 2-2-1 F Function (Feed Function) .....                                       | 2 - 4        |
| 2-2-2 S Function (Spindle Function) .....                                    | 2 - 5        |
| 2-2-3 T Function (Tool Function) .....                                       | 2 - 7        |
| 2-2-4 M Function (Miscellaneous Function) List (TS15, HT20RIII/23RIII) ..... | 2 - 11       |
| 2-3 Details of G Function .....  | 2 - 33       |
| 2-3-1 List of G Function (SEICOS-S10L/20L) .....                             | 2 - 33       |
| 2-3-2 G50 Maximum Spindle Speed Setting .....                                | 2 - 36       |
| 2-3-3 G00 Positioning .....  | 2 - 36       |
| 2-3-4 G01 Linear Cutting .....   | 2 - 38       |
| 2-3-5 G02, G03 Circular Cutting .....  | 2 - 40       |
| 2-3-6 G04 Dwell .....  | 2 - 44       |
| 2-3-7 G09 Exact Stop .....   | 2 - 44       |
| 2-3-8 G61 Exact Stop .....   | 2 - 45       |
| 2-3-9 G10 Programmable Date Input .....                                      | 2 - 45       |
| 2-3-10 G20, G21 Inch Input/Metric Input .....                                | 2 - 46       |
| 2-3-11 G22, G23 Stored Stroke Limit .....                                    | 2 - 47       |
| 2-3-12 Stroke Limit Check Before Move .....                                  | 2 - 50       |
| 2-3-13 G27 Reference Point Return Check .....                                | 2 - 51       |
| 2-3-14 G28 Automatic Reference Point Return .....                            | 2 - 52       |
| 2-3-15 G30 2nd Reference Point Return .....                                  | 2 - 52       |
| 2-3-16 G31 Skip Function .....   | 2 - 55       |
| 2-3-17 G54 Work Coordinate System Setting (Work Length) .....                | 2 - 56       |
| 2-3-18 Canned Cycle .....  | 2 - 57       |
| 2-3-19 G70, G71, G72, G73, G74, G75 Compound Repetitive Cycle (Option) ..... | 2 - 66       |
| 2-3-20 G32, G92, G76 Thread Cutting .....                                    | 2 - 82       |

|   |              |
|---|--------------|
| 2-3-21 Continuous Thread Cutting .....  | 2 - 106      |
| 2-3-22 G34 Variable Lead Thread Cutting (Option) .....  | 2 - 106      |
| 2-3-23 Multi-thread Cutting (Option) .....  | 2 - 107      |
| 2-3-24 G150, G151, G152 Groove Width Compensation .....   | 2 - 108      |
| <b>3. AUTOMATIC CALCULATING FUNCTION OF TOOL NOSE RADIUS<br/>COMPENSATION .....</b>   | <b>3 - 1</b> |
| 3-1 Outline .....   | 3 - 1        |
| 3-2 Preparation to Execute the Automatic Calculating Function of<br>Tool Nose Radius Compensation .....                         | 3 - 2        |
| 3-3 Three Conditions of Nose Radius Compensation .....  | 3 - 3        |
| 3-3-1 Tool Nose Radius Compensation Block (During Cutting) .....  | 3 - 4        |
| 3-3-2 Start-up Block and Compensation Cancel Block (Approach/Retreat) .....   | 3 - 6        |
| 3-4 Caution Point of Approach to Workpiece .....  | 3 - 10       |
| 3-5 Tool Nose Radius Compensation to Direct Designation G Code (G141, G142) .....   | 3 - 11       |
| <b>4. PROGRAM EXAMPLE (NC PROGRAM) .....</b>  | <b>4 - 1</b> |
| 4-1 Chuck Work .....  | 4 - 1        |
| 4-1-1 Machining Drawing .....   | 4 - 1        |
| 4-1-2 Chuck Work Program .....  | 4 - 2        |
| 4-2 Center Work .....   | 4 - 6        |
| 4-2-1 Machining Drawing .....   | 4 - 6        |
| 4-2-2 Center Work Program .....   | 4 - 7        |
| 4-3 Bar Work .....  | 4 - 9        |
| 4-3-1 Machining Drawing .....   | 4 - 9        |
| 4-3-2 Bar Work Program .....  | 4 - 10       |
| 4-4 Grooving .....  | 4 - 12       |
| 4-4-1 OD Grooving .....   | 4 - 12       |
| 4-4-2 ID Grooving .....   | 4 - 13       |
| 4-4-3 End Face Grooving .....   | 4 - 15       |
| 4-5 1st and 2nd Process Continuous Machining Method .....   | 4 - 16       |
| 4-5-1 Machining Method by Single Program .....  | 4 - 17       |
| 4-5-2 Machining Method by Subprogram Calling .....  | 4 - 18       |
| 4-6 Operation Example of Many Short Length Works .....  | 4 - 19       |
| <b>5. REFERENCE MATERIALS .....</b>   | <b>5 - 1</b> |
| 5-1 How to Calculate the Tool Nose Radius Compensation Amount<br>Without Using the Tool Nose Radius Compensation Function ..... | 5 - 1        |
| 5-2 Calculation Formulas .....  | 5 - 10       |
| 5-2-1 How to Obtain Side and Angle of Right Triangle .....  | 5 - 10       |
| 5-2-2 How to Obtain Side and Angle of Inequilateral Triangle .....  | 5 - 12       |
| 5-2-3 How to Obtain Taper and Intersecting Point of Circular Arc .....  | 5 - 13       |

|  |              |
|--|--------------|
| 5-2-4 Others .....   | 5 - 17       |
| <b>6. SPECIFICATIONS OF C-AXIS CONTROL (SEIKI-SEICOS S21L) .....</b>                             | <b>6 - 1</b> |
| 6-1 Outline .....  | 6 - 1        |
| 6-2 Standard Specifications .....  | 6 - 1        |
| 6-3 Program .....  | 6 - 3        |
| 6-3-1 Coordinate Axis .....  | 6 - 3        |
| 6-3-2 Plane Selection of G17, G18, G19 .....   | 6 - 3        |
| 6-3-3 Miscellaneous Function for Rotating Tool (M Code) .....                                    | 6 - 4        |
| 6-3-4 Fixed Cycle for Hole Making G80~G89, G831, G841, G861 .....                                | 6 - 5        |
| 6-3-5 Program Example .....  | 6 - 16       |
| 6-4 Polar Coordinate Interpolation Function .....  | 6 - 23       |
| 6-4-1 Polar Coordinate Function .....  | 6 - 23       |
| 6-4-2 G Function .....   | 6 - 23       |
| 6-4-3 Program Example (X-axis : Linear axis/C-axis : Rotating axis) .....                        | 6 - 25       |
| 6-5 G40, G41, G42, G140, G143, G145 Tool Radius Compensation Function .....                      | 6 - 26       |
| 6-5-1 Direction of Tool Radius Compensation .....  | 6 - 26       |
| 6-5-2 Movement of Tool Radius Compensation .....   | 6 - 27       |
| 6-6 Program Example (Polar Coordinate Interpolation,<br>Tool Radius Compensation Function) ..... | 6 - 30       |
| 6-7 G824, G843 Direct Tapping .....  | 6 - 32       |
| 6-8 G271 Cylindrical Interpolation .....   | 6 - 35       |
| <b>7. REFERENCE(SPECIFICATIONS OF C-AXIS CONTROL) .....</b>                                      | <b>7 - 1</b> |
| 7-1 How to Calculate C-axis Feed Rate for Long Hole Machining .....                              | 7 - 1        |
| 7-2 How to Calculate the Number of Rotation and Feed Rate of the Rotating Tool .....             | 7 - 3        |



# 1. PREPARATION FOR TOOL LAYOUT

There are limit of range of travel and other limits according to the machine specifications and safety.

Refer to “Specifications Manual” of each machine type for stroke, work operation range, tool interference diagram and Q setter•work interference diagram of the machine, which should be fully understood as they are premises for machine operation, programming and tool layout.

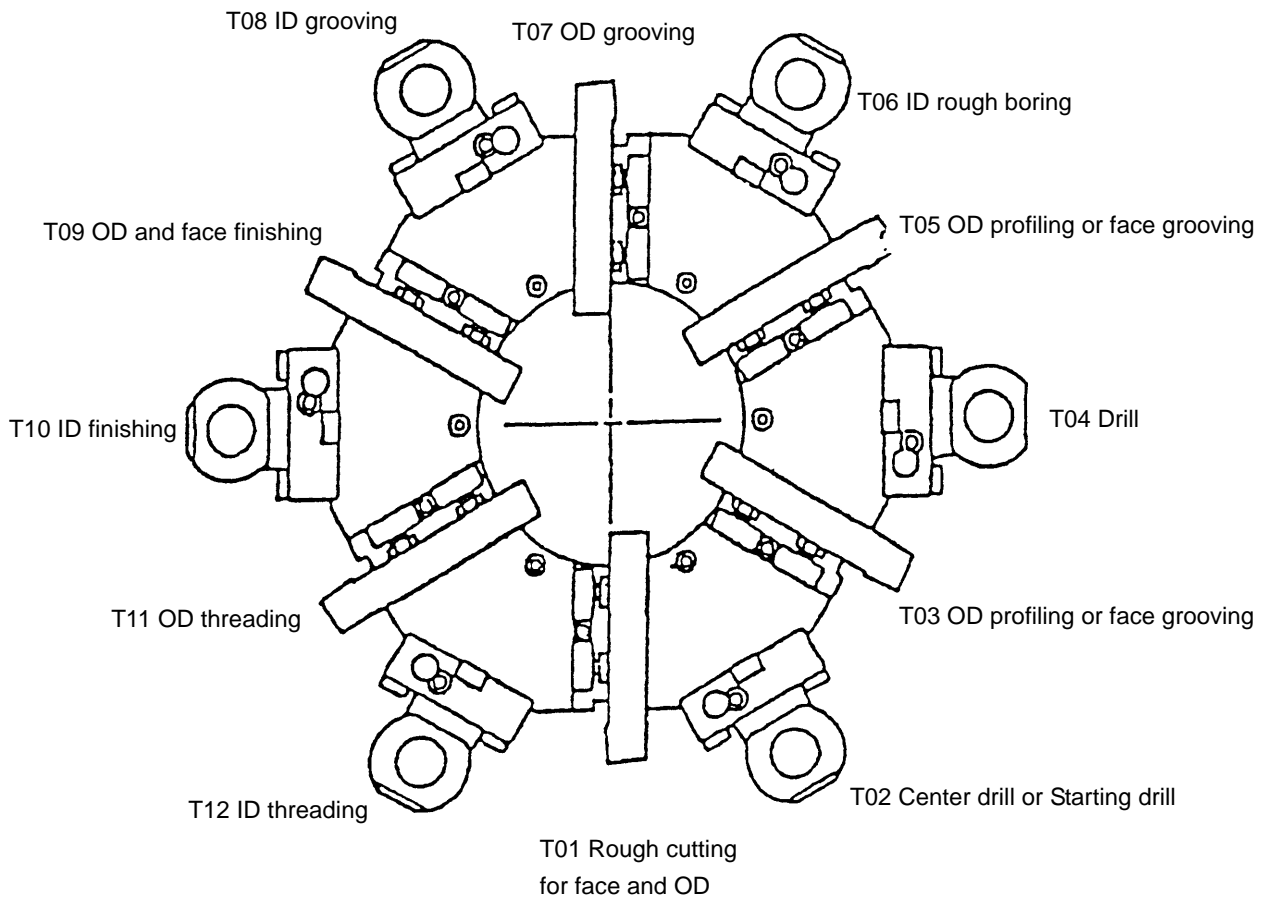
# 1-1 Tool Set

## Standard Tool Set

In order to keep operation procedure of the work and to avoid interference of the tool and the chuck large tools such as the base holder shall be set permanently.

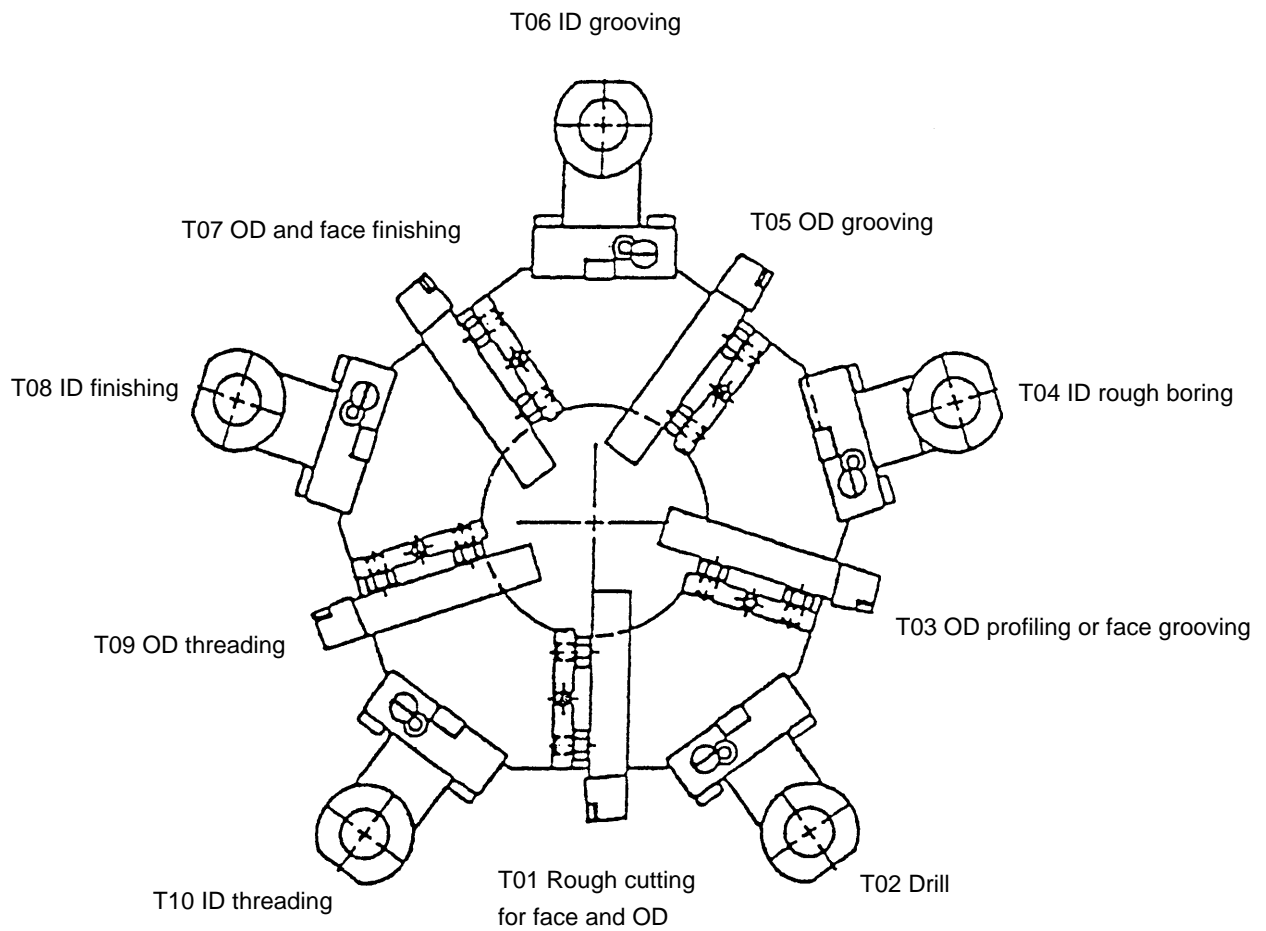
Further, set the tools as you like in order to satisfy the operation accuracy of the small tools such as the boring bar, and also to perform the turret indexing by one rotation.

The standard tool set is shown as below.



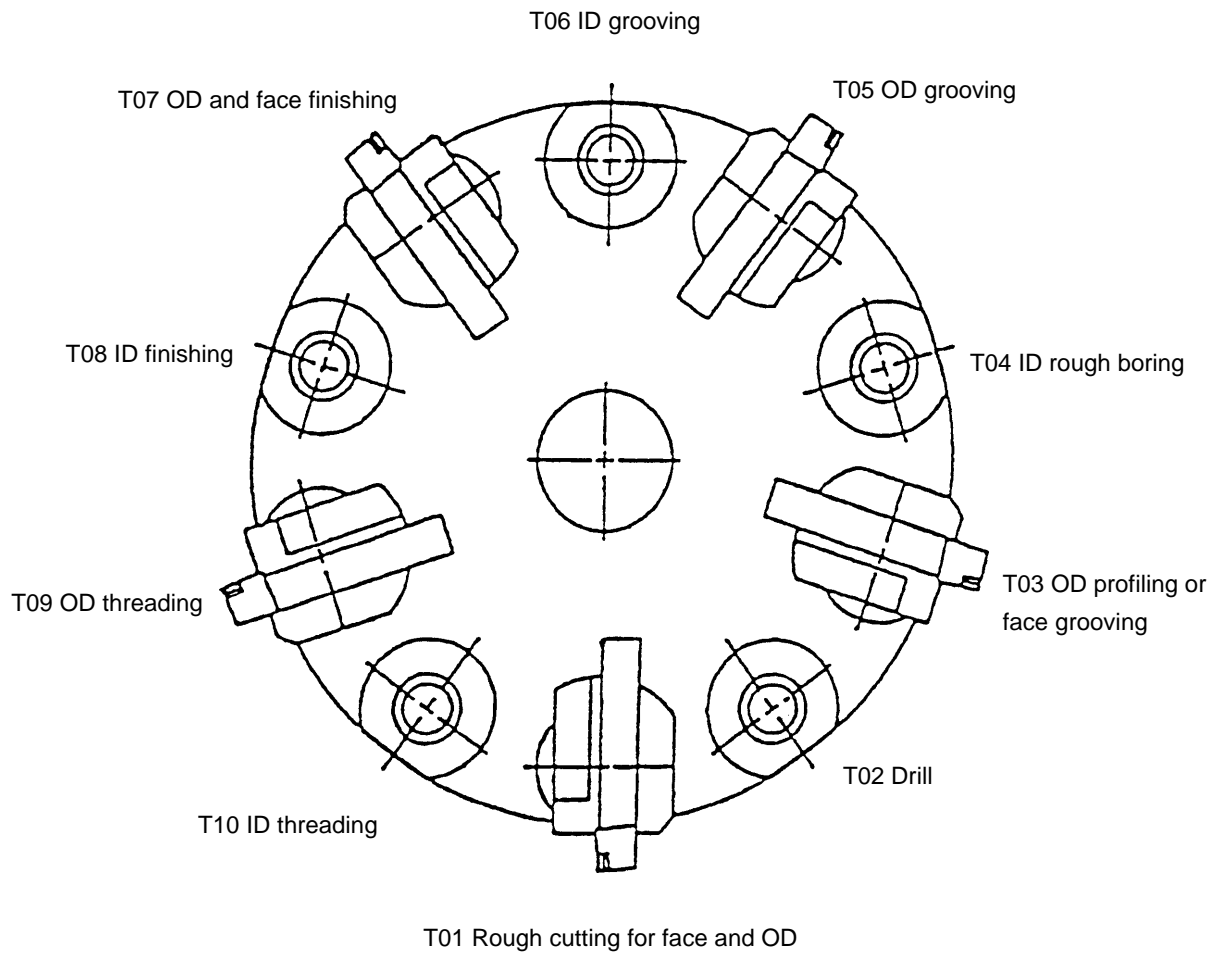
Specifications of 12-station Variable turret

# Standard Tool Set



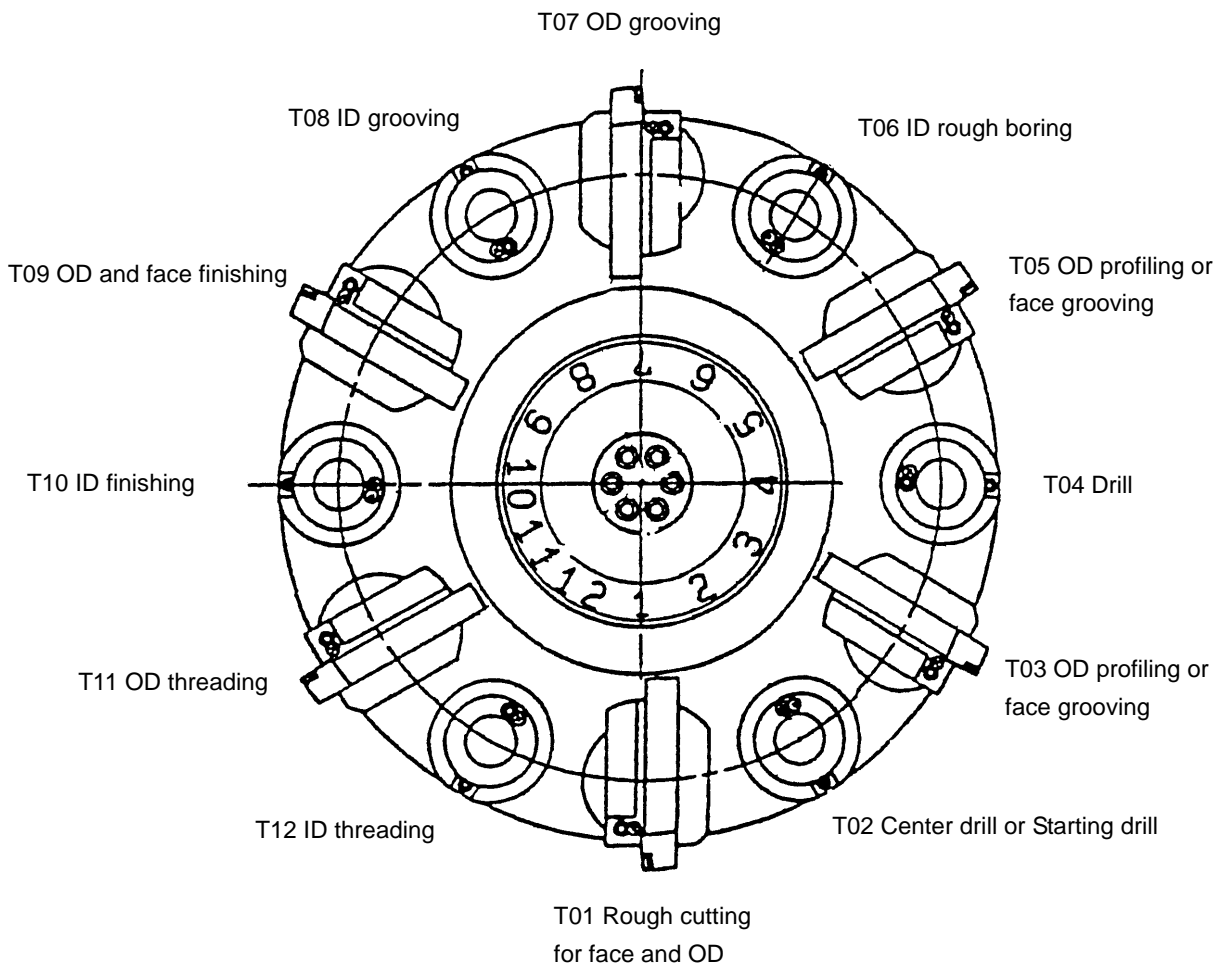
Specifications of 10-station Variable turret

# Standard Tool Set



Specifications of 10-station QCT turret

Standard Tool Set

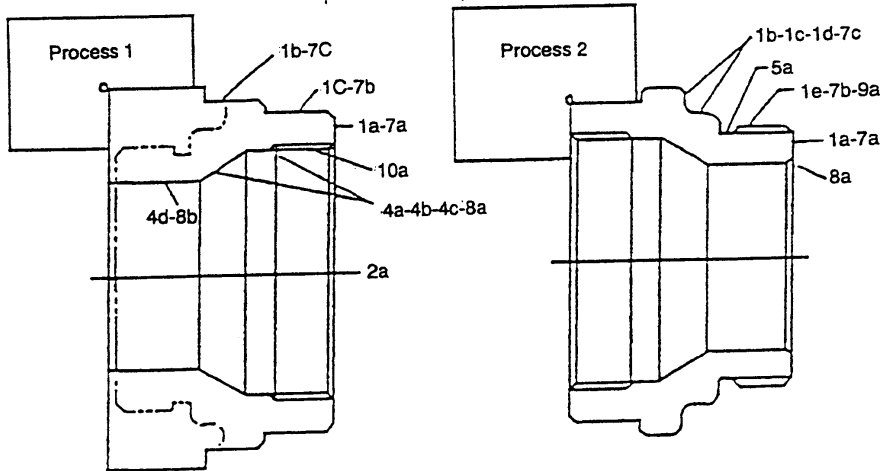
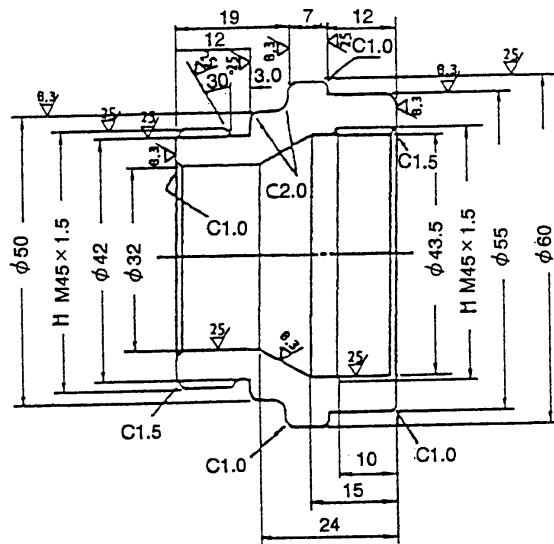


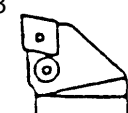
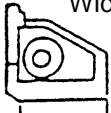
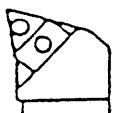
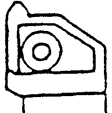
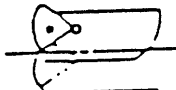



Specifications of 12-station QCT turret

# 1-2 Tool Layout

|                                       |         |  |
|---------------------------------------|---------|--|
| Example of tool layout for chuck work |         | <br>CNC LATHE:<br>TOOL LAYOUT DRAWING |
| Process :<br>Process 1, 2             | NC unit |  |

|                  |  |
|------------------|--|
| Part name SAMPLE |  |
| Material<br>S48C |  |



|  |  |  |  |   |
|--|--|--|--|---|
| T1   | T3   | T5<br>Width 2mm  | T7   | T9  |
| R0.8<br><br>OD roughing |  | <br>OD grooving | R0.8<br><br>OD finishing    | <br>OD threading     |
| T2   | T4   | T6   | T8   | T10   |
| <br>φ30                 | R0.8<br><br>φ20 ID roughing |  | R0.8<br><br>φ20 ID finishing | <br>φ25 ID threading |

### 1-3 NC Address and Range of Command Value

| Function             | Address  | Range of command value                                    |  |
|----------------------|--|---|--|
| Program No.          | O  | 1~99999999  |  |
| Sequence No.         | N  | 1~99999999  |  |
| Preparatory function | G  | 0~999   |  |
| Coordinate value     | X, Y, Z, U, V,<br>W, I, J, K, Q,<br>R, A, B, C | $\pm 99999.999(\text{mm})$<br>$\pm 99999.999(\text{deg})$ | $\pm 9999.999(\text{inch})$<br>$\pm 99999.999(\text{deg})$ |
| Feedrate             | F  | 0.001~999.999(m/rev)                                      | 0.0001~99.9999(inch/rev)                                   |
| Spindle function     | S  | 0~99999999  |  |
| Tool function        | T  | 0~999999  |  |
| Auxiliary function   | M  | 0~99999999  |  |
| Dwell                | P, X, U  | 0~99999.999(sec)  |  |
| Call up program No.  | P  | 1~99999999  |  |
| Number of repetition | L  | 1~99999999  |  |



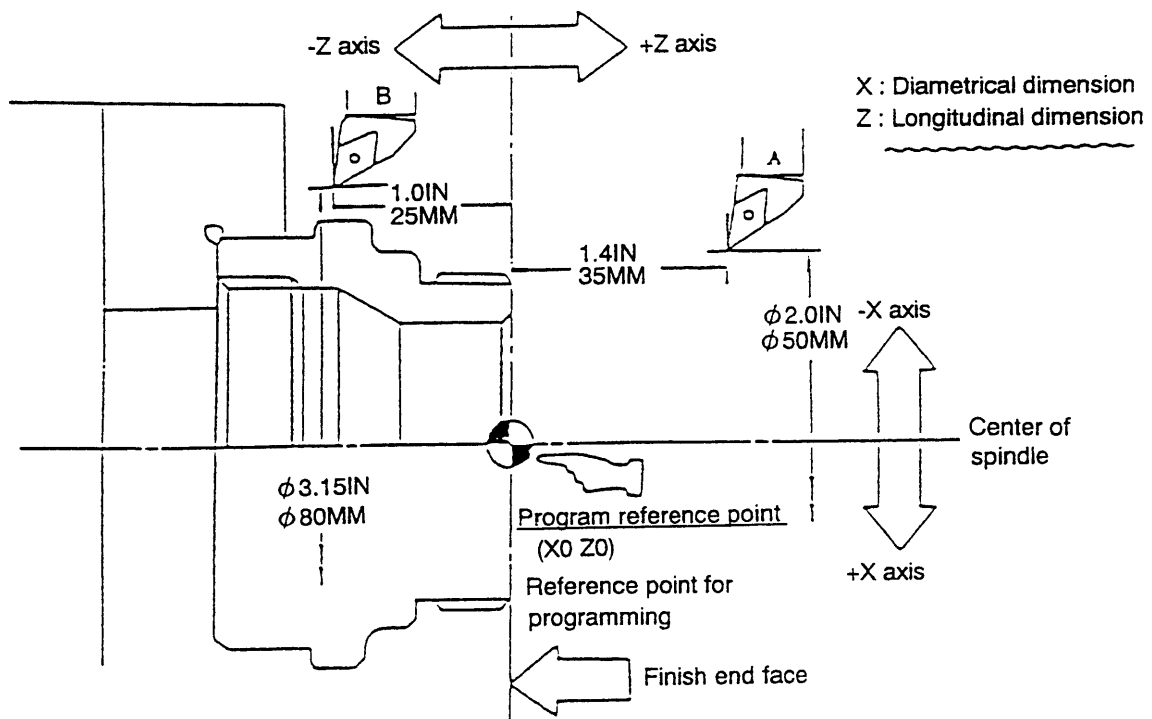
## 2. PROGRAMMING

### 2-1 Basis for Programming

#### 2-1-1 Program Reference Point and Coordinate Values

For a CNC lathe, coordinate axes X and Z are set on the machine and their intersecting point is called a "program reference point". The X axis assumes a spindle center line to be a position of "X0", and the Z axis assumes a workpiece finish end face on the tail stock side to a position of "Z0".

To move a tool, specify its moving position, adding signs "+" and "-" to both X and Z axes, with this program reference point as a datum point.



#### •Position of the tool A ...

Since it is located a plus 50 dia. on the X-axis and plus 35mm (1.4") on the Z-axis,  
X50.0 Z35.0 ..... (Omit the plus sign)

#### •Position of the tool B ...

Since it is located a plus 80 dia. on the X-axis and minus 25mm (1.0") on the Z-axis,  
X80.0 Z-25.0

## **2-1-2 Regarding Machine Zero Point**

Properly speaking, the machine zero point and reference point is a different position, however, as for our NC lathe make the both points the same position.

Therefore, here in after the reference point calls as the machine zero point in this manual.

It is a position which is the machine proper and the machine zero point which is the basis of program set the end of each axis.

This machine zero point utilizes an electrically identical point, a grid point, and stop a servo motor at the certain point.

Turn on the power at the starting time in the morning, it can be entered a program operation by execution of the zero return.

## 2-1-3 Program Example

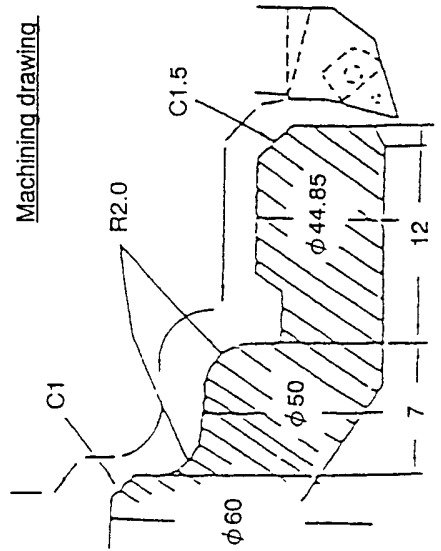
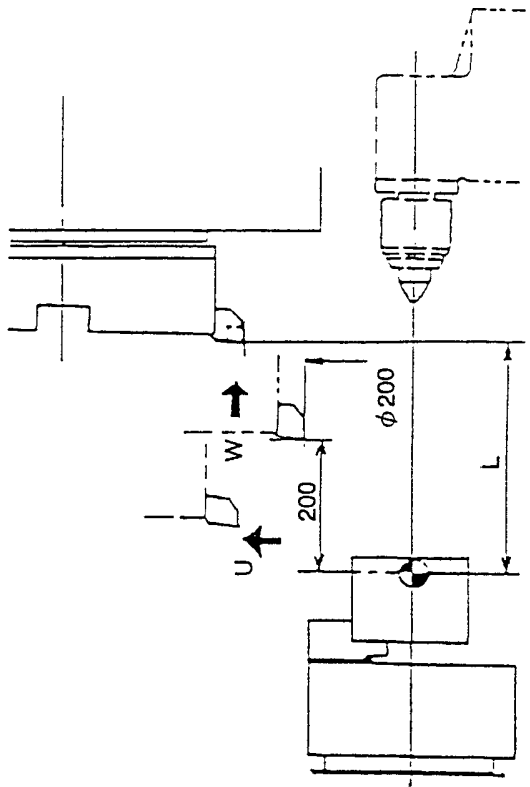
NC Program

|  |   |
|--|---|
| <p>05802<br/> N1 G28 U0<br/> N2 G28 W0 T0100<br/> N3 G50 S2000<br/> N4 G00 X200.0 Z200.0<br/> N5 M01</p>   | <p>Represents a "program number" and used to distinguish from other programs. Do not use an identical program number.</p> <p>Limits a maximum spindle speed to 2,000 rpm.</p> <p>Calls the turret head to be used.</p>                    |
| <p>N701 T07000<br/> N702 G97 S1500 M08<br/> N703 G00 X30.0 Z10.0 M03<br/> N704 G01 G96 Z0 F1.0 S150<br/> N705 X41.85 F0.2<br/> N706 X44.85 Z-1.5<br/> N707 Z-12.0</p>  | <p>Sets a spindle speed per minute to 1,590min.<sup>-1</sup></p> <p>Sets a cutting speed to 150 m/min. This is called constant surface speed control.</p> <p>Used when performing circular cutting in the counterclockwise direction.</p> |
| <p>N708 X46.0<br/> N709 G03 X50.0 Z-14.0 R2.0<br/> N710 G01 Z-17.0<br/> N711 G02 X54.0 Z-19.0 R2.0<br/> N712 G01 X58.0<br/> N713 X62.0 Z-21.0<br/> N714 G00 X70.0<br/> N715 G97 X200.0 Z200.0 S680<br/> N716 M01<br/> N6 G28 U0 W0 T0100<br/> N7 M30</p> | <p>Changes to linear cutting. chamfering and tapering are also linear cutting.</p> <p>Used when performing circular cutting in the clockwise direction.</p> <p>Cancels circular cutting by G01.</p>                                       |
| <p>%</p>   |   |

This is normally called a sequence No. and identifies a block. Avoid using an identical number. Even if a sequence No. is not placed, it is not affected the motion of machine.

Changes to linear cutting. chamfering and tapering are also linear cutting.

Used when performing circular cutting in the clockwise direction.



## 2-2 Details of F, S, T and M Functions

### 2-2-1 F Function (Feed Function)

G99 mode      F □□□.□□□(Up to 6 digits in increment of 0.001)

mm/rev Specify a cutting “feed rate” per spindle revolution or a lead of the threading.

(Example) 0.3 mm/rev = F0.3 or F30

1.0 mm/rev = F1.0 or F100

1.5 P thread = F1.5 or F150

In case of thread cutting, it is possible to command down to 5 digits of decimals.

F□□□.□□□□□(0.00001 unit; max. 8 digits)

Whether lead designation or thread number designation should be selected for the address of E depends on parameter setting.

When 8th place from the right of the parameter No.2403 is 0 ..... Lead designation.

(Example) In case of 14 threads per inch

$$\text{Feed rate} = \frac{25.4}{14 \text{ threads}} = 1.8142857$$

$$1.81429\text{mm/rev} \quad \text{F1.81429}$$

When 8th place from the right of the parameter No.2403 is 1 ..... (Thread number designation)

(Example) E14.0

Max. feed rate 10,000mm/min.

A maximum feed rate depends on the spindle speed used.

$$\text{Assuming the spindle speed to be } N; \quad \frac{5000}{N}$$

(Example) When the spindle speed is 1,000 rpm, the maximum feed rate is;

$$\frac{5000}{1000} = 5.0 \quad \text{F} = 5.0 \text{ mm/rev}$$

G98 mode      F□□□□□□ A decimal point cannot be used.

mm/min Feed rate per minute

Generally, you specify a feed rate per spindle revolution for in case of turning. However, if specified in the G98 mode, a feed rate per minute is set.

(Example) 200 mm/min = F200

- Notes)**
1. Since the G99 mode is set when turning on the power, you do not have to specify it, unless G98 is to be used.
  2. A cutting feed in taper cutting or circular cutting is that of a tool advance direction (tangent direction).
  3. If a cutting feed in G98 mode (G01, G02, G03) is specified, the turret head moves even if the spindle is not running.
  4. When commanding G98 from G99 mode or G99 mode from G98, be sure to command

*F .... as well.*

*In case of F command is missing in the block, F value is effective which is designated just preceding block in G98, G99 mode respectively.*

*To be concrete, it becomes as follows:*

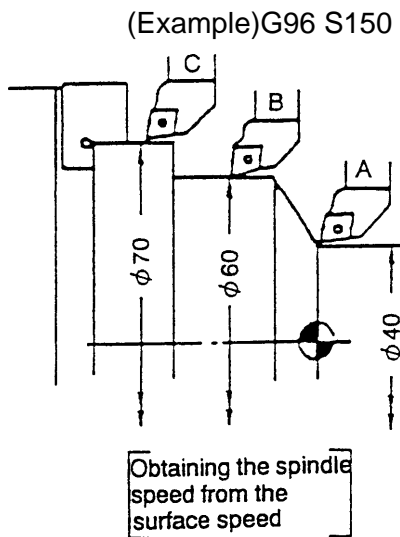
*Indicate “F” that becomes effective in that block with [ ] .*

|                           | (Feed per minute) | (Feed per revolution) |
|---------------------------|-------------------|-----------------------|
| When the power is turn ON | 0                 | 0.00                  |
| N1 G99 F1.23 ;            | 0                 | [1.23]                |
| N2 — ;                    | 0                 | [1.23]                |
| N3 G98 F1000 ;            | [1000]            | 1.23                  |
| N4 — ;                    | [1000]            | 1.23                  |
| N5 G32 F2.34567 ;         | 1000              | [2.34567]             |
| N6 — ;                    | 1000              | [2.34567]             |
| N7 G99 ;                  | 1000              | [2.34]                |
| N8 — ;                    | 1000              | [2.34]                |
| N9 G98 ;                  | [1000]            | 2.34                  |
| N10 — ;                   | [1000]            | 2.34                  |
| N11 G32 ;                 | 1000              | [2.34567]             |

### 2-2-2 S Function (Spindle Function)

Specify a spindle speed or surface speed (cutting speed) with S 4-digit numeral (S□□□□).

| Command  | Description   |
|----------|---|
| G50S□□□□ | Max. spindle speed limit<br>(Example) G50 S1800 : A maximum spindle speed is limited to 1,800 (mim <sup>-1</sup> )  |
| G97S□□□□ | Constant surface speed cancel<br>Specify a spindle revolution with S□□□□ .<br>(Example) G97 S1000 : A spindle speed per minute is set to 1,000 (mim <sup>-1</sup> ) |
| G96S□□□□ | Constant surface speed control<br>When performing constant surface speed control, specify a cutting speed “V” (m/min) with an S 4-digit code (S□□□□ ).              |



: A spindle speed is controlled to 150  
 150 m/min cutting speed at the cutting point.  
 ..... Refer to the left figure.  
 \* Formula for calculating the spindle speed from the surface speed

$$N = \frac{1000 \times V}{\pi \times D}$$

V : Surface speed (m/min)

$\pi$  : 3.14

D : Tool nose position ( $\phi$  mm)

N : Spindle speed ( $\text{mim}^{-1}$ )

$$\text{Spindle speed "N" at the position A} = \frac{1000 \times 150}{3.14 \times 40\phi} = 1193 (\text{mim}^{-1})$$

$$\text{Spindle speed "N" at the position B} = \frac{1000 \times 150}{3.14 \times 60\phi} = 795 (\text{mim}^{-1})$$

$$\text{Spindle speed "N" at the position C} = \frac{1000 \times 150}{3.14 \times 70\phi} = 682 (\text{mim}^{-1})$$

As mentioned above, an automatic change of the spindle speed relating to the work diameter is called as the constant surface speed control.

- Notes)**
1. Considering a workpiece chucking condition, specify the maximum spindle speed limit with S 4-digit code in a G50 block at the beginning of a program.
  2. When roughing with G96, calculate maximum and minimum spindle speeds so that cutting will be performed in a constant power range as much as possible.
  3. When changing over from G96 to G97 and vice versa, specify not only a G code, but also an S code.
  4. When changed over from G96 to G97 and no S code is specified, the spindle is run with the speed specified in the latest S code in G96 mode.
  5. When changed over from G96 to G97 and no S code is specified, the spindle turns with the previously used surface constant speed is S code had been specified in G96 mode.  
 Also, when no S code is specified in G96 mode, S results in 0.
  6. The following interlocks are provided as the rotating conditions of spindle.
    - (1) The direction of the chuck inner clamp and outer clamp key shall be the same direction as that of chuck clamping.
    - (2) Q-setter shall be stored.
    - (3) Rotating speed shall be command with G96 Sxxx.
    - (4) The lamp of advance or retract of center support shall be on. (Option)
    - (5) The door shall be closed.

In case of rotary tool, there are four additional interlocks as follows.

- (1) The connection of C-axis shall be in the status of OFF (M40 command). (Option)
- (2) The connection of rotating tool shall be in the status of OFF (M45 command). (Option)
- (3) Set up of the ACT shall be cancel condition. (Option)
- (4) The safety door of the ATC magazine shall be closed. (Option)

### 2-2-3 T Function (Tool Function)

The tool used and its offset No. can be selected with a 4-digit number following "T".

T□□△△

Turret face selection \_\_\_\_\_ ↑ ↑ \_\_\_\_\_ Offset No.

Face 01 ~ maximum number of faces

#### 1. Setting Coordinate of Tool-nose Position

As a general usage, it is not necessary to command of offset No. Only command of calling of turret as shown below can set the tool-nose position.

Example) If the turret No. 3 is to be called, program as follows:

T0300

#### 2. Setting Coordinate of Tool-nose Position for Arbitrary Offset No. When using an arbitrary offset No., program as follows.

Setting is done with the tool mounting position (diameter, length) of the offset No. 13.

Example)

T0313

Turret No. 3 \_\_\_\_\_ ↑ ↑ \_\_\_\_\_ Offset No.

selected

**Note** 1. Be sure to input the tool-nose point on the tool layout screen.

2. Input "9" to the tool-nose point for drilling end-milling tool. (When a rotating tool is equipped.)

### **Caution**

When "T□□△△" command is specified on the same line as the axis travel command, the indexing of turret is made simultaneously with traveling and a coordinate is set after completion of traveling.

Be careful not to command T function together with the travel command.

### 3. Compound Offset

When an adjustment is made on diametrical dimension of 50 and 70mm respectively at the following workpiece, two or more offset can be applied on one tool.

Example 1)

```

T0900
G97 S2546 M08
G00 X50.0 Z10.0 M03
G96 Z3.0 S200
G01 Z-15.0 F0.2

X70.0 T0919 ← Compound offset
              Z-40.0      Offset No. 19
X84.0 T0900 ← Compound offset
G00              cancel
G97 X200.0 Z200.0 S500
M01
    
```

Example) Input status of dimension adjustment when the part  $\phi 70$  is made larger by 0.03.

TOOL LAYOUT

| OFFSET |      |   |   |   |
|--------|------|---|---|---|
|        | X    | Z | R | T |
| 19     | 0.03 | 0 | 0 | 0 |

**Note)** Be sure to input zero for R and T.

Example 2) Cutting with taper of  $-0.3$  at  $\phi 30$  part

```

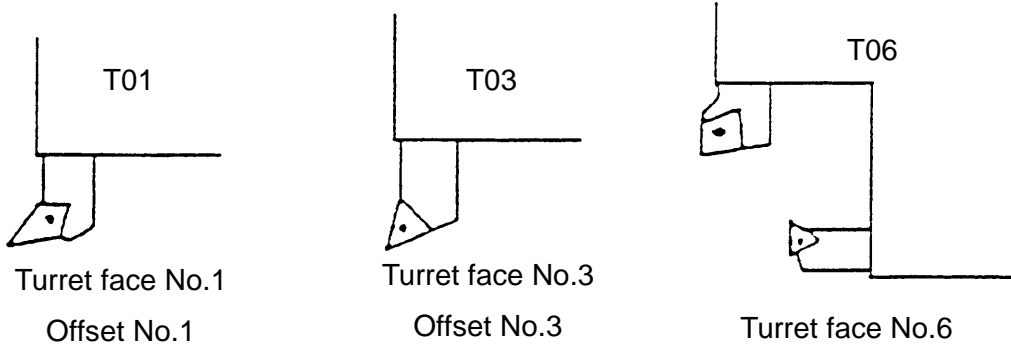
T0500
G97 S2000 M08
G00 Z3.0 M03
      X40.0
G01 X24.0 F1.0
      Z1.0 F0.2
      X30.0 Z-2.0
      Z-90.0 T0525 ← Compound offset
                          Offset No. 25
      X57.00 T0500 ← Compound offset
      X62.0 Z-92.5      cancel
G00 X200.0 Z5.0
M01
    
```

OFFSET

25 X-0.3 Z0 R0 T0



D. Program example



(Compound compensation 33, 34)

(Offset No.6, 36)

```


N100 T0100      The turret face No.1 is indexed and setting-up is
      |
      M01
N300 T0300      The turret face No.3 is indexed and setting-up is
      |
      G01 Z- ○ T0333      Compound compensation ON (Offset No.33)
      |
      X ○ Z ○ T0334      Compound compensation ON (Offset No.34)
      |
      T0300      Cancel compound compensation
      |
      M01
N600 T0600      The turret face No.6 is indexed and setting-up is
      |
      T0636      Multi tool compensation ON (Offset No.36)
      |
      M01
    
```

Example of compensating data

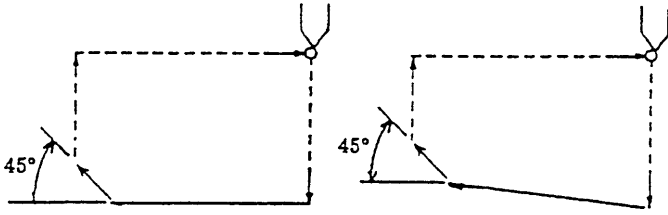
| No. | X                      | Z                      | R   | T |
|-----|------------------------|------------------------|-----|---|
| 01  | Q-setter               | Q-setter               | 0.8 | 3 |
| 03  | Q-setter               | Q-setter               | 0.8 | 3 |
| 06  | Q-setter               | Q-setter               | 0.4 | 2 |
| 33  | Extremely small amount | Extremely small amount | 0   | 0 |
| 34  | Extremely small amount | Extremely small amount | 0   | 0 |
| 36  | Q-setter               | Q-setter               | 0.4 | 2 |

## 2-2-4 M Function (Miscellaneous Function) List (TS15, HT20RIII/23RIII)

Please refer to the details on the Delivery specifications as to the discrimination between Standard or Option.

| M code | Function                | Description   |
|--------|-------------------------|---|
| M00    | Program stop            | This code can stop the machine during its operation, when measuring a workpiece or removing cutting chips. (The spindle and coolant also stop.) To restart, press the CYCLE START key. However, since the spindle and coolant are being suspended, specify M03/M08 in a subsequent block.   |
| M01    | Optional stop           | <p>Same function as M00.</p> <p>An M01 command on a program can be either executed or ignored by means of the OPTIONAL STOP key on the operation panel.</p> <p>Executed when a lamp is lit up.<br/>(optional stop is effective)</p> <p>Ignored when a lamp is lit off.<br/>(optional stop is not effective)</p> <p>Sheet key </p> |
| M02    | Program end             | This code is used in the tape operation and is programmed at the end of the program. It stops the spindle and coolant, and resets NC.   |
| M03    | Spindle forward start   | Viewing from the tailstock side, this code starts the spindle in the counterclockwise direction.  |
| M04    | Spindle reverse start   | Viewing from the tailstock side, this code starts the spindle in the clockwise direction.   |
| M05    | Spindle stop            | This code stops the spindle. When changing over spindle revolution from forward to reverse (or the other way), stop the spindle once with M05, and then specify M04 (M03).  |
| M08    | Coolant start           | This code starts discharging coolant.   |
| M09    | Coolant stop            | This code stops discharging coolant.  |
| M12    | Work count (tool count) | Normally, this code starts a work counter or tool counter to count up.  |

- Note) :**
- M05 and M09 are executed after the completion of the axes travel.
  - Do not specify M codes in the same block duplicately.

| M code | Function                                       | Description   |
|--------|--|---|
| M18    | Release the spindle Positioning                | Release the spindle positioning.  |
| M19    | Spindle Positioning                            | The spindle can be position at the one point.   |
| M23    | Chamfering ON<br>(automatic thread chamfering) | <p>This code performs automatic thread chamfering during a threading cycle (G92). A chamfering length can be set in the parameter in increment of 0.1 L.</p>  <p>When M23 is specified.</p> |
| M24    | Chamfering OFF                                 | This code cancels M23.  |
| M25    | Tailstock low speed advance                    | Low speed advance until advance end. The command is completed when work pushed and hydraulic become ON. But, alarm is on when work is not under pushing within the time set by timer.(D1006).   |
| M26    | Tailstock high speed retract                   | Completed when tailstock retracted at high speed for the time set by timer setting table. (TMR011)  |
| M27    | Tailstock high speed advance                   | Completed when tailstock advanced at high speed for the time set by timer setting table. (TMR007)<br>Don't touch work by command of high speed advance.   |
| M28    | Tailstock retract end                          | The tailstock moves the retract end.  |
| M30    | Program end<br>(memory operation)              | This code is used instead of M02 in case of memoryoperation. In addition to the function of M02, this code returns the program to the top.<br>(Specify in an independent block.)  |
| M31    | No-workpiece chuck<br>Number check             | <p>1) Tool life check.</p> <p>2) Machined work number check by preset type workcounter.</p> <p>3) When a bar feeder is equipped, non blank check.</p>   |

| M code | Function   | Description  |
|--------|--|--|
| M32    | Top cut chuck  | Block ship ON, however, block skip becomes OFF by the top cut signal ON.   |
| M33    | Top cut reset  | Reset the top cut signal.  |
| M34    | Programmable tailstock advance                             | Programmable tailstock pushes work.  |
| M35    | Programmable tailstock retract                             | Programmable tailstock stops pushing work.   |
| M36    | Power off is effective at program stop<br>Power off is not | Power is off by command of M00, M01, M02 or M30 when the power cut off is ON.  |
| M37    | effective at program stop                                  | Power does not off even the command of M00, M01, M02 or M03 when the power cut off is ON.<br>Discharge the air at the live center section. |
| M38    | Center air blow ON   | Air is blown to the center.  |
| M39    | Center air blow OFF  | Stop the air.  |
| M40    | TS15   |  |
| M41    | M40  | Main spindle low-speed gear selection 30~1000min <sup>-1</sup>   |
|        | M41  | Main spindle high-speed gear selection 30~6000min <sup>-1</sup>  |
|        | HT20RIII   |  |
|        | M40  | Main spindle low-speed gear selection 30~1500min <sup>-1</sup>   |
|        | M41  | Main spindle high-speed gear selection 30~5000min <sup>-1</sup>  |
|        | HT23RIII   |  |
|        | M40  | Main spindle low-speed gear selection 30~1300min <sup>-1</sup>   |
|        | M41  | Main spindle high-speed gear selection 30~4000min <sup>-1</sup>  |
| M46    | Spindle override is effective                              | The spindle override can be applied.   |
| M47    | Spindle override is not effective                          | The spindle override is ignores.   |
| M48    | feedrate override is effective                             | The feedrate override can be applied.  |
| M49    | feedrate override is not effective                         | The feedrate override is ignores.  |


| M code | Function                                | Description                                       |
|--------|---|---|
| M51    | Spindle air blow ON                     | Discharge the air at the chuck section.           |
| M52    | Spindle air blow OFF                    | Stop the air.                                     |
| M53    | Tool edge measuring sensor air blow ON  | Air is blown to the measuring sensor section.     |
| M54    | Tool edge measuring sensor air blow OFF | Air blow at the sensor section stops.             |
| M55    | Tool edge measuring arm OUT             | Measuring sensor swings out.                      |
| M56    | Tool edge measuring arm RETURN          | Measuring sensor is stored.                       |
| M61    | Auto door open                          | The door opens by a program command.              |
| M62    | Auto door close                         | Closes the door.                                  |
| M63    | Unloader advance                        | Catch a workpiece by protrusion of the unloader.  |
| M64    | Unloader retract                        | Retract the unloader.                             |
| M66    | Chuck clamping pressure is low          | The pressure of spindle chuck shift to low side.  |
| M67    | Chuck clamping pressure is high         | The pressure of spindle chuck shift to high side. |
| M68    | Chuck side close                        | The spindle chuck closes.                         |
| M69    | Chuck side open                         | The spindle chuck opens.                          |
| M70    | Call light ON                           | Call light is lit.                                |
| M71    | Work measuring arm OUT                  | Work measuring sensor swings out.                 |
| M72    | Work measuring arm RETURN               | Work measuring sensor is stored.                  |
| M73    | Work measuring sensor air blow ON       | Air is blown to work measuring sensor.            |

| M code | Function  | Description  |
|--------|---|--|
| M74    | Work measuring sensor air blow OFF                  | Air blow at the measuring sensor stops.  |
| M75    | Chip conveyor start                                 | Chip conveyor rotates to normal direction.   |
| M76    | Chip conveyor stop                                  | Chip conveyor stops.   |
| M81    | Robot service 1                                     | Robot start 1  |
| M82    | Robot service 2                                     | Robot start 2  |
| M83    | Tool edge measuring arm Check condition ineffective | When measuring arm swings, chuck open/close condition is neglected.  |
| M84    | Tool edge measuring arm Check condition ineffective | When measuring arm swings, chuck open/close condition becomes effective.   |
| M85    | Index chuck activated                               | Turn by 90° from index chuck indexing position, 0° and 90°.  |
| M86    | Index chuck 45°                                     | Turn by 45° from present indexing position.  |
| M87    | Index chuck 90°                                     | Turn by 90° from present indexing position.  |
| M88    | Machine proper standby                              | The machine proper standby from the robot.   |
| M89    | Release standby of robot                            | The robot stops until release a standby from the machine.  |
| M98    | Subprogram calling                                  | This code switches program from a main program to a subprogram.  |
| M99    | Sub program end                                     | This code returns control from a subprogram to a main program. If specified in the main program, the program returns to its top. |
| M122   | Air blow in spindle ON                              | Air is blown from inside spindle   |
| M123   | Air blow in spindle OFF                             | Air blow from inside spindle stops.  |

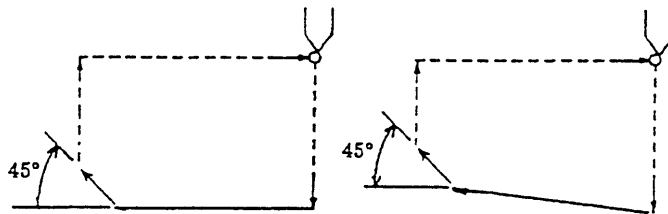
| M code | Function            | Description                 |
|--------|---------------------|-----------------------------|
| M124   | Turret air blow ON  | Air is blown from turret.   |
| M125   | Turret air blow OFF | Air blow from turret stops. |

## M Function (Miscellaneous Function) List (TF25)

Please refer to the details on the Delivery specifications as to the discrimination between Standard or Option.

| M code | Function                        | Description   |
|--------|---------------------------------|---|
| M00    | Program stop                    | This code can stop the machine during its operation, when measuring a workpiece or removing cutting chips. (The spindle and coolant also stop.) To restart, press the CYCLE START key. However, since the spindle and coolant are being suspended, specify M03/M08 in a subsequent block.   |
| M01    | Optional stop                   | <p>Same function as M00.</p> <p>An M01 command on a program can be either executed or ignored by means of the OPTIONAL STOP key on the operation panel.</p> <p>Executed when a lamp is lit up.<br/>(optional stop is effective)</p> <p>Sheet key  Ignored when a lamp is lit off.<br/>(optional stop is not effective)</p> |
| M02    | Program end                     | This code is used in the tape operation and is programmed at the end of the program. It stops the spindle and coolant, and resets NC.   |
| M03    | Spindle forward start           | Viewing from the tailstock side, this code starts the spindle in the counterclockwise direction.  |
| M04    | Spindle reverse start           | Viewing from the tailstock side, this code starts the spindle in the clockwise direction.   |
| M05    | Spindle stop                    | This code stops the spindle. When changing over spindle revolution from forward to reverse (or the other way), stop the spindle once with M05, and then specify M04 (M03).  |
| M08    | Coolant start                   | Cutting oil and chip sweeping coolant is sent out.  |
| M09    | Coolant stop                    | This code stops discharging coolant.  |
| M12    | Work count (tool count)         | Normally, this code starts a work counter or tool counter to count up.  |
| M18    | Release the spindle Positioning | Release the spindle positioning.  |

- Note) :**
- M05 and M09 are executed after the completion of the axes travel.
  - Do not specify M codes in the same block duplicately.

| M code | Function                                       | Description   |
|--------|--|---|
| M19    | Spindle Positioning                            | The spindle can be position at the one point.   |
| M23    | Chamfering ON<br>(automatic thread chamfering) | <p>This code performs automatic thread chamfering during a threading cycle (G92). A chamfering length can be set in the parameter in increment of 0.1 L.</p>  <p>When M23 is specified.</p> |
| M24    | Chamfering OFF                                 | This code cancels M23.  |
| M25    | Tailstock low speed advance                    | <p>Low speed advance until advance end. The command is completed when work is pushed and hydraulic become ON.</p> <p>But, alarm is on when work is not under pushing within the time set by timer. (D1006)</p>  |
| M26    | Tailstock high speed retract                   | Completed when tailstock retracted at high speed for the time set by timer setting table. (D1012)   |
| M27    | Tailstock high speed advance                   | <p>Completed when tailstock advanced at high speed for the time set by timer setting table. (D1034)</p> <p>Don't touch work by command of high speed advance.</p>   |
| M28    | Tailstock backward end                         | Tailstock moves backward until stroke end, and movement is finished. (high speed)   |
| M30    | Program end<br>(memory operation)              | <p>This code is used instead of M02 in case of memory operation. In addition to the function of M02, this code returns the program to the top.</p> <p>(Specify in an independent block.)</p>  |
| M31    | No-workpiece chuck<br>Number check             | <p>1) Tool life check.</p> <p>2) Machined work number check by preset type work counter.</p> <p>3) When a bar feeder is equipped, non blank check.</p>  |
| M32    | Top cut chuck                                  | Block ship ON, however, block skip becomes OFF by the top cut signal ON.  |


| M code | Function                                   | Description  |
|--------|--|--|
| M33    | Top cut reset                              | Reset the top cut signal.  |
| M34    | Programmable tailstock advance             | Programmable tailstock pushes work.  |
| M35    | Programmable tailstock retract             | Programmable tailstock stops pushing work.   |
| M36    | Power off is effective at program stop     | Power is off by command of M00, M01, M02 or M30 when the power cut off is ON.  |
| M37    | Power off is not effective at program stop | Power does not off even the command of M00, M01, M02 or M03 when the power cut off is ON.<br>Discharge the air at the live center section. |
| M38    | Center air blow ON                         | Air is blown to the center.  |
| M39    | Center air blow OFF                        | Stop the air.  |
| M40    | Main spindle low-speed gear selection      | TF25 30~450~1200 min <sup>-1</sup>   |
| M41    | Main spindle high-speed gear selection     | TF25 30~1200~4000 min <sup>-1</sup>  |
| M46    | Spindle override is effective              | The spindle override can be applied.   |
| M47    | Spindle override is not effective          | The spindle override is ignores.   |
| M48    | feedrate override is effective             | The feedrate override can be applied.  |
| M49    | feedrate override is not effective         | The feedrate override is ignores.  |
| M51    | Spindle air blow ON                        | Discharge the air at the chuck section.  |
| M52    | Spindle air blow OFF                       | Stop the air.  |

| M code | Function                                | Description                                       |
|--------|---|---|
| M53    | Tool edge measuring sensor air blow ON  | Air is blown to the measuring sensor section.     |
| M54    | Tool edge measuring sensor air blow OFF | Air blow at the sensor section stops.             |
| M55    | Tool edge measuring arm OUT             | Measuring sensor swings out.                      |
| M56    | Tool edge measuring arm RETURN          | Measuring sensor is stored.                       |
| M61    | Auto door open                          | The door opens by a program command.              |
| M62    | Auto door close                         | Closes the door.                                  |
| M63    | Unloader advance                        | Catch a workpiece by protrusion of the unloader.  |
| M64    | Unloader retract                        | Retract the unloader.                             |
| M66    | Chuck clamping pressure is low          | The pressure of spindle chuck shift to low side.  |
| M67    | Chuck clamping pressure is high         | The pressure of spindle chuck shift to high side. |
| M68    | Chuck side close                        | The spindle chuck closes.                         |
| M69    | Chuck side open                         | The spindle chuck opens.                          |
| M70    | Call light ON                           | Call light is lit.                                |
| M71    | Work measuring arm OUT                  | Work measuring sensor swings out.                 |
| M72    | Work measuring arm RETURN               | Work measuring sensor is stored.                  |
| M73    | Work measuring sensor air blow ON       | Air is blown to work measuring sensor.            |
| M74    | Work measuring sensor air blow OFF      | Air blow at the measuring sensor stops.           |
| M75    | Chip conveyor start                     | Chip conveyor rotates to normal direction.        |

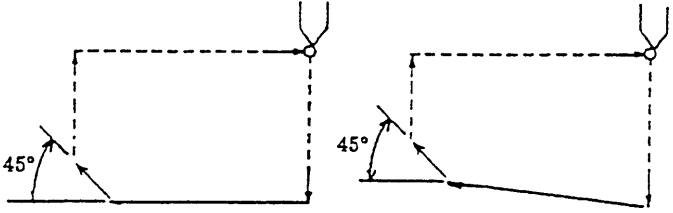
| M code | Function   | Description  |
|--------|--|--|
| M76    | Chip conveyor stop   | Chip conveyor stops.   |
| M81    | Robot service 1  | Robot start 1  |
| M82    | Robot service 2  | Robot start 2  |
| M83    | Tool edge<br>measuring arm<br>Check condition<br>ineffective | When measuring arm swings, chuck open/close condition is neglected.  |
| M84    | Tool edge<br>measuring arm<br>Check condition<br>ineffective | When measuring arm swings, chuck open/close condition becomes effective.   |
| M85    | Index chuck<br>activated                                     | Turn by 90° from index chuck indexing position, 0° and 90°.  |
| M86    | Index chuck 45°  | Turn by 45° from present indexing position.  |
| M87    | Index chuck 90°  | Turn by 90° from present indexing position.  |
| M88    | Machine proper<br>standby                                    | The machine proper standby from the robot.   |
| M89    | Release standby of<br>robot                                  | The robot stops until release a standby from the machine.  |
| M98    | Subprogram calling   | This code switches program from a main program to a subprogram.  |
| M99    | Sub program end  | This code returns control from a subprogram to a main program. If specified in the main program, the program returns to its top. |
| M122   | Air blow in spindle<br>ON                                    | Air is blown from inside spindle   |
| M123   | Air blow in spindle<br>OFF                                   | Air blow from inside spindle stops.  |
| M124   | Turret air blow ON   | Air is blown from turret.  |
| M125   | Turret air blow OFF  | Air blow from turret stops.  |

## M Function (Miscellaneous Function) List (HT25G/30G)

Please refer to the details on the Delivery specifications as to the discrimination between Standard or Option.

| M code | Function                | Description   |
|--------|-------------------------|---|
| M00    | Program stop            | This code can stop the machine during its operation, when measuring a workpiece or removing cutting chips. (The spindle and coolant also stop.) To restart, press the CYCLE START key. However, since the spindle and coolant are being suspended, specify M03/M08 in a subsequent block.   |
| M01    | Optional stop           | <p>Same function as M00.</p> <p>An M01 command on a program can be either executed or ignored by means of the OPTIONAL STOP key on the operation panel.</p> <p>Executed when a lamp is lit up.<br/>(optional stop is effective)</p> <p>Ignored when a lamp is lit off.<br/>(optional stop is not effective)</p> <p>Sheet key </p> |
| M02    | Program end             | This code is used in the tape operation and is programmed at the end of the program.<br>It stops the spindle and coolant, and resets NC.  |
| M03    | Spindle forward start   | Viewing from the tailstock side, this code starts the spindle in the counterclockwise direction.  |
| M04    | Spindle reverse start   | Viewing from the tailstock side, this code starts the spindle in the clockwise direction.   |
| M05    | Spindle stop            | This code stops the spindle.<br>When changing over spindle revolution from forward to reverse (or the other way), stop the spindle once with M05, and then specify M04 (M03).   |
| M08    | Coolant start           | This code starts discharging coolant.   |
| M09    | Coolant stop            | This code stops discharging coolant.  |
| M12    | Work count (tool count) | Normally, this code starts a work counter or tool counter to count up.  |

- Note) :**
- M05 and M09 are executed after the completion of the axes travel.
  - Do not specify M codes in the same block duplicately.

| M code | Function                                       | Description   |
|--------|--|---|
| M18    | Release the spindlePositioning                 | Release the spindle positioning.  |
| M19    | Spindle Positioning                            | The spindle can be positione at the one point.  |
| M23    | Chamfering ON<br>(automatic thread chamfering) | <p>This code perfotms automatic thread chamfering during a thresding cycle (G92). A chamfering length can be set in the parameter in increment of 0.1 L.</p>  <p>When M23 is specified.</p> |
| M24    | Chamfering OFF                                 | This code cancels M23.  |
| M25    | Tailstock advance                              | <p>The tailstock advances by the command of M25 during program operation.</p> <p>However, ot works when the spindle is stopped.</p>   |
| M26    | Tailstock retract                              | The quill retracts when the spindle is stopped.   |
| M30    | Program end<br>(memory operation)              | <p>This code is used instead of M02 in case of memory operation. In addition to the function of M02, this code returns the program to the top.</p> <p>(Specify in an independent block.)</p>  |
| M31    | No-workpiece chuck<br>Number check             | <p>1) Tool life check.</p> <p>2) Machined work number check by preset type work counter.</p> <p>3) When a bar feeder is equipped, non blank check.</p>  |
| M32    | Top cut chuck                                  | Block ship ON, however, block skip becomes OFF by the top cut signal ON.  |
| M33    | Top cut reset                                  | Reset the top cut signal.   |
| M34    | Programmable tailstock advance                 |   |
| M35    | Programmable tailstock retract                 |   |


| M code | Function   | Description  |
|--------|--|--|
| M36    | Power off is effective at program stop                         | Power is off by command of M00, M01, M02 or M30 when the power cut off is ON.  |
| M37    | Power off is not effective at program stop                     | Power does not off even the command of M00, M01, M02 or M03 when the power cut off is ON.<br>Discharge the air at the live center section. |
| M38    | Center air blow ON   | Air blow is discharged to live center section.   |
| M39    | Center air blow OFF  | Stop the air.  |
| M40    | Main spindle low speed rotation area                           | TG25: 280~1111 rotations<br>TG30: 280~1111 rotations   |
| M41    | Main spindle high rotation area                                | TG25: 908 – 3600 rotations<br>TG30: 735 – 2500 rotations   |
| M46    | Spindle override is effective                                  | The spindle override can be applied.   |
| M47    | Spindle override is not effective                              | The spindle override is ignores.   |
| M48    | feedrate override is effective                                 | The feedrate override can be applied.  |
| M49    | feedrate override is not effective                             | The feedrate override is ignores.  |
| M51    | Spindle air blow ON  | Discharge the air at the chuck section.  |
| M52    | Spindle air blow OFF   | Stop the air.  |
| M53    | Open/close condition neglect of tool tip measurement check ON  |  |
| M54    | Open/close condition neglect of tool tip measurement check OFF |  |

| M code | Function                                       | Description                                      |
|--------|--|--|
| M55    | Work measurement skip signal is effective.     |  |
| M56    | Work measurement skip signal is not effective. |  |
| M57    | Center pressure is low.                        |  |
| M58    | Center pressure is high.                       |  |
| M59    | Air blow in spindle ON                         |  |
| M60    | Air blow in spindle OFF                        |  |
| M61    | Auto door open                                 | The door opens by a program command.             |
| M62    | Auto door close                                | Closes the door.                                 |
| M63    | Unloader advance                               | Catch a workpiece by protrusion of the unloader. |
| M64    | Unloader retract                               | Retract the unloader.                            |
| M66    | Chuck clamping pressure is low                 | The pressure of spindle chuck shift to low side. |
| M67    | Chuck clamping pressure is high                | Switch over to strong side                       |
| M68    | Chuck side close                               | The spindle chuck closes.                        |
| M69    | Chuck side open                                | The spindle chuck opens.                         |
| M73    | Work measuring sensor air blow ON              |  |
| M74    | Work measuring sensor air blow OFF             |  |
| M75    | Tool-nose measuring arm swing in               |  |

| M code | Function   | Description  |
|--------|--|--|
| M76    | Tool-nose measuring arm return                           |  |
| M77    | Tool-nose measuring sensor air blow ON                   |  |
| M78    | Tool-nose measuring sensor air blow OFF                  |  |
| M79    | Unloader cylinder out                                    |  |
| M80    | Unloader cylinder return                                 |  |
| M81    | Robot service 1  | Robot start 1  |
| M82    | Robot service 2  | Robot start 2  |
| M83    | Chuck interlock of tool tip measurement is not effective |  |
| M84    | Chuck interlock of tool tip measurement is effective     |  |
| M88    | Machine proper standby                                   | The machine proper standby from the robot.   |
| M89    | Release standby of robot                                 | The robot stops until release a standby from the machine.  |
| M98    | Subprogram calling                                       | This code switches program from a main program to a subprogram.  |
| M99    | Main program return                                      | This code returns control from a subprogram to a main program. If specified in the main program, the program returns to its top. |

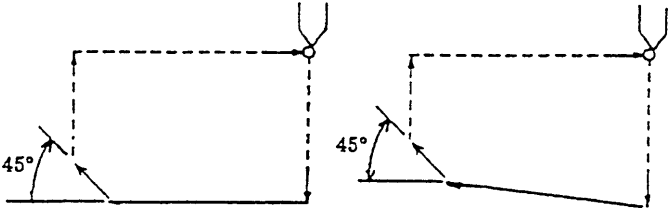
## M Function (Miscellaneous Function) List (HT40G/50G)

Please refer to the details on the Delivery specifications as to the discrimination between Standard or Option.

| M code | Function                        | Description  |
|--------|---------------------------------|--|
| M00    | Program stop                    | This code can stop the machine during its operation, when measuring a workpiece or removing cutting chips. (The spindle and coolant also stop.) To restart, press the CYCLE START key. However, since the spindle and coolant are being suspended, specify M03/M08 in a subsequent block.  |
| M01    | Optional stop                   | <p>Same function as M00.</p> <p>An M01 command on a program can be either executed or ignored by means of the OPTIONAL STOP key on the operation panel.</p> <p>Executed when a lamp is lit up.<br/>(optional stop is effective)</p> <p>Sheet key  Ignored when a lamp is lit off.<br/>(optional stop is not effective)</p> |
| M02    | Program end                     | This code is used in the tape operation and is programmed at the end of the program.<br>It stops the spindle and coolant, and resets NC.   |
| M03    | Spindle forward start           | Viewing from the tailstock side, this code starts the spindle in the counterclockwise direction.   |
| M04    | Spindle reverse start           | Viewing from the tailstock side, this code starts the spindle in the clockwise direction.  |
| M05    | Spindle stop                    | This code stops the spindle.<br>When changing over spindle revolution from forward to reverse (or the other way), stop the spindle once with M05, and then specify M04 (M03).  |
| M08    | Coolant start                   | This code starts discharging coolant.  |
| M09    | Coolant stop                    | This code stops discharging coolant.   |
| M12    | Work count (tool count)         | Normally, this code starts a work counter or tool counter to count up.   |
| M18    | Release the spindle Positioning | Release the spindle positioning.   |

**Note) :**

- M05 and M09 are executed after the completion of the axes travel.
- Do not specify M codes in the same block duplicately.

| M code | Function                                       | Description   |
|--------|--|---|
| M19    | Spindle Positioning                            | The spindle can be positione at the one point.  |
| M23    | Chamfering ON<br>(automatic thread chamfering) | <p>This code perfotms automatic thread chamfering during a thresding cycle (G92). A chamfering length can be set in the parameter in increment of 0.1 L.</p>  <p style="text-align: center;">When M23 is specified.</p> |
| M24    | Chamfering OFF                                 | This code cancels M23.  |
| M25    | Tailstock advance                              | <p>The tailstock advances by the command of M25 during program operation.</p> <p>However, ot works when the spindle is stopped.</p> <p>It advances by tape chuck mode, too.</p>   |
| M26    | Tailstock retract                              | The quill retracts when the spindle is stopped.   |
| M30    | Program end<br>(memory operation)              | <p>This code is used instead of M02 in case of memory operation. In addition to the function of M02, this code returns the program to the top.</p> <p>(Specify in an independent block.)</p>  |
| M31    | No-workpiece chuck<br>Number check             | <p>1) Tool life check.</p> <p>2) Machined work number check by preset type work counter.</p> <p>3) When a bar feeder is equipped, non blank check.</p>  |
| M32    | Top cut chuck                                  | Block skip ON, however, block skip becomes OFF by the top cut signal ON.  |
| M33    | Top cut reset                                  | Reset the top cut signal.   |
| M36    | Power off is<br>effective at program<br>stop   | Power is off by command of M00, M01, M02 or M30 when the power cut off is ON.   |

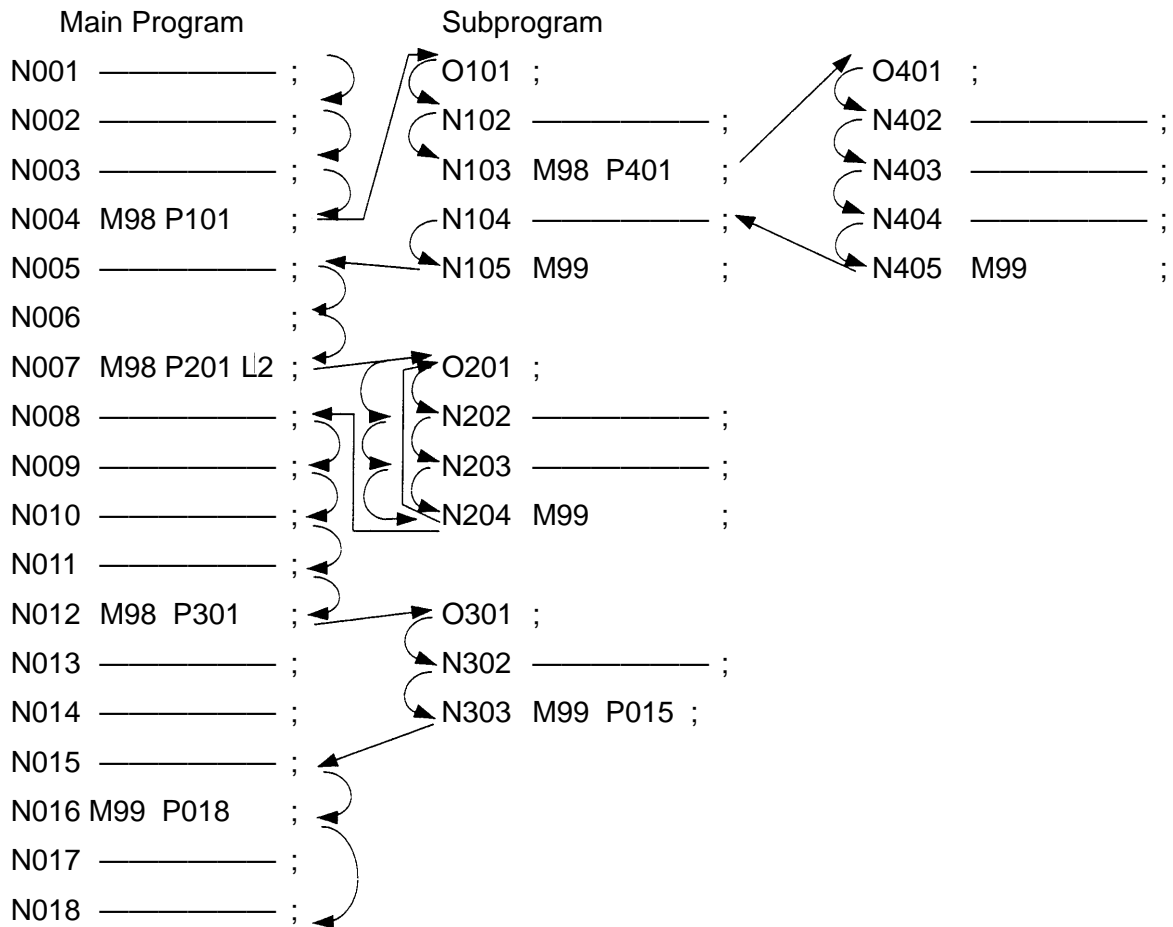
| M code | Function                                   | Description  |
|--------|--|--|
| M37    | Power off is not effective at program stop | Power does not off even the command of M00, M01, M02 or M03 when the power cut off is ON.<br>Discharge the air at the live center section. |
| M38    | Center air blow ON                         | Air is blown to the center.  |
| M39    | Center air blow OFF                        | Stop the air.  |
| M40    | HT40G                                      |  |
| M41    | M40  | Main spindle low-speed gear selection 20~120 ~ 355min <sup>-1</sup>  |
| M42    | M41  | Main spindle middle-speed gear selection 65~353 ~1035min <sup>-1</sup>   |
|        | M42  | Main spindle high-speed gear selection 120~821 ~2000min <sup>-1</sup>  |
|        | HT50G                                      |  |
|        | M40  | Main spindle low-speed gear selection 20~120 ~ 414min <sup>-1</sup>  |
|        | M41  | Main spindle high-speed gear selection 53~330 ~1100min <sup>-1</sup>   |
| M46    | Spindle override is effective              | The spindle override can be applied.   |
| M47    | Spindle override is not effective          | The spindle override is ignores.   |
| M48    | feedrate override is effective             | The feedrate override can be applied.  |
| M49    | feedrate override is not effective         | The feedrate override is ignores.  |
| M51    | Spindle air blow ON                        | Discharge the air at the chuck section.  |
| M52    | Spindle air blow OFF                       | Stop the air.  |
| M53    | Tool edge measuring sensor air blow ON     | Air is blown to the measuring sensor section.  |
| M54    | Tool edge measuring sensor air blow OFF    | Air blow at the sensor section stops.  |
| M55    | Tool edge measuring arm OUT                | Measuring sensor swings out.   |

| M code | Function                           | Description                                       |
|--------|------------------------------------|---|
| M56    | Tool edge measuring arm RETURN     | Measuring sensor is stored.                       |
| M58    | Automatic anti-swing closed        | Anti-swing arm swings out.                        |
| M59    | Automatic anti-swing loosened      | Anti-swing arm returns.                           |
| M61    | Auto door open                     | The door opens by a program command.              |
| M62    | Auto door close                    | Closes the door.                                  |
| M63    | Unloader advance                   | Catch a workpiece by protrusion of the unloader.  |
| M64    | Unloader retract                   | Retract the unloader.                             |
| M66    | Chuck clamping pressure is low     | The pressure of spindle chuck shift to low side.  |
| M67    | Chuck clamping pressure is high    | The pressure of spindle chuck shift to high side. |
| M68    | Chuck side close                   | The spindle chuck closes.                         |
| M69    | Chuck side open                    | The spindle chuck opens.                          |
| M71    | Work measuring arm OUT             | Work measuring sensor swings out.                 |
| M72    | Work measuring arm RETURN          | Work measuring sensor is stored.                  |
| M73    | Work measuring sensor air blow ON  | Air is blown to work measuring sensor.            |
| M74    | Work measuring sensor air blow OFF | Air blow at the measuring sensor stops.           |
| M75    | Chip conveyor start                | Chip conveyor rotates to normal direction.        |
| M76    | Chip conveyor stop                 | Chip conveyor stops.                              |
| M81    | Robot service 1                    | Robot start 1                                     |
| M82    | Robot service 2                    | Robot start 2                                     |

| M code | Function   | Description  |
|--------|--|--|
| M83    | Tool edge measuring arm<br>Check condition ineffective | When measuring arm swings, chuck open/close condition is neglected.  |
| M84    | Tool edge measuring arm<br>Check condition ineffective | When measuring arm swings, chuck open/close condition becomes effective.   |
| M88    | Machine proper standby                                 | The machine proper standby from the robot.   |
| M89    | Release standby of robot                               | The robot stops until release a standby from the machine.  |
| M98    | Subprogram calling                                     | This code switches program from a main program to a subprogram.  |
| M99    | Sub program end  | This code returns control from a subprogram to a main program. If specified in the main program, the program returns to its top. |
| M122   | Air blow in spindle ON                                 | Air is blown from inside spindle   |
| M123   | Air blow in spindle OFF                                | Air blow from inside spindle stops.  |
| M124   | Turret air blow ON                                     | Air is blown from turret.  |
| M125   | Turret air blow OFF                                    | Air blow from turret stops.  |

## Example of Subprogram Call

(Example)



**Note** 1) Another subprogram can be called from one subprogram.

Although the example above calls subprograms doubly, they can be call quadruply at most.

2) One call command can repeatedly call the subprogram for 99999999 times running.

3) When the subprogram ends, if a sequence number is specified with P, control does not return to next to the program called by a parent, but to the sequence number specified with P'.

## 2-3 Details of G Function


### 2-3-1 List of G Function (SEICOS-Σ10L/20L)

Please refer to the details on the Delivery specifications as to the discrimination between Standard or Option.

| Group | G code | Function   |   |
|-------|--------|--|---|
| 01    | G00    | Positioning (Rapid traverse)   |   |
|       | G01    | Linear interpolation   |   |
|       | G02    | Circular arc interpolation/Helical interpolation CW                  |   |
|       | G03    | Circular arc interpolation/Helical interpolation CCW                 |   |
| 00    | G04    | Dwell  |   |
|       | G07    | Hypothetical axis interpolation                                      |   |
|       | G09    | Exact stop   |   |
|       | G10    | Data setting   |   |
|       | G11    | Data setting mode cancel   |   |
| 02    | G17    | Xp - Yp plane designation  | Xp: X-axis } or its<br>Yp: Y-axis } parallel<br>Zp: Z-axis } axis |
|       | G18    | Zp - Xp plane designation  |   |
|       | G19    | Yp - Zp plane designation  |   |
| 06    | G20    | Inch input   |   |
|       | G21    | Metric input   |   |
| 04    | G22    | Stored stroke check ON   |   |
|       | G23    | Stored stroke check OFF  |   |
| 00    | G27    | Reference point return check   |   |
|       | G28    | Reference point return   |   |
|       | G29    | Return from reference point  |   |
|       | G30    | 2nd, 3rd and 4th reference point                                     |   |
|       | G301   | Floating reference point return                                      |   |
|       | G31    | Skip function  |   |
| 01    | G32    | Thread cutting   |   |
|       | G34    | Variable lead thread cutting   |   |
| 00    | G38    | Tool tip R compensation/Tool radius compensation vector retention    |   |
|       | G39    | Tool tip R compensation/Tool radius compensation corner circular arc |   |
| 07    | G40    | Tool radius compensation cancel                                      |   |
|       | G41    | Tool radius compensation left side                                   |   |
|       | G42    | Tool radius compensation right side                                  |   |
| 00    | G50    | Coordinate system setting/Setting of maximum high speed of spindle   |   |
|       | G52    | Back face machining mode   |   |
|       | G53    | Machine coordinate system selection                                  |   |
| 12    | G54    | Work length alteration 1   |   |
|       | G55    | Work length alteration 2   |   |
| 00    | G59    | Local coordinate system setting                                      |   |

| Group | G code | Function                                   |
|-------|--------|--|
| 13    | G61    | Exact stop mode                            |
|       | G62    | Automatic corner override mode             |
|       | G63    | Tapping mode                               |
|       | G64    | Cutting mode                               |
| 00    | G65    | Macro calling                              |
| 14    | G66    | Macro module calling                       |
|       | G67    | Macro module calling cancel                |
| 00    | G70    | Finishing cycle                            |
|       | G71    | OD/ID roughing cycle                       |
|       | G72    | End face roughing cycle                    |
|       | G73    | Closed loop turning cycle                  |
|       | G74    | End face cutting-off cycle                 |
|       | G75    | ID/OD cutting-off cycle                    |
|       | G76    | Multi-type thread cutting cycle            |
| 09    | G80    | Drilling cycle cancel                      |
|       | G81    | Drilling cycle, Spot drilling cycle        |
|       | G82    | Drilling cycle, Counter boring cycle       |
|       | G83    | Peck drilling cycle                        |
|       | G831   | Peck drilling cycle                        |
|       | G84    | Tapping cycle                              |
|       | G841   | Reverse tapping cycle                      |
|       | G842   | Direct tapping cycle                       |
|       | G843   | Reverse direct tapping cycle               |
|       | G85    | Boring cycle                               |
|       | G86    | Boring cycle                               |
|       | G861   | Fine boring cycle                          |
|       | G87    | Back boring cycle                          |
|       | G88    | Boring cycle                               |
|       | G89    | Boring cycle                               |
| 01    | G90    | OD/ID turning cycle                        |
|       | G92    | Single type thread cutting cycle           |
|       | G94    | End face turning cycle                     |
| 17    | G96    | Constant surface speed control             |
|       | G196   | Constant surface speed control (Back face) |
|       | G97    | Constant surface speed control cancel      |
| 05    | G98    | Feed per minute (mm/min)                   |
|       | G99    | Feed per rotation (mm <sup>-1</sup> )      |
| 22    | G120   | Polar coordinate interpolation mode cancel |
|       | G121   | Polar coordinate interpolation mode        |

| Group | G code | Function   |
|-------|--------|--|
| 00    | G128   | Scroll cutting speed control   |
| 18    | G130   | Tool life management OFF   |
|       | G131   | Tool life management ON  |
| 27    | G140   | Automatic tool tip R compensation/Tool radius compensation cancel mode |
|       | G143   | Automatic tool tip R compensation effective mode                       |
|       | G144   | Automatic tool tip R compensation effective mode (G144 = G143)         |
|       | G145   | Tool radius compensation effective mode                                |
| 00    | G141   | Automatic tool tip R compensation left side                            |
|       | G142   | Automatic tool tip R compensation right side                           |
| 16    | G150   | Groove width compensation cancel                                       |
|       | G151   | Groove width compensation for end face                                 |
|       | G152   | Groove width compensation for OD/ID                                    |
| 25    | G170   | Front face machining mode  |
|       | G171   | Back face machining mode   |
| 10    | G198   | Initial point return of fixed cycle for drilling                       |
|       | G199   | R point return of fixed cycle for drilling                             |
| 00    | G251   | Multi-buffer   |
|       | G261   | S designation for spindle  |
|       | G262   | S designation for rotating tool  |
|       | G263   | S designation for sub spindle  |
|       | G271   | Cylindrical interpolation  |
| 15    | G501   | Programmable mirror image reset  |
|       | G511   | Programmable mirror image set  |

- Note** 1) When the source power is switched on, those G codes marked  are set.
- 2) G codes of 00 group indicate those which are not modal, and are effective to the blocks indicated.
- 3) When G codes which are not listed in G Code List are commanded, alarm is displayed, and when G codes which don't have corresponding options, alarm is displayed.
- 4) Any numbers of G codes can be commanded in the same block, if they belong to different groups.  
When two or more of G codes which belong to the same group are commanded, G code later commanded becomes effective.

## 2-3-2 G50 Maximum Spindle Speed Setting

Using a command "G50 S ..... ;", you can directly specify the upper limit value of a spindle speed ( $\text{min}^{-1}$ ) with a 4-digit numerical value following an address S.

When a S beyond the upper limit has commanded after this command, it is clamped at this upper limit.

Even in constant surface speed control (G96 mode), the spindle rotation speed for the specified surface speed ( $\text{m}/\text{min.}$  or  $\text{ft}/\text{min.}$ ) will be clamped to this upper limit.

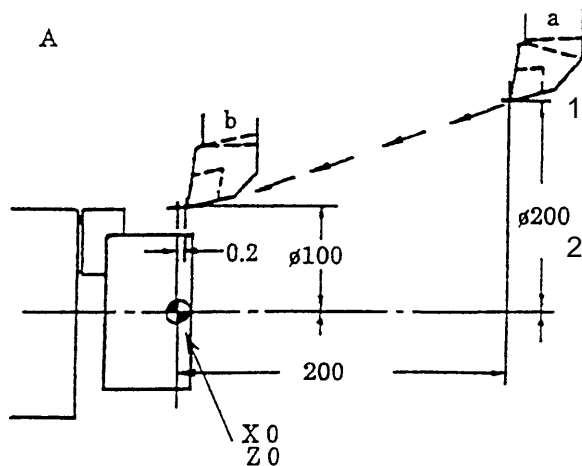
(Example) G50 S2000 ;

Fixes the maximum spindle speed to  $2,000 \text{ min}^{-1}$

**Note:** Depending on a workpiece loading state, specify the maximum spindle speed (G50 Sxxxx) at the beginning of the program.

## 2-3-3 G00 Positioning

Specify this G code when feeding a tool by rapid traverse. This is used when approaching the tool to the workpiece or when retreating it after cutting is completed.



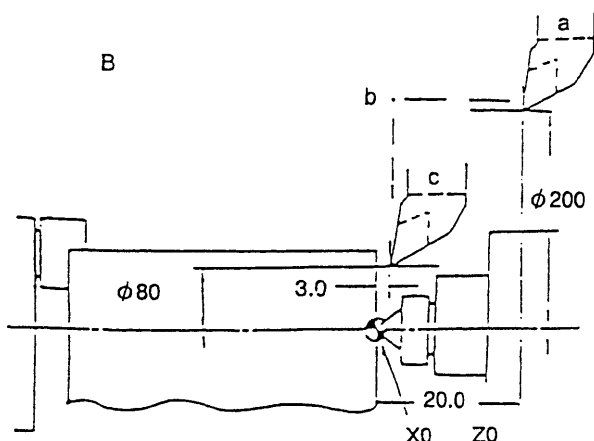
### Chuck work

1. Specify 2 axes simultaneously from the point "a" to the point "b".

G00 X100.0 Z0.2

2. From the point "b" to the point "a"

G00 X200.0 Z200.0



### Center work

Specify one axis when there is any interference.

3. From the point "a" to the point "b"

G00 Z3.0

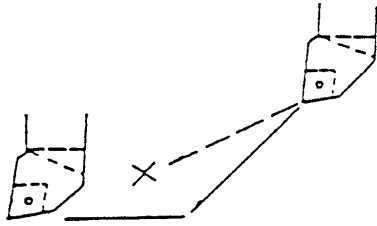
From the point "d" to the point "c" X80.0

4. From the point "c" to the point "b"

G00 X200.0

From the point "b" to the point "a" Z20.0

**Note:** When simultaneously positioning both the X and Z axes, the tool does not linearly move from a current position to a specified position, because their rapid traverse rates differ from each other. Therefore, you must be careful when there is an interfering substance halfway a tool path.



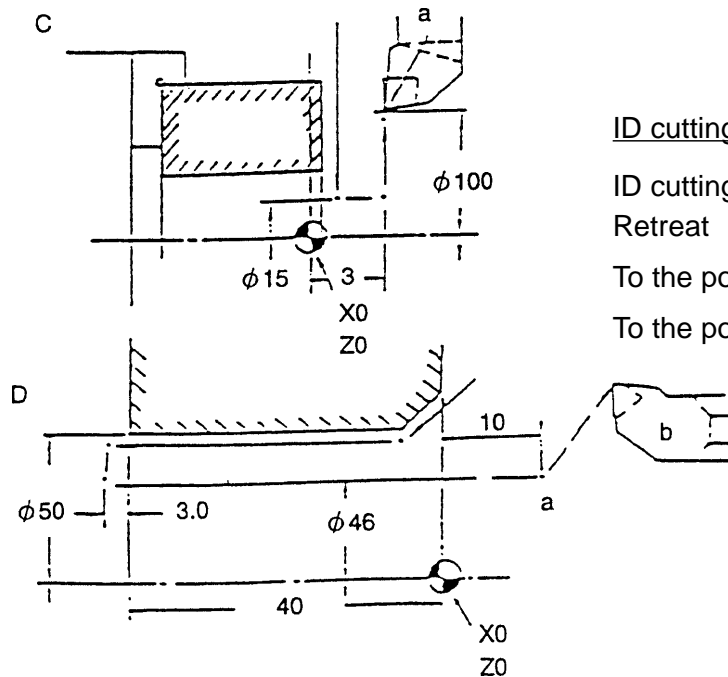
After one of 2 axes (X and Z) has completed its move, the other one moves to a specified point. The tool does not move linearly as shown with a dotted line in the left figure.

When moving to the next cutting position

When moving the tool to the next cutting position, do so at a rapid traverse rate after retreating it by about 2 to 3 mm from a cut surface.

End face OD cutting

|                  |                |
|------------------|----------------|
| End facing       | G01 X45.0 F0.3 |
| Retreat          | Z3.0           |
| To the point "a" | G00 X100.0     |
| Next command     | G01 Z □□□□     |



ID cutting

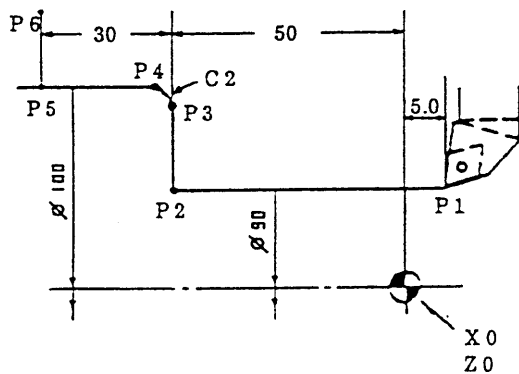
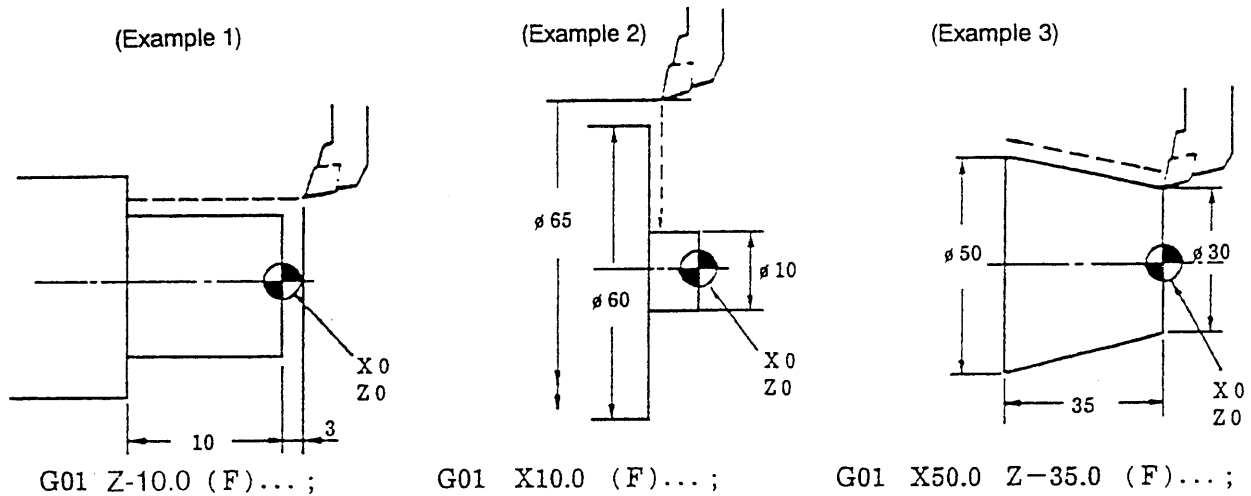
|                  |                 |
|------------------|-----------------|
| ID cutting       | G01 Z-43.0 F0.2 |
| Retreat          | X46.0           |
| To the point "a" | G00 Z10.0       |
| To the point "b" | X200.0 Z200.0   |

## 2-3-4 G01 Linear Cutting

(1) Specify this G code when performing linear cutting (ordinary cutting).

Chamfering and taper cutting are also considered linear cutting.

Use an F code to specify a feeding rate.



### Absolute programming

```
A P1 G00 X90.0 Z5.0
P2 G01 Z-50.0 F0.3
P3 X96.0
P4 X100.0 Z-52.0
P5 Z-80.0 F0.2
P6 G00 X□□□.□□
```

\* Absolute programming means to specify the position with X and Z coordinates from work coordinate zero point.

### \* Incremental programming

```
P1 G00 X90.0 Z5.0
P2 G01 W-55.0 F0.3
P3 U6.0
P4 X100.0 W-2.0
P5 W-28.0
P6 G00 X□□□.□□
```



This is also called an incremental value programming method. This method specifies tool strokes from a current position (start point) to the next point with U (X axis) and W (Z axis).

An end point of the previous block becomes a start point of the next block.

The end point of a previous block becomes the start point of the next block. X and W (or U and Z) can be used in the same block.

- Note:** 1) Be sure to specify an F function in the first G01 command in the program.  
2) Even if the G01 command is reset, the feed rate given with an F code is kept.

(2) Chamfering, corner R command

When there is chamfering (45° chamfering) or corner R (quarter circle) between 2 blocks which are parallel with the X or Z and cross with each other at a right angle, specify as follows:

For chamfering

(a) G01 X ... K ... F ...

(b) G01 Z ... I ... F ...

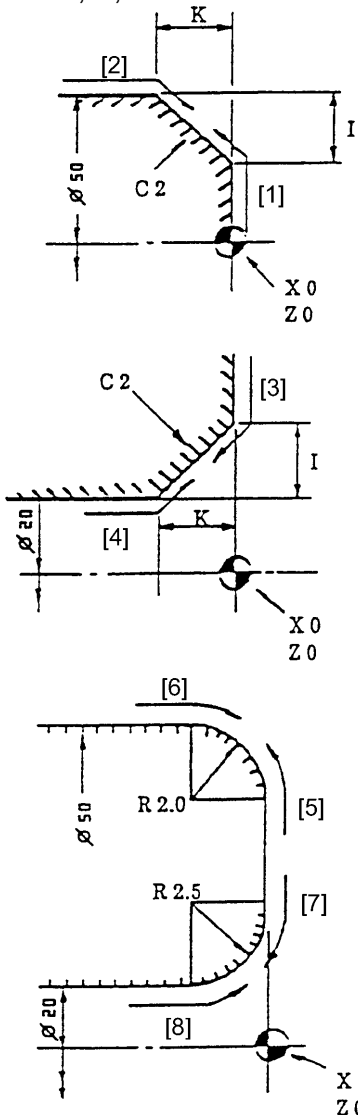
For corner R

(c) G01 X ... R ... F ...

(d) G01 Z ... R ... F ...

X and Z coordinate values .. Position after chamfering or corner R cutting

I, K, R ..... Radius designation. Signs “+” and “-” represent directions from the starting point, and a numerical value is a size of chamfering or corner R.



For the direction [1]

G01 X50.0 K-2.0

For the direction [2]

G01 Z0 I-2.0

For the direction [3]

G01 X20.0 K-2.0

For the direction [4]

G01 Z0 I2.0

For the direction [5]

G01 X50.0 R-2.0

For the direction [6]

G01 Z0 R-2.0

For the direction [7]

G01 X20.0 R-2.5

For the direction [8]

G01 Z0 R2.5

**Note:** 1) When specifying a tool movement with G01 for chamfering or corner R, it must be either one axis of X and Z.

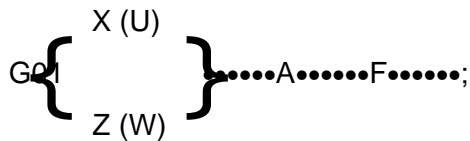
In the next block, the other one axis of X and Z, which crosses the former axis at a right angle, must be given.

2) The stop point by a single block operation is a point after chamfering or corner R cutting.

3) Specify I, K and R values for chamfering and corner R smaller than a specified axial amount.

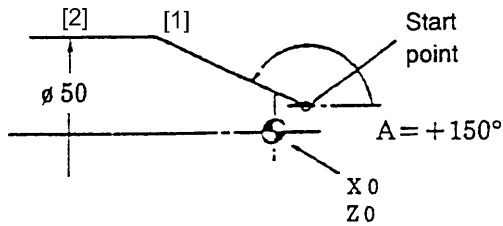
### (3) Angle designated linear interpolation

The angle designated linear interpolation can be performed by designating the angle A formed by the X or Z axes and +Z-axis.



The range of the angle is  $-360.0 \leq A \leq 360.0$  (deg).

CCW angle from +Z-axis is regarded as plus and the CW angle is as minus.

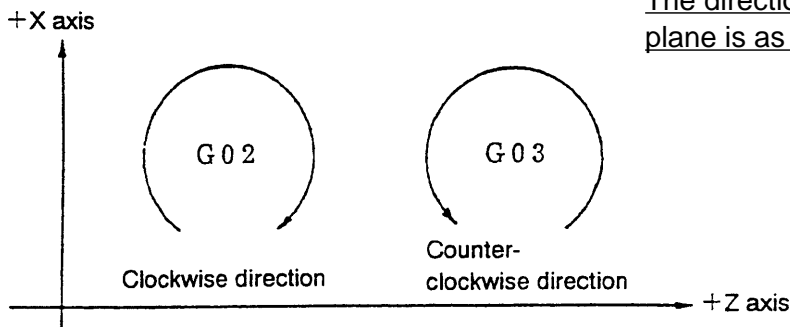


(1) G01 X50.0 A150.0 F0.3;

(2) G01 Z-100.0 A-180.0;

### 2-3-5 G02, G03 Circular Cutting

Specify either G02 or G03 when performing circular cutting.

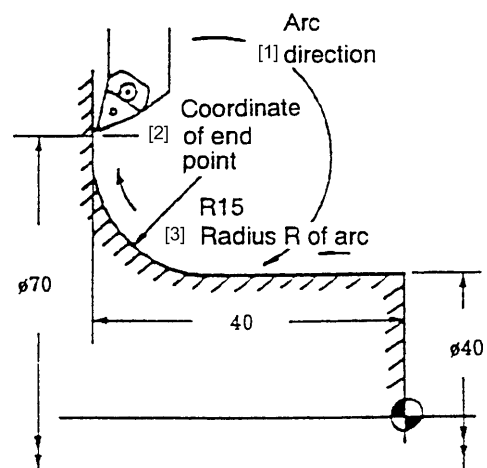
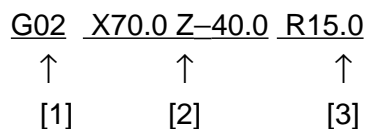


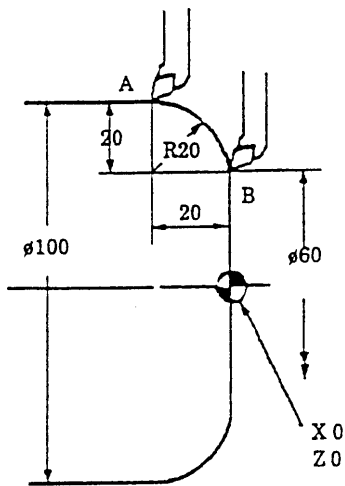
The direction of arc rotation on the X-Z plane is as illustrated below.

A circular command consists of the following 3 factors:

- [1] Circular arc direction G02 or G03
- [2] X and Z coordinate values of a circular arc end point
- [3] Circular arc radius R (radius designation)

Example : G01 Z-25.0

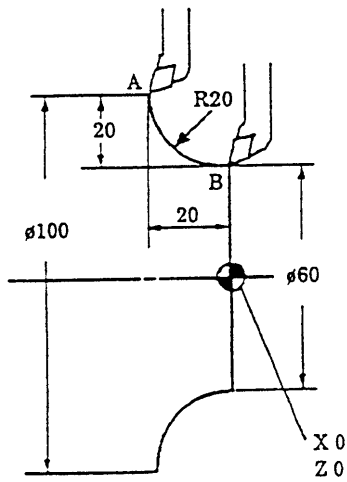




(Example 1)

When moving from the point A to the point B  
 $G02 X60.0 Z0 R20.0 F...$ ;

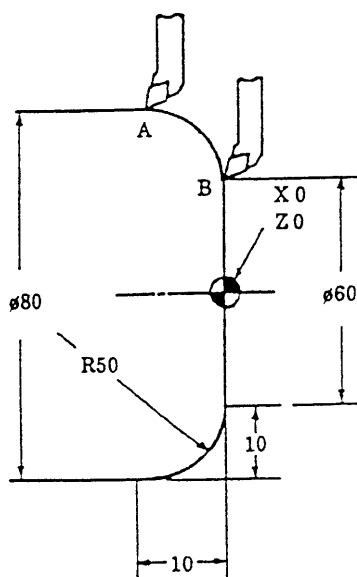
When moving from the point B to the point A  
 $G03 X100.0 Z-20.0 R20.0 F...$ ;



(Example 2)

When moving from the point A to the point B  
 $G03 X60.0 Z0 R20.0 F...$ ;

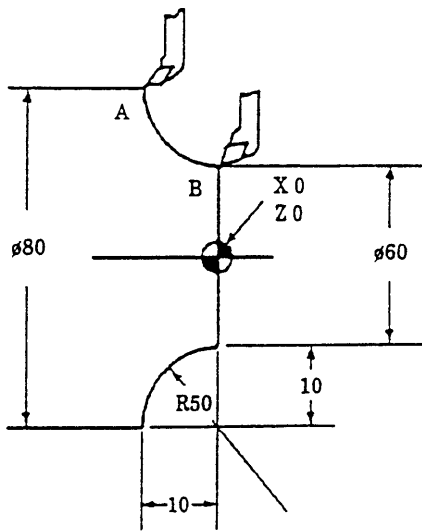
When moving from the point B to the point A  
 $G02 X100.0 Z-20.0 R20.0 F...$ ;



(Example 3)

When moving from the point A to the point B  
 $G02 X60.0 Z0 R50.0 F...$ ;

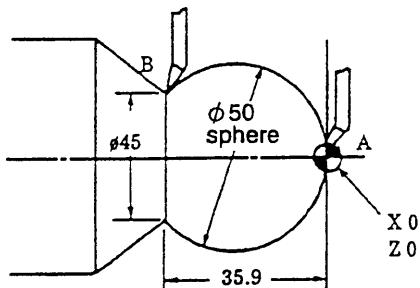
When moving from the point B to the point A  
 $G03 X80.0 Z-10.0 R50.0 F...$ ;



(Example 4)

When moving from the point A to the point B  
 G03 X60.0 Z0 R50.0 F...;

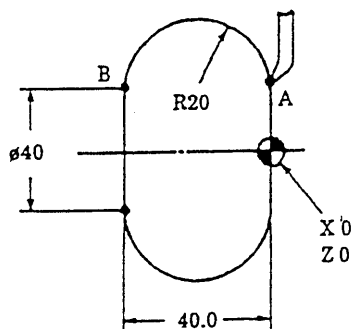
When moving from the point B to the point A  
 A02 X80.0 Z-10.0 R50.0 F...;



(Example 5)

When moving from the point A to the point B  
 G03 X45.0 Z-35.9 R25.0 F...;

When moving from the point B to the point A  
 G02 X0.0 Z0 R25.0 F...;



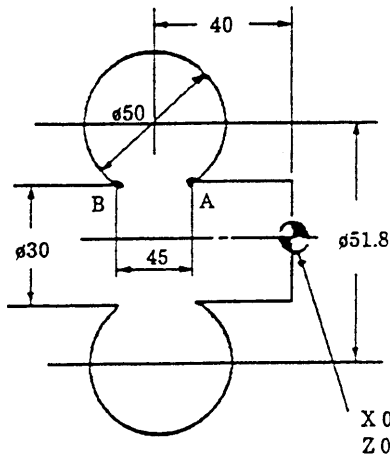
(Example 6)

When moving from the point A to the point B  
 G03 X40.0 Z-40.0 R20.0 F...;

When moving from the point B to the point A  
 G02 X40.0 Z0 R20.0 F...;

• Circular command exceeding 180°

When specifying a circular arc exceeding 180°, give a minus sign such as R-ΔΔ. ΔΔ.



When moving from the point A to the point B

G03 X30.0 Z-62.5 R-25.0 F...;

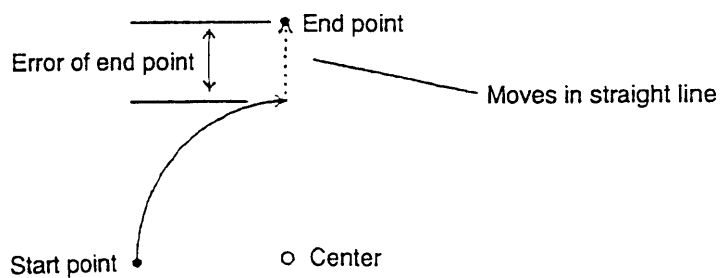
When moving from the point B to the point A

G02 X30.0 Z-17.5 R-25.0 F...;

Cutting feed rate

The cutting feed rate commanded by F code becomes the speed that a tool moves on a circular arc.

- Note:**
- 1) When F code has not feed commanded in G02 and G03 blocks or before that, an alarm will occur.
  - 2) Exponent type acceleration/deceleration is engaged.
  - 3) When radius of circular arc = 0 is commanded, an alarm will occur.
  - 4) If the end point is not located on the circular arc, the tool will move on the remainder in straight line after moving in circular, when an error of the end point of circular interpolation is within the parameter setting value. And when it is out of the parameter setting value, an alarm will occur.

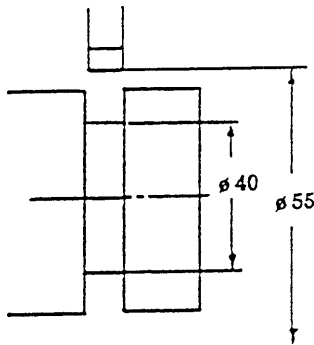


- 5) When I, J, K and R are commanded in the same block, R has priority.

### 2-3-6 G04 Dwell

A tool can be rested during a command time.

(Example)



When stopping the tool for 2 seconds

G04 U2.0;

In order to stabilize the diameter of the groove shown in the left figure, it is necessary to dwell the tool for 1 revolution or more at the bottom of the groove.

Assuming the spindle speed "N" to be 600 rpm, the time "T" required for 1 revolution is;

$$T = \frac{60}{N} = \frac{60}{600} = 0.1 \text{ second}$$

Therefore, stop the tool for 0.1 second or more.

G01 X40.0 F...;

G04 U0.2      In this case, the feed was made interrupted for 0.2 second.

X55.0 F...;

### 2-3-7 G09 Exact Stop

When G09 is commanded in the same block with a moving command, the machine is decelerated to stop and the next block is executed after checking that the position of the machine is within the range designated as a command position.

Only commanded block is effective.

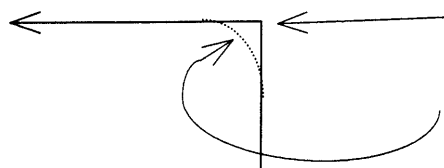
(1) Command form

G09 — ;

(2) Program example

N1 G09 G01 U50. F ;

N2 G01 W \_ 50. ;



When commanding G09, an edge is created on the corner.

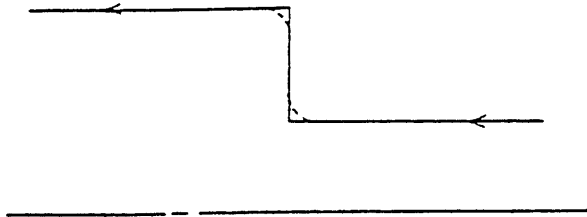
When not commanding G09, a round is created on the corner.

### 2-3-8 G61 Exact Stop

The machine is decelerated to stop at the end point until G62, G63 and G64 etc. are commanded after commanding G61, and the next block is executed after checking that the position of the machine is within the range commanded.

Program example

```
G61 G01 Z-100.0 F0.2
      X20.0
      Z-150.0
```



G61 mode effective :

An edge is created on the corner.

G61 mode ineffective :

A round is created on the corner.

### 2-3-9 G10 Programmable Data Input

It is possible to change various data for work shift and offset on the N/C program.

(1) Work shift amount input

```
G10 P00 X (U) Z_ (W)_ ;
```

P00 : Work shift amount input designation

X (U) : Work shift amount of X-axis

Z (W) : Work shift amount of Z-axis

Generally, setting of an offset amount is performed for only the value of Z.

Don't perform work shift for other axes.

(Example) G10 P00 Z512.368;

(2) Form offset amount input

```
G10 L10 P_ X (U)_ Z (W)_ R_ Q_ H_ ;
```

L10 : Form offset input designation

P : Offset No. (0 ~ Maximum offset sets)

X (U) : Form offset amount of X-axis

Z (W) : Form offset amount of Z-axis

R : Tool nose R (Absolute)

Q : Virtual tool nose point (0 ~ 9)

H : Tool width (Absolute)

(3) Wear offset amount input

G10 L11 P\_ X (U)\_ Z (W)\_ R\_ H \_ ;

- L11 : Wear offset amount input designation
- P : Offset No. (0 ~ Maximum offset sets)
- X (U) : Wear offset amount of X-axis
- Z (W): Wear offset amount of Z-axis
- R : Tool nose R (Absolute)
- H : Tool width (Absolute)

**Note** 1) *Only when absolute input is performed by the form offset input, the wearoffset amount of the address input is cleared to 0.*

2) *R (Tool nose R) and H (Tool width) are performed only by absolute input.*

### 2-3-10 G20, G21 Inch Input/Metric Input

It is possible to select the input unit of a program command either in inch input or in metric input by G20 or G21 command.

Command form

G20 ; Input unit is inch input

G21 ; Input unit is metric input

The following units are changed by the G20/G21 command.

- (a) Feed rate command by F (E is included for thread cutting).
- (b) Commands related to positions.
- (c) Work reference point shift amount.
- (d) Tool offset amount.
- (e) A part of parameters.
- (f) The unit of one graduation of the manual pulse generator.

- (1) The G20/G21 command shall be commanded to the head of the program in the single block.
- (2) When the G20/G21 command is executed, conduct the coordinate system preset.
- (3) This function is for selecting the unit of numerical value programmed either in metric or in inch.  
Inch ⇔ Metric conversion isn't performed.

## 2-3-11 G22, G23 Stored Stroke Limit

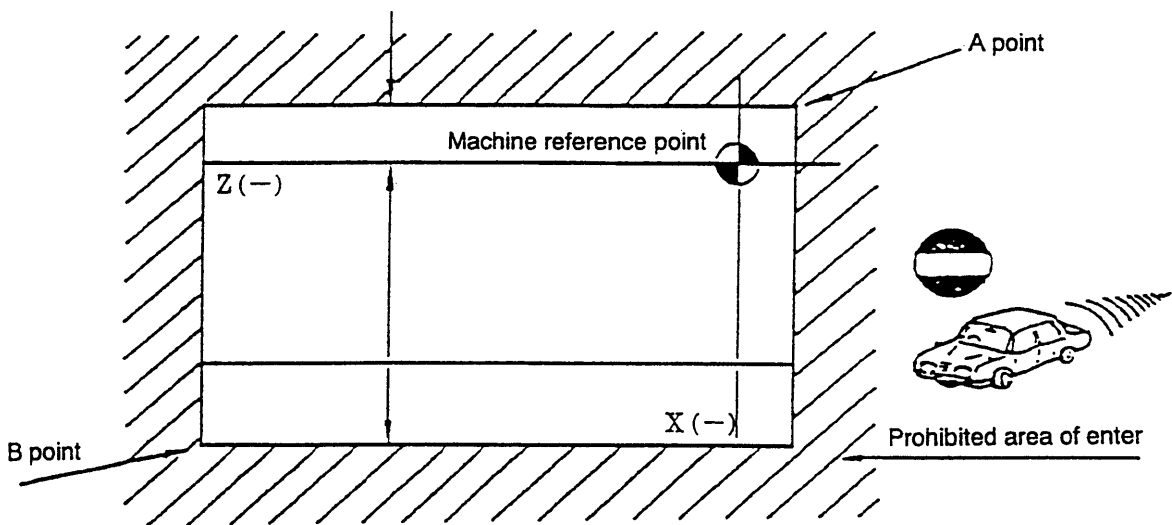
This machine is provided the stored stroke limit, which can be set the entering prohibition of tool in the movable area (Within the machine stroke) of the machine for safety operation by whether automatic or manual operation, as standard feature.

This function is different from the mechanical stroke end and there are following three kinds.

### 1. The first prohibited area

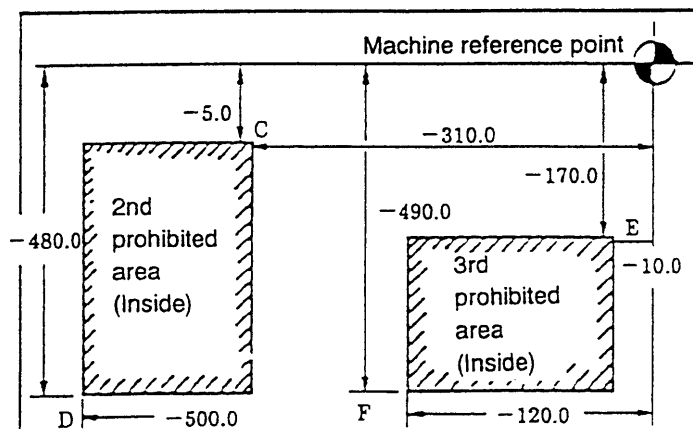
This is set the maximum stroke of the machine by the parameter and not changeable.

Points A and B are set by the distance from the machine reference point by the parameter and the hatching area is prohibited entering always.



### 2. The 2nd, 3rd prohibited area

Set the second and third stroke limit at any places without restraint by commanding a distance and direction from the machine reference point. The inside or the outside can be selected in the second stroke limit. Only the inside becomes effective in the third stroke limit.



(1) Selection of prohibited area

A prohibited area can be selected by the parameter No.1300 to close which side of in or outside of a frame determined by the points C, D and E, F.

Usually, the inside becomes the prohibited area in the second area. In the third area, always the inside is prohibited.

|  |              |  |
|--|--------------|--|
| No.1300 - bit 0<br>(First bit from<br>right) | In case of 0 | Inside of stroke limit 2 is a prohibited area  |
|  | In case of 1 | Outside of stroke limit 2 is a prohibited area |

(2) Setting of area by parameter and confirmation

| Prohibit area | No.          | Setting position | Setting example |
|---------------|--------------|------------------|-----------------|
| Second area   | 1322         | X of point C     | -5.000          |
|               | •            | Z of point C     | -310.000        |
|               | •            |                  | 0.000           |
|               | •            |                  |                 |
|               | •            |                  |                 |
|               | 1323         | X of point D     | -480.000        |
| •             | Z of point D | -500.000         |                 |
| •             |              | 0.000            |                 |
| •             |              |                  |                 |
| Third area    | 1324         | X of point E     | -170.000        |
|               | •            | Z of point E     | -10.000         |
|               | •            |                  | 0.000           |
|               | •            |                  |                 |
|               | 1325         | X of point F     | -490.000        |
|               | •            | Z of point F     | -120.000        |
| •             |              | 0.000            |                 |
| •             |              |                  |                 |

**Note)** Setting units 0.001mm. Decimal point input is not available.  
Value X is a diametrical value.

(3) Setting of the second or third stroke limit by MDI or program

Example:

G22 X-170.0 Z-10.0 I-490.0 K-120.0 (Refer to the sketch on the previous page.)

Command of entering prohibition into the second stroke limit and the second or third stroke limit is set.

Example:

G23; Entering is possible into the second area.

**Note** 1) *When G23 has commanded, G22 should be commanded in the individual block to make a setting area entering prohibition again.*

2) *When G22 X\_ Z\_ I\_ K\_ ; is commanded, the parameter changes automatically to the commanded value.*

3) *The last commanded G22 and G23 is kept even if the power off.*

4) *If G22 has commanded at the time of power on, entering to the zero becomes prohibition immediately after execution of manual reference point return.*

5) *If entering to the prohibited area manually, it can be escaped from the prohibited area by moving the opposite direction.*

*The reset button should be pressed after an escape.*

6) *During automatic operation, if the end point of travel locates in the prohibited area, it becomes an alarm before move and stop the automatic operation. It can be released by the reset button.*

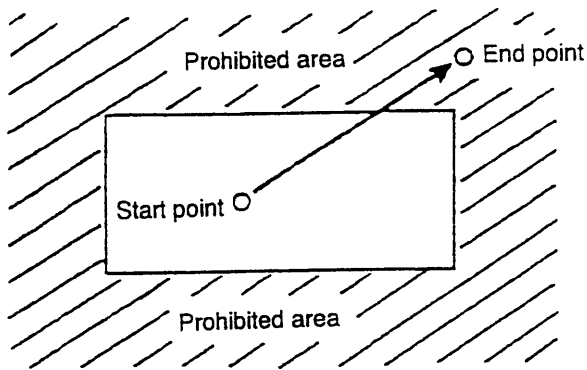
*If the middle of the way to the end point to travel is the prohibited area, it becomes an alarm when entering into the prohibited area and stop the automatic operation.*

(4) Setting in other than program command

Refer to Operation Manual.

### 2-3-12 Stroke Limit Check Before Move

If the end point of the block to be executed the automatic operation locates in the prohibited area, stop the axis travel and make an alarm. Execute a check regarding all effective matters by the stroke limit 1, 2 and 3.



Interrupt a travel if the end point of executing block locates in the prohibited area.

- (1) If the alarm of stroke limit before move is issued, release the alarm by pressing the reset button.
- (2) The end point of executing block can be calculated by the "Machine coordinate" + "Remaining amount of travel" at this time.  
**Note)** *If the travel time of one block is very short, some times becomes an alarm before setting the remaining amount of travel.*
- (3) Precautions
  - (a) Concerning a traveling path of block, a check is not executed.
  - (b) Concerning an axis which is machine lock condition, a check is not executed.
  - (c) Checking of a block of G31 is not executed.
  - (d) Check the axis which has completed the reference point return only.
  - (e) If the end point locates very close to the prohibited area, it becomes an alarm occasionally.

### 2-3-13 G27 Reference Point Return Check

The G27 command positions to the designated position by a program then check the position whether it is the first reference point or not and it becomes alarm if it is not.

(1) Form of command

G27 X\_ Z\_ ..... ;

(2) Program example

G27 X100.0 Z-50.0 ;

Move to the X-axis 100.0 and Z-axis -50.0 then check the X and Z axes return to the first reference point or not.

(3) Precautions

(a) An arriving position by G27 command include tool position offset, nose R compensation and tool radius compensation etc.

(b) Check the axis which has commanded by G27 block only.

(c) The 2nd ~ 4th reference point return check can be done by commanding a P.

G27 P\_ X\_ Z\_ ..... ;

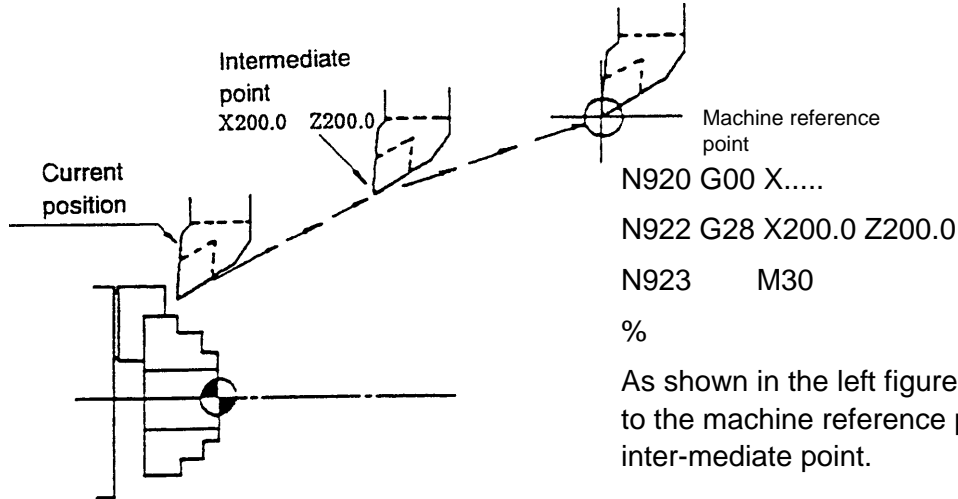
(Designate it according to the 2nd ~ 4th reference point. )

(d) An axis which is the condition of machine lock on does not execute the reference point return check.

(e) G27 command checks an imposition after positioning.

## 2-3-14 G28 Automatic Reference Point Return

With a command “G28 X (U)□□□. □□ Z (W)□□□. □□”, the tool automatically returns to the machine reference point after moving to the position (intermediate point) specified with X (U), Z (W). G28 assumes the same rapid traverse rate as G00. After returning to the machine reference point, the machine reference point lamp lights up.



As shown in the left figure, the tool returns to the machine reference point via the inter-mediate point.

**Note)** Difference from “G28 U0 W0 ;”

Since U0 W0, which is incremental programming, means that a tool stroke is 0(zero), the current position becomes the intermediate point as it is, and the tool returns to the machine reference point from that position.

## 2-3-15 G30 2nd Reference Point Return

(1) A commanded axis can be returned the 2nd reference point automatically.

The 2nd reference point can be set either setting by the parameter for the distance from the machine zero point previously or process of the 2nd reference point setting.

Refer to the instruction manual for the process of the 2nd reference point setting.

Exactly same motion as the automatic zero return by G28 is executed except returning to the 2nd reference point by the parameter setting.

Program example

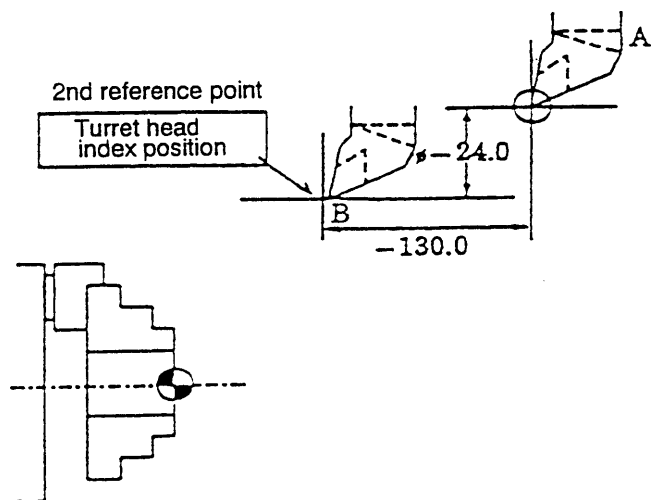
```

O5801
N1 G28 U0
N2 G28 W0 T0100
N3 G50 S1500
N4 G30 U0 W0(G00 X200:0 Z150.0)
N5     M01

N101 G30 U0 W0
N102     T0100
N103 G97 S530 M08
N104 G00 X72.0 Z10.0 M03
N105 G01 G96 Z0.2 F3.0 S120
N106     X0 F0.2
N107     Z3.0
N*     ..
N*     ..
N*     ..
N118 G00 G97 X70.0 S545
N119 G30 U0 W0 (G00 X200.0 Z150.0)
N120     M01
..
..
..
N1001 G30 U0 W0
N1002     T1000
N1003 G97 S695 M08
N1004 G00 X30.0 Z15.0 M03
N1005 G01 G96 Z7.0 F3.5 S150
N*
N*     ..
N*     ..
N1013 G00 G97 Z15.0 S.....
N1014 G30 U0 W0 (G00 X200.0 Z150.0)
N1015     M01
N6 G28 U0 W0 T0100
N7     M30
    
```

A program example at left uses the 2nd reference point (G30) as the turret index position.

A setting of the 2nd reference point execute on the 2nd reference point setting screen after the turret with maximum protruded tool is moved the position (B point) which is not interfered position with a machining workpiece or the chuck etc.



**Caution**

If the 2nd reference point is used correctly, it makes the safest program. However, when the turret head index position (2nd reference point) is altered due to a process change or preparatory plan change, set the second reference point again each time.

**Note** 1) Before specifying G30, perform automatic reference point return at least once by manual reference point return operation or a G28 command after turning on the power.

(2) The third, fourth reference point

G30 Pn X (U) ... Z (W) ...;

(Pn=P2, P3, P4)

Execute a positioning of the second, third or fourth reference point after positioning at intermediate point commanding by the above command.

{ P2 : The second reference point  
 P3 : The third reference point  
 P4 : The fourth reference point

If Pn is omitted, it becomes the second reference point.

**Caution**

Be careful, if the coordinate command of an axis is omitted, the axis does not move.

(3) Position of each reference point

The position of each reference point is set previously by

the parameter  $\left\{ \begin{array}{l} \text{P2 : No.1226~} \\ \text{P3 : No.1227~} \\ \text{P4 : No.1228~} \end{array} \right\}$  is a distance from the machine reference point.

<Program example>

`G30 P3 U~40. W30. ;` X and Z axes return to the third reference point.

## 2-3-16 G31 Skip Function

If the skip signal is entered from the outside while linear interpolation is executed by G31 command, the travel is stopped, the remaining travel amount is left and the next block is proceeded.

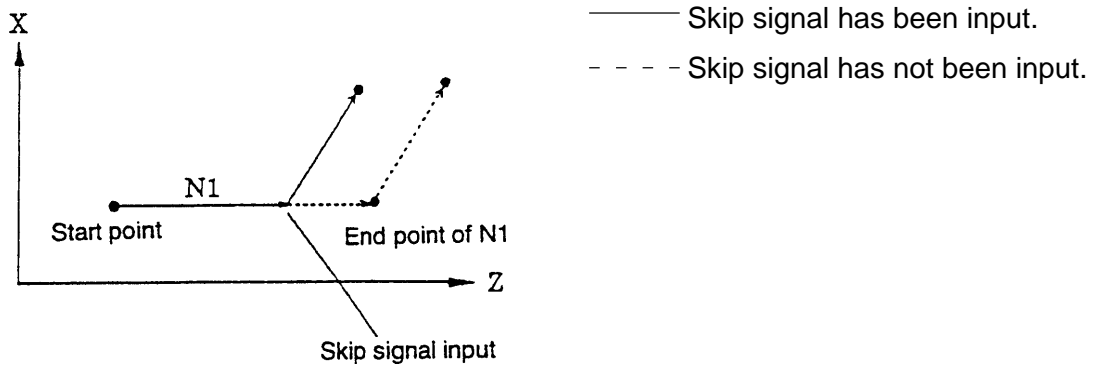
(1) Form of command

G31 X\_ Z\_ . . . . F\_ ;

(2) Program example

N1 G98 G31 W50. F100 ;

N2 G01 U50. W25. ;



(3) Caution

- (a) The federate set in the parameter can be obtained regardless F of the program by parameter setting.
- (b) G31 cannot be commanded during nose R compensating mode.
- (c) When the next block of G31 is commanded by an increment, the next block moves incrementally from the interrupted position by a skip signal.
- (d) G31 is effective only for the commanded block.

## 2-3-17 G54 Work Coordinate System Setting (Work Length)

Work length shall be set as the value following address Z by the command

G54 Z\_\_\_\_\_

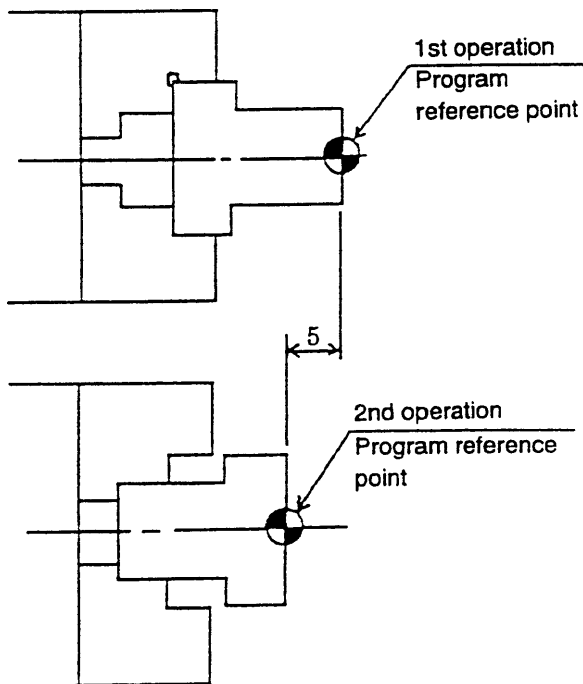
Correct distance is displayed of the tool position from the machine origin by following procedures.

1. When tool is indexed by T code in program (available by MDI as well).
2. When rotate the turret by pushing the turret index button while manual mode.
3. When applied Q setter or Z setter (Option).

An incremental amount of the work length can be designated by G54 W\_ command.

This function is used for the case when 1st and 2nd operations are continuously machined.

(Example)



```

Oxxxx
G28 U0
G28 W0
G54 Z0 ..... Reference point shift amount cancel
G50 S2000
G30 U0 W0
M01

N100
T0100
G97 S1230 M08
    } 1st operation machining
G28 U0
G28 W0
M00..... Work turn-over

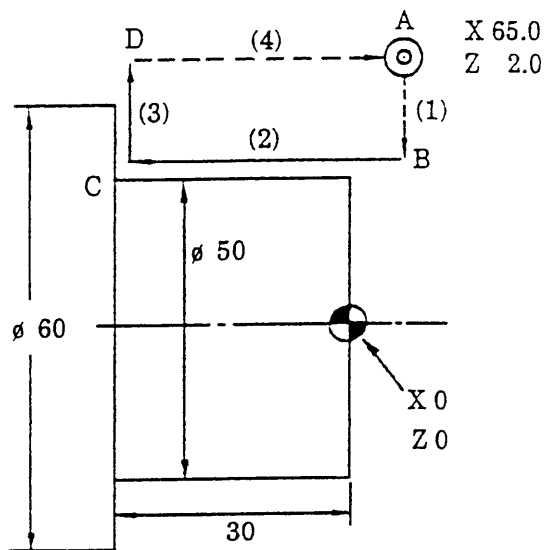
G28 U0
G28 W0
G54 Z-5.0 .... Difference from the finishing end of
                the work at 1st operation

G50 S1500
G30 U0 W0
M01

N150
T0100
G97 S730 M08
    } 2nd operation machining
G54 Z0 ..... Reference point shift amount cancel
G28 U0
G28 W0
M30
    
```

## 2-3-18 Canned Cycle

Using a canned cycle, machine functioning equivalent to 4 blocks of “cutting-in → cutting (or threading) → retreat → return” in a regular program can be specified as 1 cycle in 1 block.

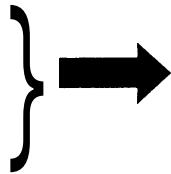


The tool starts from the point A (X65.0, Z2.0) and returns to the point A via the points B, C and D respectively.

If the canned cycle is used, the program will be changed as follows:

### Regular Program

- (1) G00 X50.0;
- (2) G01 Z-30.0 F;
- (3) X65.0;
- (4) G00 Z2.0;



### Program with Canned Cycle

G90 X50.0 Z-30.0 F;

The machine works in the same manner as in the 4-block program on the left.

Accordingly, when a large cutting allowance is required, or when the number of blocks is many as in a threading program, the canned cycle is useful because it can simplify the program. There are the following 3 kinds of canned cycles available:

1. G90 ..... OD/ID cutting cycle
2. G92 ..... Threading cycle
3. G94 ..... End face/side cutting cycle

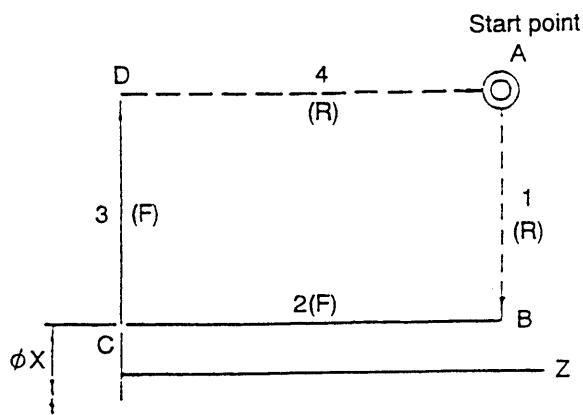
### 1. G90 OD/ID cutting cycle

G90 enables OD/ID straight cutting or taper cutting.

The tool moves via a specified point from its start point, cuts the workpiece at a feed rate specified with an F code and returns to the start point again.

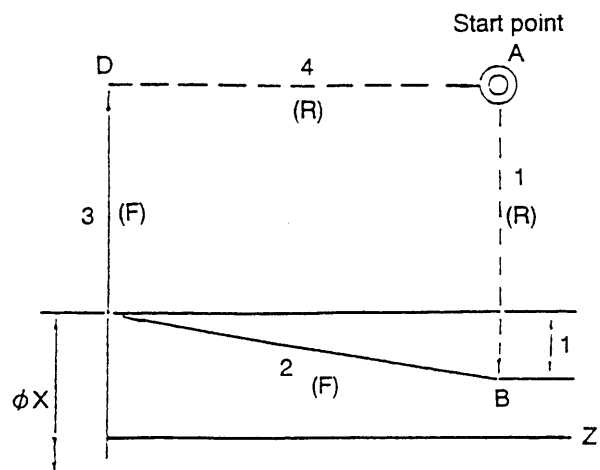
## G90 cycle patterns

### (1) Straight cutting



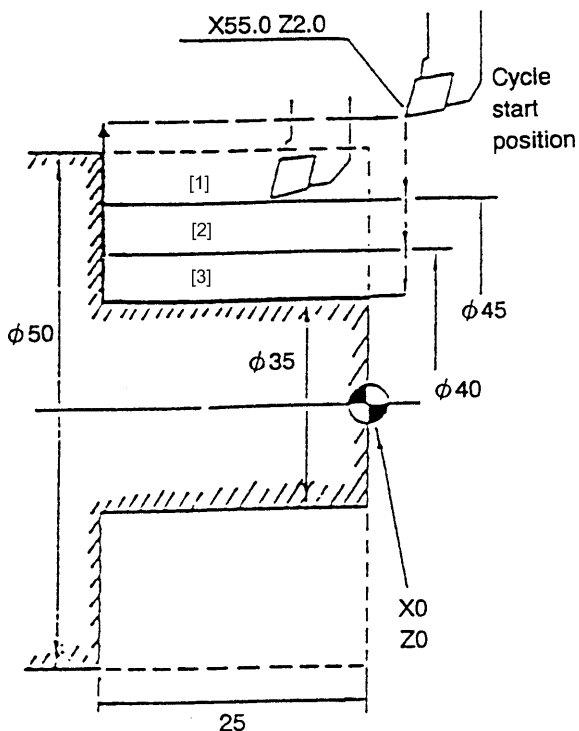
G90 X...Z...F...; (I=0)  
 R : Rapid traverse  
 F : Cutting feed  
 (specified with an F code)

### (2) Taper cutting



G90 X...Z...I...F...;  
 (Pay attention to a sign of I.)

### 1. Example of straight cutting

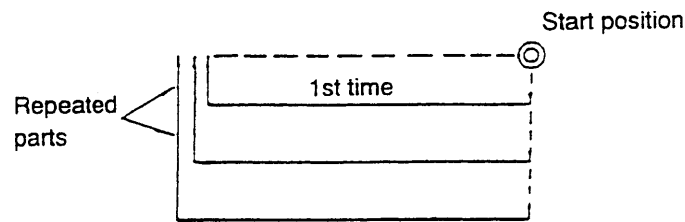


When machining a  $\phi 50$  blank as shown in the left figure, with its start point at X55.0 and Z2.0 and a depth of cut of 2.5 mm, the program is as follows:

```
N101 T0100 M40
N102 G97 S695 M08
N103 G00 X55.0 Z10.0 M03
N104 G01 G96 Z2.0 F2.5 S120
N105 G90 X45.0 Z-25.0 F0.35-[1]
N106 X40.0 [2]
N107 X35.0 [3]
N108 G00 G97 X200.0 Z200.0 S695
N109 M01
```

- Note**
- 1) As G90 is modal, once it is specified, it can be neglected from the next block. Accordingly, cycle operation is executed by only specifying the cutting depth of X-axis from the next block on.
  - 2) After completing the canned cycle, cancel G90 with another G code, such as G00 belonging to the same group.
  - 3) For the T, S and M functions which serve as cutting conditions, be sure to specify them in a block preceding the one where G90 is to be specified.

In the above-mentioned program, the tool returns to the same start point after completing each cycle. At that time, a machining time is wasted because the same parts are repeatedly machined in side cutting as shown in the figure below. Therefore, the machining time can be saved by shifting the cycle start position per cycle as shown in the program below, after completing each cycle.



```

N105 G90 X45.0 Z-25.0 F0.35 [1]
N106 G00 X47.0
N107 G90 X40.0 Z-25.0 [2]
N108 G00 X42.0
N109 G90 X35.0 Z-25.0 [3]
N110 G00 .....

```

Since the start position is shifted by G00 after completing the canned cycle, it is canceled each time.

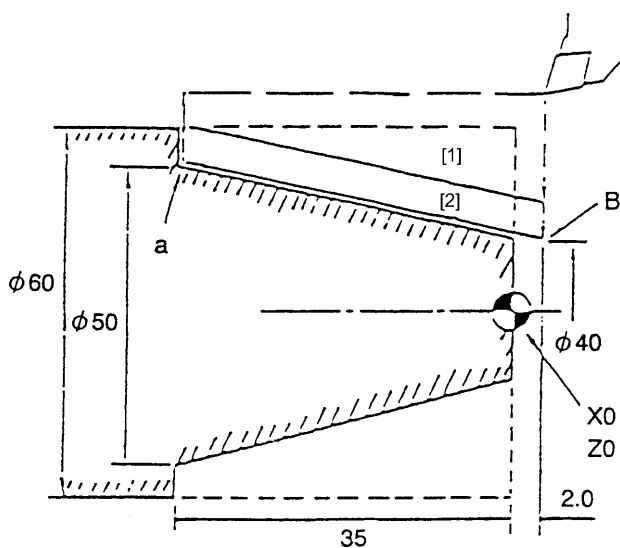
Therefore, you must specify a G90 command and coordinate values each time.

## 2. Example of taper cutting

When machining a  $\phi 60$  blank as shown in the figure below, with the cycle start position at X65.0 and Z2.0 and a depth of cut of 5 mm, the program is as follows:

First, obtain an amount of I.  $I = \frac{50-40}{2} = 5 \text{ mm}$

The sign of I(“+” or “-”) is determined as a direction from point “a” to the point B. Accordingly;  $I = -5.0$

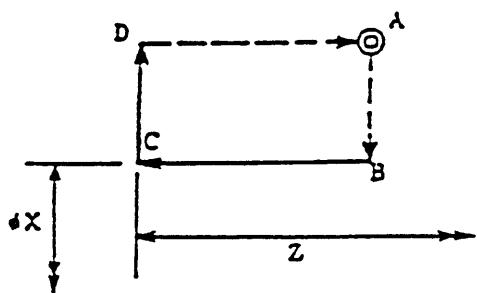
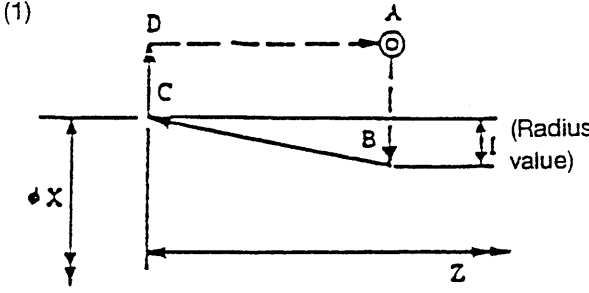
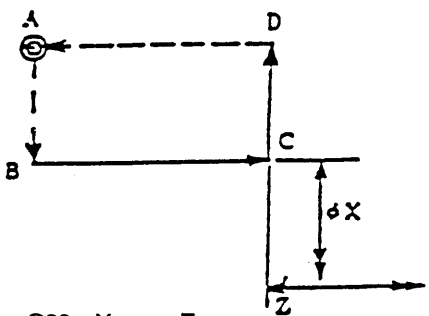
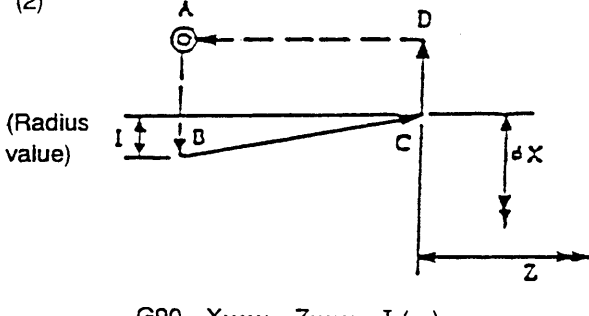
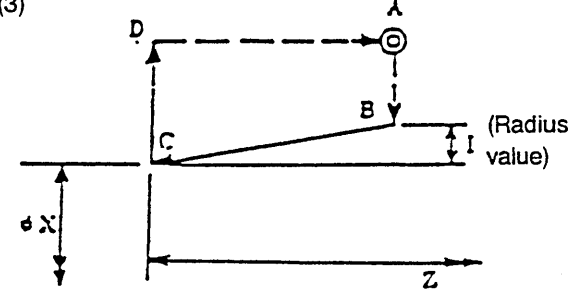
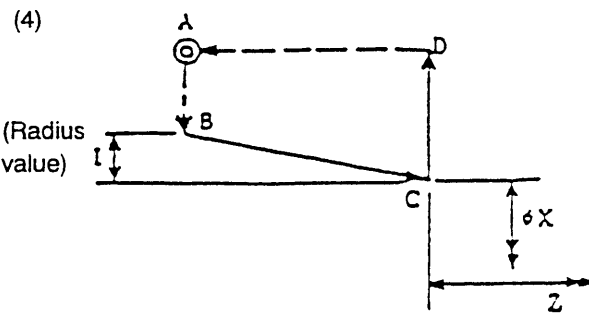


```

N104 G01 G96 X65.0 Z2.0 S120
N105 G90 X60.0 Z-35.0 I-5.0 F0.3 → [1]
N106 X50.0 → [2]
N107 G00 X..... Z.....
or
N104 G01 G96 X65.0 Z2.0 S120
N105 G90 X60.0 Z-35.0 I-5.0 F0.3 → [1]
N106 G00 X55.0
N107 G90 X50.0 Z-35.0 I-5.0 → [2]
N108 G00 X..... Z.....

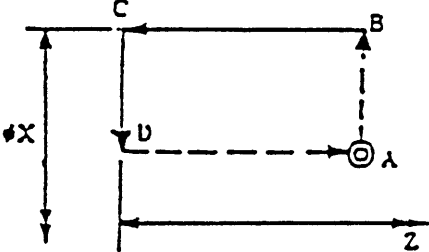
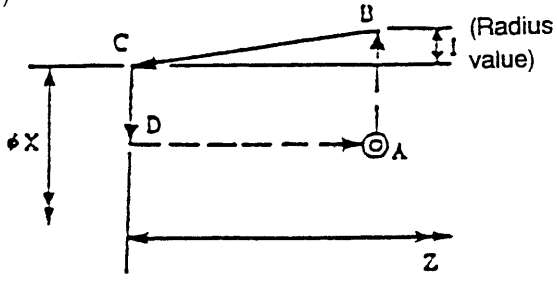
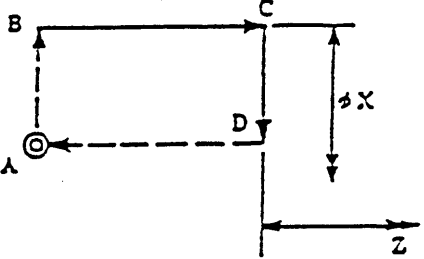
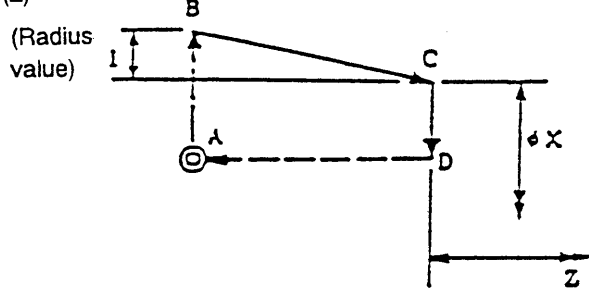
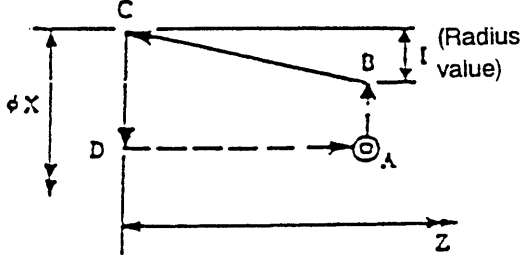
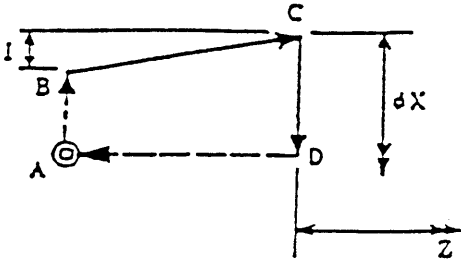
```

### G90 Cycle Patterns (OD)

| Straight  | Taper   |
|---|---|
|  <p style="text-align: center;">G90 X..... Z.....</p>  | <p>(1)</p>  <p style="text-align: center;">G90 X..... Z..... I (-) .....</p>  |
|  <p style="text-align: center;">G90 X..... Z.....</p> | <p>(2)</p>  <p style="text-align: center;">G90 X..... Z..... I (-) .....</p> |
|   | <p>(3)</p>  <p style="text-align: center;">G90 X..... Z..... I.....</p>     |
|   | <p>(4)</p>  <p style="text-align: center;">G90 X..... Z..... I</p>          |

The sign (+, -) of I is determined as a direction viewing the point B from the point C.  
 For a cutting diameter, specify a dimension at the point C.

### G90 Cycle Patterns (ID)

| Straight   | Taper   |
|--|---|
| <p>(1)</p>  <p style="text-align: center;">G90 X..... Z.....</p>  | <p>(1)</p>  <p style="text-align: center;">G90 X..... Z..... I.....</p>       |
| <p>(2)</p>  <p style="text-align: center;">G90 X..... Z.....</p> | <p>(2)</p>  <p style="text-align: center;">G90 X..... Z..... I.....</p>      |
|  | <p>(3)</p>  <p style="text-align: center;">G90 X..... Z..... I(-) .....</p> |
|  | <p>(4)</p>  <p style="text-align: center;">G90 X..... Z..... I(-) .....</p> |

The sign (+, -) of I is determined as a direction viewing the point B from the point C.  
 For cutting diameter, specify a dimension at the point C.

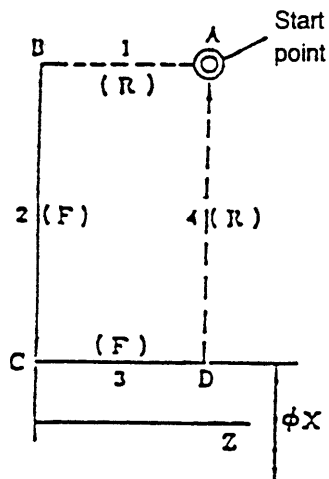
## 2. G94 End face and side cutting cycle

G94 enables straight/taper cutting of the end face and side.

The tool moves via a specified point from its start point, cuts the workpiece at a feed rate specified with an F code and returns to the start point.

### G94 cycle patterns

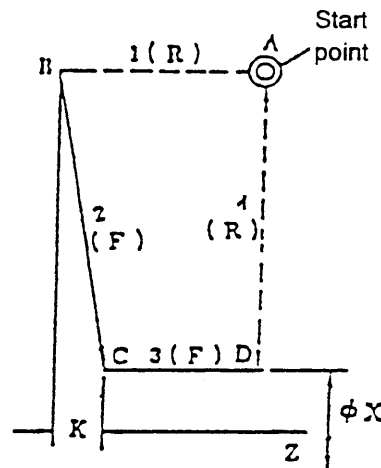
(1) Straight cutting



G94 X...Z...F...  
(K=0)

R: Rapid traverse  
F: Cutting feed  
(specified with  
an F code)

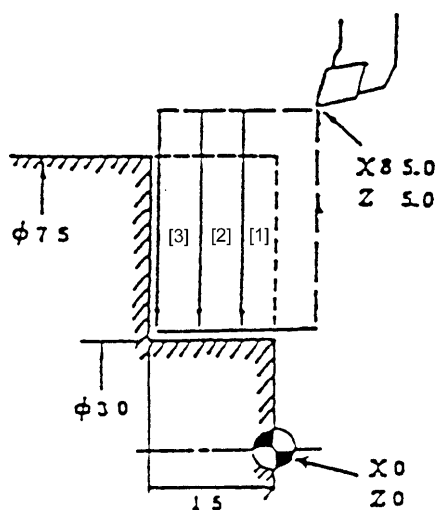
(2) Taper cutting



G94 X...Z...K...F...

(Pay attention to a sign of K)

### 1. Example of straight cutting

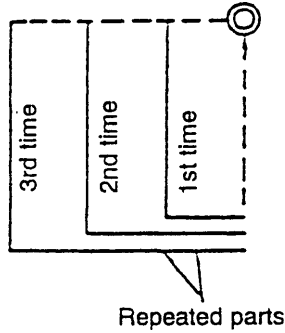


When machining a  $\phi 75$  blank as shown in the left figure, with its cycle start position at X85.0 and Z5.0 and a depth of cut of 5 mm, the program is as follows:

```

N101 T0100 M40
N102 G97 S450 M08
N103 G00 X85.0 Z10.0 M03
N104 G01 G96 Z5.0 F3.0 S120
N105 G94 X30.0 Z-5.0 F0.2 ..... [1]
N106 Z-10.0 ..... [2]
N107 Z-15.0 ..... [3]
N108 G00 G97 X200.0 Z200.0 S150
N109 M01
    
```

- Note** 1) Since G94 is modal, specify it just once. You do not have to specify it again thereafter. Accordingly, cycle operation is executed by only giving Z-axis depth of cut from the next block on.
- 2) After completing the canned cycle, cancel G94 with another G code, such as G00, belonging to the same group.
- 3) For the T, S and M functions which serve as cutting conditions, be sure to specify them in a block preceding the one where G94 is to be specified.



In the above-mentioned program, the tool returns to the same start point after completing each cycle. At that time, a machining time is wasted because the same parts are repeatedly machined in OD cutting as shown in the left figure.

Therefore, the machining time can be saved by shifting the cycle start position per cycle as shown in the program below.

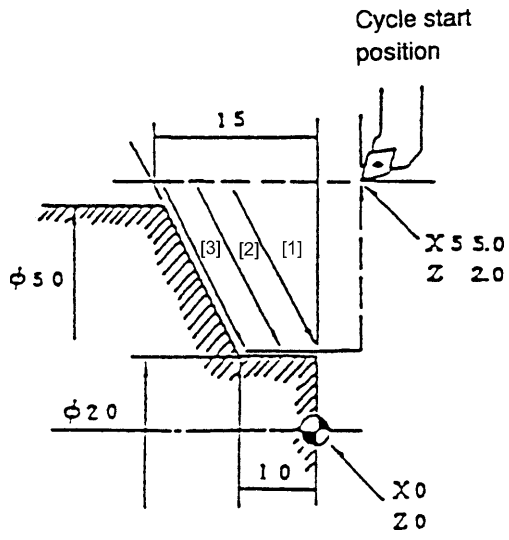
```

N105 G94 X30.0 Z-5.0 F0.2 ..... [1]
N106 G00 Z-3.0
N107 G94 X30.0 Z-10.0 ..... [2]
N108 G00 Z-8.0
N109 G94 X30.0 Z-15.0 ..... [3]
N110 G00 X.... Z....

```

Since the start position is shifted by G00 after completing the canned cycle, it is canceled each time. Therefore, you must specify a G94 command and coordinate values each time.

**2. Example of taper cutting**



When machining a  $\phi 50$  blank as shown in the left figure, with its cycle start position at X55.0 and Z2.0 and a depth of cut of 5 mm, the program is as follows:  
First, obtain a size of K.

$$K = 15 - 10 = 5$$

Determine a sign of K, viewing its cycle pattern.

Accordingly; K-5.0

```

N104 G01 G96 X55.0 Z2.0 S120
N105 G94 X20.0 Z0 K-5.0 F0.2 ..... [1]
N106 Z-5.0 ..... [2]
N107 Z-10.0 ..... [3]
N108 G00 X.... Z....

```

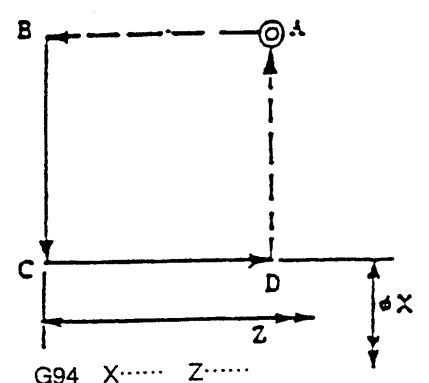
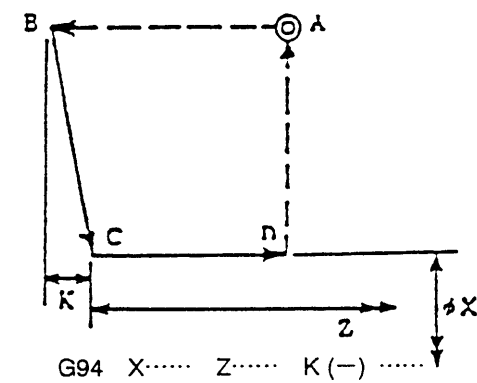
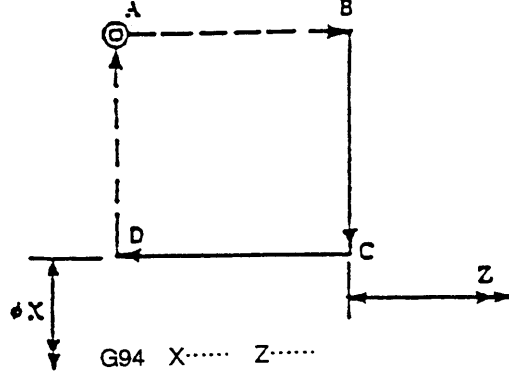
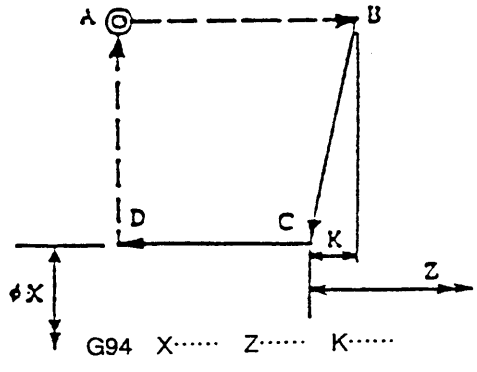
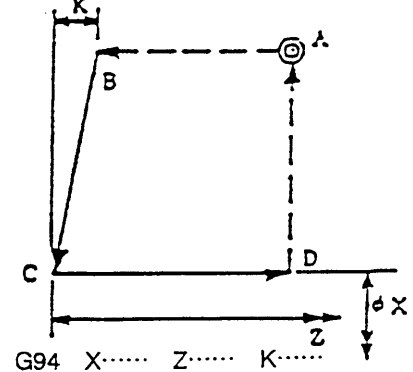
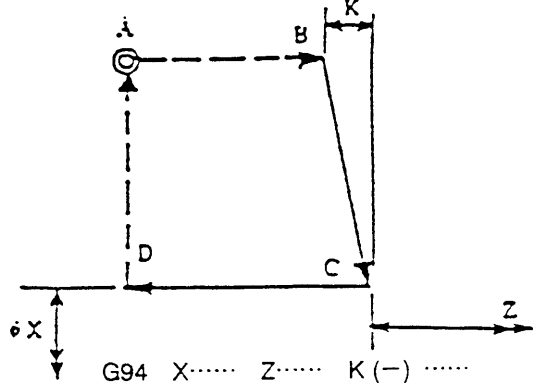
or

```

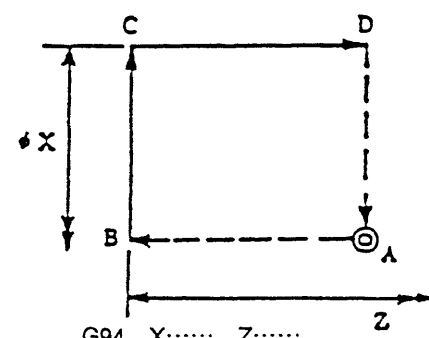
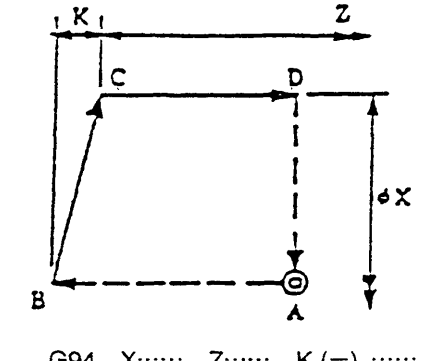
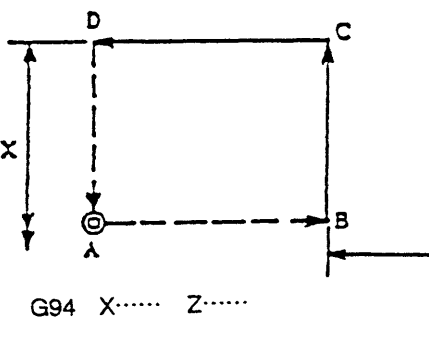
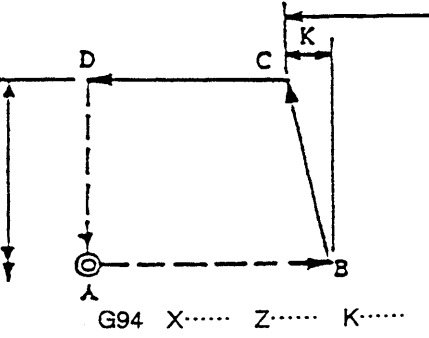
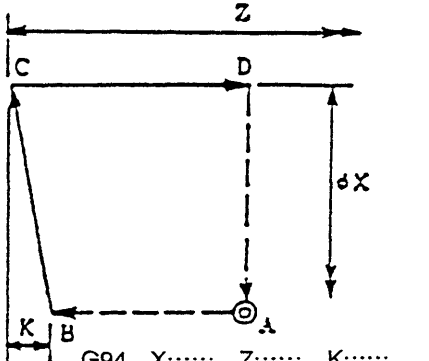
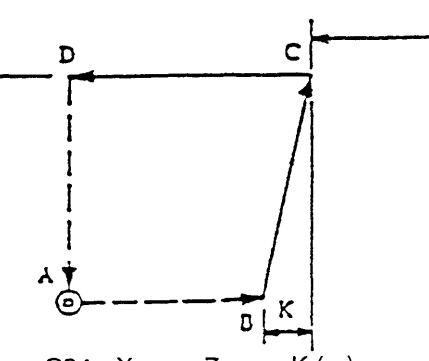
N104 G01 G96 X55.0 Z2.0 S120
N105 G94 X20.0 Z0 K-5.0 F0.2 ..... [1]
N106 G00 Z-3.0
N107 G94 X20.0 Z-5.0 K-5.0 ..... [2]
N108 G00 Z-8.0
N109 G94 X20.0 Z-10.0 K-5.0 ..... [3]
N110 G00 X.... Z....

```

### G94 Cycle Patterns (OD)

| Straight  | Taper  |
|---|--|
| <p>(1)</p>  <p style="text-align: center;">G94 X..... Z.....</p>           | <p>(1)</p>  <p style="text-align: center;">G94 X..... Z..... K(-).....</p>   |
| <p>(2)</p>  <p style="text-align: center;">G94 X..... Z.....</p>          | <p>(2)</p>  <p style="text-align: center;">G94 X..... Z..... K.....</p>     |
| <p>(3)</p>  <p style="text-align: center;">G94 X..... Z..... K.....</p> | <p>(4)</p>  <p style="text-align: center;">G94 X..... Z..... K(-).....</p> |

G94 Cycle Patterns (ID)

| Straight   | Taper  |
|--|--|
| <p>(1)</p>  <p>G94 X..... Z.....</p>  | <p>(1)</p>  <p>G94 X..... Z..... K(-).....</p>   |
| <p>(2)</p>  <p>G94 X..... Z.....</p> | <p>(2)</p>  <p>G94 X..... Z..... K.....</p>     |
|  | <p>(3)</p>  <p>G94 X..... Z..... K.....</p>    |
|  | <p>(4)</p>  <p>G94 X..... Z..... K(-).....</p> |

## 2-3-19 G70, G71, G72, G73, G74, G75 Compound Repetitive Cycle (Option)

A canned cycle with G90, G92 or G94 cannot simplify the program sufficiently. However, if you use a multiple repetitive cycle, the program can be greatly reduced by specifying a finish shape, such as enabling roughing and finishing.

As shown in the table below, 7 kinds of multiple repetitive cycles are available:

| G code | Name                       | Remarks                                |  |
|--------|----------------------------|--|--|
| G70    | Finishing cycle            |  |  |
| G71    | OD roughing cycle          |  |  |
| G72    | End face roughing cycle    | Finishing enabled<br>by G70            | Tool nose radius<br>compensation enabled |
| G73    | Closed loop cutting cycle  |  |  |
| G74    | End face cutting-off cycle | Tool nose radius compensation disabled |  |
| G75    | OD cutting-off cycle       |  |  |
| G76    | Automatic threading cycle  |  |  |

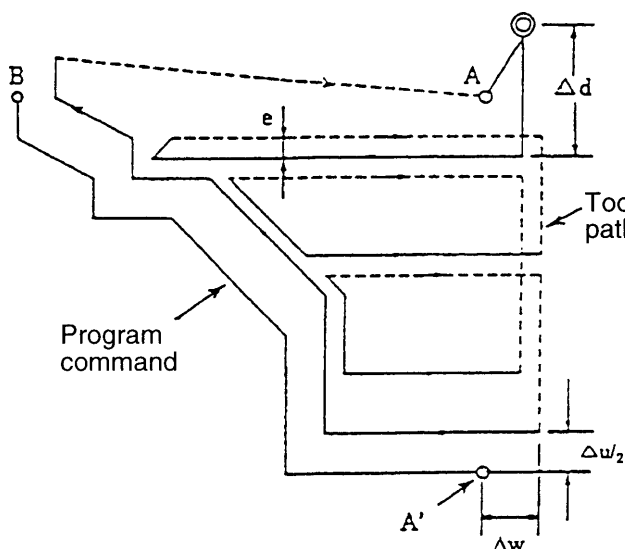
1. The G codes above are all unmoral ones belonging to the \* group.
2. Finish shape programs specified by G71, G72 and G73 are stored in the NC unit's internal memory. Its maximum capacity is 45 blocks which store a set of programs.
3. When chamfering or corner R is commanded, it is equivalent to the two blocks.

### 1. G71 ID/OD roughing cycle (Type I and TypeII) (Option)

If the Z-axis command (Z or W) is not placed at the first block of the finish shape, it becomes type I or it becomes type II if it is command.

#### (1) Type I

As shown in the figure below, if a finish shape between A and B via A' with a tape command, the tool cuts away a section specified with the depth of cut  $\Delta d$ , leaving the finishing allowances  $\Delta U/2$  and  $\Delta W$ .



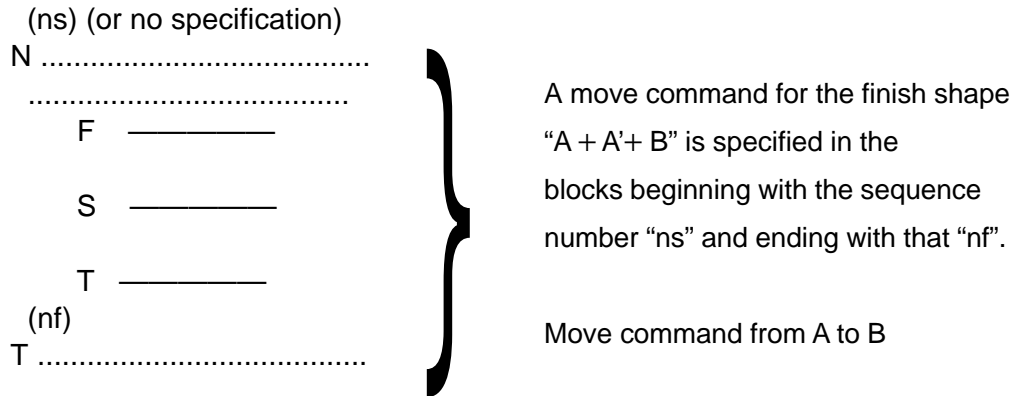
- An escape after turning retract at 45° direction.
  - Retract amount  $e = 0.5\text{mm}$
- Designate it at the parameter No.5145.

First, the tool cuts in parallel to its Z axis with the depth of cut  $\Delta d$ , and finally, it cuts in parallel to the tape command.

Create the tape command as follows:

•G71P (ns) Q (nf) U± W± I± K± D F S ;

•Rough finishing cycle is omitted when the 4 bit = 1 of the parameter No.5102.

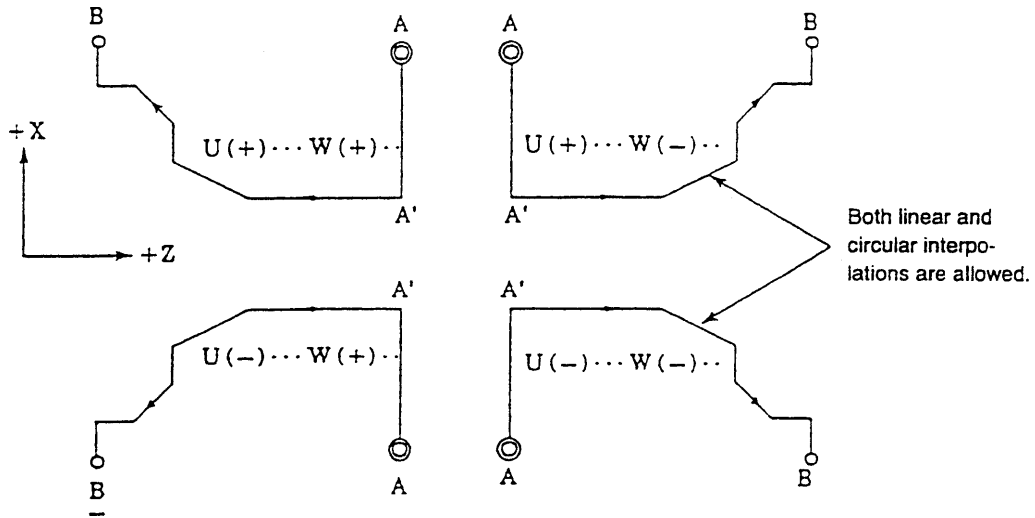


- P : First sequence No. (ns) of a group of finishing profile blocks
- Q : Last sequence No. (nf) of a group of finishing profile blocks
- U : X-directional finishing allowance and direction ( $\Delta U$ ) ... Diameter designation ( $\Delta U/2$  for radius designation)
- W : Z-directional finishing allowance and direction ( $\Delta W$ )
- D : Depth of cut ( $\Delta d$ ). Specify with no sign.
- I : X directional rough finishing allowance and direction ( $\Delta I$ ).. radius value
- K : Z directional rough finishing allowance and direction ( $\Delta K$ ).
- F, S : F and S function in any block within sequence No. P and Q in the cycle is ignored and those in the block of G71 or designated before G71 are effective. Also, in case of constant surface speed control, G96 or G97 during move command between A and B is ignored and those in the block of G71 or designated before G71 are effective.

The following 4 patterns are likely as to a profile to be cut with G71.

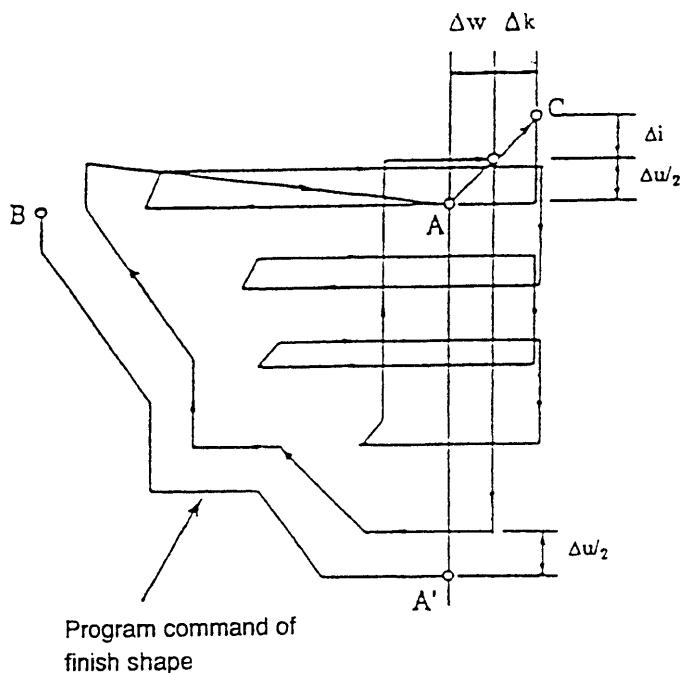
In any case, the workpiece is cut by tool movements in parallel with the Z axis of the tool.  
Signs of  $\Delta U$  and  $\Delta W$  are as follows:

The nose R compensation is not engaged in the type I of G71.



- Between A and A', a move command is given by the block with the sequence number "ns". Z-axis command cannot be included.
- Between A' and B, cutting must assume either monotonous incremental/decremental pattern in both X and Z directions.
- When a command between A and A' is of G00 mode, cutting -in along A to A' is also performed in the G00 mode.
- When a command between A and A' is of G01 mode, cutting -in along A to A' is also performed in the G01 mode.
- The coordinate value of X-axis of point A and B should be same.

### Execution of rough finishing cycle



At the last of this program, cut along with the shape leaving a finishing allowance, however, it can be omitted an execution by setting of 1 at the fourth bit of parameter No.5102 4BIT (NRC).

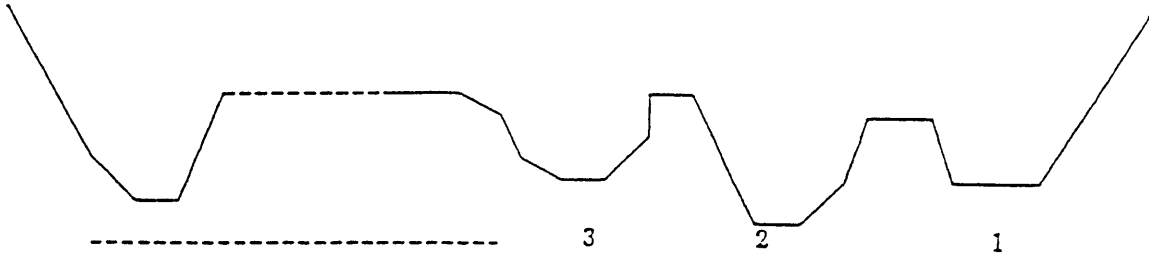
Also, even if the above case, execute rough cutting leaving a cutting amount designated by I and K and finally cut leaving a finishing allowance along with the shape by the command of I and K in the same block with G71. In this case, rough finishing cycle is executed after returning to the start point of cycle once.

(2) Type II

Type II differs from type I in the following points.

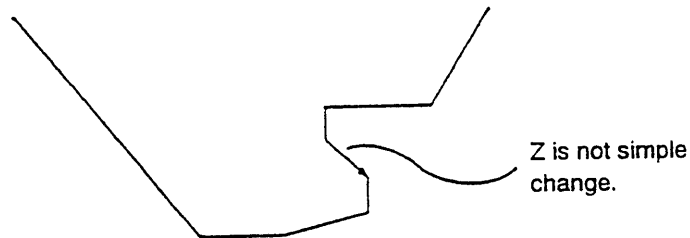
- (i) The shape is not necessary to be simple increase in X direction and it may have as many pockets as possible.

The first block of finishing shape requires movement of Z-axis.

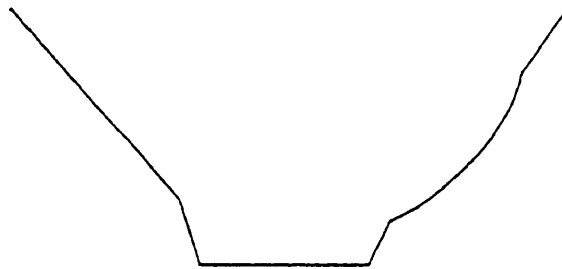


However, Z direction must be simple change.

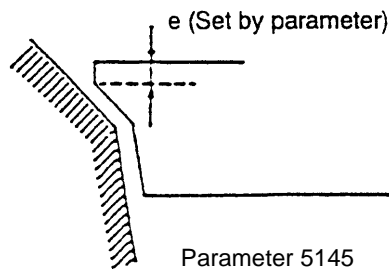
The following shape cannot be cut.



- (ii) The cutting at the beginning may not be vertical and the shape does not matter as long as Z-axis direction is simple change.

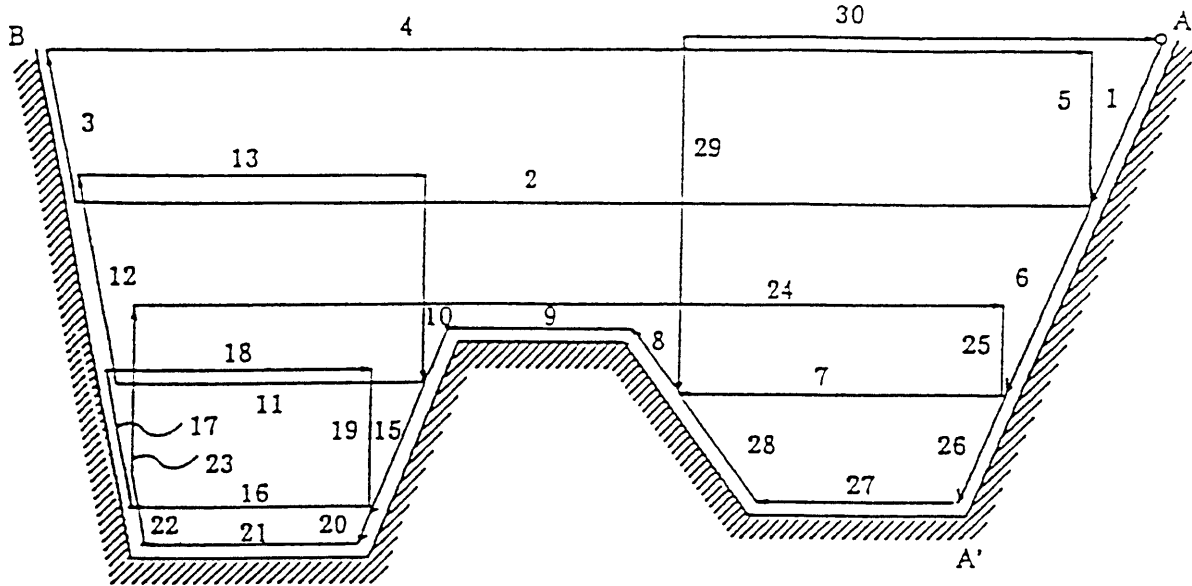


- (iii) The release after turning is performed on straight line after turning along the shape. (e)



Clearance e after cutting up is set to e = 0.5mm

(iv) The cutting path becomes as the following example.



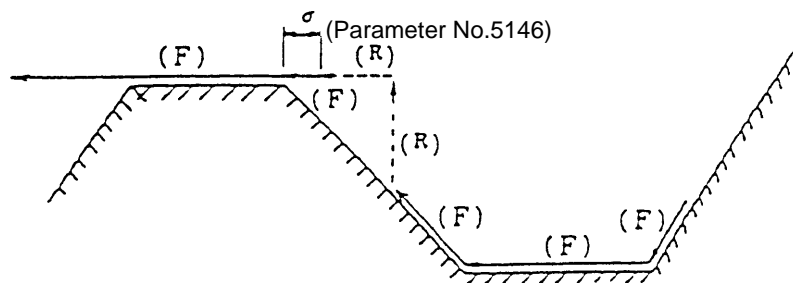
Between A and A' is commanded in the block with sequence No. (ns) and should be included the Z-axis command.

Even if no movement on Z-axis, command W0.

When moving amount of Z-axis is zero between A and A', cutting along with A and A' becomes the same cutting method with finish shape (G00 or G01).

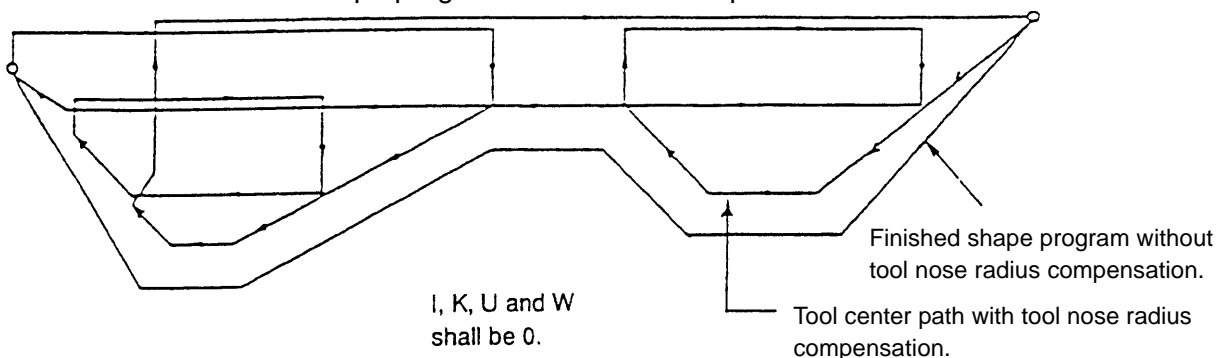
If Z-axis motion is included between A and A', indeed is done by G01.

- Cutting after one pocket is done, it becomes as follows.



(a) Coexistence of nose R compensation

This cycle can be executed with nose R compensation. In this case, a finish shape at the start point of cycle is cancel condition of compensation and each cutting motion is done by commanded finish shape program with nose R compensation.



(b) Execution of rough cutting finishing cycle

At the last part of this cycle, cutting is performed along the shape, leaving the finishing allowance.

By commanding I and K in the same block as G71, rough cutting is done, leaving the allowance specified in I and K, and finally cutting along the shape is performed, leaving the finishing allowance. In this case, rough cutting finishing cycle is executed after returning to the cycle start point once.

## 2. G72 End face rough cutting cycle (Type I and Type II) (Option)

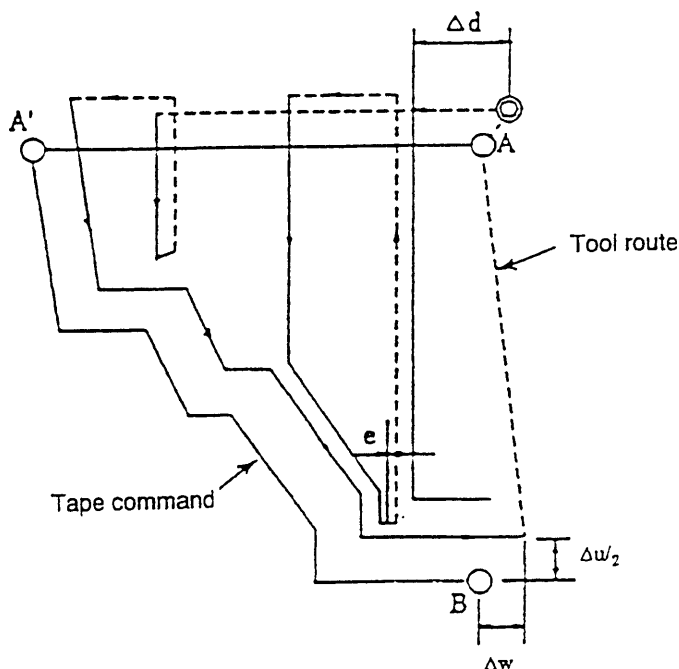
•G72P\_(ns) Q\_(nf) U±\_\_ W±\_\_ I±\_\_ K±\_\_ D\_\_ F\_\_ S\_\_ ;

- P : First sequence No. (ns) of the finishing pattern block
- Q : The last sequence No. (nf) of the finishing pattern block
- U : Finishing allowance in the X direction ( $\Delta u$ )
- W : Finishing allowance in the Z direction ( $\Delta w$ )
- I : Cutting allowance in the X direction for the rough cutting and finishing cycle ( $\Delta i$ )
- K : Cutting allowance in the Z direction for the rough cutting and finishing cycle ( $\Delta k$ )
- D : Cutting depth ( $\Delta d$ ) ..... To be commanded without symbol
- e : Release amount (parameter No.6213)

- F, S and T commands commanded between (ns) and (nf) are ignored and the block of G72 or the command commanded before that block is effective.

Cutting is executed by parallel motion with X axis as shown in the sketch.

There are type I and type II same as OD rough cutting cycle.

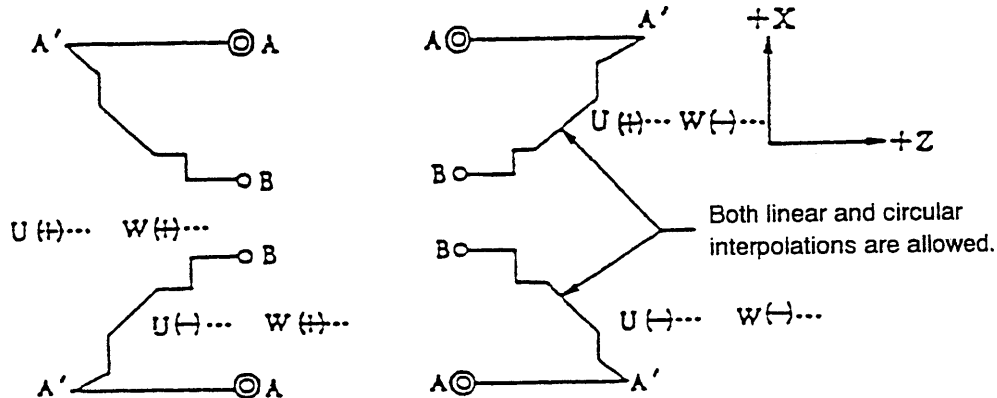


- An escape after turning retract at 45° direction.
  - Retract amount  $e = 0.5\text{mm}$
- Designate it at the parameter No.5145.

### (1) Type I

The following 4 patterns are likely as to a profile to be cut with G72. In any case, the workpiece is cut by repeating tool movements in parallel with the X axis of the tool. Signs of  $\Delta U$  and  $\Delta W$  are as follows:

- Tool movement between A and A' is commanded by the block of sequence No. "ns". An X-axis command cannot be contained.
- Tool movement between A' and -B must be a monotonous incremental or decremental pattern as to both X and Z axes.



- Whether a cutting mode along A to A' is G00 or G01 depends on a command between A and A'.

### (2) Type II

If command X and Z axes in the first block (In brief, the heading block of finish shape program) of repetitive section, the tool moves parallel to Z axis then along finish shape and retract parallel to X axis by retract amount ( $e=0.5\text{mm}$ ) then shift to next turning.

Outline of tape command is the same as type I except the command of X and Z axes are existed in the first block of repetitive section.

#### (a) Cutting shape

In case of type II, finish shape is not required simple change and regardless of the number of cavity section (Pocket).

However, must be kept simple change in X direction.

#### (b) Heading block of finish shape program

The first indeed section, in short, move command of heading block of finish shape program should be G00 or G01 and simple change on X axis direction.

The indeed section after that, regardless of any shape if simple change is kept in X direction.

**Note** 1) The coordinate value of Z-axis of point A and B should be the same.

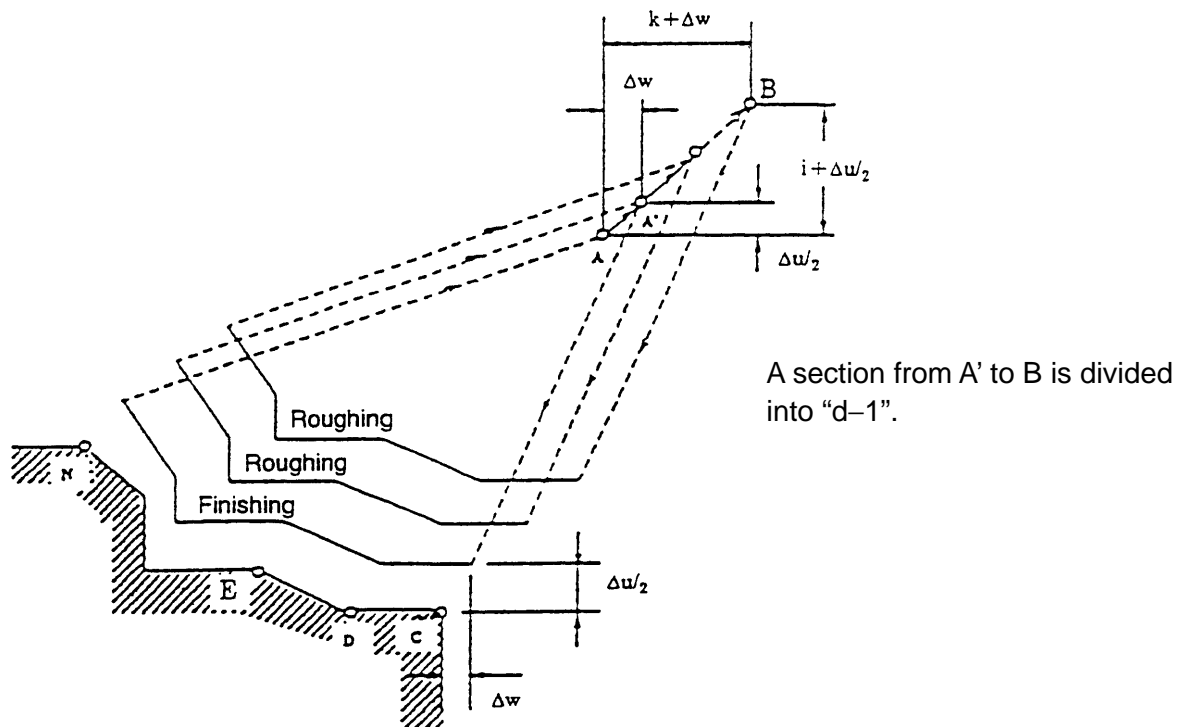
#### 2) Execution of rough finishing cycle

*Cut along the final shape of this cycle leaving finishing shape. Rough cut leaving allowance designated by I and K by commanding I and K in the same block of G72, then cut along the final shape leaving the finishing allowance.*

*In this case, semi-finishing cycle is executed after returning start point each time.*

### 3. G73 Closed loop cutting cycle (Option)

This G code can repeat a fixed cutting pattern, shifting a tool position little by little. With this cycle used, you can efficiently cut a workpiece whose material shape has been made in pre-machining such as forging or casting.



Pattern Specified by Tape

Point A → Point C → point D → Point E ..... → point N  
 Create a tape command as follows:

•G73P (ns) Q (nf) I±\_\_ K±\_\_ U±\_\_ W±\_\_ D\_\_ F\_\_ S\_\_ ;

(ns)

N.....

.....

(nf)

N.....

} Commanded by the blocks of sequence No. "ns" to "nf". Move command from A to N

P : Sequence No. (ns) of the first block of a group of finishing profile blocks

Q : Sequence No. (nf) of the last block of a group of finishing profile blocks

I : X directional all roughing allowance ( $\Delta i$ ) ..... Radius designation

K : Z directional all roughing allowance ( $\Delta k$ )

U : X directional finishing allowance ( $\Delta U$ ) ..... Diameter designation

W : Z-directional finishing allowance ( $\Delta W$ )

D : Number of divisions (d) ..... Equal to roughing times. (A decimal point is not available.)

F : Even if the F function is contained in any block between P and Q, it is ignored and the F function which is designated in the G73 block or previous block becomes effective.

Since there are four patterns for cutting shape, at the time to prepare a program for a machining set a center of nose R i.e.  $\Delta U$ ,  $\Delta W$ ,  $\Delta i$ , and  $\Delta K$  to start point, a nose R compensation is add on the  $\Delta U$  and  $\Delta W$  same as G71, if cutting is made by G73 with nose R compensation.

#### 4. G70 Finishing cycle

When rough cutting is performed with G71, G72 or G73, the following command allows finish cutting:

N \_\_ G70 P (ns) Q (nf) ;

P: Sequence No. (ns) of the first block of a group of finishing profile blocks

Q: Sequence No. (nf) of the last block of a group of finishing profile blocks

**Note)** • *The F and T functions specified in G71, G72 and G73 blocks are ignored. However, those specified between sequence No. "ns" and "nf" become valid.*

- *Be sure to execute multiple repetitive cycles (G70~G73) with memory operation.*
- *The P and Q blocks specified with G70 are searched for from the beginning of the memory, and a command between the P and Q blocks found first is executed. Therefore, see to it that the same sequence No. is not repeated in the memory.*
- *When roughing is executed by G71, G72 and G73, up to 3 pcs. of the memory address of P and Q blocks can be stored.*

*Eventually, when executing G70, the blocks specified by P and Q can be immediately searched without searching from the beginning of the memory for P and Q blocks. Further, the finishing cycles of G70 can be performed together after several roughing cycles of G71, G72 and G73 are executed.*

*In this case, the roughing cycles after 4th becomes longer the cycle time of because P and Q blocks are searched by searching the memory.*

G71P100Q200...;

N100...

...

...

N200

G71P300Q400...;

N300

...

...

N400

...

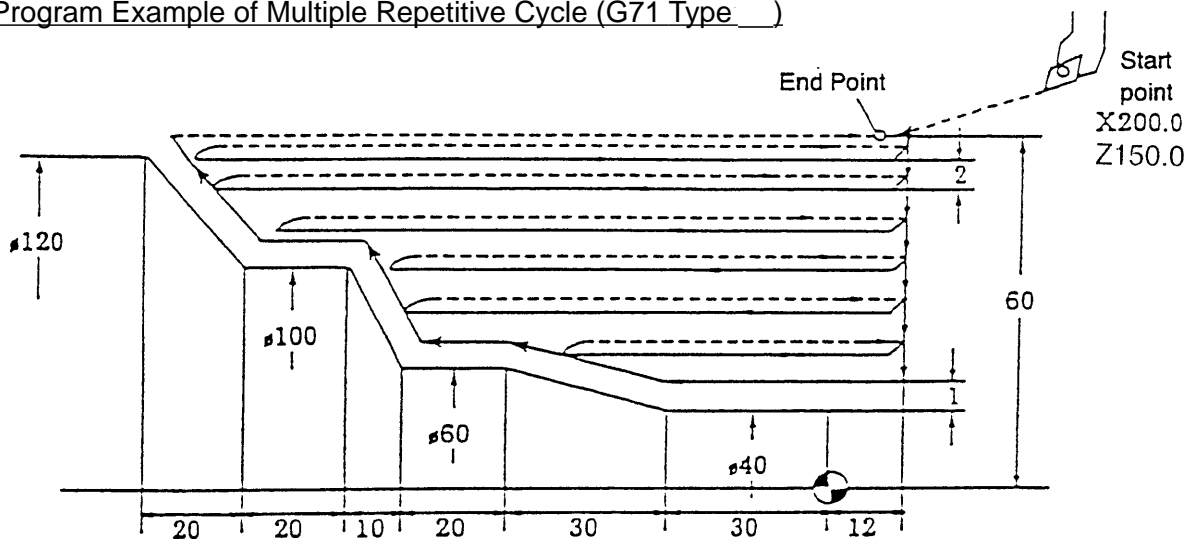
...

G70P100Q200; Up to 3 is executed without search.

G70P300Q400; Over 4 is executed after searching.

- When the cycle is completed, the tool returns to a start point at a rapid traverse rate. For NC command data, a block next to the G70 cycle is read.
- A subprogram cannot be called between the sequence No. "ns" and "nf" used for G70~G73.
- The memory addresses stored by the roughing cycle of G71~G73 are erased after executing G70. Moreover, all the memory addresses stored by reset are erased.
- If the reset button is pressed after the G71~G73 cycles were executed, the finish cycle of G70 is disabled to perform.

Program Example of Multiple Repetitive Cycle (G71 Type )

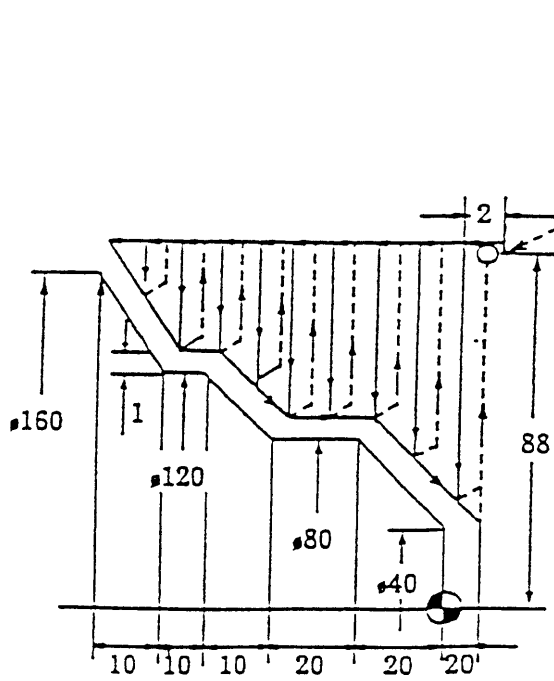


```

N100 (OD-R)
N101 T0100;
N102 G97 S240 M08;
N103 G00 X120.0 Z10.0 M03;
N104 G96 S120;
N105 G71 P106 Q112 U2.0 W2.0 D2.0 F0.3;
N106 G00 X40.0 F0.15;
N107 G01 Z-30.0;
N108 X60.0 Z-60.0 ;
N109 Z-80.0;
N110 X100.0 Z-90.0;
N111 Z-110.0;
N112 X120.0 Z-130.0;
N113 G00 G97 X200.0 Z150.0 S500;

```

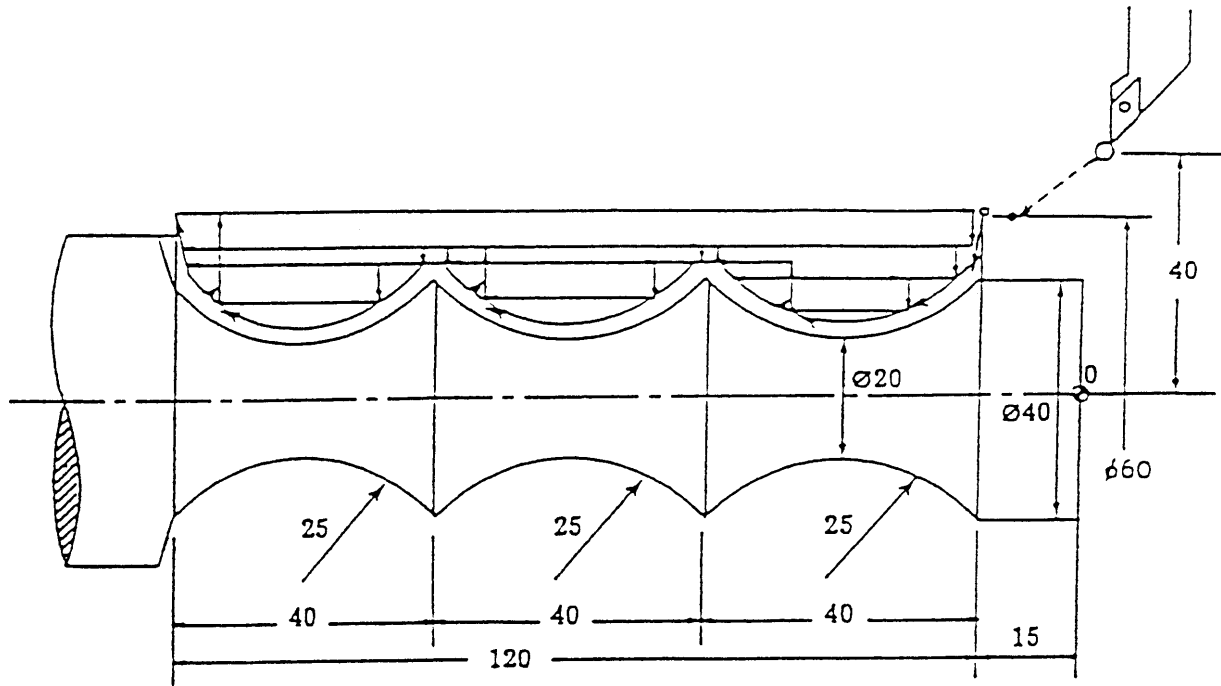
Program Example of Multiple Repetitive Cycle(G72)



```

N100 (FA-R)
N101 T0100;
N102 G97 S220 M08;
N103 G00 X176.0 Z2.0 M03;
N104 G96 S120;
N105 G72 P106 Q111 U2.0 W0.5 D2.0 F0.3;
N106 G00 Z-70.0 F0.15;
N107 G01 X120.0 Z-60.0;
N108     Z-50.0 ;
N109     X80.0 Z-40.0;
N110     Z-20.0;
N111     X36.0 Z2.0;
N112 G00 G97 X200.0 Z150.0 S500;
N114 M01 ;
    
```

Program Example of Compound Canned Cycle (G71 Type II)



```

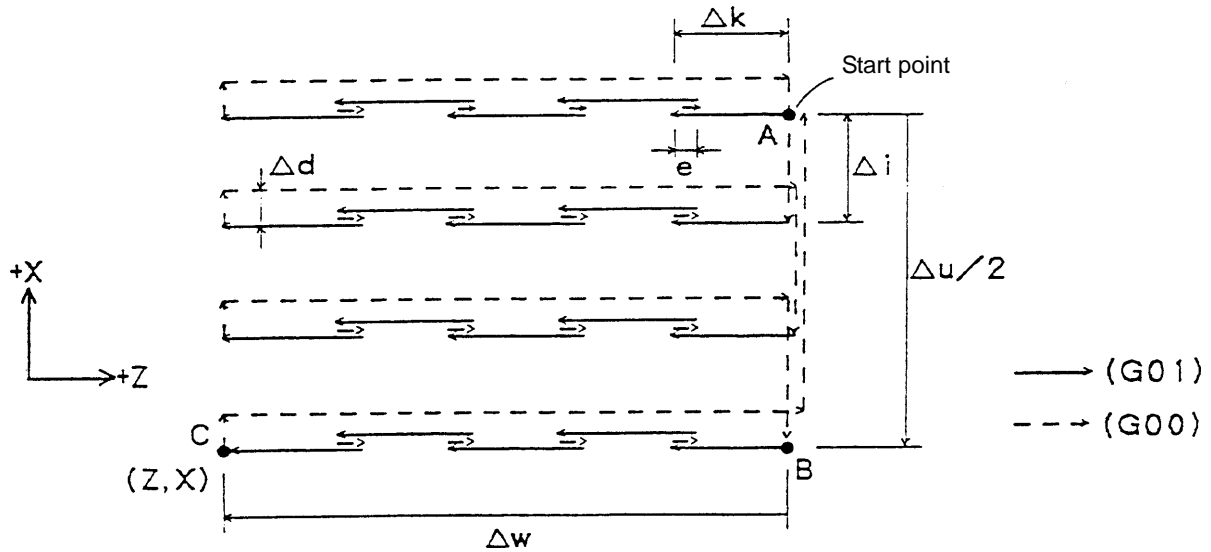
N010 T0300;
N011 G97 S1650 M08;
N012 G00 X60.0 Z-15.0 M03;
N013 G71 P014 Q018 U0.5 D5.0 F0.3;
N014 G01 X40.0 W0 F0.15;
N015 G02 Z-55.0 R25.0;
N016     Z-95.0 R25.0;
N017     Z-135.0 R25.0;
N018 G01 X60.0;
    
```

The tool nose radius for offset No. 03 shall be 2 mm and the start point shall be the same as the tool nose center.



## 5. G74 End face cutting-off cycle

By this command, chip disposal in end face cutting-off can be functioned. Also, if X(U) and I are omitted, peck drilling cycle in Z axis direction is effected.



G74 X(U)\_\_\_ Z(W)\_\_\_ I\_\_\_ K\_\_\_ D\_\_\_ F\_\_\_ R\_\_\_ ;

X : Point C

U : A→C X-direction incremental amount ( $\Delta u$ )

Z : Point C

W : A→C Z-direction incremental amount ( $\Delta w$ )

I : One-time cutting amount in X-direction (absolute value) ( $\Delta i$ )

K : One-time cutting amount in Z-direction (absolute value) ( $\Delta k$ )

D : Tool relief amount at the cutting bottom ( $\Delta d$ )

F : Feed rate

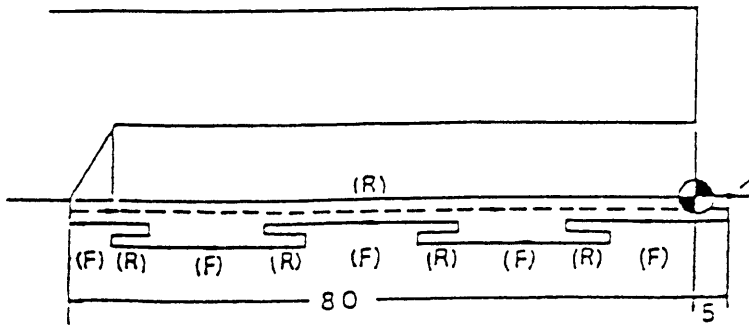
e : Return amount (parameter No.5147)

R : If commanded with value "0", or the value is omitted, retreat is effected by the value "e" in Z-axis return action. If commanded with value "1", retreat is effected to the start point (A→B) in each action of Z-direction return.

**Note)** At the cutting bottom, tool relief is in the direction of B→A. When there is no X-direction movement, the relief is in the direction of the sign of the value of  $\Delta d$ .

Program example of peck drilling cycle (G74)

In case of peck drilling cycle, omit I and D.

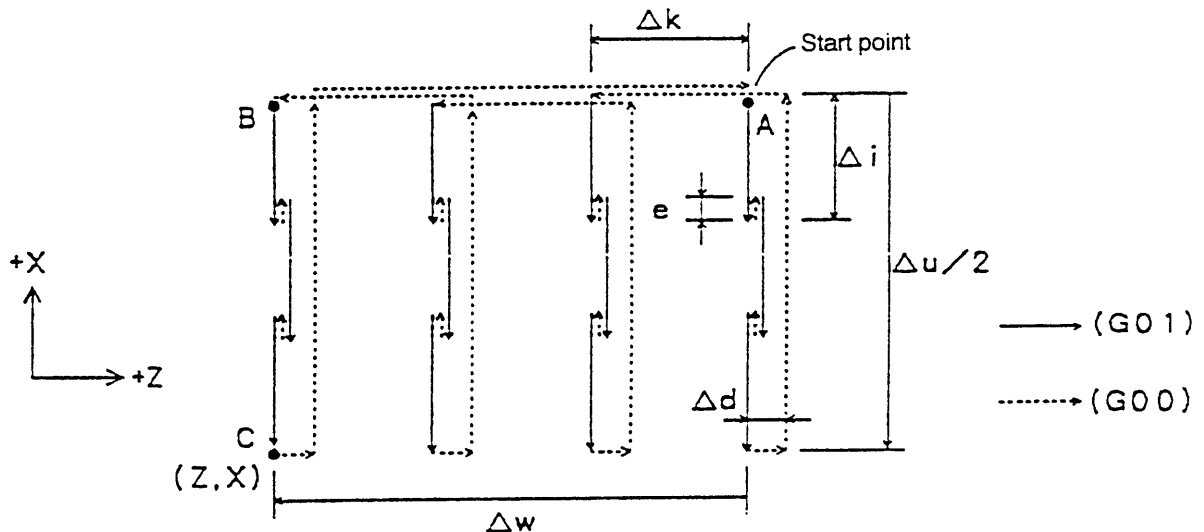


```

N201 T0200;
N202 G97 S300 M08;
N203 G00 X0 Z5.0 M03;
N204 G74 Z-80.0 K20.0 F0.15;
N205 G00 X200.0 Z100.0;
N206 M01;
    
```

**6. Outside diameter cutting-off cycle**

By the command, chip disposal in end face cutting off can be functioned. Also, if Z(W) and K are omitted, grooving and end cutting-off can be effected.

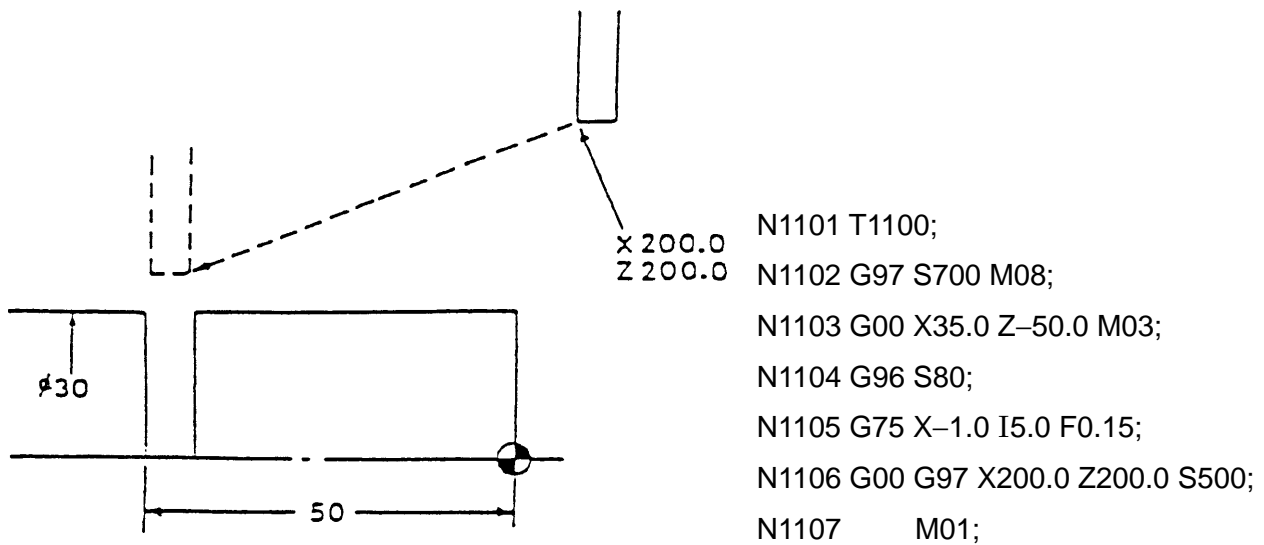


```
G75 X(U)___ Z(W)___ I___ K___ D___ F___ R___ ;
```

- X : Point C
- U : A→C X-direction incremental amount ( $\Delta u$ )
- Z : Point C
- W : A→C Z-direction incremental amount ( $\Delta w$ )
- I : One-time cutting amount in X-direction (absolute value) ( $\Delta i$ )
- K : One-time cutting amount in Z-direction (absolute value) ( $\Delta k$ )
- D : Tool relief amount at the cutting bottom ( $\Delta d$ )
- F : Feed rate
- e : Return amount (parameter No.5147)
- R : If commanded with value "0", or the value is omitted, retreat is effected by the value "e" in X-axis return action. If commanded with value "1", retreat is effected to the start point (A→B) in each action of X-direction return.

**Note)** At the cutting bottom, tool relief is in the direction of B→A. When there is no Z-direction movement, the relief is in the direction of the sign of the value of  $\Delta d$ .

### Program example of OD grooving cycle (G75)



### Precautions for Multiple Repetitive Cycles(G70-G76)

- (1) In the blocks where multiple repetitive cycles are to be specified, you must correctly specify necessary parameters, such as P, Q, X, Z, U, W, I, K, D and A, block by block.
- (2) In the G71, G72 and G73 blocks with the sequence number specified with P, you must specify G00 or G01 of the group 01 without fail.
- (3) It is not allowed to give a G70-G76 command by MDI.
- (4) You cannot specify an M98/M99 command in the block where G70, G71, G72 or G73 was specified, and between the blocks of the sequence numbers specified with P and Q of G70, G71, G72, or G73.
- (5) G codes which can be specified in the blocks excluding the sequence numbers specified with P and Q of G70, G71, G72 and G73 are G00, G01, G02, G03, G04, G09, G61, G64, G96, G97, G98, G99 and G196.
- (6) Although you can make manual operation intervene in the multiple repetitive cycle (G70-G76) halfway its execution by suspending it, when resuming the multiple repetitive cycle, be sure to return the machine to the position where manual operation intervened.  
If you resume the multiple repetitive cycle without returning the machine, a machine stroke by manual operation is not added to an absolute value even if the manual absolute switch is turned on, and thereafter, the machine functions with a shift of the stroke by manual operation.
- (7) When executing G70, G71, G72 or G73, the sequence number-for a G70 block and those specified with P and Q should not be the same as those in the memory.
- (8) In a G71 command, the X coordinate value of the start point must be equal to that of the end point of a finish shape.
- (9) In a G72 command, the Z coordinate value of the start point must be equal to that of the end point of a finish shape.
- (10) Do not command the program which contains chambering or rounding at the end of last moving command of finishing pattern block group with P and Q of G70, G71, G72, G73.

## 2-3-20 G32, G92, G76 Thread Cutting

A G32 command enables straight/taper/face thread cutting and tapping, and G92 and G76 (option) commands enable straight/taper-thread cutting.

- Threading code and lead programmable range

Specify a lead with a numerical value following F.

| G code | Description                             | Command address | Lead range            |                        |
|--------|---|-----------------|-----------------------|------------------------|
|        |   |                 | Metric input          | Inch input             |
| G32    | Threading                               | F               | mm/rev                | inch/rev               |
| G92    | Canned cycle for threading              | E               | 0.00001<br>~999.99999 | 0.000001<br>~99.999999 |
| G76    | Multiple repetitive cycle for threading |                 |                       |                        |

When designating number of thread in case of cutting inch thread, it is possible to set 7 bits of the parameter No. 3403 to 1 and to command E14.0 (14 thread per inch) etc.

G32 Z\_ E14.0

### Limitation of spindle speed

The following limitation must be observed in threading by G32/G92/G76.

$$P = \frac{5000}{N} \quad P: \text{Lead or pitch(mm)}$$

$$N = \frac{5000}{P} \quad N: \text{Spindle speed(min}^{-1}\text{)}$$

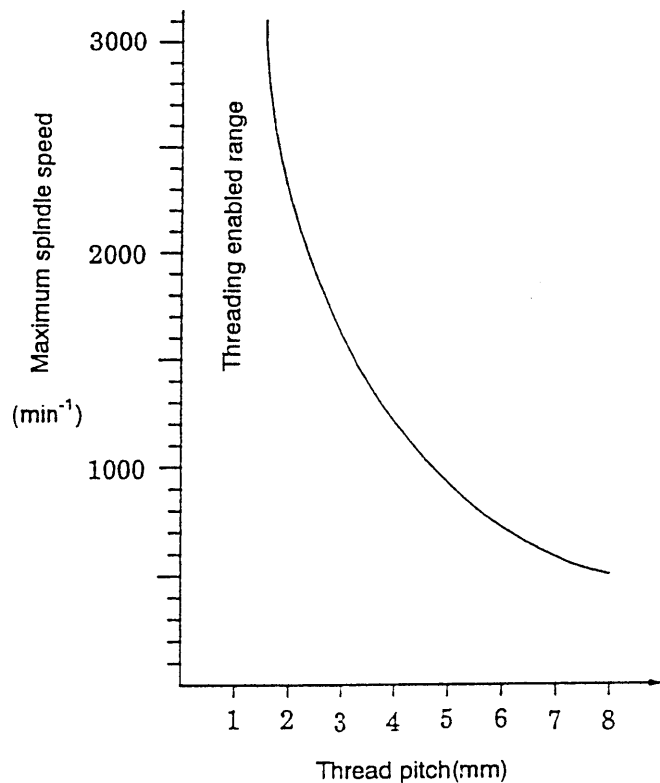
(Example)

When a thread pitch is 3 mm, a spindle speed is; (min<sup>-1</sup>)

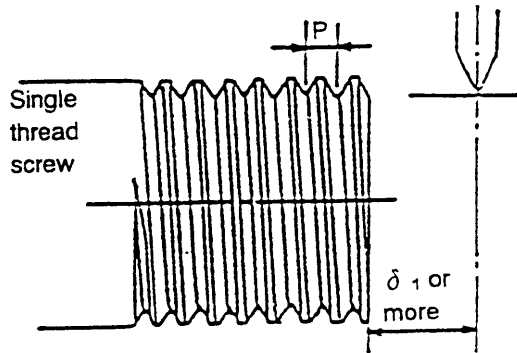
$$N = \frac{5000}{P} = \frac{5000}{3} = 1666 \text{ (min}^{-1}\text{)}$$

Therefore, when cutting threads with a pitch of 3mm, use a spindle speed of 1666 rpm or less.

**Note)** The above-mentioned limitation does not apply to an oil groove, etc. which do not require an accurate lead.

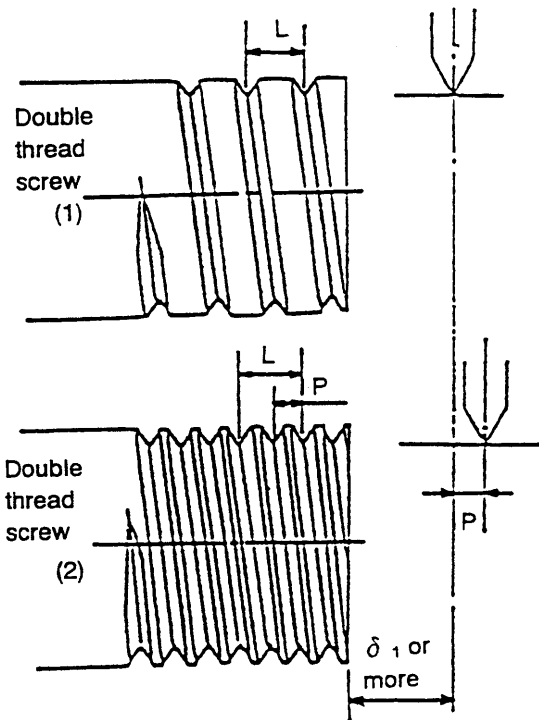


1. Cutting the single thread screw



For a single thread screw, cut at a threading feed rate of  $P$  mm/rev from an arbitrary position by  $\delta_1$  or more away from the end face of a thread part.

2. Cutting the multiple thread screw



Cut the first thread of a double thread screw at a threading feed rate of  $L$  mm/rev from an arbitrary position by  $\delta_1$  or more away from the end face of a thread part.

Cut the second thread of the double thread screw at a threading feed rate of  $L$  mm/rev from a position by  $P$  mm away from the cutting start position of the first thread. This also applies to an  $n$ -thread screw.

Important Formulas for Thread

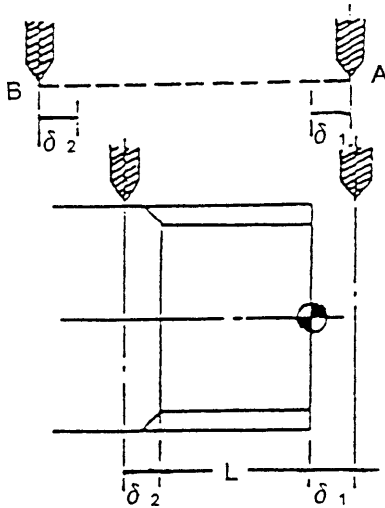
$$l = n \cdot P \quad P = \frac{l}{n} \quad \left( \begin{array}{ll} l: \text{Thread lead} & P: \text{Pitch} \\ n: \text{No. of threads} & \alpha: \text{Twisting angle} \\ & \beta: \text{Lead angle} \end{array} \right)$$

$$\alpha + \beta = 90^\circ \quad \tan \beta = \frac{l}{2\pi r} \quad \cot \alpha = \frac{l}{2\pi r}$$

$$\text{Effective sectional area} = \frac{\pi}{4} \left( \frac{\text{Effective diameter} + \text{Thread bottom diameter}}{2} \right)^2$$

$$\text{Thread efficiency } \eta = \frac{\tan \beta}{\tan (\rho + \beta)} \quad \left( \begin{array}{l} \rho: \text{Friction angle of} \\ \text{helicoidal surface} \end{array} \right)$$

<Incomplete thread>



When cutting the thread from the point A to the point B, it causes shorter leads(pitches) of  $\delta_1$  and  $\delta_2$  at the cutting start point A due to acceleration and at the cutting end point B due to deceleration, respectively. Therefore, when obtaining an effective thread length "1", a threading length of "1+ $\delta_1$ + $\delta_2$ " is required.

<How to determine  $\delta_1$ >

Obtain  $\delta_1$  from the spindle speed and thread lead (pitch) used for threading, using the following formula:

$$\delta_1 \text{ (mm)} = 0.0015 \times R(\text{min}^{-1}) \times L(\text{mm})$$

Example) When cutting a JIS Class-1 thread with a pitch of 1.5 at a spindle speed of 800 rpm;

$$\delta_1 \text{ (mm)} = 0.0015 \times 800 \times 1.5 = 1.8(\text{mm})$$

<How to determine  $\delta_2$ >

Obtain  $\delta_2$  from the spindle speed and thread lead (pitch) used for threading, using the following formula:

$$\delta_2 \text{ (mm)} = 0.00042 \times R(\text{min}^{-1}) \times L(\text{mm})$$

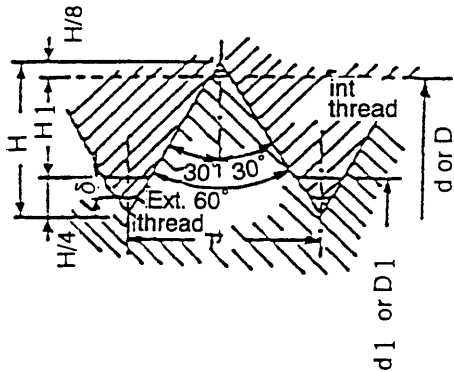
Example) When cutting a JIS Class-1 thread with a pitch of 1.5 at a spindle speed of 800min<sup>-1</sup>

$$\delta_2 \text{ (mm)} = 0.00042 \times 800 \times 1.5 = 0.504(\text{mm})$$

**Note)** As mentioned above,  $\delta_1$  and  $\delta_2$  values are determined by the spindle speed and thread lead. Accordingly, when cutting one screw, the spindle speed must not be changed to the last.

(A threading section would be shifted.)

## Thread Cutting Method



<Metric coarse and fine threads>

$$H = 0.866025P$$

$$H_1 = 0.541266P$$

$$d_1 = D_1 = d - 2 \times H_1 = d - 1.082532P$$

(1) The following shows formulas used for calculating reference thread shapes for metric coarse/fine and unified coarse/fine threads:

<Unified coarse and fine threads>

$$P = 25.4/n$$

$$H = 0.866025/n \times 25.4$$

$$H_1 = 0.541266/n \times 25.4$$

$$d = (d) \times 25.4$$

$$d_1 = D_1 = (d - 1.082532/n) \times 25.4$$

$n$  : No. of threads per 25.4mm

(d) : Nominal size for thread

## Number of cutting times by pitch in threading

The number of cutting times is calculated as follows:

$$NE = K \times P + 2.5 \quad (P > 0)$$

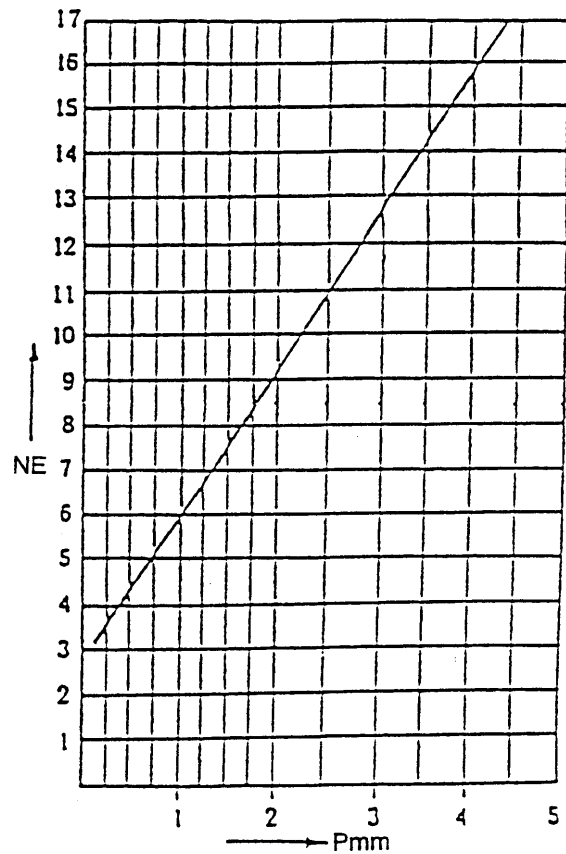
$$= 3.3P + 2.5$$

**Note)** After calculating NE, raise its decimal places to a unit.

NE : No. of cutting times

K : Constant based on various conditions (assumed to be 3.3 in this case)

P : Thread pitch to be cut (mm)



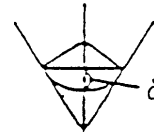
You must determine a depth of cut, depending on the nose R of a tip used. As shown in the right figure, assuming a relief amount to be  $\delta$  and a relief cutting part to be an arc (nose R);

$$\delta = \frac{1}{4} H - R = \frac{1}{4} P \cos 30^\circ - R \quad \dots\dots (1)$$

external thread

$$\delta = \frac{1}{8} H - R = \frac{1}{8} P \cos 30^\circ - R$$

internal thread



A maximum allowable value for R is;

$$R_{\max} = \frac{P}{4 \times \tan 60^\circ}, \quad R_{\max} = \frac{P}{8 \times \tan 60^\circ} \quad \dots\dots (2)$$

external thread                      internal thread

Example) When  $P = 2.5$ , a maximum nose R is;  $R_{\max}$  (external thread) = 0.36,  $R_{\max}$  (internal thread) = 0.18

Therefore ; when R0.2 is used for the external thread,

$$\delta = 0.34, \text{ and depth of cut} = H_1 + \delta = 1.353 + 0.34 = 1.694.$$

When R0.1 is used for the internal thread,

$$\delta = 0.17, \text{ and depth of cut} = H_1 + \delta = 1.353 + 0.17 = 1.524.$$

<Depth of cut and No. of Cutting Times for 60° Triangular Thread>

| P                                  | 1.0                            |                                | 1.25                           |                                | 1.5                            |                                | 1.75                           |                                | 2.0                            |                                | 2.5                            |                                | 3.0                            |                                | 3.5                            |                                |
|------------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|
| H <sub>1</sub>                     | 0.541                          |                                | 0.677                          |                                | 0.812                          |                                | 0.947                          |                                | 1.083                          |                                | 1.353                          |                                | 1.624                          |                                | 1.894                          |                                |
|                                    | Ext. thread                    | Int. thread                    | Ext. thread                    | Int. thread                    | Ext. thread                    | Int. thread                    | Ext. thread                    | Int. thread                    | Ext. thread                    | Int. thread                    | Ext. thread                    | Int. thread                    | Ext. thread                    | Int. thread                    | Ext. thread                    | Int. thread                    |
| Max. Nose R RMAX                   | 0.14                           | 0.07                           | 0.18                           | 0.09                           | 0.21                           | 0.10                           | 0.25                           | 0.12                           | 0.29                           | 0.14                           | 0.36                           | 0.18                           | 0.43                           | 0.21                           | 0.50                           | 0.25                           |
| Calculated Nose R                  | 0.1                            | 0.07                           | 0.1                            | 0.09                           | 0.2                            | 0.1                            | 0.2                            | 0.1                            | 0.2                            | 0.1                            | 0.3                            | 0.1                            | 0.3                            | 0.2                            | 0.4                            | 0.2                            |
| d Relief                           | 0.12                           | 0.04                           | 0.17                           | 0.04                           | 0.12                           | 0.06                           | 0.18                           | 0.09                           | 0.23                           | 0.11                           | 0.24                           | 0.17                           | 0.35                           | 0.12                           | 0.36                           | 0.18                           |
| d + H <sub>1</sub><br>Depth of Cut | 0.661                          | 0.581                          | 0.847                          | 0.717                          | 0.932                          | 0.872                          | 1.127                          | 1.037                          | 1.313                          | 1.193                          | 1.593                          | 1.523                          | 1.973                          | 1.741                          | 2.251                          | 2.074                          |
| n                                  | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ | $\frac{\Delta X(n)}{\Delta W}$ |
| *1                                 | 0.45\ (0.641\)                 | 0.45\ (0.541\)                 | 0.5\ (0.722\)                  | 0.5\ (0.605\)                  | 0.55\ (0.729\)                 | 0.55\ (0.679\)                 | 0.6\ (0.821\)                  | 0.6\ (0.751\)                  | 0.65\ (0.900\)                 | 0.65\ (0.815\)                 | 0.7\ (1.035\)                  | 0.7\ (0.989\)                  | 0.75\ (1.171\)                 | 0.75\ (0.027\)                 | 0.8\ (1.226\)                  | 0.8\ (1.128\)                  |
| 2                                  | 0.878\ 0.124                   | 0.765\ 0.091                   | 1.02\ 0.150                    | 0.856\ 0.103                   | 1.031\ 0.139                   | 0.96\ 0.118                    | 1.162\ 0.162                   | 1.066\ 0.135                   | 1.273\ 0.180                   | 1.153\ 0.145                   | 1.414\ 0.206                   | 1.398\ 0.202                   | 1.657\ 0.122                   | 1.453\ 0.203                   | 1.734\ 0.270                   | 1.595\ 0.230                   |
| 3                                  | 1.075\ 0.057                   | 0.937\ 0.05                    | 1.25\ 0.066                    | 1.049\ 0.056                   | 1.261\ 0.067                   | 1.177\ 0.063                   | 1.423\ 0.075                   | 1.305\ 0.069                   | 1.559\ 0.082                   | 1.412\ 0.075                   | 1.793\ 0.095                   | 1.712\ 0.091                   | 2.03\ 0.107                    | 1.779\ 0.194                   | 2.124\ 0.112                   | 1.954\ 0.103                   |
| 4                                  | 1.242\ 0.048                   | 1.082\ 0.042                   | 1.444\ 0.056                   | 1.211\ 0.047                   | 1.457\ 0.057                   | 1.358\ 0.052                   | 1.643\ 0.064                   | 1.507\ 0.058                   | 1.800\ 0.070                   | 1.631\ 0.063                   | 2.071\ 0.080                   | 1.977\ 0.096                   | 2.363\ 0.090                   | 2.055\ 0.88                    | 2.453\ 0.095                   | 2.256\ 0.087                   |
| 5                                  | 1.282\                         | 1.122\                         | 1.614\ 0.049                   | 1.354\ 0.041                   | 1.628\ 0.049                   | 1.519\ 0.046                   | 1.837\ 0.056                   | 1.685\ 0.051                   | 2.013\ 0.061                   | 1.823\ 0.055                   | 2.315\ 0.070                   | 2.211\ 0.067                   | 2.62\ 0.08                     | 2.297\ 0.07                    | 2.742\ 0.083                   | 2.523\ 0.077                   |
| 6                                  | 1.322\                         | 1.162\                         | 1.654\                         | 1.394\                         | 1.784\ 0.045                   | 1.664\ 0.042                   | 2.013\ 0.051                   | 1.846\ 0.046                   | 2.205\ 0.055                   | 1.897\ 0.050                   | 2.536\ 0.064                   | 2.422\ 0.061                   | 2.87\ 0.072                    | 2.517\ 0.063                   | 3.004\ 0.075                   | 2.763\ 0.070                   |
| 7                                  |                                |                                | 1.674\                         | 1.434\                         | 1.824\                         | 1.704\                         | 2.174\ 0.046                   | 1.994\ 0.043                   | 2.381\ 0.051                   | 2.157\ 0.046                   | 2.739\ 0.058                   | 2.616\ 0.056                   | 3.1\ 0.066                     | 2.718\ 0.058                   | 3.245\ 0.069                   | 2.985\ 0.014                   |
| 8                                  |                                |                                |                                |                                | 1.864\                         | 1.744\                         | 2.214\ 0.047                   | 2.034\ 0.043                   | 2.546\ 0.047                   | 2.306\ 0.043                   | 2.928\ 0.054                   | 2.796\ 0.052                   | 3.314\ 0.062                   | 2.906\ 0.054                   | 3.469\ 0.064                   | 3.191\ 0.060                   |
| 9                                  |                                |                                |                                |                                |                                |                                | 2.254\                         | 2.074\                         | 2.586\                         | 2.346\                         | 3.106\ 0.051                   | 2.966\ 0.043                   | 3.515\ 0.058                   | 3.082\ 0.051                   | 3.673\ 0.861                   | 3.385\ 0.056                   |
| 10                                 |                                |                                |                                |                                |                                |                                |                                |                                | 2.626\                         | 2.386\                         | 3.146\                         | 3.006\                         | 3.705\ 0.055                   | 3.249\ 0.048                   | 3.898\ 0.057                   | 3.568\ 0.053                   |
| 11                                 |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                | 3.186\                         | 3.046\                         | 3.866\ 0.046                   | 3.408\ 0.045                   | 4.067\ 0.054                   | 3.742\ 0.050                   |
| 12                                 |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                | 3.906\                         | 3.448\                         | 4.248\ 0.052                   | 3.908\ 0.048                   |
| 13                                 |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                | 3.946\                         | 3.488\                         | 4.422\ 0.050                   | 4.068\ 0.046                   |
| 14                                 |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                | 4.462\                         | 4.108\                         |
| 15                                 |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                |                                | 4.502\                         | 4.148\                         |

\* Since calculated values within parenthese at the bottom are too large, use corrected values at the top.

When Cutting Straight (External Thread)

| G   | X     | Z      | F    | Remarks                              |
|-----|-------|--------|------|--------------------------------------|
| G00 | X...  | Z...   |      |                                      |
| G92 | X9.55 | ZΔΔ.ΔΔ | F1.0 | $d-\Delta X(1) = 10 - 0.45 = 9.55$   |
|     | X9.12 |        |      | $d-\Delta X(2) = 10 - 0.878 = 9.122$ |
|     | X8.92 |        |      | $d-\Delta X(3) = 10 - 1.075 = 8.925$ |
|     | X8.76 |        |      | $d-\Delta X(4) = 10 - 1.242 = 8.758$ |
|     | X8.72 |        |      | $d-\Delta X(5) = 10 - 1.282 = 8.718$ |
|     | X8.68 |        |      | $d-\Delta X(6) = 10 - 1.322 = 8.678$ |
| G00 | X...  | Z...   |      |                                      |

For M10,P1.0

When Cutting Straight (Internal Thread)

| G   | X     | Z      | F    | Remarks                               |
|-----|-------|--------|------|---------------------------------------|
| G00 | X...  | Z...   |      |                                       |
| G92 | X9.25 | ZΔΔ.ΔΔ | F1.0 | $d+\Delta X(1) = 8.8 + 0.45 = 9.25$   |
|     | X9.57 |        |      | $d+\Delta X(2) = 8.8 + 0.765 = 9.565$ |
|     | X9.73 |        |      | $d+\Delta X(3) = 8.8 + 0.937 = 9.737$ |
|     | X9.88 |        |      | $d+\Delta X(4) = 8.8 + 1.082 = 9.882$ |
|     | X9.92 |        |      | $d+\Delta X(5) = 8.8 + 1.122 = 9.922$ |
|     | X9.96 |        |      | $d+\Delta X(6) = 8.8 + 1.162 = 9.962$ |
| G00 | X...  | Z...   |      |                                       |

When Cutting Along Helicoidal Surface (External Thread)

| G             | X     | Z      | F    | Remarks                              |
|---------------|-------|--------|------|--------------------------------------|
| G00           | X...  | Z...   |      |                                      |
| G92           | X9.55 | ZΔΔ.ΔΔ | F1.0 | $d-\Delta X(1) = 10 - 0.45 = 9.55$   |
| G01<br>or G00 |       | W-0.12 |      | $\Delta W = 0.124 \quad 0.12$        |
| G92           | X9.12 | ZΔΔ.ΔΔ |      | $d-\Delta X(2) = 10 - 0.878 = 9.122$ |
| G01<br>or G00 |       | W-0.06 |      | $\Delta W = 0.057 \quad 0.06$        |
| G92           | X8.92 | ZΔΔ.ΔΔ |      | $d-\Delta X(3) = 10 - 1.075 = 8.925$ |
| G01<br>or G00 |       | W-0.05 |      | $\Delta W = 0.048 \quad 0.05$        |
| G92           | X8.76 | ZΔΔ.ΔΔ |      | $d-\Delta X(4) = 10 - 1.242 = 8.758$ |
|               | X8.72 |        |      | $d-\Delta X(5) = 10 - 1.282 = 8.718$ |
|               | X8.68 |        |      | $d-\Delta X(6) = 10 - 1.322 = 8.678$ |
| G00           | X...  | Z...   |      |                                      |

When Cutting Along Helicoidal Surface (Internal Thread)

| G             | X     | Z      | F    | Remarks                               |
|---------------|-------|--------|------|---------------------------------------|
| G00           | X...  | Z...   |      |                                       |
| G92           | X9.25 | ZΔΔ.ΔΔ | F1.0 | $d+\Delta X(1) = 8.8 + 0.45 = 9.25$   |
| G01<br>or G00 |       | W-0.09 |      | $\Delta W = 0.091 \quad 0.09$         |
| G92           | X9.57 | ZΔΔ.ΔΔ |      | $d+\Delta X(2) = 8.8 + 0.765 = 9.565$ |
| G01<br>or G00 |       | W-0.05 |      | $\Delta W = 0.05$                     |
| G92           | X9.73 | ZΔΔ.ΔΔ |      | $d+\Delta X(3) = 8.8 + 0.937 = 9.737$ |
| G01<br>or G00 |       | W-0.04 |      | $\Delta W = 0.042$                    |
| G92           | X9.88 | ZΔΔ.ΔΔ |      | $d+\Delta X(4) = 8.8 + 1.082 = 9.882$ |
|               | X9.92 |        |      | $d+\Delta X(5) = 8.8 + 1.122 = 9.922$ |
|               | X9.96 |        |      | $d+\Delta X(6) = 8.8 + 1.162 = 9.962$ |
| G00           | X...  | Z...   |      |                                       |

When Cutting ZigZag (External Thread)

| G             | X     | Z                             | F    | Remarks                     |
|---------------|-------|-------------------------------|------|-----------------------------|
| G00           | X...  | Z...                          |      |                             |
| G92           | X9.55 | Z $\Delta\Delta.\Delta\Delta$ | F1.0 | $d-DX(1)=10-0.45=9.55$      |
| G01 or<br>G00 |       | W-0.12                        |      | $\Delta W=0.124 \quad 0.12$ |
| G92           | X9.12 | Z $\Delta\Delta.\Delta\Delta$ |      | $d-AX(2)=10-0.818=9.122$    |
| G01 or<br>G00 |       | W(+) $0.06$                   |      | $\Delta W=0.057 \quad 0.06$ |
| G92           | X8.92 | Z $\Delta\Delta.\Delta\Delta$ |      | $d-AX(3)=10-1.075=8.925$    |
| G01 or<br>G00 |       | W-0.05                        |      | $\Delta W=0.048 \quad 0.05$ |
| G92           | X8.76 | Z $\Delta\Delta.\Delta\Delta$ |      | $d-AX(4)=10-1.082=8.758$    |
|               | X8.72 |                               |      | $d-AX(5)=10-1.282=8.718$    |
|               | X8.68 |                               |      | $d-AX(6)=10-1.322=8.678$    |
| G00           | X...  | Z...                          |      |                             |

When Cutting ZigZag (Internal Thread)

| G             | X     | Z                             | F    | Remarks                     |
|---------------|-------|-------------------------------|------|-----------------------------|
| G00           | X...  | Z...                          |      |                             |
| G92           | X9.25 | Z $\Delta\Delta.\Delta\Delta$ | F1.0 | $d+AX(1)=8.8+0.45=9.25$     |
| G01 or<br>G00 |       | W-0.09                        |      | $\Delta W=0.091 \quad 0.09$ |
| G92           | X9.57 | Z $\Delta\Delta.\Delta\Delta$ |      | $d+AX(2)=8.8+0.765=9.565$   |
| G01 or<br>G00 |       | W(+) $0.05$                   |      | $\Delta W=0.05$             |
| G92           | X9.73 | Z $\Delta\Delta.\Delta\Delta$ |      | $d+AX(3)=8.8+0.937=9.737$   |
| G01 or<br>G00 |       | W-0.04                        |      | $\Delta W=0.042$            |
| G92           | X9.88 | Z $\Delta\Delta.\Delta\Delta$ |      | $d+AX(4)=8.8+1.082=9.882$   |
|               | X9.92 |                               |      | $d+AX(5)=8.8+1.122=9.922$   |
|               | X9.96 |                               |      | $d+AX(6)=8.8+1.162=9.962$   |
| G00           | X...  | Z...                          |      |                             |

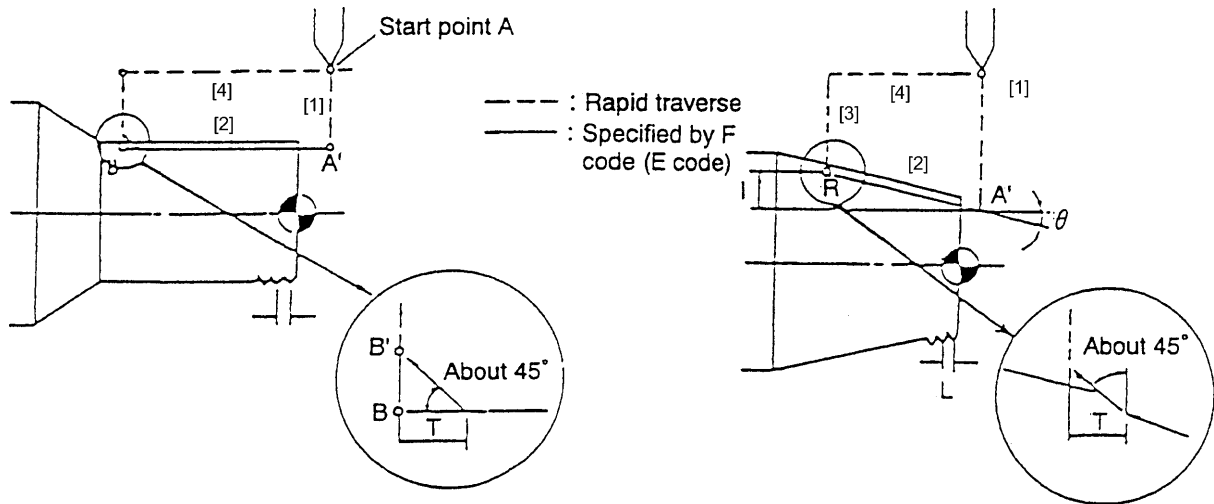
## Thread chamfering

Automatic thread chamfering is enabled in G92 and G76 threading cycles.

### 1. M functions for chamfering selection

M23 ..... chamfering ON (chamfering performed)

M24 ..... chamfering OFF



Details of thread chamfering

Details of thread chamfering

A range of chamfering value  $r$  of thread cutting is  $0 \sim 12.7L$  ( $L$  is a lead of thread) and any value of  $0.1L$  increment can be selected by parameter No.5111. Normally, it has been set to

$1.0L$ .

Thread chamfering angle can be selected by parameter No.5112.

Normally, it is set at  $45^\circ$ .

**Note** 1) When turning on the power, M24 is set.

*It chamfering is required, specify M23 in the block prior to the one which starts threading.*

2) Setting of chamfering width is set at parameter No. 5111.

*Setting of chamfering angle is set at parameter No. 5112.*

3) *The starting point of thread cutting must be designated larger than the end point (B') of chamfering at external thread, or smaller than the end point of chamfering at internal thread, otherwise NC unit issues alarm.*

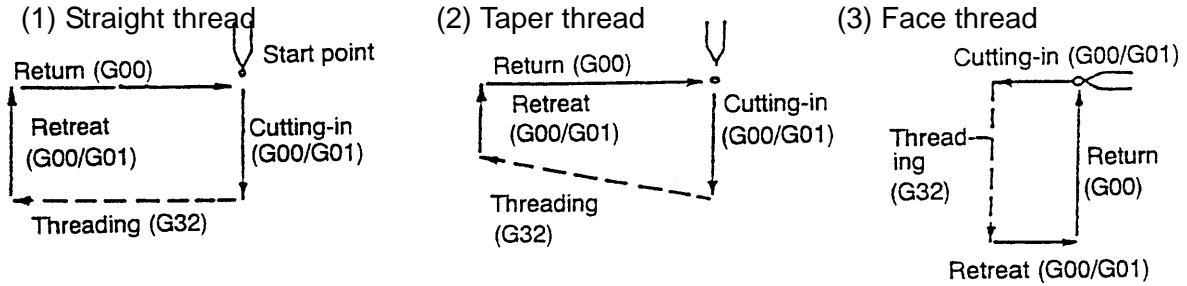
4) *The command M23 should be placed before thread cutting command.*

# 1. G32 Threading

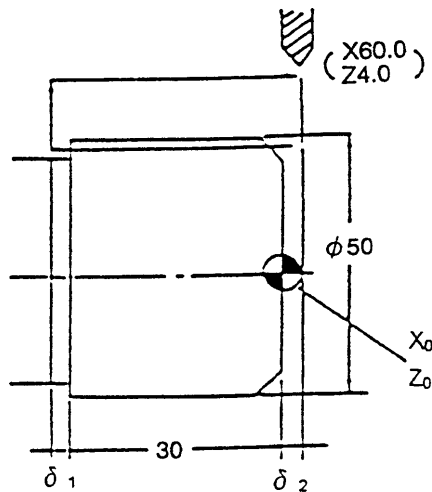
The tool cuts a thread at a feed rate (pitch or lead) specified with F or E as far as a position of X... Z... in the block where G32 was specified.

G32 does not allow cycle operation. Therefore, blocks before and after threading require programs for cutting retreat and return.

- Program cutting retreat and return with G00 or G01.



## (1) Example of straight threading



Lead : 4mm  
 $\delta_1 = 4\text{mm}$   
 $\delta_2 = 2\text{mm}$

M50 x P4.0 thread shown in the left figure

```

N901 T0900
N902 G97 S540 M08
N903 G00 X60.0 Z4.0 M03
N904 X49.12
N905 G32 Z-32.0 F4.0...(W-36.0)
N906 G00 X60.0
N907 Z4.0
N908 X48.39
X909 G32 Z-32.0...(W-36.0)
N910 G00 X60.0
N911 Z4.0
N912 X48.03
N913 G32 Z-32.0...(W-36.0)
N914 G00 .....
.. .....
.. .....
N965 G32 Z-32.0(W-36.0)
N966 G00 X60.0
N967 X200.0 Z200.0 M09
N968 M01

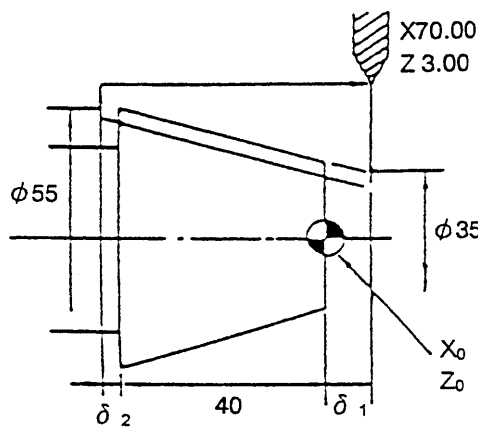
```

\* This example cuts a thread from an outer diameter of 50.0 mm.

- For the threading depth and number of threading times, refer to the number of threading list.
- U... and W... within parentheses specify strokes (incremental programming) from a threading start point to an end point.

Although either programming (incremental or absolute) will do, note that command values will change in case of “G32 U... W...”.

(2) Example of taper threading Program example for  $\phi 35/55$  taper threading shown in the left figure.



Lead : 2mm  
 $\delta_1 = 3\text{mm}$   
 $\delta_2 = 2\text{mm}$

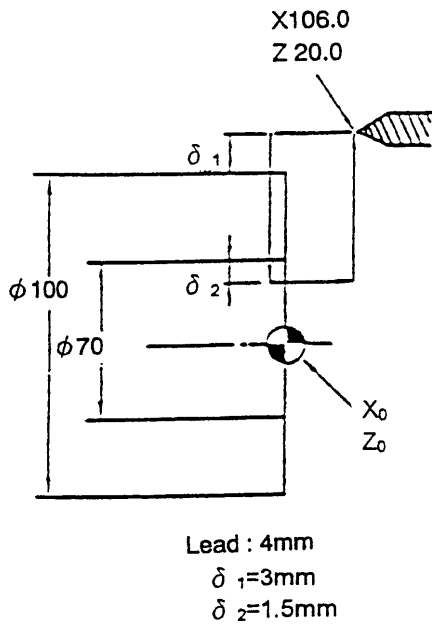
```

N901 T0900
N902 G97 S600 M08
N903 G00 X70.0 Z3.0 M03
N904 X34.33
N905 G32 X54.33 Z-42.0 F2.0 (U20.0 W-45.0)
N906 G00 X70.0
N907 Z3.0
N908 X33.96
N909 G32 X53.96 Z-42.0...(U20.0 W-45.0)
N910 G00 X70.0
N911 Z3.0
N912 X33.72
N913 G32 Z53.72 Z-42.0...(U20.0 W-45.0)
N914 G00 .....
.. .....
.. .....
N940 G32 X52.83 Z-42.0...(U20.0 W-45.0)
N941 G00 X70.0
N942 X200.0 Z200.0 M09
N943 M01

```

(3) Example of face threading

Program example for face threading shown in the left figure, with each depth of cut set to 0.5 mm.



```

N301    T0300
N302 G97  S300 M08
N303 G00  X106.0 Z20.0 M03
N304      Z-0.5
N305 G32  X67.0 F4.0...(U-39.0))
N306 G00  X20.0
N307      X106.0
N308      Z-1.0
X309 G32  X67.0...(U-39.0)
N310 G00  X20.0
N311      X106.0
N312      Z-1.5
N313 G32  X67.0...(U-39.0)
N314 G00  .....
        .. .....
        .. .....
N340 G32  X67.0...(U-39.0)
N341 G00  Z20.0
N342      X200.0 Z200.0 M09
N343      M01
    
```

Instead of “G32 X... Z...”, you can use a command “G32 U... W...”.

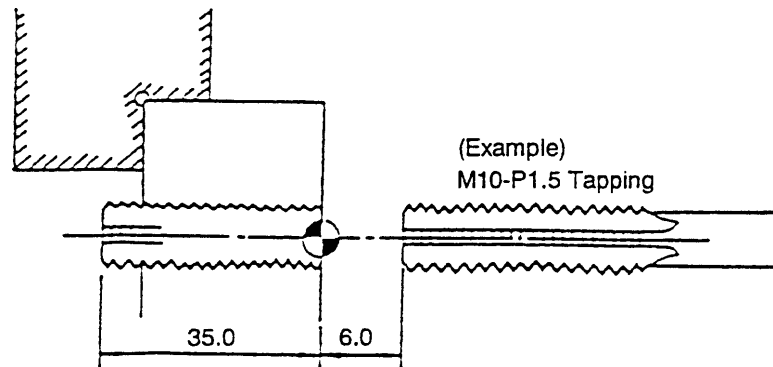
Command values in this case specify strokes from the threading start point to the threading end point.

Refer to the command values “U... W...” within parentheses.

## 2. G32 Tapping

When a tap feed rate (pitch, lead) is specified with G01, if the FEEDRATE OVERRIDE switch on the operation panel is not set to 100%, the feed rate (pitch, lead) specified in the program cannot be obtained because of its change.

To avoid this, if you specify tapping with G32, machining will be performed at the same feed rate as specified in the program for safe operation, ignoring a feed rate override.



Program Example:

```
N601 T0600
N602 G97 S255 M08
N603 G00 X0 Z20.0 M03
N604 G01 Z6.0 F5.0
N605 G32 Z-35.0 F1.5 M05...
N606 G04 U0.5
N607 G32 Z10.0 M04
N608 G04 U0.5
N609 G00 X200.0 Z200.0 M05
N610 M01
```

- N605 G32 Z-35.0 F1.5 M05

The above-mentioned program stops the spindle (M05) when its Z axis is at a position of -35.0 mm. However, when a spindle speed is high, it takes some time for the spindle to stop.

- N607 G32 Z10.0 M04

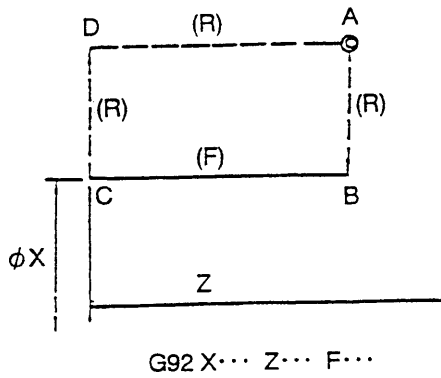
The spindle runs in the reverse direction, and then, the Z axis moves to a position of +10 mm. To retreat the tap, the safer, the bigger a command value for the Z-axis position is.

- When tapping, use a special purpose taper.

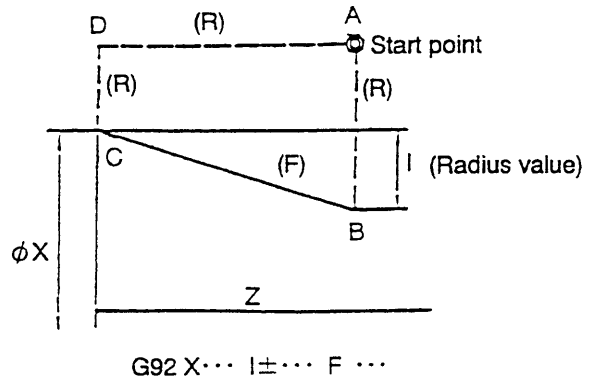
### 3. G92 Threading Cycle

From a threading start point, four actions of cutting-in, threading, retreat and return to the start point can be specified in one block as one cycle.

(1) Straight thread



(2) Taper thread

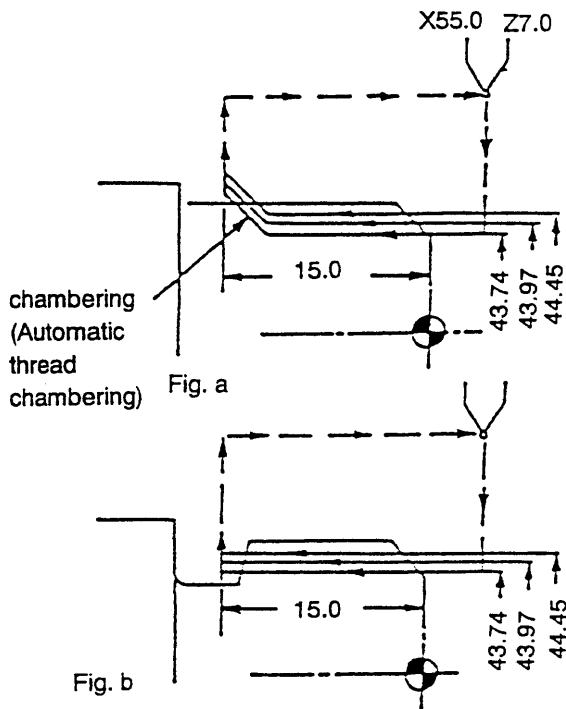


- An incomplete thread part is included within a Z-axis moving range.

R : Rapid traverse  
F : Threading

(1) Example of straight threading

Program example for M45-P1.5 threading (left figure)



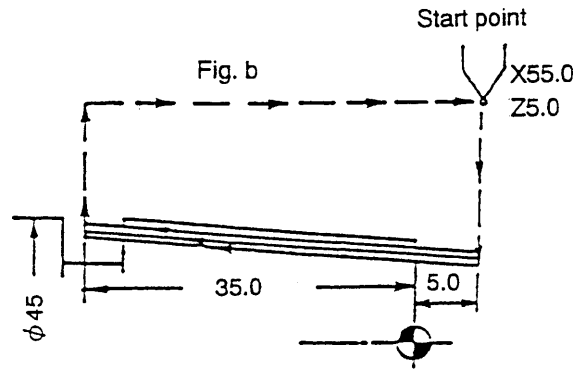
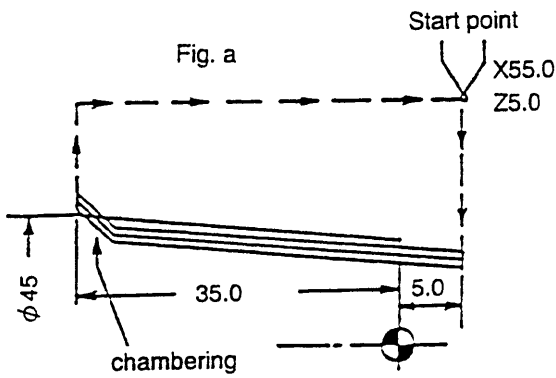
```

N901    T0900
N902G97  S565 M08
N903 G00 X55.0 Z7.0 M03
* N904    M23
N905 G92 X44.45 Z-15.0 F1.5
N906    X43.97
N907    X43.74
N908    X43.54
X909    X43.37
N910    X43.22
N911    X43.18
N912    X43.14
* N913    M24
N914 G00 X200.0 Z200.0
N915    M01
    
```

|      |                           |  |  |                        |
|------|---------------------------|--|--|------------------------|
| N905 | <u>G92</u>                | <u>X44.45</u>                                  | <u>Z-15.0</u>  | <u>F1.5</u>            |
|      | ↙ Threading cycle command | ↙ Cutting diameter dimension for 1st threading | ↙ Threading end position in the longitudinal direction | ↙ Thread pitch or lead |

- The above-mentioned program example executes chambering (automatic thread chambering) as shown in Fig. a above.
- When chambering is not required as shown in Fig. b above, delete blocks marked with “\*” (N904 and N913).
- Since G92 is modal, you can omit it from the next block on, if once specified. In the above-mentioned program, therefore, specify an X-axis cutting diameter dimension after N906 to execute a threading cycle until N912.
- After completing a canned cycle (G92), be sure to cancel it with G00.

(2) Example of taper threading

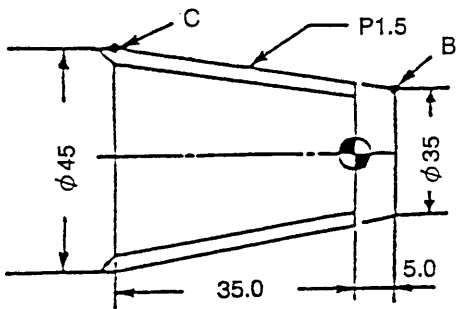


When cutting a taper thread as shown in the left figure, obtain a size of I first.

$$I = \frac{45-40}{2} = 2.5\text{mm}$$

Next, determine a sign (+, -) of I based on a cycle pattern. (direction of the point B viewed from the point C)

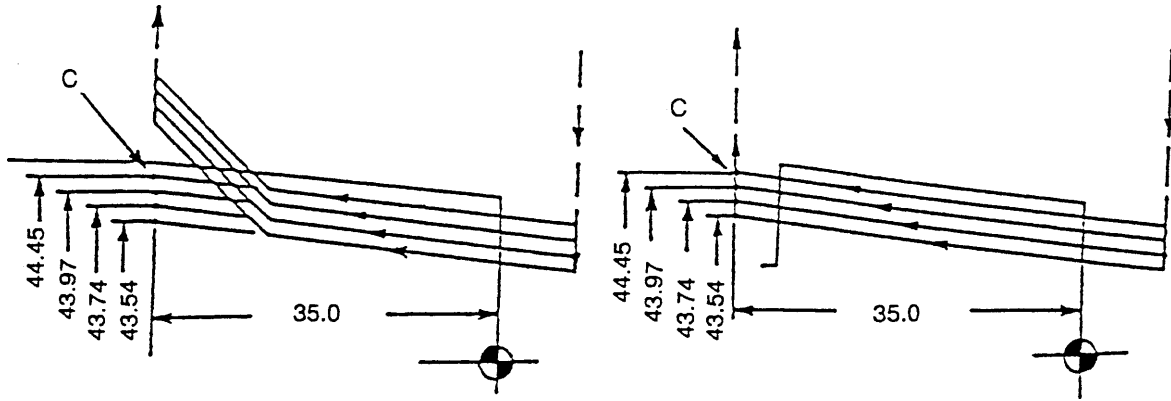
Therefore; I = -2.5



```

N901      T0900
N902 G97  S500 M08
N903 G00  X55.0 Z5.0 M03
* N904      M23
N905 G92  X44.45 Z-35.0 I-2.5 F1.5...(W-40.0)
N906      X43.97
N907      X43.74
N908      X43.54
X909      X43.37
N910      X43.22
N911      X43.18
N912      X43.14
* N913      M24
N914 G00  X200.0 Z200.0
N915      M01
    
```

- Specify the dimension of the point C as to a cutting diameter dimension.
- The program example on a preceding page executes chambering as shown in Fig. a.
- When chambering is not required as shown in Fig. b, delete blocks marked with "\*" (N904 and N913). (Refer to the preceding page.)
- Specify the dimension of the point C as to the cutting diameter dimension for threading.



Point C: A position of end point of threading on the extend line of taper.

|               | Straight thread | Taper thread        |
|---------------|-----------------|---------------------|
| O<br>D<br>(1) | <p>G92 X Z</p>  | <p>G92 X Z I(-)</p> |
| O<br>D<br>(2) | <p>G92 X Z</p>  | <p>G92 X Z I(-)</p> |
| O<br>D<br>(3) |                 | <p>G92 X Z I</p>    |
| O<br>D<br>(4) |                 | <p>G92 X Z I</p>    |

G92 Cycles

|               | Straight thread | Taper thread        |
|---------------|-----------------|---------------------|
| I<br>D<br>(1) | <p>G92 X Z</p>  | <p>G92 X Z I</p>    |
| I<br>D<br>(2) | <p>G92 X Z</p>  | <p>G92 X Z I</p>    |
| I<br>D<br>(3) |                 | <p>G92 X Z I(-)</p> |
| I<br>D<br>(4) |                 | <p>G92 X Z I(-)</p> |

**Note)**

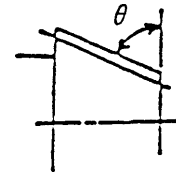
1. A lead becomes inaccurate with a constant surface speed applied.  
Be sure to cut a thread with G97.
2. A cutting feed rate override is always fixed at 100%.
3. If G92 threading cycle is performed in the single block mode, the tool will return to its start point and stop there after completing one cycle.
4. Machine operation cannot be suspended during threading. It stops after executing the first non-threading cycle following the threading mode.
5. A taper thread lead is specified with a length in the longitudinal direction.

[Example] G32 X\_\_ Z\_\_ F4.0

When  $\theta = 45^\circ$ , a lead of Z-axis direction cut by 4mm.

When  $\theta < 45^\circ$ , a lead of X-axis direction cut by 4mm.

Therefore, when  $\theta=30^\circ$ , a lead of Z-axis direction becomes  $4 \times \tan 30^\circ = 2.31\text{mm}$ .

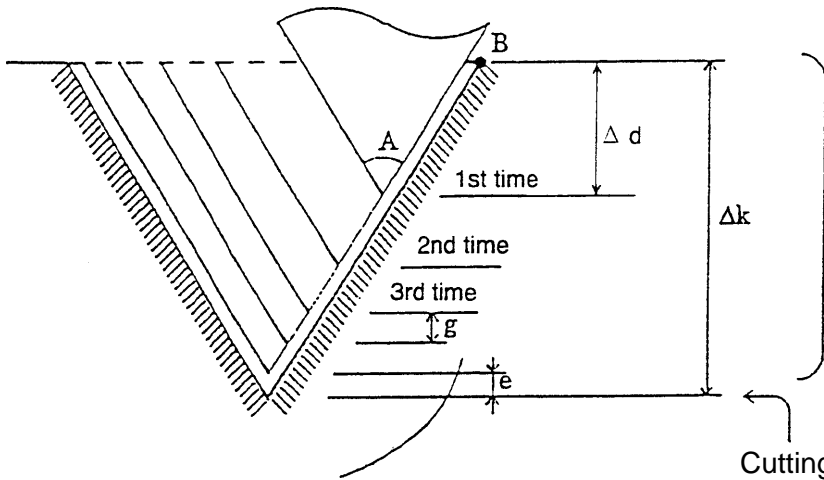


6. Tool nose radius compensation is not allowed in threading.
7. The lengths  $\delta_1$  and  $\delta_2$  of an incomplete thread part are determined by the spindle speed and lead as mentioned above. Therefore, when cutting one screw, the spindle speed must be kept constant to the last.  
(A thread section would be shifted.)



Cutting method

(1) Constant cutting amount, single edge (P1 designation)



Parameter No.5149  
(In case of 1st cutting amount <g)

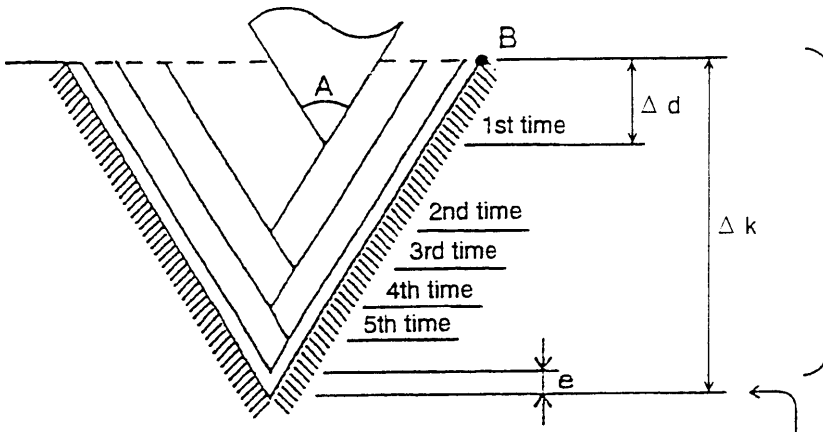
In H command, the cutting is completed with the process of H times.

Cuttings are repeated the number of times as set by Parameter No.5129.

(Finishing)

- 1st cutting amount .....  $\Delta d$
- 2nd cutting amount .....  $\Delta d\sqrt{2}$
- 3rd cutting amount .....  $\Delta d\sqrt{3}$
- 4th cutting amount .....  $\Delta d\sqrt{4}$
- 5th cutting amount .....  $\Delta d\sqrt{5}$

(2) Constant cutting amount, Staggered cutting (P2 designation)



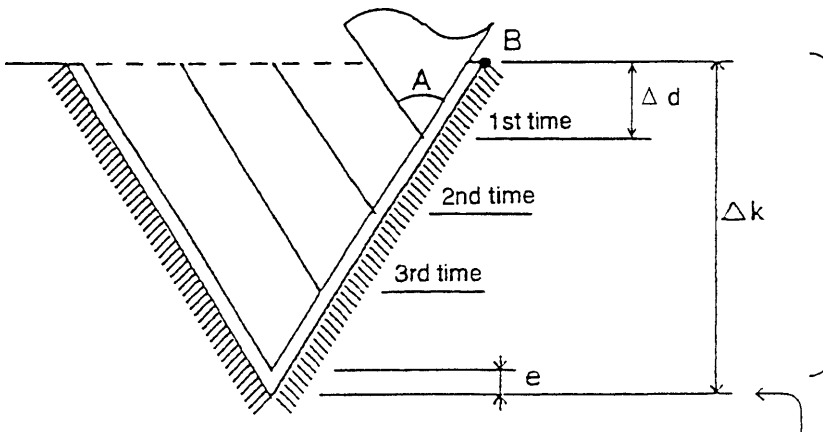
In H command, the cutting is completed with the process of H times.

Cuttings are repeated the number of times as set by Parameter No.5129.

- 1st cutting amount .....  $\Delta d \cdot \frac{\sqrt{2}}{2}$
- 2nd cutting amount .....  $\Delta d \sqrt{2}$
- 3rd cutting amount .....  $\Delta d \cdot \frac{\sqrt{2} + \sqrt{4}}{2}$
- 4th cutting amount .....  $\Delta d \sqrt{4}$
- 5th cutting amount .....  $\Delta d \cdot \frac{\sqrt{4} + \sqrt{6}}{2}$
- 5th cutting amount .....  $\Delta d \sqrt{6}$

(Finishing)

(3) Constant cutting amount, Single edge cutting P3 designation)



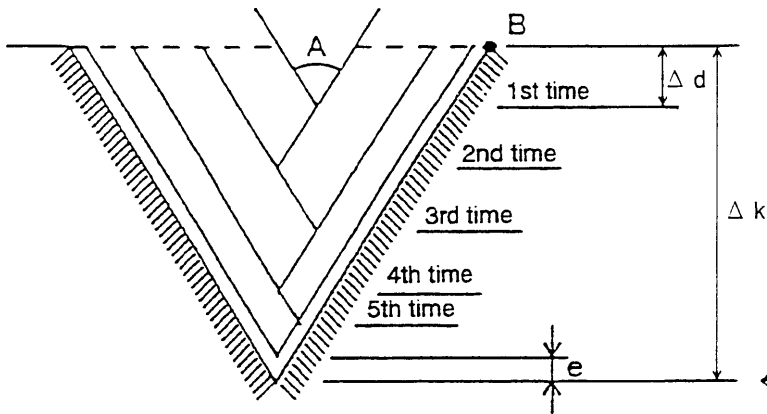
In H command, the cutting is completed with the process of H times.

Cuttings are repeated the number of times as set by Parameter No.5129.

- 1st cutting amount .....  $\Delta d$
- 2nd cutting amount .....  $\Delta d \cdot 2$
- 3rd cutting amount .....  $\Delta d \cdot 3$
- 4th cutting amount .....  $\Delta d \cdot 4$

(Finishing)

(4) Constant cutting amount, Staggered cutting (P4 designation)



In H command, the cutting is completed with the process of H times.

Cuttings are repeated the number of times as set by Parameter No.5129.

- 1st cutting amount .....  $\Delta d$
- 2nd cutting amount .....  $\Delta d \cdot 2$
- 3rd cutting amount .....  $\Delta d \cdot 3$
- 4th cutting amount .....  $\Delta d \cdot 4$

(Finishing)

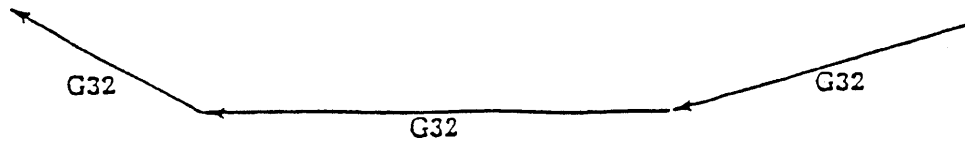
**Note)**

- 1) In case of single block, single-block-stop at points A and A'.
- 2) Finish margin "e" is set by Parameter No.5149.
- 3) The No. of times of last finishing can be set by Parameter No. 5129.
- 4) When commanding P2, P4 (staggered cutting), "H" is to be commanded by an even number (2,4,6,.....).
- 5) The last 2 times of rough cuttings in P2, P4, a half of the remaining margin is to be cut off each time.
- 6) In entire process of G76, spindle override becomes ineffective.

### 2-3-21 Continuous Thread Cutting

Continuous thread cutting in which thread cutting blocks are continuously commanded is available.

As it is controlled so that synchronism with the spindle will be shifted minimumly at a joint of blocks, it is possible to cut a special thread whose lead or shape changes halfway machining.



Even when repeatedly threading the same position, changing each depth of cut, it is properly machined without breaking threads.

### 2-3-22 G34 Variable Lead Thread Cutting (Option)

A variable lead thread cutting can be done by commanding of increase or decrease amount per revolution of the thread.

(1) Form of command

G34  $\alpha$   $\beta$  F K ;

$\alpha$ ,  $\beta$  : Any axis

F : A lead at starting time of thread cutting of longitudinal direction.

K : Increase amount per revolution (When it is negative, it is decrease amount)

Command range of K is as follows:

Metric command :  $\pm 0.001 \sim \pm 99999.999$  (mm/rev)

Inch command :  $\pm 0.0001 \sim \pm 9999.9999$  (inch/rev)

**Note)** Command unit of K is decided by parameter No. 1007.

(2) Program example

Straight thread cutting with variable lead

(Lead of starting time of thread cutting : 2.0mm, Increase amount per revolution : 0.3mm)

G34 W-10. F2.0 K0.3 ;

**Note** 1) If K is omitted in the command of G34, it becomes even lead thread cutting of command of G32.

## 2-3-23 Multi-thread Cutting (Option)

Cutting of multiple thread is performed by synchronous feed of starting pulse from the spindle plus generator and start the other thread from the spindle rotate by designated degree after starting pulse.

Command

G32 X(U) .... Z(W) .... F .... Q .... ;

G92 X(U) .... Z(W) .... I .... Q .... F ....

By the command above mentioned, thread cutting is performed up to X(U), Z(W) by lead F from rotating the spindle designated angle by Q from starting pulse of the spindle pulse generator.

The address Q commanded by multi-thread cutting is as follows:

Least increment  $1^\circ$

Commanding range 0 Q  $360^\circ$

Don't enter a decimal point in Q command.

**Note** 1) Unit of shifted angle can be chosen by parameter setting.  $1^\circ$  or  $0.001^\circ$ .

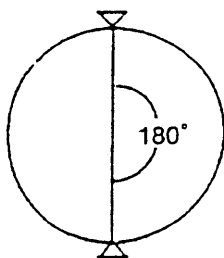
But when  $0.001^\circ$  chosen, actual resolving power is  $0.088^\circ$ .

2) When macro parameter is used in Q, choose  $0.001^\circ$  as unit of Q.

Number of thread of multi-thread and command

It is a principle that the cutting start point on the circumference becomes the point that the circumference ( $360^\circ$ ) is equally divided by the number of thread.

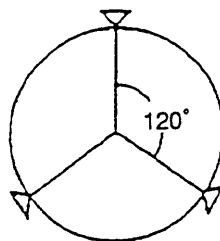
Double thread screw  
Cut-in start point



1st thread: Without  
Q command  
2nd thread: Q180

(a) 2 thread screw

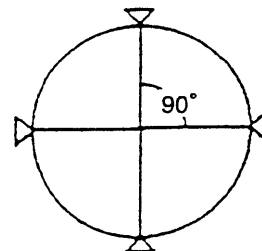
Three thread screw  
Cut-in start point



1st thread: Without  
Q command  
2nd thread: Q120  
3rd thread: Q240

(b) 3 thread screw

Four thread screw  
Cut-in start point



1st thread: without  
Q command  
2nd thread: Q90  
3rd thread: Q180  
4th thread: Q270

(c) 4 thread screw

## 2-3-24 G150, G151, G152 Groove Width Compensation

Groove width compensation is changing the tool point by shifting the coordinate system to the amount of tool width through reading the data of tool width and tool point in the tool layout screen by command of G151.

(Shift to the amount of tool width x 2 for end surface.)

G150 Groove compensation, OFF

G151 Groove compensation for end surface, ON

G152 Groove compensation for OD/ID, ON

Compensating amount of tool width to be set at H data in the tool compensation screen.

In case of using grooving tool on T03 with tool width 2.0mm.

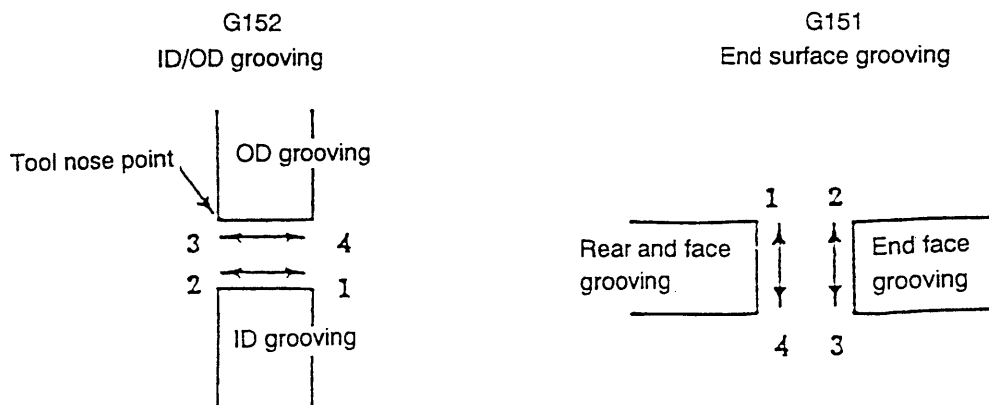
| TOOL(OFFSET) |       |      |       | 05000 |        |      |       |       |         |      |       |
|--------------|-------|------|-------|-------|--------|------|-------|-------|---------|------|-------|
|              |       |      |       | N0004 |        |      |       |       |         |      |       |
| SHAPE        |       | WEAR |       | SHAPE |        | WEAR |       | SHAPE |         | WEAR |       |
| 01           |       |      |       | 02    |        |      |       | 03    |         |      |       |
| X            |       | U    | 0.000 | X     | 1.999  | U    | 0.000 | X     | 100.000 | U    | 0.000 |
| Y            | 2.000 | V    | 0.000 | Y     | 20.000 | V    | 0.000 | Y     | 200.000 | V    | 0.000 |
| Z            | 2.000 | W    | 0.000 | Z     | 30.000 | W    | 0.000 | Z     | 200.000 | W    | 0.000 |
| R            | 0.000 | Q    | 0.000 | R     | 2.499  | Q    | 0.000 | R     | 0.000   | Q    | 0.000 |
| T            | J     |      |       | T     | Z      |      |       | T     | S       |      |       |
| H            | 0.000 | J    | 0.000 | H     | 0.000  | J    | 0.000 | H     | 2.000   | J    | 0.000 |

Tool width 2.0

If groove width compensation (G151, G152) is commanded, tool point will be shifted as follows.

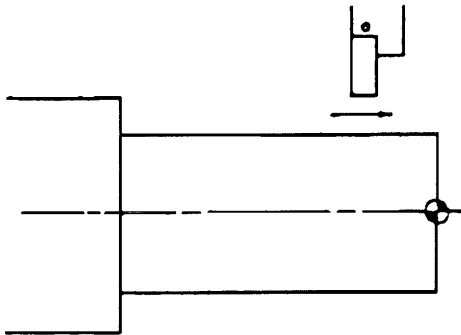
Tool width compensation is canceled by G150 or T command.

Refer to "3. Tool nose R compensation automatic calculating function" about tool nose point.



(Example 1) In case of OD grooving tool (tool nose point 3)

Tool width 6.0



| POSITION |         | ABSOLUTE |         | DIST TO GO |       |
|----------|---------|----------|---------|------------|-------|
| ABSOLUTE |         | ABSOLUTE |         | N          |       |
| X        | 50.000  | X        | 50.000  | X          | 0.000 |
| Z        | -26.000 | Z        | -26.000 | Z          | 0.000 |
|          |         | RELATIVE |         | MACHINE    |       |
| U        | 0.000   | X        | 0.000   |            |       |
| W        | 0.000   | Z        | 0.000   |            |       |

G152 ON



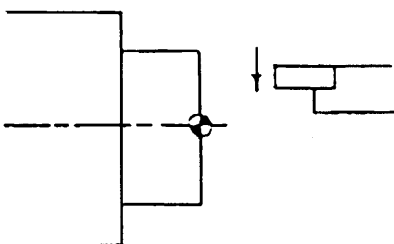
- The coordinate system of the Z-axis is shifted by the tool width amount.
- The tool nose point is shifted internally from 3 to 4.

| POSITION |         | ABSOLUTE |         | DIST TO GO |       |
|----------|---------|----------|---------|------------|-------|
| ABSOLUTE |         | ABSOLUTE |         | N          |       |
| X        | 50.000  | X        | 50.000  | X          | 0.000 |
| Z        | -20.000 | Z        | -26.000 | Z          | 0.000 |
|          |         | RELATIVE |         | MACHINE    |       |
| U        | 0.000   | X        | 0.000   |            |       |
| W        | 0.000   | Z        | 0.000   |            |       |

Coordinate system changes

(Example 2) In case of end grooving tool (tool nose point 2)

Tool width 5.0



| POSITION |        | ABSOLUTE |        | DIST TO GO |       |
|----------|--------|----------|--------|------------|-------|
| ABSOLUTE |        | ABSOLUTE |        | N          |       |
| X        | 50.000 | X        | 50.000 | X          | 0.000 |
| Z        | 20.000 | Z        | 20.000 | Z          | 0.000 |
|          |        | RELATIVE |        | MACHINE    |       |
| U        | 0.000  | X        | 0.000  |            |       |
| W        | 0.000  | Z        | 0.000  |            |       |

G151 ON



- The coordinate system of the X-axis is shifted by tool width  $\times 2$ .
- The tool nose point is shifted internally from 2 to 3.

Coordinate system changes

| POSITION |        | ABSOLUTE |        | DIST TO GO |       |
|----------|--------|----------|--------|------------|-------|
| X        | 40.000 | X        | 40.000 | X          | 0.000 |
| Z        | 20.000 | Z        | 20.000 | Z          | 0.000 |
|          |        | RELATIVE |        | MACHINE    |       |
| U        | 0.000  | X        | 0.000  |            |       |
| W        | 0.000  | Z        | 0.000  |            |       |

- Note**
- 1) Except when the tip point is at 1~4, alarm is produced.
  - 2) With G151/G152 are continuously commanded in a program, the current correction is canceled and a new one is applied.
  - 3) With a spindle shift command given simultaneously with G150~G150, an alarm is produced.
  - 4) Correction is canceled on resetting while in correction, where, however, no change takes place in the coordinate axes.

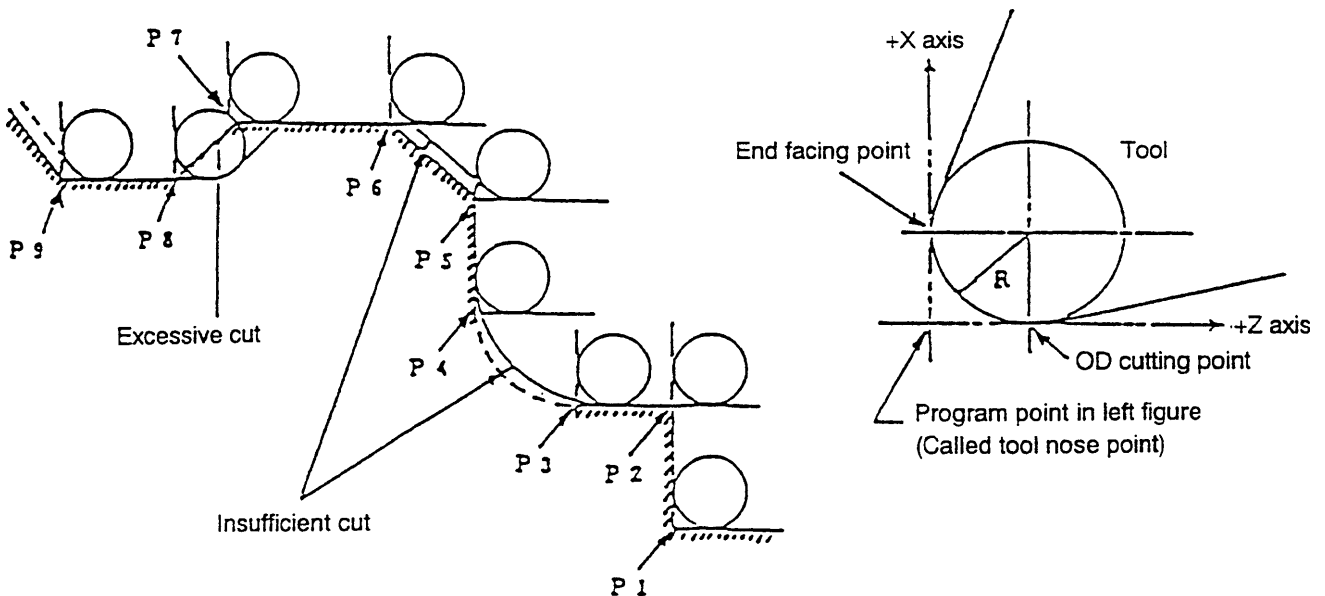
### 3. AUTOMATIC CALCULATING FUNCTION OF TOOL NOSE RADIUS COMPENSATION

#### 3-1 Outline

Normally, a tool nose is programmed as one point. However, an actual tool has nose R.

Although it can be ignored when cutting in parallel to axes, such as an end face, outer diameter and inner diameter, when chamfering or cutting a slope and circular arc, the workpiece tends to be cut insufficiently or excessively due to this nose R.

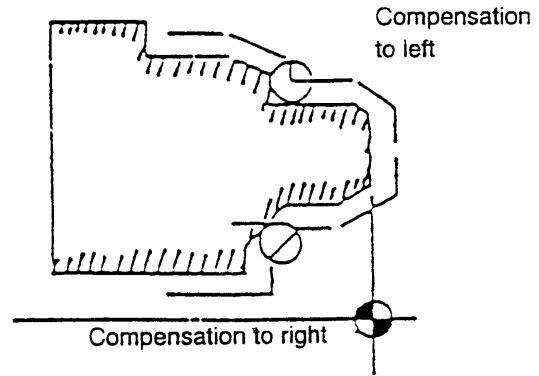
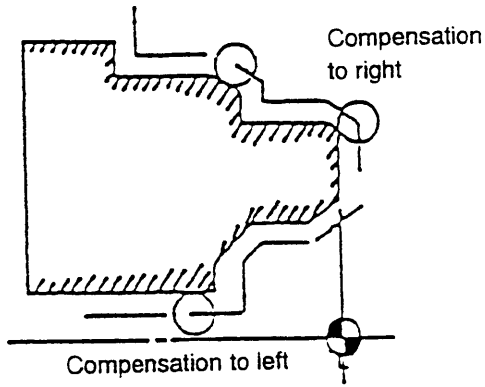
Tool nose radius compensation automatic calculating function makes operation processing done inside the NC unit, automatically controls the tool nose and prevents insufficient and excessive cutting.



P1 through p9 are program points.

When tool nose radius compensation is under way, the following 3 states exist:

| State                     | Tool route  |
|---------------------------|---|
| Compensation cancel state | The tool moves on a programmed route.                                     |
| Compensation to left      | The tool moves on the left side of a programmed route advance direction.  |
| Compensation to right     | The tool moves on the right side of a programmed route advance direction. |



### 3-2 Preparation to Execute the Automatic Calculating Function of Tool Nose Radius Compensation

The following setting is required to do a nose R compensation.

These are set in the tool offset screen.

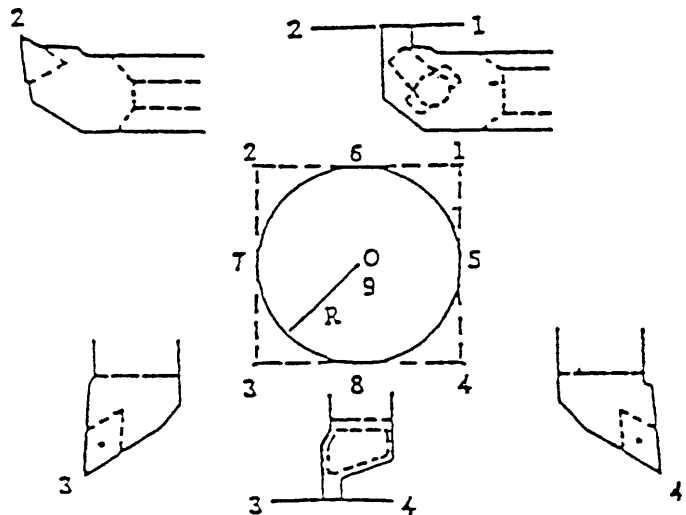
1. Tool tip point (refer to the lower sketch) ... Input at the T\_ of tool offset screen.
2. Size of nose R ..... Input at the R\_ of tool offset screen.

Input of a tool tip point is done by inputting of designated address of tool as lower sketch.

#### Tool tip point

A programmed point in nose R is called like this.

A program or method of nose R compensation become completely different depend on the setting method.



### 3-3 Three Conditions of Nose Radius Compensation

When performing tool nose radius compensation, its program starts from a tool nose radius compensation cancel state and proceeds to a tool nose radius compensation state via a start-up state, and then, it returns to the initial compensation cancel state.

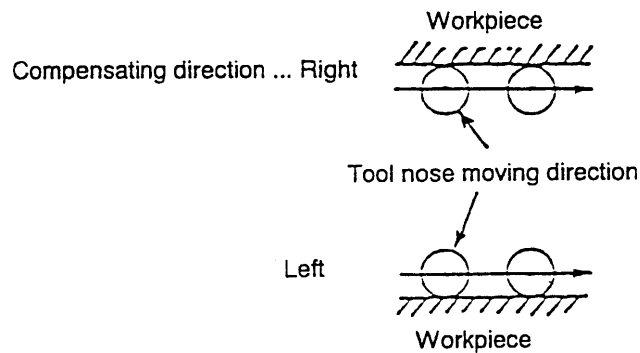
These are divided into three states and each block is called as follows:

1. Start-up block .... Block changing over from rapid traverse to cutting feed (G00 → G01)
2. Tool nose radius compensation block .... Continuous block for cutting feed (G01•G02•G03  
↔ G01•G02•G03)
3. Compensation cancel block .... Block changing over from cutting feed to rapid traverse (G01  
→ G00)

### 3-3-1 Tool Nose Radius Compensation Block (During Cutting)

A tool nose radius compensating method during cutting is determined by the tool nose point and a tool nose moving direction. A list is given below.

- Compensating direction by tool nose point and tool nose moving direction



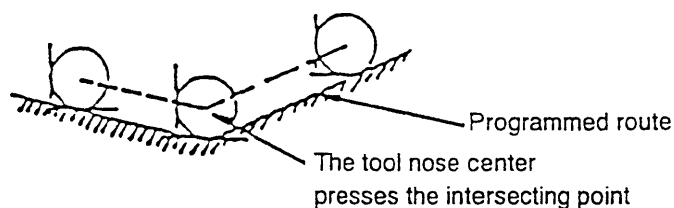
: Follows the compensating direction in a preceding block (because the compensating direction cannot be determined).

: Does not compensate the tool nose radius (the tool nose moves as programmed).

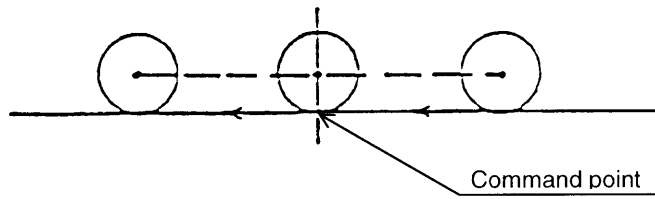
| Moving direction \ Nose point | Moving direction |       |       |       |       |       |       |       |
|-------------------------------|------------------|-------|-------|-------|-------|-------|-------|-------|
|                               | →                | ↗     | ↑     | ↖     | ←     | ↙     | ↓     | ↘     |
| 1                             | Right            |       | Left  | Left  | Left  |       | Right | Right |
| 2                             | Right            | Right | Right |       | Left  | Left  | Left  |       |
| 3                             | Left             |       | Right | Right | Right |       | Left  | Left  |
| 4                             | Left             | Left  | Left  |       | Right | Right | Right |       |
| 5                             |                  | Left  | Left  | Left  |       | Right | Right | Right |
| 6                             | Right            | Right |       | Left  | Left  | Left  |       | Right |
| 7                             |                  | Right | Right | Right |       | Left  | Left  | Left  |
| 8                             | Left             | Left  |       | Right | Right | Right |       | Left  |
| 0•9                           |                  |       |       |       |       |       |       |       |

“During cutting” means to be in the G01/G02 G03 mode.

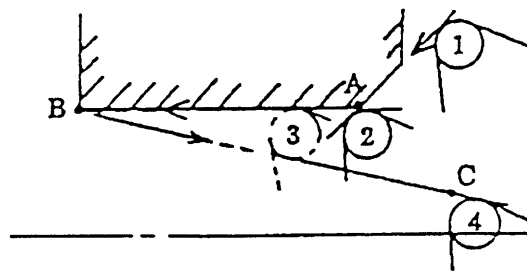
- a) When a tangent angle is 180° or less (inner corner), an intersecting point is operated and the tool nose center moves to that intersecting point.



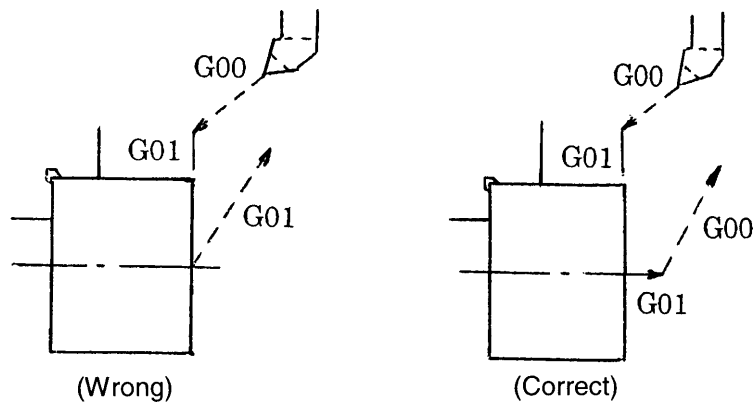
b) When the tangent angle is  $180^\circ$ , the tool nose center comes on the normal of a command point.



c) Do not command a wedge shape with an obtuse angle



In case of the path  $A \rightarrow B \rightarrow C$  is commanded by G01, a tool tip does not move further than condition [3] even a command of point B.



In case of simultaneous two axis moving on cutting feed, tool nose does not reach to the commanded point.

### 3-3-2 Start-up Block and Compensation Cancel Block (Approach/Retreat)

Concretely, the start-up block and compensation cancel block refer to blocks changing over from G00 to G01 (approach) and G01 to G00 (retreat).

#### How to determine the compensating direction in approaching/retreating

[1] i) When a specified stroke is  $| \frac{X}{2} | > | Z |$ , create a virtual line parallel to the Z axis.

(  $| \frac{X}{2} | > | Z |$  means the case when a moving axis direction makes an angle larger

than 45° with the Z axis.)

ii) When a specified stroke is  $| \frac{X}{2} | < | Z |$ , create a virtual line parallel to the X axis.

(  $| \frac{X}{2} | < | Z |$  means the case when a moving axis direction makes an angle of 45° or less with the Z axis.)

[2] Viewing the compensating direction of the moving axis, determine either “+” or “-” (determination of a virtual line direction).

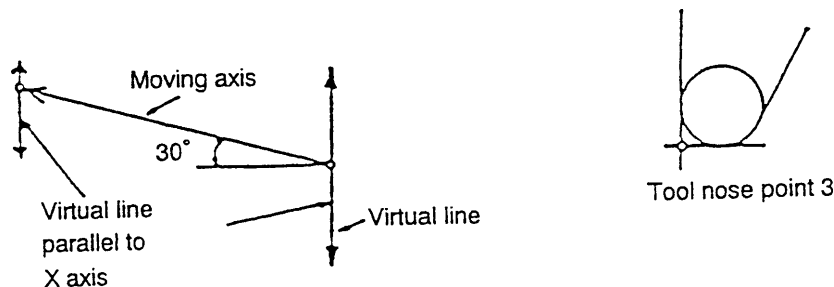
[3] Determine the compensating direction (right or left) to the virtual line.

[4] Calculate the intersecting point.

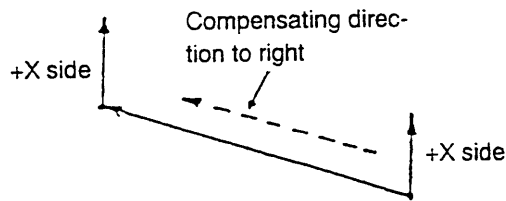
**Note)** When you cannot determine the compensating direction to the virtual line in [3] (when the tool nose point is 5 through 8), select the same compensating direction as in [2].

#### **Example 1) For the tool nose point 3**

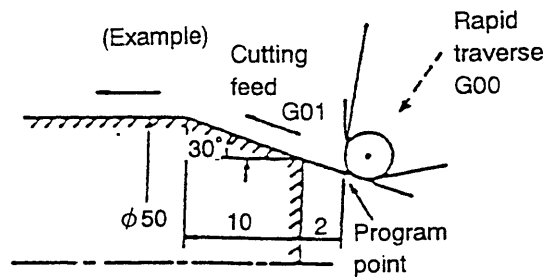
[1] Since  $| \frac{X}{2} | < | Z |$ , create the virtual line parallel to the X axis.



[2] Determine a virtual line direction in the same direction as the compensating direction of the moving axis (+X side because the compensating direction is to the right).

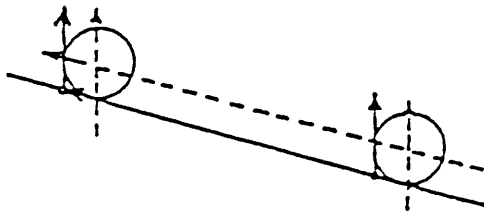


[3] Determine the compensating direction of the virtual line, and then, the intersecting point.



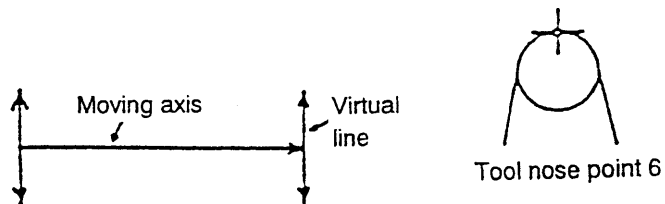
```

N100 T0100 M40 ;
N101 G97 S1060 M08 ;
N102 X36.14 Z2.0 M03 ;
N103 G01 G96 X50.0 Z-10.0 F0.2
S120 ;
N104 Z-XXXX ;
    
```

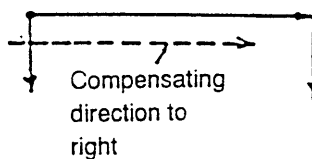


**Example 2) For the tool nose point 6**

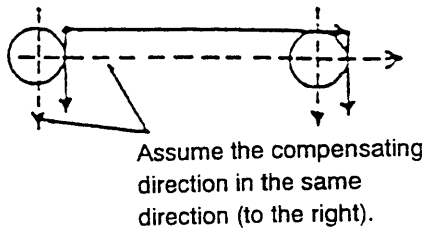
[1] Since  $|\frac{X}{2}| < |Z|$ , create the virtual line parallel to the X axis.



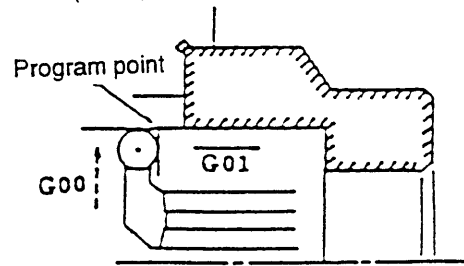
[2] Determine a virtual line direction to the -X side, because the compensating direction of the moving axis is to the right.



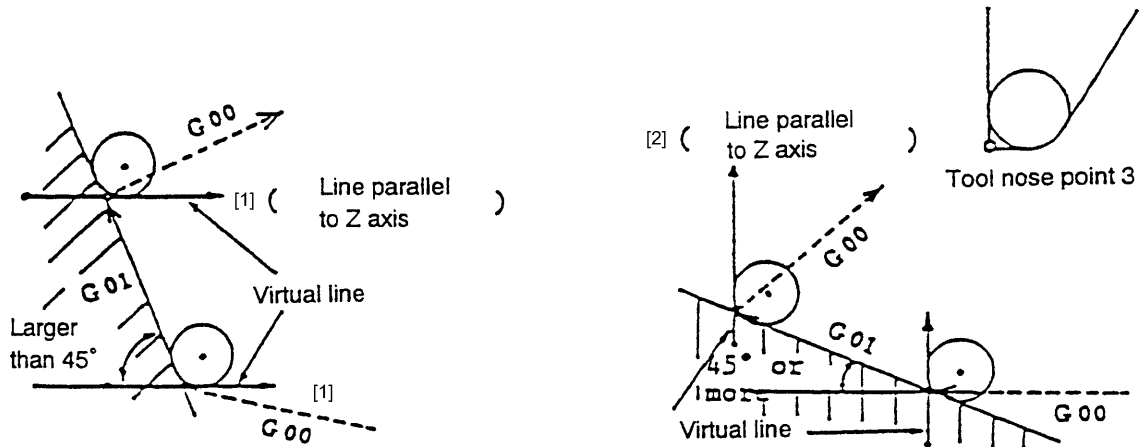
\*[3] Since the compensating direction of the virtual line cannot be determined, assume it in the same direction as the compensating direction of the moving axis.



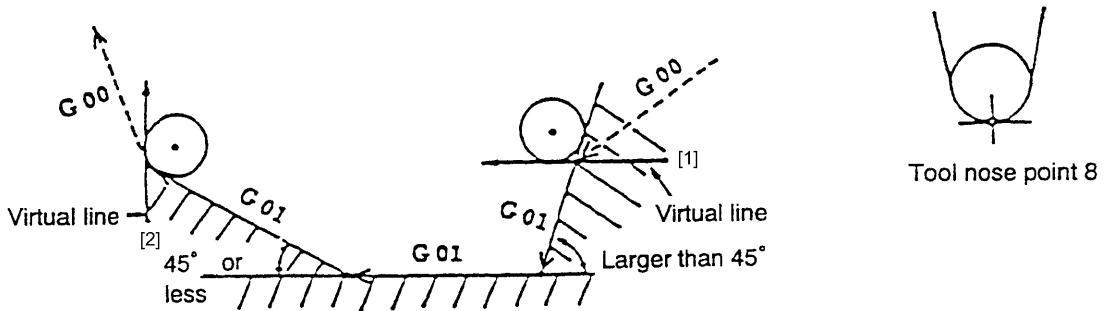
(Example)



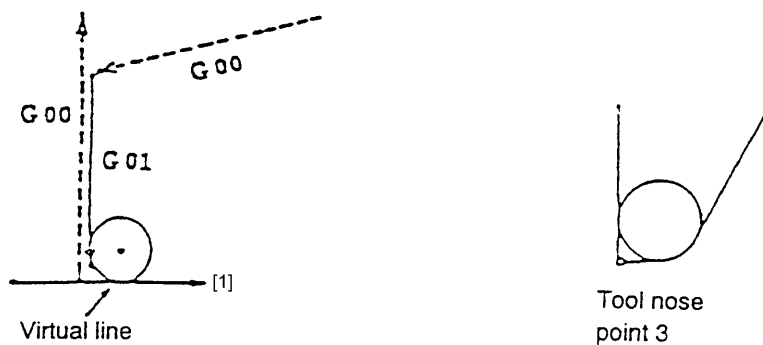
**Example 3) A. For the tool nose point 3**



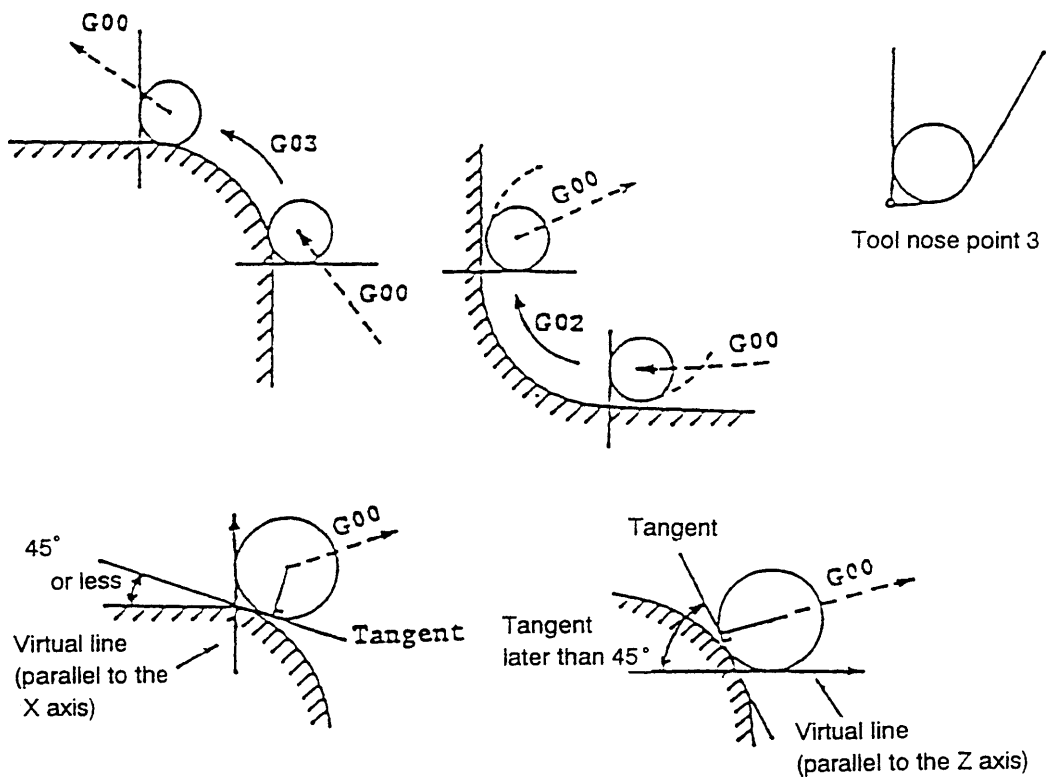
**B. For the tool nose point 8**



**C. For the tool nose point 3 in grooving (when returning only a single axis)**



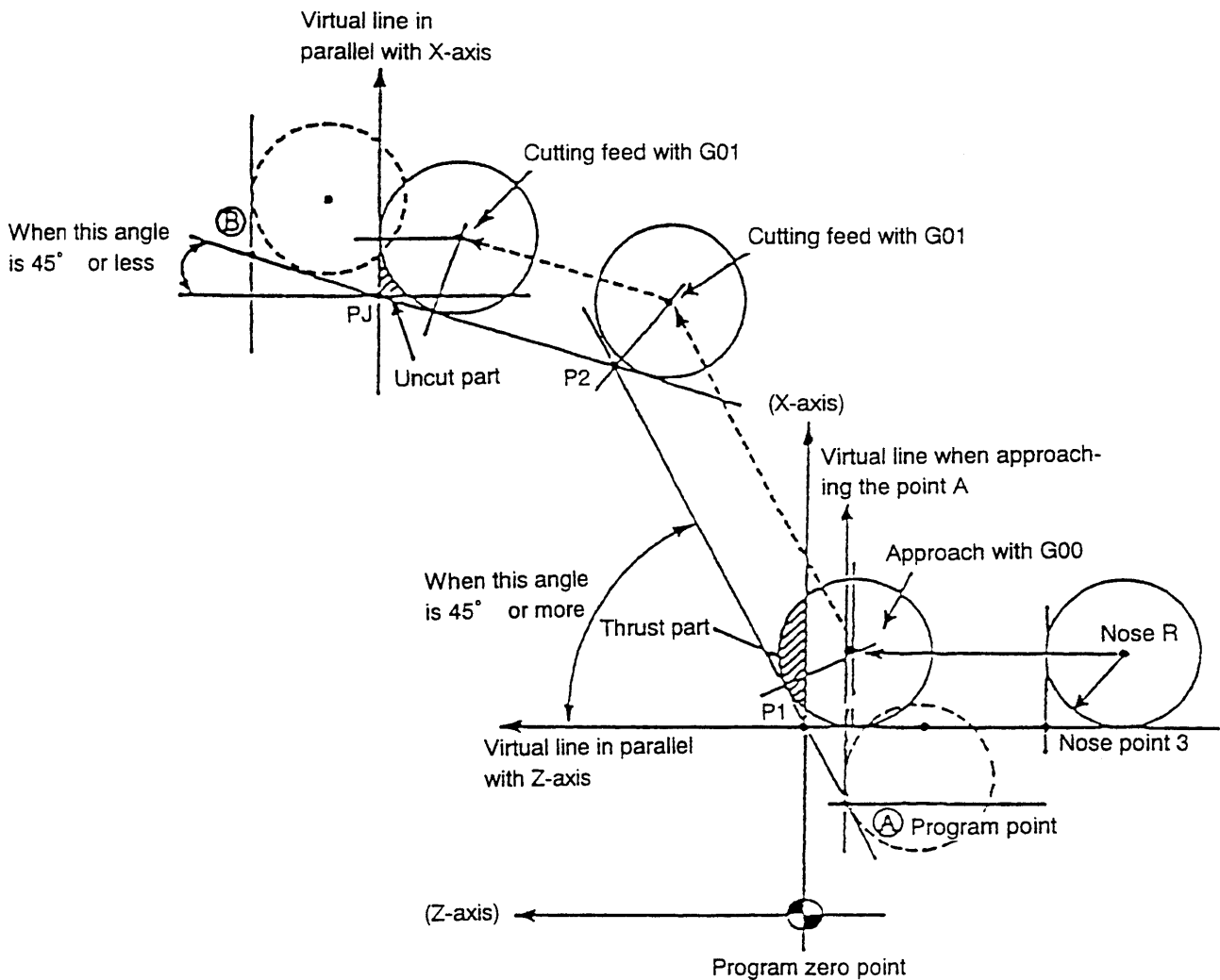
D. For the tool nose point in approaching to an arc and retreating



When commanding either of the following modes in the status of compensation currently the compensation is canceled.

- [1] Axial travel is performed in the plane by G00.
- [2] Coordinate system setting by T command.

### 3-4 Caution Point of Approach to Workpiece



In the figure above, when the tool approach P1 by G00 then P2 by federate, tool point may over cut against command point because tool nose R compensation is executed at G01 block.

In addition, after cutting feed to P3, tool nose R compensation is turned off in G00 block, the uncut part may occur.

Therefore, it is necessary to take care not of have this type of shape by checking the program point for approach and escape.

As to countermeasures, set program points to points A and B in the figure.

### 3-5 Tool Nose Radius Compensation to Direct Designation G Code (G141, G142)

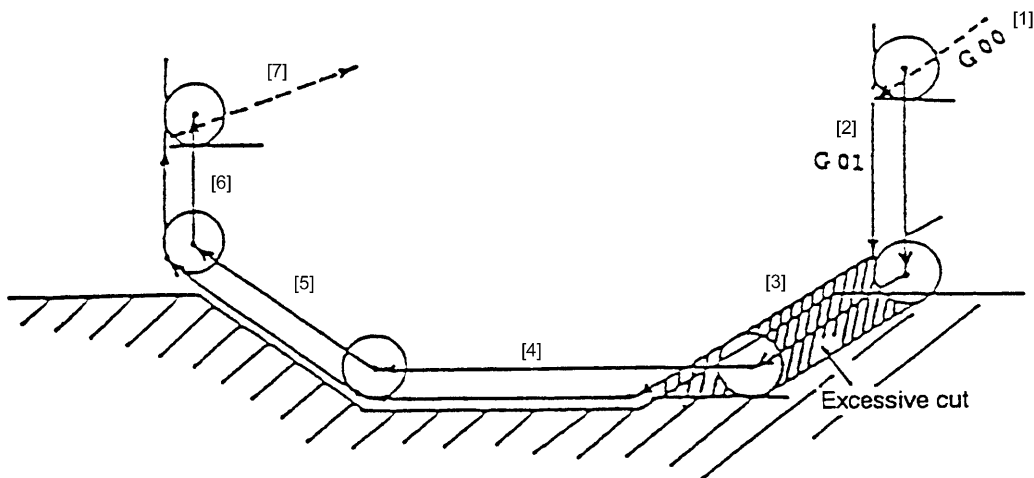
In indenting, there is no particular problem for finishing. In roughing, however, specify a compensation direction with the following G codes:

G141 Tool nose radius compensation direction to left

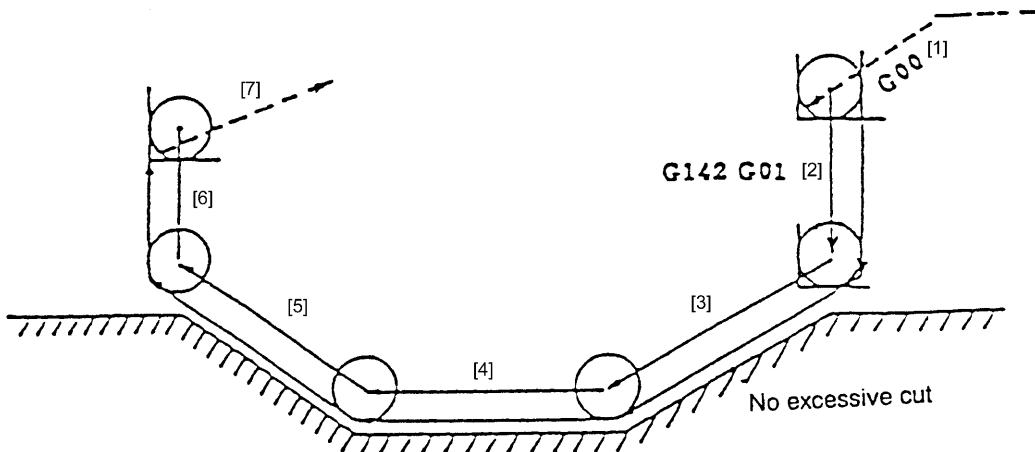
G142 Tool nose radius compensation direction to right

Effective designated one block only.

#### Example 1) For the tool nose point 3



#### Example 2) For the tool nose point 3



#### Program

##### Example 1

```
[1] G00 X_ Z_
[2] G01 X_ F...
[3] X_ Z_
[4] Z_
[5] X_ Z_
[6] X_
[7] G00 X_ Z_
```

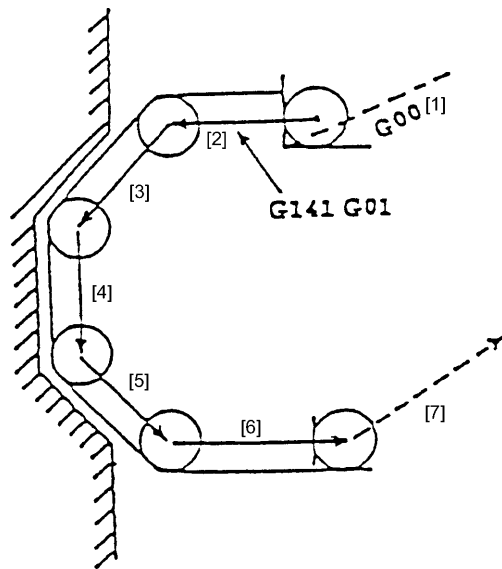
##### Example 2

```
[1] G00 X_ Z_
[2] G142 G01 X_ F...
[3] X_ Z_
[4] Z_
[5] X_ Z_
[6] X_
[7] G00 X_ Z_
```

In Example 1, the command [2] moves the tool in a direction of “↓”, the compensation direction is specified to the left, assuming this as end facing. For the command [3], as the compensation direction follows the previous block because this command moves the tool in a direction of “↓”, excessive cutting is caused. To prevent this in indenting, specify G142 (compensation to right) as shown in Example 2 to specify the compensation direction to the right.

This solution also applies to end face indenting. (Example 3)

**Example 3)**



```
[1] G00 X__ Z__
[2] G141 G01 Z__ F...
[3] X__ Z__
[4] X__
[5] X__ Z__
[6] Z__
[7] G00 X__ Z__
```

**Overall Precautions**

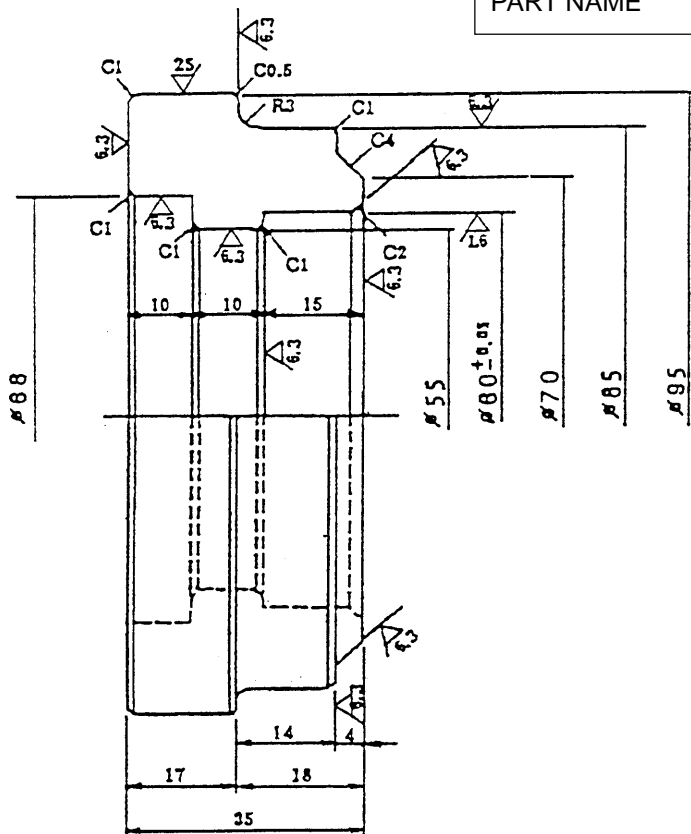
1. An error will result if you specify a tool move inside an arc smaller than a tool nose radius or a groove width up to 2 times or less of the tool nose radius while executing automatic tool nose R compensation.
2. Tool nose R compensation is not performed by data input operation.  
It is available only by a program command.
3. During tool nose R compensation, if you continuously specify 3 or more blocks which do not have any move commands, compensation will be temporarily canceled.
4. In case of axis move command and T code is placed in the same block during tool nose R compensation, canceled the previous coordinate system after completion of movement then perform new coordinate system setting.
5. In the start-up block, you cannot specify a moving direction for which the compensation direction cannot be determined. If specified, excessive or insufficient cutting may be caused.

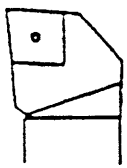
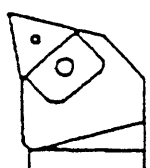
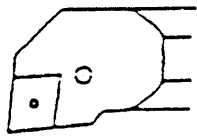
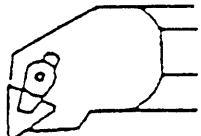
# 4. PROGRAM EXAMPLE (NC PROGRAM)

## 4-1 Chuck Work

### 4-1-1 Machining Drawing

|                       |              |                   |
|-----------------------|--------------|-------------------|
| EXAMPLE               | <b>SEIKI</b> | CNC LATHE         |
| PROCESS : 2ND NC UNIT |              | TOOL LAYOUT SHEET |
| PART NAME             |              | MATERIAL S48C     |

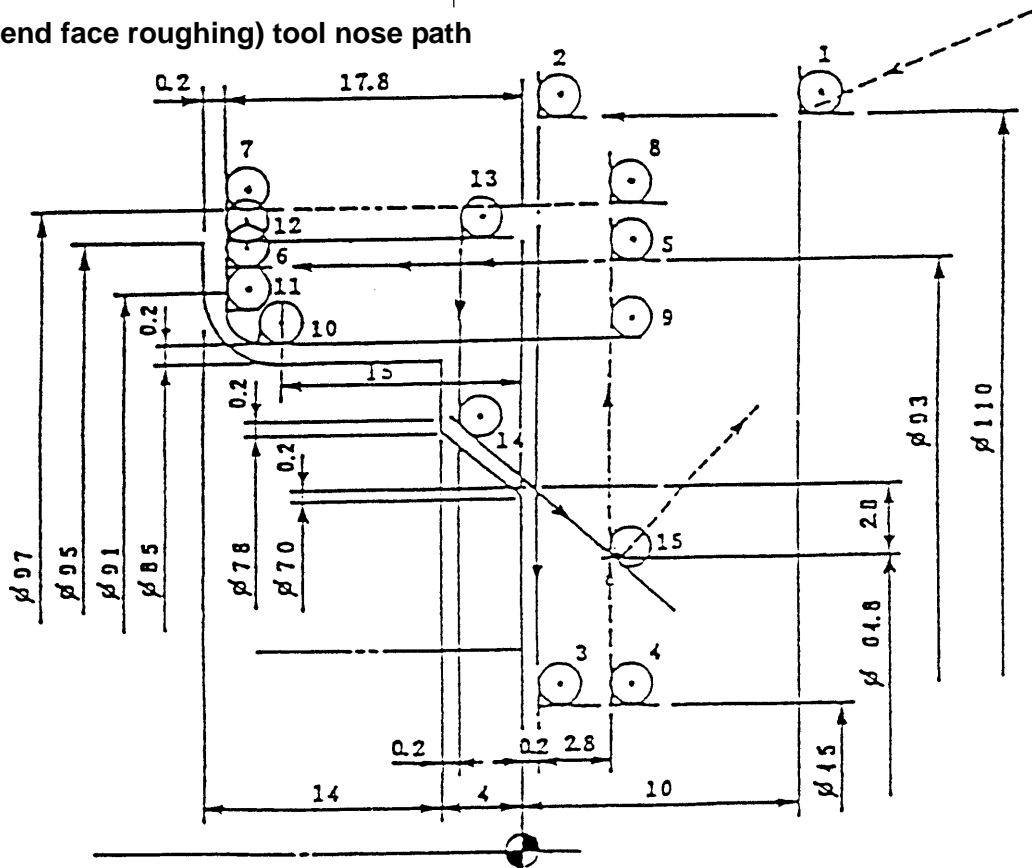


|   |   |    |  |     |
|---|---|----|--|-----|
| T1  | T3  | T5 | T7   | T9  |
|  |   |    |  |     |
| T2  | T4  | T6 | T8   | T10 |
|   |  |    |  |     |

## 4-1-2 Chuck Work Program

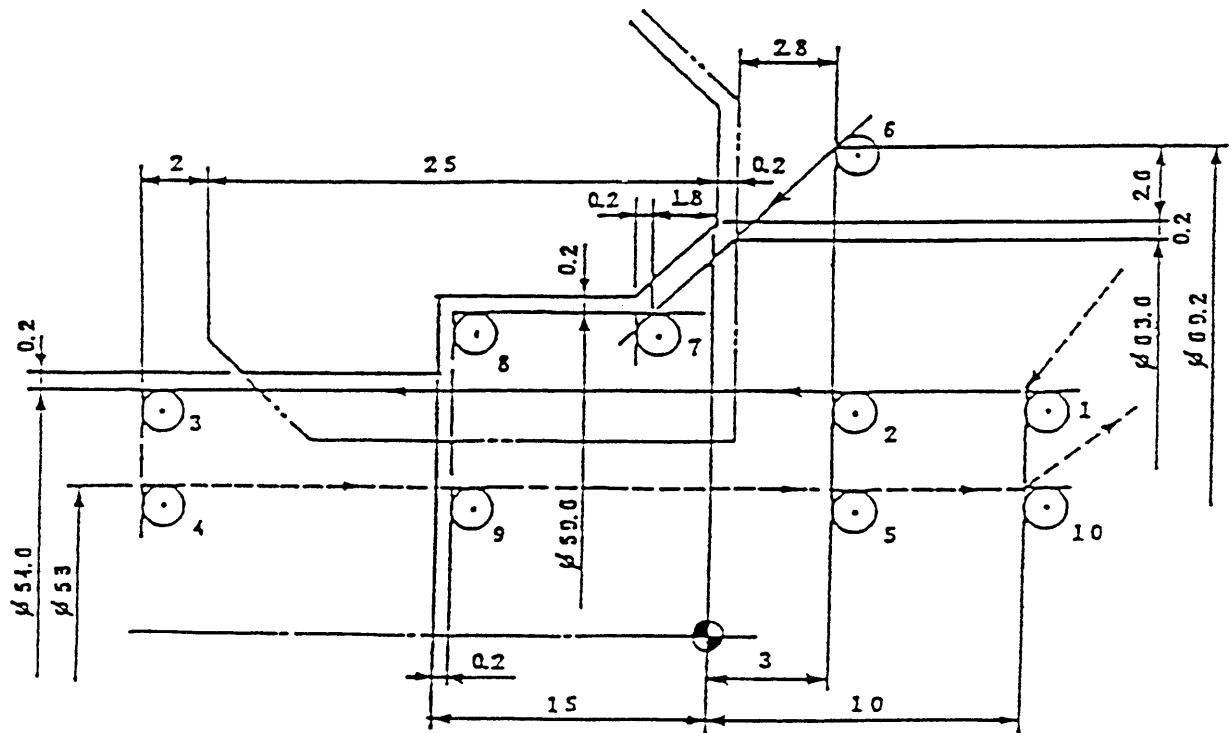
| Programming                 | Description                               |
|-----------------------------|---|
| O0052                       | Program No. Be sure to provide it.        |
| N1 G28 U0                   | Automatic reference point return (X axis) |
| N2 G28 W0 T0100             | Automatic reference point return (Z axis) |
|                             | Setting of T01 coordinate system          |
| N3 G50 S2000                | Maximum spindle speed clamp (2,000 rpm)   |
| N4 G00 X200.0 Z200.0        | Move to the index position.               |
| N5 M01                      | End of process                            |
| N101 T0100 M40              |   |
| N102 G97 S350 M08           |   |
| N103 G00 X110.0 Z10.0 M03   |   |
| N104 G01 G96 Z0.2 F3.0 S120 |   |
| N105 X45.0 F0.2             |   |
| N106 Z3.0                   |   |
| N107 G00 G97 X93.0 S400     |   |
| N108 G01 Z-17.8 F0.3        |   |
| N109 X97.0                  |   |
| N110 G00 Z3.0               |   |
| N111 X85.4                  |   |
| N112 G01 Z-15.0             |   |
| N113 G02 X91.0 Z-17.8 R2.8  |   |
| N114 G01 X95.0              |   |
| N115 G00 Z-3.8              |   |
| N116 G01 X78.4 F0.3         |   |
| N117 X64.8 Z3.0             |   |
| N118 G00 X200.0 Z200.0      |   |
| N119 M01                    |   |

### 1. T01 (OD end face roughing) tool nose path




N401 T0400 M40  
 N402 G97 S650 M08  
 N403 G00 X54.6 Z10.0 M03  
 N404 Z3.0  
 N405 G01 Z-27.0 F0.4  
 N406 X53.0  
 N407 G00 Z3.0  
 N408 X69.2  
 N409 G01 X59.6 Z-1.8 F0.3  
 N410 Z-14.8 F0.4  
 N411 X53.0  
 N412 G00 Z10.0  
 N413 X260.0 Z100.0  
 N414 M01

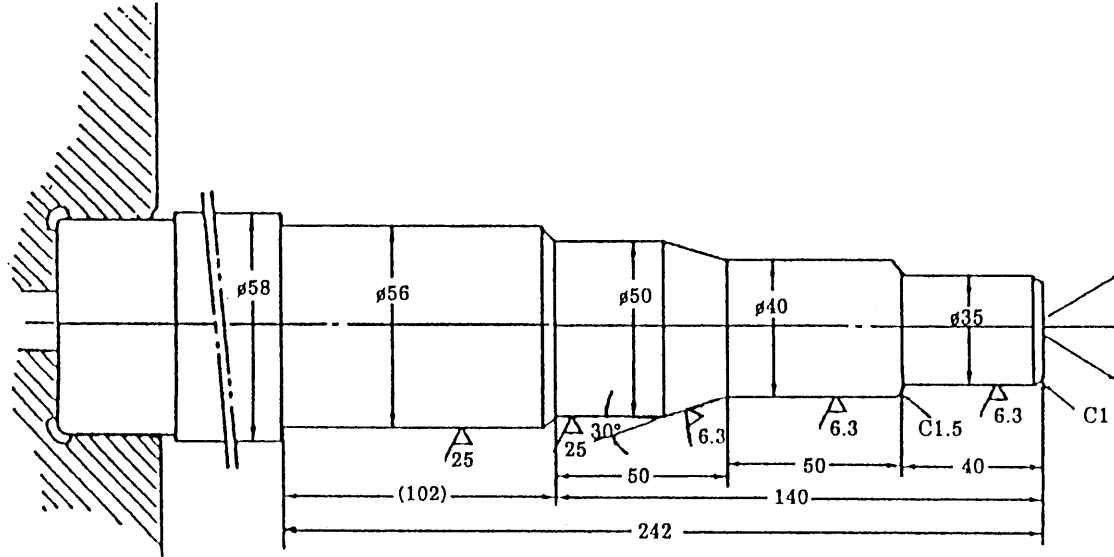
**2. T04 (ID roughing) tool nose route**



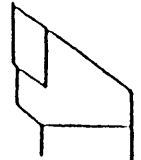
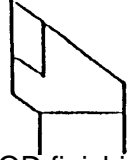




|                       |   |                   |                   |
|-----------------------|---|-------------------|-------------------|
| CENTER WORK EXAMPLE   |  | CNC LATHE         | PART NAME : SHAFT |
| PROCESS : 1ST NC UNIT |   | TOOL LAYOUT SHEET | MATERIAL S48C     |



4 - 6

|    |   |    |  |     |
|----|---|----|--|-----|
| T1 | T3  | T5 | T7   | T9  |
|    | <br>OD roughing |    | <br>OD finishing |     |
| T2 | T4  | T6 | T8   | T10 |
|    |   |    |  |     |

## 4-2-2 Center Work Program

|                           |                                    |
|---------------------------|------------------------------------|
| O0003                     |                                    |
| N1 G28 U0                 |                                    |
| N2 G28 W0 T0300           |                                    |
| N3 G50 S2000              |                                    |
| N4 G00 X200.0 Z10.0       |                                    |
| N5 M01                    |                                    |
| OD roughing               |                                    |
| N301 T0300 M40            | Selecting the turret face No.3     |
| N302 G97 S635 M08         |                                    |
| N303 G00 Z2.0 M03         |                                    |
| N304 ZX65.0               |                                    |
| N305 G96 S130             | Constant surface speed V 130 m/min |
| N306 X52.0                | Approach to a cutting position     |
| N307 G01 Z-139.1 F0.4     | Machining                          |
| N308 X56.4 Z-140.8        |                                    |
| X309 Z-241.8              |                                    |
| N310 X63.0                |                                    |
| N311 G00 Z2.0             |                                    |
| N312 X46.0                | Cutting-in                         |
| N313 G01 Z-89.8 F0.4      | Machining                          |
| N314 X56.0 Z-91.2         |                                    |
| N315 G00 Z2.0             |                                    |
| N316 X40.0                | Cutting-in                         |
| N317 G01 Z-89.8 F0.4      | Machining                          |
| N318 X50.4 Z-98.66        |                                    |
| N319 Z-139.8              |                                    |
| N320 X61.0                |                                    |
| N321 G00 Z2.0             |                                    |
| N322 X44.0                |                                    |
| N323 X29.4                | Cutting-in                         |
| N324 G01 X35.4 Z-1.0 F0.4 | Machining                          |
| N325 Z-39.8               |                                    |

|      |                       |                                      |
|------|-----------------------|--------------------------------------|
| N326 | X37.4                 |                                      |
| N327 | X42.4 Z-42.3          |                                      |
| N328 | G00 X50.0             |                                      |
| N329 | G97 X200.0 Z10.0 S825 | Canceling the constant surface speed |
| N330 | M01                   |                                      |

OD finishing

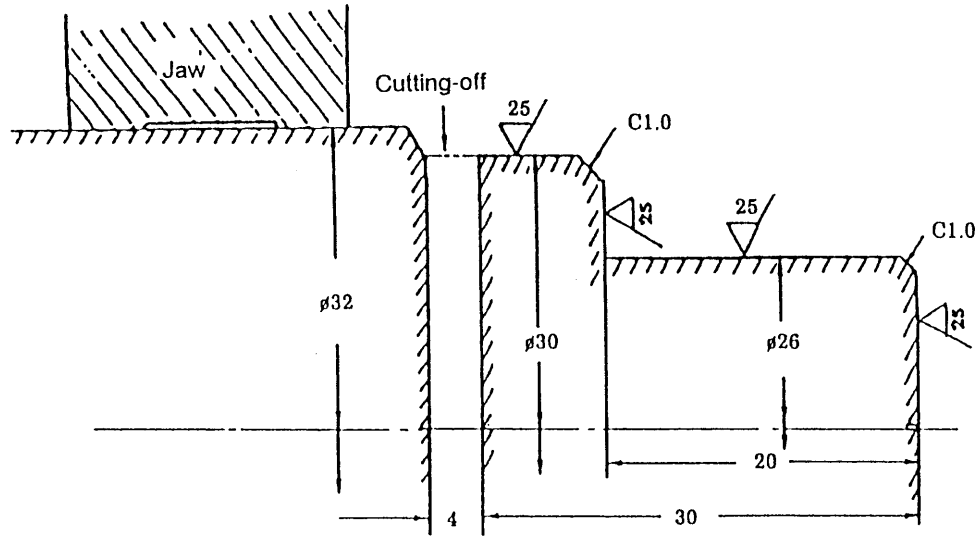
|      |                           |                                      |
|------|---------------------------|--------------------------------------|
| N701 | T0700 M40                 | Selecting the turret face No.7       |
| N702 | G97 S1350 M08             |                                      |
| N703 | G00 X210.0 Z2.0 M03       |                                      |
| N704 | X40.0                     |                                      |
| N705 | G96 S170                  | Constant surface speed               |
| N706 | X29.0                     | Approach to the cutting position     |
| N707 | G01 X35.0 Z-1.0 F0.15     | Machining                            |
| N708 | Z-40.0                    |                                      |
| N709 | X37.0                     |                                      |
| N710 | X40.0 Z-41.5              |                                      |
| N711 | Z-90.0                    |                                      |
| N712 | X50.0 Z-98.66             |                                      |
| N713 | Z-140.0 F0.2              |                                      |
| N714 | X54.0                     |                                      |
| N715 | X56.0 Z-141.0             |                                      |
| N716 | Z-242.0                   |                                      |
| N717 | X 65.0                    |                                      |
| N718 | G00 G97 X200.0 Z10.0 S835 | Canceling the constant surface speed |
| N719 | M01                       |                                      |

|    |                 |                                  |
|----|-----------------|----------------------------------|
| N6 | G28 U0 W0 T0300 | Automatic reference point return |
| N7 | M30             | End of program & rewind          |
| %  |                 |                                  |

# 4-3 Bar Work

## 4-3-1 Machining Drawing

|                  |         |              |                   |                 |
|------------------|---------|--------------|-------------------|-----------------|
| BAR WORK EXAMPLE |         | <b>SEIKI</b> | CNC LATHE         |                 |
| PROCESS :        | NC UNIT |              | TOOL LAYOUT SHEET | MATERIAL S48C-D |



4 - 9

|                               |    |    |    |                            |
|-------------------------------|----|----|----|----------------------------|
| T1                            | T3 | T5 | T7 | T9                         |
| <p>OD end facing<br/>R0.8</p> |    |    |    | <p>4mm<br/>Cutting-off</p> |
| T2                            | T4 | T6 | T8 | T10                        |
|                               |    |    |    | <p>Stopper</p>             |

## 4-3-2 Bar Work Program

|  |  |
|--|--|
| <p>O005</p> <p>N1 G28 U0</p> <p>N2 G28 W0 T1000</p> <p>N3 G50 S2000</p> <p>N4 G00 X200.0 Z200.0</p> <p>N5 M01</p> <p style="text-align: center;">Material sizing</p>   |  |
| <p>N1001 T1000 M40</p> <p>N1002 G97 S200</p> <p>N1003 G00 X0 Z10.0 M03</p> <p>N1004 G01 Z-33.0 F5.0</p> <p>N1005 M69</p> <p>N1006 G04 U2.0</p> <p>N1007 G01 Z1.0 F5.0</p> <p>N1008 M68</p> <p>N1009 G04 U3.0</p> <p>N1010 G00 Z10.0</p> <p>N1011 X200.0 Z200.0</p> <p>N1012 M31</p> <p>N1013 G04 U0.5</p> <p>N1014 M01</p> <p style="text-align: center;">OD cutting</p> | <p>Selecting the turret face No.10</p> <p>Stopper approach</p> <p>Chuck open</p> <p>Dwell 2 seconds (chuck opening time)</p> <p>Material loading and sizing</p> <p>Chuck close</p> <p>Dwell 3 seconds (chuck closing time)</p> <p>Retreat</p> <p>Retract to index position</p> <p>No-workpiece check</p> <p>Dwell 0.5 second</p> |
| <p>N101 T0100</p> <p>N102 G97 S1005 M08</p> <p>N103 G00 X38.0 Z10.0 M03</p> <p>N104 G96 Z0 S120</p> <p>N105 G01 Z-1.6 F0.2</p> <p>N106 Z3.0</p> <p>N107 G00 X18.4</p> <p>N108 G01 X26.4 Z-1.0 F0.3</p> <p>N109 Z-19.8</p> <p>N110 X28.4</p> <p>N111 X30.4 Z-20.8</p>   | <p>Selecting the turret face No.1</p> <p>Constant surface speed V 120 m/min</p> <p>End facing</p> <p>OD cutting</p>  |

N112 Z-36.0  
 N113 X34.4 Z-38.0  
 N114 G00 X40.0  
 N115 Z3.0  
 N116 X18.0  
 N117 G01 X26.0 Z-1.0 F0.3  
 N118 Z-20.0  
 N119 X28.0  
 N120 X30.0 Z-21.0  
 N121 Z-35.0  
 N122 G00 X40.0  
 N123 G97 X200.0 Z200.0 S955  
 N124 M01

OD finishing

Cutting-off

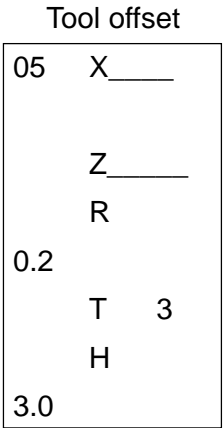
N901 T0900 M40  
 N902 G97 S795 M08  
 N903 M63  
 N904 G04 U1.0  
 N905 G00 X45.0 Z-25.0 M03  
 N906 G01 X40.0 Z-34.0 F2.0  
 N907 G96 X-0.5 F0.1 S100  
 N908 X40.0 F1.0 M09  
 N909 G00 MG97 X200.0 Z200.0 S795 M05  
 N910 M01  
  
 N10 M64  
 N11 G04 U0.5  
 N12 M12  
 /N13 P1001 M99  
 N14 G28 U0 W0  
 N15 M30  
 %

Selecting the turret face No.9  
  
 Unloader advance  
 Dwell 1 second (unloader operating time)  
 Positioning  
 Approach to a cutting-off position  
 Cutting-off  
 Retreating a cutting-off tool  
 Canceling the constant surface speed  
  
  
 Returning the unloader  
 Unloader operating time  
 Work count  
 Return to N1001 and remachining start  
 Block skip and reference point return  
 Block skip end program & rewind

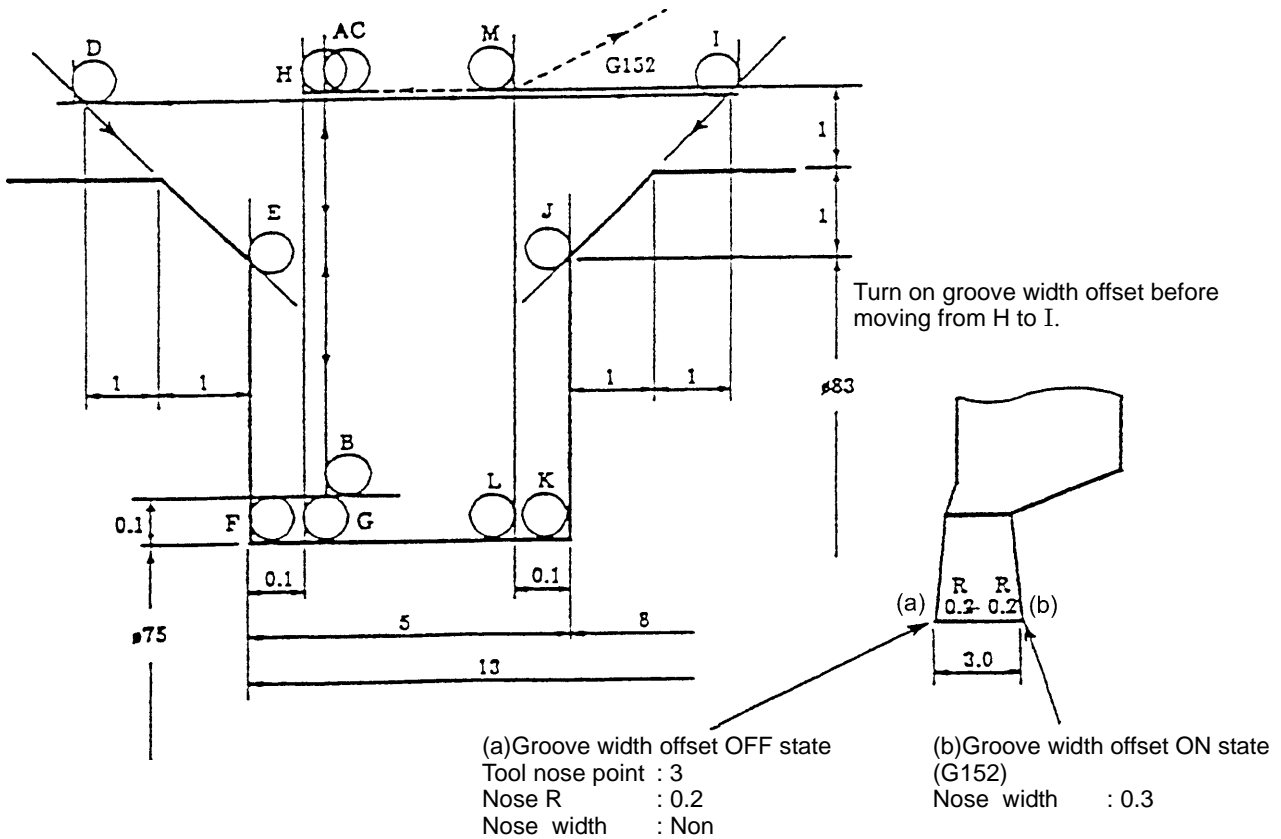
## 4-4 Grooving

### 4-4-1 OD Grooving

| Programming                     | Description  |
|---------------------------------|--|
| N501 T0500 M40                  | No.5 turret face calling                                 |
| N502 G97 S360 M08               |  |
| N503 G150                       | Groove width offset OFF                                  |
| N504 G00 X87.0 Z10.0 M03        |  |
| N505 G01 G96 Z-12.0 F5.0 S100   |  |
| A→B N506 X75.2 F0.1             |  |
| B→C N507 X87.0 F5.0             |  |
| C→D N508 Z-15.0                 |  |
| D→E N509 X83.0 Z-13.0 F0.1      |  |
| E→F N510 X75.0                  |  |
| F→G N511 Z-12.9                 |  |
| G→H N512 X87.0 F5.0             |  |
| N513 G152                       | Groove width offset ON.<br>Change to a program point "b" |
| H→I N514 Z-6.0                  |  |
| I→J N515 X83.0 Z-8.0 F0.1       |  |
| J→K N516 X75.0                  |  |
| K→L N517 Z-8.1                  |  |
| L→M N518 X87.0 F5.0             |  |
| N519 G150                       | Groove width offset OFF                                  |
| N520 G00 G97 X200.0 Z200.0 S365 |  |
| N521 M01                        |  |



**1. T05 (OD grooving) Tool width : 3mm**



**4-4-2 ID Grooving**

| Programming                   | Description              |
|-------------------------------|--------------------------|
| N601 T0600 M40                | No.6 turret face calling |
| N602 G97 S400 M08             |                          |
| N603 G150                     | Groove width offset OFF  |
| N604 G00 X78.0 Y0 Z10.0 M0.3  | Tool offset              |
| N605 G01 G96 Z-9.75 F5.0 S100 | 06 X____                 |
| A→B N606 X86.0 F0.1           | Z____                    |
| B→C N607 X79.0 F1.0           | R 0.2                    |
| C→D N608 Z-10.7               | T 2                      |
| D→E N609 X80.4 Z-10.0 F0.1    | H 2.5                    |
| E→F N610 X86.0                |                          |
| F→G N611 Z-9.8                |                          |
| G→H N612 X79.0 F1.0           |                          |

N613 G152

Groove width offset ON.

Change to a program point "b"

H→I N614 Z-6.3

I→J N615 X80.4 Z-7.0

J→K N616 X86.0 F0.1

K→L N617 Z-7.2

L→M N618 X78.0 F1.0

N619 G00 Z10.0

N620 G150

Groove width offset OFF

N621 G00 G97 X260.0 Z100.0 S400

N622 M01

(a) Groove width offset OFF state

Tool nose point : 2

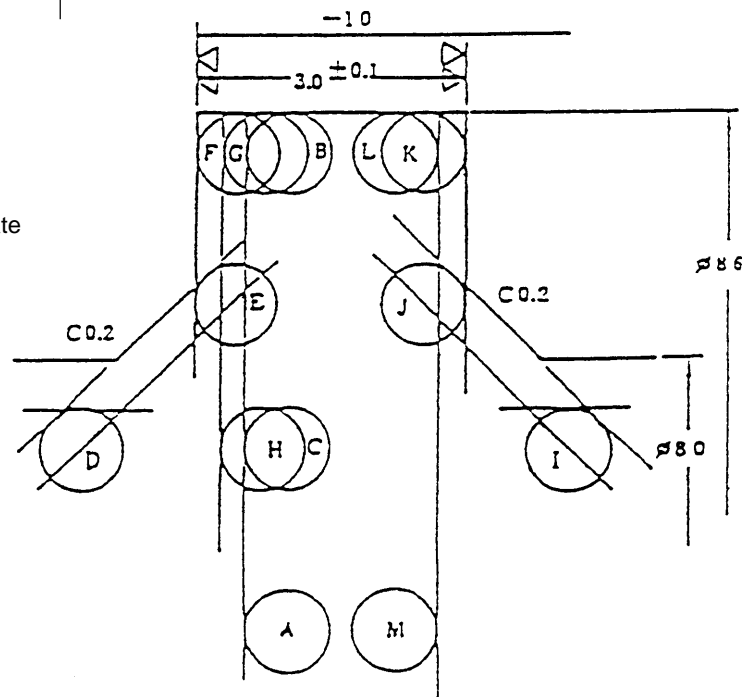
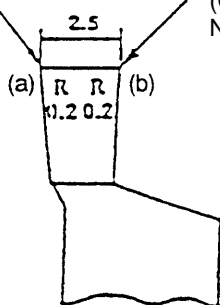
Nose R : 0.2

Nose width : Non

(b) Groove width offset ON state

(G152)

Nose width : 2.5

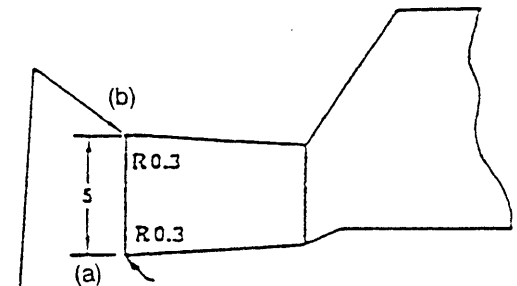
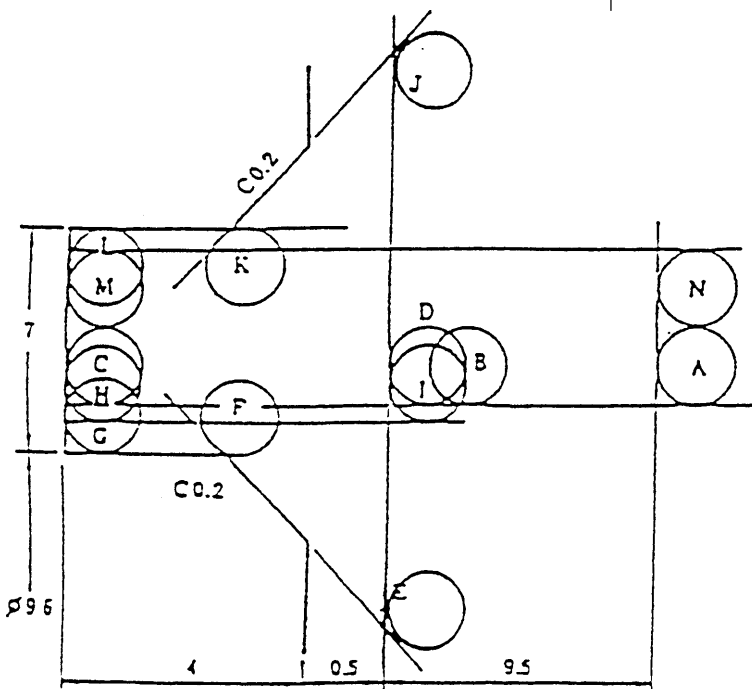


### 4-4-3 End Face Grooving

| Programming                    | Description              |
|--------------------------------|--------------------------|
| N301 T0300 M40                 | No.3 turret face calling |
| N302 G97 S330 M08              |                          |
| N303 G150                      | Groove width offset OFF  |
| N304 G00 X97.0 Z10.0 M03       |                          |
| A→B N305 G01 Z1.0 F8.0         |                          |
| B→C N306 Z-4.0 F0.1            |                          |
| C→D N307 Z0.5 F1.0             |                          |
| D→E N308 X94.6                 |                          |
| E→F N309 X96.0 Z0.2 F0.1       |                          |
| F→G N310 Z-4.0                 |                          |
| G→H N311 X96.5                 |                          |
| H→I N312 Z0.5 F1.0             |                          |
| N313 G151                      | Groove width offset ON.  |
| Change to a program point "b"  |                          |
| I→J N314 G00 X111.4            |                          |
| J→K N315 G01 X110.0 Z-0.2 F0.1 |                          |
| K→L N316 Z-4.0                 |                          |
| L→M N317 X109.5                |                          |
| M→N N318 G00 Z10.0             |                          |
| N319 G150                      | Groove width offset OFF  |
| N320 G00 X260.0 Z100.0         |                          |
| N321 M01                       |                          |

Tool offset

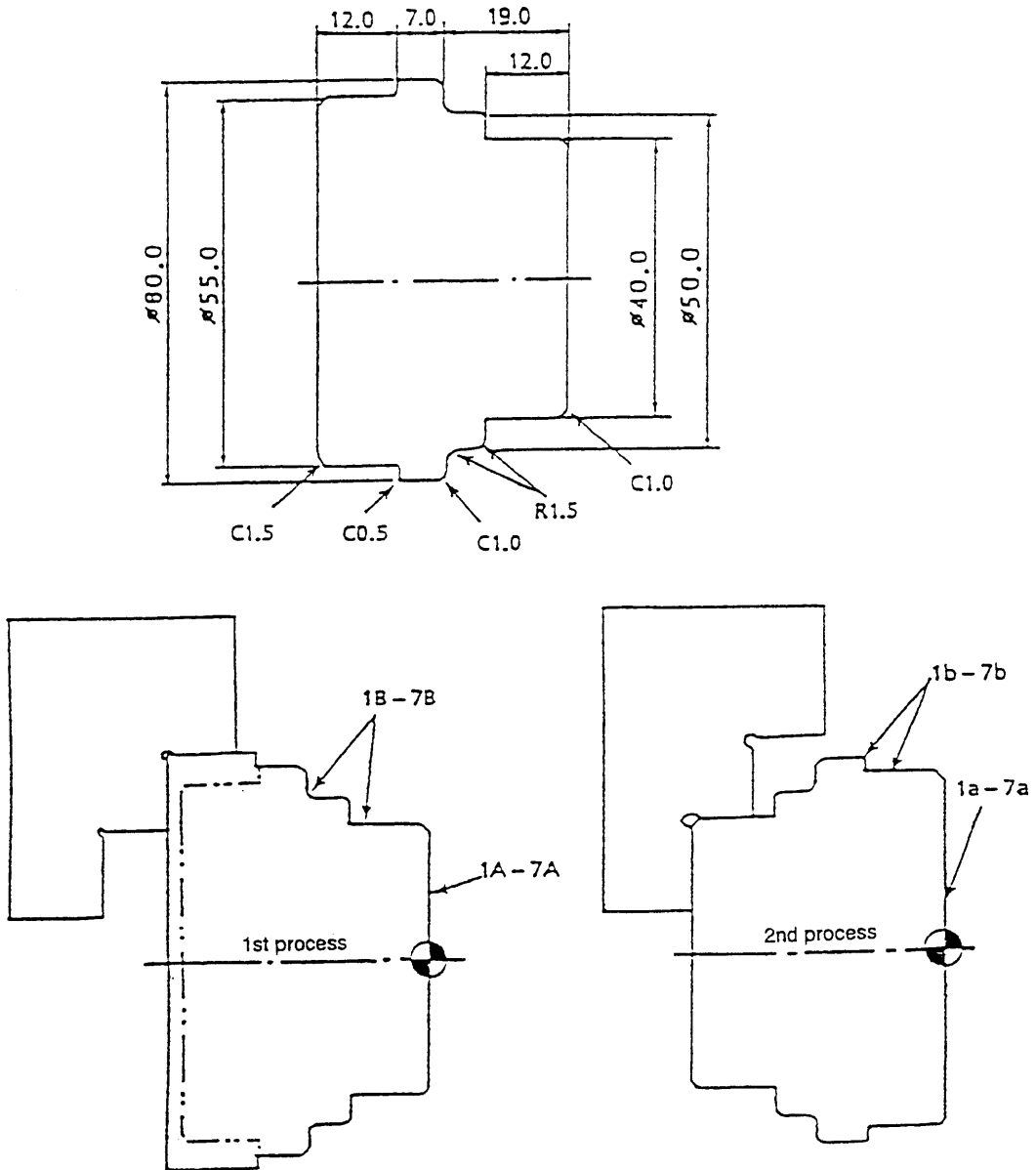
|    |   |       |
|----|---|-------|
| 03 | X | _____ |
|    | Z | _____ |
|    | R | 0.2   |
|    | T | 3     |
|    | H | 5.0   |



- (a) Groove width offset OFF state  
 Tool nose point : 3  
 Nose R : 0.3  
 Nose width : Non
- (b) Groove width offset ON state  
 (G151)  
 Nose width : 5.0

### 4-5 1st and 2nd Process Continuous Machining Method

One example for programming method of consecutive machining as process 1st and 2nd is introduced as follows:



|                         |    |    |                          |     |
|-------------------------|----|----|--------------------------|-----|
| T1                      | T3 | T5 | T7                       | T9  |
| R0.8<br><br>OD roughing |    |    | R0.8<br><br>OD finishing |     |
| T2                      | T4 | T6 | T8                       | T10 |
|                         |    |    |                          |     |

## 4-5-1 Machining Method by Single Program

O1111 (1st process)

N1 G28 U0

N2 G28 W0 T0100

N3 G54 Z0  Reference point shift cancel.

N4 G50 S1800

N5 G00 X200.0 Z175.0

N6 M01

N100 (OD-R)

N101 T0100 M40

N102 G97 S545

N103 G00 X70.0 Z10.0 M03

N104 G01 Z0.2 F1.5 M08

M105 G96 X-1.2 F0.2 S120

N106 ..

N...

N... M01

N700 (OD-F)

N701 T0700 M40

N702 G97 S... M08

N703 G00 X... Z...M03

N704 ..

N...

N... G00 G97 X200.0 Z175.0

N... M01

N8 G28 U0 W0

N9 M00 ← Turning over the workpiece.

(2nd process)

N11 G28 U0 W0

N12 G54 Z-12.0  \* (Difference from the finishing end face of 1st process.)

N13 G50 S1800

N14 G00 X200.0 Z175.0

N15 M01

N5100 (OD-R)

N5101 T0100 M40

N5102 G97 S545

N5103 G00 X... Z...M03

N... M08

N...

N... G30 U0 W0

N... M01

N5700 (OD-F)

N5701 T0700 M40

N5702 G97 S...M08

N5703 X... Z...M03

N5704 ..

N...

N... G00 X200.0 Z175.0

N... M01

N20 G54 Z0  Reference point shift cancel.

N21 G28 U0 W0

N22 M30

%

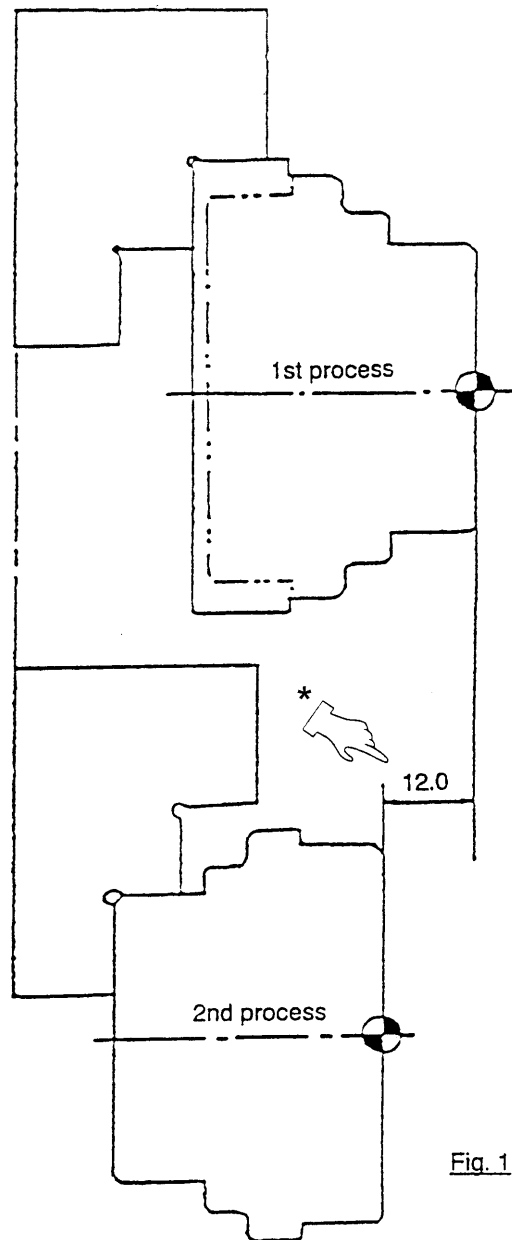


Fig. 1

Reference point for programming is the 1st process finishing end face.

## 4-5-2 Machining Method by Subprogram Calling

Executing method of continuous machining when call subprogram by main program. 1st and 2nd process machining program are stored separately as subprograms.

\*\*\* Main program\*\*\*

O2222 Refer to Fig. 1.

(OP-1)

N1 M98 P0001 ..... For calling 1st process program

N2 M00 ..... Turning over the workpiece

(OP-2)

N3 M98 P0002 ..... For calling 2nd process program


N4 M30

%

(Program for 1st process)

O0001 (OP-1)

N1 G28 U0

N2 G28 W0 T0100  Reference point shift cancel.

N3 G54 Z0

N4 G56 S1800

N5 G00 X200.0 Z175.0

N6 M01

N100 (OD-R)

N101 T0100 M40

N102 G97 S545

N103 G00 X70.0 Z10.0 M03

N104 G01 Z0.2 F1.5 M08

N105 G96 X-1.2 F0.2 S120

N106 Z...

N...

N...

N... M01

N700 (OD-F)

N701 T0700 M40

N702 G97 S...M08

N703 G00 X... Z...M..

N704 ..

N...

N...

N... G00 G97 X200.0 Z175.0

N... M01

N8 G28 U0 W0

/N9 M99


N10 M30

%

(Program for 2nd process)

O0002 (OP-2)

N20 G28 U0 W0

N21 G54 Z-12.0  \* (Difference from the finishing end face of 1st process.)

N22 G50 S1800

N23 G00 X200.0 Z175.0

N24 M01

N5100 (OD-R)

N5101 T0100 M40

N5102 G97 S545

N5103 G00 X... Z...M03

N... M08

N...

N... G00 X200.0 Z175.0

N... M01

N5700 (OD-F)

N5701 T0700 M40

N5702 G97 S...M08

N5703 X... Z...M03

N5704 ..

N...

N...

N... G00 X200.0 Z175.0

N... M01

N26 G54 Z0  Reference point shift cancel.

N27 G28 U0 W0

/N28 M99

N29 M30

%

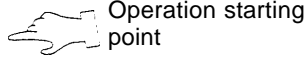
## 4-6 Operation Example of Many Short Length Works

O1111 (Main program)

N1 G28 U0

N2 G28 W0

N3 G10 P00 Z200.0



Operation starting point

N4 T1000

N5 G50 S2000

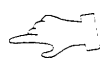
N6 G00 X200.0 Z50.0

N7 M01

N1000 T1000 M40

N1001 G00 Z1.0

N1002 X0



Positioning of works (manual)

N1003 M00

N1004 G00 X200.0 Z50.0

N1005 M01

N8 M98 P2222 L3

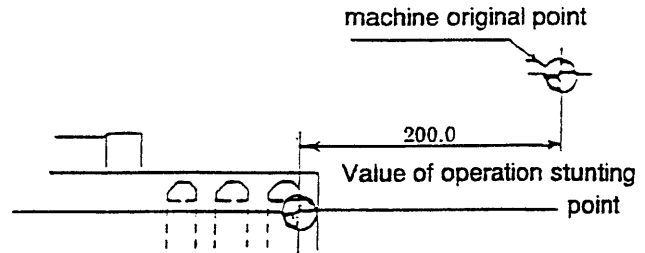


Call of operation program and setting

N9 G28 U0 M09

N10 G28 W0 M05

N11 M30



O2222 (Sub-program)

N100 T0100 M40

N101 G96 S120 M08

N102 G00 X... Z...M03

┆

Operation program

┆

Cutting-in program

┆

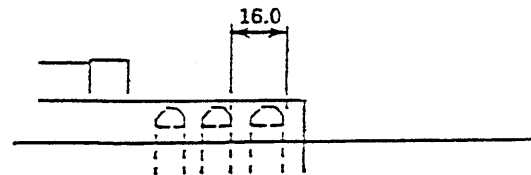
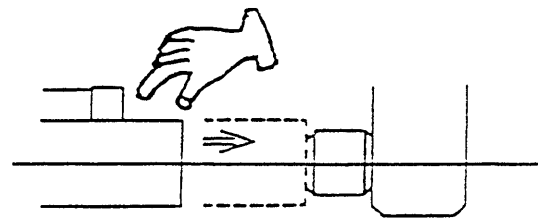
N... M01

N12 G10 P00 W16.0

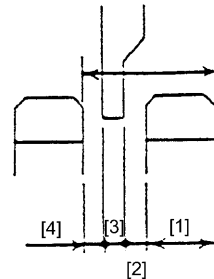


Shifted amount

N13 M99



**Note)** Shifted amount is [1]finished length of work +[2]cutting length of back surface +[3]width of cutting in tool +[4]cutting length of surface.





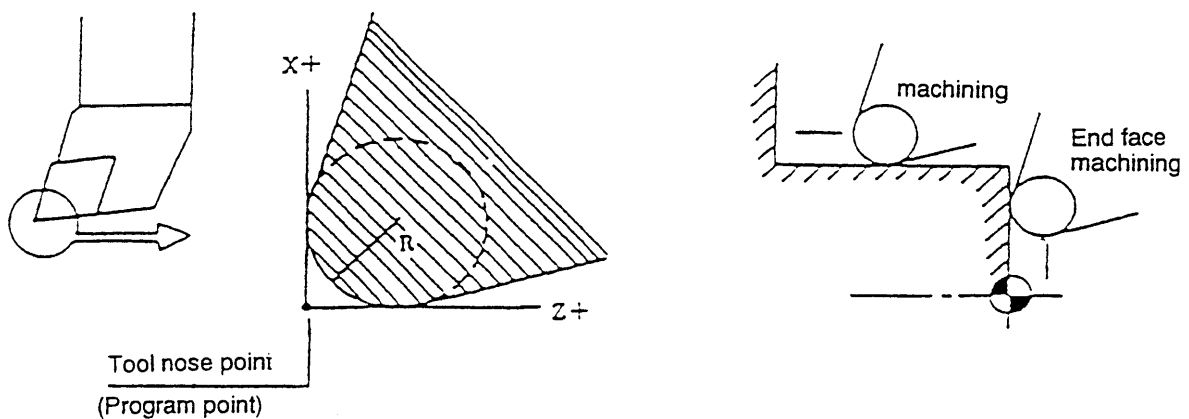
## 5. REFERENCE MATERIALS

### 5-1 How to Calculate the Tool Nose Radius Compensation Amount Without Using the Tool Nose Radius Compensation Function

At the normal program, since it becomes a program which is a program point coincide a point on the drawing if nose R compensation function is used, preparation time of program is shortened, however, in this section explain about a method without calculation function of nose R compensation, i.e. direct command of tool nose point.

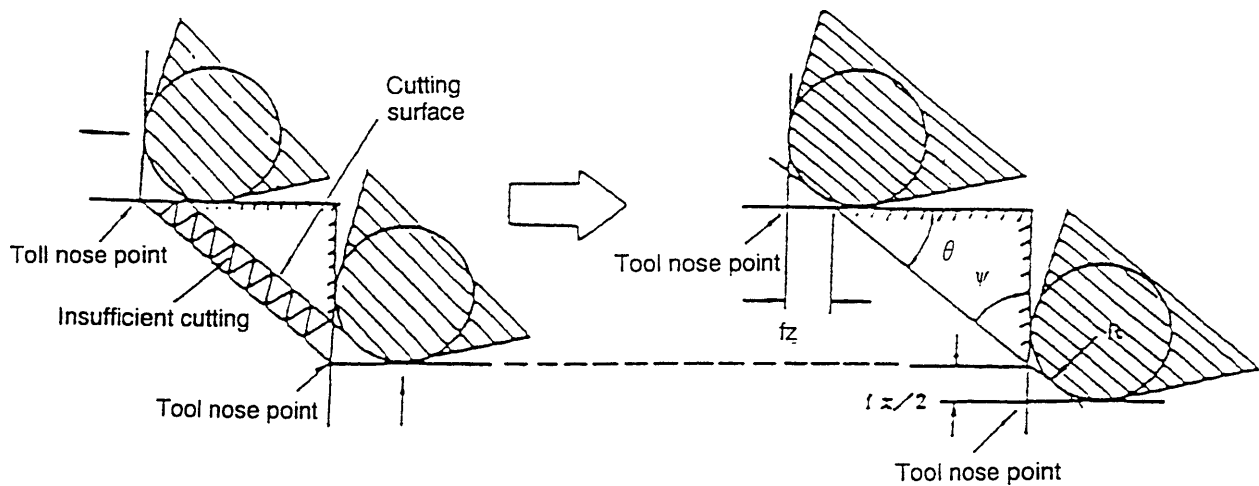
#### 1. Tool nose radius compensation amount

A tool nose has roundness called nose R. When cutting an outer diameter, inner diameter or end face in parallel with an axis, it can be cut as per the drawing, even if the tool nose is programmed as one point (tool nose point).



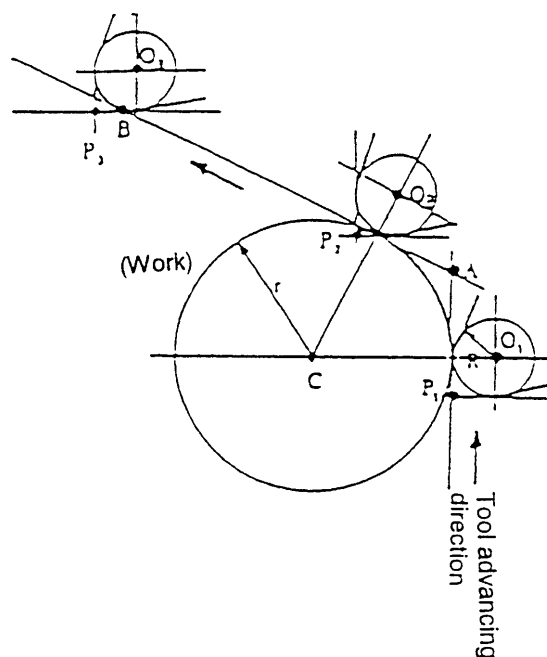
It can be cut according to the drawing even program by tool nose point.

However, a tool position to be cut is differ and become "Left behind or over cut" since a tool position is different by a tool nose point program when chamfering, tapering or circular cutting. To avoid this left behind cutting, according to an angle of chamfer and taper or size of nose R of tool a command which is shifted a tool nose at the X and Z direction with finding a nose R compensation amount ( $f_x$ ,  $f_z$  in lower sketch) by manual calculation.



## 2. Calculating procedure of tool nose position

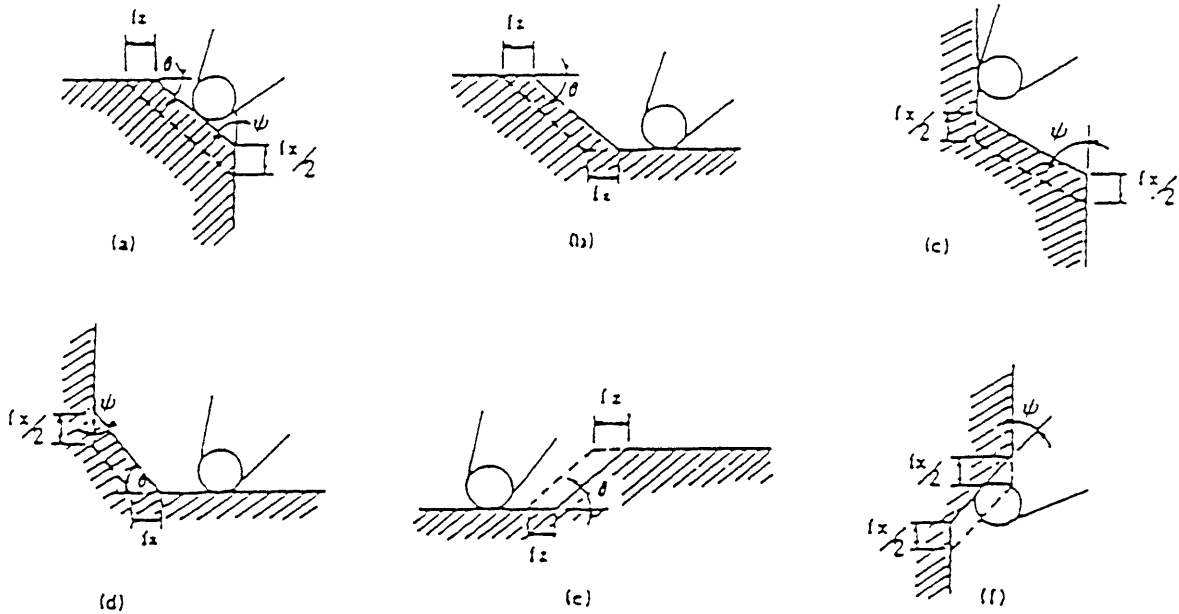
1. Calculate the coordinate values of the intersecting points of a straight line and those of the center of a circular arc. (in the above-mentioned figure, coordinate values of the points A, B and C)
2. Calculate the center coordinate values of the nose R to each intersecting point or contact point, and a radius value (I, K) in circular cutting. (in the below-mentioned figure, coordinate values of the points  $O_1$ ,  $O_2$  and  $O_3$  and a distance between the points  $O_1$  and C)
3. Transfer the center coordinate values of each nose R obtained in the step 2 to the coordinate values of the program point. (in the above-mentioned figure, coordinate values of the points  $P_1$ ,  $P_2$  and  $P_3$ )



### 3. How to obtain tool nose radius compensation amount in chamfering and taper cutting

To prevent insufficient cutting, calculate the tool nose radius compensation amount ( $f_x$ ,  $f_z$ ) out of an angle and a nose R size, and shift the tool by the amount when programming.

Although the following tool paths (a) through (f) are available, make programming so that the tool will take a path indicated by a broken line to each desired machining profile (full line).



Calculating formula of tool nose R compensation amount

**Note)** Except (e) and (f)  $R = \text{nose } R$

$$f_x = 2R \left( 1 - \tan \frac{\psi}{2} \right)$$

$$f_z = R \left( 1 - \tan \frac{\theta}{2} \right) \quad (\text{where; } \psi = 90 - \theta)$$

**Note)** For (e) and (f), use the following formulas respectively, because a cutting edge is reversed.

$$(f) \quad f_x = 2R \left( 1 + \tan \frac{\psi}{2} \right)$$

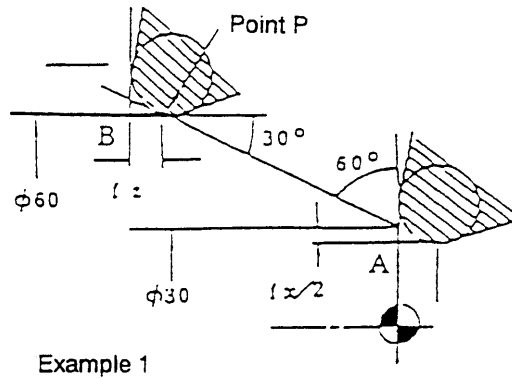
$$(e) \quad f_z = R \left( 1 + \tan \frac{\theta}{2} \right)$$

Indicate the value found typical angle and size of nose R by the above formula in the table (next page).

| Tool nose<br>R (radius) |    | 0.2   | 0.4   | 0.5   | 0.8   | 1.0   | 1.2   | 1.6   |
|-------------------------|----|-------|-------|-------|-------|-------|-------|-------|
| Angle                   |    |       |       |       |       |       |       |       |
| θ mm 5°                 | fx | 0.033 | 0.067 | 0.084 | 0.134 | 0.167 | 0.201 | 0.268 |
|                         | fz | 0.191 | 0.383 | 0.478 | 0.765 | 0.956 | 1.148 | 1.530 |
| 10°                     | fx | 0.064 | 0.129 | 0.161 | 0.257 | 0.322 | 0.386 | 0.515 |
|                         | fz | 0.183 | 0.365 | 0.456 | 0.730 | 0.913 | 1.095 | 1.460 |
| 15°                     | fx | 0.093 | 0.186 | 0.233 | 0.372 | 0.465 | 0.558 | 0.745 |
|                         | fz | 0.174 | 0.347 | 0.434 | 0.695 | 0.868 | 1.042 | 1.389 |
| 20°                     | fx | 0.120 | 0.240 | 0.300 | 0.480 | 0.600 | 0.719 | 0.959 |
|                         | fz | 0.165 | 0.329 | 0.412 | 0.659 | 0.824 | 0.988 | 1.318 |
| 25°                     | fx | 0.145 | 0.290 | 0.363 | 0.581 | 0.726 | 0.871 | 1.161 |
|                         | fz | 0.156 | 0.311 | 0.389 | 0.623 | 0.778 | 0.934 | 1.245 |
| 30°                     | fx | 0.169 | 0.338 | 0.423 | 0.676 | 0.845 | 1.014 | 1.352 |
|                         | fz | 0.146 | 0.293 | 0.366 | 0.586 | 0.732 | 0.878 | 1.171 |
| 35°                     | fx | 0.192 | 0.384 | 0.479 | 0.767 | 0.959 | 1.151 | 1.534 |
|                         | fz | 0.137 | 0.274 | 0.342 | 0.548 | 0.685 | 0.822 | 1.096 |
| 40°                     | fx | 0.213 | 0.427 | 0.534 | 0.854 | 1.067 | 1.281 | 1.708 |
|                         | fz | 0.127 | 0.254 | 0.318 | 0.509 | 0.636 | 0.763 | 1.018 |
| 45°                     | fx | 0.234 | 0.469 | 0.586 | 0.937 | 1.172 | 1.406 | 1.875 |
|                         | fz | 0.117 | 0.234 | 0.293 | 0.469 | 0.586 | 0.703 | 0.937 |
| 50°                     | fx | 0.254 | 0.509 | 0.636 | 1.018 | 1.272 | 1.526 | 2.035 |
|                         | fz | 0.107 | 0.213 | 0.267 | 0.427 | 0.534 | 0.640 | 0.747 |
| 55°                     | fx | 0.274 | 0.548 | 0.685 | 1.096 | 1.369 | 1.643 | 2.191 |
|                         | fz | 0.096 | 0.192 | 0.240 | 0.384 | 0.479 | 0.575 | 0.767 |
| 60°                     | fx | 0.293 | 0.586 | 0.732 | 1.171 | 1.464 | 1.757 | 2.343 |
|                         | fz | 0.085 | 0.169 | 0.211 | 0.338 | 0.423 | 0.507 | 0.676 |
| 65°                     | fx | 0.311 | 0.623 | 0.778 | 1.245 | 1.557 | 1.868 | 2.491 |
|                         | fz | 0.073 | 0.145 | 0.181 | 0.290 | 0.363 | 0.436 | 0.581 |
| 70°                     | fx | 0.329 | 0.659 | 0.824 | 1.318 | 1.647 | 1.977 | 2.636 |
|                         | fz | 0.060 | 0.120 | 0.150 | 0.240 | 0.300 | 0.360 | 0.480 |
| 75°                     | fx | 0.347 | 0.695 | 0.868 | 1.389 | 1.737 | 2.084 | 2.779 |
|                         | fz | 0.047 | 0.093 | 0.116 | 0.186 | 0.233 | 0.279 | 0.372 |
| 80°                     | fx | 0.365 | 0.730 | 0.913 | 1.460 | 1.825 | 2.190 | 2.920 |
|                         | fz | 0.032 | 0.064 | 0.080 | 0.129 | 0.161 | 0.193 | 0.257 |
| 85°                     | fx | 0.383 | 0.765 | 0.956 | 1.530 | 1.913 | 2.295 | 3.060 |
|                         | fz | 0.017 | 0.033 | 0.042 | 0.067 | 0.084 | 0.100 | 0.134 |

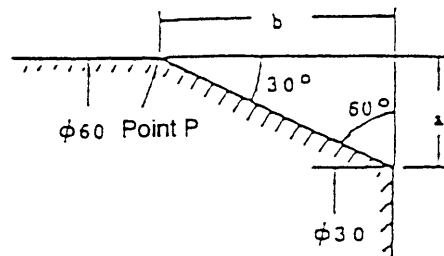
The case (e) and (f) on the previous page are excluded.

**4. Example of tool nose radius compensation amount calculation in chamfering and taper cutting**



When the tool is located at the positions A and B in the above figure, the tool nose radius compensation amount ( $f_x$ ,  $f_z$ ) is obtained as follows. (However, the nose radius of a tool used shall be 0.8.)

- (1) For the Z-axis position at the point P in the above-mentioned figure, draw a triangle as shown below and obtain lengths of the sides "a" and "b".



The length of the side "a" is ;

$$a = \frac{\phi 60 - \phi 30}{2} = 15$$

The length of the side "b" is ;

$$b = \tan 60^\circ \times a$$

$$= 1.732 \times 15 = 25.98$$

Therefore, the position of the point P is X60.0 and Z-25.98.

(2) Tool nose R compensation amount (fx, fz)

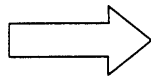
$$\begin{aligned}
 f_x &= 2R \left(1 - \tan \frac{\psi}{2}\right) \\
 &= 2 \times 0.8 \left(1 - \tan \frac{60^\circ}{2}\right) \\
 &= 2 \times 0.8 (1 - \tan 30^\circ) \\
 &= 2 \times 0.8 (1 - 0.57735) \\
 &= 2 \times (0.42265) \\
 &= 2 \times 0.338 \\
 &= 0.676
 \end{aligned}$$

$$\begin{aligned}
 f_z &= R \left(1 - \tan \frac{\theta}{2}\right) \\
 &= 0.8 \times \left(1 - \tan \frac{30^\circ}{2}\right) \\
 &= 0.8 \times (1 - \tan 15^\circ) \\
 &= 0.8 \times (1 - 0.268) \\
 &= 0.8 \times 0.732 \\
 &= 0.5856
 \end{aligned}$$

(3) X and Z coordinate value of tool nose point

Tool nose point position of the tool A

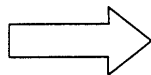
$$\begin{aligned}
 X &= \phi 30 - f_x \\
 &= 30 - 0.676 \\
 &= 29.324 \\
 &29.32
 \end{aligned}$$



Coordinate value  
X29.32  
Z0

Tool nose point position of the tool B

$$\begin{aligned}
 Z &= P - f_z \\
 &= -25.98 - 0.5856 \\
 &= -26.5656 \\
 &-26.57
 \end{aligned}$$



Coordinate value  
X60.0  
Z-26.57

(4) Program example

G00 X..... Z ..... T□□□△△

G01 Z0 F .....

X29.32 ←———— A

X60.0 Z-26.57 ←———— B

Z-.....

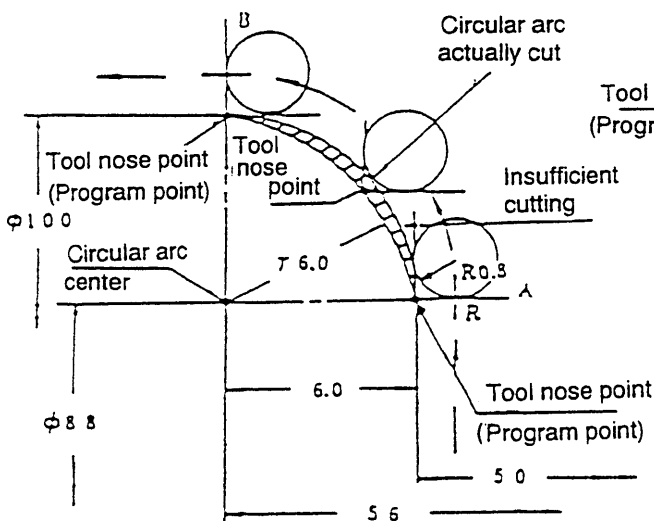
To perform cutting shown in the above-mentioned example 1, program the tool nose point positions of the A and tool B, shifting them by the tool nose radius compensation amount (fx, fz).

## 5. How to obtain tool nose radius compensation amount in circular cutting

(1) Program example without considering tool nose R compensation amount In circular cutting, a tool cuts a workpiece along its circular arc "r" with the nose R being in contact with the arc.

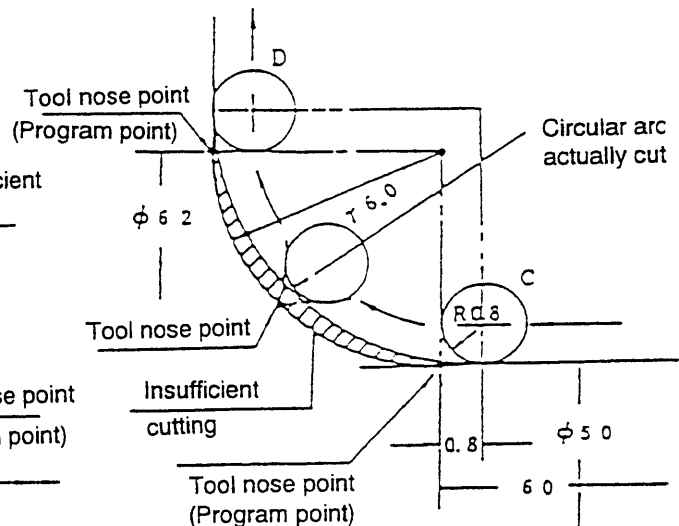


Due to this, insufficient cutting will be caused as shown in the following figure, if the nose R is not taken into account in case of circular cutting as well.



(Ex.) 1/4 circular arc, nose R 0.8

Exempli 2



(Ex.) 1/4 circular arc, nose R 0.8

Example 3

Program for Example 2

```
G01 Z-50.0 F0.2
      X88.0 ←———— A
G03 X100.0 Z-56.0 R6.0 ← B
G01 Z-ΔΔ. Δ
```

Program for Example 3

```
G01 X50.0 F0.2
      Z-60.0 ←———— C
G02 X62.0 Z-66.0 R6.0 ← D
G01 XΔΔ. Δ
```

Since the virtual tool nose point (program point) is different from a cutting edge position for actual cutting, insufficient cutting is caused by the programs for Examples 2 and 3.

To prevent insufficient cutting of a convex circular arc;



Calculate the positions of the nose R center and virtual tool nose point at the start point and end point of the circular arc, and command the position of the virtual tool nose point by the program.



For the convex circular arc, command a circular arc "r" larger by a tool nose radius.

To prevent insufficient cutting of a concave circular arc;

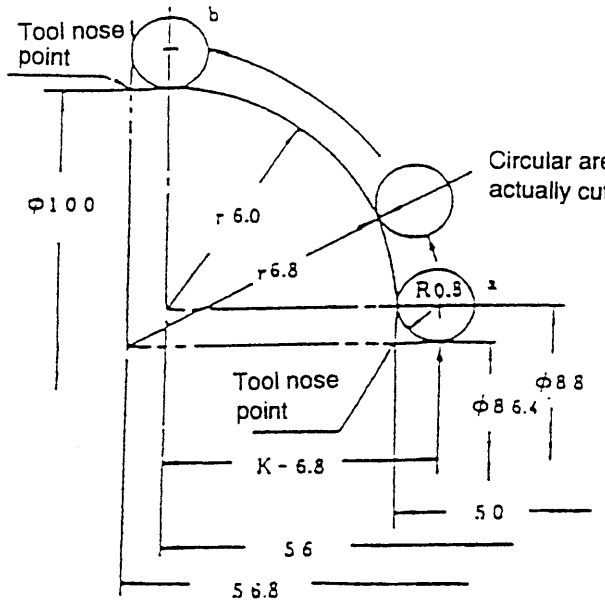


Calculate the positions of the nose R center and virtual tool nose point at the start point and end point of the circular arc, and command the position of the virtual tool nose point by the program.

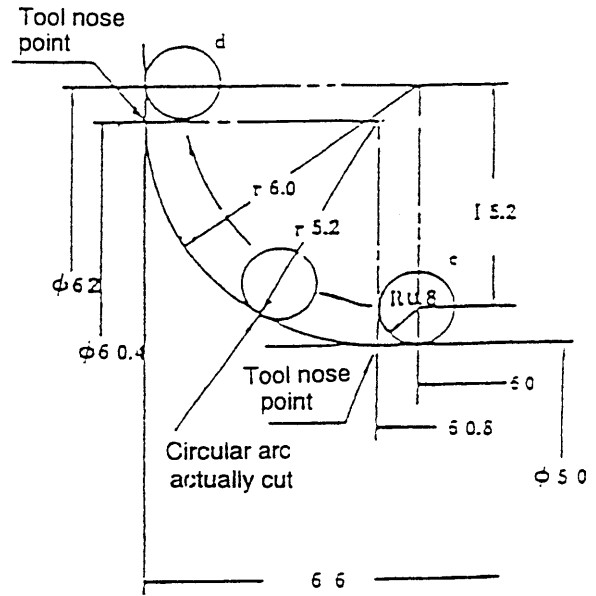


For the concave circular arc, command a circular arc "r" smaller by a tool nose radius.

(2) Program example with considering tool nose R compensation amount



Example 4



Example 5

```
G01 Z-50.0 F0.2
      X86.5 ← a
G03 X100.0 Z-56.8 R6.8 ← b
G01 Z-ΔΔ. Δ
```

```
G01 X50.0 F0.2
      Z-60.8 ← c
G02 X60.4 Z-66.0 R5.2 ← d
G01 XΔΔ. Δ
```

(3) When commanding the circular arc "r" by I and K instead of using R command a distance as far as the center of the circular arc "r", viewed from the center of the nose R at a circular cutting start point.

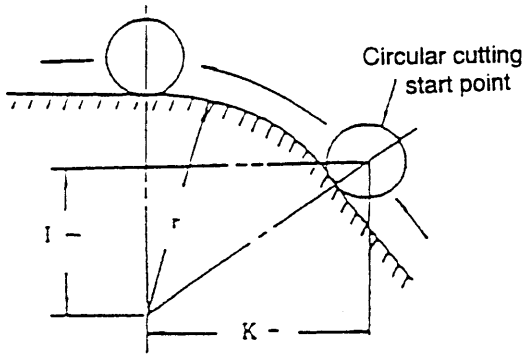
I : Command an element in the X-axis direction in terms of radius value.

K : Command an element in the Z-axis direction.

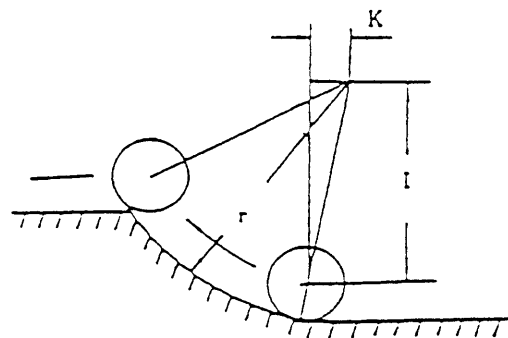
Programming of I and K for Examples 4 & 5 (1/4 circular arc)

|                             |                           |
|-----------------------------|---------------------------|
| G01 Z-50.0 F0.2             | G01 X-50.0 F0.2           |
| X86.4 ← a                   | Z-60.8 ← c                |
| G03 X100.0 Z-56.8 K-6.8 ← b | G02 X60.4 Z-66.0 I5.2 ← d |
| G01 Z-Δ. Δ                  | G01 XΔ. Δ                 |

As for the circular arc "r" other than a quarter circle, program I and K as a 2-axis command.



G03 X\_ Z\_ I-\_ K-

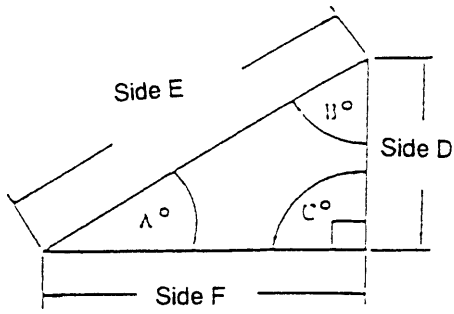


G02 X\_ Z\_ I+\_ K+

## 5-2 Calculation Formulas

### 5-2-1 How to Obtain Side and Angle of Right Triangle

If all inside angles of any triangle are added, a sum will be 180°. Therefore, as far as a right triangle is concerned, if 2 side lengths or 2 angles or 1 side length 2nd 1 angle are known, its all angles and side lengths can be known.



$$\sin A^\circ = \frac{D}{E}$$

$$\cos A^\circ = \frac{F}{E}$$

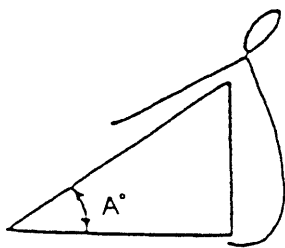
$$\tan A^\circ = \frac{D}{F}$$



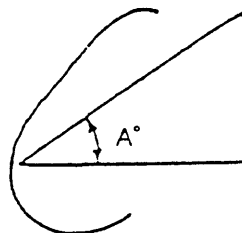
Bottom of the stream can be seen after a water mill stops.

Formulas (for right triangle)  $A^\circ + B^\circ + C^\circ = 180^\circ$

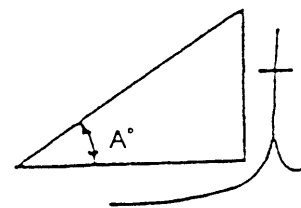
| Side and angle given   | Formula obtaining side or angle       |                              |
|------------------------|---------------------------------------|------------------------------|
| Angle "A" and side "D" | $E = \frac{D}{\sin A^\circ}$          | $F = \frac{D}{\tan A^\circ}$ |
| Angle "A" and side "E" | $D = E \times \sin A^\circ$           | $F = E \times \cos A^\circ$  |
| Angle "A" and side "F" | $D = F \times \tan A^\circ$           | $E = \frac{F}{\cos A^\circ}$ |
| Angle "B" and side "D" | $E = \frac{D}{\cos B^\circ}$          | $F = D \times \tan B^\circ$  |
| Angle "B" and side "E" | $D = E \times \cos B^\circ$           | $F = E \times \sin B^\circ$  |
| Angle "B" and side "F" | $D = F \times \frac{1}{\tan B^\circ}$ | $E = \frac{F}{\sin B^\circ}$ |
| Sides "D" and "E"      | $\sin A^\circ = \frac{D}{E}$          | $F = \sqrt{E^2 - D^2}$       |
| Sides "D" and "F"      | $\tan A^\circ = \frac{D}{F}$          | $E = \sqrt{D^2 + F^2}$       |
| Sides "E" and "F"      | $\sin B^\circ = \frac{F}{E}$          | $D = \sqrt{E^2 - F^2}$       |



Sine



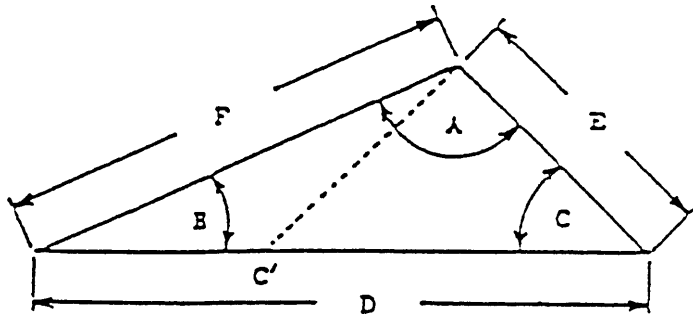
Cosine



Tangent

### 5-2-2 How to Obtain Side and Angle of Inequilateral Triangle

If some of sides and angles of a triangle are known, calculate remaining sides and angles as follows:



$$A^\circ + B^\circ + C^\circ = 180^\circ$$

(1) When 3 sides (E, F and D) are known;

$$\cos A^\circ = \frac{E^2 + F^2 - D^2}{2 \times E \times F} \quad \cos B^\circ = \frac{D^2 + F^2 - E^2}{2 \times D \times F} \quad C = 180^\circ - A^\circ - B^\circ$$

(2) When 2 sides (E and F) and an angle ( $A^\circ$ ) between them are known;

$$D = \sqrt{E^2 + F^2 - 2 \times E \times F \times \cos A^\circ}$$

(3) When 2 sides (E and F) and 1 opposite angle ( $B^\circ$ ) are known;

$$\sin C^\circ = \frac{F}{E} \times \sin B^\circ \quad A^\circ = 180^\circ - B^\circ - C^\circ$$

**Note)** Pay attention to existence of double solution as  $c'$  for a solution of  $c$ .

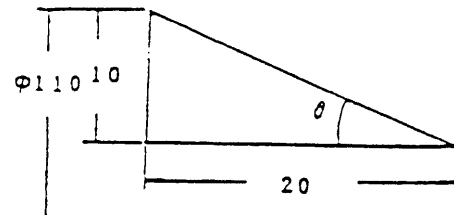
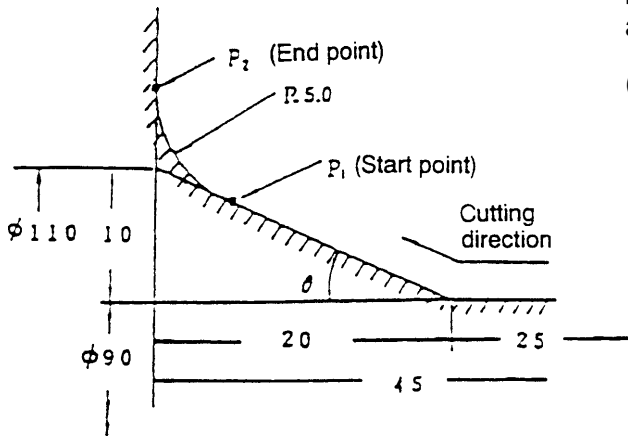
(4) When 1 side (D) and 2 angles are known;

$$E = \frac{\sin B^\circ}{\sin A^\circ} \times D \quad F = \frac{\sin C^\circ}{\sin A^\circ} \times D$$

### 5-2-3 How to Obtain Taper and Intersecting Point of Circular Arc

Obtain the command values of the start point ( $P_1$ ) and end point ( $P_2$ ) of the circular arc shown in the left figure.

- (1) Obtain the taper angle " $\theta$ " in the left figure.

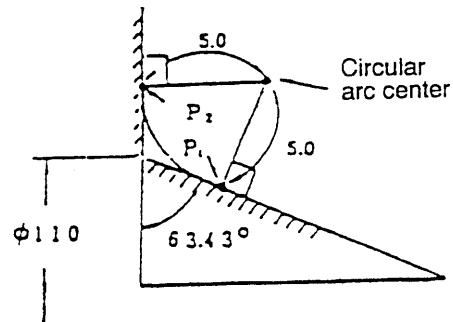
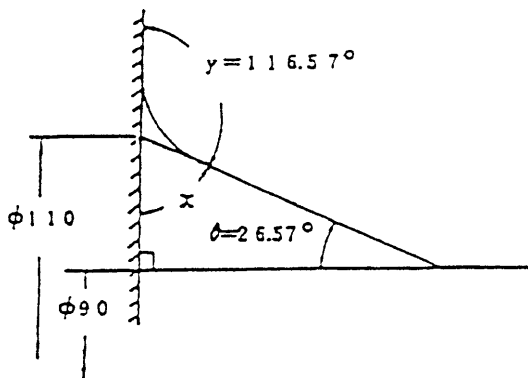


$$\theta = \tan^{-1} \frac{10}{20}$$

$$\theta = \tan^{-1} 0.5 = 26.57^\circ$$

- (2) Obtain the following angles " $x$ " and " $y$ " from " $\theta$ ".

- (3) From the start point and the end point of the circular arc to the center of the circular arc, draw lines which are as long as a radius of 5.0.

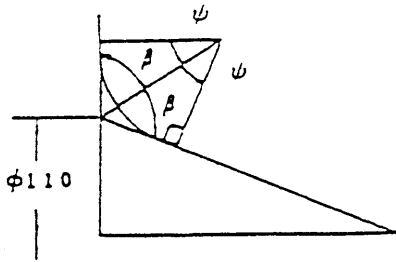


$$x = 90^\circ - 26.57^\circ = 63.43^\circ$$

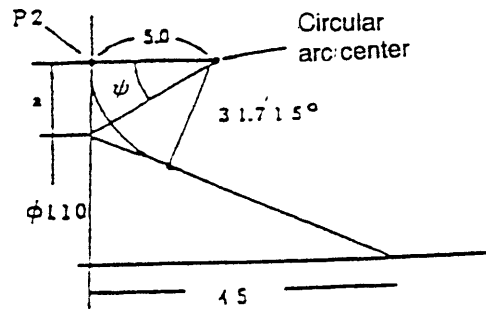
$$y = 180^\circ - x$$

$$y = 180^\circ - 63.43^\circ = 116.57^\circ$$

(4) Divide the thus created fan shape into two equally and obtain the angles “β” and “ψ”.



(5) Obtain the length of the side “a”.



$$a = \tan 31.715^\circ \times 5.0 = 3.08987$$

$$a \quad 3.09$$

$$\beta = 116.57^\circ \div 2 = 58.285^\circ$$

$$\psi = 90^\circ - 58.285^\circ = 31.715^\circ$$

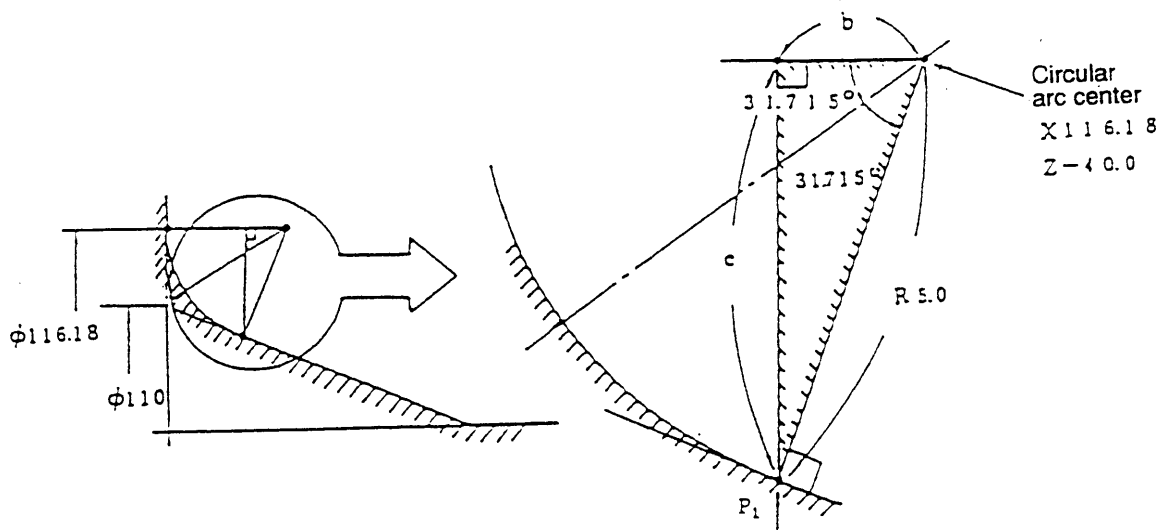
(6) The position of the end point (P<sub>2</sub>) of the circular arc is ; X of P<sub>2</sub> = 3.09×2+φ110=φ116.18

|                              |
|------------------------------|
| Point P <sub>2</sub> X116.18 |
| Z-45.0                       |

(7) The center position of the circular arc is;

|         |
|---------|
| X116.18 |
| Z-40.0  |

- (8) To obtain the position of the circular arc start point, create another right triangle and obtain the lengths of the sides "b" and "c".



Length of the side "b"

$$\begin{aligned}
 b &= 5.0 \times \cos(31.715^\circ + 31.715^\circ) \\
 &= 5.0 \times \cos 63.43^\circ \\
 &= 5.0 \times 0.47729 \\
 &= 2.24
 \end{aligned}$$

Length of the side "c"

$$\begin{aligned}
 c &= 5.0 \times \sin(31.715^\circ + 31.715^\circ) \\
 &= 5.0 \times \sin 63.43^\circ \\
 &= 5.0 \times 0.8943 \\
 &= 4.47
 \end{aligned}$$

- (9) Based on the calculations on the left, obtain the position of the circular arc start point ( $P_1$ ) from the circular arc center.

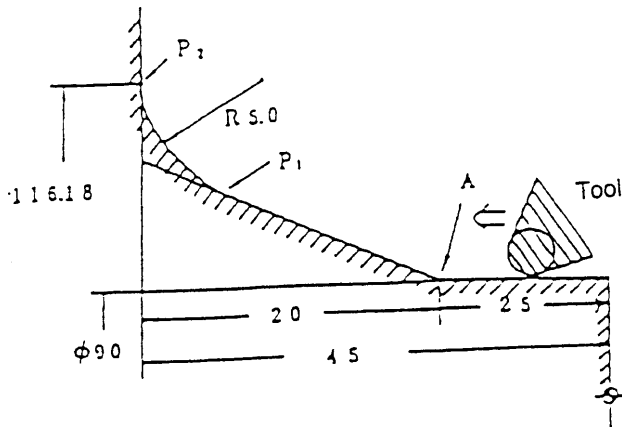
$$\begin{aligned}
 X \text{ of } P_1 &= \phi 116.18 - (C \times 2) \\
 &= \phi 116.18 - (4.47 \times 2) \\
 &= \phi 107.24
 \end{aligned}$$

$$Z \text{ of } P_1 = -40 - 2.24 = -42.24$$

|             |         |
|-------------|---------|
| Point $P_1$ | X107.24 |
|             | Z-42.24 |

A program with automatic calculation function of tool nose R compensation


Program the position of each intersecting point obtained by the above-mentioned calculations.



```

T..... M
G97 S..... M08
G00 X90.0 Z10.0 M03
G01 G96 Z3.0 F..... S.....
      Z-25.0 F..... ← A
      X107.24 Z-42.24 ← P1
G02 X116.18 Z-45.0 R5.0 ← P2
G01 X.....
    
```

### 5-2-4 Others

| Classification              | Calculation formula   | Remarks   |            |                       |            |  |
|-----------------------------|---|---|------------|-----------------------|------------|--|
| Spindle                     | Cutting speed "V" $V = \frac{\pi \cdot D \cdot N}{1000}$  | V. Cutting speed (m/min)  |            |                       |            |  |
|                             | Spindle speed "N" $N = \frac{V \cdot 1000}{\pi \cdot D}$  | N. Spindle speed (rpm)  |            |                       |            |  |
|                             | Tool nose Position "φD" $D = \frac{V \cdot 1000}{\pi \cdot N}$  | π. Number π(3.1416)<br>D. Workpiece diameter (mm)   |            |                       |            |  |
| Feed                        | Max. cutting feed "F" $F = \frac{5000}{N}$  | F. Feed per revolution (mm/rev)   |            |                       |            |  |
|                             | Approach feed rate "F" $F = \frac{2000}{N}$   | f. Feed per minute (mm/min)   |            |                       |            |  |
|                             | Feed rate per minute "f" $f = F \times N$   | L. Total cutting length (depth)<br>T. Time  |            |                       |            |  |
| Machining time              | Cutting time "T" $T = \frac{\pi \cdot D \cdot L}{V \cdot F \cdot 1000} = \frac{L}{N \cdot F}$   |   |            |                       |            |  |
| Thread cutting              | Thread lead $L = n \times P$  | L. Thread lead  |            |                       |            |  |
|                             | Lead limit $L(P) = \frac{5000}{N}$  | P. Thread pitch   |            |                       |            |  |
|                             | Spindle speed limit $N = \frac{5000}{L(P)}$   | n. No. of threads<br>N. Spindle speed   |            |                       |            |  |
|                             | Relation between the spindle speed and thread lead  | $5000 = L(P) \times N$<br>$\delta_1$ . Incomplete thread area (before machining)<br>$\delta_2$ . Incomplete thread area (after machining) |            |                       |            |  |
|                             | Incomplete thread area  | $\delta_1 = K_1 \times N \times L(P)$<br>$\delta_2 = K_2 \times N \times L(P)$<br>$K_1, K_2$ : constant                                   |            |                       |            |  |
|                             | Depth of thread cutting (diameter value)  |   |            |                       |            |  |
|                             | <table border="1"> <tr> <td>1. Metric thread 60°</td> <td>0.6495×P×2</td> </tr> <tr> <td>2. Unified thread 60°</td> <td>0.6134×P×2</td> </tr> </table>  | 1. Metric thread 60°  | 0.6495×P×2 | 2. Unified thread 60° | 0.6134×P×2 | <br>Convert unified and Whitworth thread pitches into millimeter. |
| 1. Metric thread 60°        | 0.6495×P×2  |   |            |                       |            |  |
| 2. Unified thread 60°       | 0.6134×P×2  |   |            |                       |            |  |
| Finishing surface roughness | Surface roughness based on feed rate and tool nose radius<br>$H_{max} = \frac{F^2}{8 \cdot R}$<br>Feed rate based on surface roughness and tool nose radius<br>$F = \sqrt{8R \cdot H_{max}}$  | H <sub>max</sub> Max. surface roughness<br>F. Feed rate (mm/rev)<br>R. Tool nose radius   |            |                       |            |  |
| Power and depth             | Power required for cutting "KW"<br>$KW = \frac{K \cdot t \cdot F \cdot V}{6120 \times 0.8}$<br>Max. depth of cut "t"<br>$t = \frac{6120 \times KW}{K \cdot V \cdot F} \times 80\%$<br>Required horsepower "HP" $HP = \frac{KW}{0.75}$ | K. Specific cutting resistance<br>t. Depth of cut (mm)<br>F. Feed rate (mm/rev)<br>V. Cutting speed (m/min)<br>Motor efficiency 80%       |            |                       |            |  |



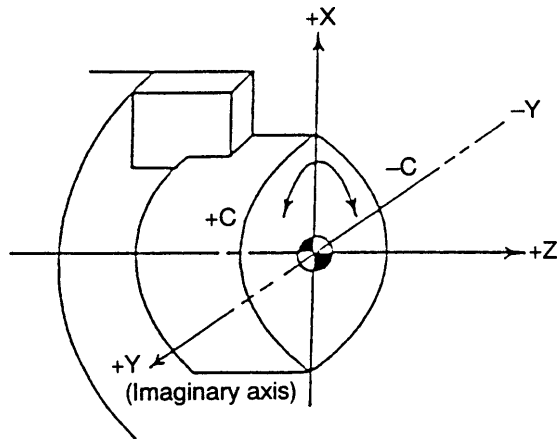


17. Reference point return .....G27: Reference point return check  
 G28: Reference point return  
 In the reference point return, rotating axis processing is performed. (The reference point return is completed within 360°)  
 G29: Return from the reference point
18. Feed per minute/feed .....G98: Feed per minute (mm/min)  
 per revolution G99: Feed per revolution (mm/REV) (synchronized with that of rotating tool spindle)  
**Note)** *In this case, it is necessary to set PC on the rotating tool spindle, and of its feedback pulse to be 4069P/REV.*
19. Tool position offset .....Tool offset for C-axis is not available.
20. Backlash compensation .....32767 pulse MAX
21. Manual feed.....Rapid traverse (RAPID)/Manual jog feed (JOG)/Manual handle feed (STEP)
22. Manual reference point return .....Processed by the high speed and reduction LS using type rotating shaft.
23. Operation support switch .....Single block  
 Block skip  
 Dry run (processed through converting the value of mm/min designated by X and Z into that of deg/min.)  
 Machine lock  
 Manual absolute (fixed to "ON" for C-axis as well as X and Z axes.)
24. Feed hold (Halt).....Effective
25. Buffering function .....Effective
26. Inch/Metric conversion .....Regarding C-axis, both inch and mm system is 0.001°.
27. The second reference point .....Possible  
 return
28. Automatic coordinate system .....At the time of manual reference point return, setting coordinate system is set automatically.
29. Work coordinate system shift .....Set it by work shift screen.
30. Stored stroke limit.....Ineffective for C-axis
31. Chamfer/Corner R .....Impossible for C-axis
32. Return to machining .....Automatic return to the position changed over to the interrupting point manual mode available for C-axis as well.

## 6-3 Program

### 6-3-1 Coordinate Axis

The C-axis is included in the ordinary cutting coordinate system. Each coordinate axis and signs are defined as follows.



As a matter of fact the Y-axis not exists, however, prepare a program as if imaginary Y-axis exists.

### 6-3-2 Plane Selection of G17, G18, G19

Designate a plane executing circular interpolation and tool radius compensation etc. by G17, G18 or G19.

A plane should be designated either turning or machining by rotating tool.

Command form

G17  $X_p$ - $Y_p$ ;  $X_p$ - $Y_p$  plane

G18  $Z_p$ - $X_p$ ;  $Z_p$ - $X_p$  plane

G19  $Y_p$ - $Z_p$ ;  $Y_p$ - $Z_p$  plane

Provided that,  $X_p$ : X-axis or it's parallel axis.

$Y_p$ : Y-axis or it's parallel axis.

$Z_p$ : Z-axis or it's parallel axis.

Select G18 (A plane of  $Z_p$ - $X_p$ ) at normal turning, G19 (A plane of  $X_p$ - $Y_p$ ) at machining from direction of Z-axis.

Since a condition at power on is G18, plane selection must be done by direction of machining.

**(Note) 1.** If a plane is not fixed at a block commanded by G17, G18 or G19, it becomes an alarm.

### 6-3-3 Miscellaneous Function for Rotating Tool (M Code)

In case of hole machining, you can use these codes to specify start, stop and reverse rotation of the tool.

M13 Rotating tool connection + Rotating tool forward rotation

M14 Rotating tool connection + Rotating tool reverse rotation

M15 Rotating tool stop + Spindle positioning

(In case of simple spindle stop, it is available the same spindle stop code M05 for the turning spindle. Never fail to use M05 when stopping the rotating tool in tapping process.)

M40 C-axis connection release

(Release is available by the low-speed range selection code of the cutting spindle.)

M41 C-axis connection release

(Release is available by the high-speed range selection code of the cutting spindle.)

M43 C-axis connection (Incl. spindle indexing)

M44 Rotating tool connection

M45 Rotating tool connection release

- Note:**
- 1) *For C-axis specifications, the hole machining canned cycles are added both to X-axis and Z-axis.*
  - 2) *H is used to specify C-axis incremental.*
  - 3) *Execute the reference point return of the C-axis after connection of the C-axis.*

### 6-3-4 Fixed Cycle for Hole Making G80~G89, G831, G841, G861

With this function, machining cycle such as drilling, tapping or boring can be commanded by one block.

Furthermore, in case of making the same hole repeatedly, just command hole position and it is very effective to simplify a program.

(1) Command form

$$\begin{bmatrix} G198 \\ G199 \end{bmatrix} G\_X\_Z\_R\_D\_Q\_P\_C(H)\_L\_F\_E\_ ;$$

G198/G199 : Return point

G\_ : G code for fixed cycle (G81~G89, G831, G841, G861)

X\_ : Hole position (Note 3)

Z\_ : Position of Z point (Note 3)

R\_ : Position of R point (Coordinate value of diameter)

D\_ : Position of D point (Incremental diametral command from R point)

Q\_ : Cutting depth of G831 or C83 or shift amount of G861, always radius value.

P\_ : Dwelling time

C\_ : Rotating angle of C-axis

L\_ : Number of times of repetition

F\_ : Cutting feedrate

E\_ : Cutting feedrate

**(Note 1)** *If omit a number of times of repetition (L), it deems as L=1. If L=0 is commanded, move to hole making position but hole making is not executed.*

**(Note 2)** *I, J or K is used at G83, G831 or G861 as well.*

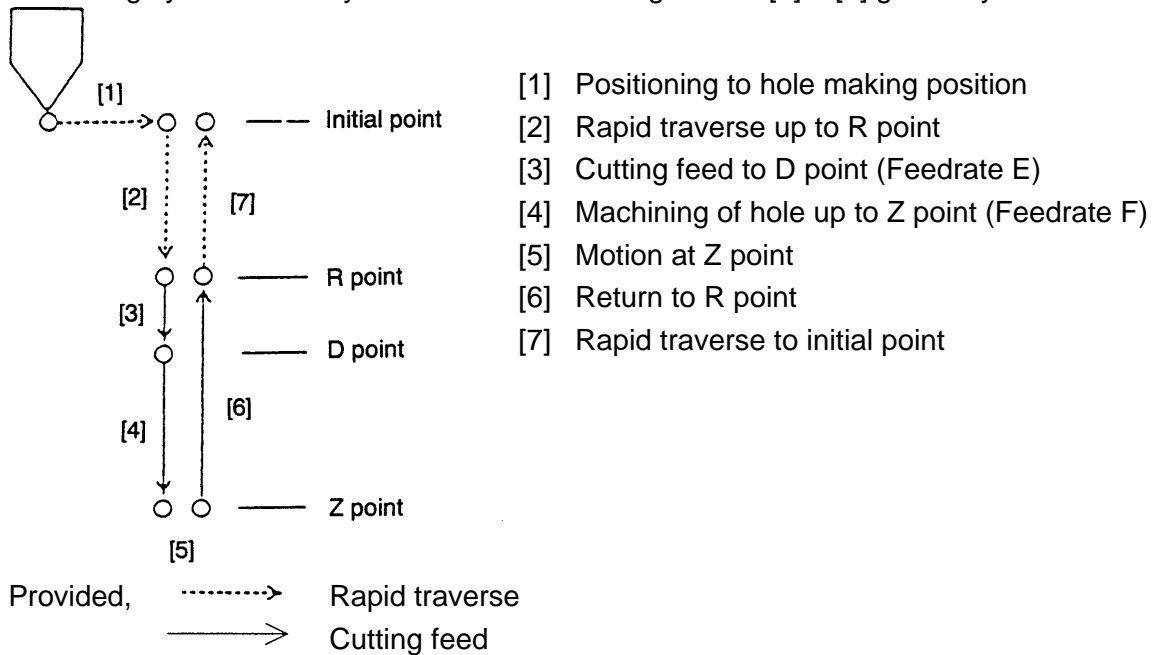
**(Note 3)** *Command a position of Z point by an address of axis of hole making axis. Command a position of hole making by an address of axis other than hole making axis.*

**(Note 4)** *R and Z points, P, Q, I, J and K etc. are modal during fixed cycle.*

**(Note 5)** *Incremental command on B-axis can't be used during fixed cycle of hole making.*

## (2) Machining cycle

Machining cycle of fixed cycle consists of following motion [1] ~ [7] generally.



**(Note 1)** Motion between [4] and [6] does not stop by single block.

**(Note 2)** If E is omitted, moving section by feedrate E moves by feedrate F.

**(Note 3)** If omit D, motion [3] is ignored.

**(Note 4)** Command E at tapping cycle G84 or G841, it becomes feedrate of motion [6].

## (3) Return point

Return point of fixed cycle command following G code.

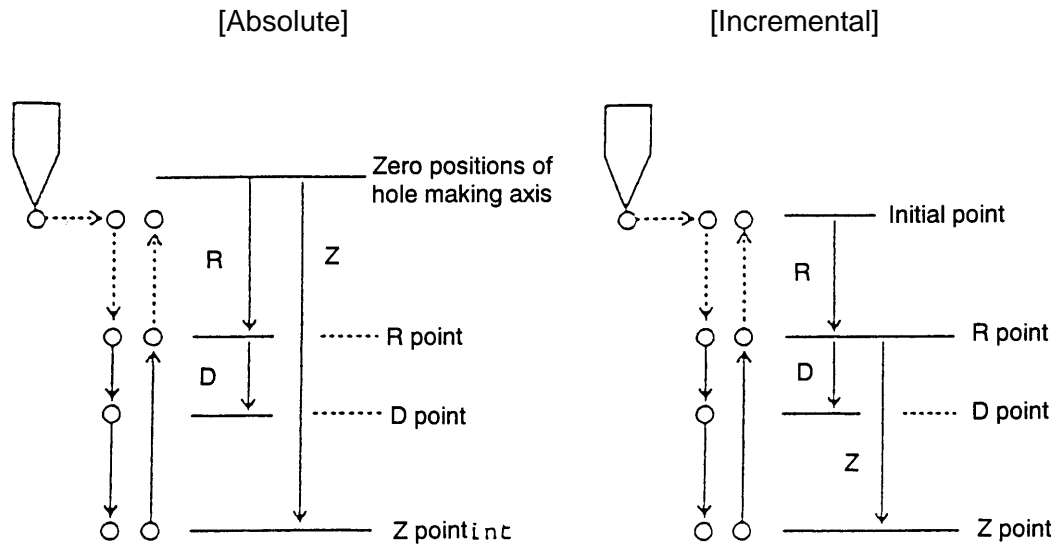
G198 Initial point level return

G199 R point level return

**(Note)** An initial point is a position of hole making axis at time of fixed cycle mode from a condition of fixed cycle cancel.

(4) "R point", "Z point" and "D point"

R and Z points are available both absolute and incremental command, however, D point is commanded by incremental always.



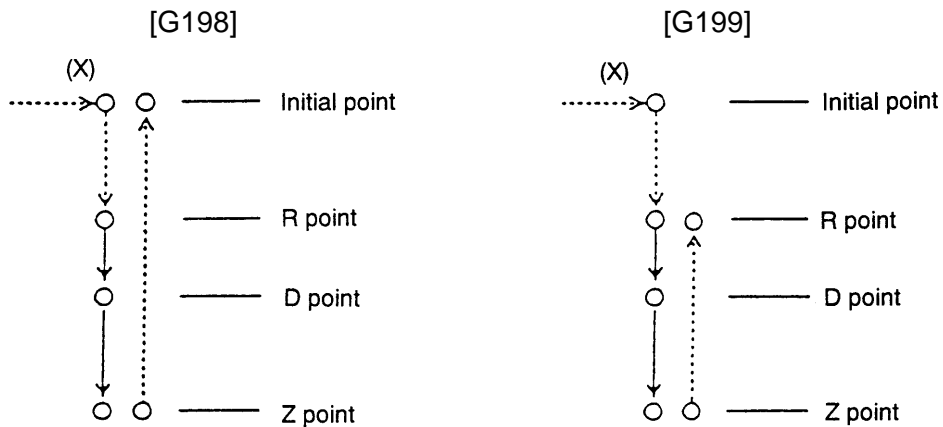
**(Note)** D point is incremental position from R point and at the time of machining from diametral direction, indicates by diametral value.  
Incremental command of the B axis by "D" is not available during fixed cycle.

(5) Explanation of motion of fixed cycle

In this explanation of motion of fixed cycle, positioning axis hole making position is X and hole making axis is Z.

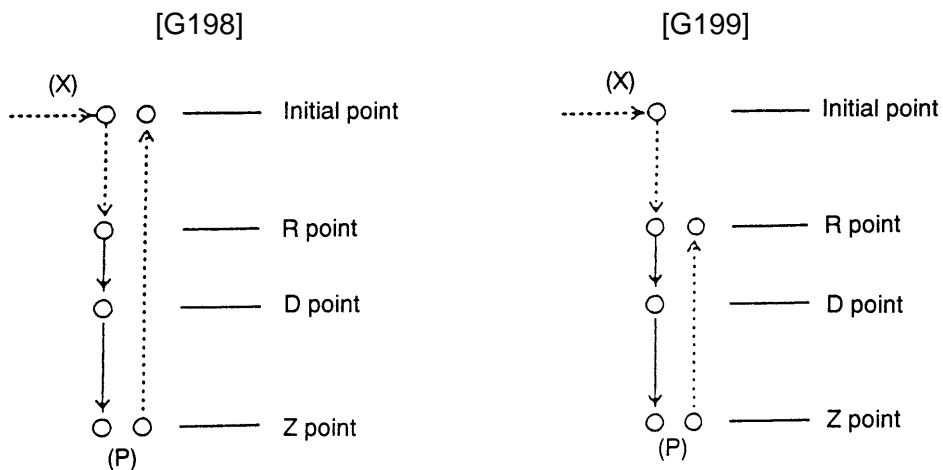
(a) G81 (Drilling, Spot drilling)

```
[ G198 ] G81 X_Z_R_D_C(H)_L_F_E_ ;
[ G199 ]
```



(b) G82 (Drilling, Counter boring)

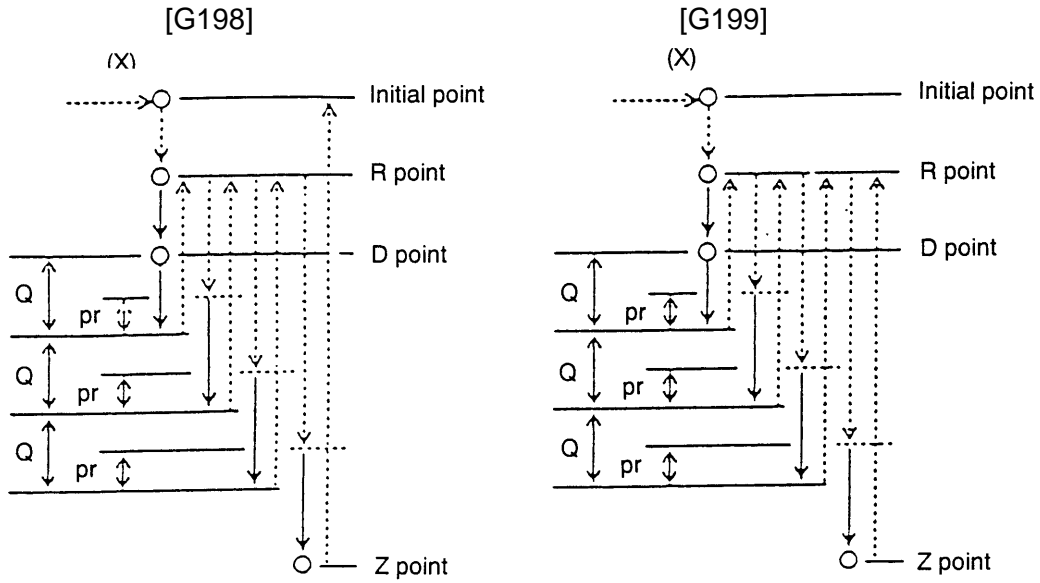
```
[ G198 ] G82 X_Z_R_D_P_C(H)_L_F_E_ ;
[ G199 ]
```



(P): Dwell

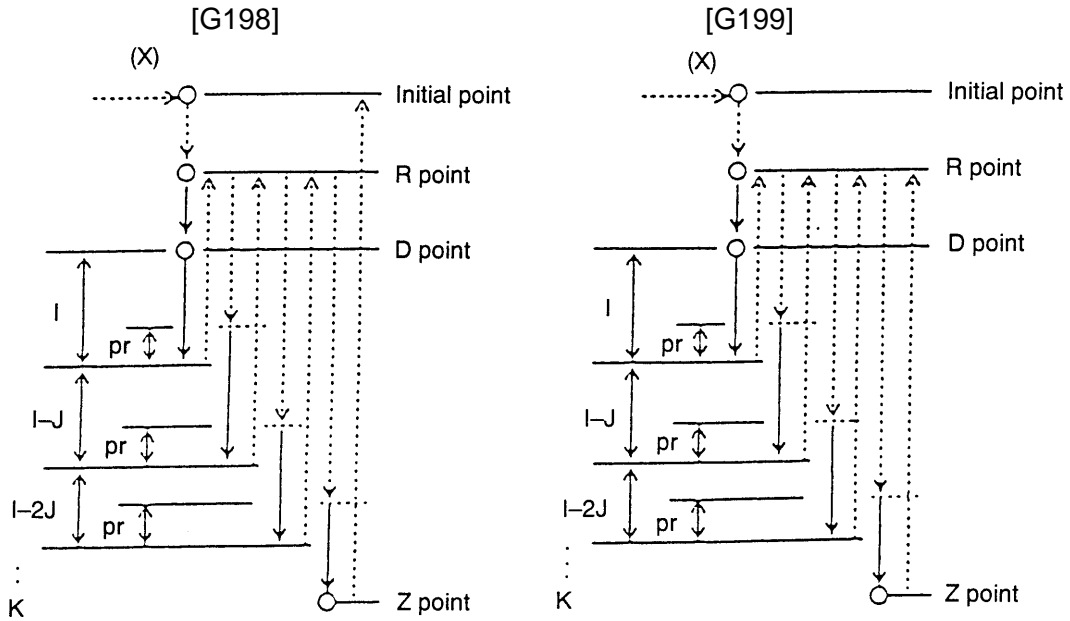
(c) G83 (Deep hole drilling)

```
[ G198 ] G83 X_Z_R_D_Q_C(H)_L_F_E_ ;
[ G199 ]
```



Set a clearance amount Pr on the parameter No.6222.

```
[ G198 ] G83 X_Z_R_D_I_J_K_C(H)_L_F_E_ ;
[ G199 ]
```



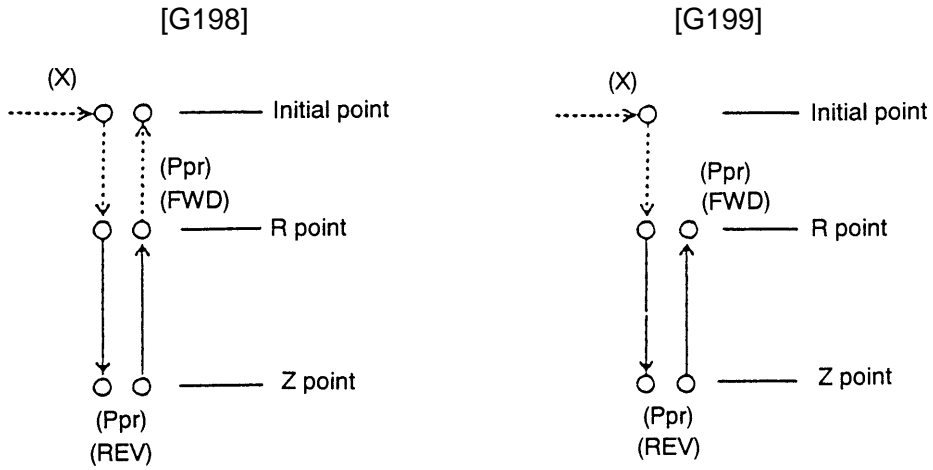
Provided; I : Initial value of cutting amount (Positive value)  
 J : Subtraction value after the second time (Positive value)  
 K : Final value of cutting amount (Positive value)

} Always radius value

**(Note)** In case of existence of command Q before command a variable pitch by I, J or K, command Q as zero.

(d) G84 (Tapping)

[ G198 ] G84 X\_Z\_R\_P\_C(H)\_L\_F\_E\_ ;  
 [ G199 ]



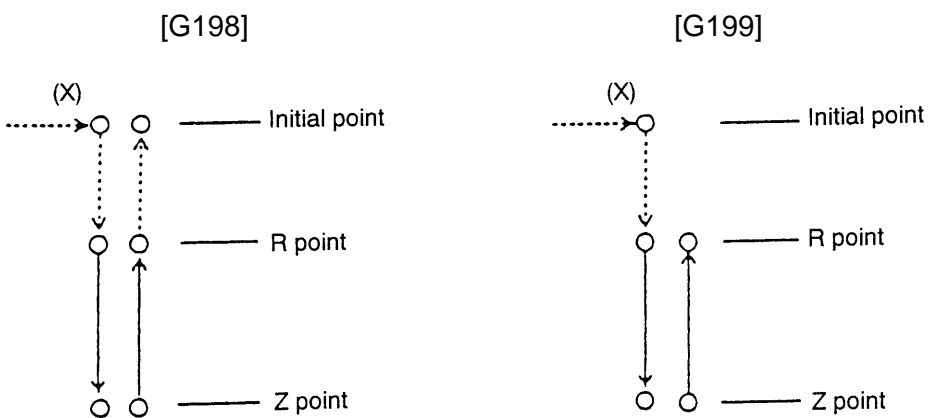
(FWD) : Forward rotation of tool  
 (REV) : Reverse rotation of tool  
 (Ppr) : Dwell (Parameter setting)

**(Note 1)** A dwell by command P can be ineffective by parameter setting.

**(Note 2)** Feed hold and feedrate override are ineffective while cutting.

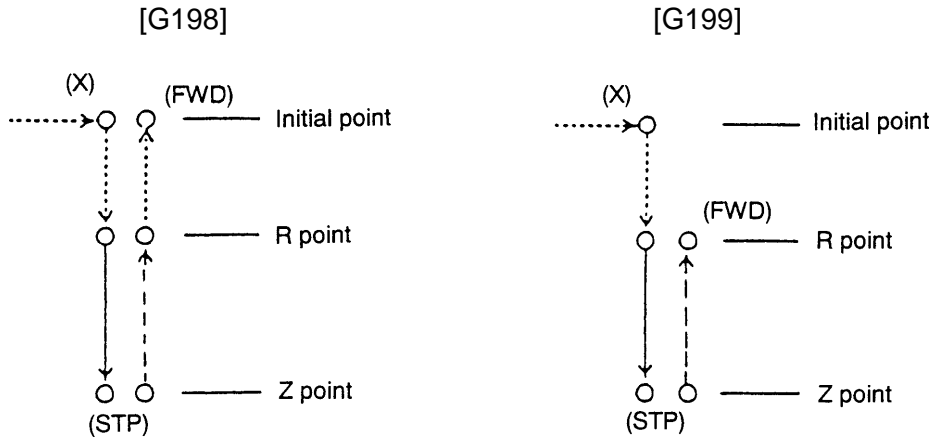
(e) G85 (Boring)

[ G198 ] G85 X\_Z\_R\_C(H)\_L\_F\_ ;  
 [ G199 ]



(f) G86 (Boring)

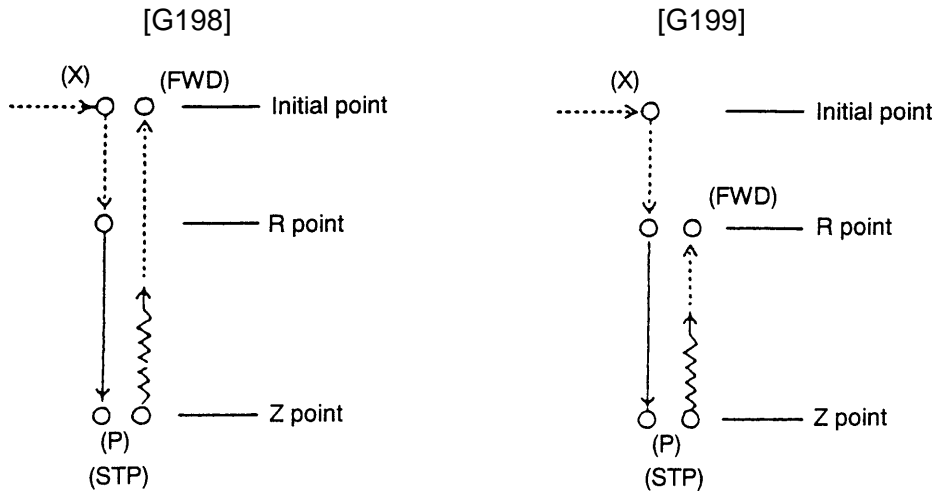
[ G198 ] G86 X\_Z\_R\_C(H)\_L\_F\_ ;  
 [ G199 ]



(FWD) : Forward rotation of tool  
 (STP) : Stop of tool

(g) G88 (Boring)

[ G198 ] G88 X\_Z\_R\_P\_C(H)\_L\_F\_ ;  
 [ G199 ]

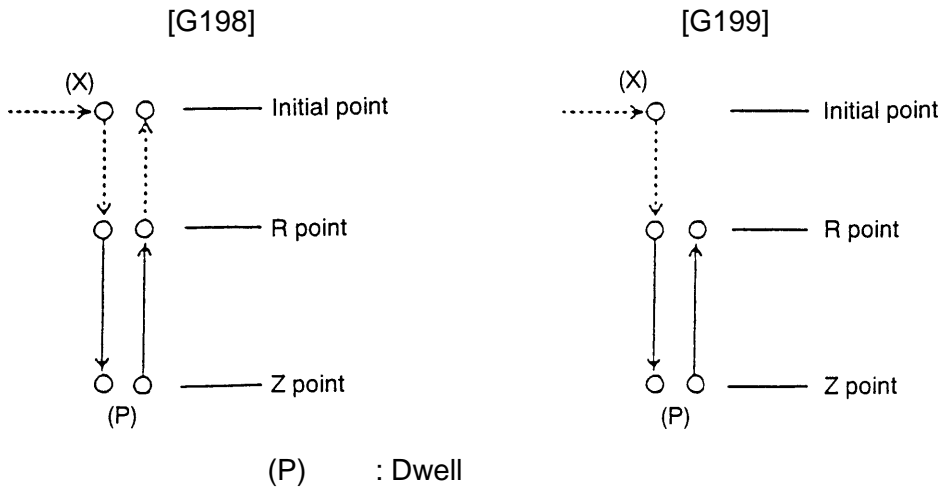


(P) : Dwell  
 (FWD) : Forward rotation of tool  
 (STP) : Stop of tool  
 ~~~~~> : Manual feed

**(Note)** A tool reaches to Z point and stop a rotation of tool after dwell, it becomes single block stop condition automatically. If select a manual mode, manual feed is available.  
 If select an automatic mode then press "START" button, restart an automatic operation.

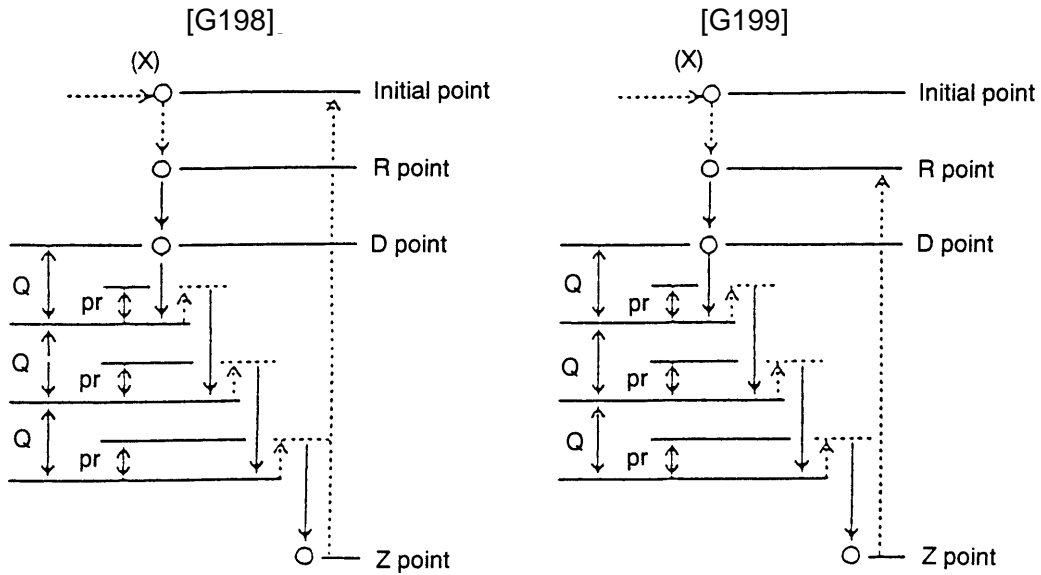
(h) G89 (Boring)

[ G198 ] G89 X\_Z\_R\_P\_C(H)\_L\_F\_ ;  
[ G199 ]



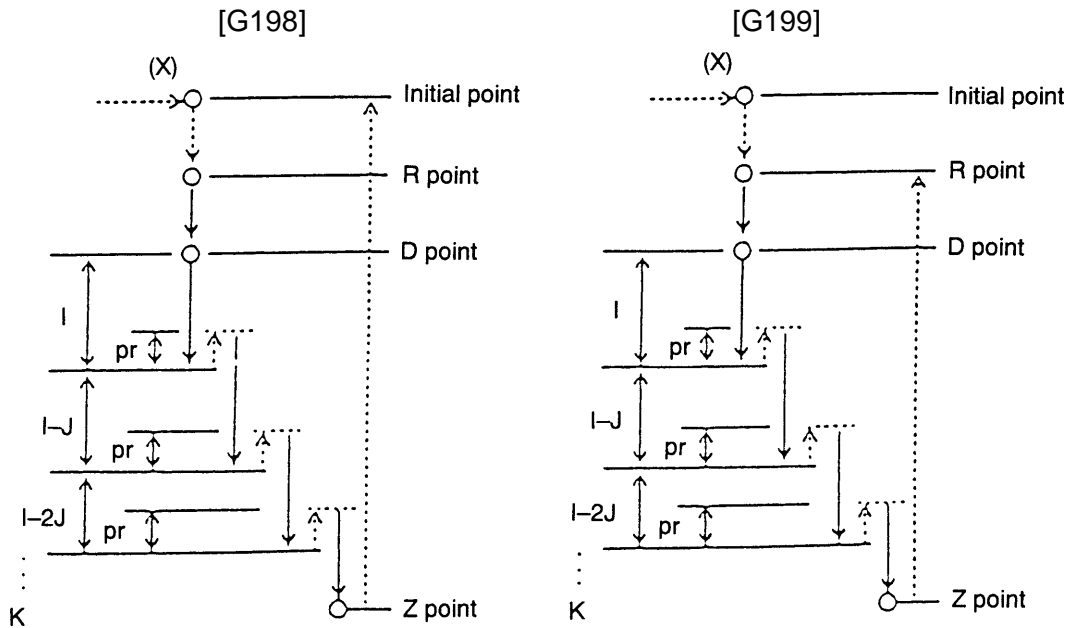
(i) G831 (High speed deep hole drilling)

```
[ G198 ] G831 X_Z_R_D_Q_C(H)_L_F_E_ ;
[ G199 ]
```



Set a clearance amount Pr on the parameter No.6222.

```
[ G198 ] G831 X_Z_R_D_I_J_K_C(H)_L_F_E_ ;
[ G199 ]
```



Provided; I : Initial value of cutting amount  
(Positive value)  
J : Subtraction value after the second time (Positive value)  
K : Final value of cutting amount  
(Positive value)

} Always radius value

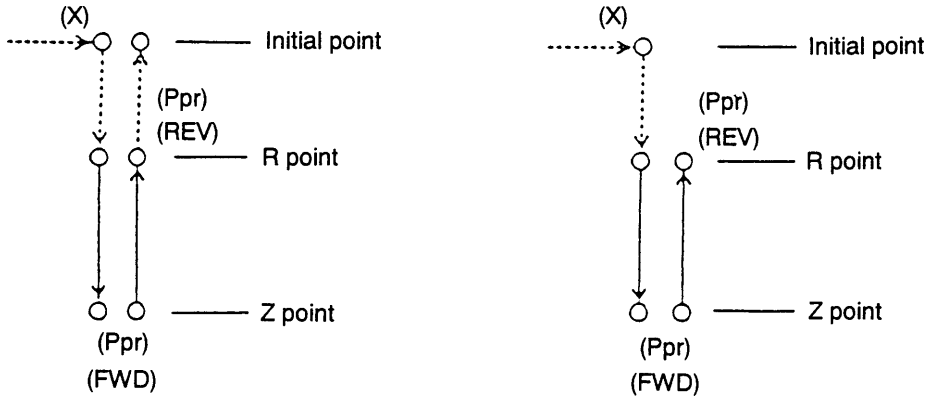
**(Note)** In case of existence of command Q before command a variable pitch by I, J or K, command Q as zero.

(j) G841 (Reverse tapping)

```
[ G198 ] G841 X_Z_R_P_C(H)_L_F_E_ ;
[ G199 ]
```

[G198]

[G199]



(FWD) : Forward rotation of tool  
 (REV) : Reverse rotation of tool  
 (Ppr) : Dwell (Parameter setting)

**(Note 1)** A dwell by command P can be ineffective by parameter setting.

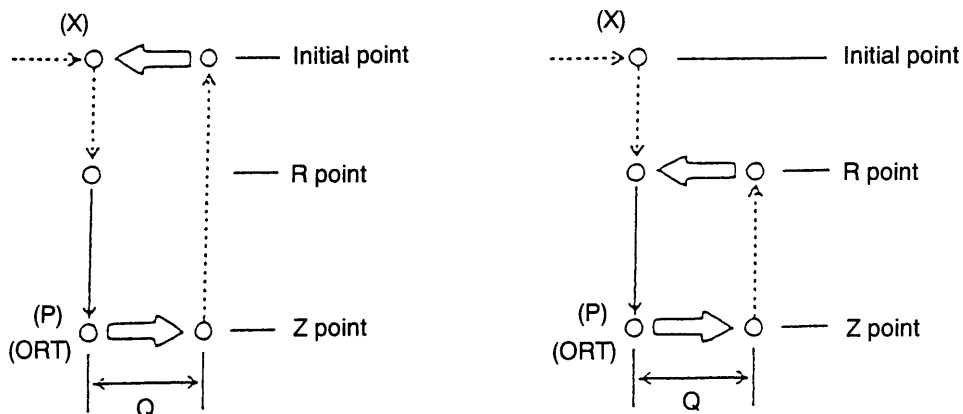
**(Note 2)** Feed hold and feedrate override are ineffective while cutting.


(k) G861 (Fine boring)

```
[ G198 ] G861 X_Z_R_P_Q_C(H)_L_F_E_ ;
[ G199 ]
```

[G198]

[G199]



(P) : Dwell  
 (ORT) : Tool orientation and stop  
 : Shift (Rapid traverse)

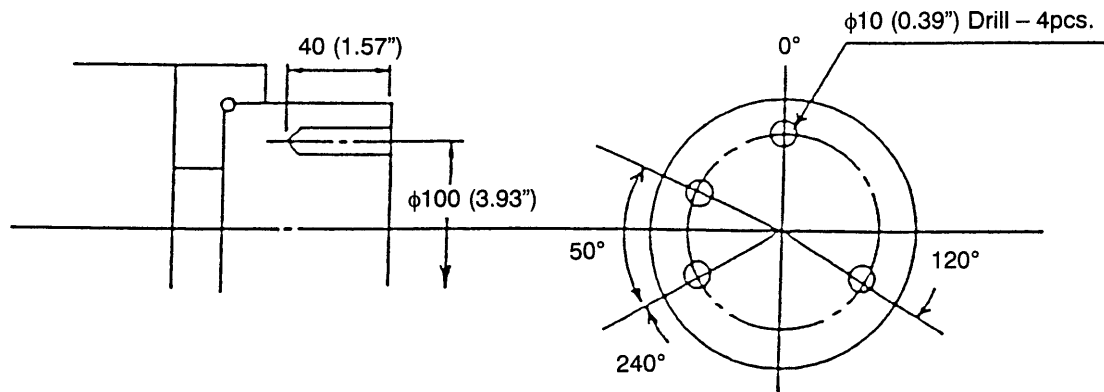
**(Note)** Set a shift amount Q at the parameter No.6212.

## (6) Precautions

- (1) When single block is ON, stop at the end point of motion [1] [2], [3] and [7]. In this case a feed hold lamp light at the end point of motion [1], [2], [3] and the end point of motion [7] if remain the number of times of repetition.  
A cycle motion between [4] and [6] other a tapping cycle (G84, G841) can be stopped on single block by setting of parameter.
- (2) If the "FEED HOLD" button is pressed at the motion [4] ~ [6] of G84 or G841, a feed hold button lights immediately and a motion stops after execution of motion continuously up to [7].
- (3) A feedrate override is fixed at 100% during a motion [4] ~ [6] of G84 or G841. Effective or ineffective of dry run can be selected by setting of parameter.
- (4) Q, P, I, J and K should command in the block contains axis command. P, Q, I, J and K are not handled as a data of fixed cycle in the other blocks.  
Also, P, Q, I, J and K are not handled as a data of fixed cycle in the block commanded by G code of 00 group except G09.
- (5) A fixed cycle mode will be cancelled by commanding G80 or G code of 01 group such as G00, G01 etc.
- (6) If M or S is commanded in a block of fixed cycle command, it issues the first motion [1], positioning to hole making position.  
If number of times of repetition (L) is commanded, M and S issues at the first time only.
- (7) Command by positive value for the numerals of P, Q, I, J, K, L, F or E etc.
- (8) Command Q, I, J and K by radius designation always.
- (9) R and D are diamtral designation.
- (10) When G17~G19 going to be changed, execute it after a fixed cycle is cancelled.

### 6-3-5 Program Example

Example 1: Drilling (Z-axis Rotating Tool)

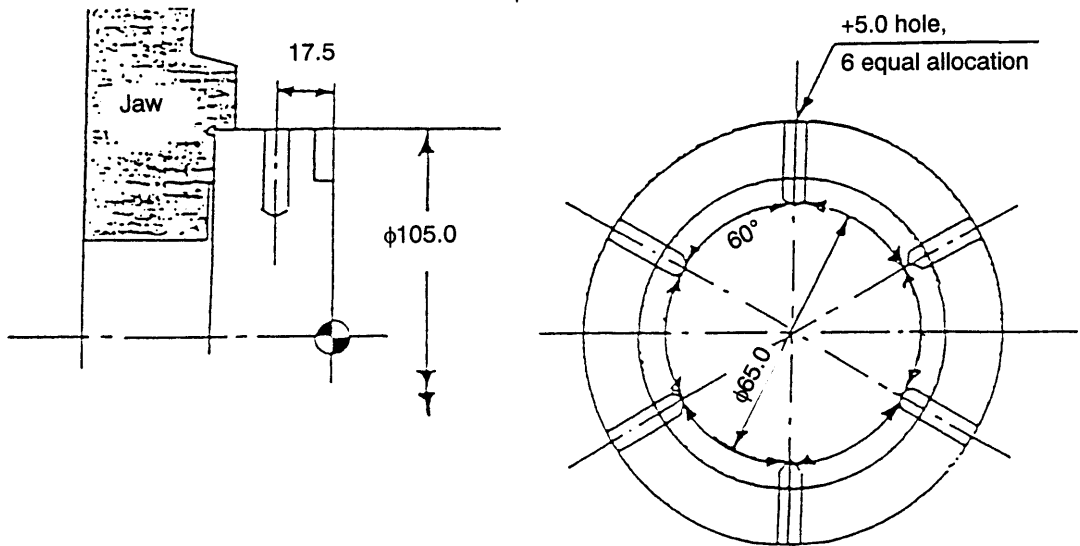
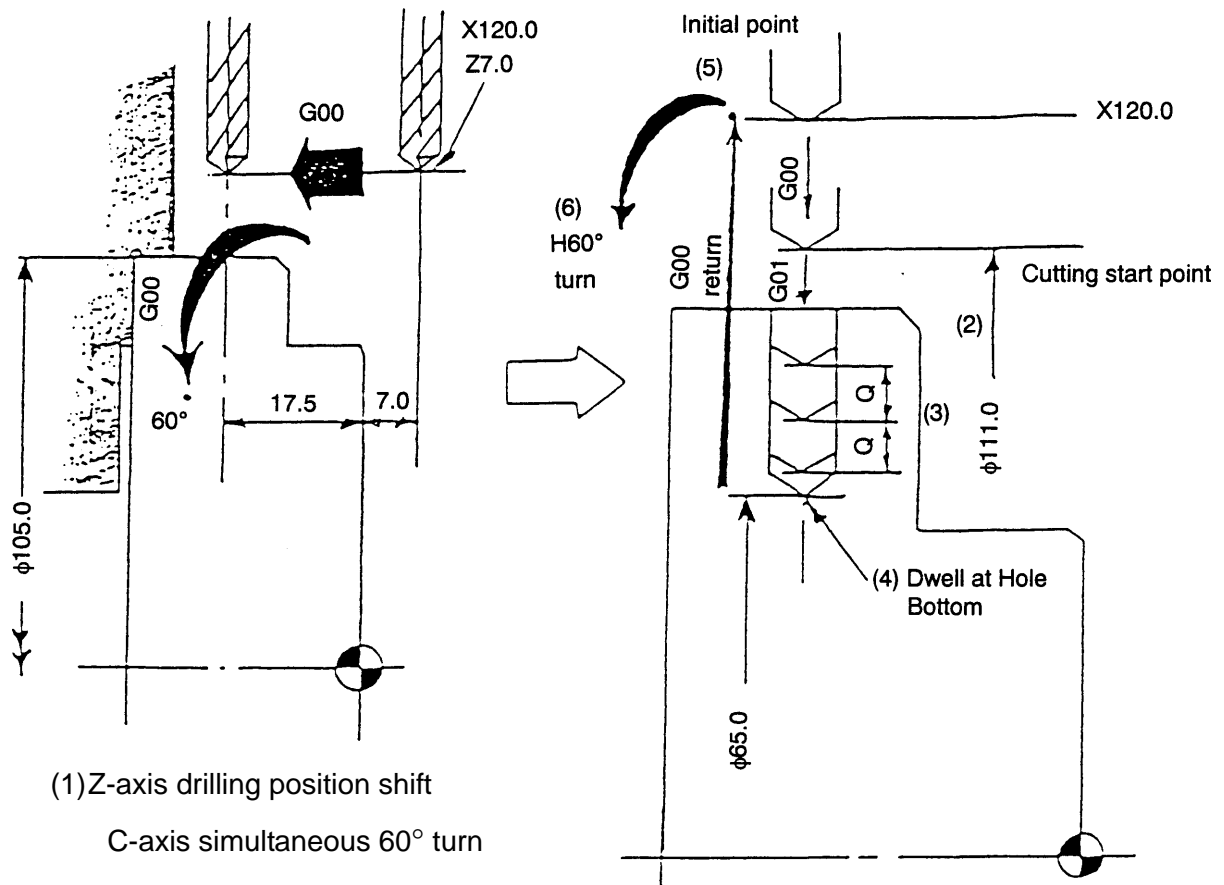


```

OΔΔΔΔ
G28 U0
G28 W0
G18 ..... X-Z plane designation
  }
Turning
  }
N600
T0600 M40 ..... 6th turret face selection•Spindle low speed sideselection
                  C-axis connection release
G17 ..... X-Z plane designation
M43 ..... C-axis connection
G28 H0 ..... C-axis zero return
G50 C0 ..... C-axis coordinate setting
G97 S800 M08 ..... Rotating tool rotating speed command. Coolant ON
G00 X100.0 Z10.0 C0 M13 ..... Axial travel + Rotating tool forward rotation start
Z2.0
G01 G98 Z-40.0 F120
G00 Z2.0
      C120.0 ..... C-axis absolute command
G01 Z-40.0
G00 Z2.0
      C240.0
G01 Z-40.0
  
```

G00 Z2.0  
    H50.0 ..... C-axis incremental command  
G01 Z-40.0  
G00 Z2.0 M09  
G00 X200.0 Z200.0 M05 ..... Return to index position + Rotating tool rotation stop  
G28 H0 ..... C-axis zero return  
M45 ..... Rotating tool connection release  
M01  
.  
.  
.  
.

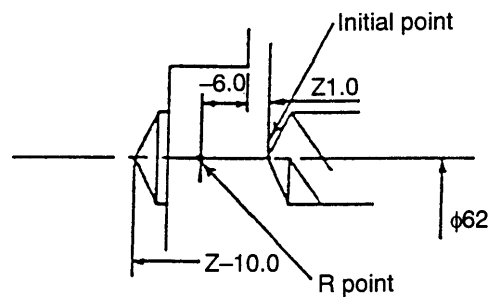
Example 2: Drilling (X-axis Rotating Tool)



|                                                    |                                            |
|----------------------------------------------------|--------------------------------------------|
| N400                                               |                                            |
| T0400 M40                                          |                                            |
| G19                                                |                                            |
| G23 .....                                          | Stored stroke 2 turned off                 |
| G97 S100 M05                                       |                                            |
| M43 .....                                          | C-axis connection                          |
| G28 U0 H0 .....                                    | X-axis and C-axis zero return              |
| G50 C0 .....                                       | C-axis coordinate setting                  |
| G98 S1270 M08 .....                                | Rotating tool spindle speed (Feed: mm/min) |
| G00 X120.0 Z7.0 M13 .....                          | Rotating tool forward rotation             |
| G198 .....                                         | Initial point return command               |
| G83 Z-17.5 H60.0 X65.0 R110.0 P0.5<br>Q3.0 L6 F127 | X-axis peck drilling cycle                 |
| G80 .....                                          | Peck drilling cycle canceled               |
| G28 U0 H0 M09                                      |                                            |
| G00 Z200.0 M05                                     |                                            |
| M45 .....                                          | Rotating tool connection release           |
| G99 M40 .....                                      | C-axis connection release (Feed: mm/rev)   |
| G22 .....                                          | Stored stroke 2 turned on                  |
| M01                                                |                                            |

Example 3: Drilling (Z-axis Rotating Tool)

N1000 (D6.5 – DRL)  
T1000 M40  
G17 ..... X-Y plane designation  
M43 ..... C-axis connection  
G28 H0 ..... C-axis zero return  
G50 C0 ..... C-axis coordinate setting  
G97 S1500 M08  
G00 X62.0 Z5.0 M13 ..... Rotating tool forward rotation  
G98 G01 Z1.0 F5000 ..... Feed per minute (mm/min)  
G198 ..... Initial point return  
G81 Z-10.0 H60.0 R-6.0 P1.0 L6 F130  
G80 ..... Fixed cycle cancel  
G00 Z5.0  
G99 M40 ..... C-axis connection release  
G00 X200.0 Z200.0 M05  
M45 ..... Rotating tool connection release  
M01





Example 5: End-milling (Z-axis Rotating Tool)

N200 (D10.0 – MIL)

T0200 M40

G17

M43

G28 H0

G50 C0

G97 S500 M08

G00 X80.0 Z5.0 C-15.0 M13

G98 G01 Z1.0 F3000

Z-5.0 F25

C15.0 F50

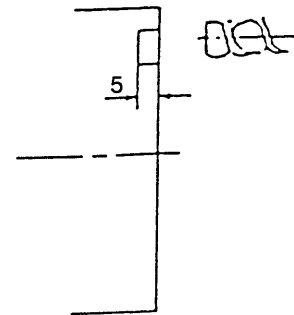
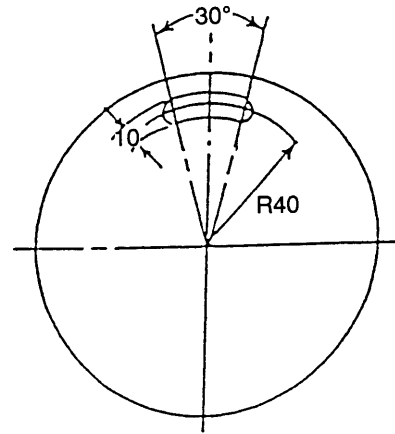
G00 Z5.0

G99 M40

G00 X200.0 Z200.0 M05

M45

M01



## 6-4 Polar Coordinate Interpolation Function

### 6-4-1 Polar Coordinate Function

A workpiece can be machined into an arbitrary shape with the linear axis (X-axis) and rotary axis (C-axis).

If G121 is specified, polar coordinate interpolation is put into effect and a virtual coordinate system is set assuming the zero point of the absolute coordinate system as that on the X-C plane.

A polar coordinate interpolation is executed on this plane.

If G121 is specified, a current C-axis position is assumed as "0". Therefore, it is necessary to return the C-axis to the program zero point before specifying G121.

Polar coordinate interpolation allows cutting by G01, G02 or G03. A feedrate is 00 mm/min. Either absolute programming (X, C) or incremental programming (U, H) is available.

Polar coordinate interpolation allows selection of diameter/radius designation for X-axis and C-axis commands.

In case of execution of circular interpolation (G02, G03), designation of radius of arc performs by the address R.

Polar coordinate interpolation is executed over a shape after tool radius compensation.

However, command G120 and G121 at the cancel mode of tool radius compensation (G40).

G120 command turns off the polar coordinate interpolation function.

### 6-4-2 G Function

There are limit of commandable G code during G121 mode.

|                    |                         |
|--------------------|-------------------------|
| Commandable G code | G00 G01 G02 G02 G03 G04 |
|                    | G09 G40 G41 G42         |
|                    | G65 G66 G67             |
|                    | G98 G99                 |

- (Note)** 1. Command the G120 or G121 in the individual block. If it is not individual, it becomes an alarm.
2. A plane (any one of the G17, G18 or G19) before the G121 has commanded is cancelled once by a command of the G121 and returns by a command of the G120.
3. During a polar coordinate interpolation should be G98 (feed per minute).
4. During the G121 mode, the T or S command is not available.
5. The following functions are not available for a block during the G121 mode.
- Program restart
  - Block restart
  - Return to the interrupted point during machining
  - Manual intervention by the manual absolute ON
6. Prepare a program the coordinate axis of the X should be corresponded to the real coordinate axis of the X.
7. The G00 command is available during the polar coordinate interpolation mode.

However, the end point only changes to the position of polar coordinate system and a path is not changed for the G00 block.

Also, a positioning system becomes a linear type and a compound travelling speed becomes a setting speed at the parameter (No.1468). A positioning system and compound travelling speed is the same for the axis have no relation with the polar coordinate interpolation.

*Example)* When G00 X100.0 G50.0 ; is commanded at the G121 mode becomes the same path which is commanded by the G01 X141.421 G45.0 ; at the G120 mode and a speed applies a parameter value and acceleration or deceleration speed becomes the same as the G00.

Also, a movement of rotary axis by G00 is a short cut (movement within  $\pm 180^\circ$ ) to reduce a machining time.

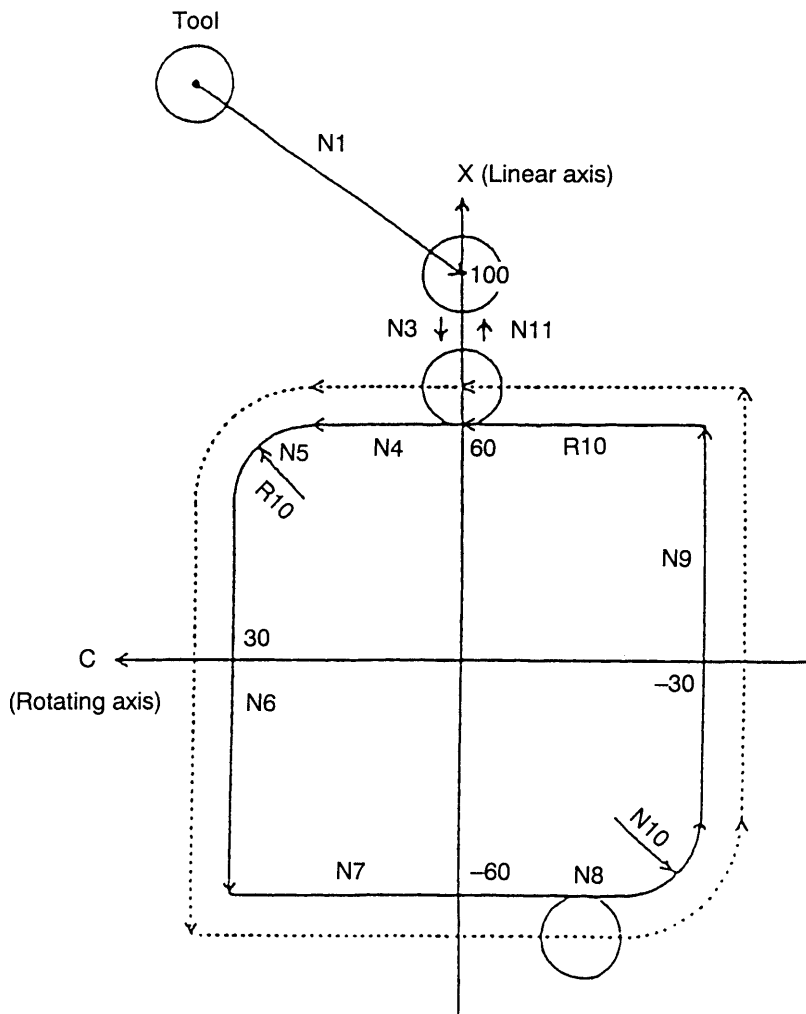
At the time of approach to a workpiece or retract from the workpiece, if the G00 is commanded during the polar coordinate interpolation mode, may be interfered to the workpiece.

Avoid a command of G00 at the polar coordinate interpolation mode if possible.

8. At the polar coordinate interpolation, a shape which is programmed by the rectangular coordinate changes a movement of a linear and rotary axes.
- Therefore, a speed of rotary axis at the movement near the center of workpiece (reference point of coordinate) becomes large and it may beyond the limit speed of the machine.
- In this case, clamp a speed of rotary axis at the setting speed of the parameter (No.1443 : clamping speed of cutting feedrate per each axis) and prevent a speed too fast.

### 6-4-3 Program Example (X-axis : Linear axis/C-axis : Rotating axis)

|                              |                                       |
|------------------------------|---------------------------------------|
| N1 G00 X100.0 C0 ;           | Positioning to the start point        |
| N2 G121 ;                    | Polar coordinate interpolation starts |
| N3 G42 G01 X60.0 F100 ;      | (Tool radius compensation right side) |
| N4 C20.0 F60 ;               |                                       |
| N5 G03 X40.0 C30.0 R10.0 ;   |                                       |
| N6 G01 X-60.0 ;              |                                       |
| N7 C-20.0 ;                  |                                       |
| N8 G03 X-40.0 C-30.0 R10.0 ; |                                       |
| N9 G01 X60.0 ;               |                                       |
| N10 C0 ;                     |                                       |
| N11 G40 X100.0 F100 ;        |                                       |
| N12 G121 ;                   | Polar coordinate interpolation cancel |



## 6-5 G40, G41, G42, G140, G143, G145 Tool Radius Compensation

### Function

Generally, an imaginary tool nose point at 0 or 9 can not be applied a tool radius compensation, however, at the time of G143 mode, a tool radius compensation can be effective by G145 at an imaginary tool nose point 9.

However, a plane designation by G17, G18 or G19 must be set previously.

G140 : Cancel mode of automatic tool nose R compensation/tool radius compensation

G143 : Automatic tool nose R compensation effective mode (At the time of power on and reset, a control becomes this mode.)

G145 : Tool radius compensation effective mode

For an information of tool radius compensation, input a compensating amount of tool radius (radius value of tool) to R and an imaginary to 1 point to T of an applied tool compensating No.

In case of an imaginary tool point 9, approach by G140 mode and a tool radius compensation becomes effective by G145.

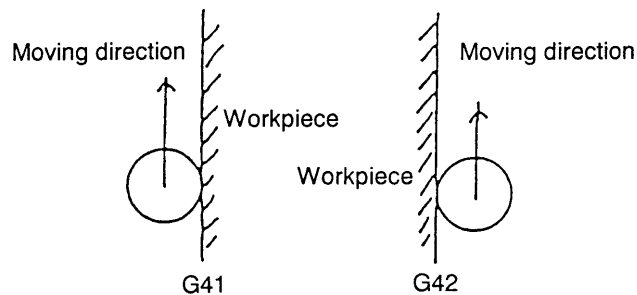
In case of a tool radius compensation effective, a tool radius compensating program is available by G40, G41 or G42.

### 6-5-1 Direction of Tool Radius Compensation

G40 : Tool radius compensation cancel (A tool moves on a path of program.)

G41 : Tool radius compensation, left side (A tool offsets of workpiece toward a moving direction of tool.)

G42 : Tool radius compensation, right side (A tool offsets right side of workpiece toward a moving direction of tool.)



## 6-5-2 Movement of Tool Radius Compensation

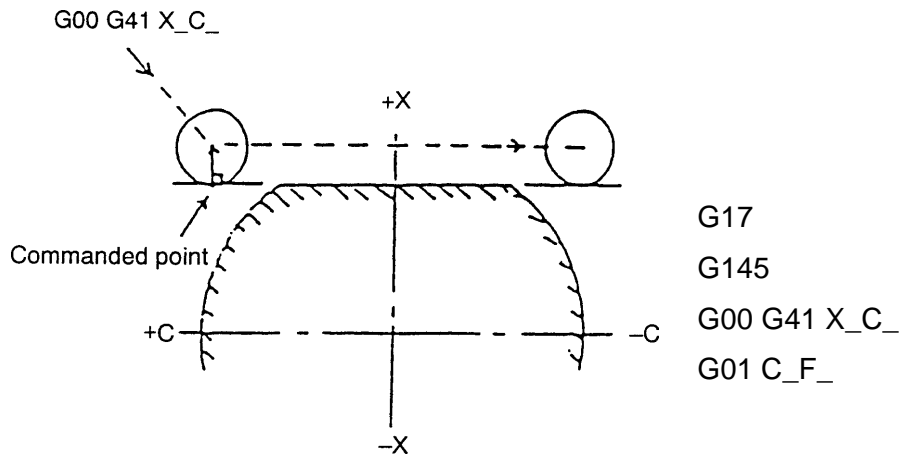
In case of execution of tool radius compensation, a program starts a status of compensation cancel (G40) and command a tool radius compensation mode (G41, G42) then completes after command a compensation cancel status again.

Divide it three conditions and each block calls as follows;

1. Start up
2. Tool radius compensation mode
3. Tool radius compensation cancel

### 1. Start up

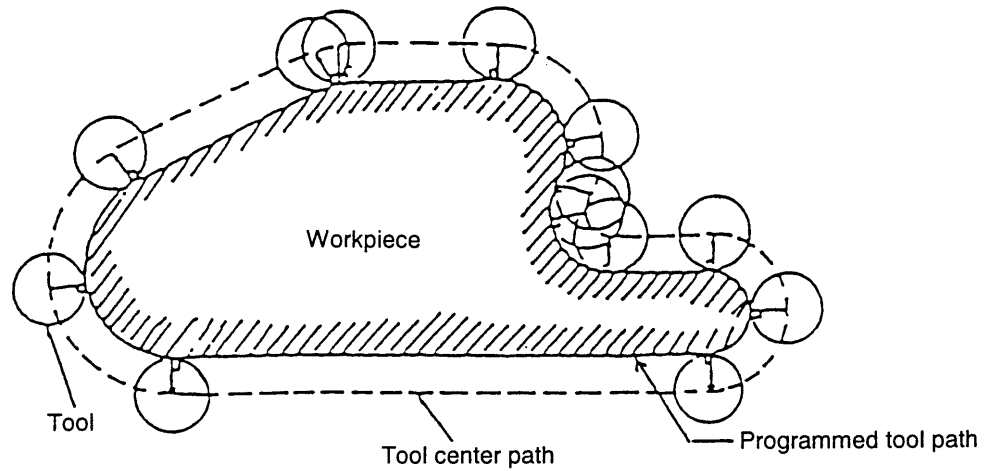
1. A block changed over from the status of tool radius compensation cancel (G40) to the status of tool radius compensation mode (G41 or G42) is called as a block of start up.
2. A center of tool moves to perpendicular position of start point of next move command at a start up block.



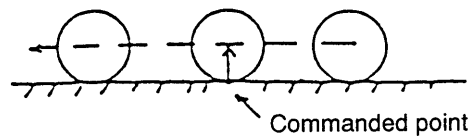
3. Circular interpolation is not acceptable for a start up block. It should be executed either G00 or G01 mode.
4. During tool radius compensation read three blocks in advance to find a stop position of movement. Therefore, block without move command such as M, S, T code or dwell etc. must not continue three blocks or more.

## 2. Tool radius compensation mode

During tool radius compensation mode, the tool moves so that the center of tool is located at the position perpendicular to the advance direction of the tool.

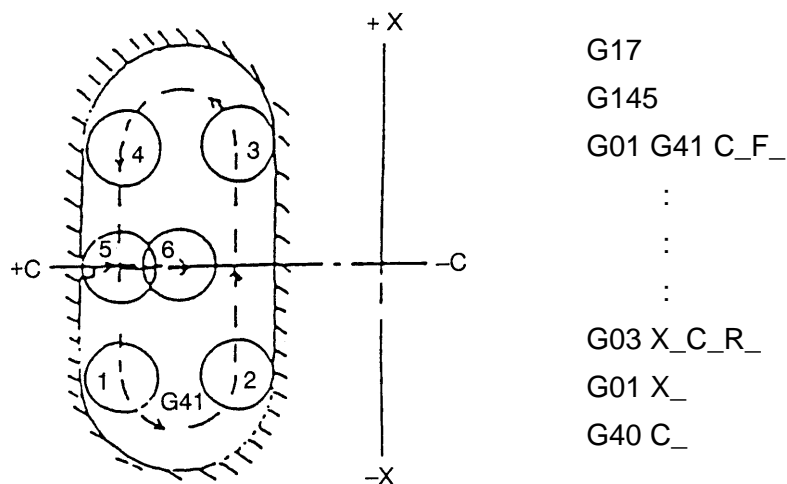


When tangent angle is  $180^\circ$ , the center of tool is located at the position perpendicular to the command point.



## 3. Tool radius compensation cancel

1. A block changed over from the status of tool radius compensation mode to the status of tool radius compensation cancel (G40) is named a cancel block.
2. When the compensation is completed, the tool moves so that the center of tool is located at the position perpendicular to the end point of the block immediately before the cancelled block.



3. In the cancelled block, the center of tool coincides with the command point.

- (Note)** 1. A plane designation should not change during tool radius compensation mode.
2. In case of changing a direction of tool radius compensation during tool radius compensation, cancel a tool radius compensation once then execute a start up.
3. Inside compensation of smaller arc than tool radius can not machining because it generates over cut.
4. Execution of rotating radius command of arc is as follows;

(a) Command  $R\_$  when rotating angle is  $0 \sim 180^\circ$ .

EX  $G02 X\_C\_R\_$

(b) Put minus sign on a value of  $R$  if rotating angle is beyond  $180^\circ$  and less than  $360^\circ$ .

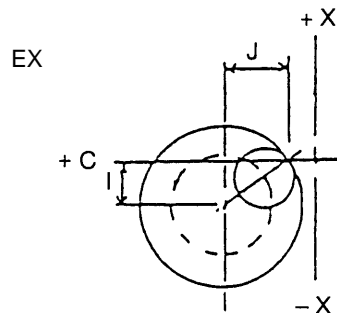
EX  $G02 X\_C\_R\_$

(c) Command by  $I, J$  or  $K$  instead of  $R$  if true circle cutting.

$I$  : X component of a center position of rotation a view from start point of cutting.

$K$  : C component of a center position of rotation a view from start point of cutting.

Put + or - sign either.



$G01 G41 X\_C\_F\_$

$G03 I\_J\_F\_$

$G01 G40 X\_C\_$

5. During tool radius compensation mode, if command a block without movement three blocks or more, it generates insufficient or over cutting.

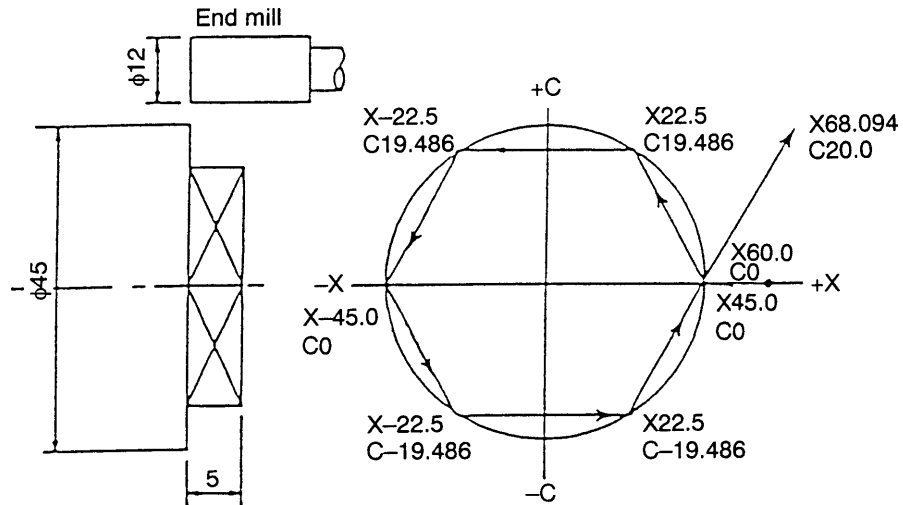
A block without move command should not continue three blocks or more.

6. Do not command the followings during tool radius compensation mode.

- G31
- G37
- G53
- Fixed cycle by G code of 09 group

## 6-6 Program Example (Polar Coordinate Interpolation, Tool Radius Compensation Function)

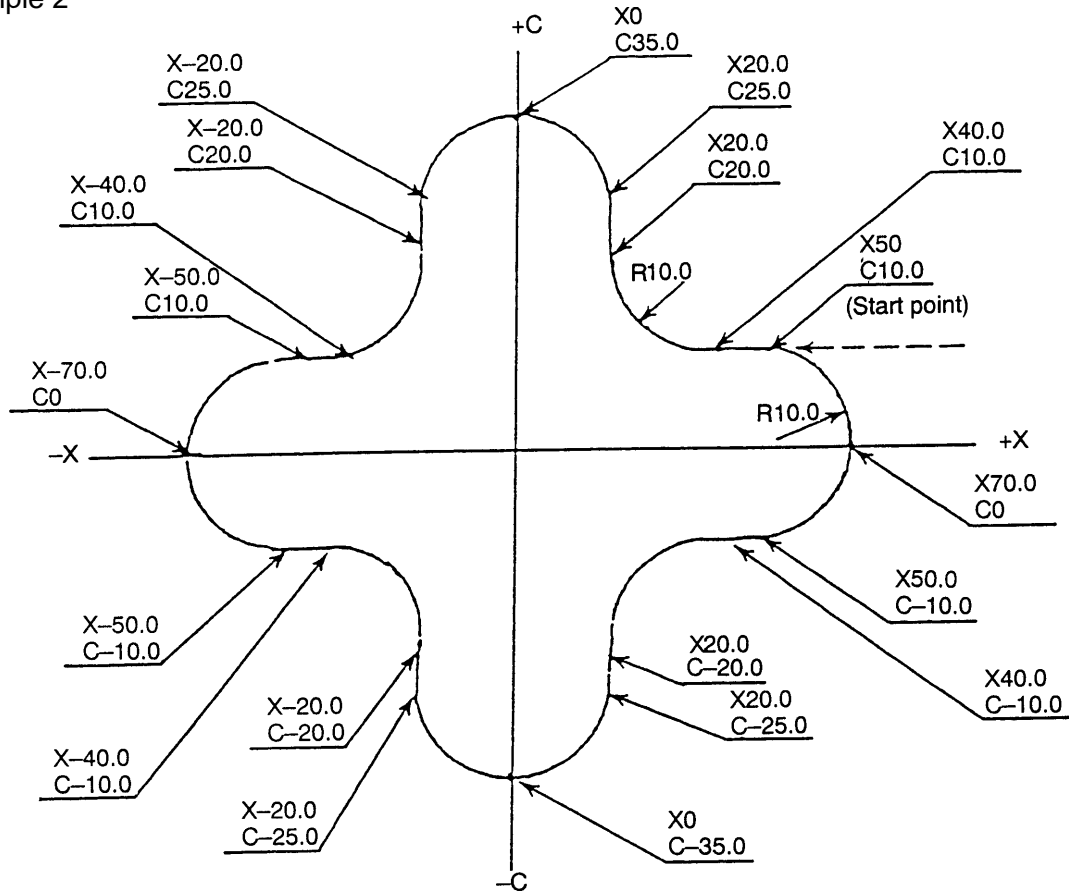
Example 1



```

N400
G28 U0
G28 W0
M43
G28 H0 ..... C-axis zero return
T0400
G17 G145 ..... X-Y plane designation, Tool radius compensation is effective.
G97 S800 M08
G00 X100.0 Z200.0 M13 ..... Rotating tool forward start
G01 G98 Z10.0 F2000
G01 Z-5.0 F1000
G121 ..... Polar coordinate interpolation ON
G01 X60.0 C0 F500
G42 X45.0 F80 ..... Tool radius compensation ON
X22.5 C19.486
X-22.5
X-45.0 C0
X-22.5 C-19.486
X22.5
X68.094 C20.0
G40 X100.0 F2000 ..... Tool radius compensation OFF
G120 ..... Polar coordinate interpolation OFF
G99 M05
M45
G00 Z200.0
M01
    
```

Example 2



|                                                        |                                                                  |
|--------------------------------------------------------|------------------------------------------------------------------|
| N1000                                                  | G03 X-50.0 C-10.0 R10.0                                          |
| M43 ..... C-axis connection                            | G01 X-40.0                                                       |
| G28 H0 ..... C-axis zero return                        | G02 X-20.0 C-20.0 R10.0                                          |
| T1000                                                  | G01 C-25.0                                                       |
| G17 G145                                               | G03 X20.0 C-25.0 R10.0                                           |
| G97 S1000 M08                                          | G01 C-20.0                                                       |
| G00 X100.0 Z20.0 M13 ..... Rotating tool forward start | G02 X40.0 C10.0 R10.0                                            |
| G01 G98 Z1.0 F1000                                     | G01 X40.0                                                        |
| G121                                                   | G03 X50.0 C10.0 R10.0                                            |
| G01 C10.0                                              | G01 X40.0                                                        |
| G42 X80.0 F300 ..... Tool radius compensation ON       | G00 Z20.0                                                        |
| X40.0 F100                                             | G40 X100.0 C50.0 F2000 M09 ..... Tool radius compensation cancel |
| G02 X20.0 C20.0 R10.0                                  | G120                                                             |
| G01 C25.0                                              | X200.0 M05                                                       |
| G03 X-20.0 C25.0 R10.0                                 | M45                                                              |
| G01 C20.0                                              | G99 Z200.0 M40                                                   |
| G02 X-40.0 C10.0 R10.0                                 | M01                                                              |
| G01 X-50.0                                             |                                                                  |

## 6-7 G824, G843 Direct Tapping

A direct tapping is performed with a spindle speed of rotating tool and feed rate of tapping axis synchronize perfectly, therefore, a floating tap holder is not required and a high accuracy tapping is available at high speed.

### (1) Command form

$$\begin{bmatrix} \text{G842} \\ \text{G843} \end{bmatrix} \begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix} \begin{bmatrix} \text{G98} \\ \text{G99} \end{bmatrix} \text{X\_C\_Z\_R\_P\_L\_S\_F(E)\_};$$

G842 : Forward direct tapping  
 G843 : Reverse direct tapping  
 G198/G199 : Return point  
 X\_/C\_ : Hole position (Note 2)  
 Z\_ : Z point position (Note 2)  
 R\_ : R point position  
 P\_ : Dwell time  
 L\_ : No. of repetition  
 F\_ : G98 mode Feed rate of tapping axis  
       G99 mode Pitch of tap  
 E\_ : No. of thread per inch (Effective at G99 mode only)  
 S\_ : Spindle speed of rotating tool

**(Note 1)** If number of repetition (L) is omitted, it deems as L=1.

If L=0 is commanded, it moves at hole position but tapping is not performed.

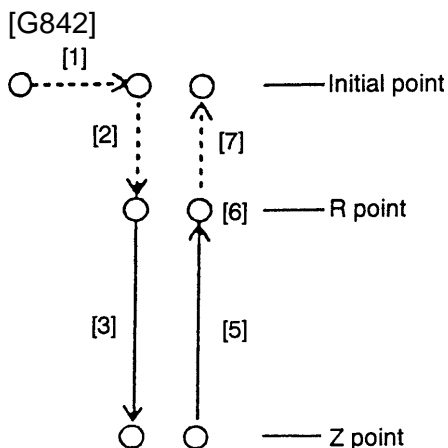
**(Note 2)** Command a position of Z point by an axis address of tapping axis.

Command a hole position by an axis address other than tapping axis.

**(Note 3)** R and Z points and P are modal during a fixed cycle.

### (2) Machining cycle

A machining cycle of direct tapping consist motions from [1] to [7].



••••• Rapid traverse, ——— Cutting feed

- [1] Positioning at hole position
- [2] Rapid traverse to the R point
- [3] Tapping to the Z point with forward rotation
- [4] Dwell by the parameter setting
- [5] Return to the R point with reverse rotation and stop the tool rotation
- [6] Dwell by the parameter setting
- [7] Rapid traverse to the initial point

**(Note)** Tool rotation is reversed at the G843.

(3) Designation of feed rate and pitch (F command)

At the direct tapping, the meaning of F command differs at the feed per minute mode (G98) and feed per revolution mode (G99).

Also, the E command is available instead of the F command at the G99 mode.

- G98 mode : The F shows a feed rate of tapping axis. (mm/min, inch/min)
- G99 mode : The F shows a pitch of tap. (mm, inch)  
The E shows a number of thread per inch. (thread/inch)

**(Note 1)** A motion becomes a feed per minute even at the G99 mode.

**(Note 2)** The number of effective digits are same for the F and E.

**(Note 3)** If the F and E are commanded in the same block, the F becomes effective.

**(Note 4)** At the G98 mode, a pitch is determined by the F and S (Number of rotation of tool).

$$\text{Pitch (min, inch)} = \frac{F \text{ (mm/min, inch/min)}}{S \text{ (min}^{-1}\text{)}}$$

(4) Magnification of returning speed

Returning speed of the direct tapping (Z point - R point) can be changed to the cutting feed rate (R point - Z point).

Set a magnification of returning speed to the cutting feed rate at the parameter. (unit 0.1).

$$\text{Returning speed} = \text{Cutting feed rate} \times \frac{\text{Parameter No.5211}}{100}$$

However, if zero is set at this parameter, a parameter value is deemed as 10 and the magnification becomes 1.

Normally, setting value is zero.

(5) Cancel of direct tapping

Command G80 or G code of 01 group (G00, G01, G02, G03 etc.).

However, do not command the other fixed cycle (G70, G71 etc.) in the same block of cancellation command.

(6) Precautions

(a) Command the direct tapping at the condition of cancellation of constant surface speed control (G97).

(b) Effective or ineffective can be selected for the dwell by the parameter setting.

Normally, it is set as effective.

(c) A feed rate override and spindle speed override are fixed at 100% while tapping.

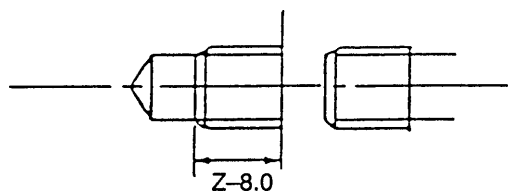
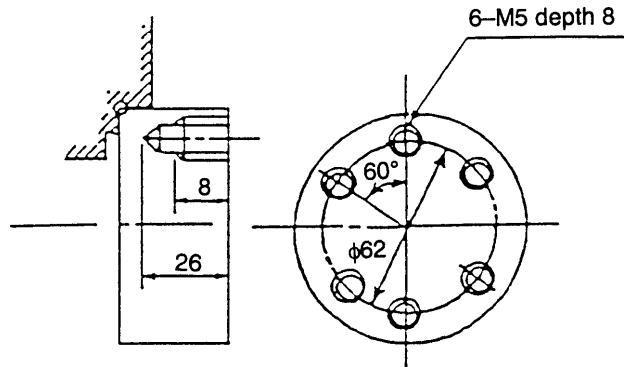
However, effective or ineffective can be selected for the dry run by the parameter

setting.

- (d) When it performs at the single block, a tool stops at the initial point or R point.
- (e) If the "Halt" button is pressed during the tapping, the halt lamp turns on immediately but the motion continues until the R point then stops.
- (f) To cancel a direct tapping, command G80 or G codes of G01 group (G00, G01, G02 . . .). However, do not command the other fixed cycle (G70, G71 etc.) in the same block of the cancellation command.
- (g) When executing the direct tapping, command the spindle speed of the rotating tool at the immediately before or the same block of the G842 or G843 command.

(7) Program example

```
N800 (M5*P0.8)
T0800 M40
G17
M43
G28 H0
G50 C0
G97 S300 M08
G00 X62.0 Z20.0 M13
G99 G01 Z5.0 F5000
G842 G198 Z-8.0 H60.0 L6 F0.8
G80
G00 Z5.0 M05
M45
G00 X200.0 Z200.0
M01
```



## 6-8 G271 Cylindrical Interpolation

When commanding a traveling amount of linear axis and angle of rotary axis by a program command, a traveling amount of rotary axis commanded by an angle converts to a distance on the circumference internally. A distance on the circumference deems a traveling amount of linear axis on the circumference, therefore, the linear or circular interpolation with the other linear axis is available.

After interpolation, it convert reversly to the angle of rotary axis.

### (1) Command form

|                                    |   |                                  |
|------------------------------------|---|----------------------------------|
| G271 C <u>Radius of cylinder</u> ; | } | Cylindrical interpolation ON     |
| ·                                  |   | Cylindrical interpolation mode   |
| G271 G0 ;                          |   | Cylindrical interpolation cancel |

(C is a rotary axis.)

A rotating angle of the rotary axis is calculated reversly from the traveling amount on the circumference.

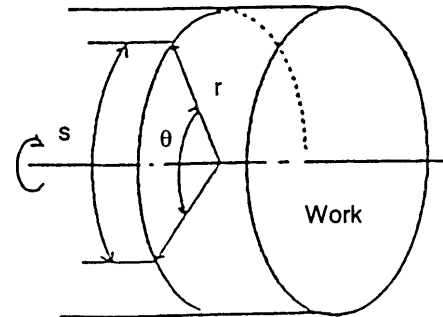
For example, if a traveling amount on the circumference at the cylinder with a radius = 50.0 is wanted to move by 100.0, find a rotating angle of the rotary axis by the following formula.

r : Radius of cylinder

$\theta$  : Rotating angle

s : Traveling amount on the circumference of cylinder

$$\begin{aligned} \text{Rotating angle} &= \frac{360 \times s \text{ (Traveling amount on the circumference)}}{2\pi \times r \text{ (Radius of cylinder)}} \\ &= \frac{360 \times 100.0}{2\pi \times 50.0} = 114.591 \end{aligned}$$



### (2) Feed rate

A feedrate during the cylindrical interpolation mode becomes a traveling speed of a tool on the outer diameter of the cylinder.

### (3) Plane selection

During the cylinder interpolation mode, the plane in which performs the cylindrical interpolation is determined as a horizontal axis is a linear axis (Z axis) and a vertical axis is a rotary axis (C axis). Therefore, since the plane for the cylindrical interpolation is determined by these parameters, the plane selection (G17 ~ G19) can not commanded during the cylindrical interpolation.

(4) Program example (X axis is a diametal designation)

(Select the C - Z plane by the parameter No. 3426 and 3427)

N400;

G28 U0;

G28 W0 M43;

G28 H0;

T0400;

G19 G98 M44;

G40 G80;

G50 C0;

G97 S600;

M145;

G00 X120.0 Z-120.0 C0 M13;

G271 C50.0;

N1 G42 G01 Z-40.0 F500;

G01 X100.0 F50;

N2 C90.0 F100;

N3 Z-100.0 C180.0;

N4 C260.0;

N5 G03 Z-80.0 C282.918 R20.0;

N6 G01 Z-60.0;

N7 G02 W20.0 H22.918 R20.0;

N8 G01 C360.0;

G00 X200.0;

N9 G40 G01 Z-120.0 F500;

G271 C0;

G00 Z50.0 C0 M05;

G143;

M45;

G18 G00 X200.0 Z200.0 M40;

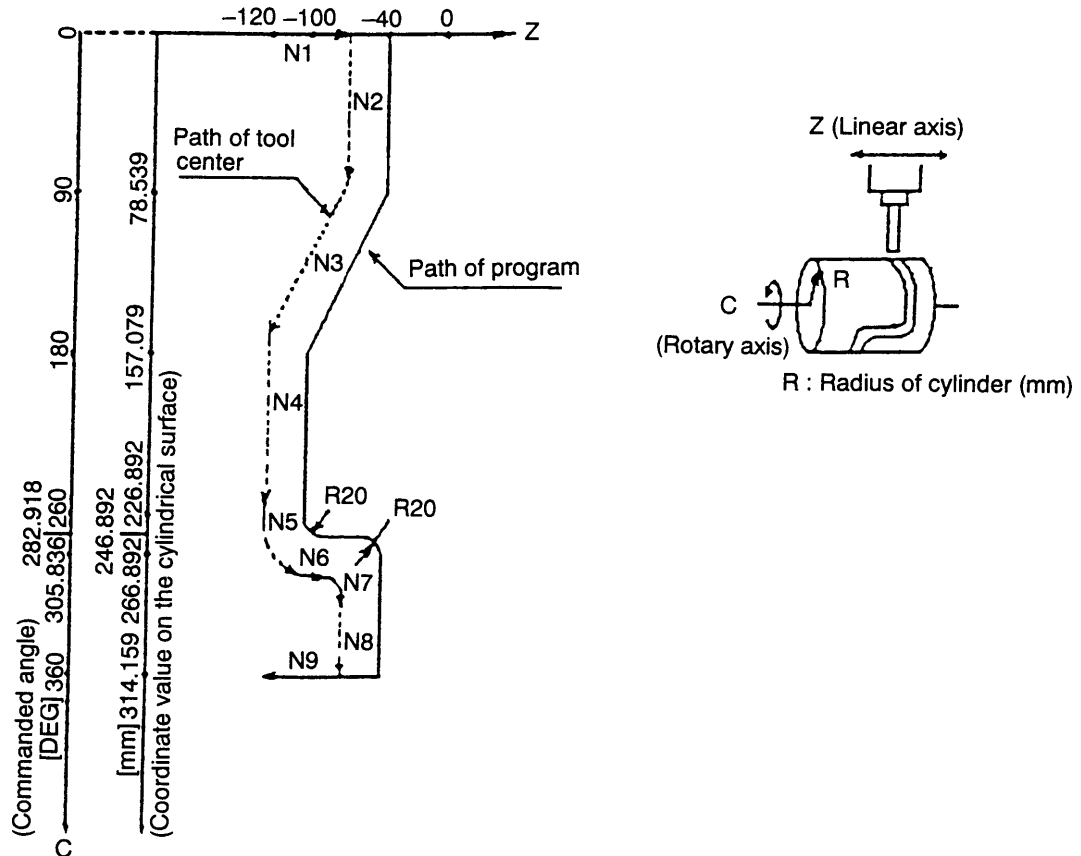
M01;

Cylindrical interpolation mode ON  
(Radius of cylinder = 50.0)

Under cylindrical interpolation mode

Cylindrical interpolation mode OFF

Unfolded drawing of cylindrical surface with radius of cylinder is 50.0



(5) Precautions

- If a tool radius compensation is commanded, start up and cancel should be done during the cylindrical interpolation mode.
- The G271 command (G271 Cxx;) should be commanded in the block individually. Also, if an axis command is missing after the G271 (G271; for example), It becomes an alarm.
- If an axis other than the axis which is set by the parameter No.7817 is commanded by the G271 command, it becomes an alarm.
- The following functions are not available.
  - Block restart
  - Return to the interrupted point of machining
  - Manual intervention by manual absolute ON
- If the following command is issued during the cylindrical interpolation mode, it becomes an alarm.
 

|                                   |                           |
|-----------------------------------|---------------------------|
| G17, G18, G19                     | Plane designation         |
| G28, G30, G53                     | Machine coordinate system |
| Thhtt ; (T command), G54, G50     | Work coordinate system    |
| G70 ~ G76, C81 ~ G89, G831 ~ C861 |                           |
| G90 ~ G94                         | Various fixed cycle       |
| G31, G121, G232                   | Others                    |

G00 (Restricted only when the rotary axis which performs the cylindrical interpolation has been commanded.)

- (f) At the cylindrical interpolation mode, convert an angle of rotary axis to the distance on the circumference then convert reversely after interpolation.

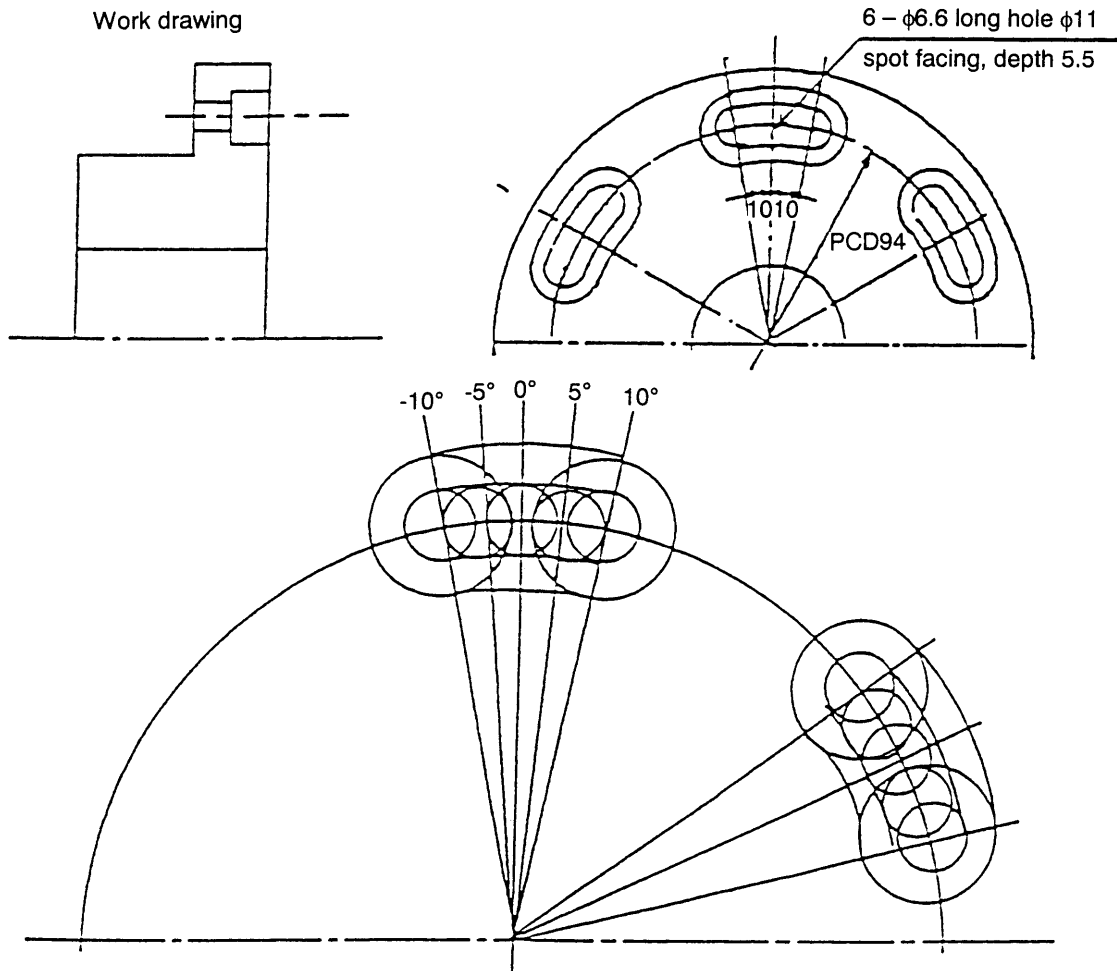
At this time a conversion error generates slightly.

- (g) By the above conversion error, if the circular interpolation for small radius is executed during the cylindrical interpolation mode, the circular interpolation alarm may occur, therefore, an attention is required when it applies. Also, a tool radius compensation alarm may occur at the tool radius compensation by the above reason.
- (h) If the cylindrical interpolation mode ON, G271 Cxx; ( $C \neq 0$ ), is commanded again during the cylindrical interpolation mode, it becomes an alarm.
- (i) A remaining traveling amount is shown a value which is traveling the outer diameter of the cylinder.

## 7. REFERENCE (SPECIFICATIONS OF C-AXIS CONTROL)

### 7-1 How to Calculate C-axis Feed Rate for Long Hole Machining

Work drawing



1) C-axis feed rate (mm/min); No decimal point allowed

$$\text{Arc length per } 1^\circ = \frac{D \times \pi}{360} = \frac{94 \times 3.14}{360} = 0.82 \text{ mm/deg}$$

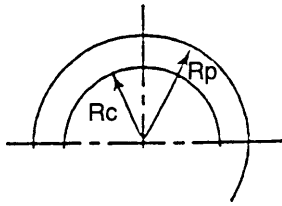
D : Cutting diameter

Feed rate per minute :

$$\begin{aligned} & \text{Feed rate} \div \text{Arc length per } 1^\circ \\ & = 50 \text{ mm/min} \div 0.82 = 60.975 \text{ deg/min} \end{aligned}$$

Where ; feed rate in normal cutting is taken as 50mm/min. Feed rate of C axis becomes a command of 61mm/min.

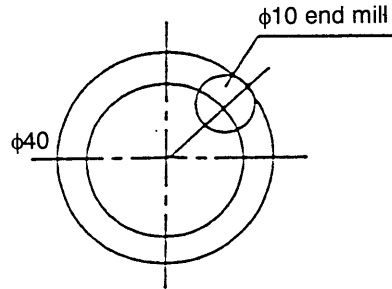
2) Feed rate inside/outside the arc



$R_p$  radius of program path

$R_c$  radius of center path of the cutter  $F = F \times \frac{R_c}{R_p}$

Example

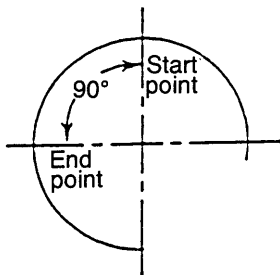


If the program path is F100

$$F = 100 \times \frac{15}{20} = 75\text{mm/min}$$

3) Feed rate of the rotating axis

Example Specify in deg/min.



Move at 300 deg/min for 90°.

$$\text{Time } \frac{300}{90} = 0.3 \text{ min.}$$

4) Feed rate at linear interpolation including linear axis and rotating axis.

Tangent rate in Cartesian coordinate of the rotating axis (deg) and the linear axis (mm) is F.

Example G00 W-20.0 C40.0 F300

Supposing C-axis 40 deg is 40 mm, distribution time will be

$$\frac{\sqrt{20^2 + 40^2}}{300} = 0.15 \text{ min}$$

C-axis rate is

$$\frac{40 \text{ deg}}{0.15 \text{ min}} = 267 \text{ deg/min}$$

## 7-2 How to Calculate the Number of Rotation and Feed Rate of the Rotating Tool

1) The number of rotation of the rotating tool

$$N = \frac{1000 V}{\pi D}$$

N = Rotation per minute ( $\text{min}^{-1}$ )

D = Diameter of the cutter (mm)

V = Cutting rate

Example) Rotation per minute when machining with D10.0 drill, V20

$$N = \frac{1000 \times 20}{3.14 \times 10} = 636.9 \quad S = 637 \text{ min}^{-1}$$

2) Feed rate per minute

For end mill

F = Feed rate (mm/min)

or drill

$$F = N \times f$$

N = Rotation per minute ( $\text{min}^{-1}$ )

f = 1 rotation feed (mm/rev)

For tap

$$F = N \times P$$

P = Pitch (mm)

Example)

Feed rate with D10.0 drill, V20 and 1 rotation feed of 0.2.

$$F = 637 \times 0.2 = 127.4 \quad F = 127$$

Example)  
320.

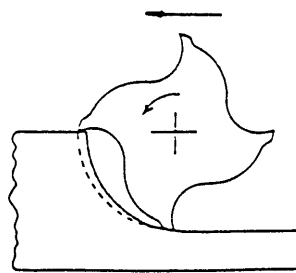
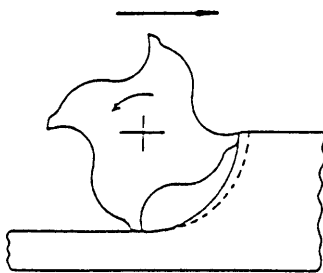
Feed rate when machining with M8 tap and the spindle rpm of

$$F = 320 \times 1.25 = 400 \quad F = 400$$

3) Up-cutting and Down-cutting

Up-cutting

Down-cutting



|          | Up-cutting                                                                                                                                                                                                                                                                                                                                                                                                                                | Down-cutting                                                                                                                                                                                                                    |
|----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Merits   | <ul style="list-style-type: none"> <li>• Undercut is easily caused.</li> <li>• Finish surface roughness is good in wet cutting.</li> <li>• A finish surface is glossy and looks fine because it is rubbed by a tool nose.</li> <li>• More advantageous than down-cutting in scale or sand caught machining.</li> </ul>                                                                                                                    | <ul style="list-style-type: none"> <li>• A tool nose flank is worn out less and a tool life is longer.</li> <li>• A cutting resistance is low.</li> <li>• Finish surface roughness is superior in dry cutting.</li> </ul>       |
| Demerits | <ul style="list-style-type: none"> <li>• Since a tool nose tends to slip, a flank is worn out more and a tool life is shorter.</li> <li>• It is necessary to firmly mount a workpiece.</li> <li>• A cutting resistance is high.</li> <li>• Finish surface roughness is inferior in dry cutting.</li> <li>• Burr tends to be caused at the end of a workpiece.</li> <li>• A hardened layer is caused for a hardenable material.</li> </ul> | <ul style="list-style-type: none"> <li>• Coolant has a less effect on finish surface roughness and may worsens it to the contrary.</li> <li>• A tool nose is likely to be damaged in scale or sand caught machining.</li> </ul> |



CNC LATHE  
INSTRUCTION MANUAL  
PROGRAMMING  
SEIKI-SEICOS Σ10L/21L  
Version 1.01

11-1999 First Edition  
02-2000 Revision