

# ST200/250 CNC LATHE PROGRAMMING

65 Edition 1.01

PM-1782-1-0300-E-1-01



***Hitachi Seiki Deutschland  
Werkzeugmaschinen GmbH***



# 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 - 4
1-3 NC Address and Range of Command Value .....	1 - 5
<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 - 8
2-2-4 M Function (Miscellaneous Function) List .....	2 - 12
2-3 Details of G Function .....	2 - 20
2-3-1 List of G Function .....	2 - 20
2-3-2 G50 Maximum Spindle Speed Setting .....	2 - 23
2-3-3 G00 Positioning .....	2 - 23
2-3-4 G01 Linear Cutting .....	2 - 25
2-3-5 G02, G03 Circular Cutting .....	2 - 27
2-3-6 G04 Dwell .....	2 - 31
2-3-7 G09 Exact Stop .....	2 - 31
2-3-8 G61 Exact Stop .....	2 - 32
2-3-9 G10 Programmable Date Input .....	2 - 32
2-3-10 G20, G21 Inch Input/Metric Input .....	2 - 33
2-3-11 G22, G23 Stored Stroke Limit .....	2 - 34
2-3-12 Stroke Limit Check Before Move .....	2 - 35
2-3-13 G28 Automatic Reference Point Return .....	2 - 36
2-3-14 G30 2nd Reference Point Return .....	2 - 36
2-3-15 G31 Skip Function .....	2 - 39
2-3-16 G54 Work Coordinate System Setting (Work Length) .....	2 - 40
2-3-17 Canned Cycle .....	2 - 41
2-3-18 Multiple Repetitive Cycle .....	2 - 50
2-3-19 G32, G92, G76 Thread Cutting .....	2 - 68
2-3-20 G32 Continuous Thread Cutting .....	2 - 88

2-3-21 Multi-thread Cutting .....	2 - 89
2-3-22 G34 Variable Lead Thread Cutting .....	2 - 90
2-3-23 G150, G151, G152 Groove Width Compensation .....	2 - 91

### **3. AUTOMATIC CALCULATING FUNCTION OF**

#### **TOOL NOSE RADIUS COMPENSATION ..... 3 - 1**

3-1 Outline .....	3 - 1
3-2 Preparation to Execute the Automatic Calculating Function of 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

### **4. PROGRAM EXAMPLE (NC PROGRAM) ..... 4 - 1**

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

### **5. REFERENCE MATERIALS ..... 5 - 1**

5-1 How to Calculate the Tool Nose Radius Compensation Amount 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

## **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.

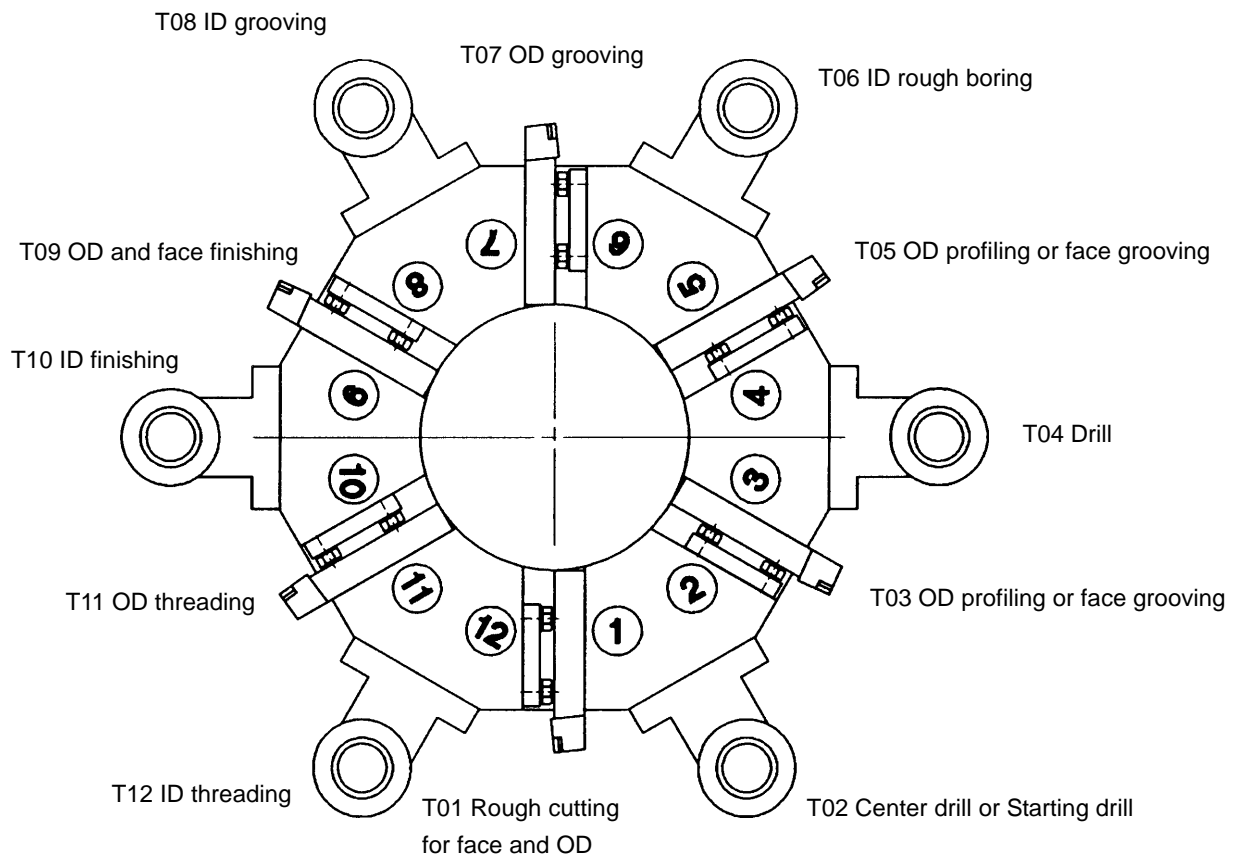
## 1-1 Tool Set

### 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.

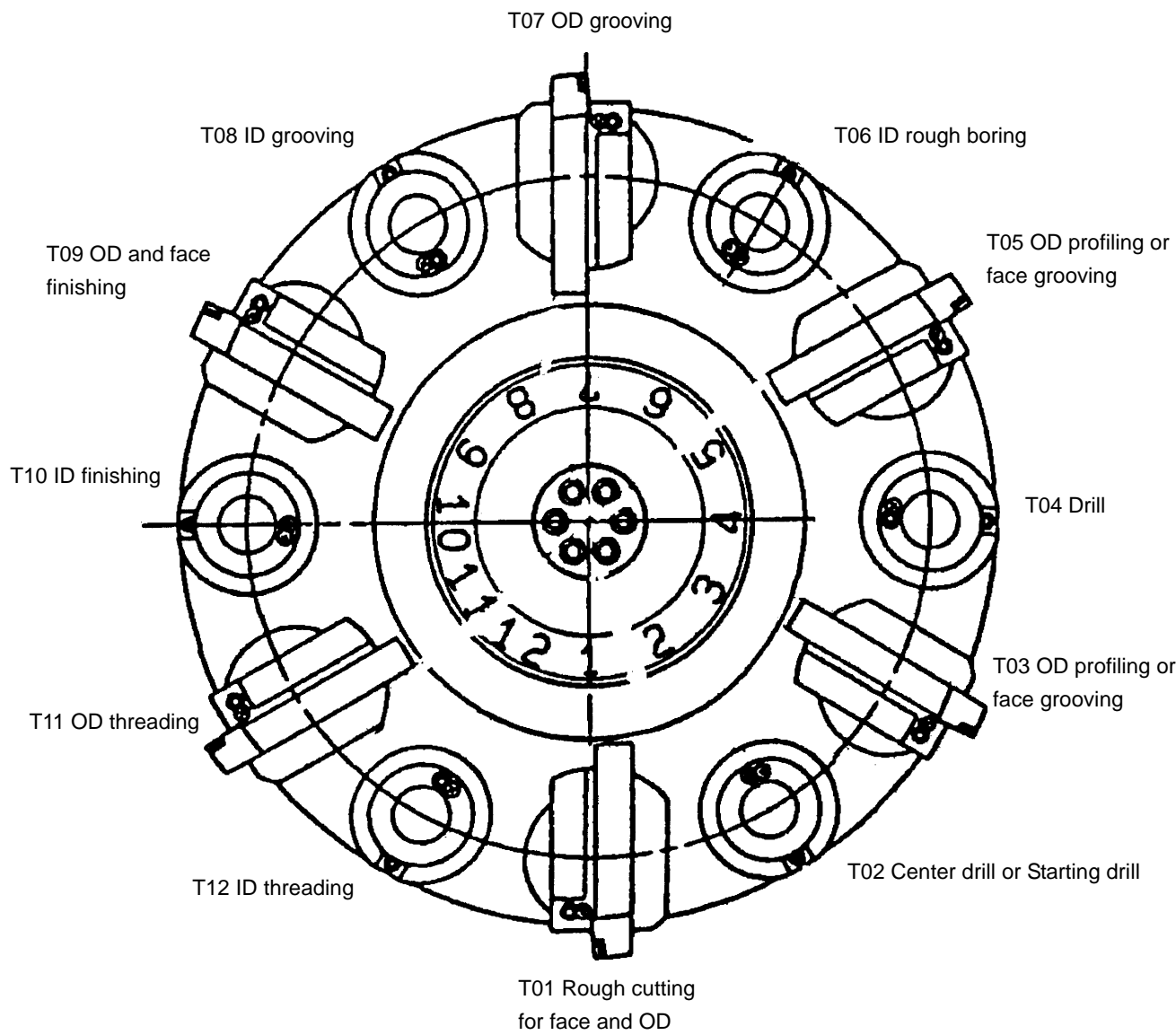
The standard tool set is shown as below.



Specifications of 12-station Variable turret



Standard Tool Set



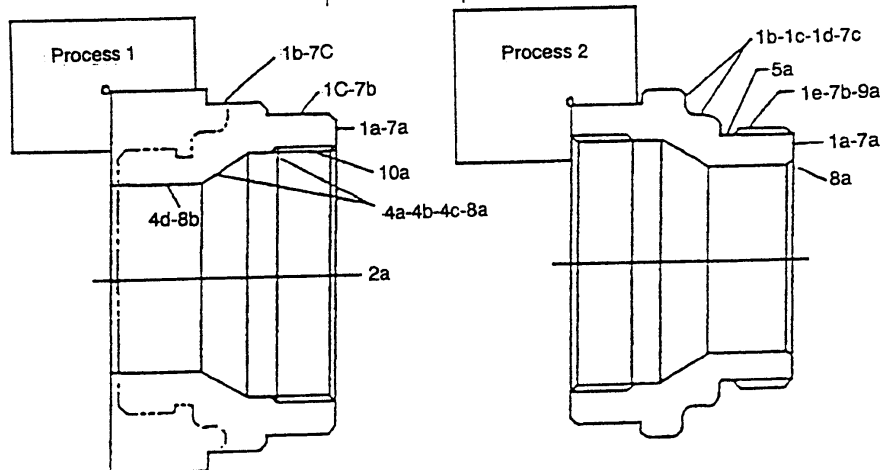
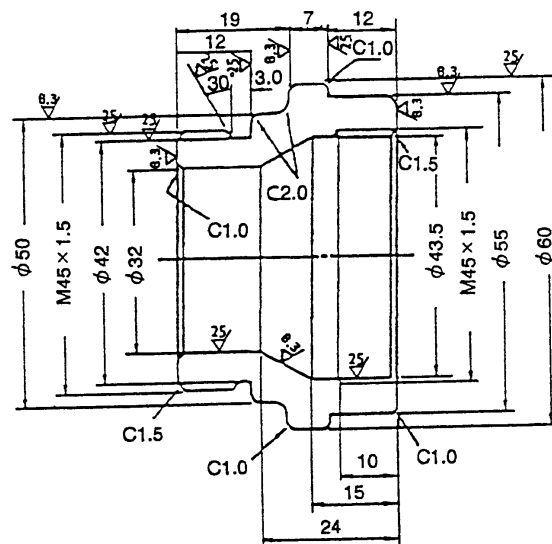
Specifications of 12-station QCT turret

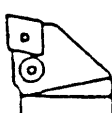
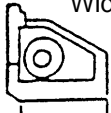

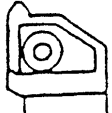
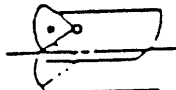



## 1-2 Tool Layout

Example of tool layout for chuck work		 CNC LATHE: TOOL LAYOUT DRAWING
Process : Process 1, 2	NC unit	

Part name SAMPLE

Material  
S48C



T1	T3	T5	T7	T9
R0.8  OD roughing		Width 2mm  OD grooving	R0.8  OD finishing	 OD threading
T2	T4	T6	T8	T10
 φ30	R0.8  φ20 ID roughing		R0.8  φ20 ID finishing	 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, W, I, J, K, Q, R, A, B, C	$\pm 99999.999(\text{mm})$ $\pm 99999.999(\text{deg})$	$\pm 9999.999(\text{inch})$ $\pm 99999.999(\text{deg})$
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	



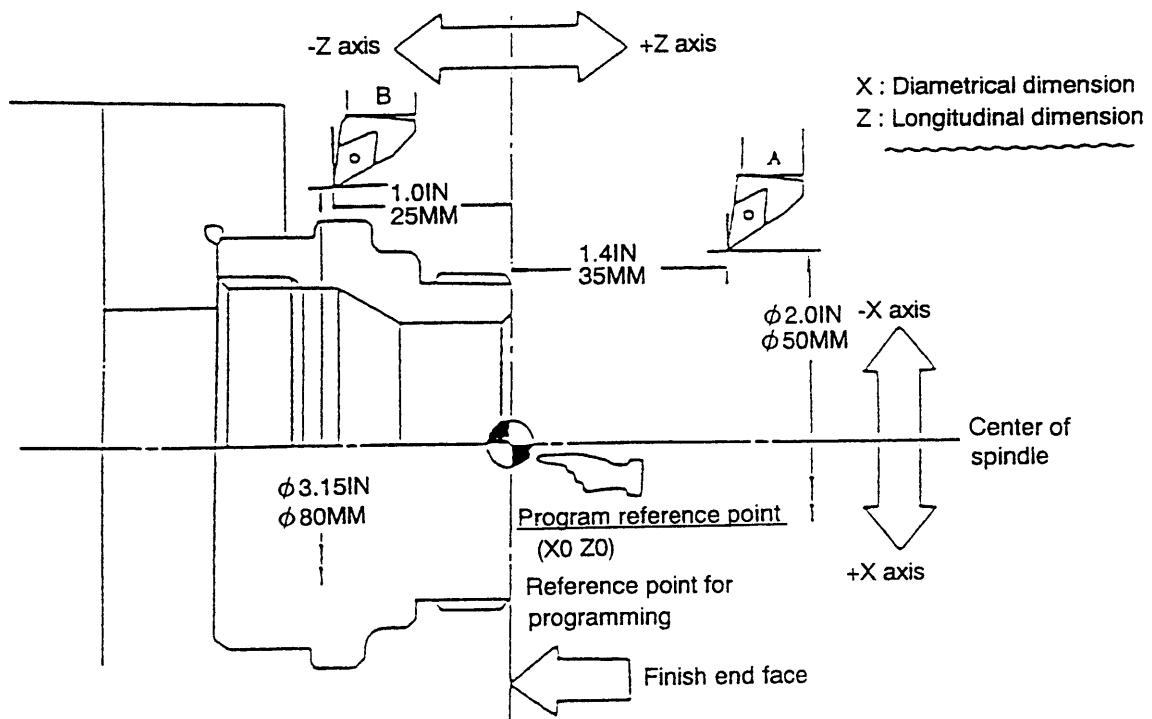
## 2. PROGRAMMING

### 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 at 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 at 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.

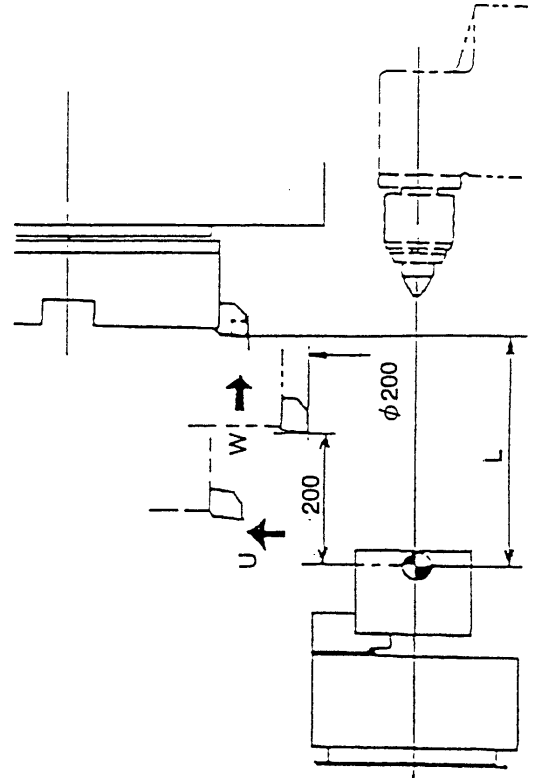
Turn on the power at the starting time in the morning, it can be entered a program operation .

NC Program

Machining drawing

The drawing shows a mechanical part with the following features and dimensions:

- Top View:**
  - Outer diameter:  $\phi 60$
  - Inner diameter:  $\phi 50$
  - Radius:  $R2.0$
  - Surface texture:  $C1$
- Front View:**
  - Outer diameter:  $\phi 44.85$
  - Inner diameter:  $\phi 50$
  - Radius:  $R2.0$
  - Surface texture:  $C1.5$
  - Dimension lines: 7, 12



## 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)

Max. feed rate 5,000mm/min.

A maximum feed rate depends on the spindle speed used.

Assuming the spindle speed to be N;

$$\frac{5000}{N}$$

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

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

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)

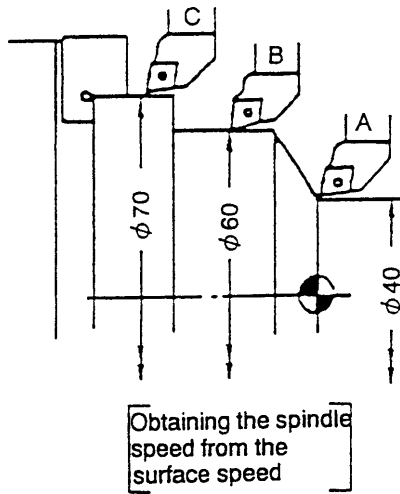
Specify a spindle speed or surface speed (cutting speed) with S 4-digit numeral (S□□□□).

Command	Description
G50S□□□□	Max. spindle speed limit (Example) G50 S1800 : A maximum spindle speed is limited to 1,800 (mim <sup>-1</sup> )
G97S□□□□	Constant surface speed cancel Specify a spindle revolution with S□□□□ . (Example) G97 S1000 : A spindle speed per minute is set to 1,000 (mim <sup>-1</sup> )

## G96S□□□□ Constant surface speed control

When performing constant surface speed control, specify a cutting speed "V" (m/min) with an S 4-digit code (S□□□□ ).

(Example)G96 S150 : 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.*

### 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

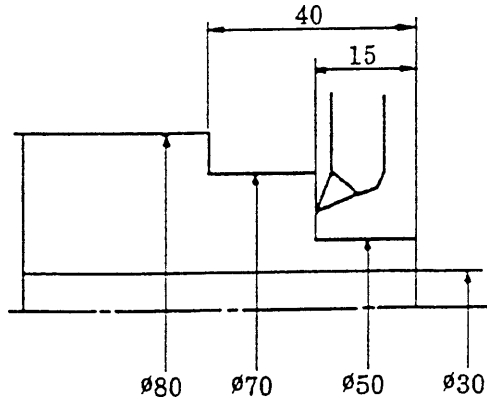
When “T□□△△” command is specified on the same line as the axis travel command, the indexing of turret is made simultaneously with traveling and a coordinate is set after completion of traveling.

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
               cancel

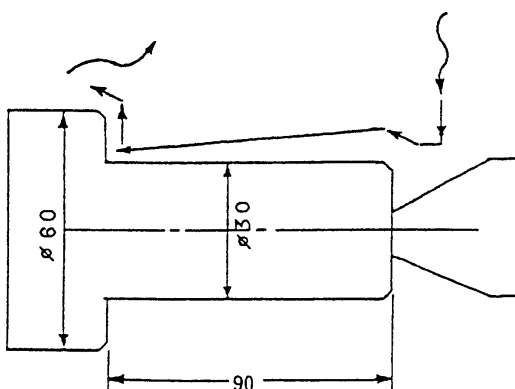
G00 G97 Z10.0
G30 U0 W0
M01
    
```

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

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



```

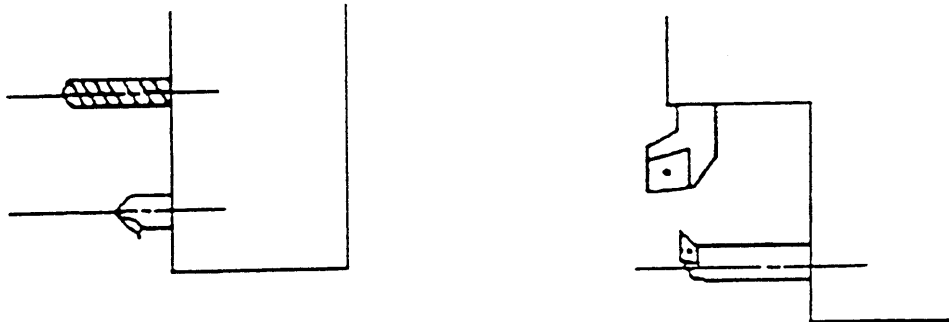
T0500
G97 S2000 M08
G00 Z3.0 M03
    X40.0
G01 X24.0 F1.0
    Z1.0 F0.2
    X30.0 Z-2.0
    Z-90.0 T0525 ← Compound offset
                  Offset No. 25
    X57.00 T0500 ← Compound offset
    X62.0 Z-92.5      cancel
    X68.0

G00 G97 Z10.0
G30 U0 W0
M01
    
```

OFFSET				
	X	Z	R	T
25	-0.3	0	0	0

#### 4. Multi tool compensation

When set up tools 2 or more on the same face on the turret described below, give plural compensation on a face and set up the coordinate for each tool respectively.



Command system of compound compensation, and furthermore, set up tools deem as different one by setting data in nose radius and control point.

(Example) N100	T0100	A tool with turret face No.1 is indexed and setting-up
	└	is performed by the data of offset No.1.
	T0131	A tool with turret face No.1 is indexed and setting-up
	└	is performed by the data of offset No.31.

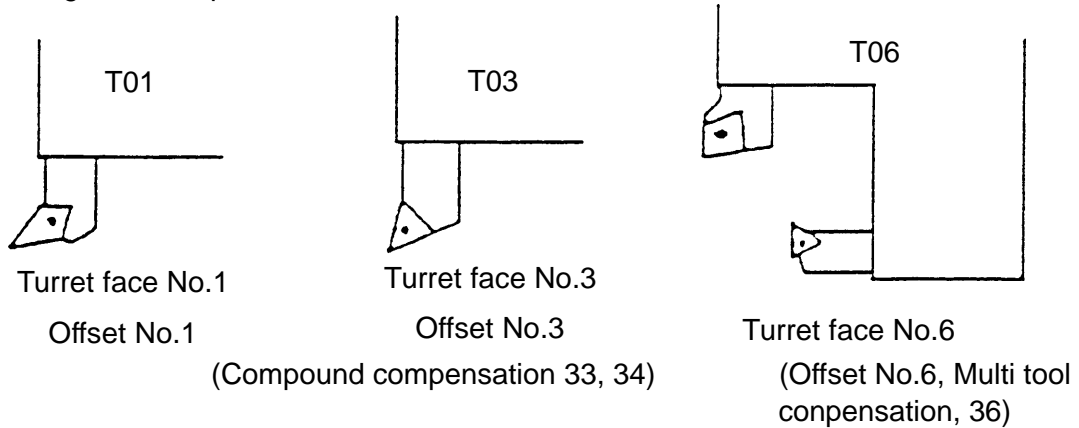
- Note** 1) When a tool, which is not required tool point and tool nose R such as drill etc., is applied to multi tool, set a tool point as 9. (Tool nose R may be set as zero.)
- 2) When set the Q setter, the cursor position of tool offset coincide with the tool No. mounted on the turret face indexed at machining position at this moment.
- Any No. can be selected by moving the cursor by cursor key.

Multi tool compensation and compound compensation is divided by data of tool point and tool nose R as follows:

Tool nose R and tool point of offset No. on effect the compound compensation and multi tool compensation.

- 1 Both tool nose R and tool point are zero → Compound compensation
- 2 Data of tool point from 1 to 9 and setting of tool nose R  
→ Multi tool cutting
- 3 Tool point is zero and set a tool nose R → Alarm (No.182)

## 5. Program example




N100	T0100	The turret face No.1 is indexed and setting-up is performed by the data of offset No.1.
	└	
	M01	
N300	T0300	The turret face No.3 is indexed and setting-up is performed by the data of offset No.3.
	└	
G01	Z- ○ T0333	Compound compensation ON (Offset No.33)
	└	
	X ○ Z ○ T0334	Compound compensation ON (Offset No.34)
	└	
	T0300	Cancel compound compensation (Offset No.3)
	└	
	M01	
N600	T0600	The turret face No.6 is indexed and setting-up is performed by the data of offset No.6.
	└	
	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

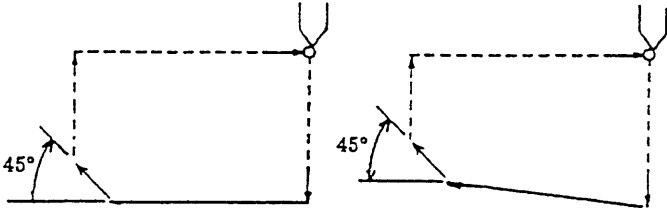
Please refer to the details on the Delivery specifications as to the discrimination between Standard or Option.

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> <div style="display: flex; align-items: center; justify-content: center;"> <div style="text-align: right; padding-right: 10px;"> <p>Executed when a lamp is lit up. (optional stop is effective)</p> <p>Sheet key</p> </div> <div style="border: 1px solid black; padding: 5px; text-align: center;">  </div> <div style="text-align: left; padding-left: 10px;"> <p>Ignored when a lamp is lit off. (optional stop is not effective)</p> </div> </div>
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 spindle motor side, this code starts the spindle in the clockwise direction.
M04	SPINDLE REVERSE START	Viewing from the spindle motor side, this code starts the spindle in the counterclockwise direction.
M05	SPINDLE STOP & ROTARY TOOL 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).
M07	OPTIONAL COOLANT START	
M08	COOLANT START	This code starts discharging coolant.
M09	COOLANT STOP	This code stops discharging coolant.
M12	WORK 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
M13	ROTARY TOOL FORWARD ROTATION	The rotary tool runs in the forward direction at C-axis coupling time.
M14	ROTARY TOOL REVERSE ROTATION	The rotary tool runs in the reverse direction at C-axis coupling time.
M15	ROTARY TOOL STOP	Stops the rotary tool spindle .
M18	SPINDLE POSITIONING OFF	Cancels M19.
M19	SPINDLE POSITIONING	Indexes the spindle by one position.
M23	CHAMFERING ON	<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 ADVANCE	
M26	TAILSTOCK RETRACT	
M28	CENTER STOPOVER RETRACT	Use when the center position detector is set.
M30	END OF PROGRAM, NC RESET & REWIND	End of the program in case of memory operation. Stops the spindle and coolant, and resets the NC unit to return the program to the beginning. Specify this code in an independent block.
M31	NO-WORKPIECE CHUCK& COUNT UP CHECK	<ol style="list-style-type: none"> <li>1. Tool life check</li> <li>2. Work quantity check of the preset work counter</li> <li>3. No-workpiece check when the bar feeder is attached</li> </ol>

M code	Function	Description
M32	TOP CUT CHECK	Block skip ON, however, block skip becomes OFF by the top cut signal ON.
M33	TOP CUT RESET	Reset the top cut signal.
M34	BAR LOAD COMMAND	
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.
M38	CENTER AIR BLOW ON	Air is blown to the center.
M39	CENTER AIR BLOW OFF	Stop the air.
M40	SPINDLE LOW WINDING SELECT & CANCEL C-AXIS COUPLING	30 ~ 1000min <sup>-1</sup>
M41	SPINDLE HIGH WINDING SELECT & CANCEL C-AXIS COUPLING	30 ~ 6000min <sup>-1</sup>
M43	C-AXIS COUPLING ON	Switches from the cutting mode to the milling (rotary tool) mode.
M44	ROTARY TOOL COUPLING ON	
M45	ROTARY TOOL COUPLING OFF	
M46	SPINDLE OVERRIDE IS EFFECTIVE	The spindle override can be applied.
M47	SPINDLE OVERRIDE IS NOT EFFECTIVE	The spindle override is ignores.

M code	Function	Description
M48	FEEDRATE OVERRIDE IS NOT EFFECTIVE	The feedrate override can be applied.
M49	FEEDRATE OVERRIDE IS 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.
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	
M64	UNLOADER RETRACT	
M65	BAR FEEDER SUPPLY COMP.CHECK(ASQ 80)	This function is the bar feeder start check of bar feeder (ASQ type) made by ALPS.
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 CLOSE	The spindle chuck closes.
M69	CHUCK OPEN	The spindle chuck opens.
M70	SPARE OUTPUT SIGNAL	For bar feeder (ASQ type) made by ALPS.

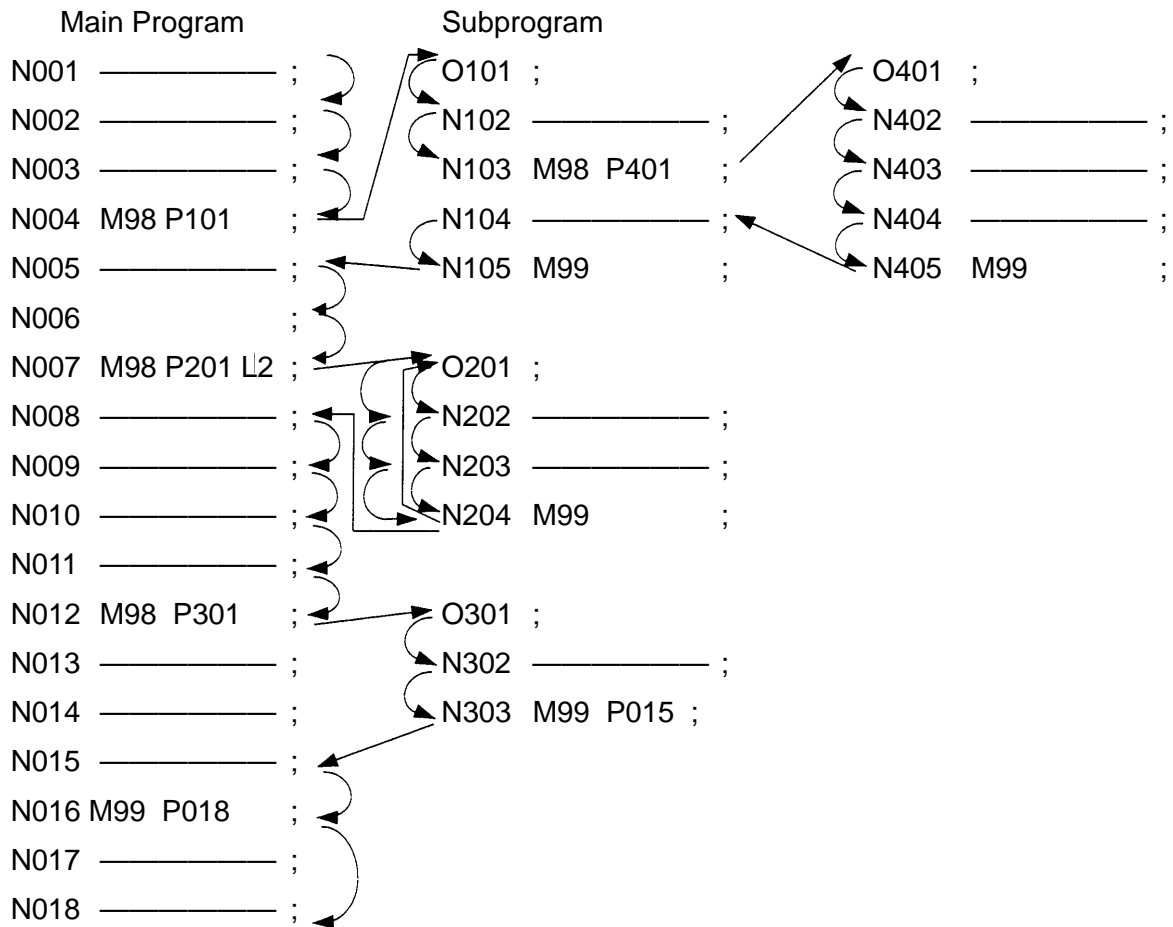
M code	Function	Description
M71	WORK MEASURING ARM OUT	Work measuring sensor swings out.
M72	WORK MEASURING ARM RETURN	Work measuring sensor is stored.
M73	WORK MEASURING SENSOR AIR BLOW ON	Air is blown to work measuring sensor. (In the custom macro of WORK MEASUREMENT FUNCTION)
M74	WORK MEASURING SENSOR AIR BLOW OFF	Air blow at the measuring sensor stops. (In the custom macro of WORK MEASUREMENT FUNCTION)
M75	CHIP CONVEYOR START	Chip conveyor rotates to normal direction.
M76	CHIP CONVEYOR STOP	Chip conveyor stops.
M81	ROBOT SERVICE REQUEST-1	ROBOT START-1
M82	ROBOT SERVICE REQUEST-2	ROBOT START-2
M83	AUTO PRESETTER CHUCK INTERLOCK OFF	When measuring arm swings, chuck open/close condition is neglected.
M84	AUTO PRESETTER CHUCK INTERLOCK ON	When measuring arm swings, chuck open/close condition becomes effective.
M88	MACHINE PROPER STANDBY	The NC unit temporarily stand by. It is restarted by cancel signal from the robot.
M89	FEEDER STANDBY OFF	Robot on standing by is restarted by cancel signal 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.
M100	C - AXIS BRAKE ON	Used during C-axis couplig. With the brake applied, spindle rotation or C-axis move are disabled.

M code	Function	Description
M101	C - AXIS BRAKE OFF	
M102	SPINDLE FORWARD START (CHUCKING CONDITION IS NEGLECTED)	
M110	TURRET HEAD AIR BLOW ON	Air is blown from turret head.
M111	TURRET HEAD AIR BLOW OFF	Air blow from iturret head stops.
M122	AIR BLOW FROM SPINDLE ON	Air is blown inside spindle.
M123	AIR BLOW FROM SPINDLE OFF	Air blow from inside spindle stops.
M132	SPINDLE THROUGH COOLANT ON	Dischage the coolant from spindle.
M133	SPINDLE THROUGH COOLANT OFF	Stops dischaging the coolant from spindle.
M140	WORK SETTING CHECK	
M141	M CODE EXTERNAL FUNCTION 1	
M142	M CODE EXTERNAL FUNCTION 2	
M143	M CODE EXTERNAL FUNCTION 3	
M144	M CODE EXTERNAL FUNCTION 4	
M145	M CODE EXTERNAL FUNCTION 5	
M162	SP. ROTATE CW + AIR BLOW ON	
M163	SP. ROTATE CW + AIR BLOW OFF	

M code	Function	Description
M167	DOOR OPEN +SP. STOP +COOLANT STOP	
M171	AUTO DOOR OPEN (ONE SHOT)	
M172	AUTO DOOR CLOSE (ONE SHOT)	
M173	CENTER FORWARD (ONE SHOT)	
M174	CENTER RETRACT (ONE SHOT)	
M201 }	ROBOT SERVICE REQUEST 1 }	
M231	ROBOT SERVICE REQUEST 31	
M260	WORK SETTING CHECK SOL ON (ONE SHOT)	
M263	UNLOADER FORWARD (ONE SHOT)	
M264	UNLOADER RETRACT (ONE SHOT)	
M285	SPINDLE SPEED CHANGE CONTROL ON	
M286	SPINDLE SPEED CHANGE CONTROL OFF	
M292	CENTER FORWARD OT LS DISABLE	
M293	CENTER FORWARD OT LS ENABLE	

## 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


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 Yp: Y-axis } parallel 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	G28	Reference point return	
	G30	2nd, 3rd and 4th reference point	
	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
	G64	Cutting mode
00	G65	Macro calling
14	G66	Macro modal calling
	G67	Macro modal 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
00	G194	External measurement compensation
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
00	G921	Work coordinate system preset

**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 ( $\text{m/min.}$  or  $\text{ft/min.}$ ) will be clamped to this upper limit.

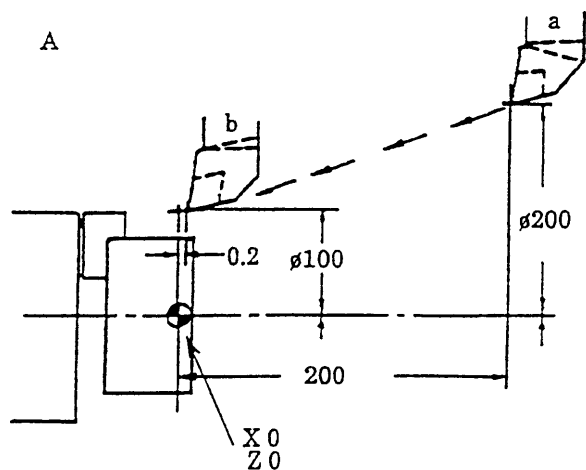
(Example) G50 S2000 ;

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.



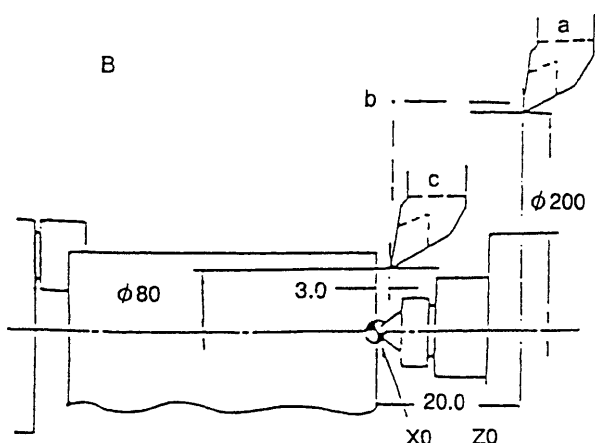
#### Chuck work

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.

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

G00 Z3.0

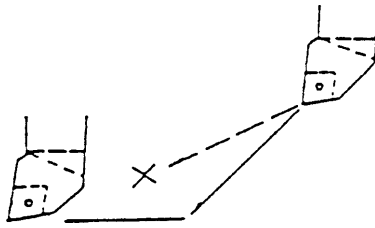
From the point "d" to the point "c" X80.0

2. 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.



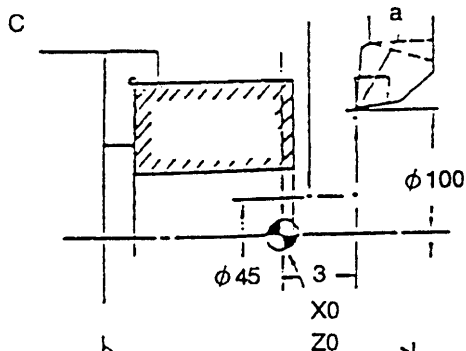
After one of 2 axes (X and Z) has completed its move, the other one moves to a specified point. The tool does not move linearly as shown with a dotted line in the left figure.

### 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.

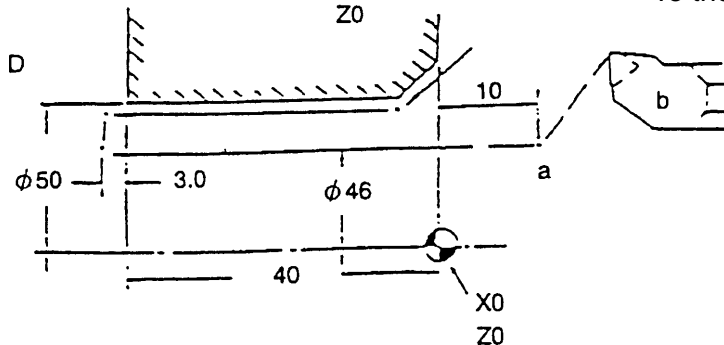
### End face OD cutting

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



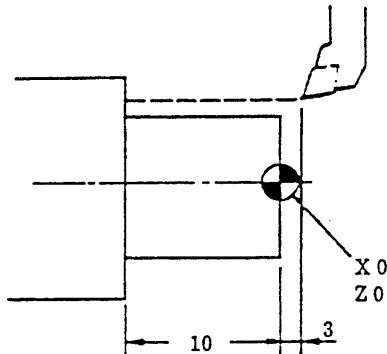
## 2-3-4 G01 Linear Cutting

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

Chamfering and taper cutting are also considered linear cutting.

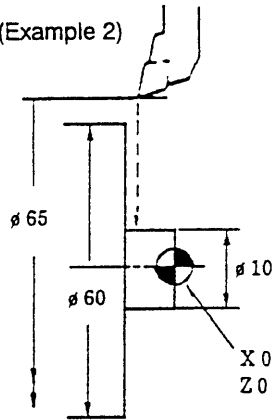
Use an F code to specify a feeding rate.

(Example 1)



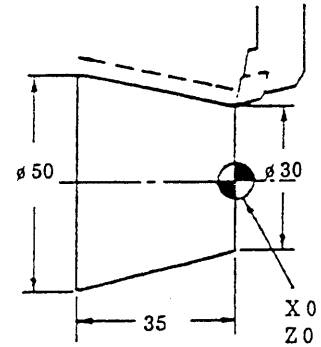
G01 Z-10.0 (F)...;

(Example 2)



G01 X10.0 (F)...;

(Example 3)



G01 X50.0 Z-35.0 (F)...;

### 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 G00 command is commanded next to G01, 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 ...

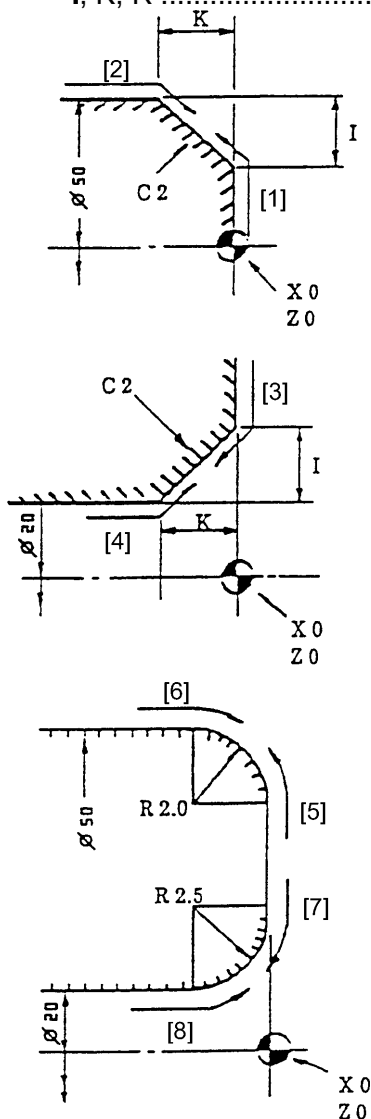
### For corner R

(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.

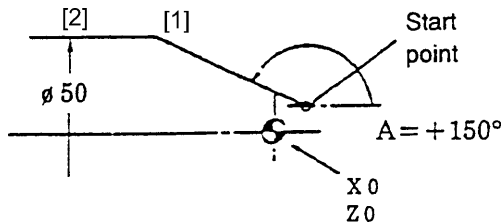
### (3) Angle designated linear interpolation

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

$$\left. \begin{array}{c} X(U) \\ Z(W) \end{array} \right\} G01 \dots A \dots F \dots ;$$

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.



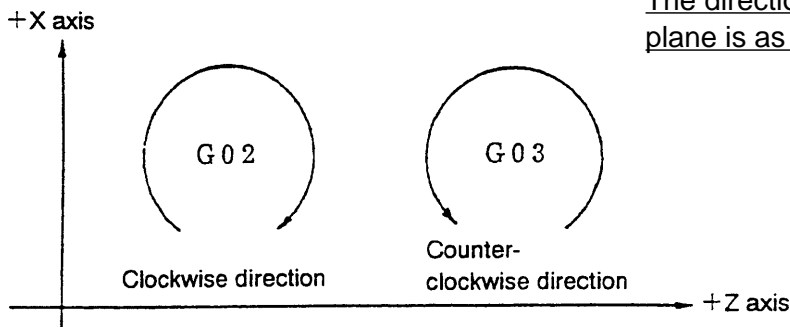
(1) G01 X50.0 A150.0 F0.3;

(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:

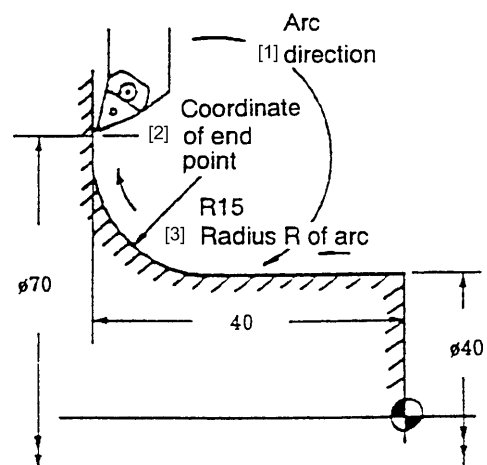
[1] Circular arc direction G02 or G03

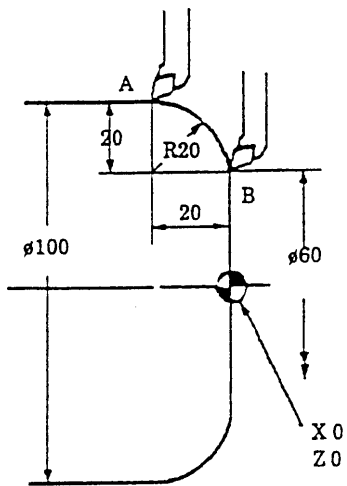
[2] X and Z coordinate values of a circular arc end point

[3] Circular arc radius R (radius designation)

Example : G01 Z-25.0

G02	X70.0	Z-40.0	R15.0
↑	↑	↑	↑
[1]	[2]	[3]	

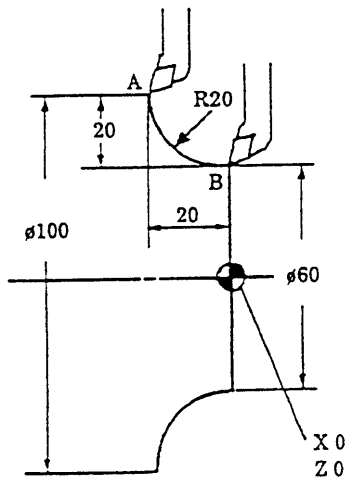




(Example 1)

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

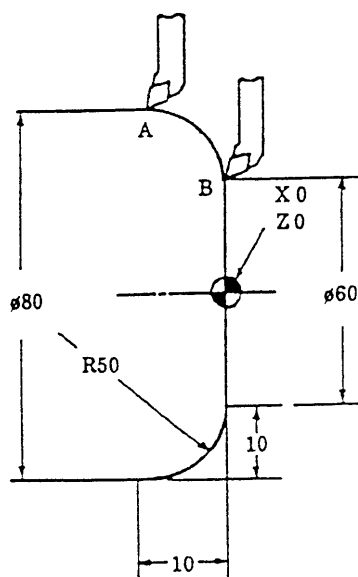
When moving from the point B to the point A  
G03 X100.0 Z-20.0 R20.0 F...;



(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...;

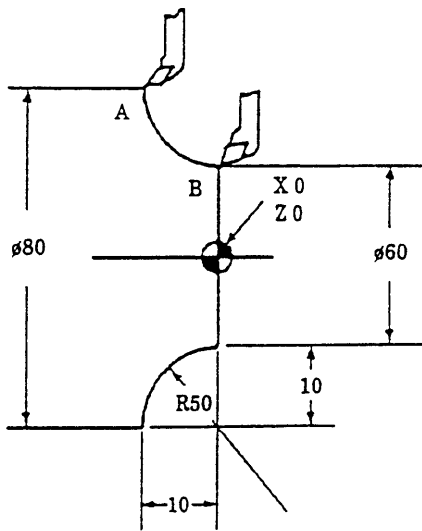


(Example 3)

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...;





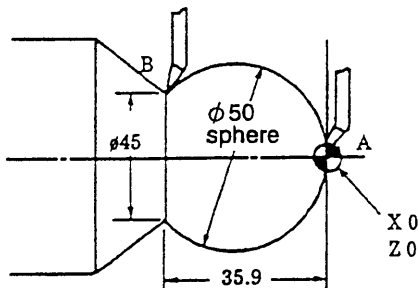
(Example 4)

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

G02 X80.0 Z-10.0 R50.0 F...;



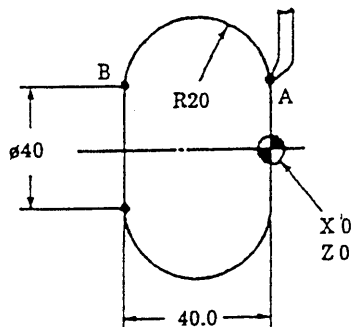
(Example 5)

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

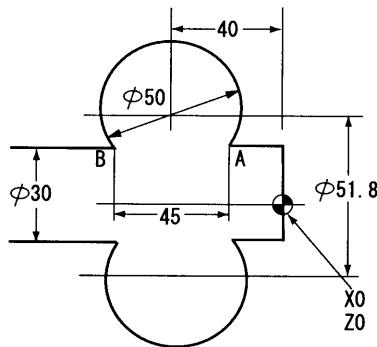
G03 X40.0 Z-40.0 R20.0 F...;

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...;

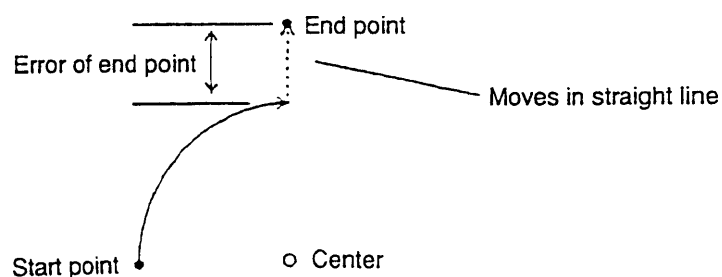
When moving from the point B to the point A

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 less than the parameter setting value(No.3459). And when it is out of the parameter setting value, an alarm will occur.

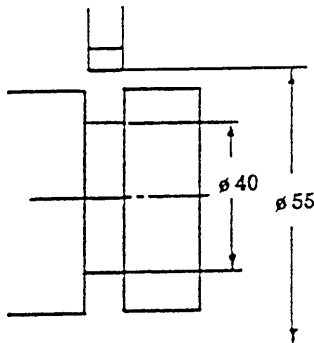


- 5) When I, J, K and R are commanded in the same block, R has priority.

### 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  $600 \text{ min}^{-1}$ , 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.

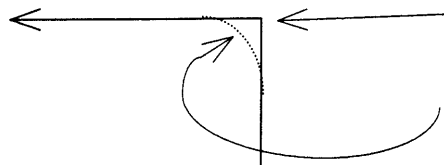
(1) Command form

G09 — ;

(2) Program example

N1 G09 G01 U50. F ;

N2 G01 W -50. ;



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

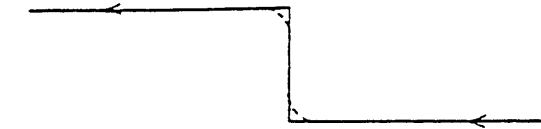
When not commanding G09, a round is created on the corner.

### 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

Setting of the second or third stroke limit can be set 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.*

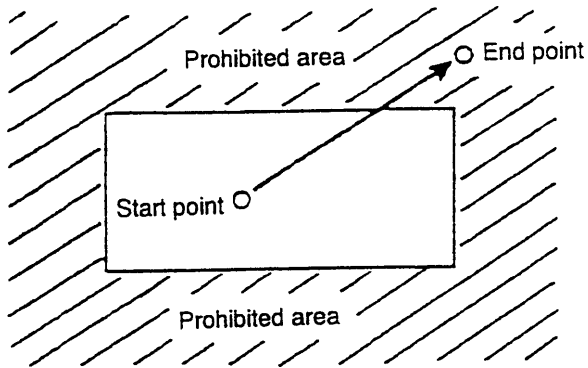
2) *When G22 X\_ Z\_ I\_ K\_ ; is commanded, the parameter changes automatically to the commanded value.*

3) *When the power is on, G22 is set automatically.*

4) *When a parameter is changed, please be sure to push "RESET" key.  
It is not changed unless it pushes "RESET" key.*

### 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.

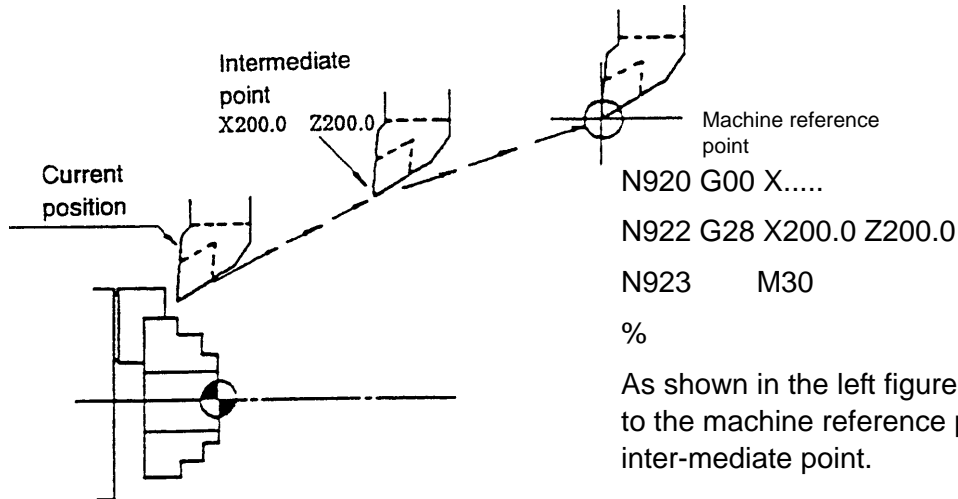
**Note)** *If the travel time of one block is very short, some times becomes an alarm before setting the remaining amount of travel.*

(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 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.

### 2-3-14 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)

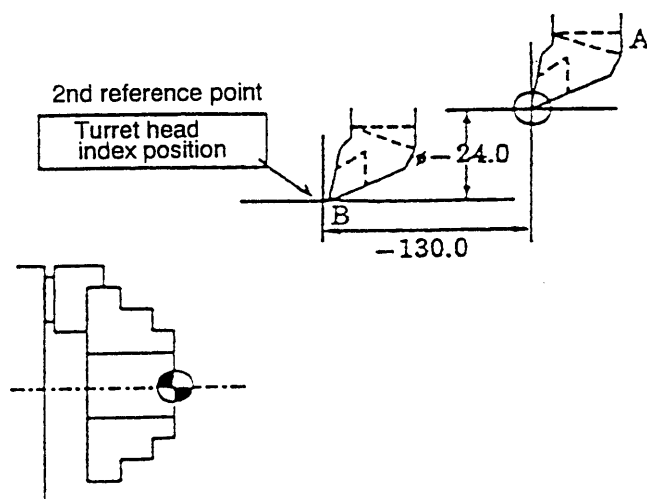
N1015 M01

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.

## (1) 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.

## (2) Position of each reference point

The position of each reference point is set previously by

the parameter { P2 : MD 34100 [0]  
P3 : MD 34100 [1]  
P4 : MD 34100 [2] } as a distance from the machine reference point.

<Program example>

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

## 2-3-15 G31 Skip Function

Linear interpolation is performed by a G31 command. If an external skip signal is input during linear interpolation, the program proceeds to the next block, stopping the axes and discarding the remaining stroke.

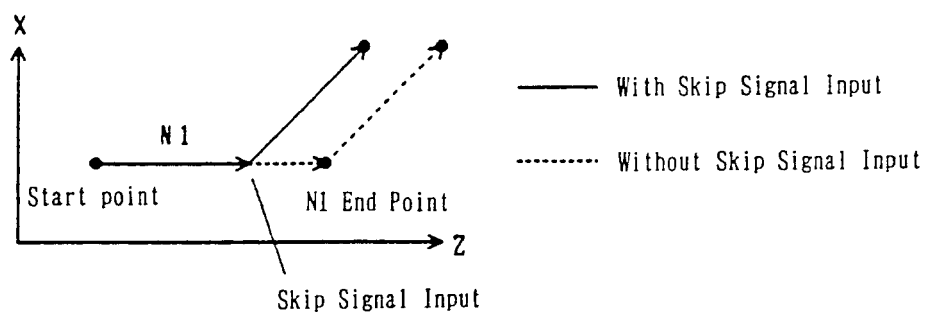
### (1) Command Format

G31 X\_\_\_ Y\_\_\_ Z\_\_\_ ..... F\_\_\_ ]

### (2) Sample Program

N1 G98 G31 W50. F100 ]

N2 G01 U50. W25. ]



### (3) Cautions

Position where skip has been input (work coordinates) can be read by the macro system variable.

## 2-3-16 G54 Work Coordinate System Setting (Work Length)

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

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

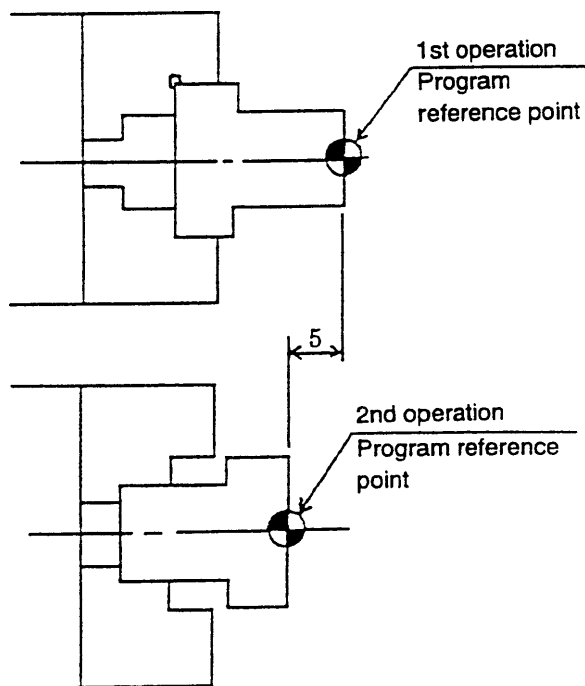
Correct distance is displayed of the tool position from the machine origin by following procedures.

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

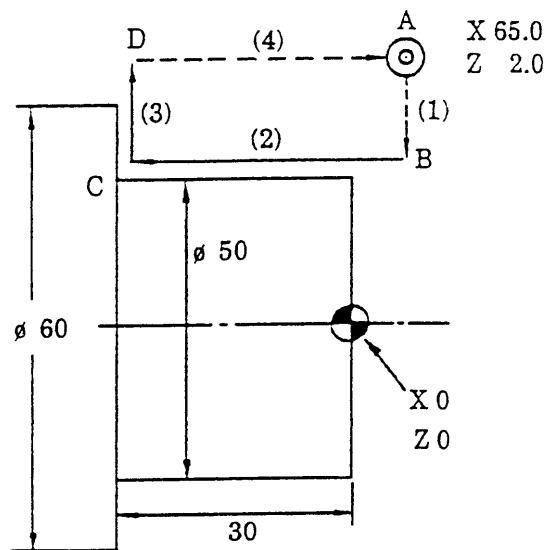
N100
T0100
G97 S1230 M08
    } 1st operation machining
G28 U0
G28 W0
M00..... Work turn-over

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
    
```

## 2-3-17 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.

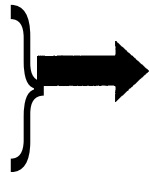


The tool starts from the point A (X65.0, Z2.0) and returns to the point A via the points B, C and D respectively.

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

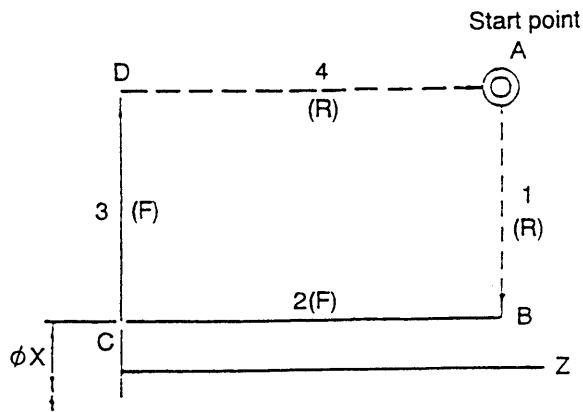
### 1. G90 OD/ID cutting cycle

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

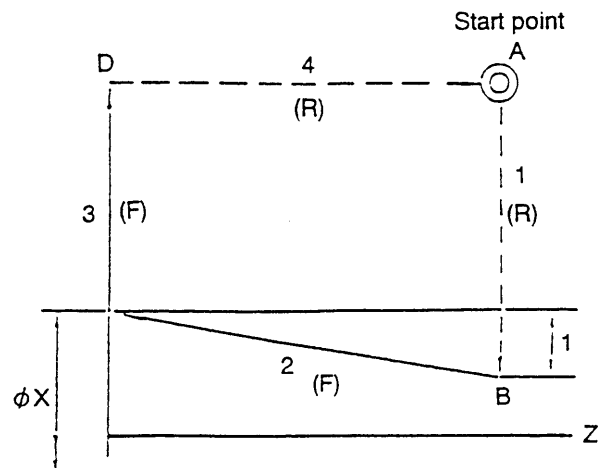


G90 X...Z...F... ] (I=0)

R : Rapid traverse

F : Cutting feed  
(specified with an F code)

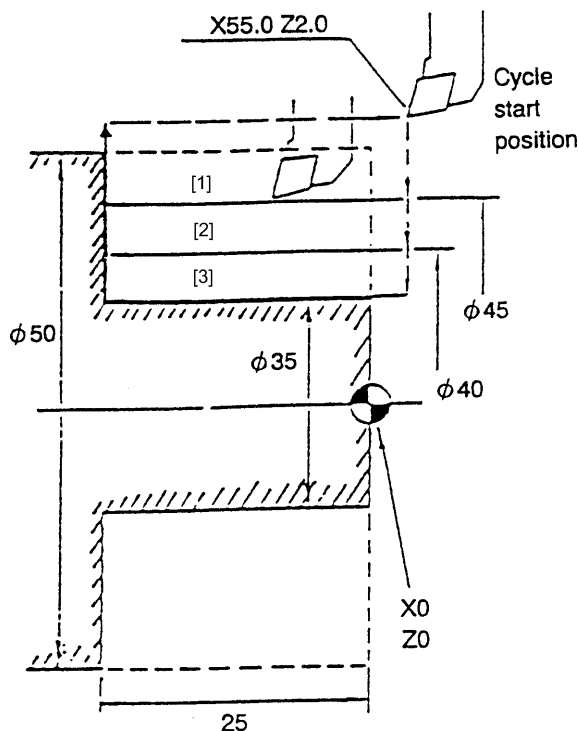
### (2) Taper cutting



G90 X...Z...I...F... ]

(Pay attention to a sign of I.)

### 1. Example of straight cutting

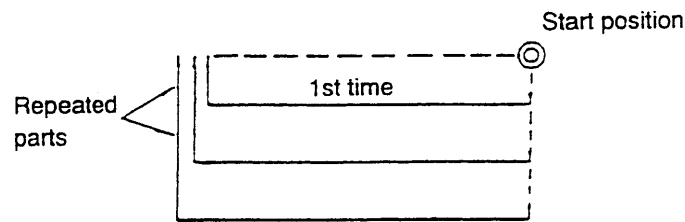


When machining a  $\phi 50$  blank as shown in the left figure, with its start point at X55.0 and Z2.0 and a depth of cut of 2.5 mm, the program is as follows:

```
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 Z10.0
N109 G30 U0 W0
N110 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.

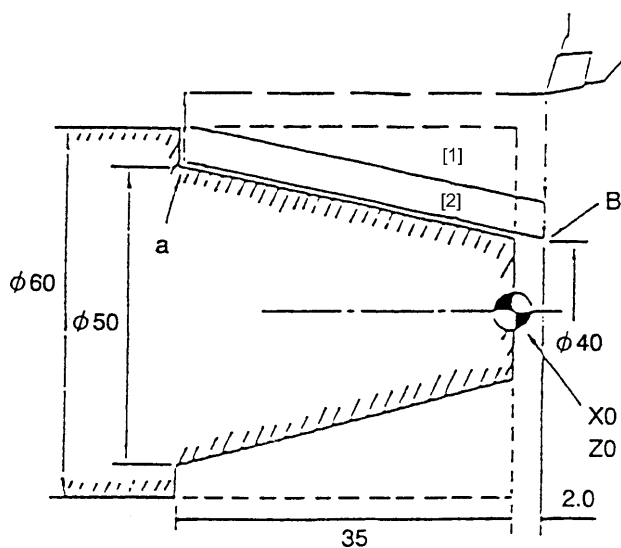
Therefore, you must specify a G90 command and coordinate values each time.

## 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;  $I = -5.0$

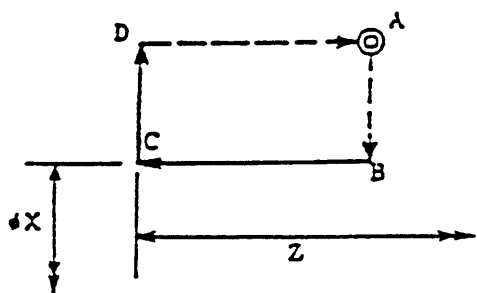
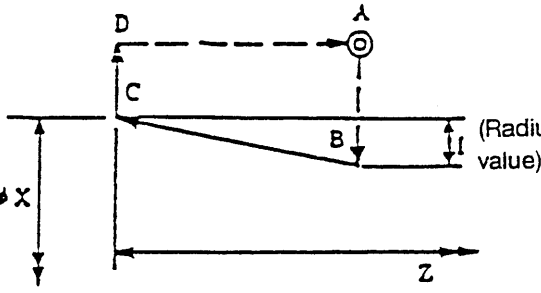
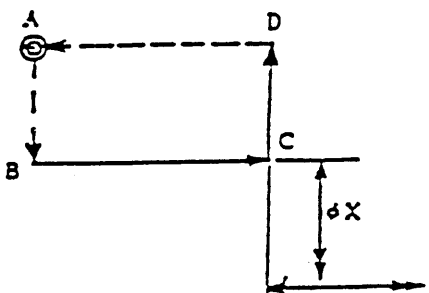
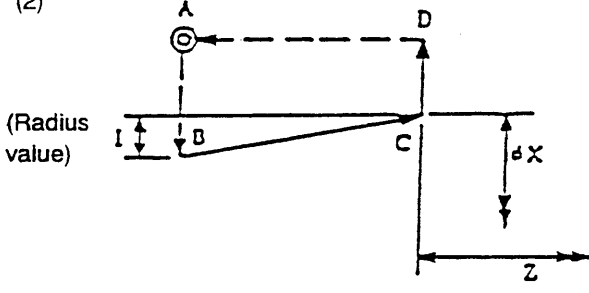
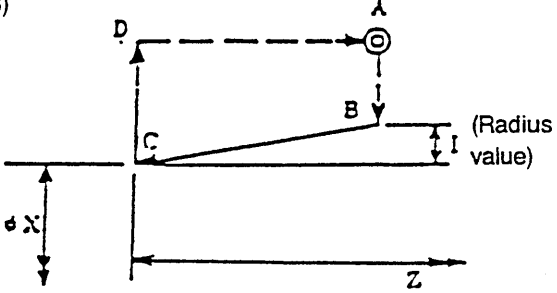
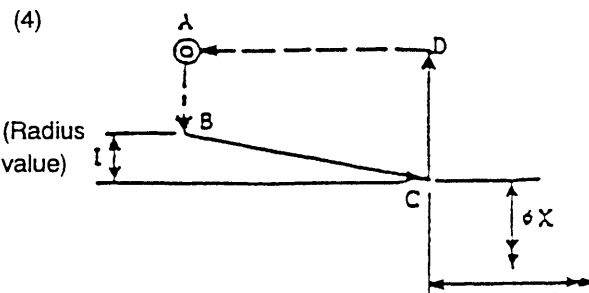


```

N103 G00 X65.0 Z10.0 M03
N104 G01 G96 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
N103 G00 X65.0 Z10.0 M03
N104 G01 G96 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.....

```

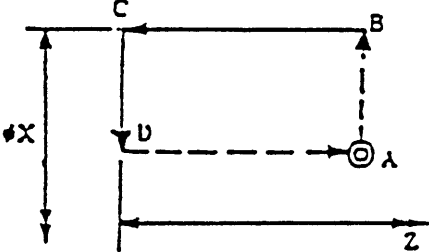
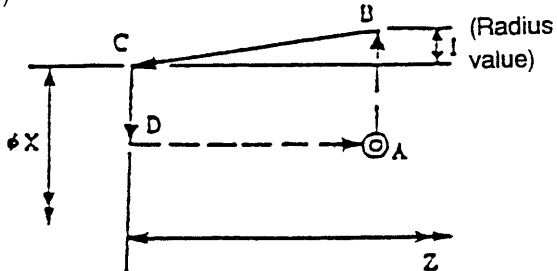
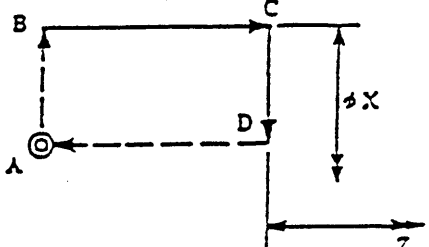
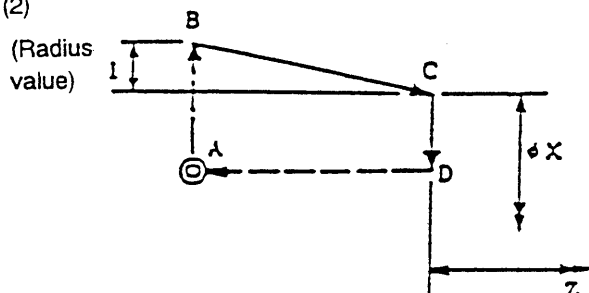
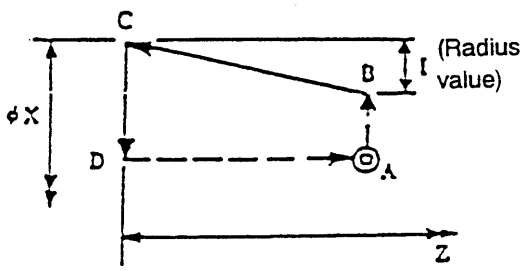
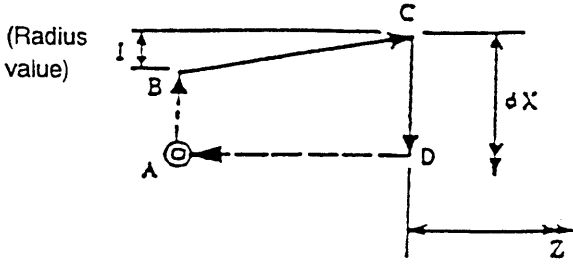
# G90 Cycle Patterns (OD)

Straight	Taper
 <p>G90 X..... Z.....</p>	<p>(1)</p>  <p>G90 X..... Z..... I (-) .....</p>
 <p>G90 X..... Z.....</p>	<p>(2)</p>  <p>G90 X..... Z..... I (-) .....</p>
	<p>(3)</p>  <p>G90 X..... Z..... I.....</p>
	<p>(4)</p>  <p>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.

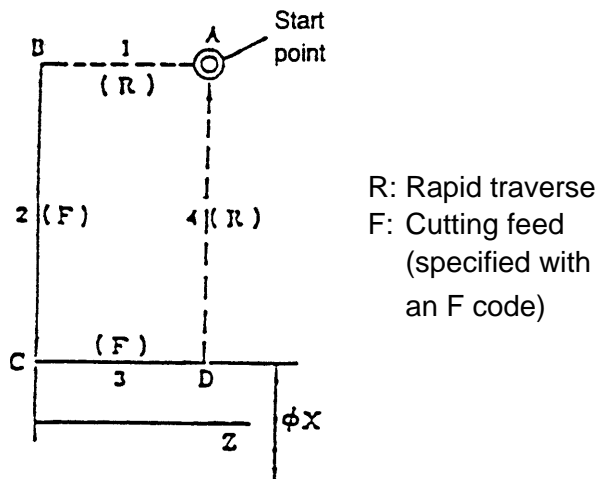
## 2. G94 End face and side cutting cycle

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

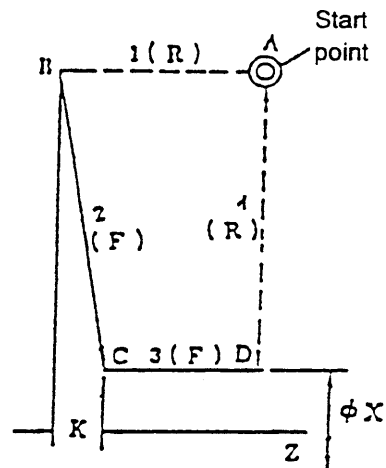
The tool moves via a specified point from its start point, cuts the workpiece at a feed rate specified with an F code and returns to the start point.

### G94 cycle patterns

#### (1) Straight cutting



#### (2) Taper cutting



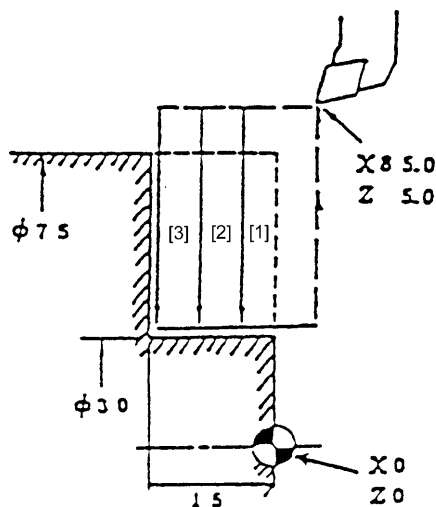
G94 X...Z...F...

(K=0)

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

(Pay attention to a sign of K)

### 1. Example of straight cutting

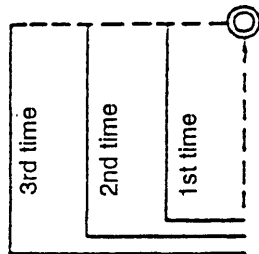


When machining a  $\phi 75$  blank as shown in the left figure, with its cycle start position at X85.0 and Z5.0 and a depth of cut of 5 mm, the program is as follows:

```

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 Z10.0
N109 G30 U0 W0
N110 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.



Repeated parts

```

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....

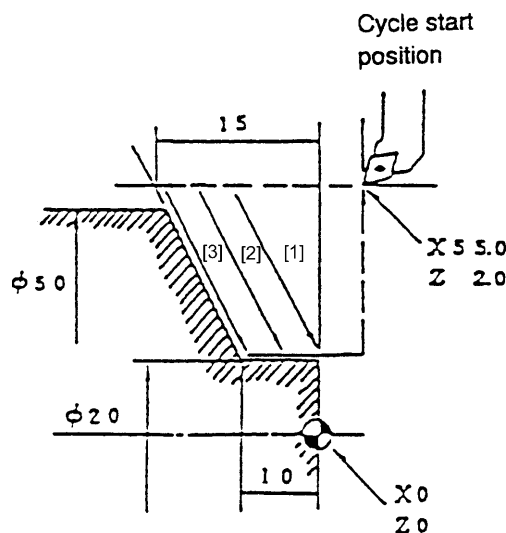
```

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.

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

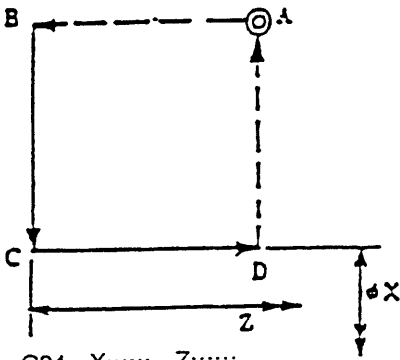
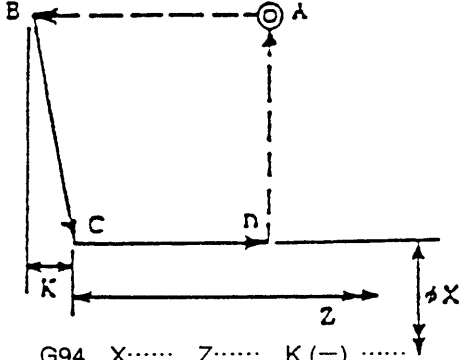
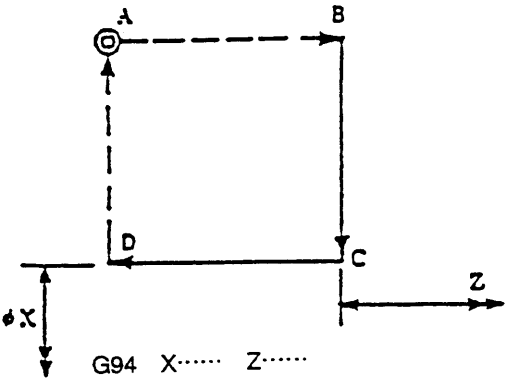
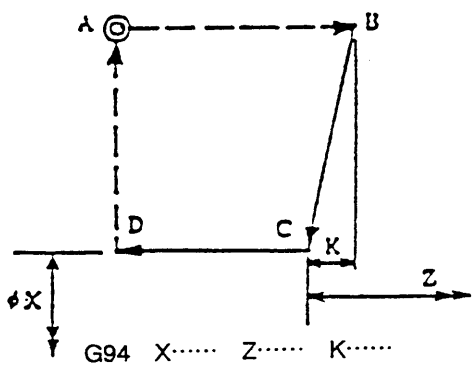
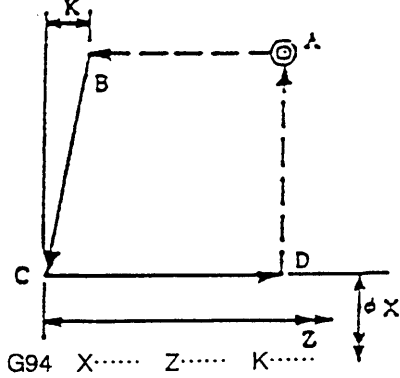
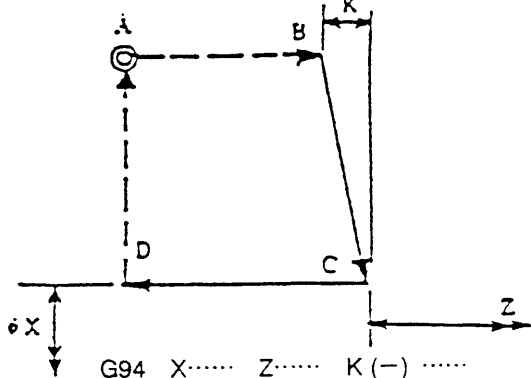
```

```

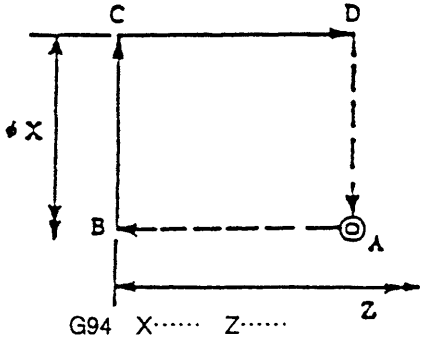
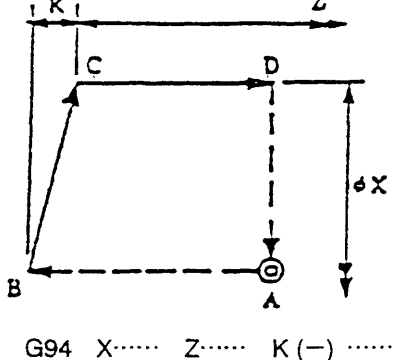
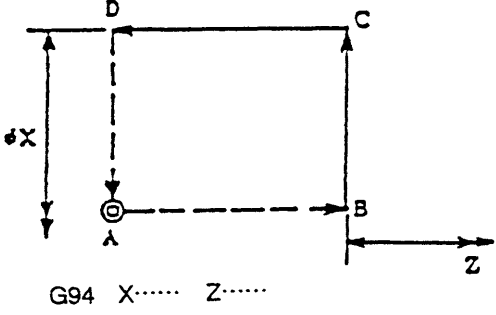
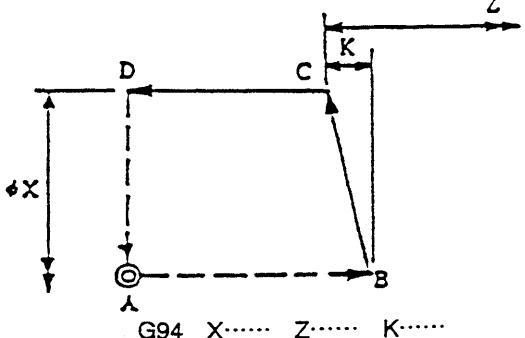
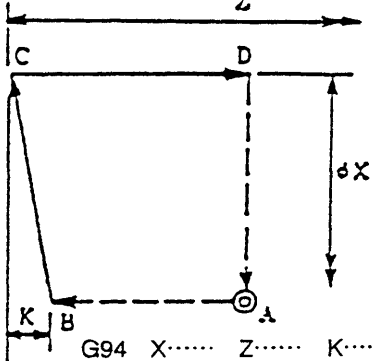
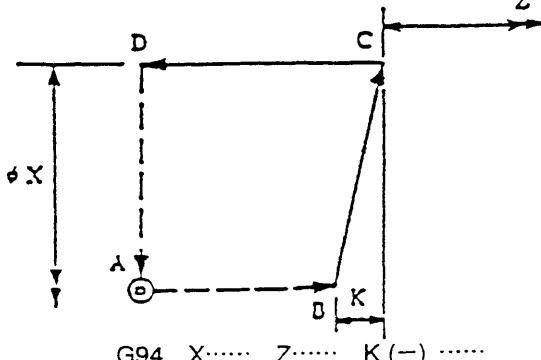
N104 G01 G96 X55.0 Z2.0 S120
N105 G94 X20.0 Z0 K-5.0 F0.2 ..... [1]
N106 G00 Z-3.0
N107 G94 X20.0 Z-5.0 K-5.0 ..... [2]
N108 G00 Z-8.0
N109 G94 X20.0 Z-10.0 K-5.0 ..... [3]
N110 G00 X.... Z....

```

# G94 Cycle Patterns (END FACE)

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>

# G94 Cycle Patterns (END FACE)

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>

### 2-3-18 Multiple Repetitive Cycle

This option consists of several fixed cycles which are preliminarily prepared to make the programs easier. For example, by giving only information on the shape of finishing, the tool path for the intermediate roughing can automatically be determined. And a fixed cycle for the thread cutting is also available.

7 kinds of fixed cycle (G70~G76) are readily available for the multiple repetitive cycle.

These G codes are all non-modal G codes.

G Code	Cycle Name	Remarks	
G70	Finishing cycle	Capable of finishing with G70	
G71	I.D./O.D. roughing cycle		
G72	End face roughing cycle		
G73	Closed loop cutting cycle		
G74	End face cutting-off cycle		
G75	Outer figure cutting-off cycle		
G76	Thread cutting cycle		

## (1) Rough Planing of Outside Diameter (G71)

There are type I and type II for the rough planing cycle.

### Type I

As shown in the figure below, if the finishing shape is given as  $A \rightarrow A' \rightarrow B$  by a program, the area specified by the cutting amount  $\Delta d$  is cut off, leaving the finish amount  $\Delta u/2$ ,  $\Delta w$ .

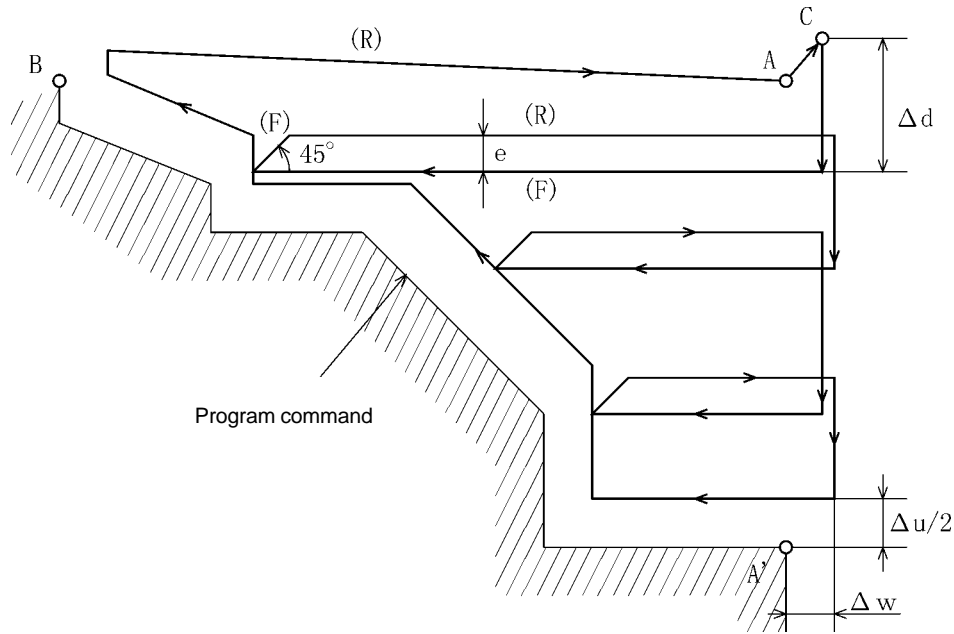


Fig. 18.1 Cutting route of rough planing during turning (Type I)

G71 P(ns) Q(nf) U( $\Delta u$ ) W( $\Delta w$ ) D( $\Delta d$ ) F(f) S(s) ]

N(ns) .....

.....

F\_\_\_\_ The shift of the finishing shape, i.e.  $A \rightarrow A' \rightarrow B$ , is commanded by the blocks  
S\_\_\_\_ of sequence No. ns to nf.

N(nf) ..... ]

$\Delta d$ : Cutting amount

Specify without a sign. The cutting direction is determined by the direction of  $AA'$  (radius specification). This specification is valid until specified next by the modal.

e: Relief amount

Parameter GUD7,\_ZSFI[31]

ns: Sequence No. of the first block in the finishing shape block group.

nf: Sequence No. of the last block in the finishing shape block group.

$\Delta u$ : Finishing amount of X axis direction (diameter specification)

$\Delta w$ : Finishing amount of Z axis

F, S: During the cycle, the F function, S function specified by the block between ns~nf are neglected, and the data of the F function, S function specified by the G71 block become enabled.

Note:

- (1) Following four patterns are conceived as the shapes cut by G71. In any case, the work is cut by the tool shift which is parallel to the Z axis, where the signs for  $\Delta u$ ,  $\Delta w$  are as follows.

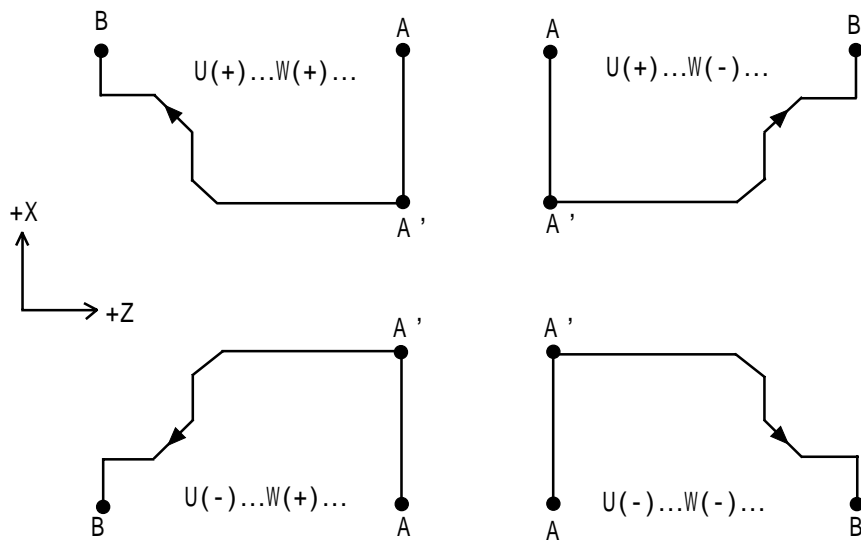


Fig. 18.2 Signs for U and W of G71

A-A' is the block of sequence No. ns, which specifies the command including G00 or G01. The shift amount of Z axis is not specified to the block of ns. The shape of A'-B' needs to simply increase or decrease for both X axis and Y axis directions.

- (2) The sub program cannot be called from the block of sequence No. ns to nf.



## Type II

Type II is different from type I in the following respect. The shape is not required to simply increase or decrease to X direction and you can have dents (pocket).

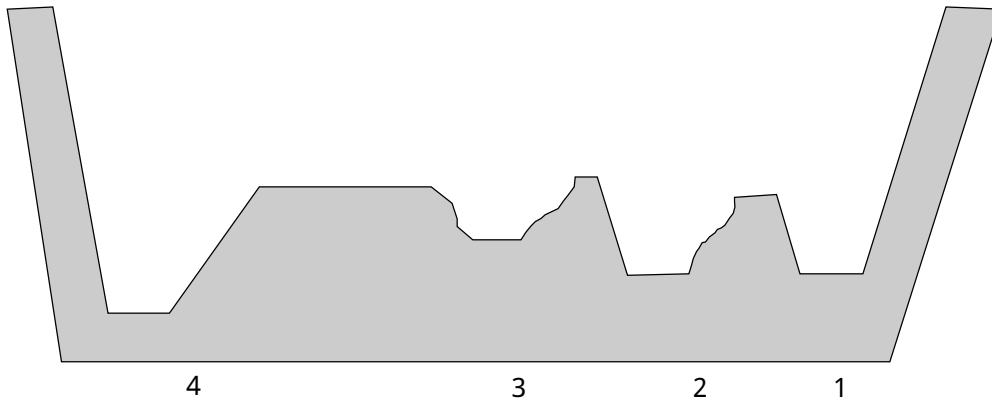


Fig. 18.3 Pockets of outside diameter rough planing (type II)

However, it must simply changes to Z direction. If the shape is like below, machining is not possible.



Fig. 18.4 Shape that cannot be processed with G71 cycle.

## Different use of type I and type II

When only X axis is specified in the first block of the finishing shape ... type I

When X axis and Y axis are specified in the first block of the finishing shape ... type II

If you want to use type II without Z moving to the first block, specify WO.

(Example)

Type I

G71 P100 Q200 ...」

N100 X(U) \_\_\_\_」

:

:

N200 .....」

Type II

G71 P100 Q200 ...」

N100 X(U) \_\_\_\_ Z(W) \_\_\_\_」

:

:

N200 .....」

## (2) Rough Planing Cycle of End Side (G72)

As shown in the figure below, this is the same as G71, but the cutting is done by the movement parallel to the X axis.

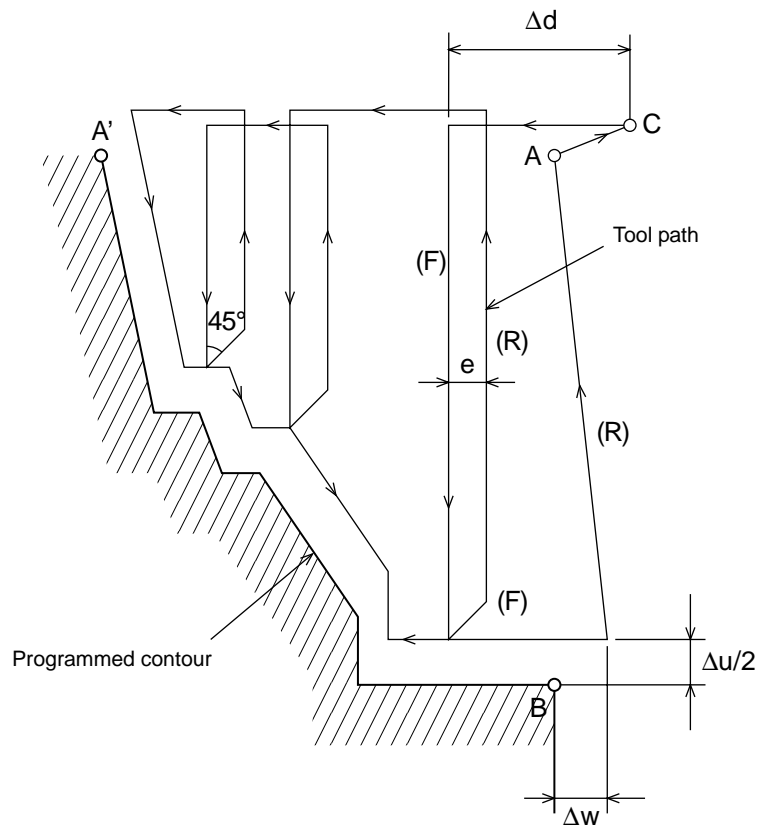


Fig. 18.5 Cutting route of face rough planing cycle

G72 P(ns) Q(nf) U( $\Delta u$ ) W( $\Delta w$ ) D( $\Delta d$ ) F(f) S(s) ]

$\Delta d$ ,  $e$ ,  $ns$ ,  $nf$ ,  $\Delta u$ ,  $\Delta w$ ,  $f$ ,  $s$  are the same as G71.

Following four patterns are conceivable as the shapes cut by G72. In any case, the work is cut by the repeated movement of the tool parallel to the X axis. Signs for  $\Delta u$  and  $\Delta w$  are as follows.

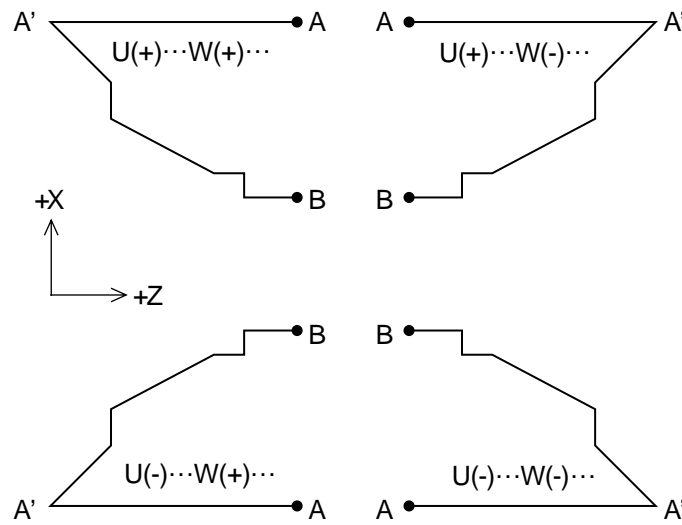


Fig. 18.6 Signs for U and W of G72

A-A' is the block of sequence No. ns, which specifies the command including G00 or G01. The X axis cannot be specified to the block of ns. The shape of A'-B needs to simply increase or decrease for both X axis and Z axis directions.

### (3) Planing Cycle of Close Loop (G73)

You can repeatedly use a certain cutting pattern gradually shifting its position. Using this cycle, you can efficiently cut a work with a material shape made by pre-machining such as forging and casting.

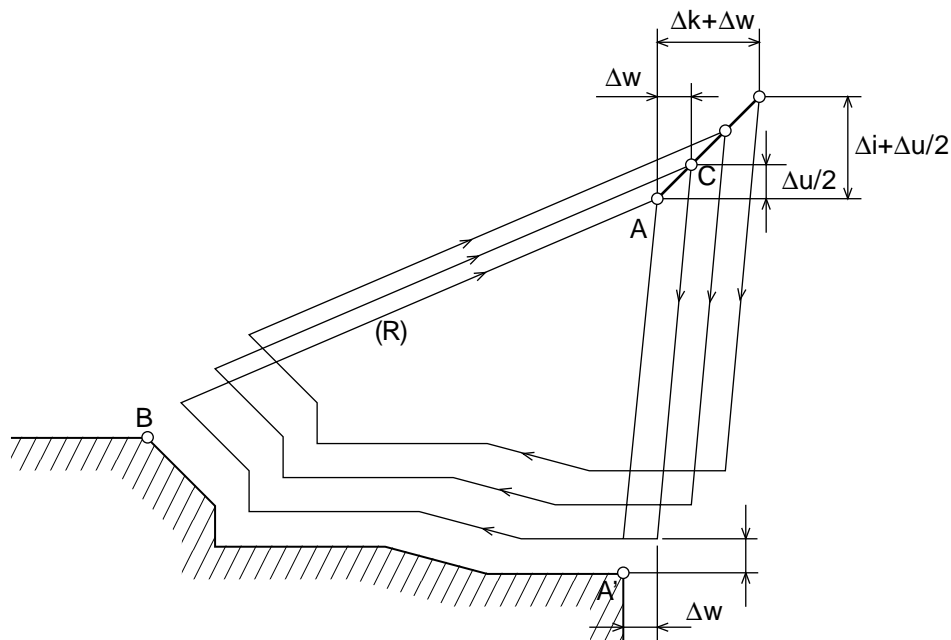


Fig. 18.7 Cutting route of close loop planing cycle

Pattern commanded by a program.

A point→A' point→B point

G73 P(ns) Q(nf) U(Δu) W(Δw) I(Δi) K(Δk) D(d) F(d) F(f) S(s) ]  
N(ns) .....

.....

F \_\_\_\_ Shift command of finishing shape for A→A'→B is specified by blocks of  
S \_\_\_\_ sequence No. ns to nf.

N(nf) ..... ]

Δi: Removal amount for X axis direction (specify radius)

Δk: Removal amount for Z axis direction

d: Times of rough planing.

ns: Sequence No. of the first block in the finishing shape block group.

nf: Sequence No. of the last block in the finishing shape block group.

Δu: Finishing amount for X axis direction (specify diameter).

Δw: Finishing amount for Z axis direction.

f, s: If F function or S function is specified to any block between ns~nf, it is neglected and the data of F function or S function specified by G73 block become enabled.

Note:

- (1) Cycle operation is executed by G73 command with P and Q specified. Since there are four patterns as the cutting shapes, take care for the signs for  $\Delta u$ ,  $\Delta w$ ,  $\Delta i$  and  $\Delta k$  when you program. When the cycle ends, it returns to the A point.

#### (4) Finish Cycle (G70)

If you have executed the rough planing with G71, G72 and G73, you can execute finish planing with a following command.

Command format

G70 P(ns) Q(nf) ;

ns: Sequence No. of the first block in the finish shape block group.

nf: Sequence No. of the last block in the finish shape block group.

Note:

- (1) With G70, the F, S function specified by the block of either G71, G72 or G73 are neglected and the F, S function specified in the sequence No. from ns to nf are enabled.
- (2) When G70 cycle ends, the tool is returned to the starting point by fast feed and the block next to the G70 cycle is read in the CNC command data.
- (3) A sub program cannot be called in the block within the sequence No. from ns to nf which are used in G70~G73.

## Example

### Outer diameter rough planing cycle (G71)

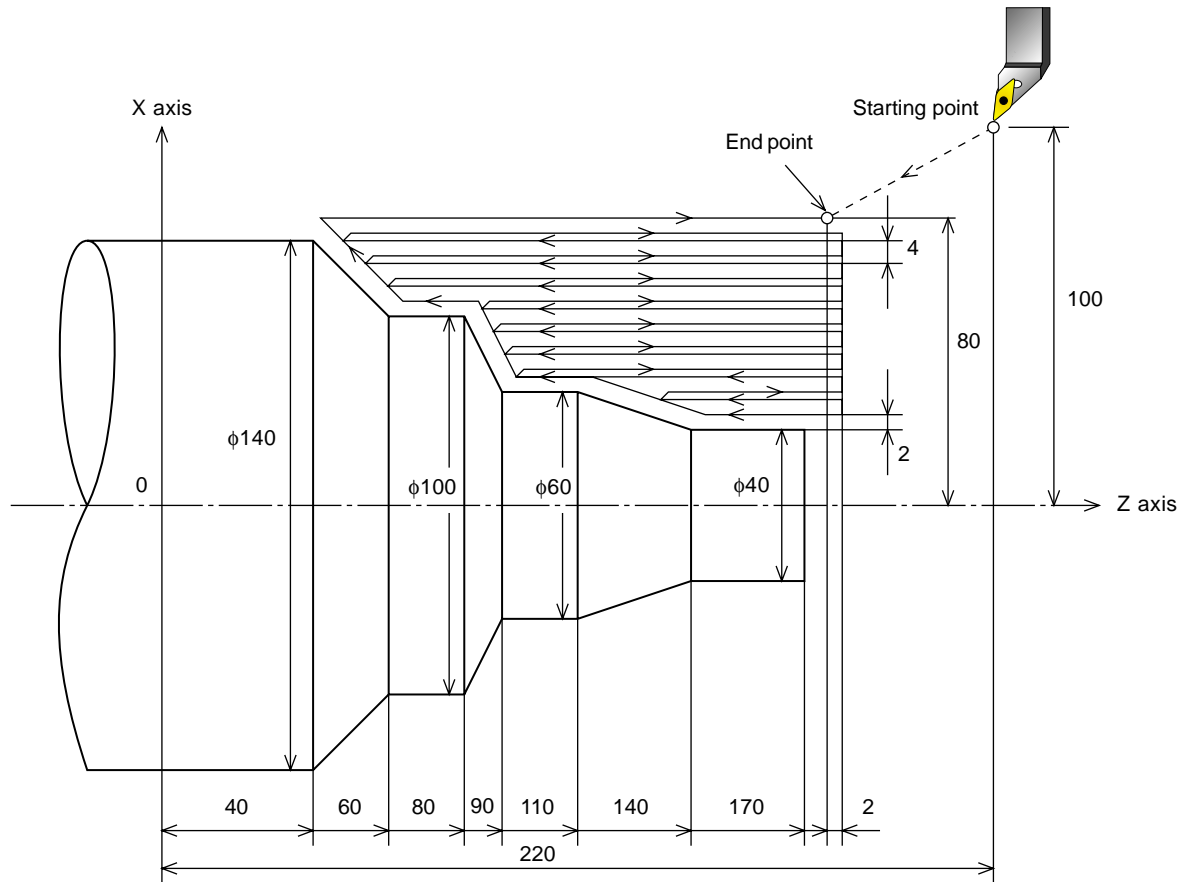


Fig. 18.8 Outer diameter rough planing cycle

(Diameter specification, millimeter input)

```

N10 G00 G90 X200.0 Z220.0 ]
N11 X142.0 Z171.0 ]
N12 G71 P13 Q19 U4.0 W2.0 D4.0 F0.3 S550 ]
N13 G00 X40.0 F0.15 S700 ]
N14 G01 Z140.0 ]
N15 X60.0 Z110.0 ]
N16 Z90.0 ]
N17 X100.0 Z80.0 ]
N18 Z60.0 ]
N19 X140.0 Z40.0 ]
N20 G70 P13 Q19 ]
N21 G00 X200.0 Z220.0 ]
    
```



## Example

### Face rough planing cycle (G72)

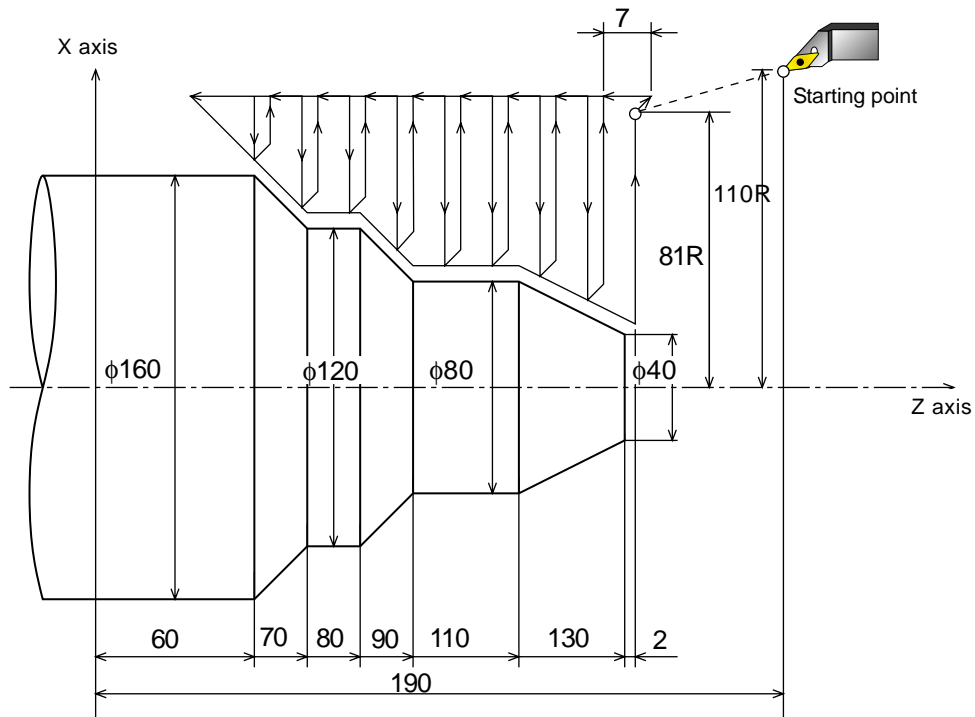


Fig. 18.9 Face rough planing cycle

(Diameter specification, millimeter input)

```

N10 G00 X220.0 Z190.0 ]
N11 G00 X162.0 Z132.0 ]
N12 G72 P13 Q18 U4.0 W2.0 D7.0 F0.3 ]
N13 G00 Z59.5 F0.15 S200 ]
N14 G01 X120.0 Z70.0 ]
N15 Z80.0 ]
N16 X80.0 Z90.0 ]
N17 Z110.0 ]
N18 X36.0 Z132.0 ]
N19 G70 P13 Q18 ]
N20 X220.0 Z190.0 ]
  
```

## Close loop planing cycle (G73)

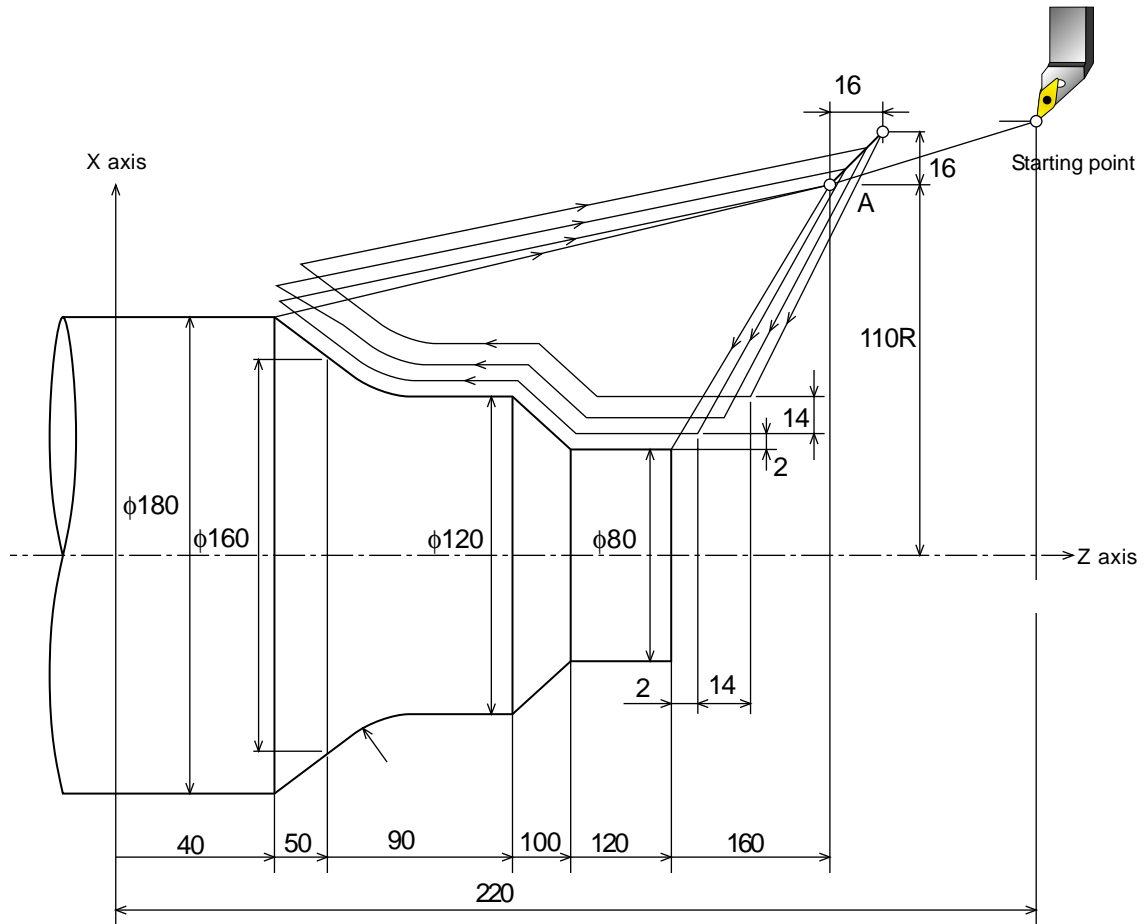


Fig. 18.10 Pattern repeat cycle

(Diameter specification, millimeter input)

```

N10 G00 X260.0 Z220.0 ]
N11 G00 X220.0 Z160.0 ]
N12 G73 P13 Q18 U4.0 W2.0 I14.0 K14.0 D3 F0.3 S180 ]
N13 G00 X80.0 Z120.0 ]
N14 G01 Z100.0 F0.15 ]
N15 X120. Z90.0 ]
N16 X70.0 ]
N17 G02 X160.0 Z50.0 R20.0 ]
N18 G01 X180.0 Z40.0 F0.25 ]
N19 G70 P13 Q18 ]
N20 G00 X260.0 Z220.0 ]
    
```

With following program commands, cutting routes are made as shown in the Fig. 18.11 below. Using this cycle, chips generated in the outer diameter cutting are disposed of. And if you omit X(U) and I, the operation is confined to the Z axis only enabling the deep hole drill cycle.



f: Feed speed.

(6) Outside, Inside Diameter Edge Cutting Cycle (G75)

With a following program command, the cutting route is made as shown in the Fig. 18.12 below, which corresponds to G74 with X and Z replaced. Using this cycle, chips coming from the face cutting can be disposed of. It also executes groove machining and edge-cut machining (omitting Z, W and K) when cutting the outside diameter.

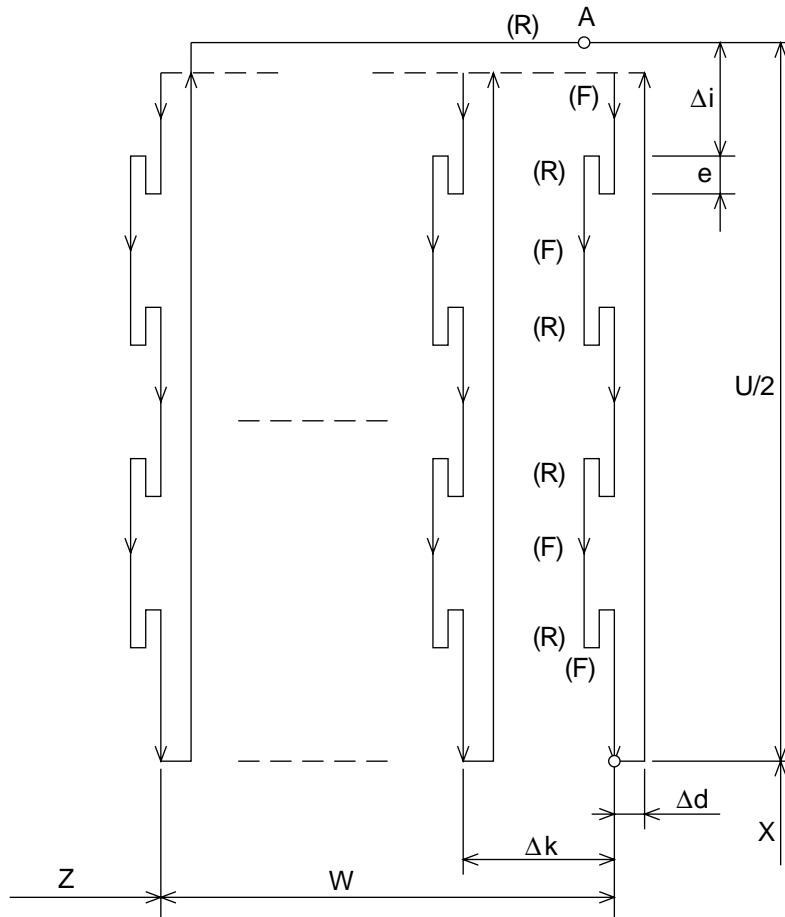


Fig. 18.12 Cutting route of outside diameter edge cutting cycle

G75 X(U)\_ Z(W)\_ I(Δi) K(Δk) R(Δd) F(f) ]

Both G74 and G75 are used for groove cutting or boring, are the cycle to automatically relieve the tool and four patterns which are symmetric to each other are conceived.

### (7) Combined Type Thread Cutting Cycle (G76)

With a following program command, the thread cutting cycle is executed as shown in the Fig. 18.13 below.

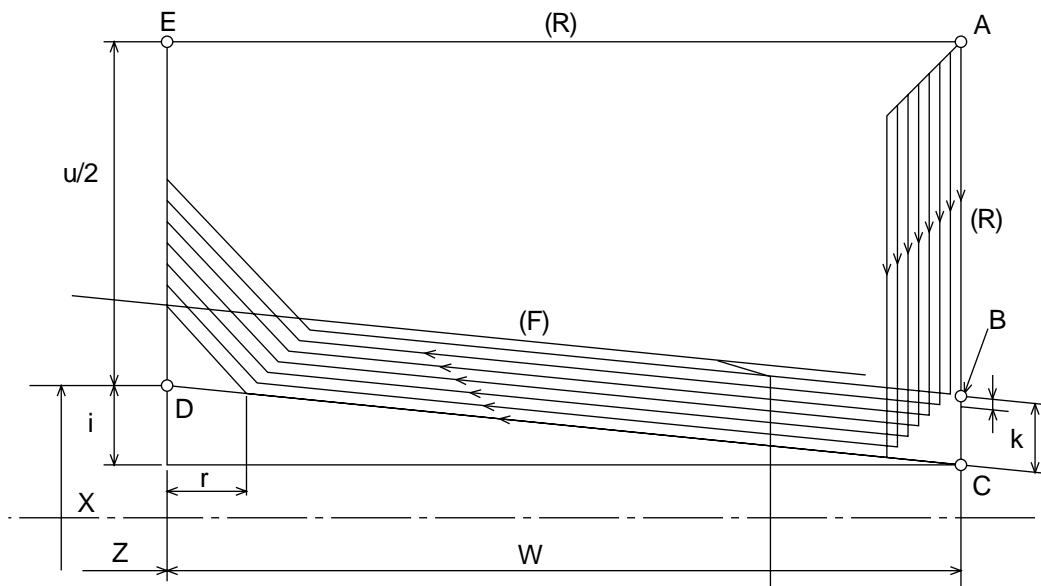


Fig. 18.13 Cutting route of automatic thread cutting cycle

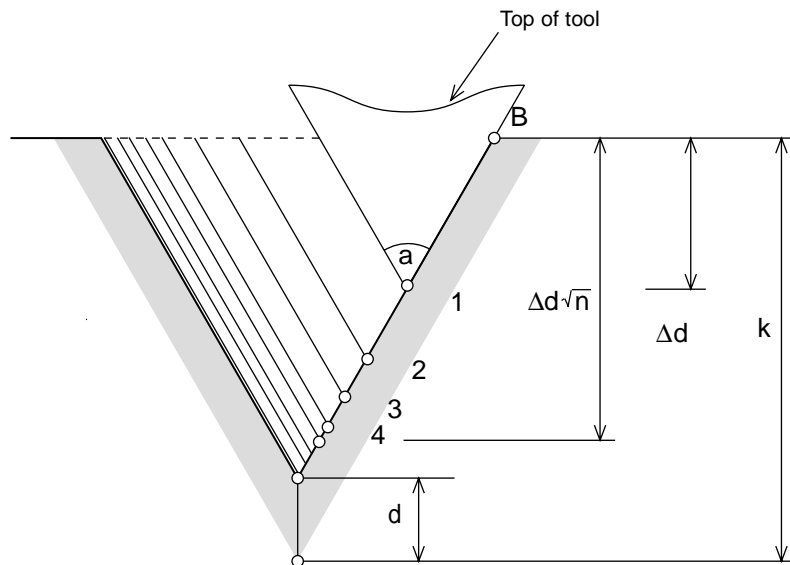


Fig. 18.14 Thread cutting

G76 X(U)\_ Z(W)\_ I(i) K(k) D( $\Delta d$ ) A(a) F(L) ]

m: Final finishing repeated times 1~99

Parameter GUD7\_,ZSFI[24]

r: Thread cutting up (chamfering) amount.

Making the lead L, specify with 2 digit value 00~99 by 0.1 increment within the range of 0.0L~9.9L. Use the parameter GUD7\_,ZSFI[26] to set.

a: Tool nose angle (angle of thread)

$\Delta d_{min}$ : Minimum cutting amount (specify by radius)

If the cutting amount ( $\Delta d_n - \Delta d_n - 1$ ) becomes less than  $\Delta d_{min}$ , it is clamped to  $\Delta d_{min}$ .

Use parameter GUD7\_, ZSFI[27] to set.

d: Finish amount

Use parameter GUD7\_, ZSFI[28] to set.

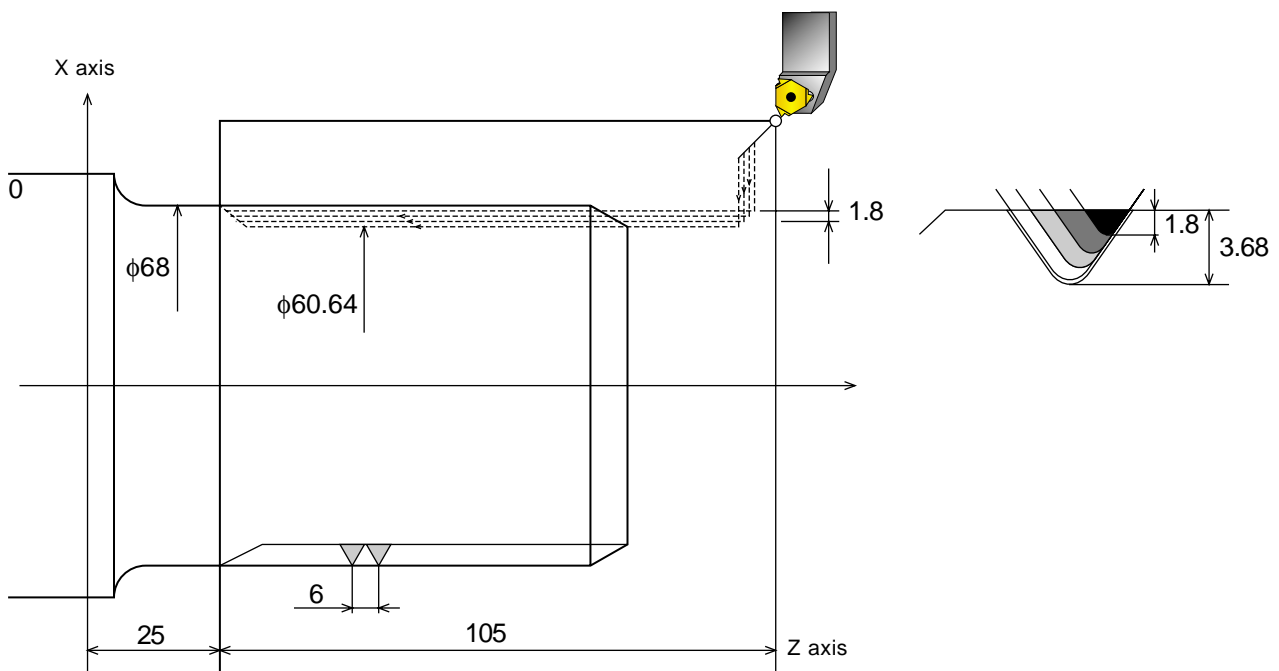
i: Taper amount (radius)

If  $i=0$ , straight thread cutting is executed.

k: Height of thread (specify the distance of X axis direction by radius).

$\Delta d$ : First cutting amount (specify by radius)

L: Lead of thread (the same as G32 thread cutting).



G76 X60640 Z25000 K3680 D1800 A60 F6. ]

Fig. 18.15 Combined type thread cutting cycle(G76)

(8) Cautions Relating to Combined Type Fixed Cycle (G70~G76)

1. It is impossible to command G70, G71, G72 and G73 in the MDI mode. If you happen to command them, it will result in alarm 1401. It is possible to command G74, G75 and G76.
2. In the block which commanded G70, G71, G72 and G73 and the blocks between the sequence numbers specified by P and Q of G70, G71, G72 and G73, M98/M99 cannot be specified.
3. The blocks between the sequence numbers specified by P and Q of G70, G71, G72 and G73 cannot make following commands.
  - One shot G code other than dwell (G04).
  - G code of group 01 other than G00, G01, G02 and G03.
  - G code of group 06.
  - M98/M99.
4. Don't specify the program which is - the last shift command in the finishing shape block group specified by P and Q of G70, G71, G72 and G73 is the chamfering and corner R.
5. Tool nose R compensation is not used in G71~G76.

## 2-3-19 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
G32	Threading
G92	Canned cycle for threading
G76	Multiple repetitive cycle for threading

### Limitation of spindle speed

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

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

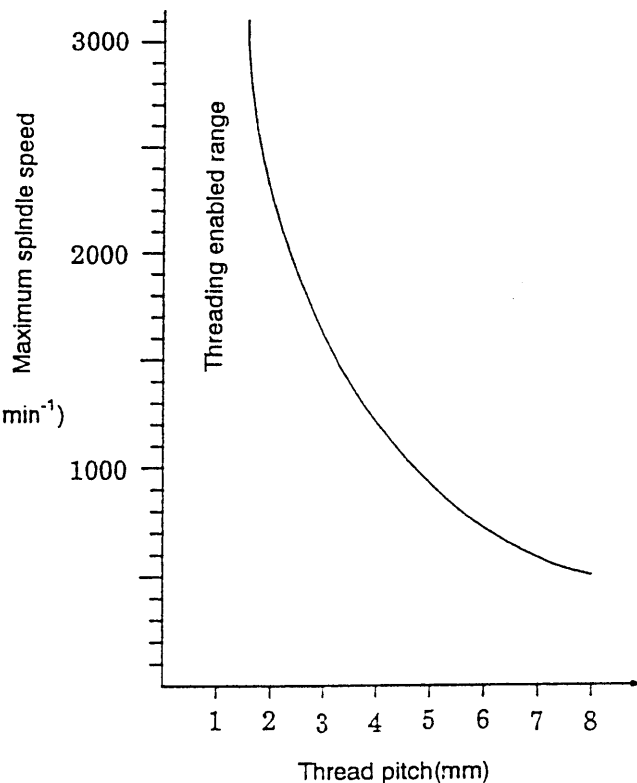
$$N \leq \frac{5000}{P} \quad N : \text{Spindle speed(min}^{-1}\text{)}$$

(Example)

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

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

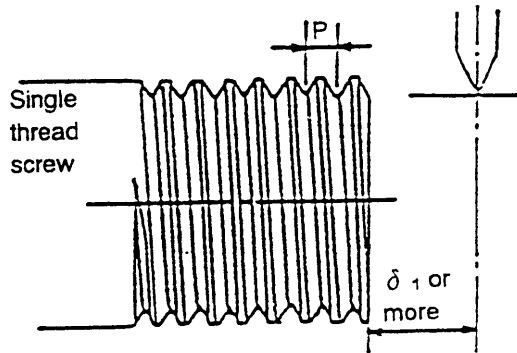
Therefore, when cutting threads with a pitch of 3mm, use a spindle speed of 1666 rpm or less.



- Note)**
1. The above-mentioned limitation does not apply to an oil groove, etc. which do not require an accurate lead.
  2. If the spindle rpm slows down due to insufficient spindle output, the threads may not be cut at accurate pitches even with the above-mentioned spindle rpm limit.

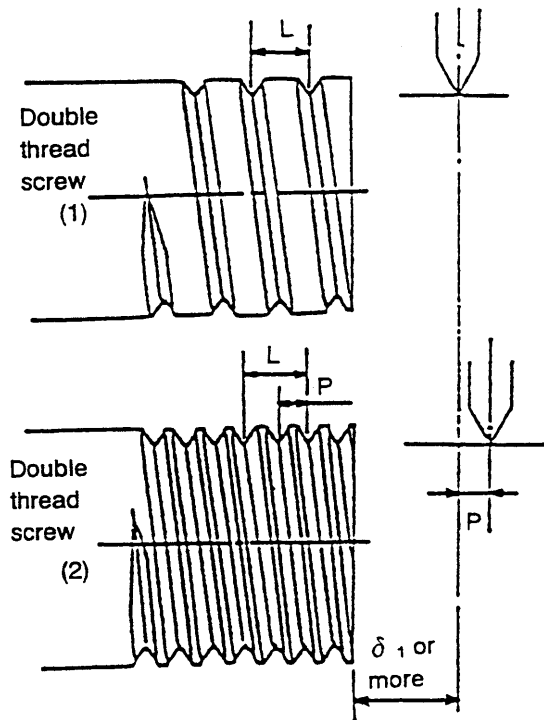


### 1. Cutting the single thread screw



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

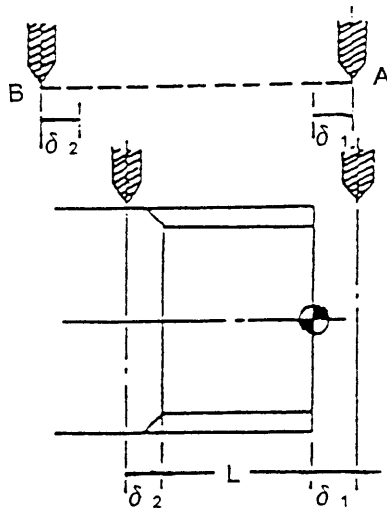
$$l = n \cdot P \quad P = \frac{l}{n} \quad \left( \begin{array}{ll} l: \text{Thread lead} & P: \text{Pitch} \\ n: \text{No. of threads} & \alpha: \text{Twisting angle} \\ & \beta: \text{Lead angle} \end{array} \right)$$

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

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

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

### <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 "L", a threading length of " $L+\delta_1+\delta_2$ " 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:

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

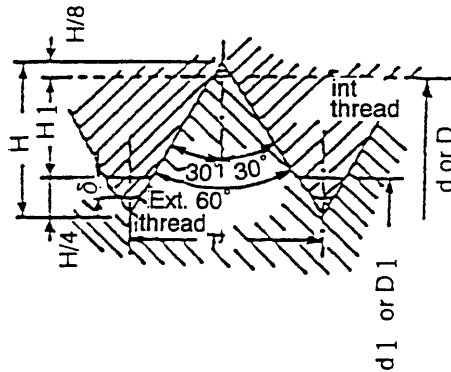
Example) When cutting a JIS Class-1 thread with a pitch of 1.5 at a spindle speed of 800min<sup>-1</sup>

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

**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:

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

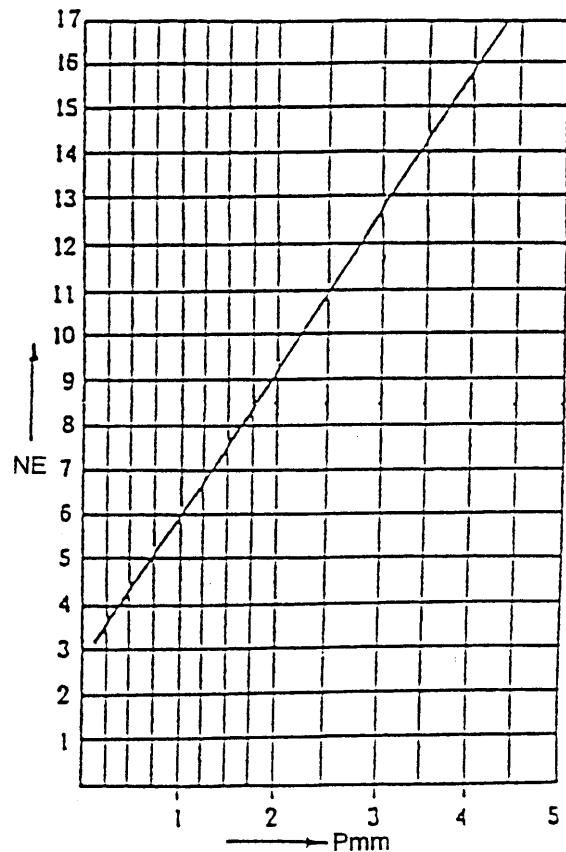
$$= 3.3P + 2.5$$

**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)

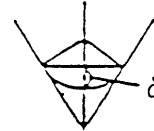
P : Thread pitch to be cut (mm)



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);

$$\left. \begin{array}{l} \delta = \frac{1}{4} H - R = \frac{1}{4} P \cos 30^\circ - R \\ \text{external} \\ \text{thread} \end{array} \right\} \dots\dots\dots (1)$$

$$\left. \begin{array}{l} \delta = \frac{1}{8} H - R = \frac{1}{8} P \cos 30^\circ - R \\ \text{internal} \\ \text{thread} \end{array} \right\}$$



A maximum allowable value for R is;

$$\left. \begin{array}{l} R_{\max} = \frac{P}{4 \times \tan 60^\circ} \\ \text{external} \\ \text{thread} \end{array} \right\}, \left. \begin{array}{l} R_{\max} = \frac{P}{8 \times \tan 60^\circ} \\ \text{internal} \\ \text{thread} \end{array} \right\} \dots\dots\dots (2)$$

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> Depth of Cut	0.661	0.581	0.847	0.717	0.932	0.872	0.127	1.037	1.313	1.193	1.593	1.523	1.973	1.741	2.251	2.074
n	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$	$\Delta X(n)/$ $\Delta W$
*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\ 0.049	1.122\ 0.049	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\ 0.045	1.162\ 0.042	1.654\ 0.042	1.394\ 0.042	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\ 0.046	1.434\ 0.043	1.824\ 0.046	1.704\ 0.043	2.174\ 0.051	1.994\ 0.046	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\ 0.047	1.744\ 0.043	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\ 0.051	2.074\ 0.043	2.586\ 0.051	2.346\ 0.043	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\ 0.055	2.386\ 0.048	3.146\ 0.055	3.006\ 0.048	3.705\ 0.055	3.249\ 0.048	3.898\ 0.057	3.568\ 0.053
11											3.186\ 0.046	3.046\ 0.045	3.866\ 0.046	3.408\ 0.045	4.067\ 0.054	3.742\ 0.050
12													3.906\ 0.052	3.448\ 0.048	4.248\ 0.052	3.908\ 0.048
13													3.946\ 0.050	3.488\ 0.050	4.422\ 0.050	4.068\ 0.046
14															4.462\ 0.050	4.108\ 0.046
15															4.502\ 0.050	4.148\ 0.046

\* 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 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 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 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 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 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 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△△.△△	F1.0	$d-DX(1)=10-0.45=9.55$
G01 or G00		W-0.12		$\Delta W=0.124 \quad 0.12$
G92	X9.12	Z△△.△△		$d-\Delta X(2)=10-0.818=9.122$
G01 or G00		W(+ )0.06		$\Delta W=0.057 \quad 0.06$
G92	X8.92	Z△△.△△		$d-\Delta X(3)=10-1.075=8.925$
G01 or G00		W-0.05		$\Delta W=0.048 \quad 0.05$
G92	X8.76	Z△△.△△		$d-\Delta X(4)=10-1.082=8.758$
	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 ZigZag (Internal Thread)

G	X	Z	F	Remarks
G00	X...	Z...		
G92	X9.25	Z△△.△△	F1.0	$d+\Delta X(1)=8.8+0.45=9.25$
G01 or G00		W-0.09		$\Delta W=0.091 \quad 0.09$
G92	X9.57	Z△△.△△		$d+\Delta X(2)=8.8+0.765=9.565$
G01 or G00		W(+ )0.05		$\Delta W=0.05$
G92	X9.73	Z△△.△△		$d+\Delta X(3)=8.8+0.937=9.737$
G01 or G00		W-0.04		$\Delta W=0.042$
G92	X9.88	Z△△.△△		$d+\Delta X(4)=8.8+1.082=9.882$
	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...		

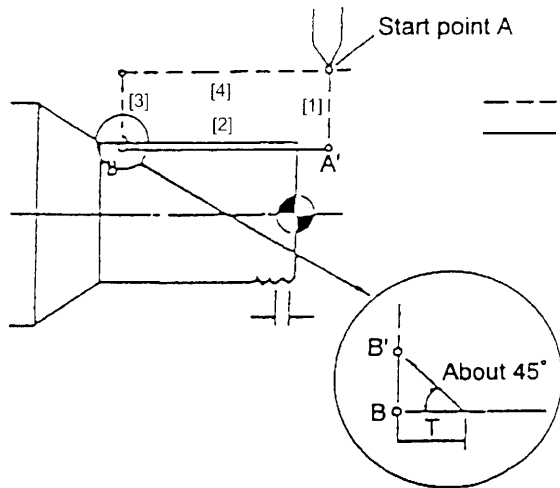
## Thread chamfering

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

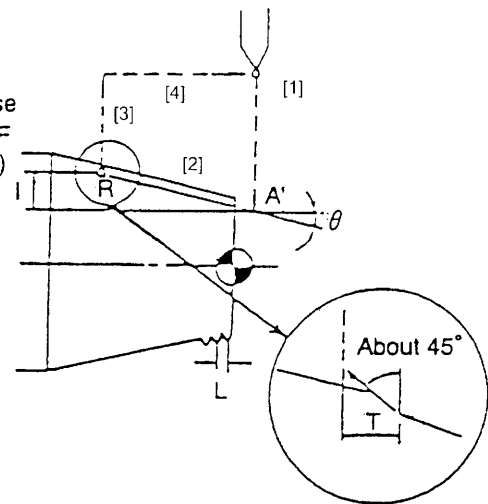
### 1. M functions for chamfering selection

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

M24 ..... chamfering OFF



Details of thread chamfering



Details of thread chamfering

Any value of thread chamfering  $r$  ( $L$  is a lead of thread) of  $0.1L$  increment can be selected by parameter No.1000.

**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. 1000.

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.



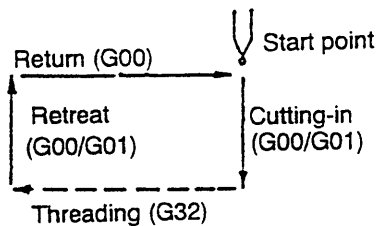
## 1. G32 Threading

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.

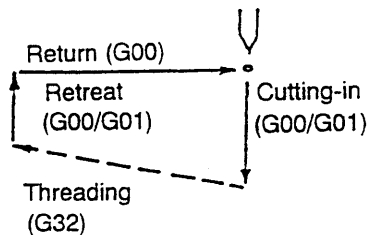
G32 does not allow cycle operation. Therefore, blocks before and after threading require programs for cutting retreat and return.

- Program cutting retreat and return with G00 or G01.

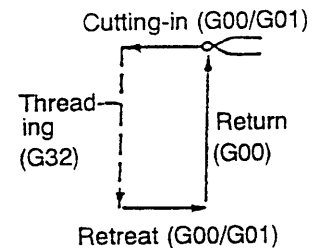
(1) Straight thread



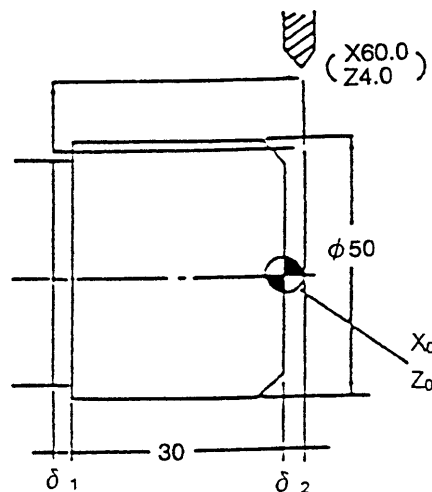
(2) Taper thread



(3) Face thread



(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 Z10.0

N968 G30 U0 W0 M09

N969 M01

• To the start point

• Cutting-in

• Threading

• Retreat

• Return

• Cutting-in

• Threading

• Retreat

• Return

• Cutting-in

• Threading

• Retreat

• Threading

• Retreat

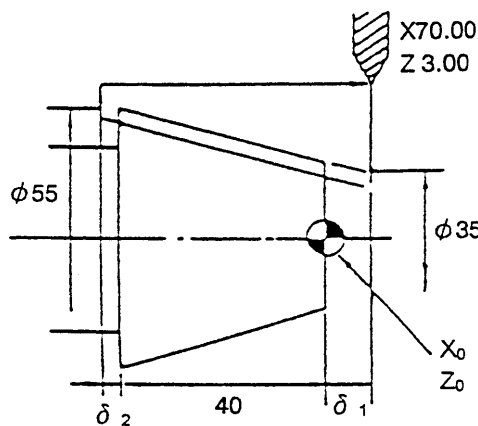
• To the index point

\* 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  
 $\delta_1 = 3\text{mm}$   
 $\delta_2 = 2\text{mm}$

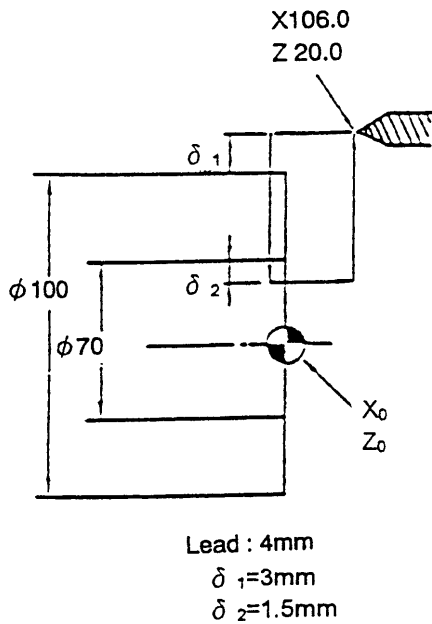
```

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      Z10.0
N943 G30  U0 W0
N944      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  Z20.0
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 G30  U0 W0 M09
N343      M01
    
```

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

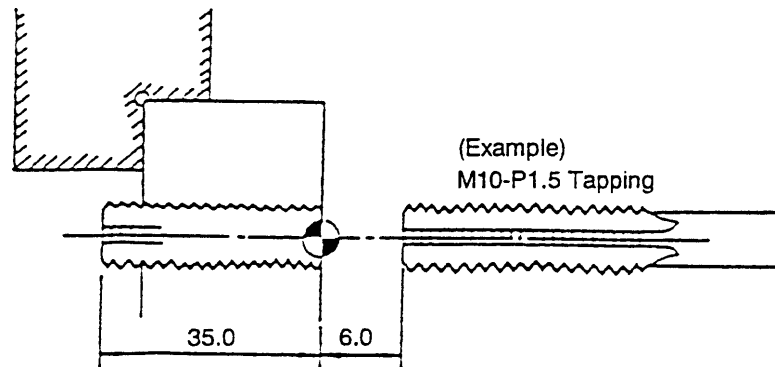
Command values in this case specify strokes from the threading start point to the threading end point.

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 G30 U0 W0 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

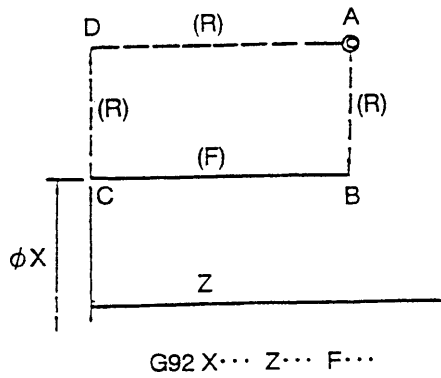
The spindle runs in the reverse direction, and then, the Z axis moves to a position of +10 mm. To retreat the tap, the safer, the bigger a command value for the Z-axis position is.

- When tapping, use a special purpose taper.

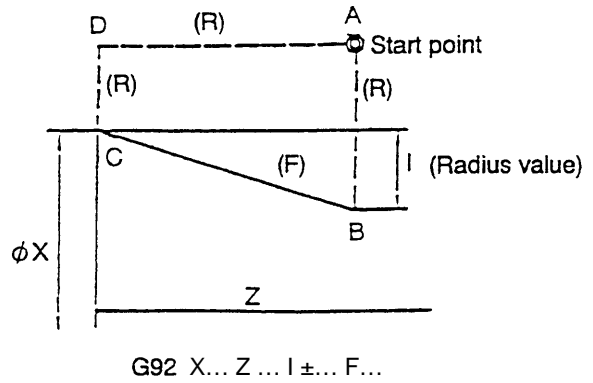
### 3. G92 Threading Cycle

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

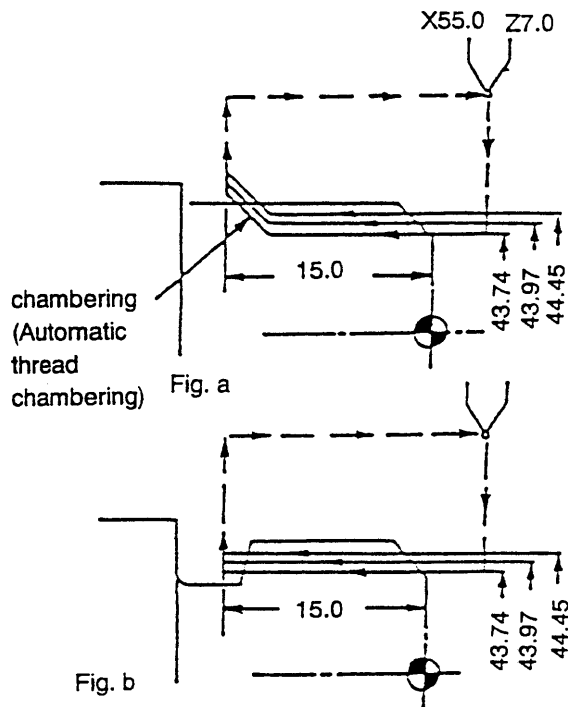


- An incomplete thread part is included within a Z-axis moving range.

R : Rapid traverse  
F : Threading

(1) Example of straight threading

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



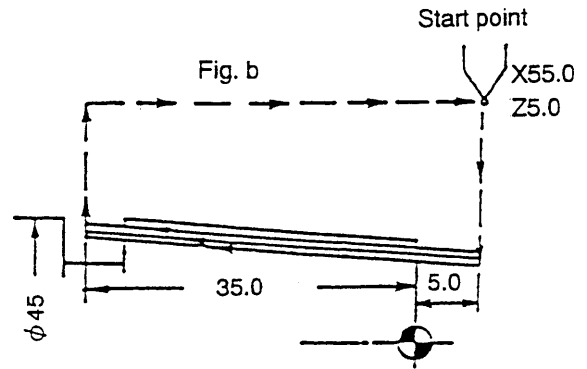
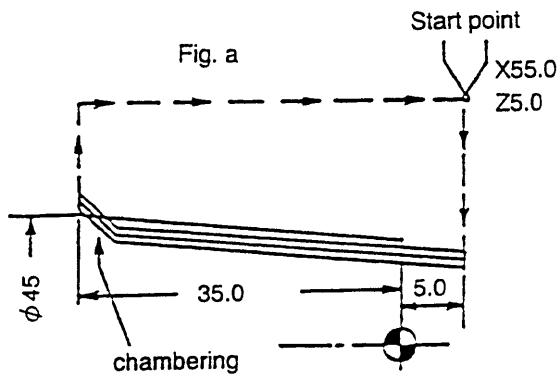
```

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 G30   U0 W0
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 chamfering (automatic thread chamfering) as shown in Fig. a above.
- When chamfering 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

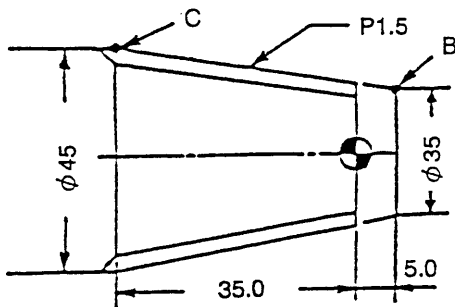


When cutting a taper thread as shown in the left figure, obtain a size of **I** first.

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

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

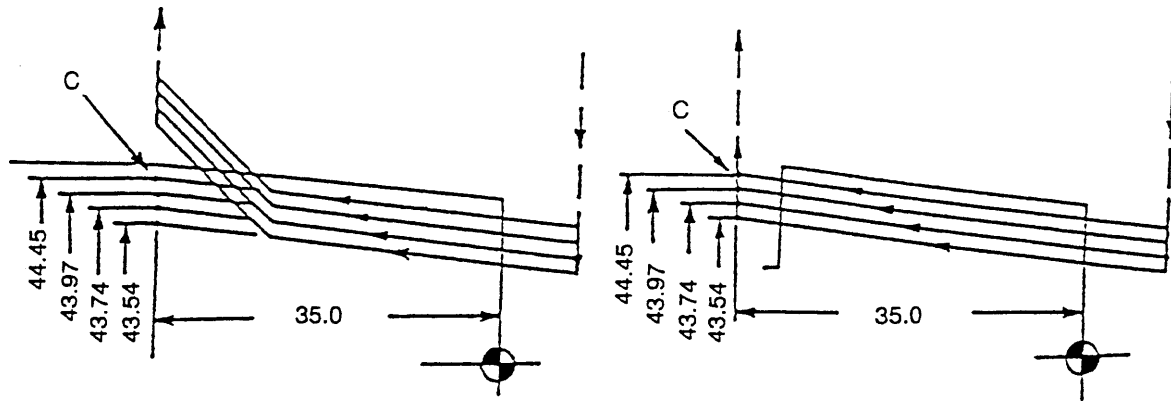
Therefore; **I** = -2.5



```

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 G30  U0 W0
N915      M01
    
```

- Specify the dimension of the point C as to a cutting diameter dimension.
- The program example on a preceding page executes chamfering as shown in Fig. a.
- When chamfering 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 D (1)	<p style="text-align: center;">Q 92 X Z</p>	<p style="text-align: center;">Q 92 X Z I (-)</p>
O D (2)	<p style="text-align: center;">Q 92 X Z</p>	<p style="text-align: center;">Q 92 X Z I (-)</p>
O D (3)		<p style="text-align: center;">Q 92 X Z I</p>
O D (4)		<p style="text-align: center;">Q 92 X Z I</p>

	Straight thread	Taper thread
I D (1)		
I D (2)		
I D (3)		
I D (4)		

**Note)**

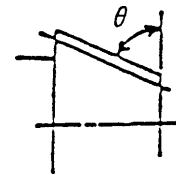
1. A lead becomes inaccurate with a constant surface speed applied.  
Be sure to cut a thread with G97.
2. A cutting feed rate override is always fixed at 100%.
3. If G92 threading cycle is performed in the single block mode, the tool will return to its start point and stop there after completing one 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.)

## 2-3-20 G32 Continuous Thread Cutting

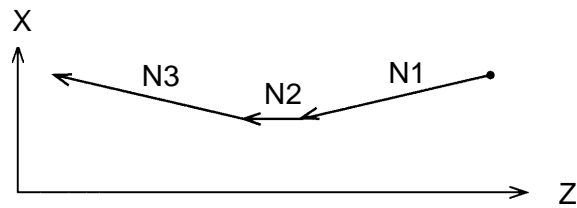
Continuous thread cutting is enabled by continuously specifying the thread cutting command blocks.

### (1) Sample Program

```
N1 G32 U-10.0 W-20.0 F3.0 ]
```

```
N2 W-10.0 ]
```

```
N3 U10.0 W-20.0 ]
```



### (2) Cautions

Stop at single block is not possible during thread cutting.

## 2-3-21 Multi-thread Cutting

If you specify Q together with the thread cutting command (G32), you can shift the thread cutting start angle by the specified shift Q.

If you execute thread cutting of the same shape after changing the Q value, you can execute multi-thread cutting.

### (1) Command Format

G32  $\alpha$ \_\_\_  $\beta$ \_\_\_ Q\_\_\_ F\_\_\_ ;

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

F : Thread lead in the direction of the longitudinal axis

Q : Thread cutting start shift angle 0 to 359999 [Unit: 0.01°]

### (2) Sample Program

G32 W-30.0 F2.0 Q180000; ;

Shift by 180° degrees with respect to the reference point.

## 2-3-22 G34 Variable Lead Thread Cutting

Variable lead threads can be cut by specifying an incremental or decremental amount per revolution of thread in the G34 command block.

### (1) Command Format

G34  $\alpha$ \_\_\_  $\beta$ \_\_\_ F\_\_\_ K\_\_\_ ]

where;  $\alpha, \beta$  : Any one axis

F : Thread lead in the longitudinal direction at thread cutting start time

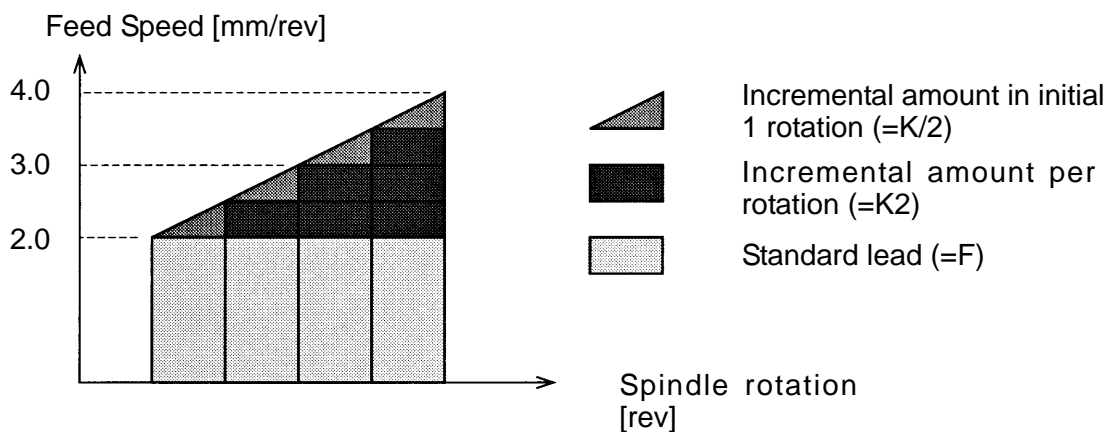
K : Incremental amount (decremental amount for a negative value) per revolution

### (2) Sample Program

Straight thread cutting at variable leads

(Thread lead at start time: 2.0 mm, Incremental amount per rev. : 0.5mm)

G34 W-10.0 F2.0 K0.5 ]



In variable lead thread cutting, feed speed is changed constantly as above so that lead is changed for every rotation.

The lead for 1st rotation in thread cutting becomes " $F \pm (K/2)$ " and the lead for 2nd rotation on is increased/decreased by K for every rotation.

### (3) Cautions

A G34 command with K omitted results in thread cutting at equal leads (G32 command).

## 2-3-23 G150, G151, G152 Groove Width Compensation

When a groove cutting tool is used, programming is done with one of the virtual tool noses (for example, 4) and an offset is input. Also, it is also necessary to offset the other virtual tool nose (for example, 4). At this time, this function performs offsetting the groove width by specifying a G code.

### (1) G Codes

G150 : Groove width compensation cancel

G151 : Groove width compensation ON for end face/back end face (X coordinate system shift)

G152 : Groove width compensation ON for outer/inner diameter (Z coordinate system shift)

### (2) Description of Compensation

#### 1. Compensation ON

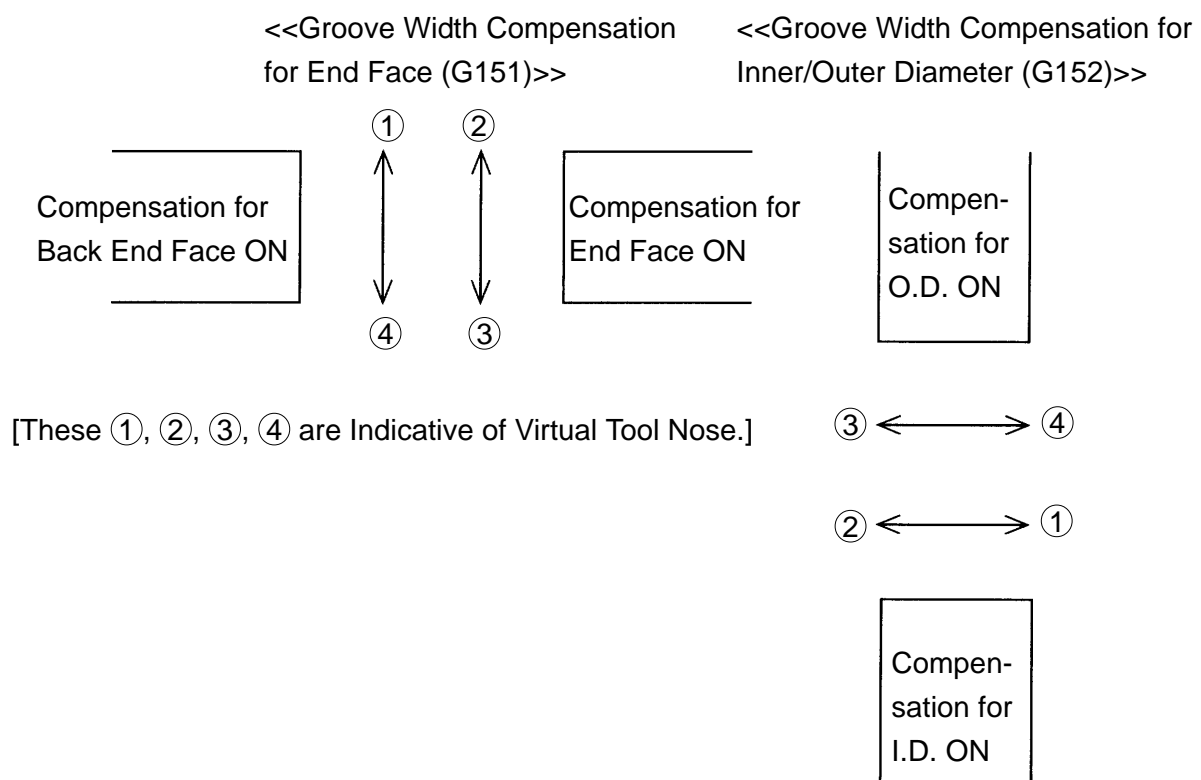
A G151/G152 command reads the tool width (H + J) and virtual tool nose point (T) in the Tool Offset screen, shifts the coordinate system by the tool width, and changes over the virtual tool nose point.

If groove width compensation (G151, G152) is specified, the virtual tool nose point is changed as shown in the figure.

For example, the G151 command changes the virtual tool nose point from 1 to 4, and changes to 1, if it is 4.

#### 2. Compensation cancel

Compensation is cancelled by G150 or a T command. (Both coordinate system and virtual tool nose point are restored to the original ones.)



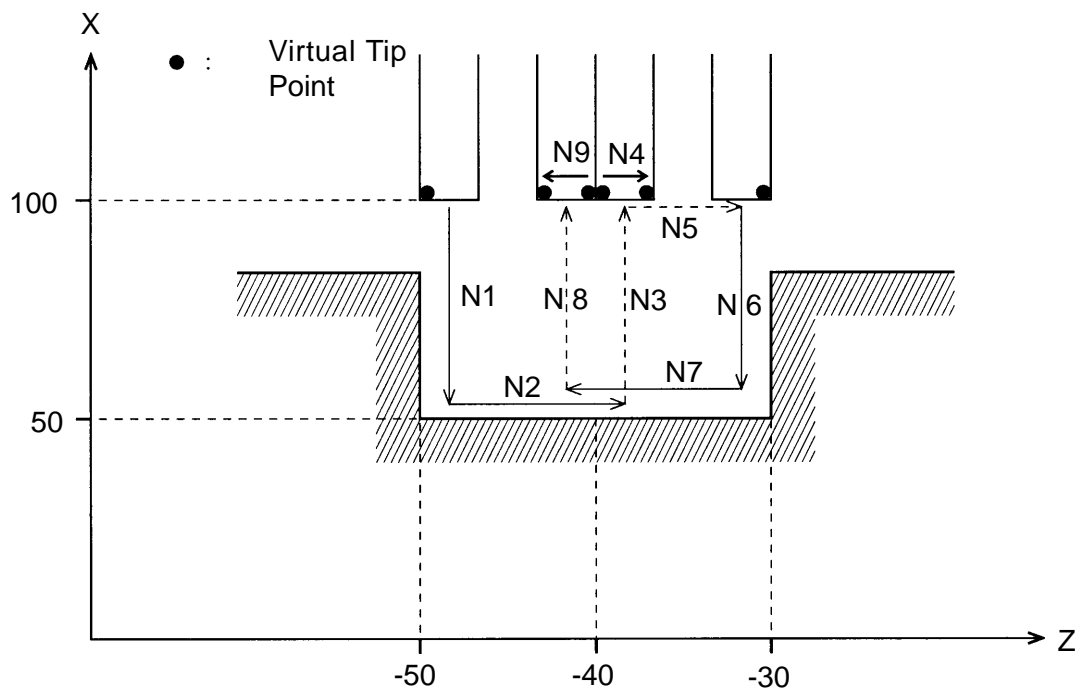
Command	Change in tip point	Change in coordinate system
G151	1 → 4	$X \rightarrow X - (H+J)$
	2 → 3	$X \rightarrow X - (H+J)$
	3 → 2	$X \rightarrow X + (H+J)$
	4 → 1	$X \rightarrow X + (H+J)$
G152	1 → 2	$Z \rightarrow Z - (H+J)$
	2 → 1	$Z \rightarrow Z + (H+J)$
	3 → 4	$Z \rightarrow Z + (H+J)$
	4 → 3	$Z \rightarrow Z - (H+J)$

(3) Sample Program (Virtual Tool Nose Point = 3)

```

G18 G00 X100.0 Z-50.0 ]
N1  G99 G01 X50.0 F0.5 ]
N2  Z-40.0 ]
N3  G00 X100.0 ]
N4  G152 ]           Groove width compensated
N5  G00 Z-30.0 ]
N6  G01 X50.0 ]       Groove width being compensated
N7  Z-40.0 ]
N8  G00 X100.0 ]
N9  G150 ]           Groove width compensated cancel

```





(4) Cautions

1. An alarm results if the virtual tool nose point is other than 1-4.
2. When G151/G152 is specified continuously in the program, cancel current compensation and apply new compensation.
3. A reset during compensation cancels the compensation.

(5) Associated Parameters

(6) Associated Alarms

No.65179 Error in the groove width compensation command.



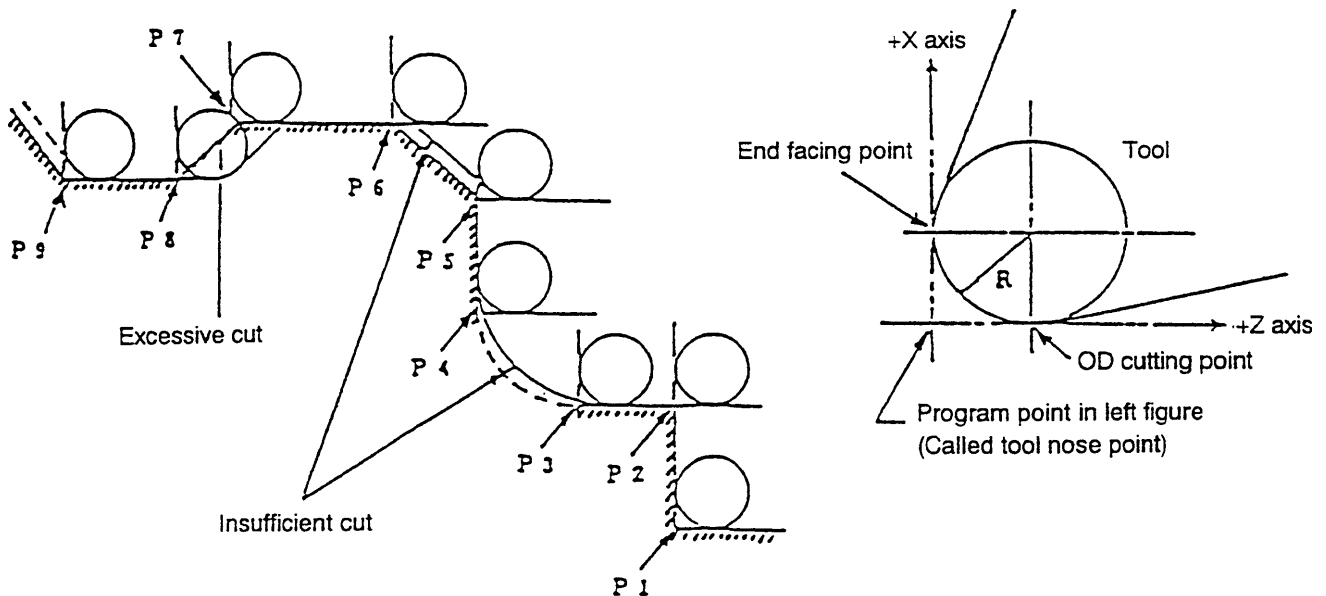
### 3. AUTOMATIC CALCULATING FUNCTION OF TOOL NOSE RADIUS COMPENSATION

#### 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.

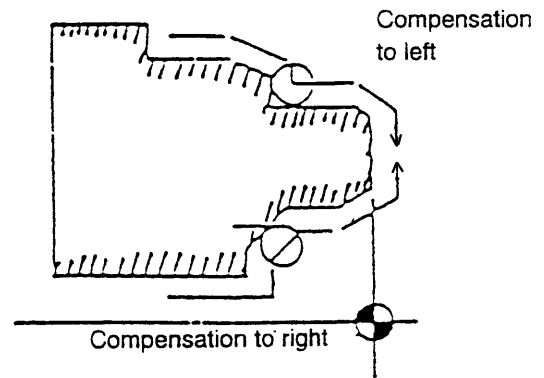
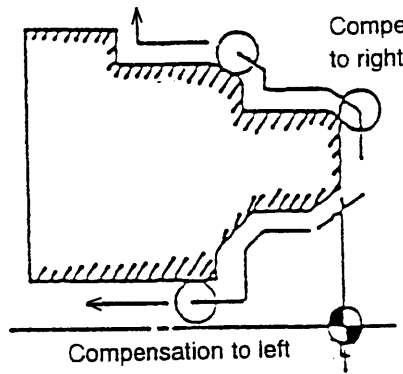
Tool nose radius compensation automatic calculating function makes operation processing done inside the NC unit, automatically controls the tool nose and prevents insufficient and excessive cutting.



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.

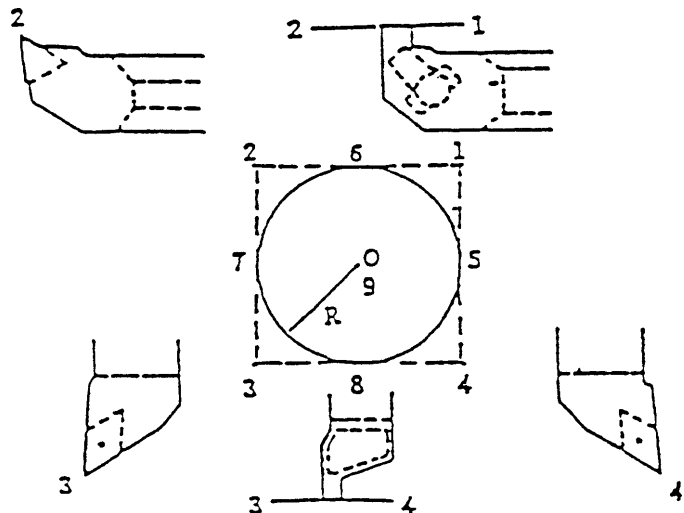
1. Tool tip point (refer to the lower sketch) ... Input at the T\_ of tool offset screen.
2. Size of nose R ..... Input at the R\_ of tool offset screen.

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.

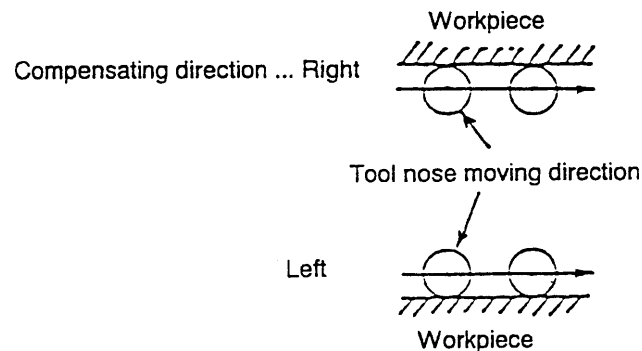
These are divided into three states and each block is called as follows:

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



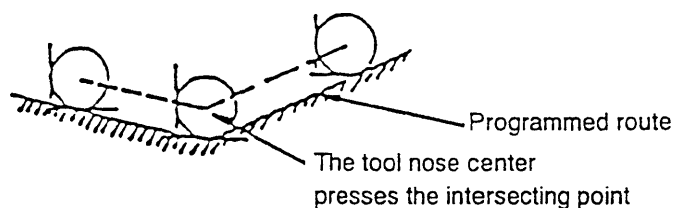
: Follows the compensating direction in a preceding block (because the compensating direction cannot be determined).

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

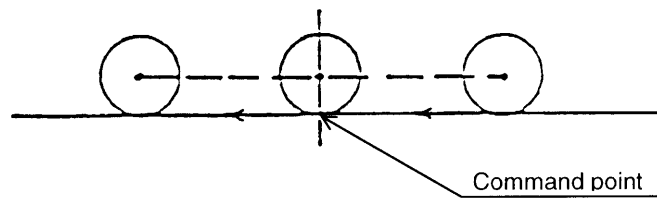
Moving direction Nose point	→	↗	↑	↖	←	↙	↓	↘
1	Right		Left	Left	Left		Right	Right
2	Right	Right	Right		Left	Left	Left	
3	Left		Right	Right	Right		Left	Left
4	Left	Left	Left		Right	Right	Right	
5		Left	Left	Left		Right	Right	Right
6	Right	Right		Left	Left	Left		Right
7		Right	Right	Right		Left	Left	Left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

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

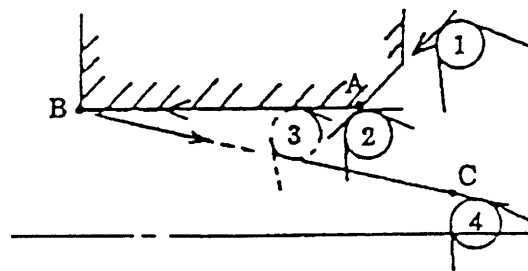
- a) When a tangent angle is  $180^\circ$  or less (inner corner), an intersecting point is operated and the tool nose center moves to that intersecting point.



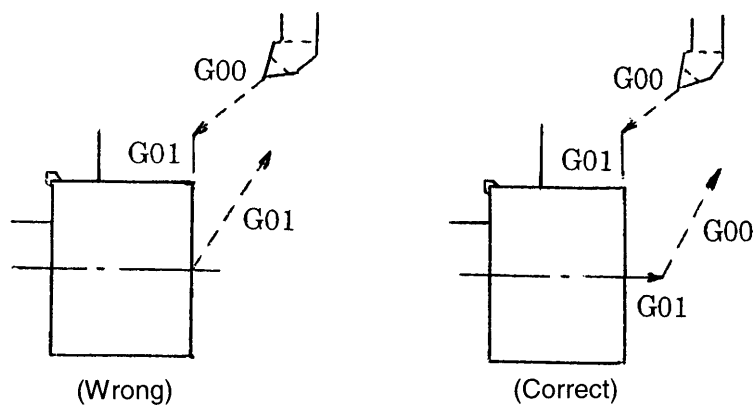
- b) When the tangent angle is  $180^\circ$ , the tool nose center comes on the normal of a command point.



- 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  $\left| \frac{X}{2} \right| > |Z|$ , create a virtual line parallel to the Z axis.

(  $\left| \frac{X}{2} \right| > |Z|$  means the case when a moving axis direction makes an angle larger than  $45^\circ$  with the Z axis.)

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

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

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

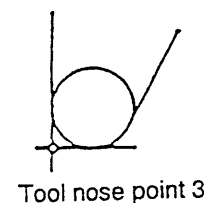
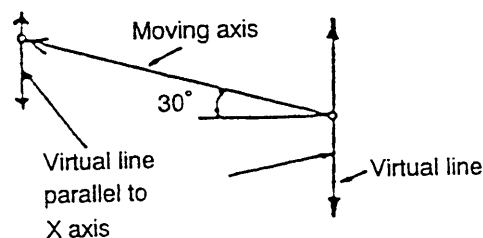
[3] Determine the compensating direction (right or left) to the virtual line.

[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].

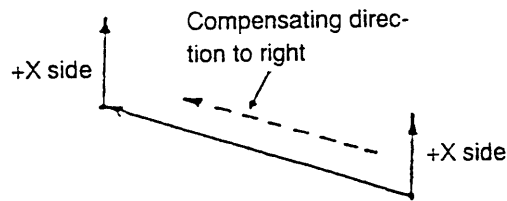
#### **Example 1) For the tool nose point 3**

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

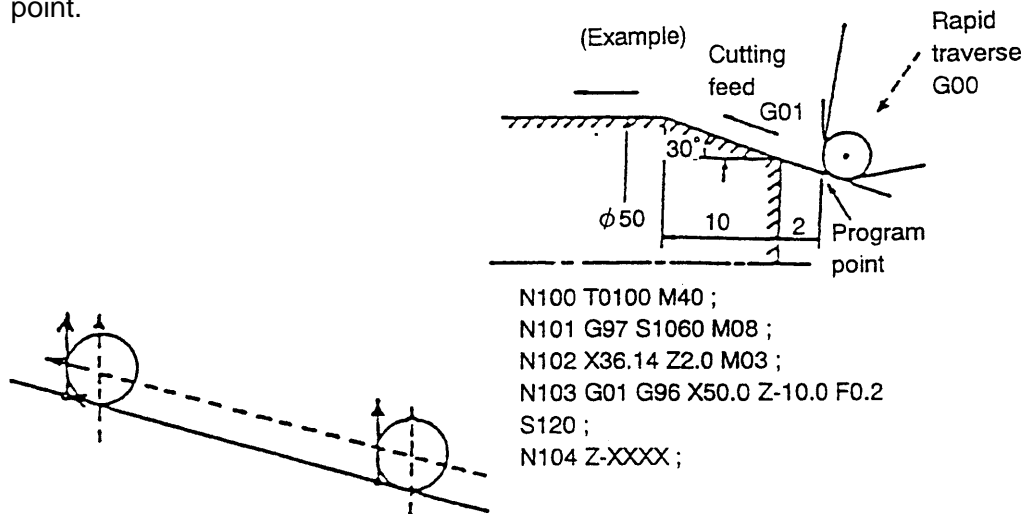




- [2] Determine a virtual line direction in the same direction as the compensating direction of the moving axis (+X side because the compensating direction is to the right).

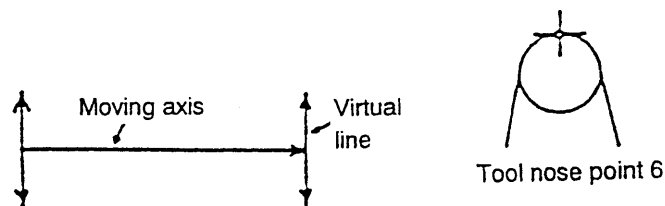


- [3] Determine the compensating direction of the virtual line, and then, the intersecting point.

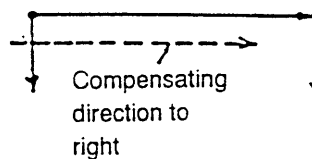


## Example 2) For the tool nose point 6

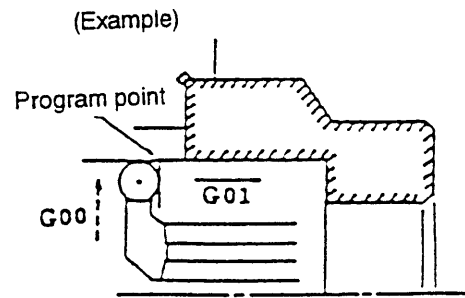
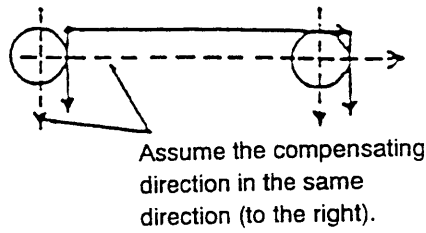
- [1] Since  $\left| \frac{X}{2} \right| < |Z|$ , create the virtual line parallel to the X axis.



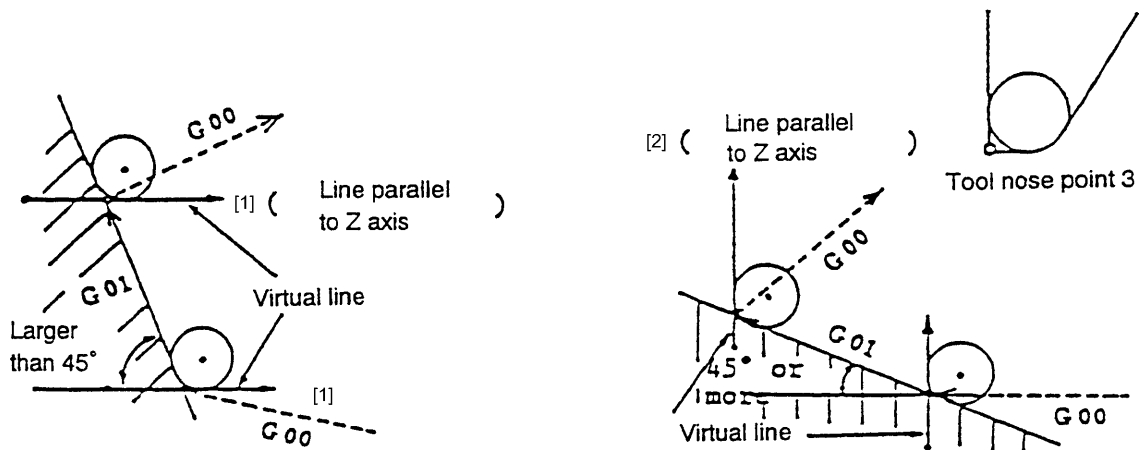
- [2] Determine a virtual line direction to the -X side, because the compensating direction of the moving axis is to the right.



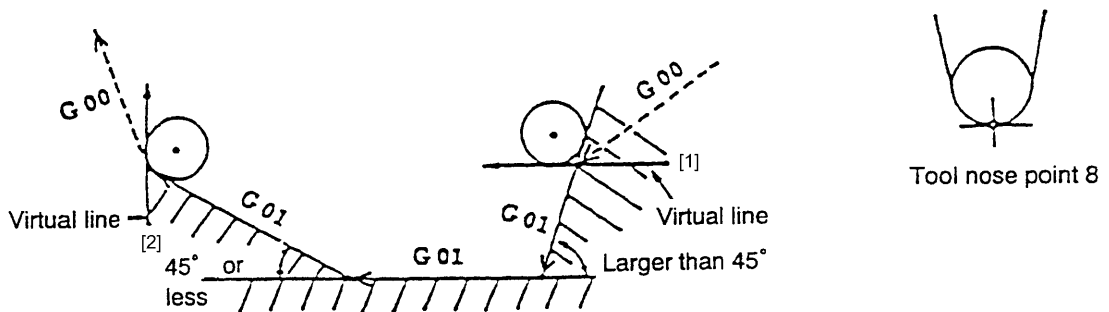
\*[3] Since the compensating direction of the virtual line cannot be determined, assume it in the same direction as the compensating direction of the moving axis.



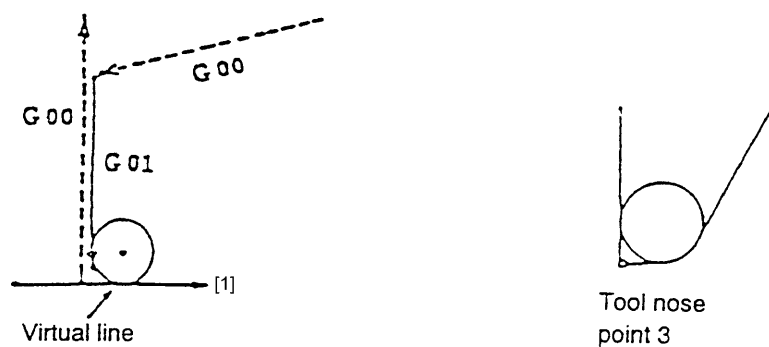
### Example 3) A. For the tool nose point 3



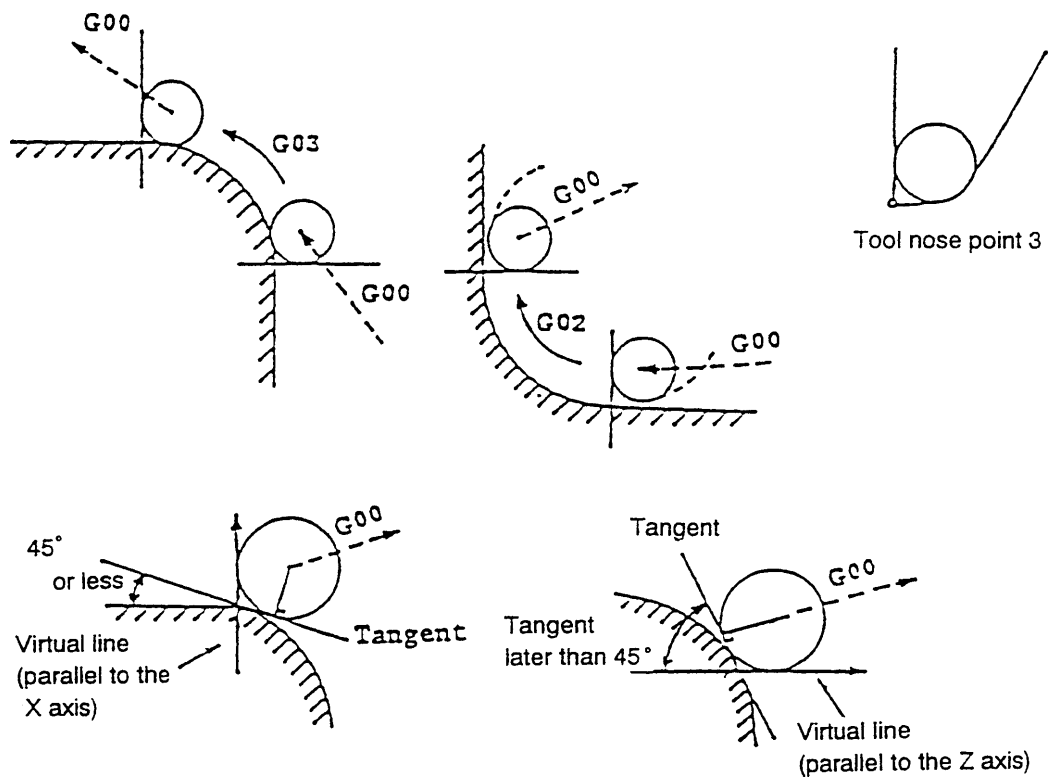
### B. For the tool nose point 8



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



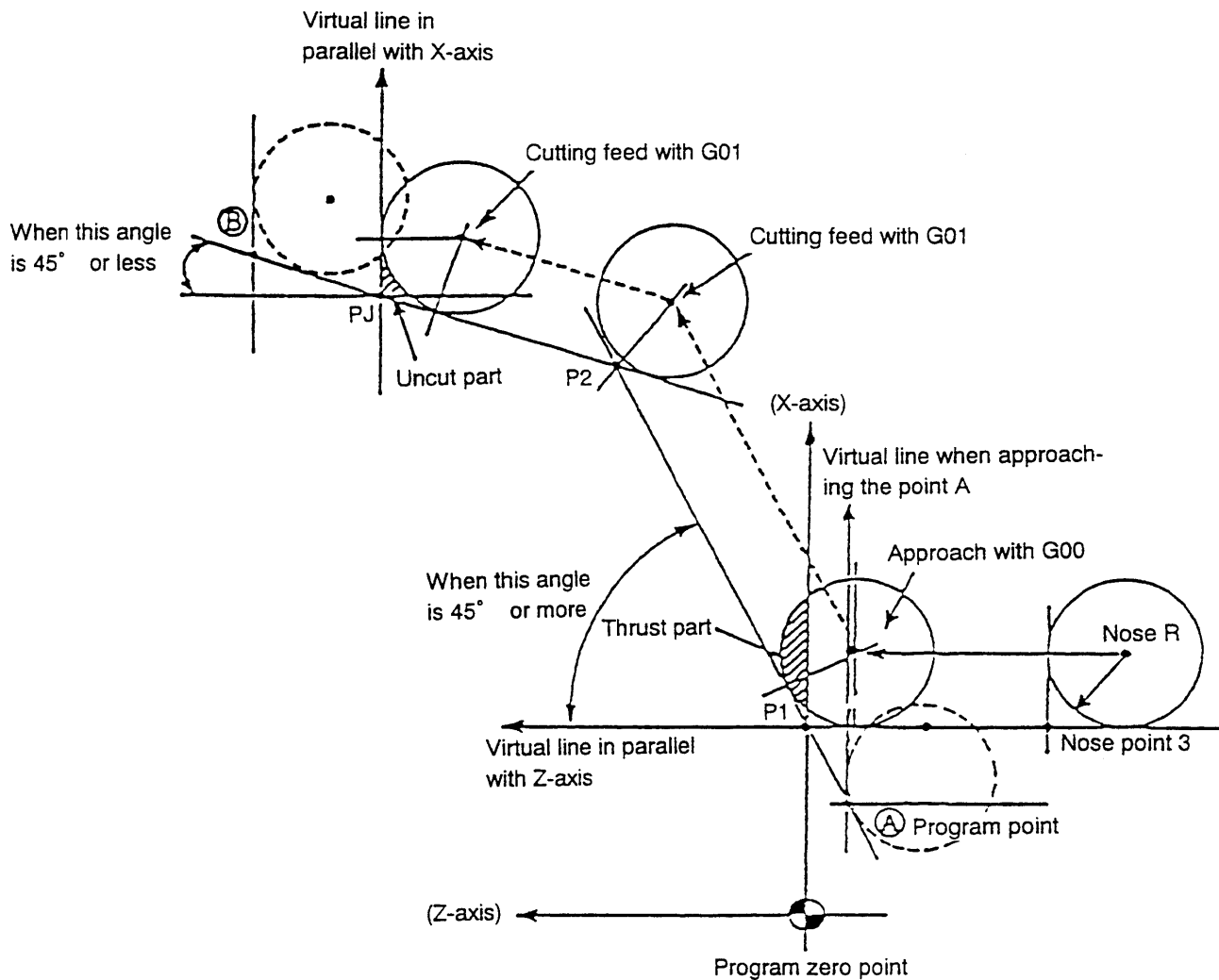
D. For the tool nose point in approaching to an arc and retreating



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)

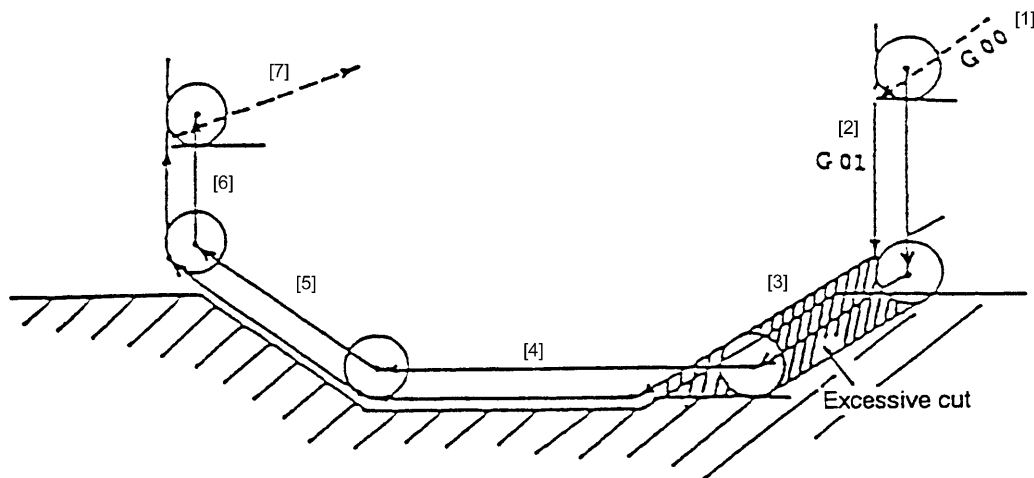
In indenting, there is no particular problem for finishing. In roughing, however, specify a compensation direction with the following G codes:

G141 Tool nose radius compensation direction to left

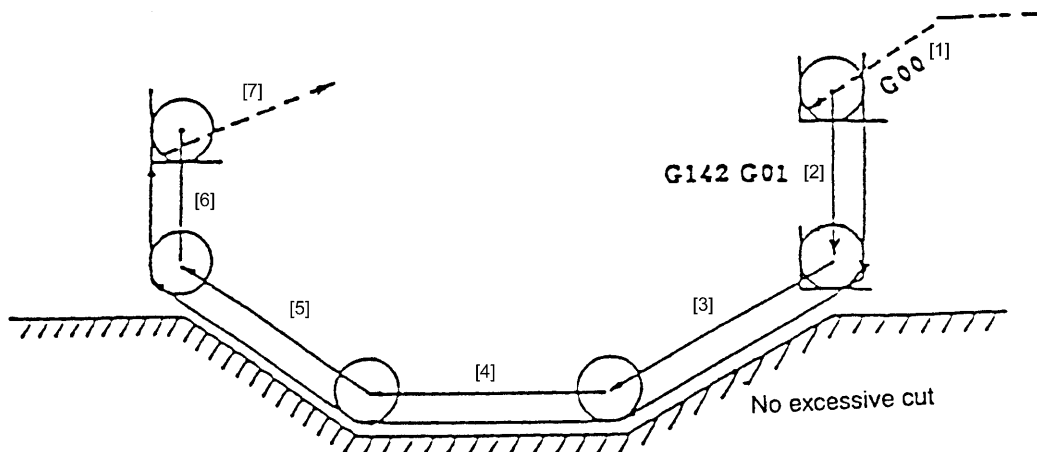
G142 Tool nose radius compensation direction to right

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_
```

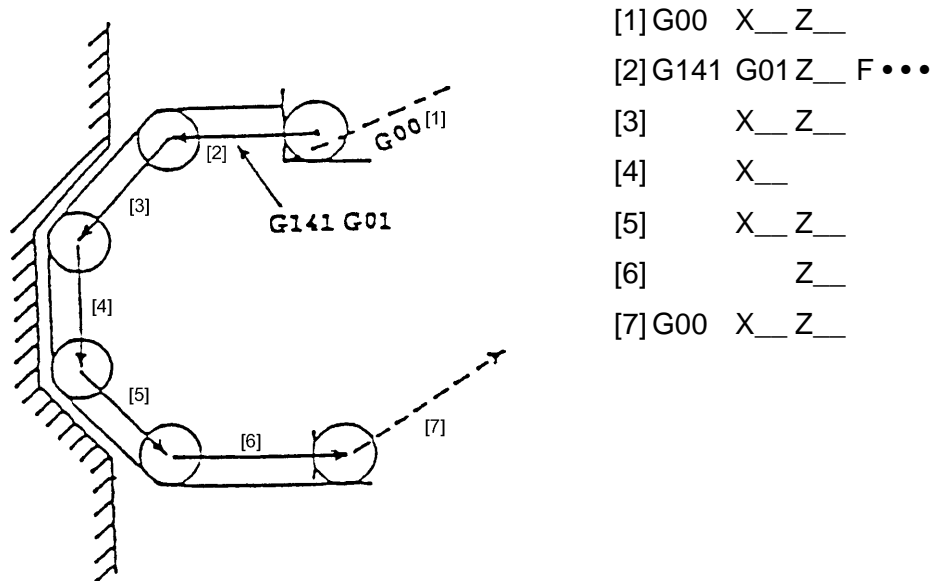
##### Example 2

```
[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)



### Overall Precautions

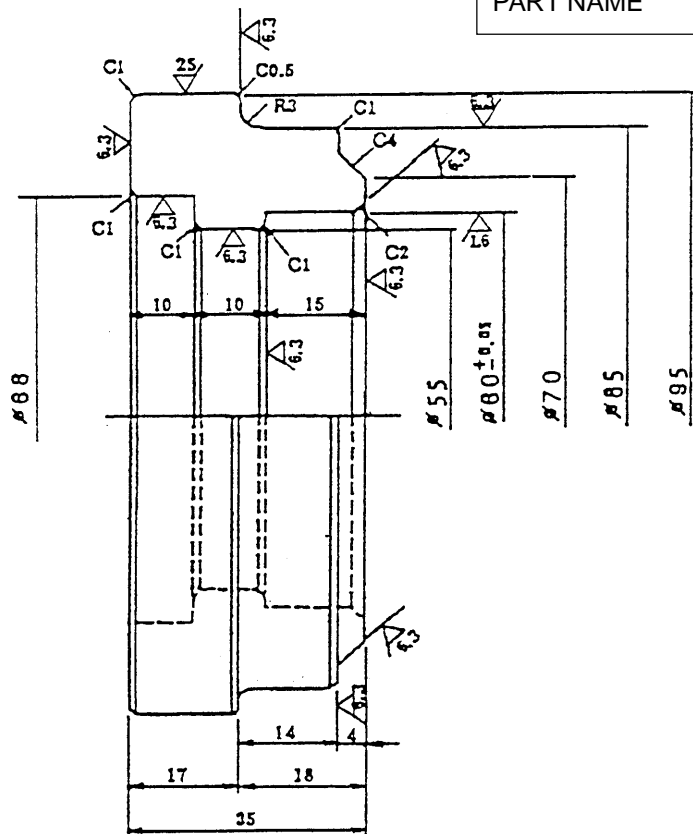
1. An error will result if you specify a tool move inside an arc smaller than a tool nose radius or a groove width up to 2 times or less of the tool nose radius while executing automatic tool nose R compensation.
2. Tool nose R compensation is not performed by data input operation.  
It is available 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			CNC LATHE
PROCESS : 2ND	NC UNIT		TOOL LAYOUT SHEET
		PART NAME	
C1	25/		MATERIAL S48C

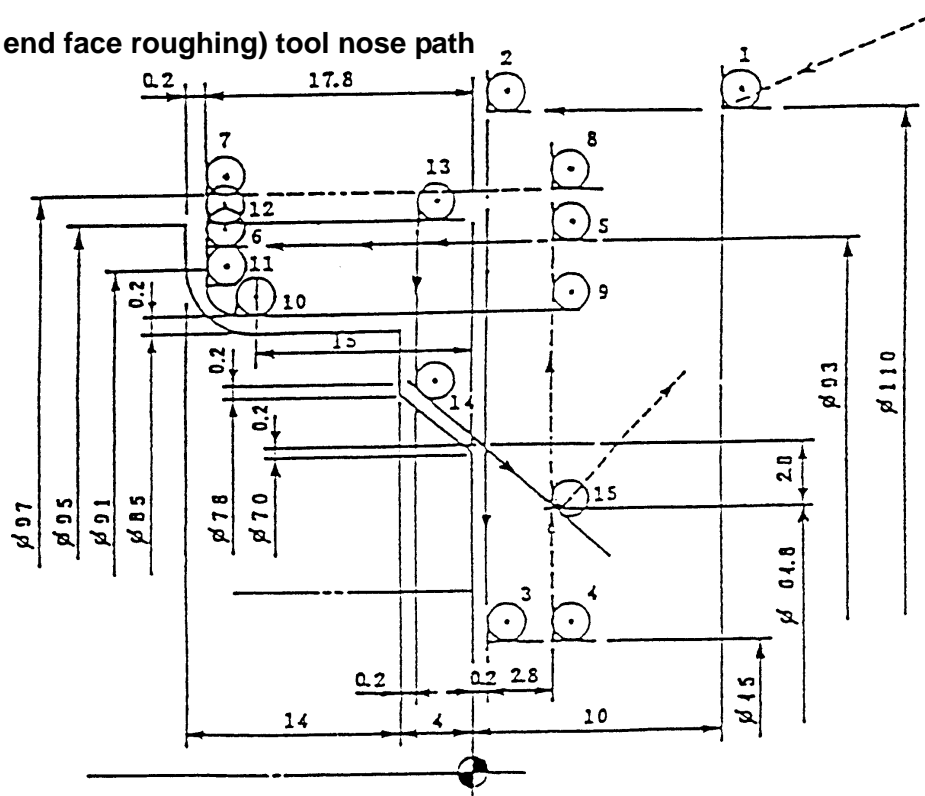


T1	T3	T5	T7	T9
T2	T4	T6	T8	T10

## 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 G97 Z10.0	
N119 G30 U0 W0	
N120 M01	

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

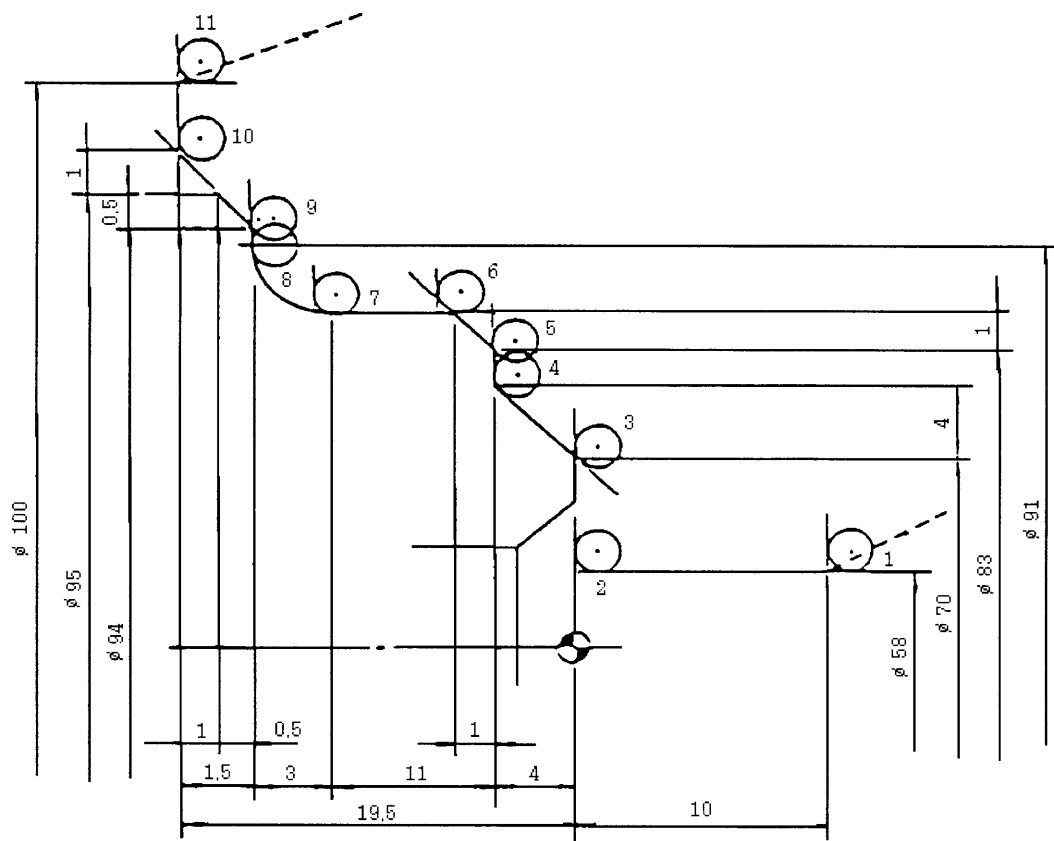






N701 T0700  
 N702 G97 S1100 M08  
 N703 G00 X58.0 Z10.0 M03  
 N704 G01 G96 Z0 F1.5 S200  
 N705 X70.0 F0.2  
 N706 X78.0 Z-4.0  
 N707 X83.0  
 N708 X85.0 Z-5.0  
 N709 Z-15.0  
 N710 G02 X91.0 Z-18.0 R3.0 F0.15  
 N711 G01 X94.0  
 N712 X97.0 Z-19.5  
 N713 X100.0  
 N714 G00 G97 Z10.0  
 N715 G30 U0 W0  
 N716 M01

### 3. T07 (OD end face finishing) tool nose route



```

N801 T0800
N802 G97 S1000 M08
N803 G00 X70.0 Z10.0 M03
N804 G01 G96 Z3.0 F1.5 S200
N805 X60.0 Z-2.0 F0.2
N806 Z-15.0 F0.15
N807 X57.0 F0.2
N808 X55.0 Z-16.0
N809 Z-27.0
N810 X53.0
N811 G00 G97 Z10.0 M09
N812 G30 U0 W0 M05
N813 M01

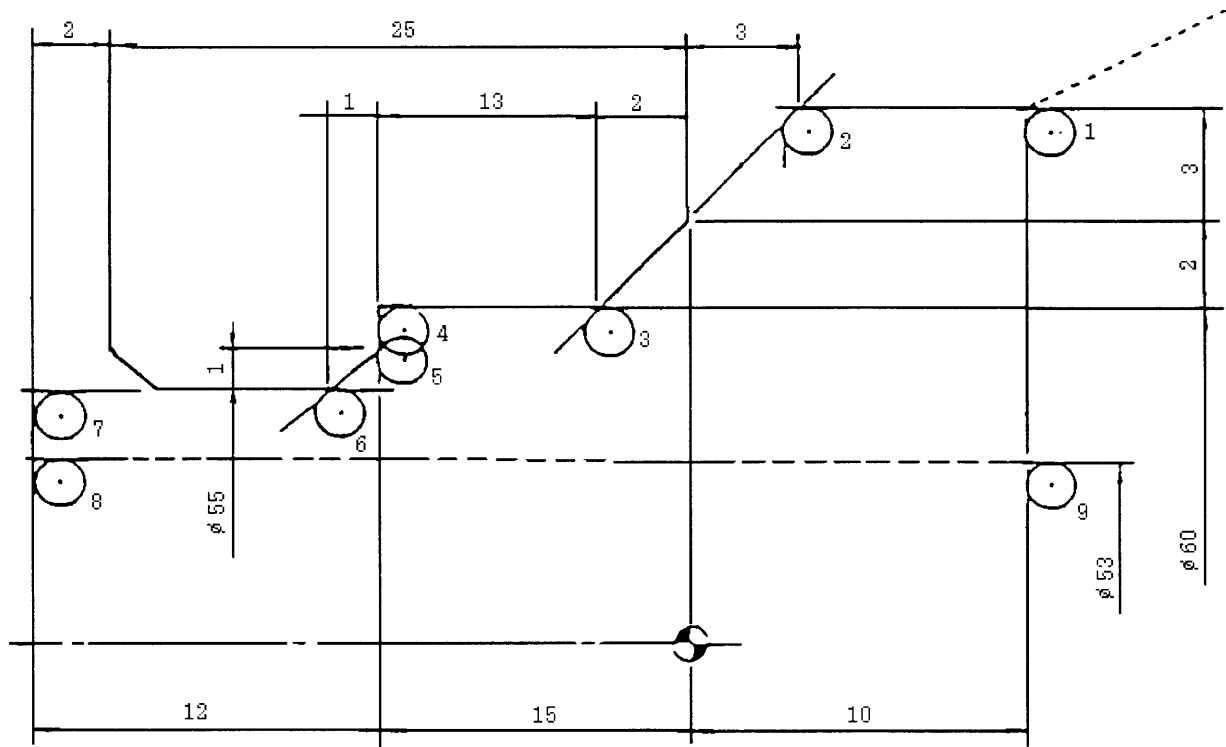
```

```

N6  G28 U0 W0 T0100
N7  M30
%
```

Automatic reference point return (X and Z axes)  
Program end & rewind  
Be sure insert a stop code

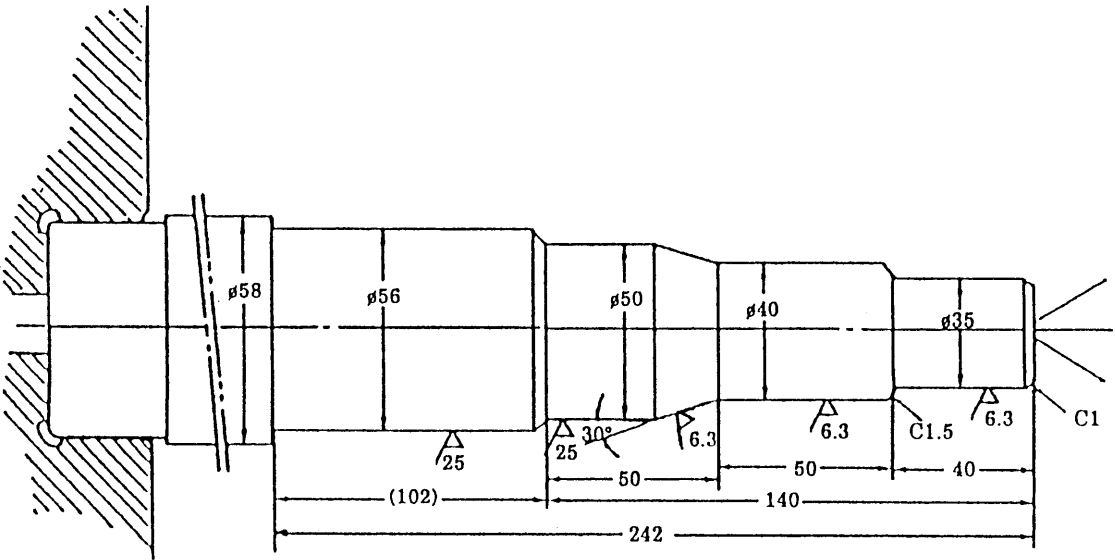
#### 4. T08 (ID finishing) tool nose route

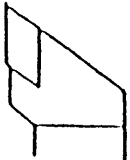
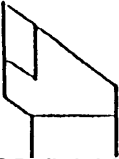


# 4-2 Center Work

## 4-2-1 Machining Drawing

CENTER WORK EXAMPLE	<div>SEIKI</div> <div>CNC LATHE</div> <div>TOOL LAYOUT SHEET</div>	PART NAME : SHAFT	
PROCESS : 1ST NC UNIT			MATERIAL S48C



T1	T3	T5	T7	T9
	 OD roughing		 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 G30 U0 W0	
N5 M01	
OD roughing	
N301 T0300	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 G97 X50.0	Canceling the constant surface speed
N329	G30 U0	
N330	G30 W0	
N331	M01	
OD finishing		
N701	T0700	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	G97 X 65.0	Canceling the constant surface speed
N718	G30 U0	
N719	G30 W0	
N720	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

PROCESS :

BAR WORK EXAMPLE

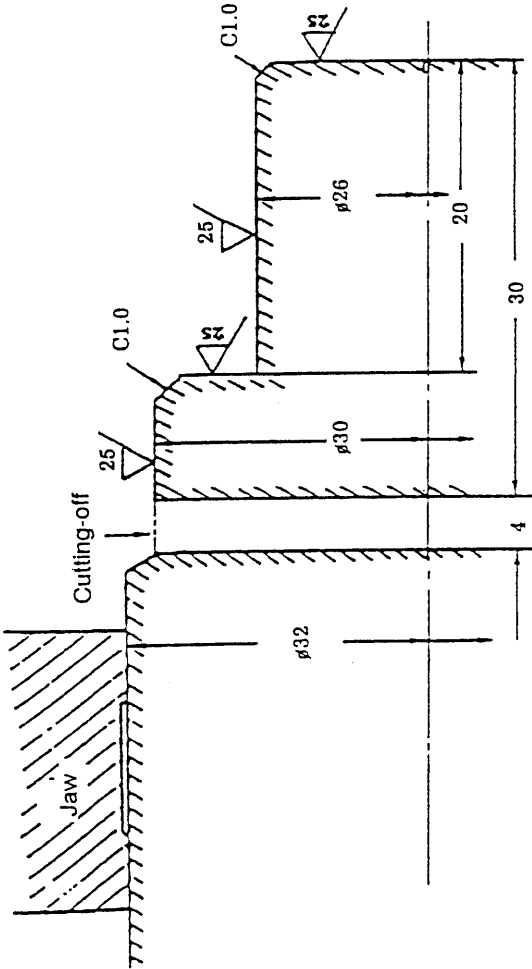


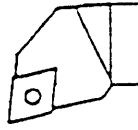
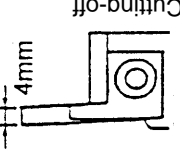
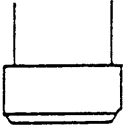
CNC LATHE  
TOOL LAYOUT SHEET

MATERIAL S48C-D

Program example for HITATI SEIKI BS65 bar feeder.

In the other maker's bar feeder, program will be different.



T1	T3	T5	T7	T9
 OD end facing R0.8				 Cutting-off 4mm
T2	T4	T6	T8	T10
				 Stopper

#### 4-3-2 Bar Work Program

program example of BS65 type.

O005	
N1 G28 U0	
N2 G28 W0 T1000	
N3 G50 S2000	
N4 G30 U0 W0	
N5 M01	
Material sizing	
N1001 T1000	Selecting the turret face No.10
N1002 G97 S200	
N1003 G00 X0 Z10.0 M03	
N1004 G01 Z-33.0 F5.0	Stopper approach
N1005 M69	Chuck open
N1006 G04 U2.0	Dwell 2 seconds (chuck opening time)
N1007 G01 Z1.0 F5.0	Material loading and sizing
N1008 M68	Chuck close
N1009 G04 U3.0	Dwell 3 seconds (chuck closing time)
N1010 G00 Z10.0	Retreat
N1011 G30 U0 W0	Retract to index position
N1012 M31	No-workpiece check
N1013 G04 U0.5	Dwell 0.5 second
N1014 M01	
OD cutting	
N101 T0100	Selecting the turret face No.1
N102 G97 S1005 M08	
N103 G00 X38.0 Z10.0 M03	
N104 G96 Z0 S120	Constant surface speed V 120 m/min
N105 G01 Z-1.6 F0.2	End facing
N106 Z3.0	
N107 G00 X18.4	
N108 G01 X26.4 Z-1.0 F0.3	OD cutting
N109 Z-19.8	
N110 X28.4	
N111 X30.4 Z-20.8	



N112	Z-36.0	OD finishing
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	G00 G97 Z10.0	
N124	G30 U0 W0	
N125	M01	
Cutting-off		
N901	T0900	Selecting the turret face No.9
N902	G97 S795 M08	
N903	M63	Unloader advance
N904	G04 U1.0	Dwell 1 second (unloader operating time)
N905	G00 X45.0 Z-25.0 M03	Positioning
N906	G01 X40.0 Z-34.0 F2.0	Approach to a cutting-off position
N907	G96 X-0.5 F0.1 S100	Cutting-off
N908	X40.0 F1.0 M09	Retreating a cutting-off tool
N909	G00 G97 Z10.0	Canceling the constant surface speed
N910	G30 U0 W0	
N911	M01	
N10	M64	Returning the unloader
N11	G04 U0.5	Unloader operating time
N12	M12	Work count
/N13	P1001 M99	Return to N1001 and remachining start
N14	G28 U0 W0	Block skip and reference point return
N15	M30	Block skip end program & rewind
%		

## 4-4 Grooving

### 4-4-1 OD Grooving

Programming	Description
N501 T0500	No.5 turret face calling
N502 G97 S360 M08	
N503 G150	
N504 G00 X87.0 Z10.0 M03	
N505 G01 G96 Z-12.0 F5.0 S100	
A→B N506 X75.2 F0.1	Groove width offset OFF  Tool offset <div style="border: 1px solid black; padding: 5px; display: inline-block;">             05 X____               Z____              R               0.2               T 3              H               3.0           </div>
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	
H→I N514 Z-6.0	
I→J N515 X83.0 Z-8.0 F0.1	
J→K N516 X75.0	Groove width offset ON. Change to a program point “b”
K→L N517 Z-8.1	
L→M N518 X87.0 F5.0	
N519 G150	
N520 G00 G97 Z10.0	
N521 G30 U0 W0	
N522 M01	
	Groove width offset OFF



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

N621 G00 G97 Z10.0

N622 G30 U0 W0

N623 M01

Groove width offset OFF

(a) Groove width offset OFF state

Tool nose point : 2

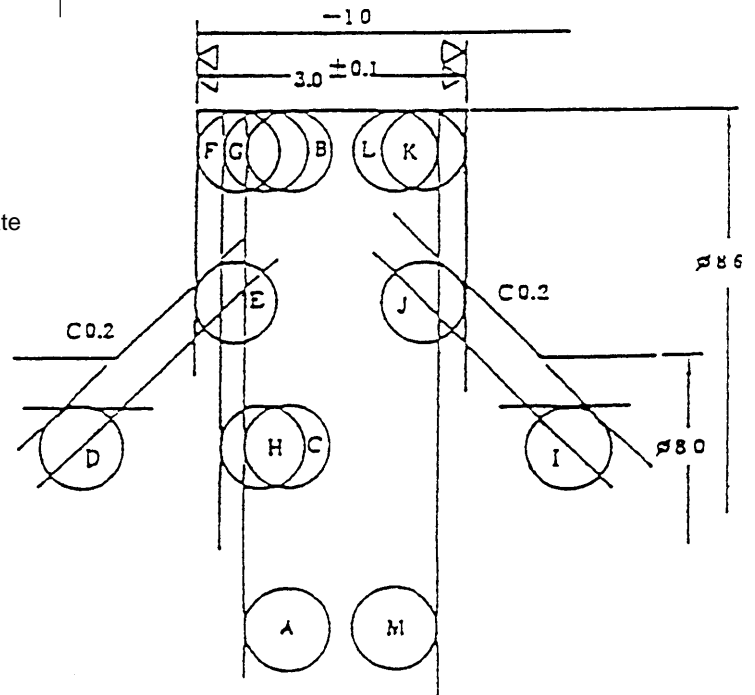
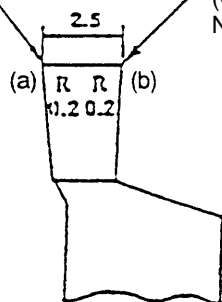
Nose R : 0.2

Nose width : Non

(b) Groove width offset ON state

(G152)

Nose width : 2.5

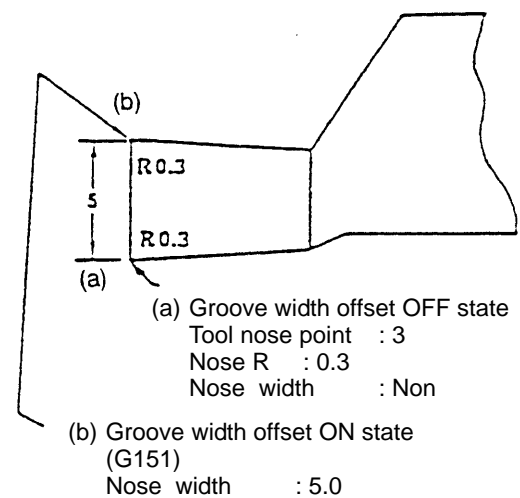
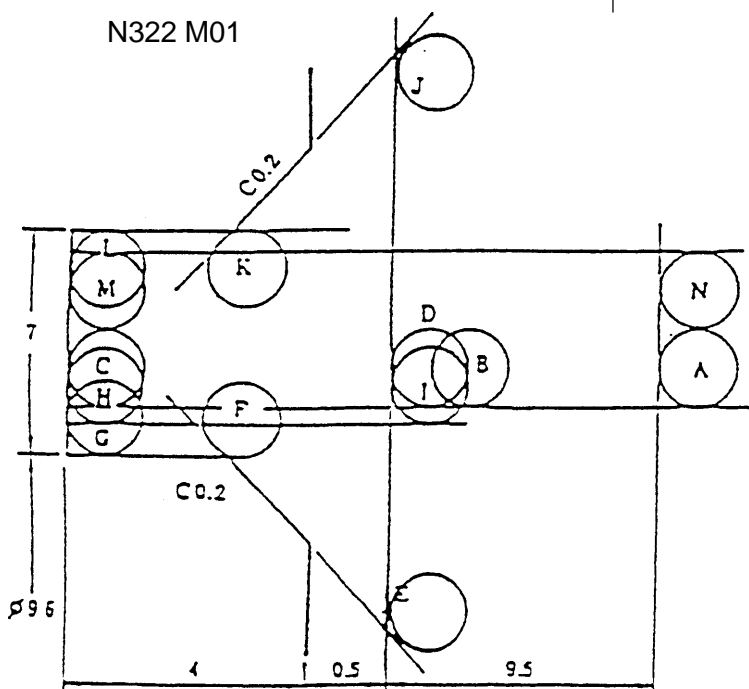


#### 4-4-3 End Face Grooving

Programming	Description
N301 T0300	No.3 turret face calling
N302 G97 S330 M08	
N303 G150	Groove width offset OFF
	Change to a program point "b"
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 G97 Z10.0	
N321 G30 U0 W0	
N322 M01	

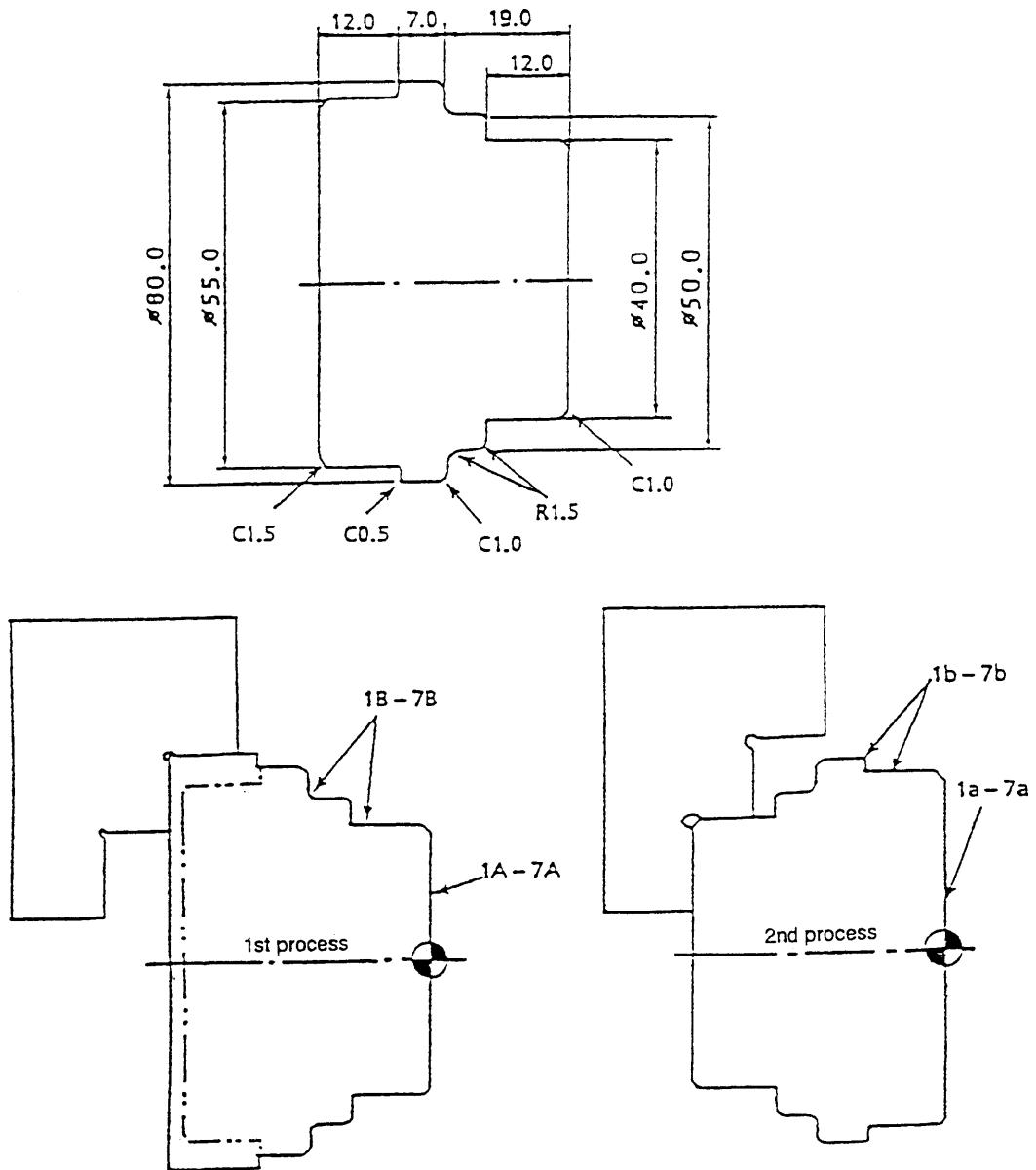
##### Tool offset

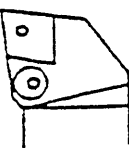
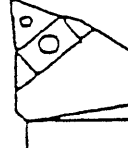
03	X	_____
	Z	_____
	R	0.2
	T	3
	H	5.0



## 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  OD roughing			R0.8  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 G30 U0 W0

N6 M01

N100 (OD-R)

N101 T0100

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

N702 G97 S... M08

N703 G00 X... Z...M03

N704 ..

N...

N... G00 G97 Z10.0

N... G30 U0 W0

N... M01

N8 G28 U0 W0

N9 M00 ← Turning over the workpiece.

(2nd process)

N11 G28 U0 W0  \* (Difference from the finishing end face of 1st process.)

N12 G54 Z-12.0

N13 G50 S1800

N14 G30 U0 W0

N15 M01

N5100 (OD-R)

N5101 T0100

N5102 G97 S545

N5103 G00 X... Z...M03

N... M08

N...

N... G30 U0 W0

N... M01

N5700 (OD-F)

N5701 T0700

N5702 G97 S...M08

N5703 X... Z...M03

N5704 ..

N...

N... G00 G97 Z10.0

N... G30 U0 W0

N... M01

N20 G54 Z0  Reference point shift cancel.

N21 G28 U0 W0

N22 M30

%

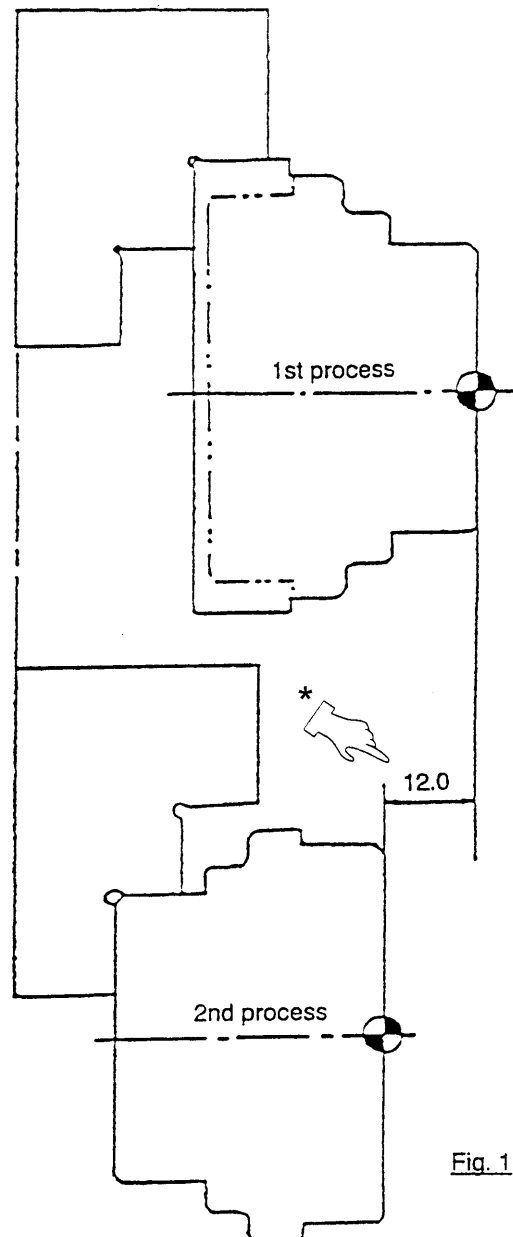


Fig. 1

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 G50 S1800

N5 G30 U0 W0

N6 M01

N100 (OD-R)

N101 T0100

N102 G97 S545 M08

N103 G00 X70.0 Z10.0 M03

N104 G01 Z0.2 F1.5

N105 G96 X-1.2 F0.2 S120

N106 Z...

N...

N...

N... M01

N700 (OD-F)

N701 T0700

N702 G97 S...M08

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

N704 ..

N...

N...

N... G00 G97 Z10.0

N... G30 U0 W0

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 G30 U0 W0

N24 M01

N5100 (OD-R)

N5101 T0100

N5102 G97 S545 M08

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...

N... G30 U0 W0

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 G30 U0 W0

N7 M01

N1000 T1000 M40

N1001 G00 Z1.0

N1002 X0



Positioning of  
works (manual)

N1004 G00 X200.0 Z50.0

N1005 M01

N8 M98 P2222 L3

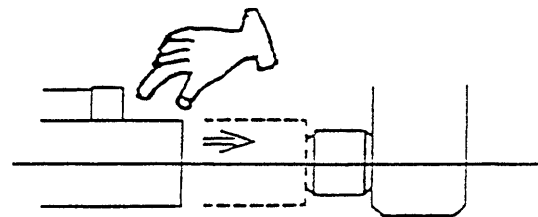
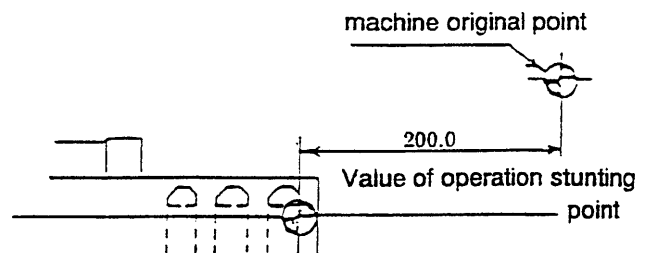


Call of operation program  
and setting

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-off program

└

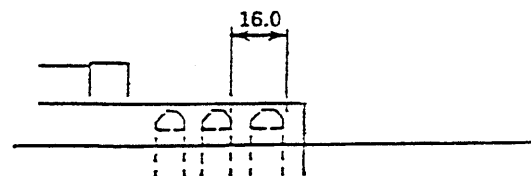
N... M01

N12 G10 P00 W16.0

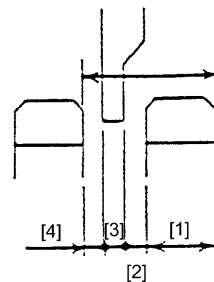


Shifted amount

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.





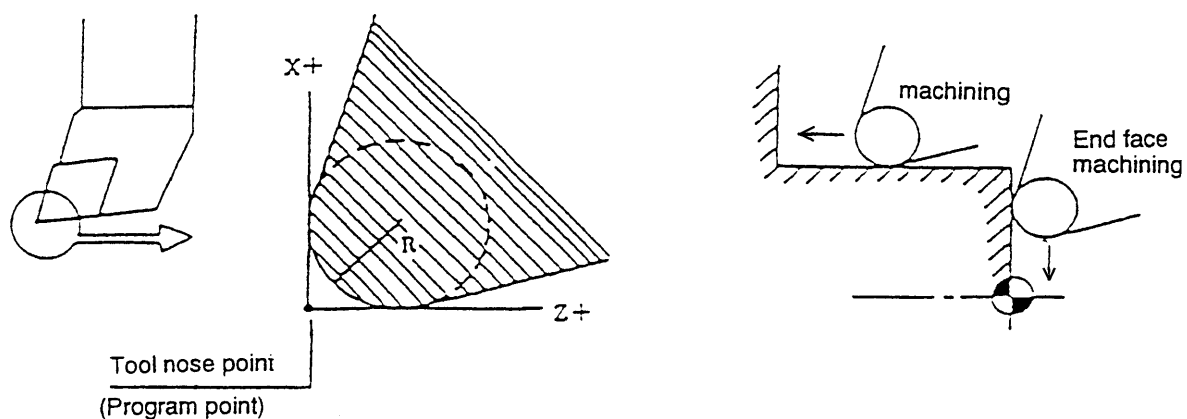
## 5. REFERENCE MATERIALS

### 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.

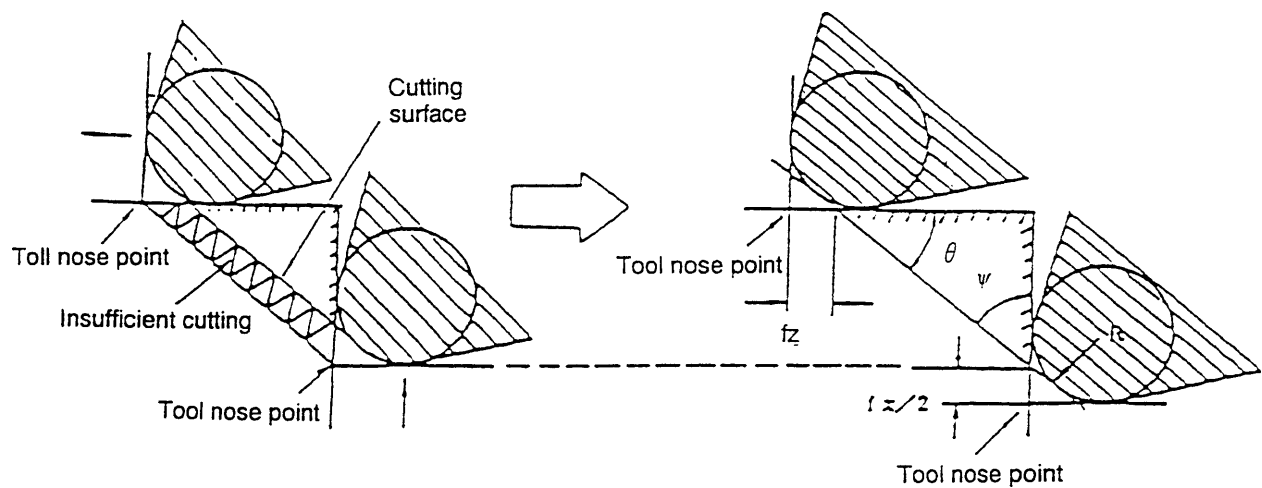
#### 1. Tool nose radius compensation amount

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).



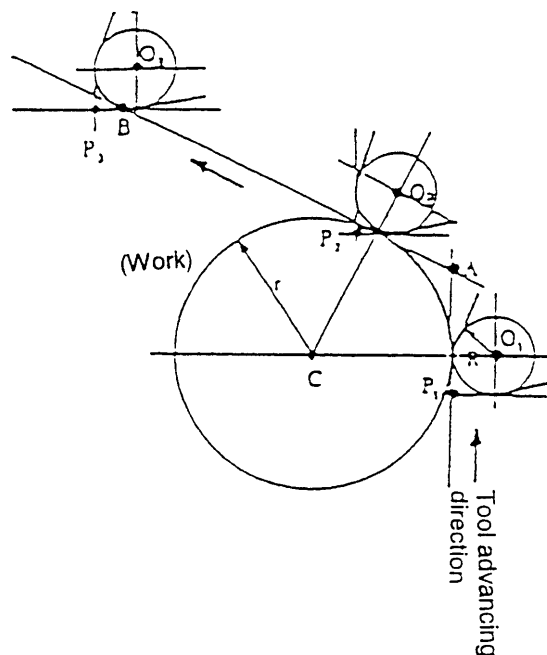
It can be cut according to the drawing even program by tool nose point.

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

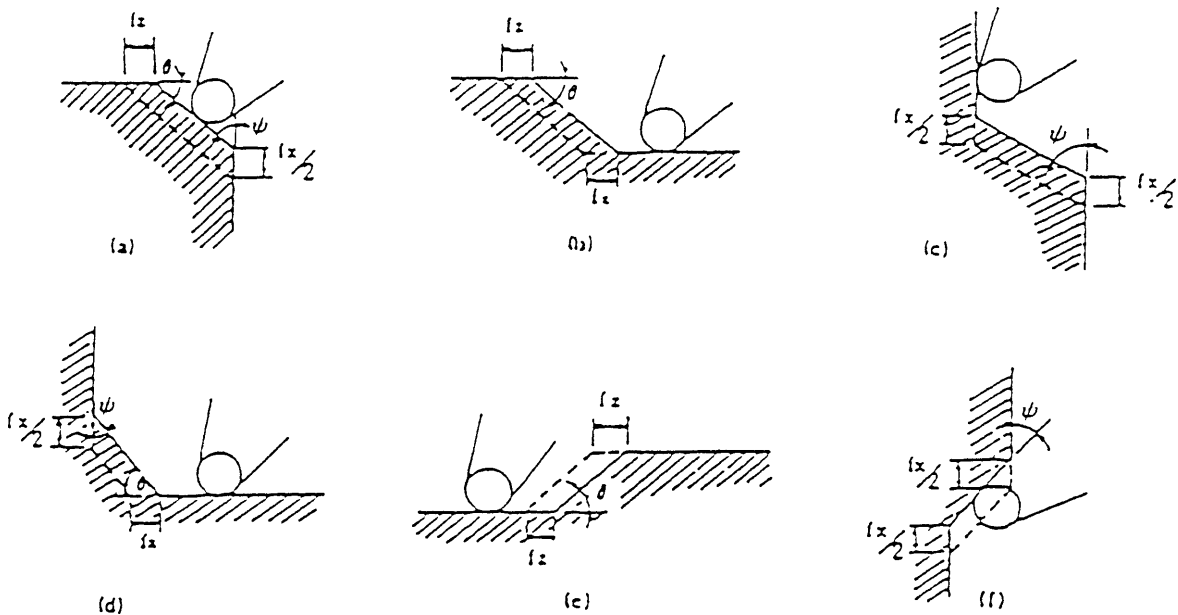
1. Calculate the coordinate values of the intersecting points of a straight line and those of the center of a circular arc. (in the above-mentioned figure, coordinate values of the points A, B and C)
2. Calculate the center coordinate values of the nose R to each intersecting point or contact point, 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.

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

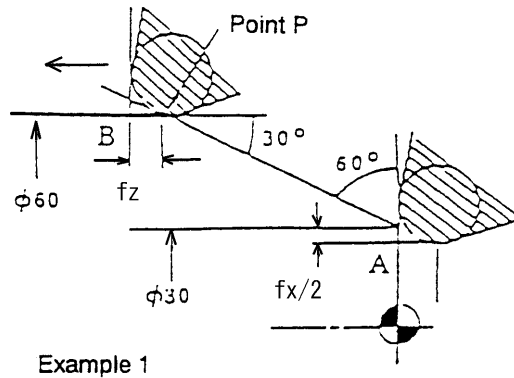
$$(e) \quad f_z = R \left( 1 + \tan \frac{\theta}{2} \right)$$

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

Tool nose R (radius)		0.2	0.4	0.5	0.8	1.0	1.2	1.6
Angle								
$\theta$ 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

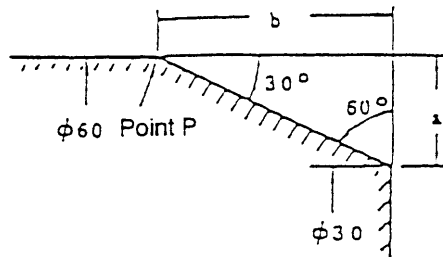
The case (e) and (f) on the previous page are excluded.

#### 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)

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

$$= 2 \times 0.8 \left( 1 - \tan \frac{60^\circ}{2} \right)$$

$$= 2 \times 0.8 (1 - \tan 30^\circ)$$

$$= 2 \times 0.8 (1 - 0.57735)$$

$$= 2 \times (0.42265)$$

$$= 2 \times 0.338$$

$$= 0.676$$

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

$$= 0.8 \times \left( 1 - \tan \frac{30^\circ}{2} \right)$$

$$= 0.8 \times (1 - \tan 15^\circ)$$

$$= 0.8 \times (1 - 0.268)$$

$$= 0.8 \times 0.732$$

$$= 0.5856$$

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

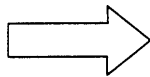
Tool nose point position of the tool A

$$X = \phi 30 - f_x$$

$$= 30 - 0.676$$

$$= 29.324$$

$$29.32$$



Coordinate value

X29.32

Z0

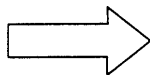
Tool nose point position of the tool B

$$Z = P - f_z$$

$$= -25.98 - 0.5856$$

$$= -26.5656$$

$$-26.57$$



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).

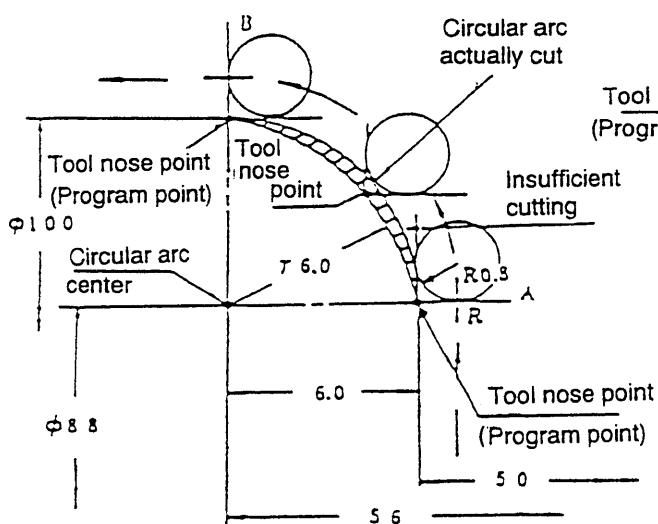


## 5. How to obtain tool nose radius compensation amount in circular cutting

- (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.

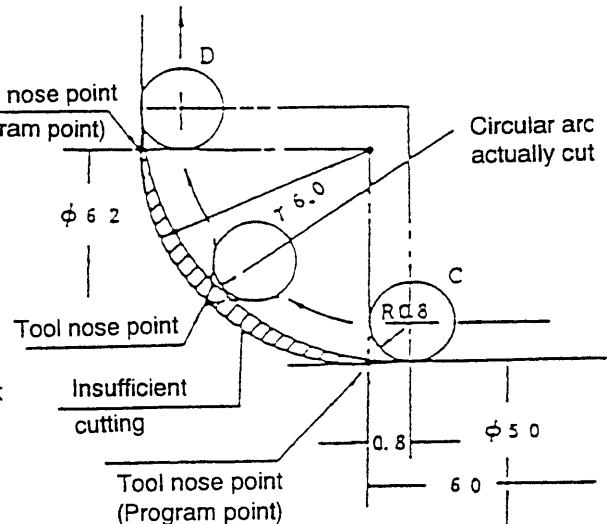


Due to this, insufficient cutting will be caused as shown in the following figure, if the nose R is not taken into account in case of circular cutting as well.



(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;



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.

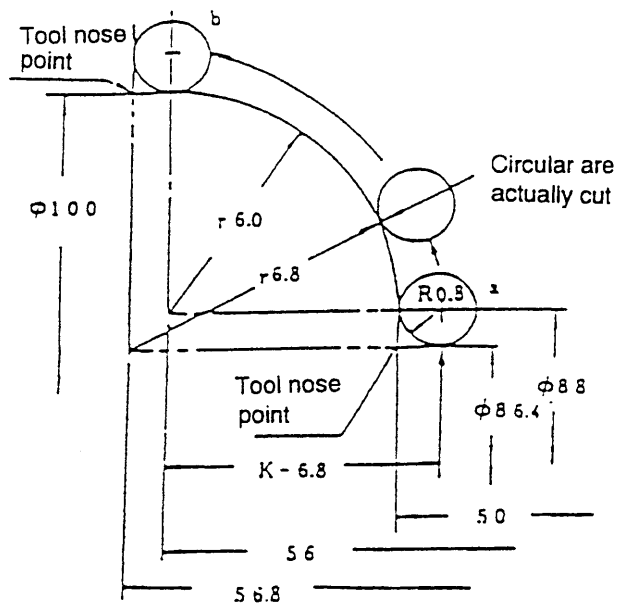


For the convex circular arc, command a circular arc “r” larger by a tool nose radius.

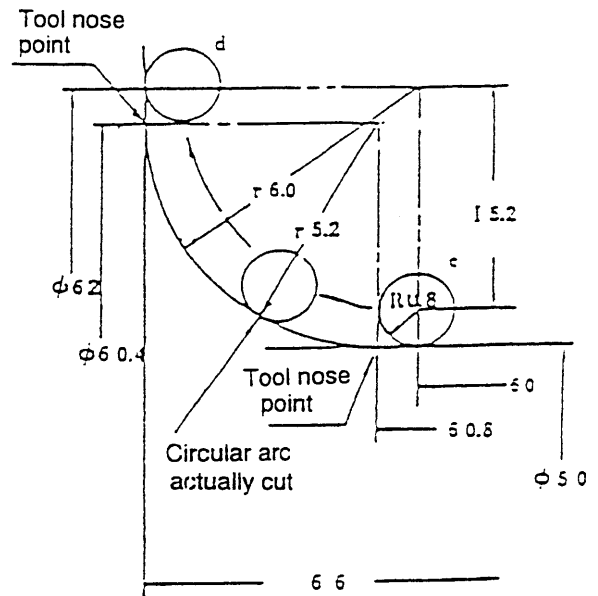


For the concave circular arc, command a circular arc “r” smaller by a tool nose radius.

(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

X86.4 ← a

G03 X100.0 Z-56.8 K-6.8 ← b

G01 Z-ΔΔ. Δ

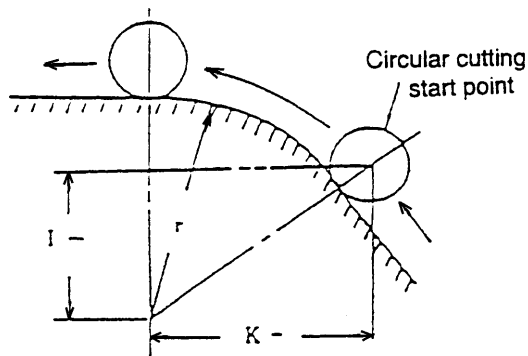
G01 X-50.0 F0.2

Z-60.8 ← c

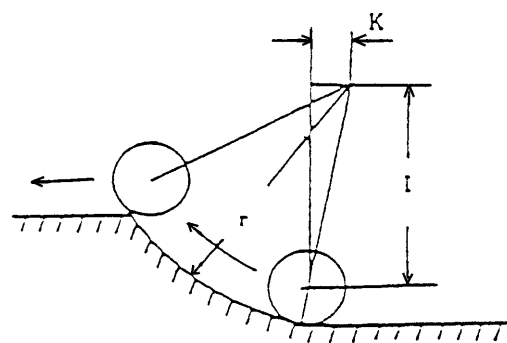
G02 X60.4 Z-66.0 I5.2 ← d

G01 XΔΔ. 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-

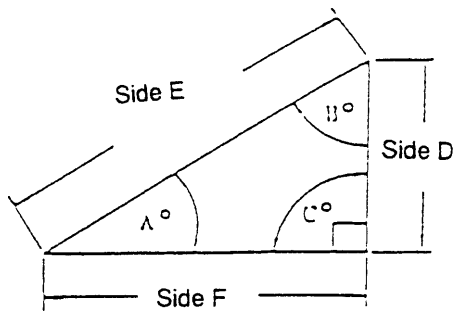


G02 X\_ Z\_ I+ K+

## 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.



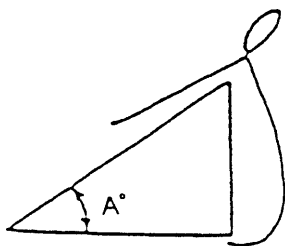
$$\sin A^\circ = \frac{D}{E}$$

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

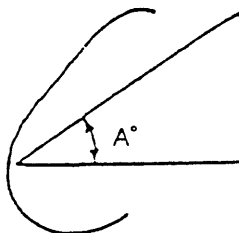
$$\tan A^\circ = \frac{D}{F}$$

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

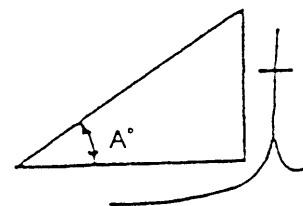
Side and angle given	Formula obtaining side or angle	
Angle "A" and side "D"	$E = \frac{D}{\sin A^\circ}$	$F = \frac{D}{\tan A^\circ}$
Angle "A" and side "E"	$D = E \times \sin A^\circ$	$F = E \times \cos A^\circ$
Angle "A" and side "F"	$D = F \times \tan A^\circ$	$E = \frac{F}{\cos A^\circ}$
Angle "B" 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



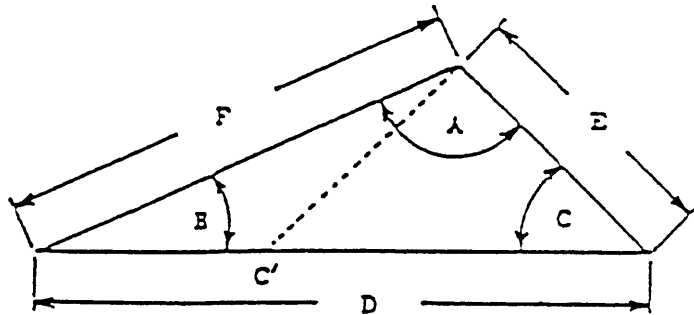
Cosine



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  $c$ .

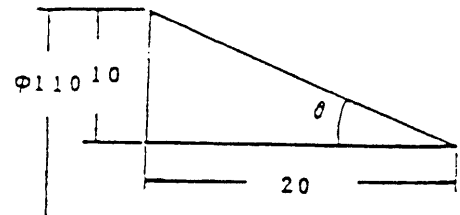
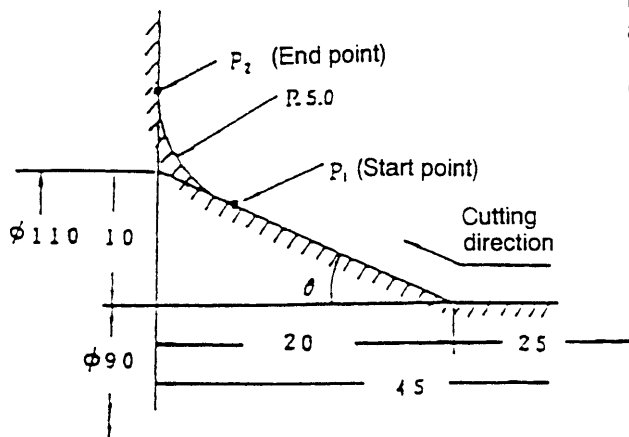
(4) When 1 side (D) and 2 angles are known;

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

### 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.

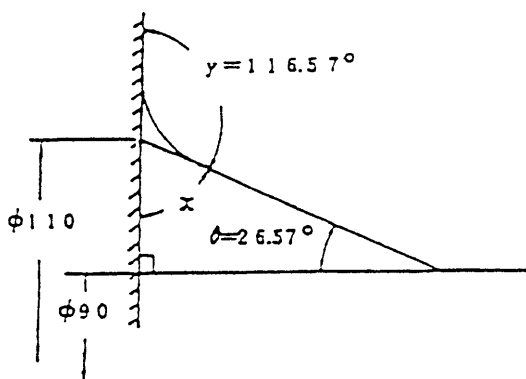
- (1) Obtain the taper angle " $\theta$ " in the left figure.



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

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

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

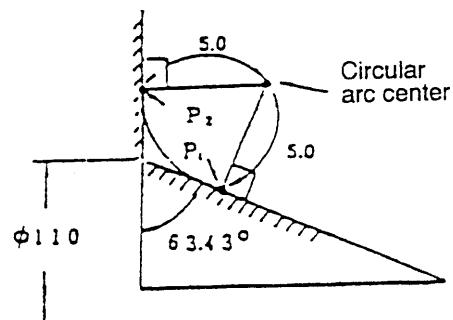


$$x = 90^\circ - 26.57^\circ = 63.43^\circ$$

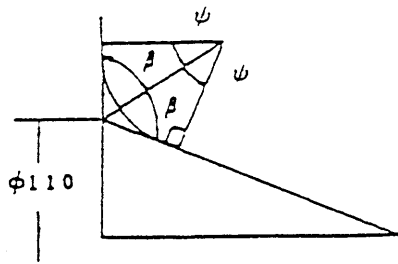
$$y = 180^\circ - x$$

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

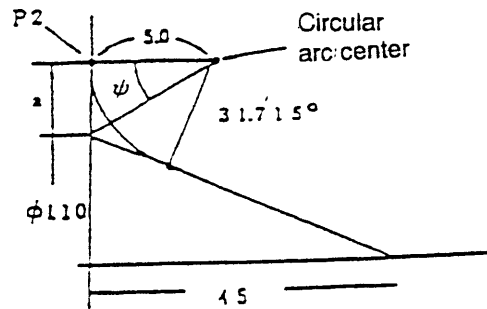
- (3) From the start point and the end point of the circular arc to the center of the circular arc, draw lines which are as long as a radius of 5.0.



(4) Divide the thus created fan shape into two equally and obtain the angles “ $\beta$ ” and “ $\psi$ ”.



(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 ( $P_2$ ) of the circular arc is ; X of  $P_2 = 3.09 \times 2 + \phi 110 = \phi 116.18$

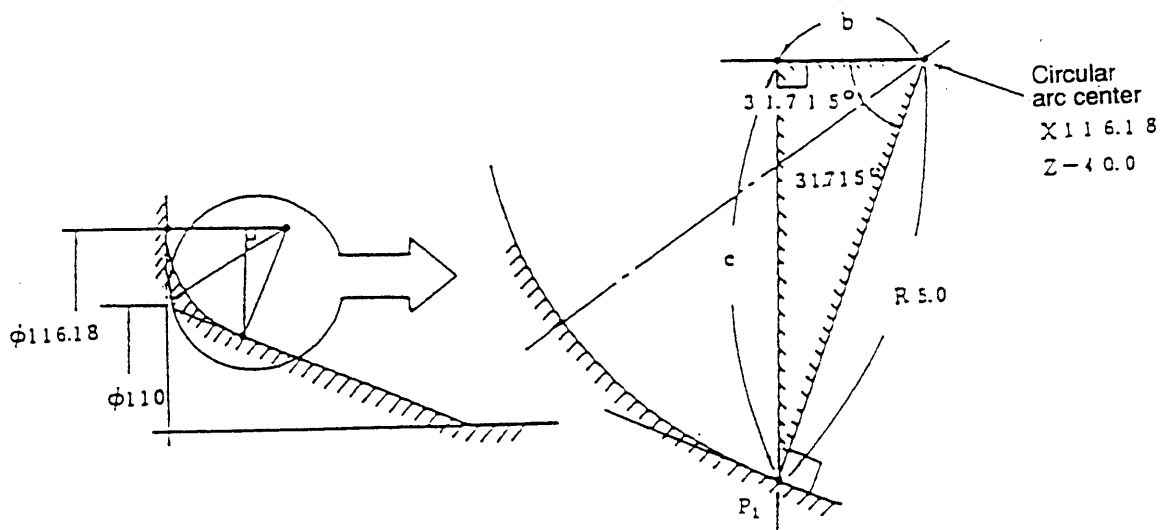
Point $P_2$ X116.18
Z-45.0

(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.

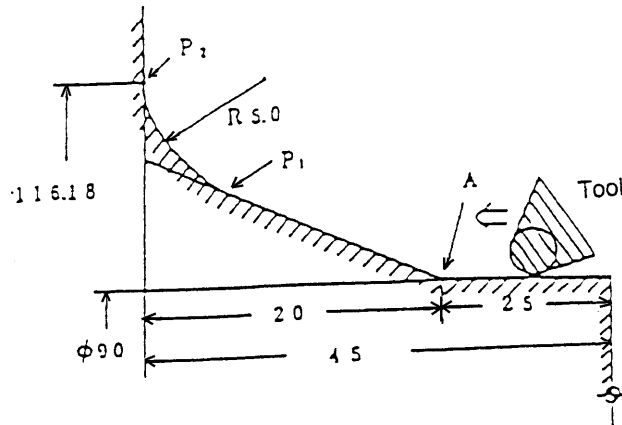
$$\begin{aligned} X \text{ of } P_1 &= \phi 116.18 - (C \times 2) \\ &= \phi 116.18 - (4.47 \times 2) \\ &= \phi 107.24 \end{aligned}$$

$$Z \text{ of } P_1 = -40 - 2.24 = -42.24$$

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) 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) 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
	Relation between the spindle speed 5000 $L(P) \times N$ and thread lead	N. Spindle speed $\delta_1$ . Incomplete thread area (before machining) $\delta_2$ . Incomplete thread area (after machining)
	Incomplete thread area $\delta_1 = K_1 \times N \times L(P)$ $\delta_2 = K_2 \times N \times L(P)$	$K_1, K_2$ . : constant Convert unified and Whitworth thread pitches into millimeter.
	Depth of thread cutting (diameter value)	
	1. Metric thread 60° $0.6495 \times P \times 2$ 2. Unified thread 60° $0.6134 \times P \times 2$	
Finishing surface roughness	Surface roughness based on feed rate and tool nose radius $H_{max} = \frac{F^2}{8 \cdot R}$ Feed rate based on surface roughness and tool nose radius $F = \sqrt{8R \cdot H_{max}}$	Hmax Max. surface roughness F. Feed rate (mm/rev) R. Tool nose radius
Power and depth	Power required for cutting "KW" $KW = \frac{K \cdot t \cdot F \cdot V}{6120 \times 0.8}$ Max. depth of cut "t" $t = \frac{6120 \times KW}{K \cdot V \cdot F} \times 80\%$ Required horsepower "HP" $HP = \frac{KW}{0.75}$	K. Specific cutting resistance t. Depth of cut (mm) F. Feed rate (mm/rev) V. Cutting speed (m/min) Motor efficiency 80%





ST200/250 CNC LATHE  
INSTRUCTION MANUAL  
PROGRAMMING  
SEICOS-pcFLexi  
Version 1.01  
8-2001

First Edition 8-2001