CNC LATHE

INSTRUCTION MANUAL

PROGRAMMING

SEIKI - SEICOS ∑10L/21L

45 Edition 1.01 11-2000

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

1. PREPARATION FOR TOOL LAYOUT

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

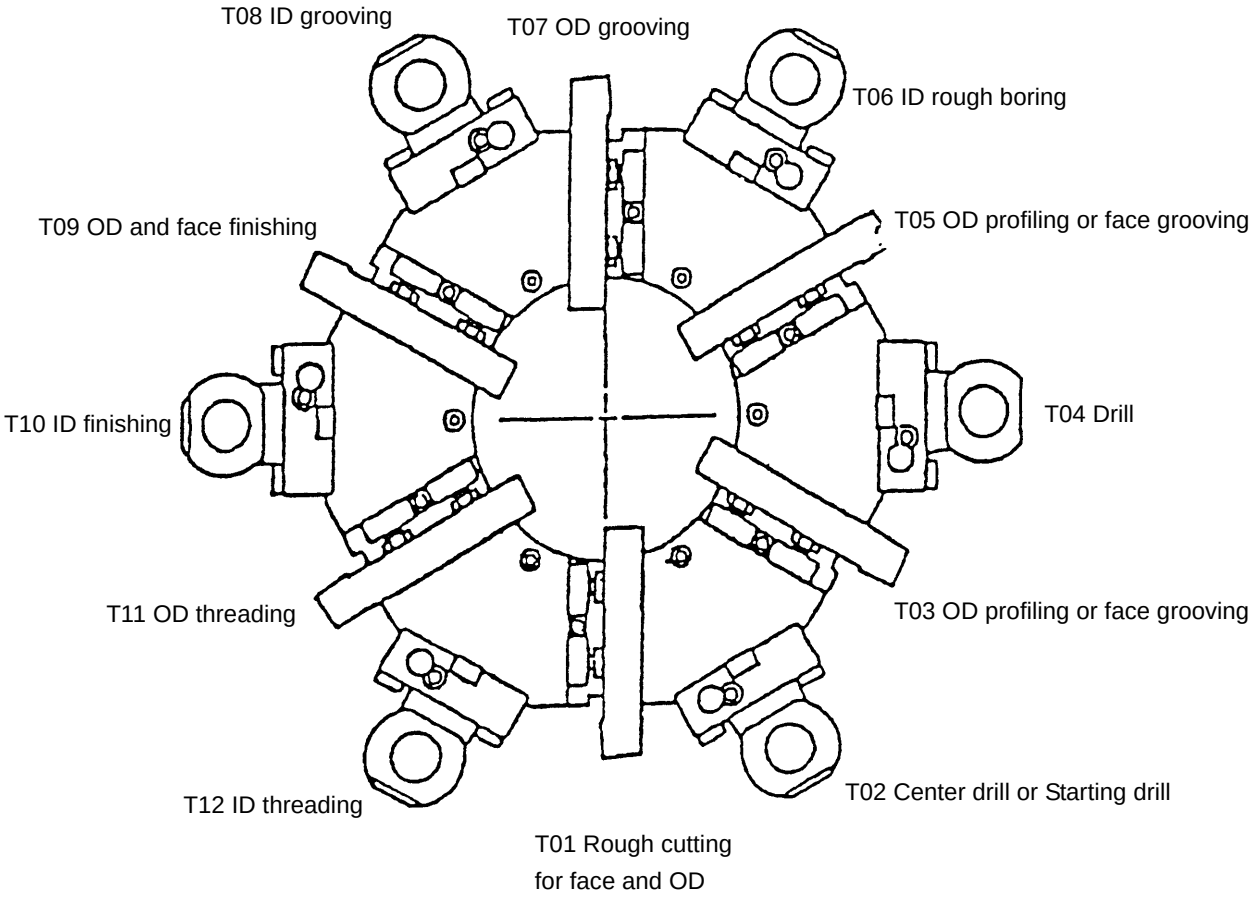
1-1 Tool Set

Standard Tool Set

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

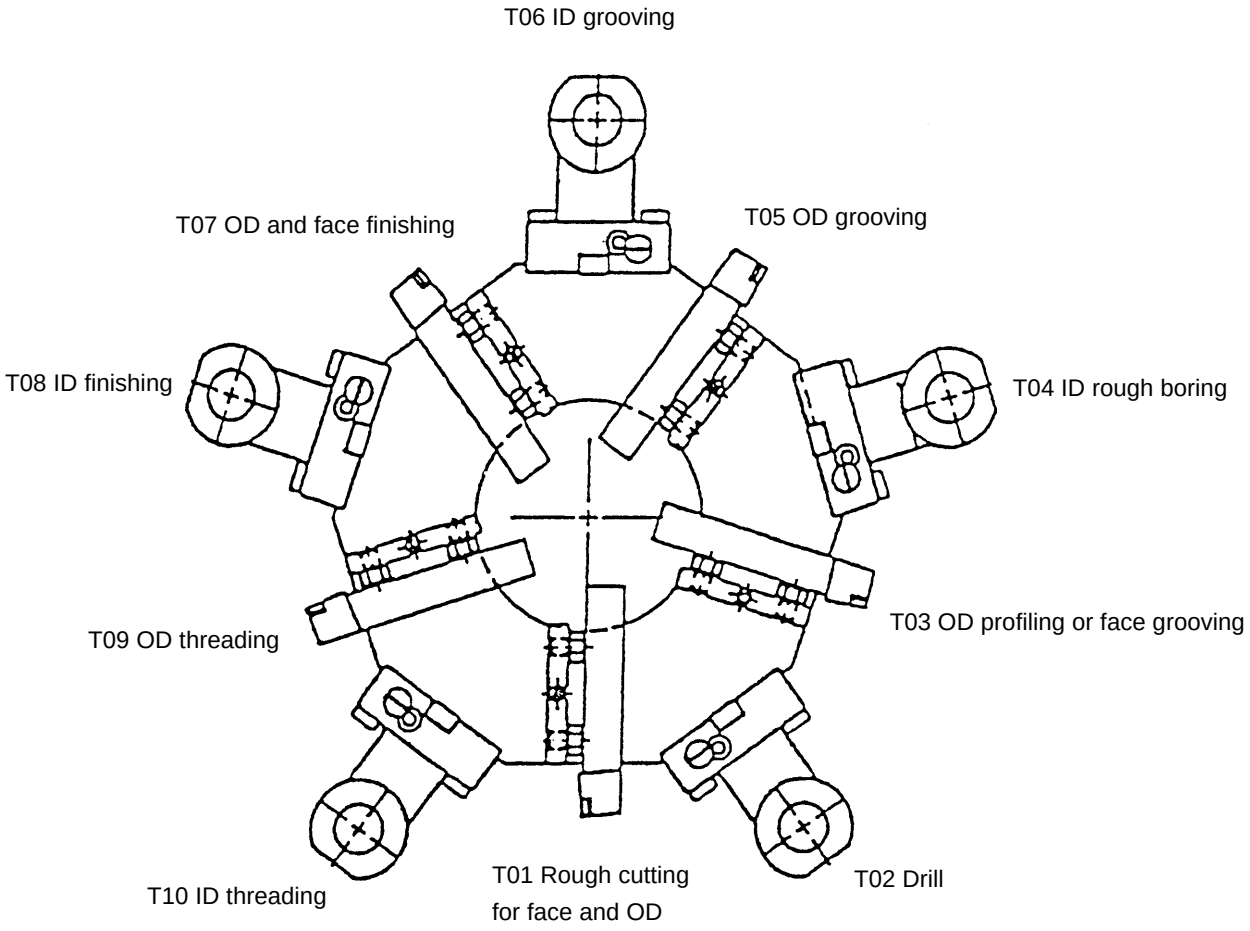
Further, set the tools as you like in order to satisfy the operation accuracy of the small tools such as the boring bar, and also to perform the turret indexing by one rotation.

The standard tool set is shown as below.



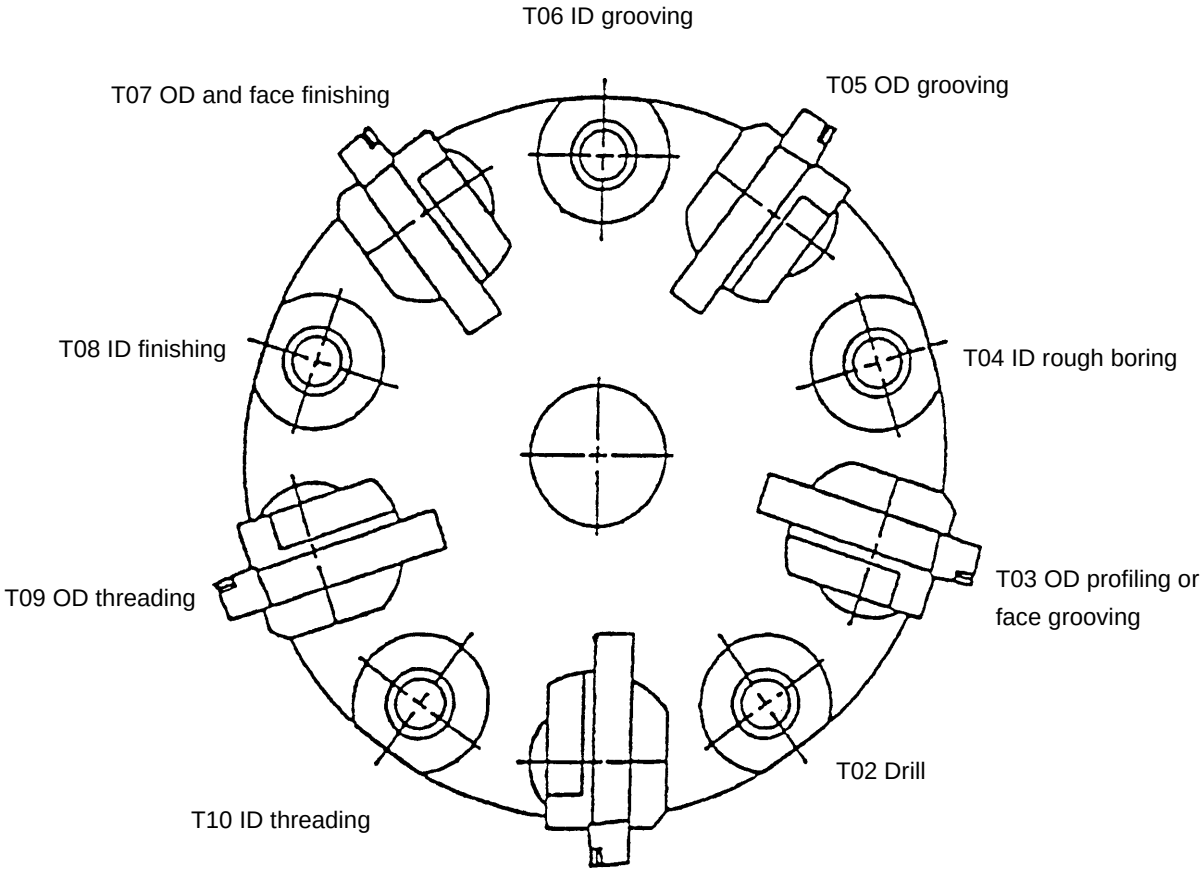
Specifications of 12-station Variable turret

Standard Tool Set



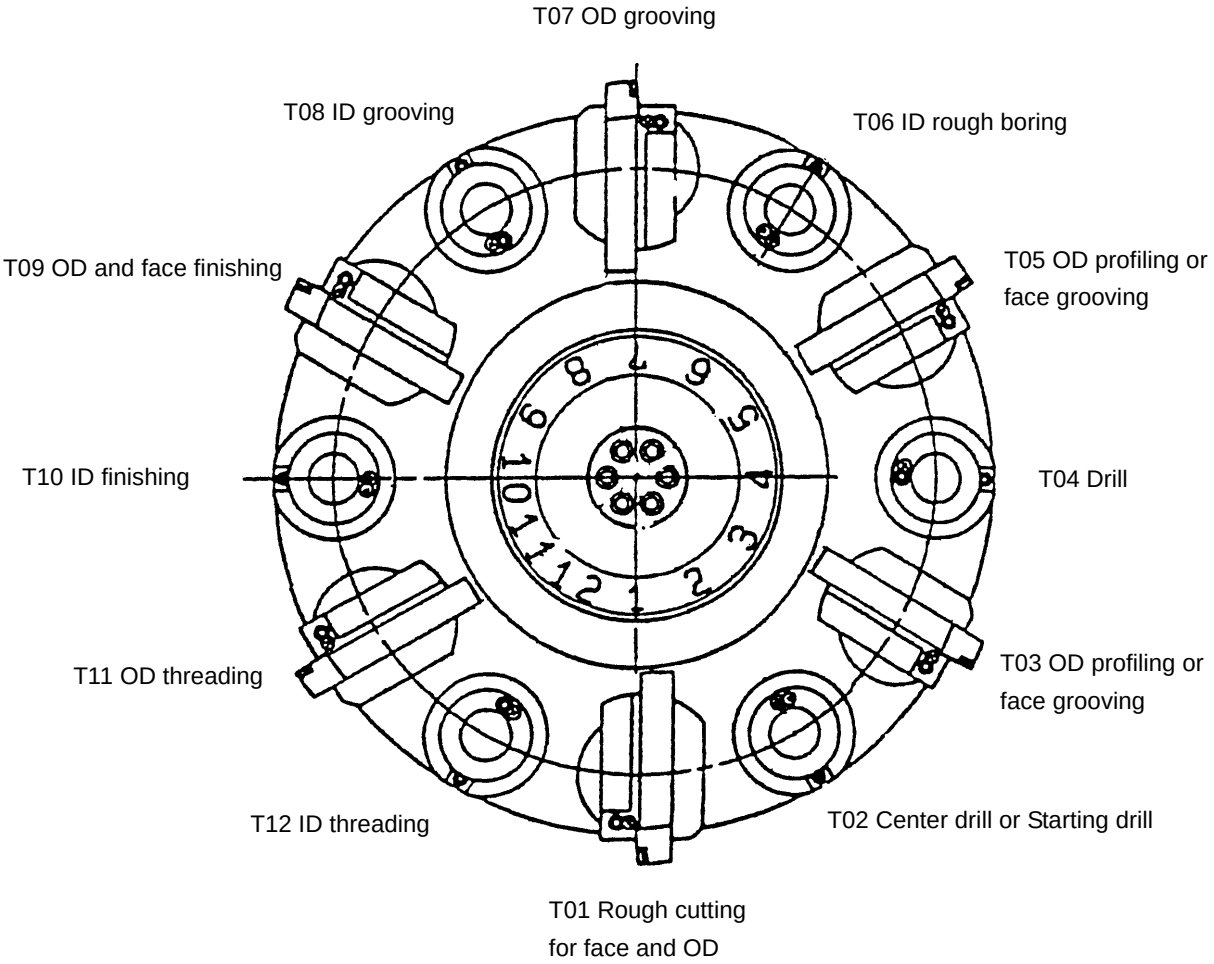
Specifications of 10-station Variable turret

Standard Tool Set



Specifications of 10-station QCT turret

Standard Tool Set

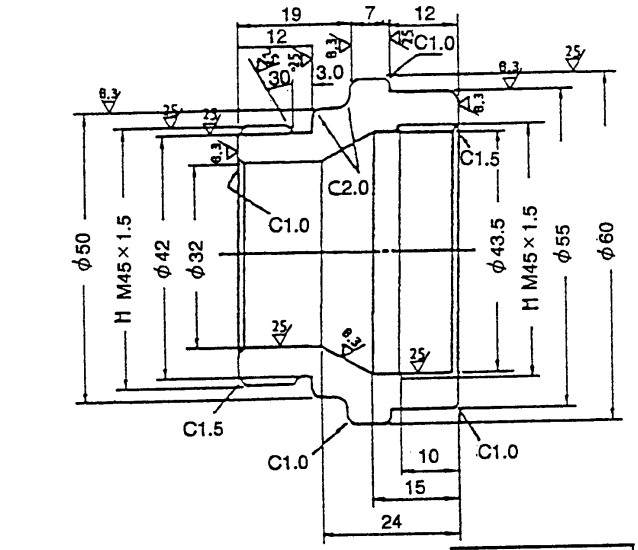


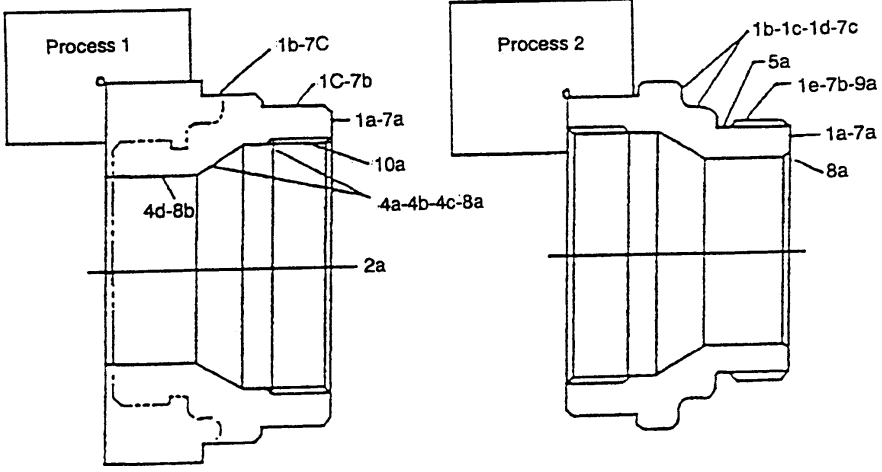
Specifications of 12-station QCT turret

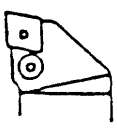
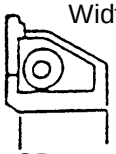
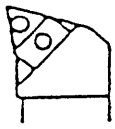
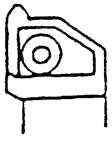
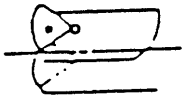
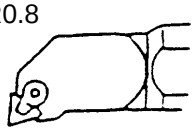
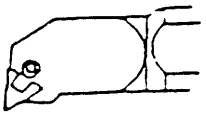
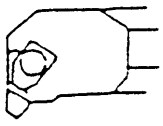
1-2 Tool Layout

Example of tool layout for chuck work		SEIKI 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	±99999.999(mm) ±99999.999(deg)	±9999.999(inch) ±99999.999(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	

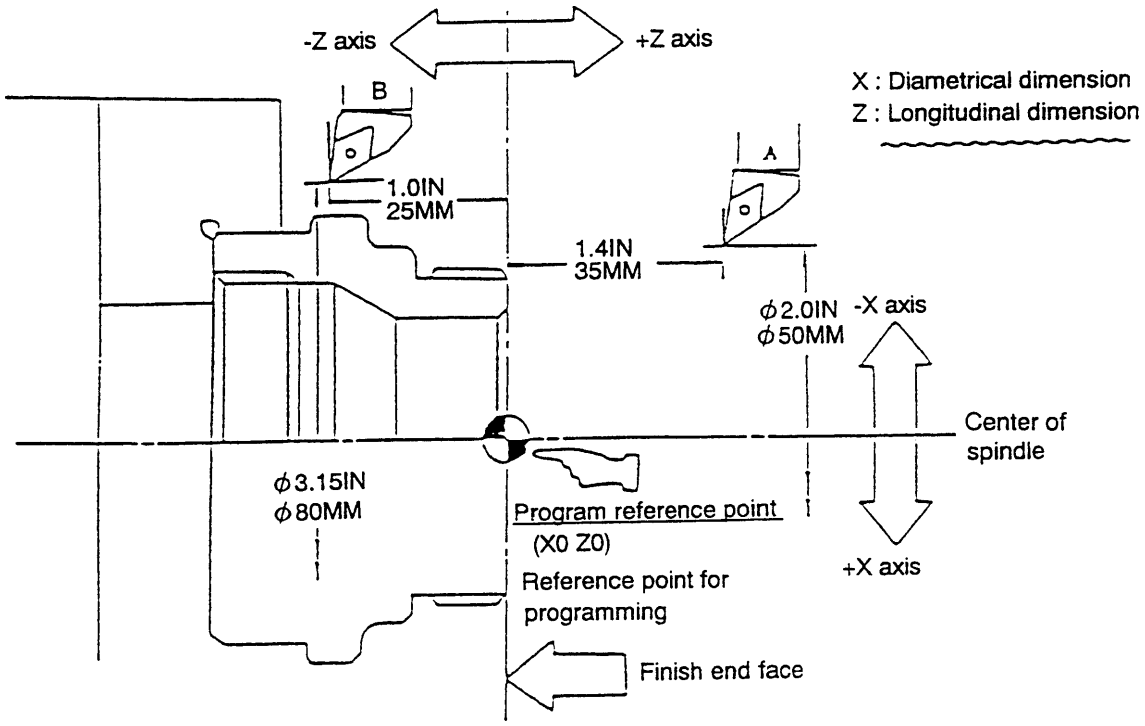
2. PROGRAMMING

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 “XO”, and the Z axis assumes a workpiece finish end face on the tail stock side to a position of “ZO”.

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 locates 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 locates a plus 80 dia. on the X-axis and minus 25mm (1.0”) on the Z-axis,

X80.0 Z–25.0

2-1-2 Regarding Machine Zero Point

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

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

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

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

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

```
05802
N1  G28  U0
N2  G28  W0  T0100
N3  G50  S2000
N4  G00  X200.0  Z200.0
N5  M01
N701  T0700
N702  G97  S1500  M08
N703  G00  X30.0  Z10.0  M03
N704  G01  G96  Z0  F1.0  S150
N705       X41.85  F0.2
N706       X44.85  Z-1.5
N707       Z-12.0
N708       X46.0
N709  G03  X50.0  Z-14.0  R2.0
N710  G01  Z-17.0
N711  G02  X54.0  Z-19.0  R2.0
N712  G01  X58.0
N713       X62.0  Z-21.0
N714  G00  X70.0
N715  G97  X200.0  Z200.0  S680
N716  M01
N6    G28  U0  W0  T0100
N7    M30
%
```

Represents a "program number" and used to distinguish from other programs.
Do not use an identical program number.

Limits a maximum spindle speed to 2,000 rpm.

Calls the turret head to be used.

Sets a spindle speed per minute to 1,590min^{-1}

Sets a cutting speed to 150 m/min. This is called constant surface speed control.

Used when performing circular cutting in the counterclockwise direction.

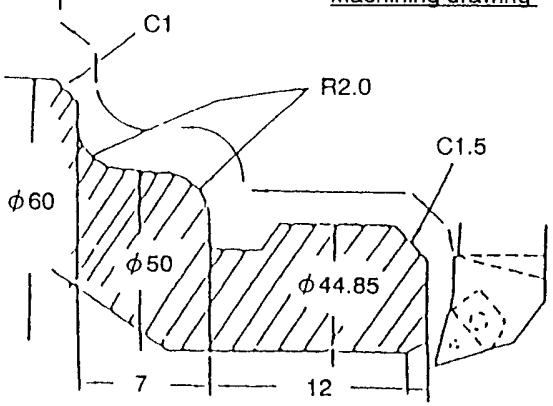
Cancels circular cutting by G01.

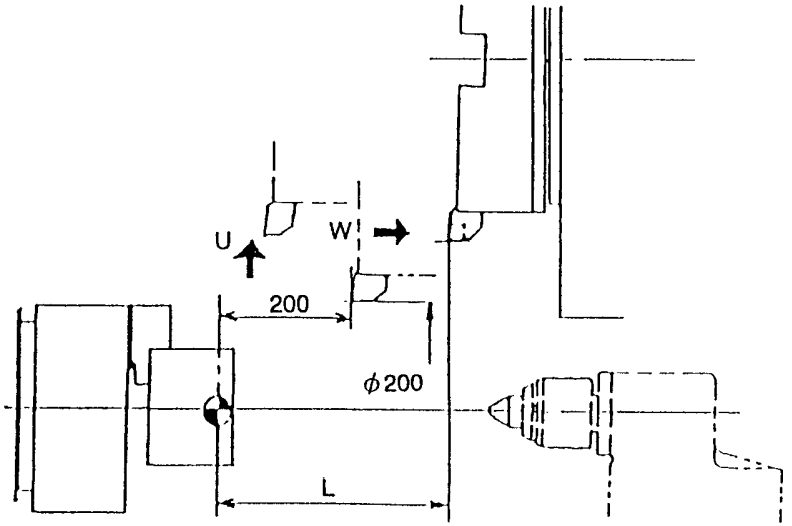
This is normally called a sequence No. and identifies a block.
Avoid using an identical number.
Even if a sequence No. is not placed, it is not affected the motion of machine.

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

Used when performing circular cutting in the clockwise direction.

Machining drawing





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

2-2-1 F Function (Feed Function)

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

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

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

1.0 mm/rev = F1.0 or F100

1.5 P thread = F1.5 or F150

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

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

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

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

(Example) In case of 14 threads per inch

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

1.81429mm/rev F1.81429

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

(Example) E14.0

Max. feed rate 10,000mm/min.

A maximum feed rate depends on the spindle speed used.

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^{-1})

G97S□□□□ Constant surface speed cancel

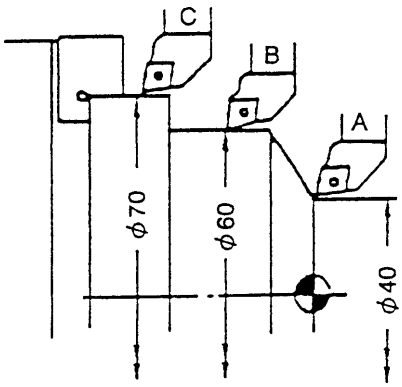
Specify a spindle revolution with S□□□□ .

(Example) G97 S1000 : A spindle speed per minute is set to 1,000 (mim^{-1})

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.



Obtaining the spindle speed from the surface speed

* Formula for calculating the spindle speed from the surface speed

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

V : Surface speed (m/min)

π : 3.14

D : Tool nose position (ϕ mm)

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

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

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

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

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

***Notes**) 1. Considering a workpiece chucking condition, specify the maximum spindle speed limit with S 4-digit code in a G50 block at the beginning of a program.*

2. When roughing with G96, calculate maximum and minimum spindle speeds so that cutting will be performed in a constant power range as much as possible.

3. When changing over from G96 to G97 and vice versa, specify not only a G code, but also an S code.

4. When changed over from G96 to G97 and no S code is specified, the spindle is run with the speed specified in the latest S code in G96 mode.

5. When changed over from G96 to G97 and no S code is specified, the spindle turns with the previously used surface constant speed is S code had been specified in G96 mode.

Also, when no S code is specified in G96 mode, S results in 0.

6. The following interlocks are provided as the rotating conditions of spindle.

(1) The direction of the chuck inner clamp and outer clamp key shall be the same direction as that of chuck clamping.

(2) Q-setter shall be stored.

(3) Rotating speed shall be command with G96 Sxxx.

(4) The lamp of advance or retract of center support shall be on. (Option)

(5) The door shall be closed.

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

(1) The connection of C-axis shall be in the status of OFF (M40 command). (Option)

(2) The connection of rotating tool shall be in the status of OFF (M45 command). (Option)

(3) Set up of the ACT shall be cancel condition. (Option)

(4) The safety door of the ATC magazine shall be closed. (Option)

2-2-3 T Function (Tool Function)

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

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.

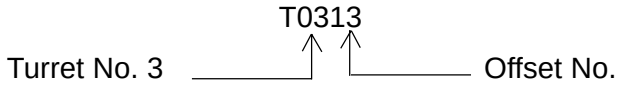
Example) If the turret No. 3 is to be called, program as follows:

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)



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.)

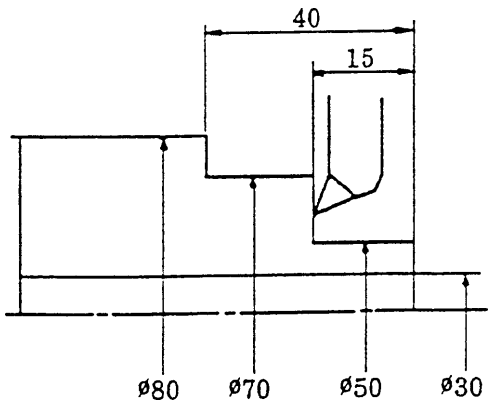
Caution When "T□□ΔΔ" command is specified on the same line as the axis travel command, the indexing of turret is made simultaneously with traveling and a coordinate is set after completion of traveling.

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

3. Compound Offset

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

Example 1)




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

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

Example) Input status of dimension adjustment when the part ϕ70 is made larger by 0.03.

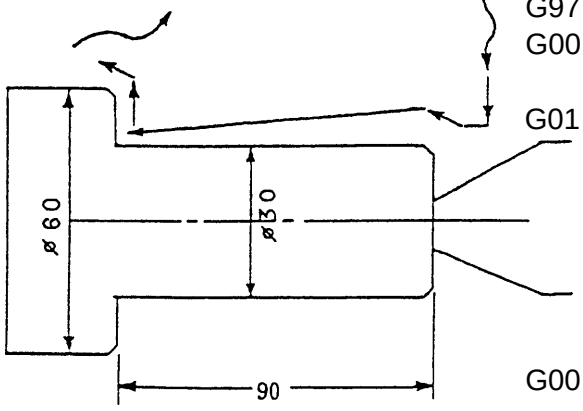
TOOL LAYOUT

OFFSET

	X	Z	R	T
19	0.03	0	0	0

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

Example 2) Cutting with taper of –0.3 at ϕ30 part



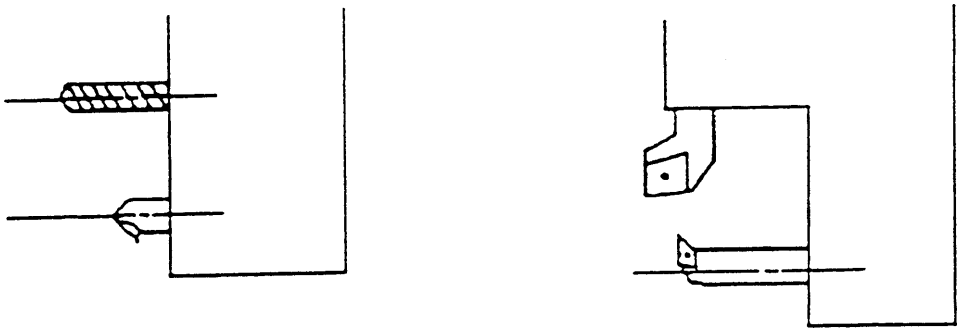

```
T0500
G97     S2000   M08
G00     Z3.0    M03
        X40.0
G01     X24.0   F1.0
        Z1.0    F0.2
        X30.0   Z–2.0
        Z–90.0  T0525  <———  Compound offset
                              Offset No. 25
        X57.00  T0500  <———  Compound offset
        X62.0   Z–92.5        cancel
G00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

OFFSET

25 X–0.3 Z0 R0 T0

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.

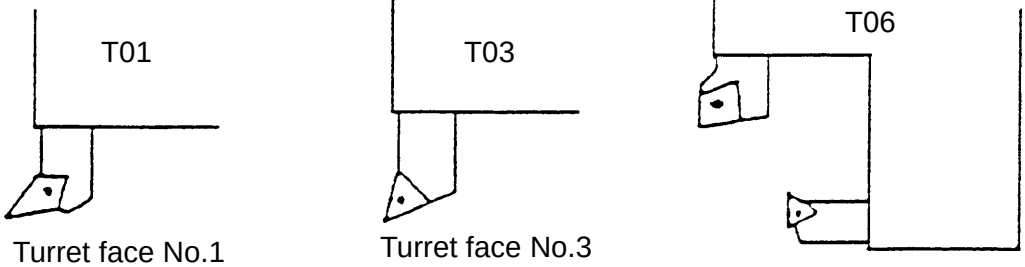
1 Both tool nose R and tool point are zero → Compound compensation

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)

D. Program example



Turret face No.1 Offset No.1

Turret face No.3 Offset No.3 (Compound compensation 33, 34)

Turret face No.6 (Offset No.6, 36)

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
	≀	
	M01	
N600	T0600	The turret face No.6 is indexed and setting-up is
	≀	Compound compensation ON (Offset No.36)
	T0636	Multi tool compensation ON (Offset No.36)
	≀	
	M01	

Example of compensating data

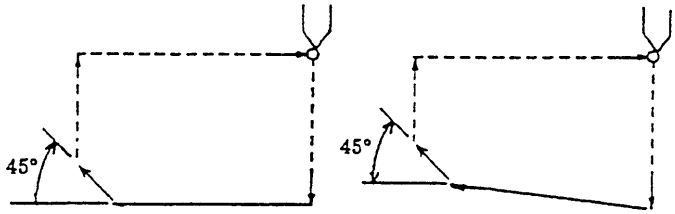
No.	X	Z	R	T
01	Q-setter	Q-setter	0.8	3
03	Q-setter	Q-setter	0.8	3
06	Q-setter	Q-setter	0.4	2
33	Extremely small amount	Extremely small amount	0	0
34	Extremely small amount	Extremely small amount	0	0
36	Q-setter	Q-setter	0.4	2

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

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

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

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

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

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

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

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

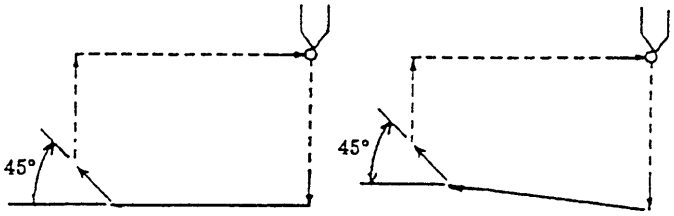
M code	Function	Description
M124	Turret air blow ON	Air is blown from turret.
M125	Turret air blow OFF	Air blow from turret stops.

M Function (Miscellaneous Function) List (TF25)

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

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

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

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

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

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

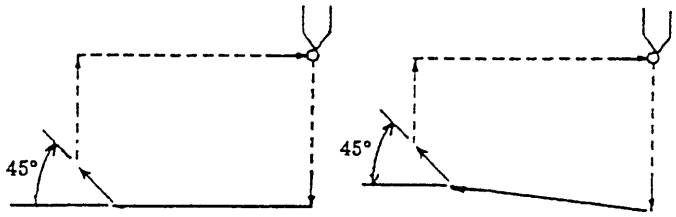
M code	Function	Description
M76	Chip conveyor stop	Chip conveyor stops.
M81	Robot service 1	Robot start 1
M82	Robot service 2	Robot start 2
M83	Tool edge measuring arm Check condition ineffective	When measuring arm swings, chuck open/close condition is neglected.
M84	Tool edge measuring arm Check condition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

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

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

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

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

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

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

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

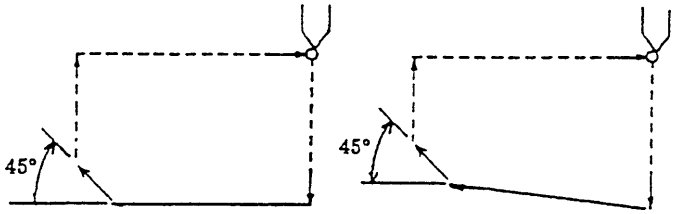
M code	Function	Description
M76	Tool-nose measuring arm return	
M77	Tool-nose measuring censor air blow ON	
M78	Tool-nose measuring censor air blow OFF	
M79	Unloader cylinder out	
M80	Unloader cylinder return	
M81	Robot service 1	Robot start 1
M82	Robot service 2	Robot start 2
M83	Chuck interlock of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

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

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

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

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

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

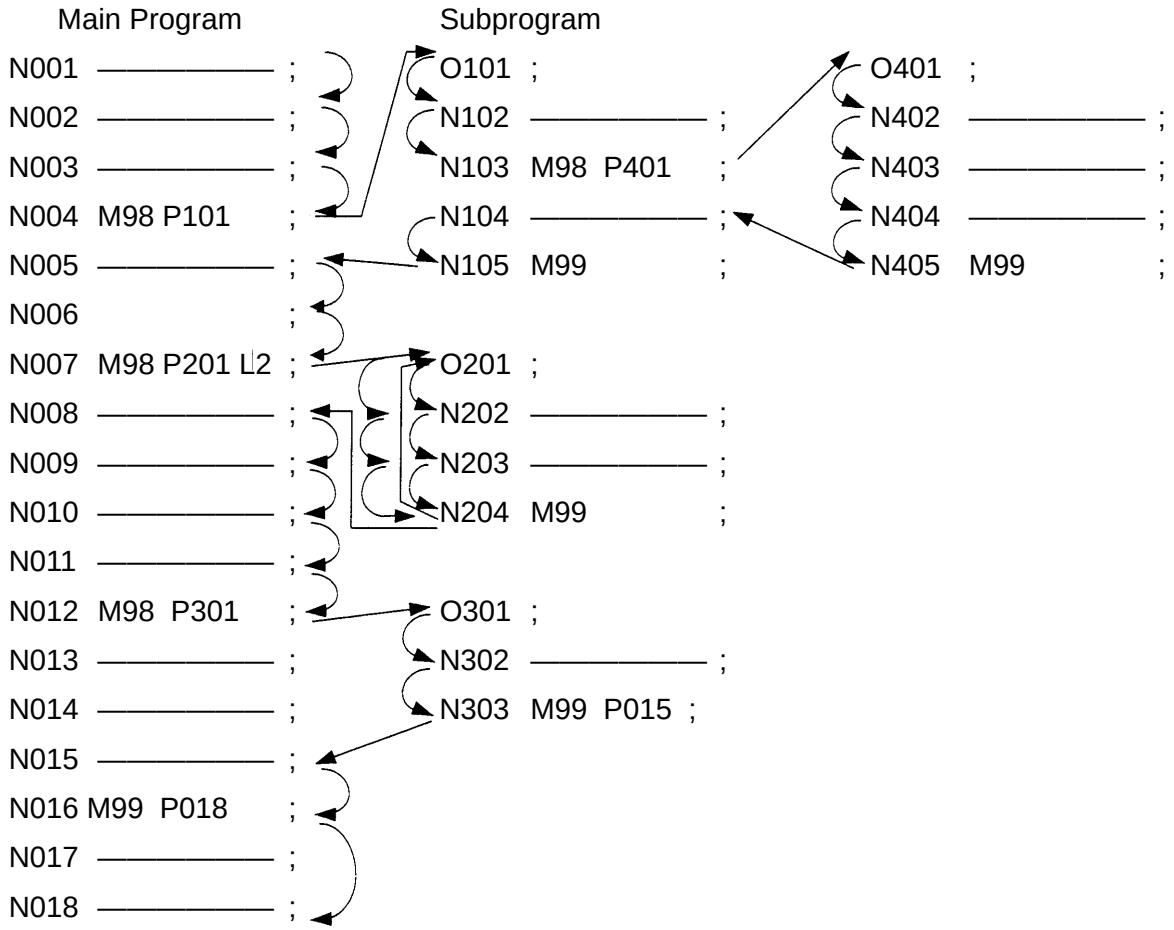
M code	Function	Description
M37	Power off is not effective at program stop	Power does not off even the command of M00, M01, M02 or M03 when the power cut off is ON. Discharge the air at the live center section.
M38	Center air blow ON	Air is blown to the center.
M39	Center air blow OFF	Stop the air.
M40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

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

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

Example of Subprogram Call

(Example)



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

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

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

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

2-3 Details of G Function

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

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

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

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

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

Note *1) When the source power is switched on, those G codes marked ◤ are set.*

2) G codes of 00 group indicate those which are not modal, and are effective to the blocks indicated.

3) When G codes which are not listed in G Code List are commanded, alarm is displayed, and when G codes which don't have corresponding options, alarm is displayed.

4) Any numbers of G codes can be commanded in the same block, if they belong to different groups.
When two or more of G codes which belong to the same group are commanded, G code later commanded becomes effective.

2-3-2 G50 Maximum Spindle Speed Setting

Using a command “G50 S ..... ;” , you can directly specify the upper limit value of a spindle speed (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 (m/min. or ft/min.) will be clamped to this upper limit.

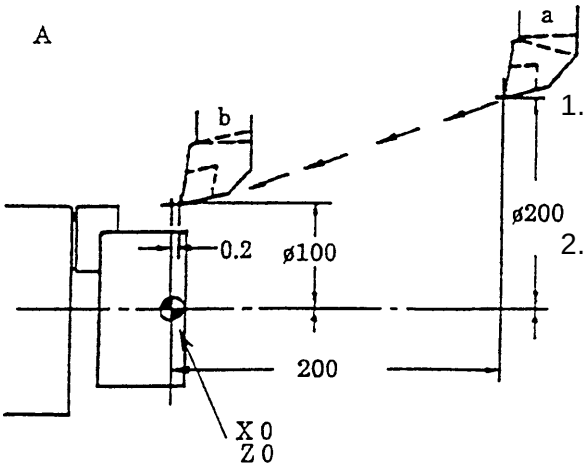
(Example) G50 S2000 ;

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

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

2-3-3 G00 Positioning

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



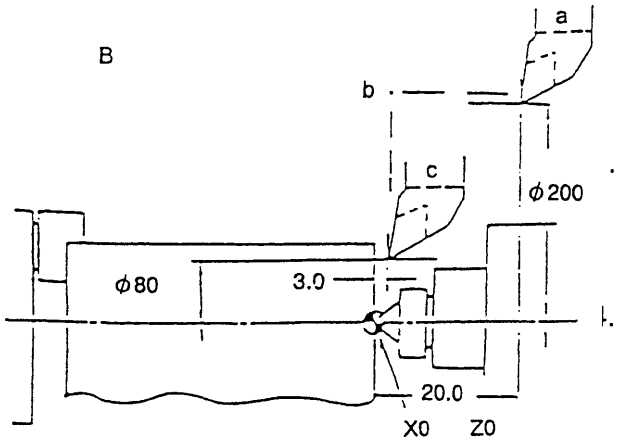
Chuck work

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

   G00 X100.0 Z0.2

2. From the point “b” to the point “a”

   G00 X200.0 Z200.0



Center work

Specify one axis when there is any interference.

3. From the point “a” to the point “b”

   G00 Z3.0

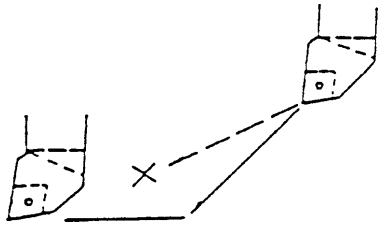
   From the point “d” to the point “c” X80.0

4. From the point “c” to the point “b”

   G00 X200.0

   From the point “b” to the point “a” Z20.0

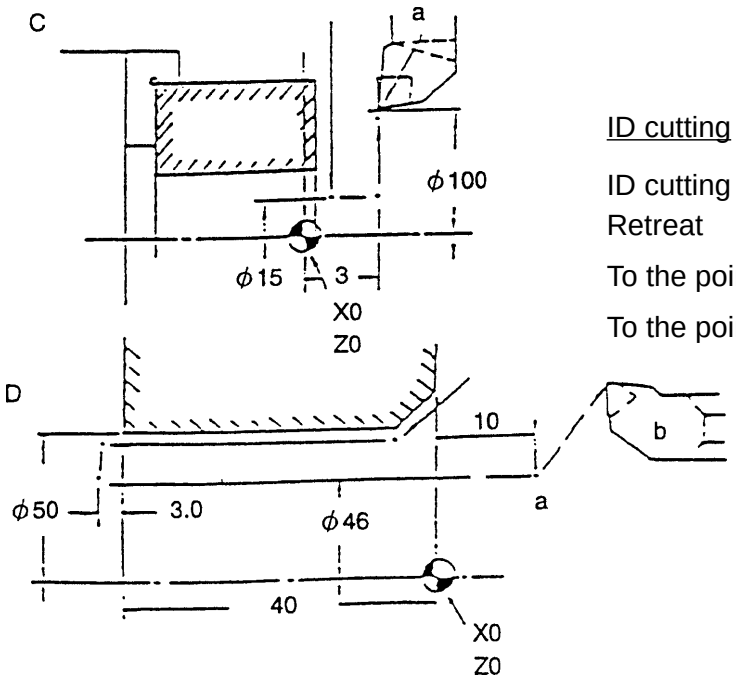
***Note**: When simultaneously positioning both the X and Z axes, the tool does not linearly move from a current position to a specified position, because their rapid traverse rates differ from each other. Therefore, you must be careful when there is an interfering substance halfway a tool path.*



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

When moving to the next cutting position

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



End face OD cutting

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

ID cutting

ID cutting	G01 Z–43.0 F0.2
Retreat	X46.0
To the point “a”	G00 Z10.0
To the point “b”	X200.0 Z200.0

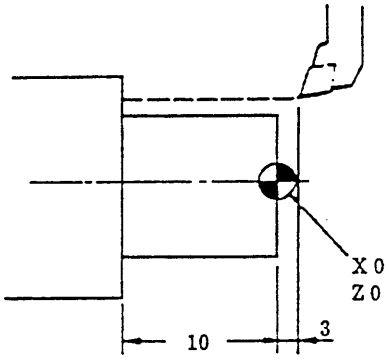
2-3-4 G01 Linear Cutting

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

Chamfering and taper cutting are also considered linear cutting.

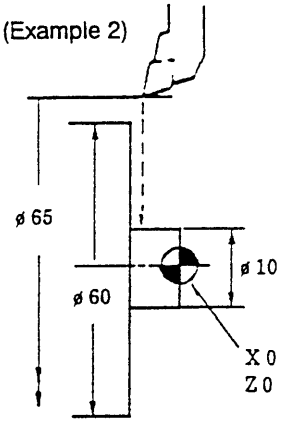
Use an F code to specify a feeding rate.

(Example 1)



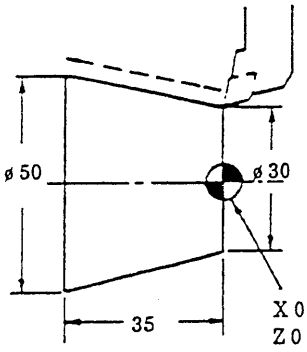
G01 Z-10.0 (F)... ;

(Example 2)

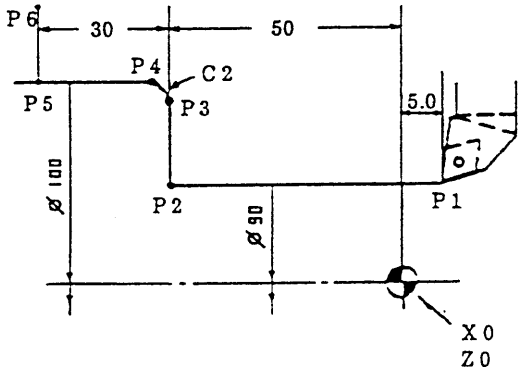


G01 X10.0 (F)... ;

(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 G01 command is reset, the feed rate given with an F code is kept.

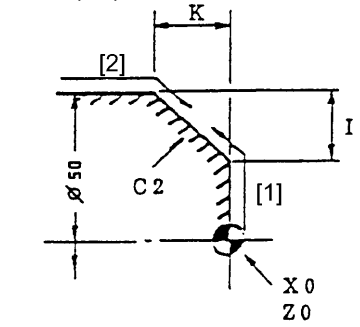
(2) Chamfering, corner R command

When there is chamfering (45°chambering) 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 chambering	For corner R
(a) G01 X ... K ... F ...	(c) G01 X ... R ... F ...
(b) G01 Z ... **I** ... F ...	(d) G01 Z ... R ... F ...

X and Z coordinate values .. Position after chambering 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.

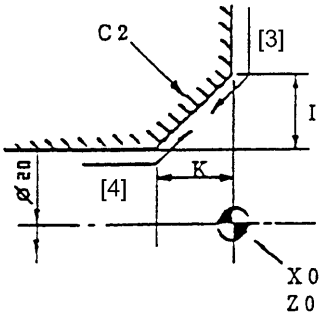


For the direction [1]

G01 X50.0 K–2.0

For the direction [2]

G01 Z0 **I**–2.0

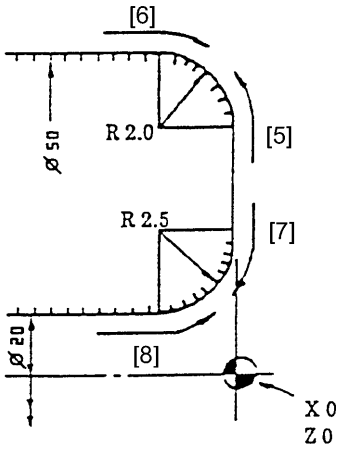


For the direction [3]

G01 X20.0 K–2.0

For the direction [4]

G01 Z0 **I**2.0



For the direction [5]

G01 X50.0 R–2.0

For the direction [6]

G01 Z0 R–2.0

For the direction [7]

G01 X20.0 R–2.5

For the direction [8]

G01 Z0 R2.5

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

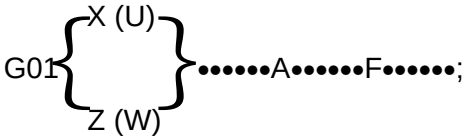
In the next block, the other one axis of X and Z, which crosses the former axis at a right angle, must be given.

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

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

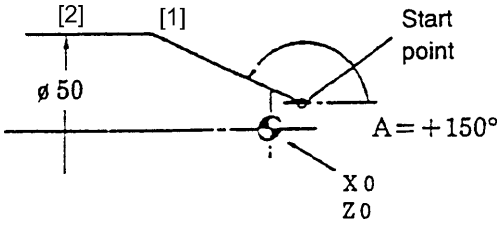
(3) Angle designated linear interpolation

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



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

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

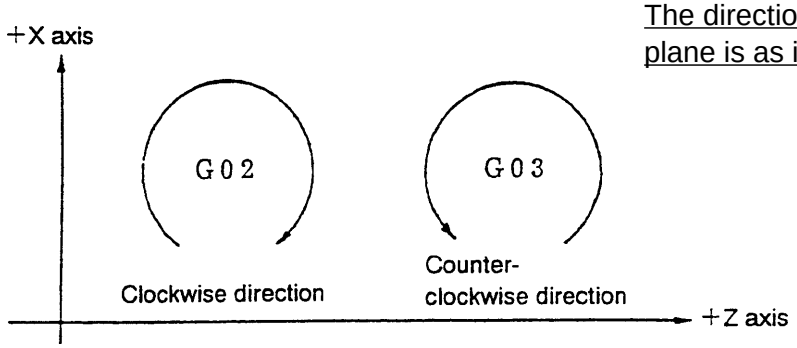


(1) G01 X50.0 A150.0 F0.3;

(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

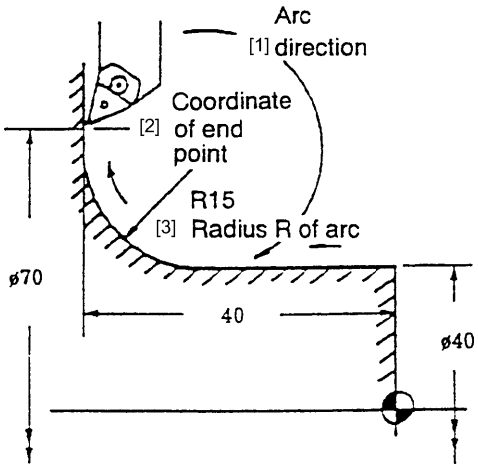
[3] Circular arc radius R (radius designation)

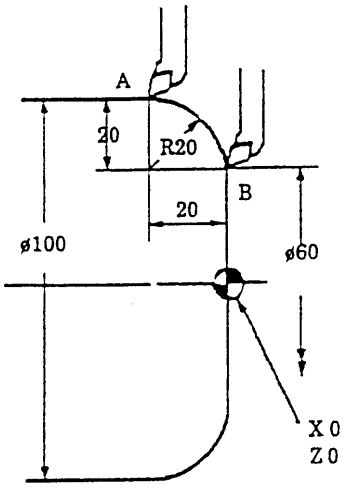
Example : G01 Z–25.0

G02 X70.0 Z–40.0 R15.0

↑ ↑ ↑

[1] [2] [3]





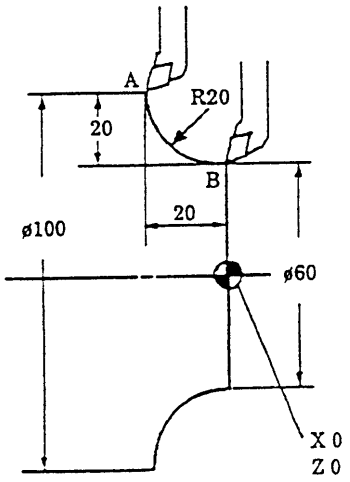
(Example 1)

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...;



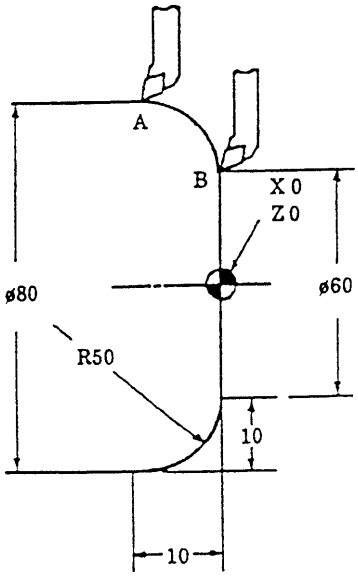
(Example 2)

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...;



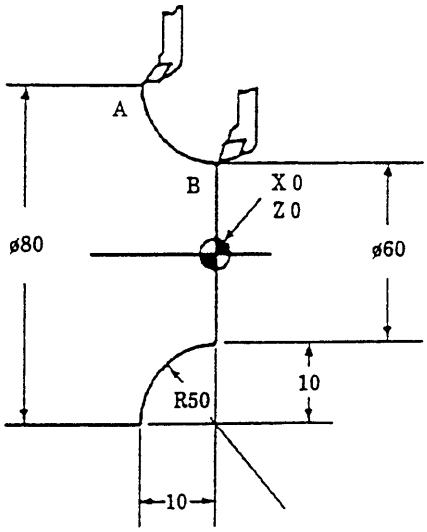
(Example 3)

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...;



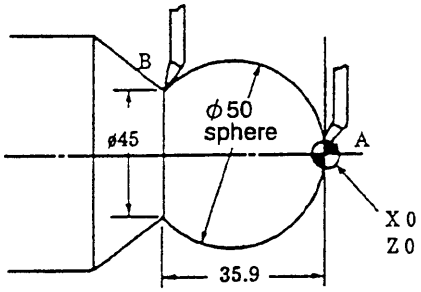
(Example 4)

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

A02 X80.0 Z–10.0 R50.0 F...;



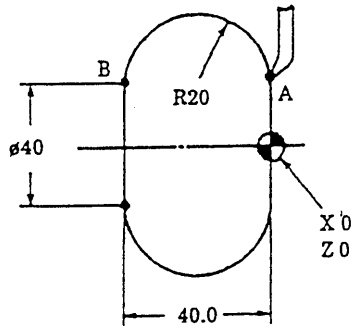
(Example 5)

When moving from the point A to the point B

G03 X45.0 Z–35.9 R25.0 F...;

When moving from the point B to the point A

G02 X0.0 Z0 R25.0 F...;



(Example 6)

When moving from the point A to the point B

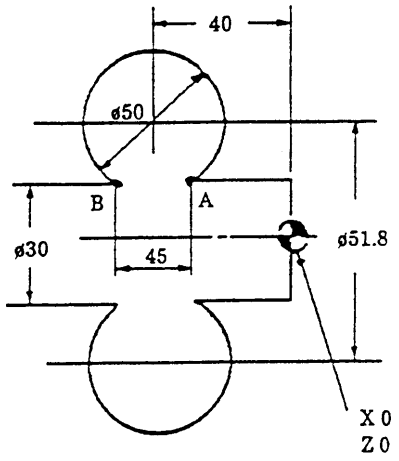
G03 X40.0 Z–40.0 R20.0 F...;

When moving from the point B to the point A

G02 X40.0 Z0 R20.0 F...;

• Circular command exceeding 180°

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



When moving from the point A to the point B

G03 X30.0 Z−62.5 R−25.0 F...;

When moving from the point B to the point A

G02 X30.0 Z−17.5 R−25.0 F...;

Cutting feed rate

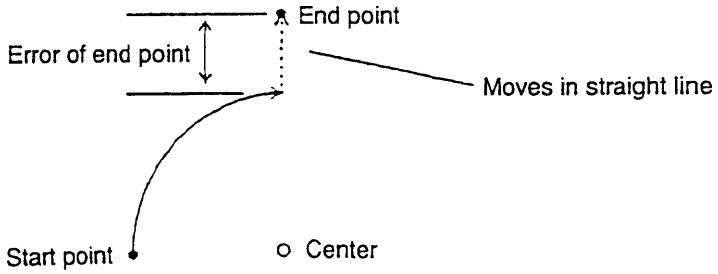
The cutting feed rate commanded by F code becomes the speed that a tool moves on a circular arc.

***Note**: 1) When F code has not feed commanded in G02 and G03 blocks or before that, an alarm will occur.*

2) Exponent type acceleration/deceleration is engaged.

3) When radius of circular arc = 0 is commanded, an alarm will occur.

4) If the end point is not located on the circular arc, the tool will move on the remainder in straight line after moving in circular, when an error of the end point of circular interpolation is within the parameter setting value. And when it is out of the parameter setting value, an alarm will occur.

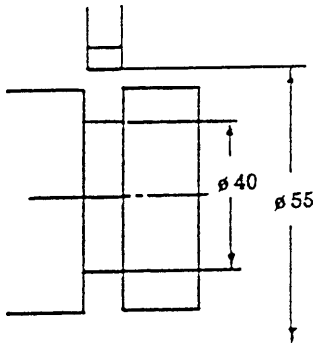


*5) When **I**, J, K and R are commanded in the same block, R has priority.*

2-3-6 G04 Dwell

A tool can be rested during a command time.

(Example)



When stopping the tool for 2 seconds

G04 U2.0;

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

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

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

Therefore, stop the tool for 0.1 second or more.

```
G01 X40.0 F...;
G04 U0.2        In this case, the feed was made
                interrupted for 0.2 second.
    X55.0 F...;
```

2-3-7 G09 Exact Stop

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

Only commanded block is effective.

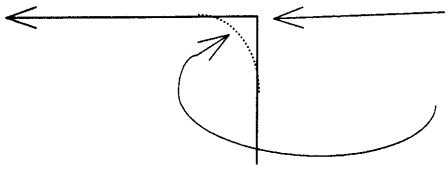
(1) Command form

G09 —— ;

(2) Program example

N1 G09 G01 U50. F ;

N2 G01 W _ 50. ;



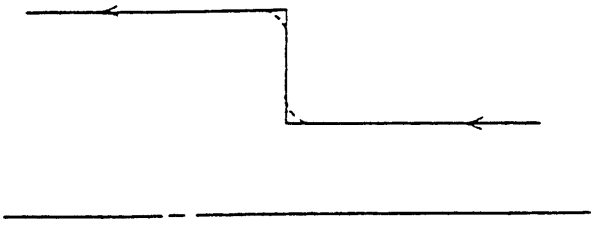
When commanding G09, an edge is created on the corner.

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

2-3-8 G61 Exact Stop

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

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



G61 mode effective :

An edge is created on the corner.

G61 mode ineffective :

A round is created on the corner.

2-3-9 G10 Programmable Date Input

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

(1) Work shift amount input

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

P00 : Work shift amount input designation

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

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

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

Don’t perform work shift for other axes.

(Example) G10 P00 Z512.368;

(2) Form offset amount input

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

L10 : Form offset input designation

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

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

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

R : Tool nose R (Absolute)

Q : Virtual tool nose point (0 ~ 9)

H : Tool width (Absolute)

(3) Wear offset amount input

G10 L11 P_ X (U)_ Z (W)_ R_ H _ ;

L11 : Wear offset amount input designation

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

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

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

R : Tool nose R (Absolute)

H : Tool width (Absolute)

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

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

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

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

Command form

G20 ; Input unit is inch input

G21 ; Input unit is metric input

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

(a) Feed rate command by F (E is included for thread cutting).

(b) Commands related to positions.

(c) Work reference point shift amount.

(d) Tool offset amount.

(e) A part of parameters.

(f) The unit of one graduation of the manual pulse generator.

(1) The G20/G21 command shall be commanded to the head of the program in the single block.

(2) When the G20/G21 command is executed, conduct the coordinate system preset.

(3) This function is for selecting the unit of numerical value programmed either in metric or in inch.

Inch ⇔ Metric conversion isn't performed.

2-3-11 G22, G23 Stored Stroke Limit

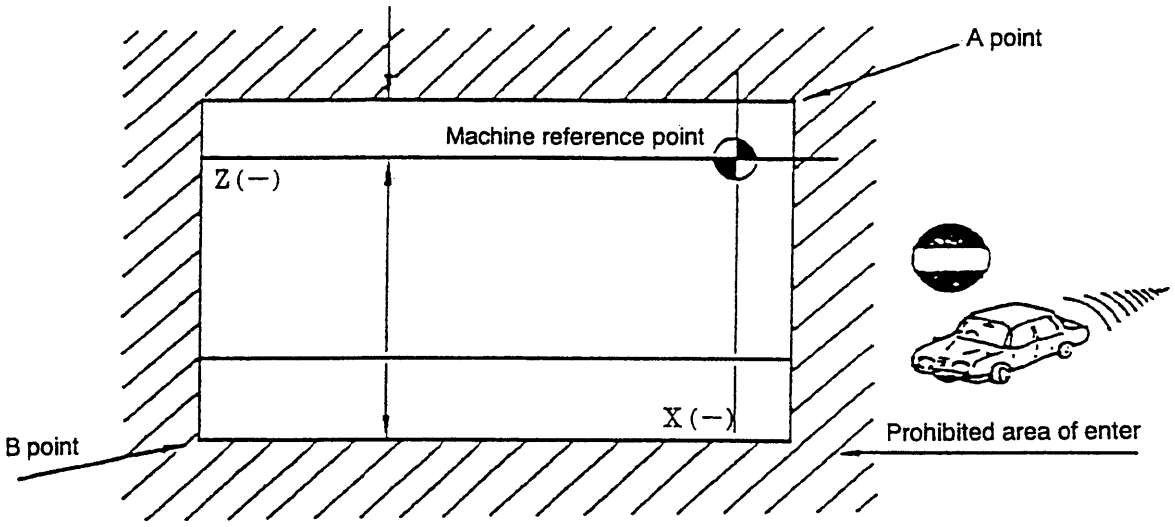
This machine is provided the stored stroke limit, which can be set the entering prohibition of tool in the movable area (Within the machine stroke) of the machine for safety operation by whether automatic or manual operation, as standard feature.

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

1. The first prohibited area

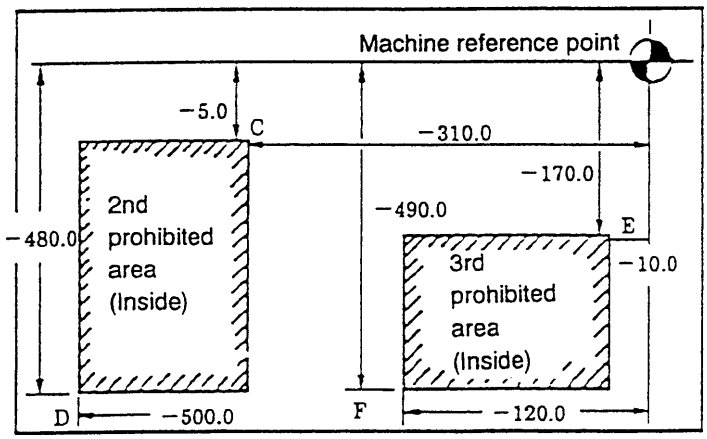
   This is set the maximum stroke of the machine by the parameter and not changeable.

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



2. The 2nd, 3rd prohibited area

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



(1) Selection of prohibited area

A prohibited area can be selected by the parameter No.1300 to close which side of in or outside of a frame determined by the points C, D and E, F.
Usually, the inside becomes the prohibited area in the second area. In the third area, always the inside is prohibited.

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

(2) Setting of area by parameter and confirmation

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

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

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

Example:

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

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

Example:

G23; Entering is possible into the second area.

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

*2) When G22 X_ Z_ **I**_ K_ ; is commanded, the parameter changes automatically to the commanded value.*

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

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

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

The reset button should be pressed after an escape.

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

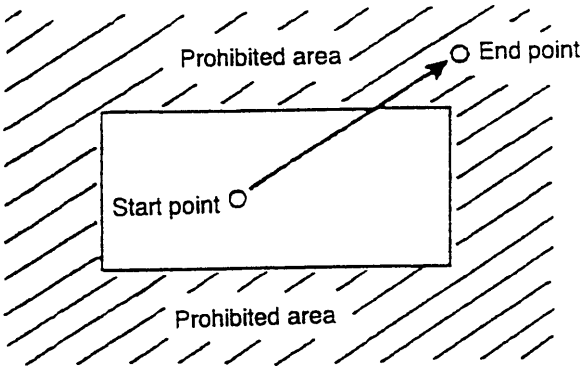
If the middle of the way to the end point to travel is the prohibited area, it becomes an alarm when entering into the prohibited area and stop the automatic operation.

(4) Setting in other than program command

Refer to Operation Manual.

2-3-12 Stroke Limit Check Before Move

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



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

(1) If the alarm of stroke limit before move is issued, release the alarm by pressing the rest button.

(2) The end point of executing block can be calculated by the “Machine coordinate” + “Remaining amount of travel” at this time.

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

(3) Precautions

(a) Concerning a traveling path of block, a check is not executed.

(b) Concerning an axis which is machine lock condition, a check is not executed.

(c) Checking of a block of G31 is not executed.

(d) Check the axis which has completed the reference point return only.

(e) If the end point locates very close to the prohibited area, it becomes an alarm occasionally.

2-3-13 G27 Reference Point Return Check

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

(1) Form of command

G27 X_ Z_ ..... ;

(2) Program example

G27 X100.0 Z–50.0 ;

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

(3) Precautions

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

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

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

G27 P_ X_ Z_ ..... ;

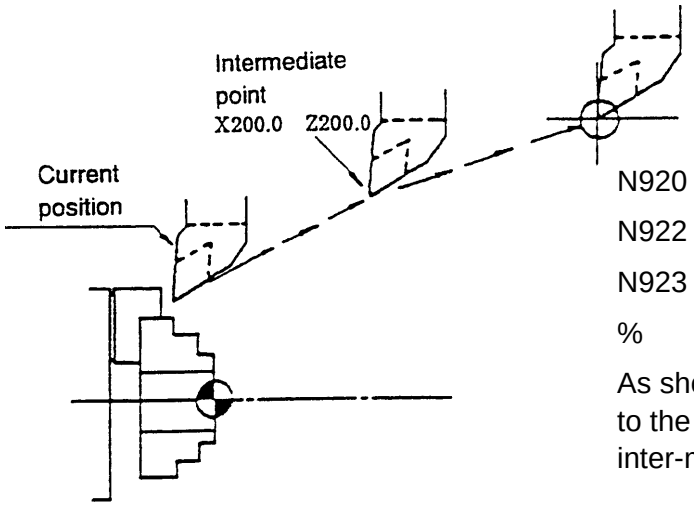
(Designate it according to the 2nd ~ 4th reference point.)

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

(e) G27 command checks an imposition after positioning.

2-3-14 G28 Automatic Reference Point Return

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




```
N920 G00 X.....
N922 G28 X200.0 Z200.0
N923       M30
%
```

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

***Note**) Difference from "G28 U0 W0 ;"*

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

2-3-15 G30 2nd Reference Point Return

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

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

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

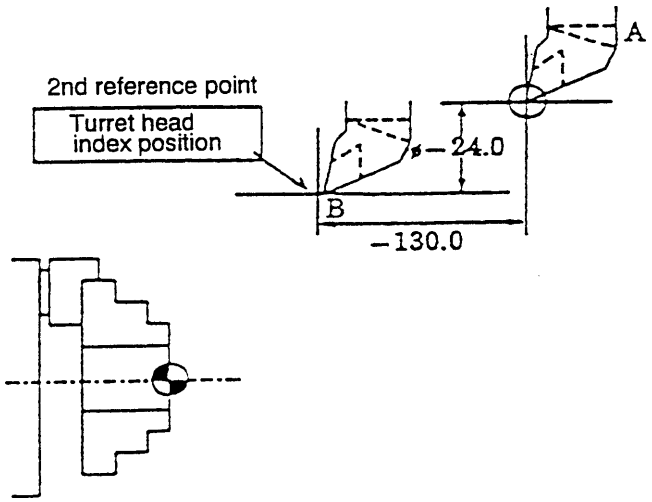
Exactly same motion as the automatic zero return by G28 is executed except returning to the 2nd reference point by the parameter setting.

Program example

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

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

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

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



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

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

(2) The third, fourth reference point

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

(Pn=P2, P3, P4)

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

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

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

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

(3) Position of each reference point

The position of each reference point is set previously by

the parameter { P2 : No.1226~ / P3 : No.1227~ / P4 : No.1228~ } 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.

2-3-16 G31 Skip Function

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

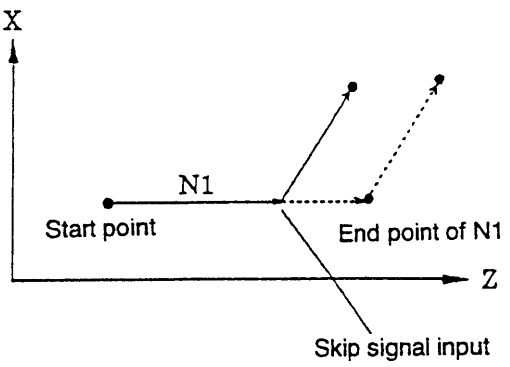
(1) Form of command

G31 X_ Z_ • • • F_ ;

(2) Program example

N1 G98 G31 W50. F100 ;

N2 G01 U50. W25. ;



——— Skip signal has been input.

– – – – Skip signal has not been input.

(3) Caution

(a) The federate set in the parameter can be obtained regardless F of the program by parameter setting.

(b) G31 cannot be commanded during nose R compensating mode.

(c) When the next block of G31 is commanded by an increment, the next block moves incrementally from the interrupted position by a skip signal.

(d) G31 is effective only for the commanded block.

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

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

G54 Z_____

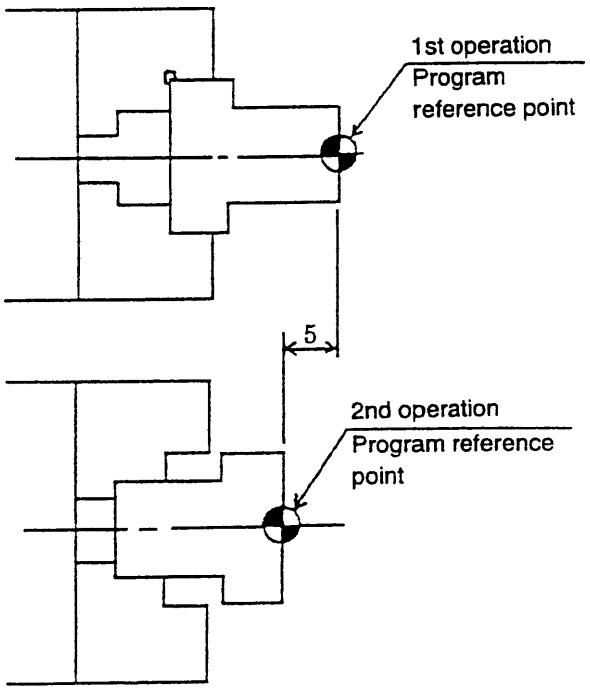
Correct distance is displayed of the tool position from the machine origin by following procedures.

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

An incremental amount of the work length can be designated by G54 W_ command.

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

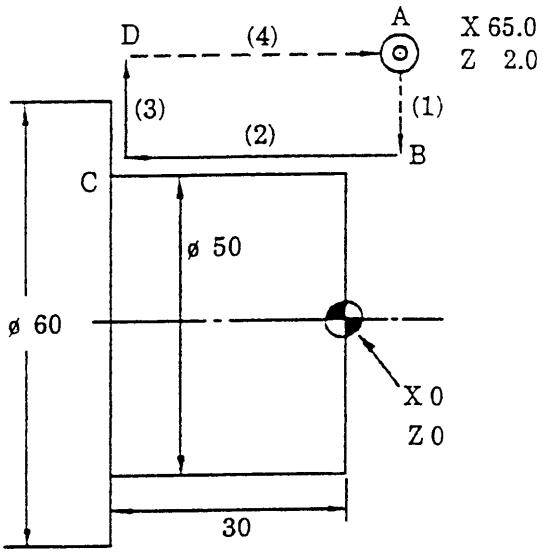
(Example)




```
O××××
G28 U0
G28 W0
G54 Z0 ..... Reference point shift amount cancel
G50 S2000
G30 U0 W0
M01

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

G28 U0
G28 W0
G54 Z–5.0 .... Difference from the finishing end of
                the work at 1st operation
G50 S1500
G30 U0 W0
M01

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

2-3-18 Canned Cycle

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



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

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

Regular Program

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

Program with Canned Cycle

G90 X50.0 Z–30.0 F;

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

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

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

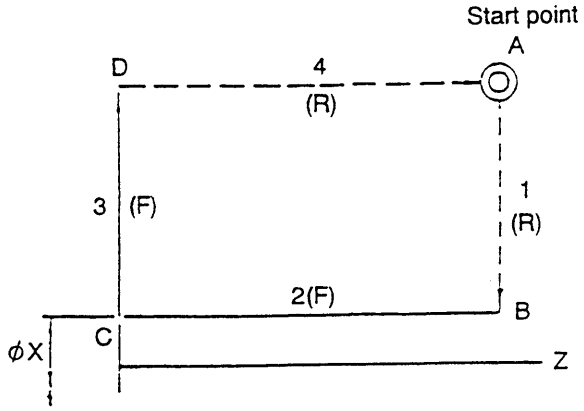
1. **G90 OD/ID cutting cycle**

G90 enables OD/ID straight cutting or taper cutting.

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

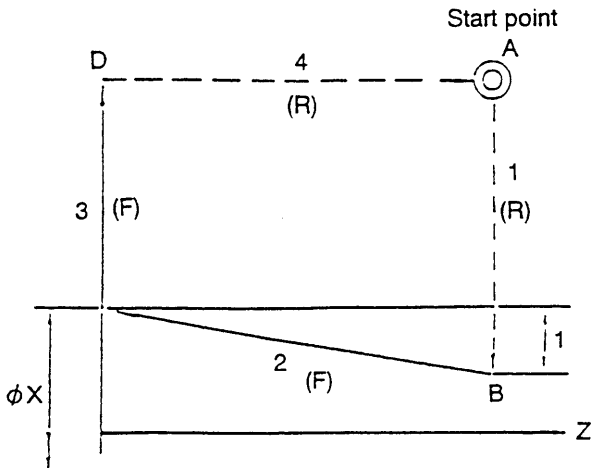
G90 cycle patterns

(1) Straight cutting



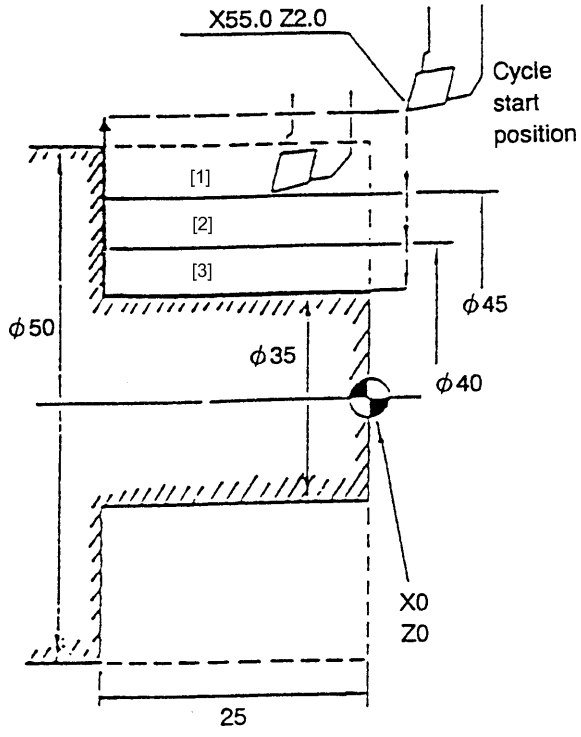
G90 X...Z...F...; (I=0)
R : Rapid traverse
F : Cutting feed
(specified with an F code)

(2) Taper cutting



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

1. Example of straight cutting



When machining a ϕ50 blank as shown in the left figure, with its start point at X55.0 and Z2.0 and a depth of cut of 2.5 mm, the program is as follows:

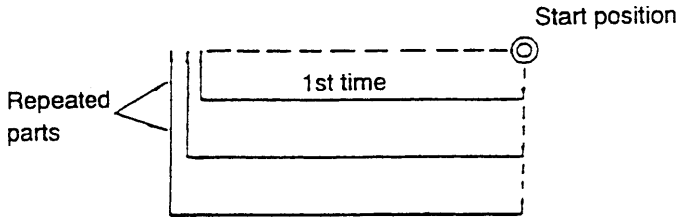
```
N101      T0100 M40
N102 G97  S695 M08
N103 G00  X55.0 Z10.0 M03
N104 G01  G96 Z2.0 F2.5 S120
N105 G90  X45.0 Z–25.0 F0.35—[1]
N106      X40.0 ————————[2]
N107      X35.0 ————————[3]
N108 G00  G97 X200.0 Z200.0 S695
N109      M01
```

Note *1) As G90 is modal, once it is specified, it can be neglected from the next block. Accordingly, cycle operation is executed by only specifying the cutting depth of X-axis from the next block on.*

2) After completing the canned cycle, cancel G90 with another G code, such as G00 belonging to the same group.

3) For the T, S and M functions which serve as cutting conditions, be sure to specify them in a block preceding the one where G90 is to be specified.

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




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

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

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

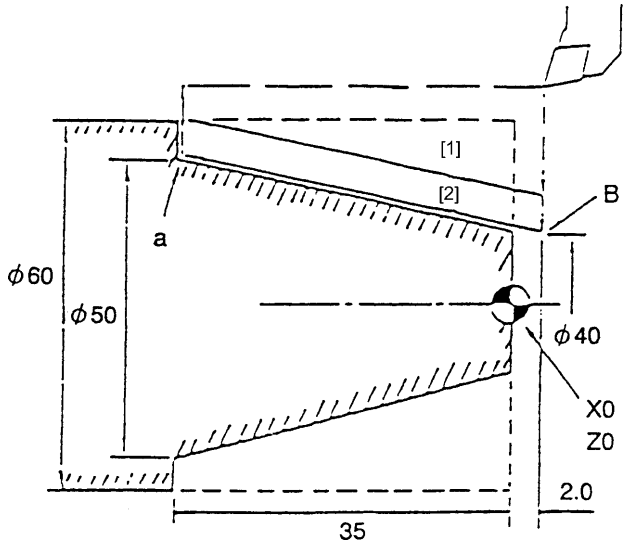
2. Example of taper cutting

When machining a ϕ60 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. $I=\frac{50-40}{2}=5$ mm

The sign of I("+" or "–") is determined as a direction from point "a" to the point B.

Accordingly; I=–5.0

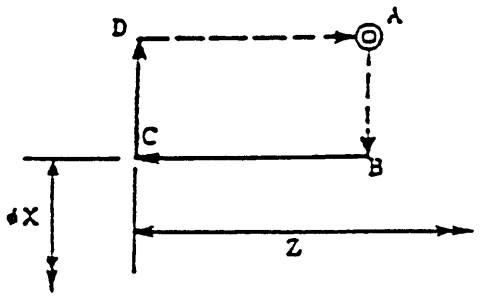
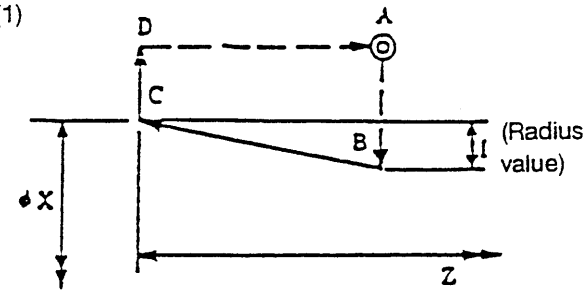
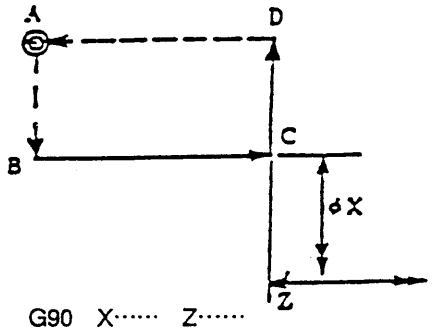
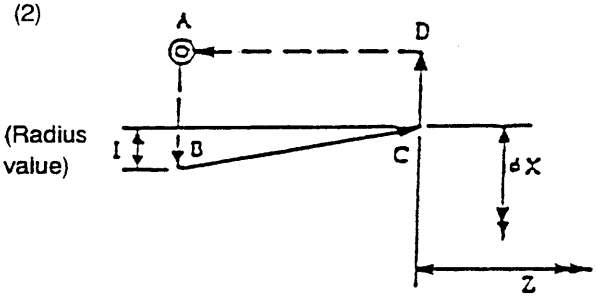
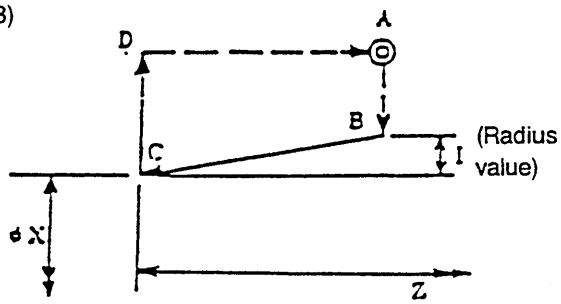
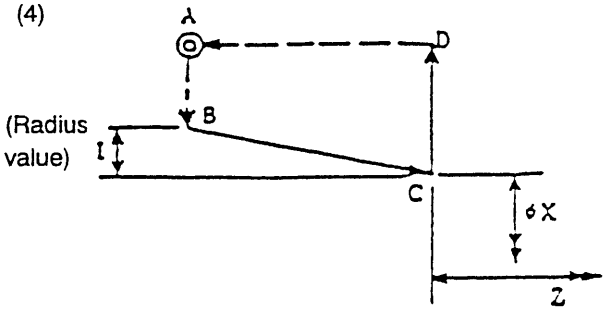



```
N104 G01 G96 X65.0 Z2.0 S120
N105 G90 X60.0 Z–35.0 I–5.0 F0.3 → [1]
N106       X50.0 → [2]
N107 G00 X..... Z.....
```

or

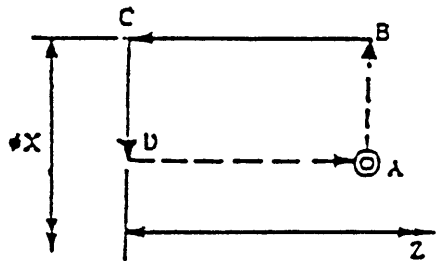
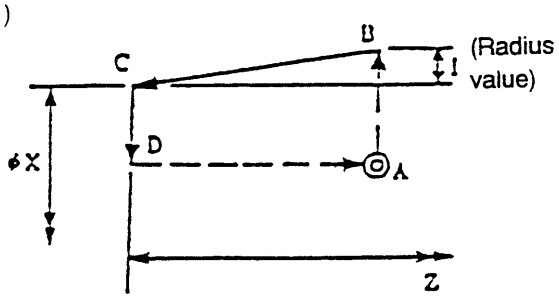
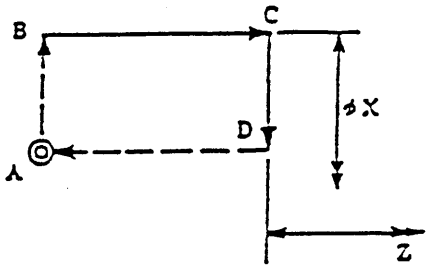
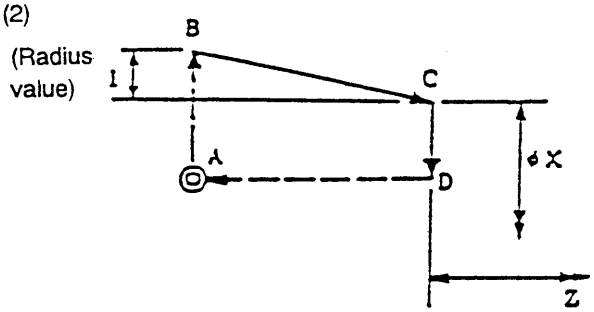
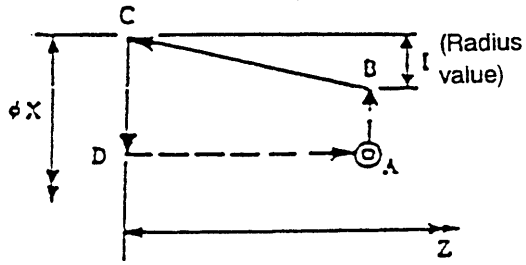
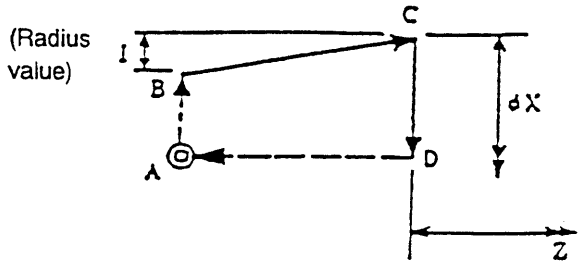
```
N104 G01 G96 X65.0 Z2.0 S120
N105 G90 X60.0 Z–35.0 I–5.0 F0.3 → [1]
N106 G00 X55.0
N107 G90 X50.0 Z–35.0 I–5.0  →[2]
N108 G00 X..... Z.....
```

G90 Cycle Patterns (OD)

Straight	Taper
 G90 X····· Z·····	(1)  G90 X····· Z····· I (−) ·····
 G90 X····· Z·····	(2)  G90 X····· Z····· I (−) ·····
	(3)  G90 X····· Z····· I·····
	(4)  G90 X····· Z····· I

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
(1)  G90 X····· Z·····	(1)  G90 X····· Z····· I·····
(2)  G90 X····· Z·····	(2)  G90 X····· Z····· I·····
	(3)  G90 X····· Z····· I (−) ·····
	(4)  G90 X····· Z····· I (−) ·····

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

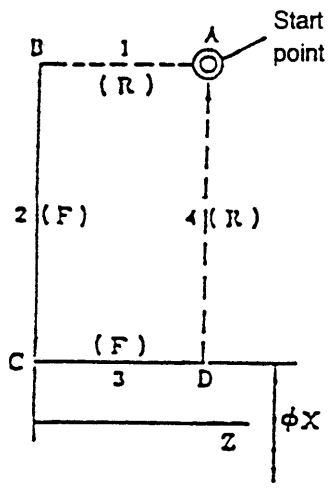
2. **G94 End face and side cutting cycle**

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

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

G94 cycle patterns

(1) Straight cutting

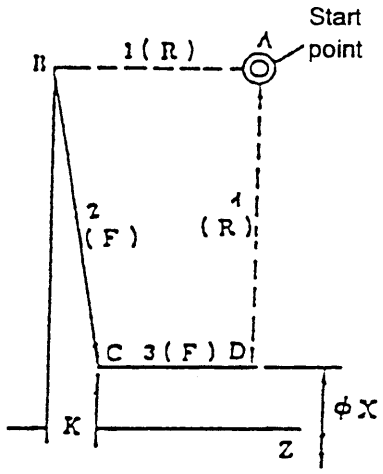


G94 X...Z...F...

(K=0)

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

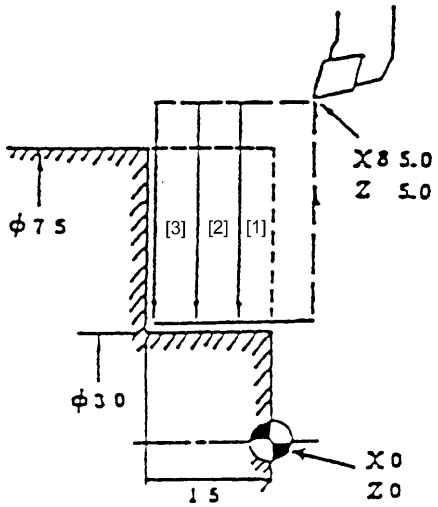
(2) Taper cutting



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

(Pay attention to a sign of K)

1. Example of straight cutting



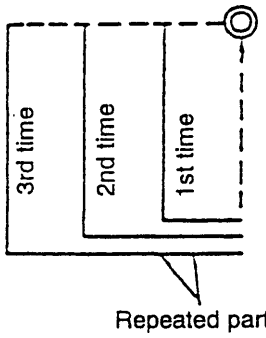
When machining a ϕ75 blank as shown in the left figure, with its cycle start position at X85.0 and Z5.0 and a depth of cut of 5 mm, the program is as follows:

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

Note *1) Since G94 is modal, specify it just once. You do not have to specify it again thereafter. Accordingly, cycle operation is executed by only giving Z-axis depth of cut from the next block on.*

2) After completing the canned cycle, cancel G94 with another G code, such as G00, belonging to the same group.

3) For the T, S and M functions which serve as cutting conditions, be sure to specify them in a block preceding the one where G94 is to be specified.



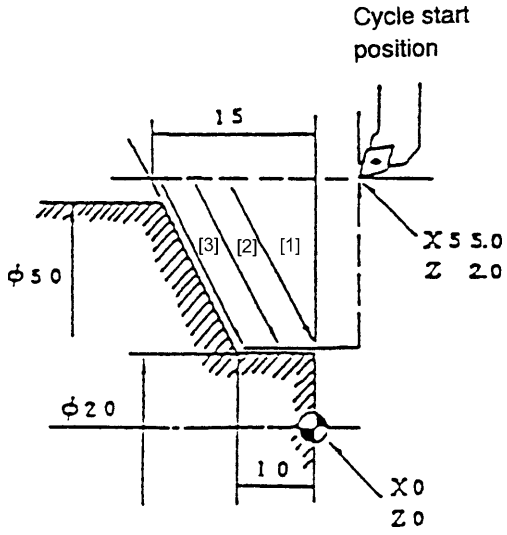
N105 G94 X30.0 Z–5.0 F0.2 ..... [1]
N106 G00 Z–3.0
N107 G94 X30.0 Z–10.0 ..... [2]
N108 G00 Z–8.0
N109 G94 X30.0 Z–15.0 ..... [3]
N110 G00 X.... Z....

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

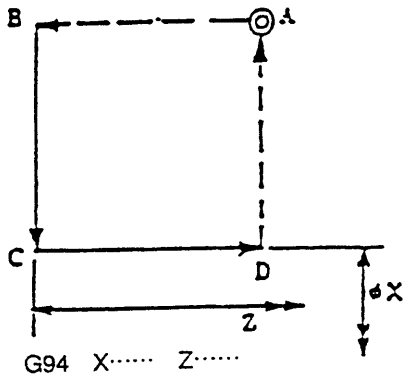
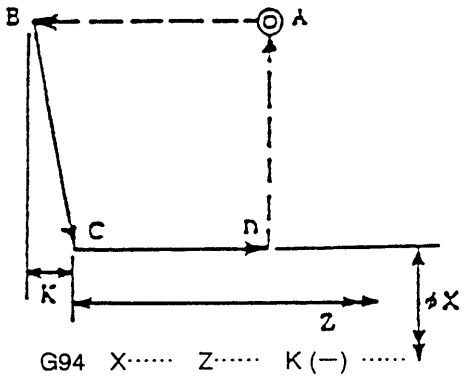
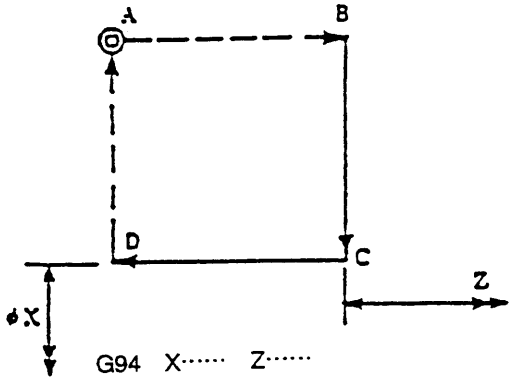
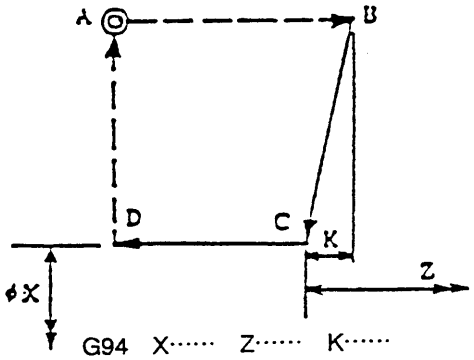
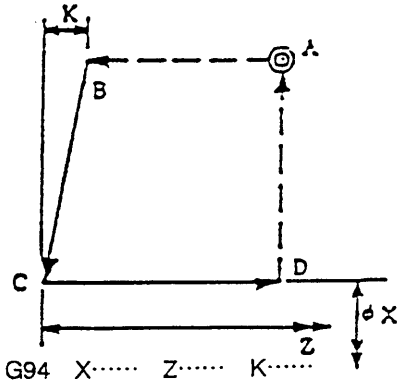
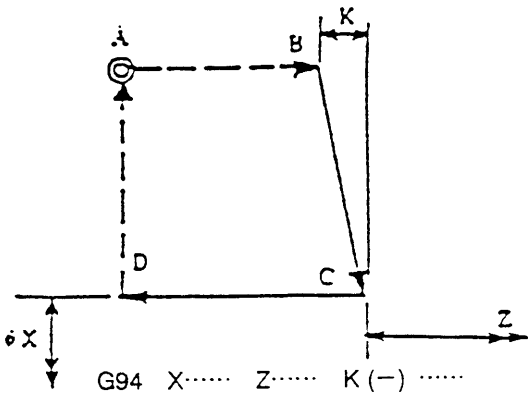


When machining a ϕ50 blank as shown in the left figure, with its cycle start position at X55.0 and Z2.0 and a depth of cut of 5 mm, the program is as follows:
First, obtain a size of K.

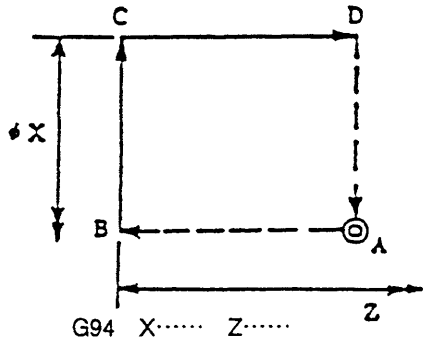
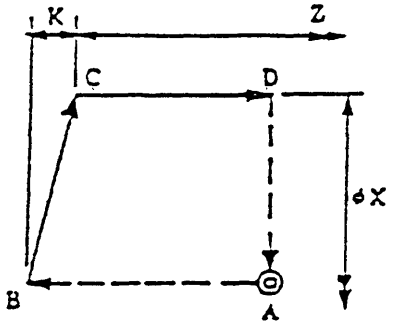
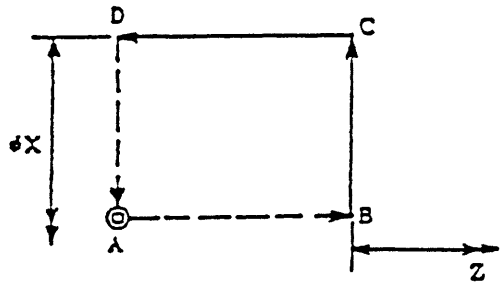
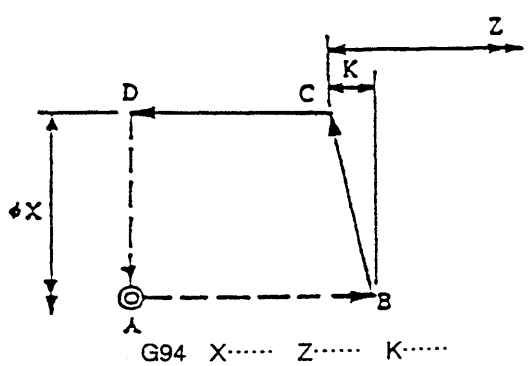
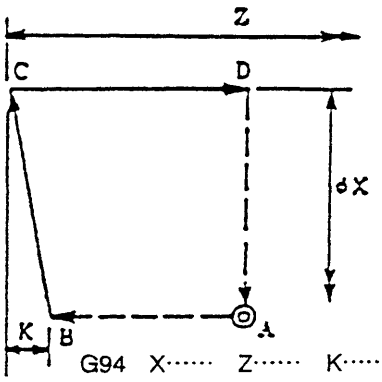
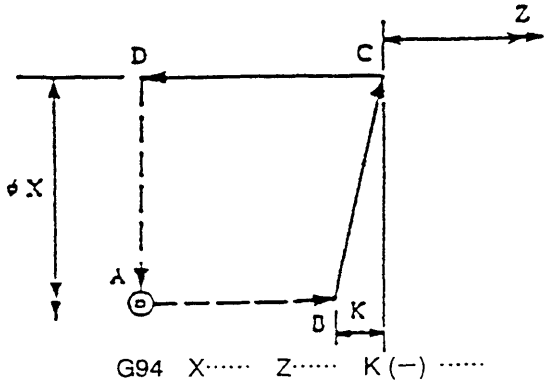
K = 15 – 10 = 5
Determine a sign of K, viewing its cycle pattern.

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

G94 Cycle Patterns (OD)

Straight	Taper
(1)  G94 X····· Z·····	(1)  G94 X····· Z····· K (−) ·····
(2)  G94 X····· Z·····	(2)  G94 X····· Z····· K·····
	(3)  G94 X····· Z····· K·····
	(4)  G94 X····· Z····· K (−) ·····

G94 Cycle Patterns (ID)

Straight	Taper
(1)  G94 X····· Z·····	(1)  G94 X····· Z····· K (−) ·····
(2)  G94 X····· Z·····	(2)  G94 X····· Z····· K·····
	(3)  G94 X····· Z····· K·····
	(4)  G94 X····· Z····· K (−) ·····

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

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

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

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

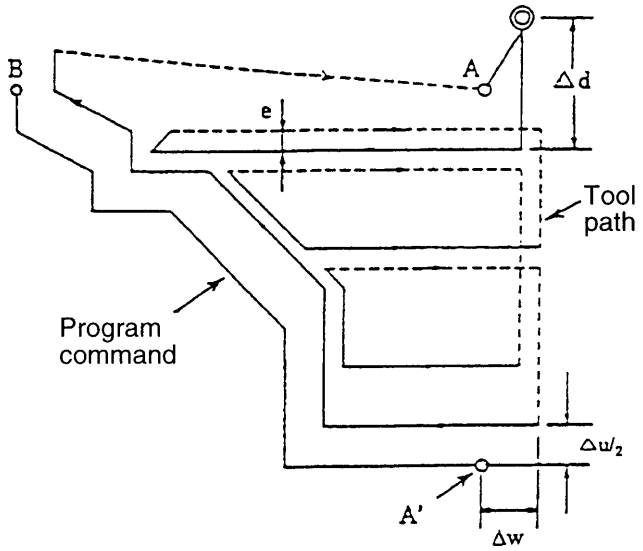
1. The G codes above are all unmoral ones belonging to the * group.
2. Finish shape programs specified by G71, G72 and G73 are stored in the NC unit's internal memory. Its maximum capacity is 45 blocks which store a set of programs.
3. When chamfering or corner R is commanded, it is equivalent to the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

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

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

(1) Type Ⅰ

As shown in the figure below, if a finish shape between A and B via A' with a tape command, the tool cuts away a section specified with the depth of cut Δd, leaving the finishing allowances ΔU/2 and ΔW.



- An escape after turning retract at 45° direction.
- Retract amount e = 0.5mm

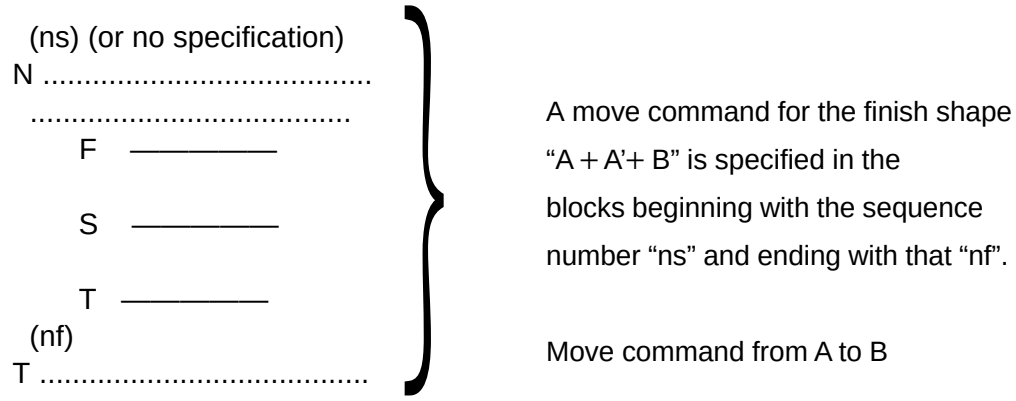
  Designate it at the parameter No.5145.

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

Create the tape command as follows:

•G71P (ns) Q (nf) U±__ W±__ I±__ K±__ D__ F__ S__ ;

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



P : First sequence No. (ns) of a group of finishing profile blocks

Q : Last sequence No. (nf) of a group of finishing profile blocks

U : X-directional finishing allowance and direction (ΔU) ... Diameter designation (ΔU/2 for radius designation)

W : Z-directional finishing allowance and direction (ΔW)

D : Depth of cut (Δd). Specify with no sign.

I : X directional rough finishing allowance and direction (ΔI).. radius value

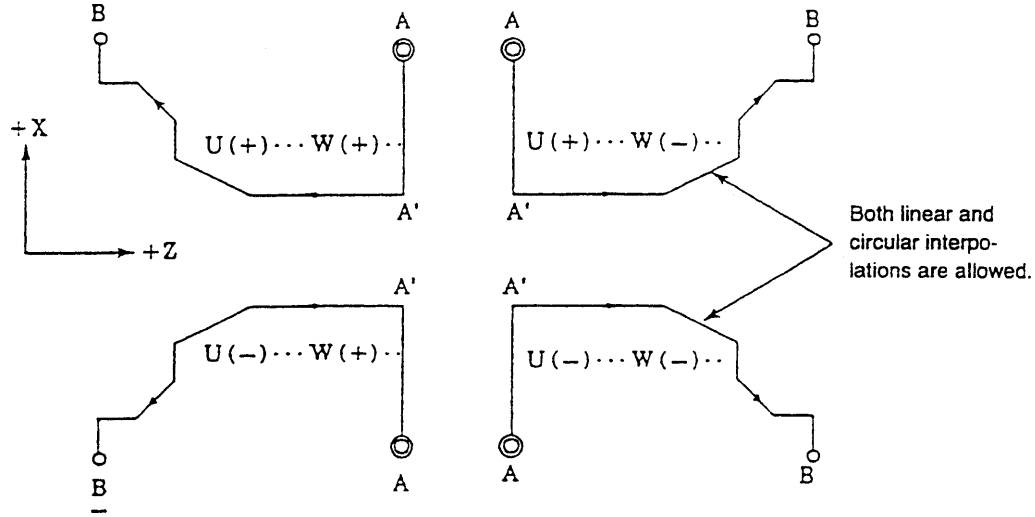
K : Z directional rough finishing allowance and direction (ΔK).

F, S : F and S function in any block within sequence No. P and Q in the cycle is ignored and those in the block of G71 or designated before G71 are effective. Also, in case of constant surface speed control, G96 or G97 during move command between A and B is ignored and those in the block of G71 or designated before G71 are effective.

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

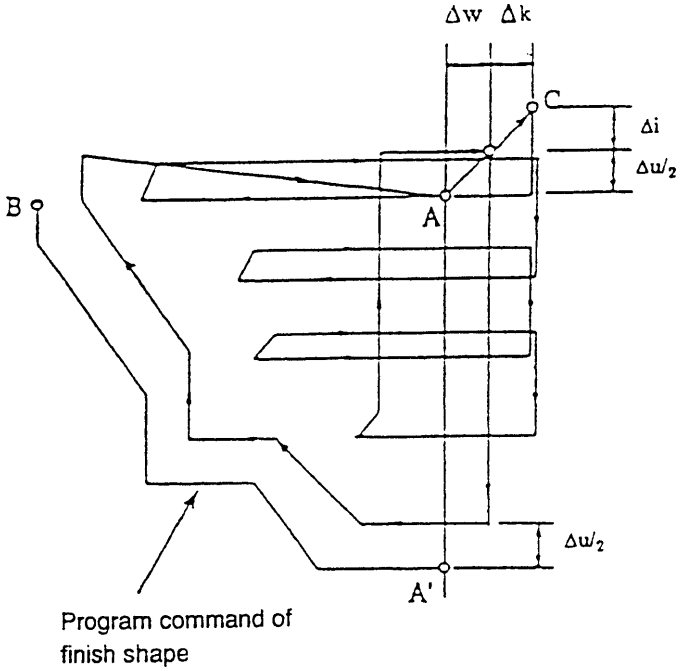
In any case, the workpiece is cut by tool movements in parallel with the Z axis of the tool. Signs of ΔU and ΔW are as follows:

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



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

Execution of rough finishing cycle



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

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

(2) Type II

Type II differs from type I in the following points.

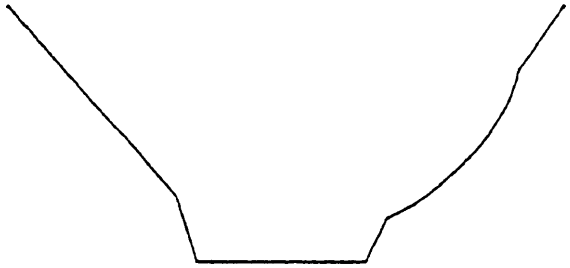
(i) The shape is not necessary to be simple increase in X direction and it may have as many pockets as possible.

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

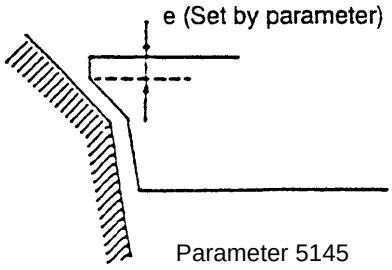
However, Z direction must be simple change.

The following shape cannot be cut.

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

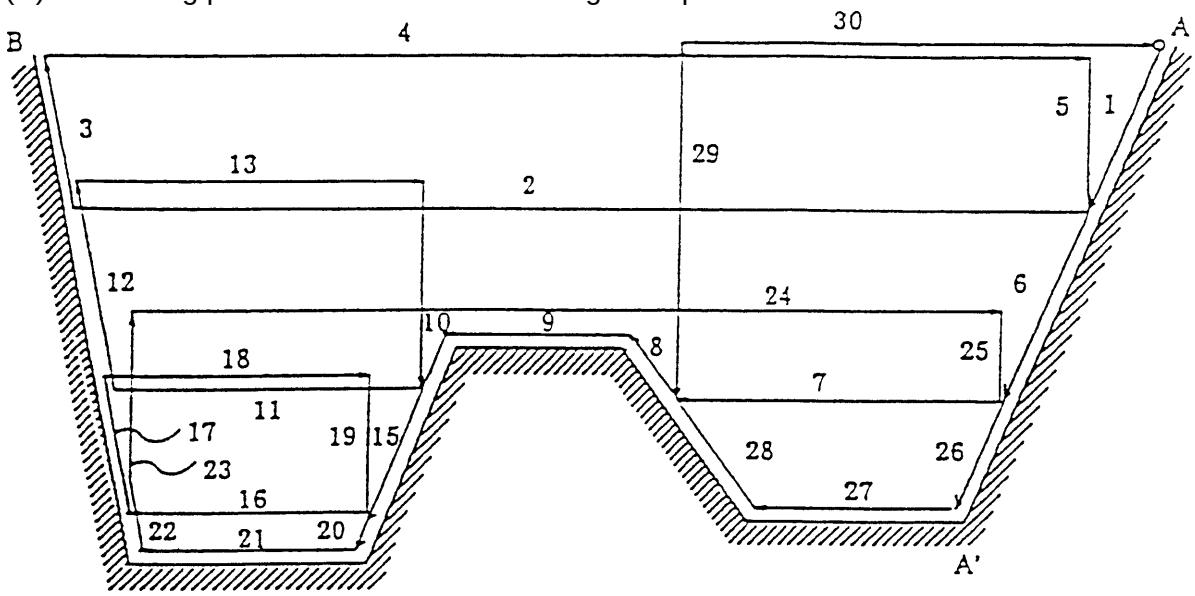


(iii) The release after turning is performed on straight line after turning along the shape. (e)



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

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



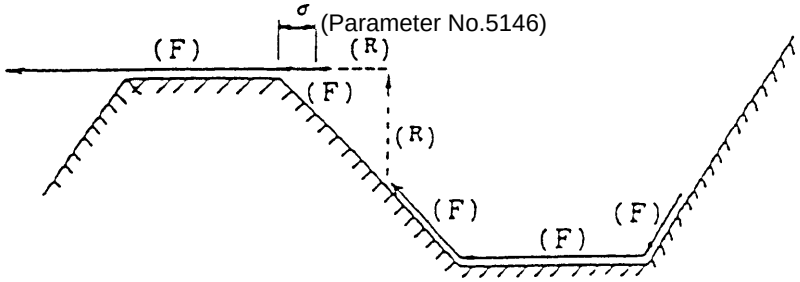
Between A and A' is commanded in the block with sequence No. (ns) and should be included the Z-axis command.

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

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

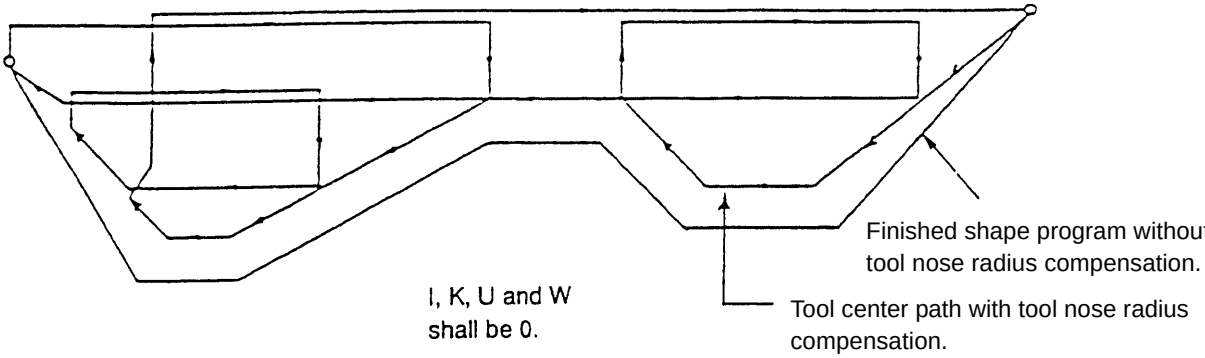
If Z-axis motion is included between A and A', indeed is done by G01.

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



(a) Coexistence of nose R compensation

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



(b) Execution of rough cutting finishing cycle

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

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

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

•G72P (ns) Q (nf) U±__ W±__ I±__ K±__ D__ F__ S__ ;

P : First sequence No. (ns) of the finishing pattern block

Q : The last sequence No. (nf) of the finishing pattern block

U : Finishing allowance in the X direction (Δu)

W : Finishing allowance in the Z direction (Δw)

I : Cutting allowance in the X direction for the rough cutting and finishing cycle (Δi)

K : Cutting allowance in the Z direction for the rough cutting and finishing cycle (Δk)

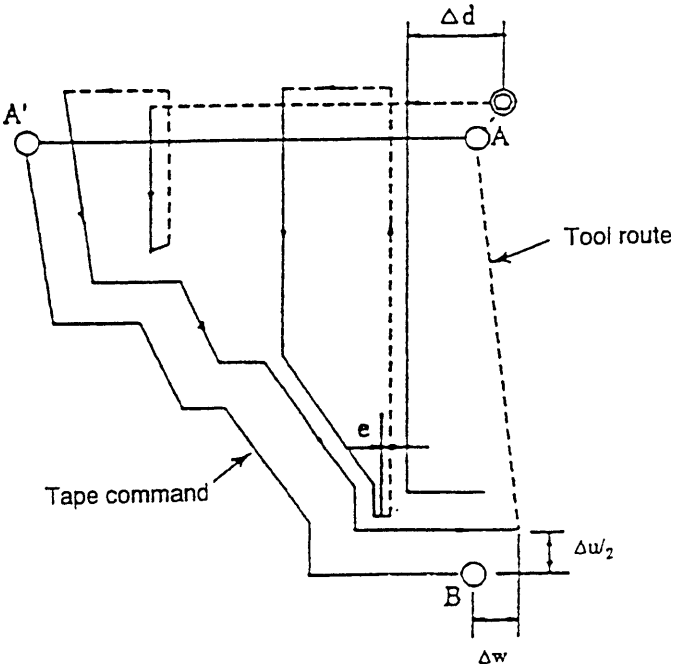
D : Cutting depth (Δd) ..... To be commanded without symbol

e : Release amount (parameter No.6213)

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

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

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



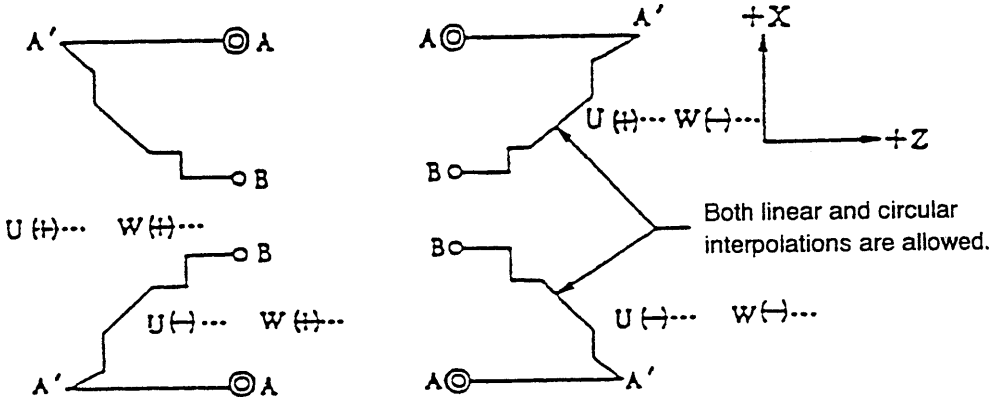
- An escape after turning retract at 45° direction.
- Retract amount e = 0.5mm

  Designate it at the parameter No.5145.

(1) Type I

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

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



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

(2) Type II

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

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

(a) Cutting shape

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

However, must be kept simple change in X direction.

(b) Heading block of finish shape program

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

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

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

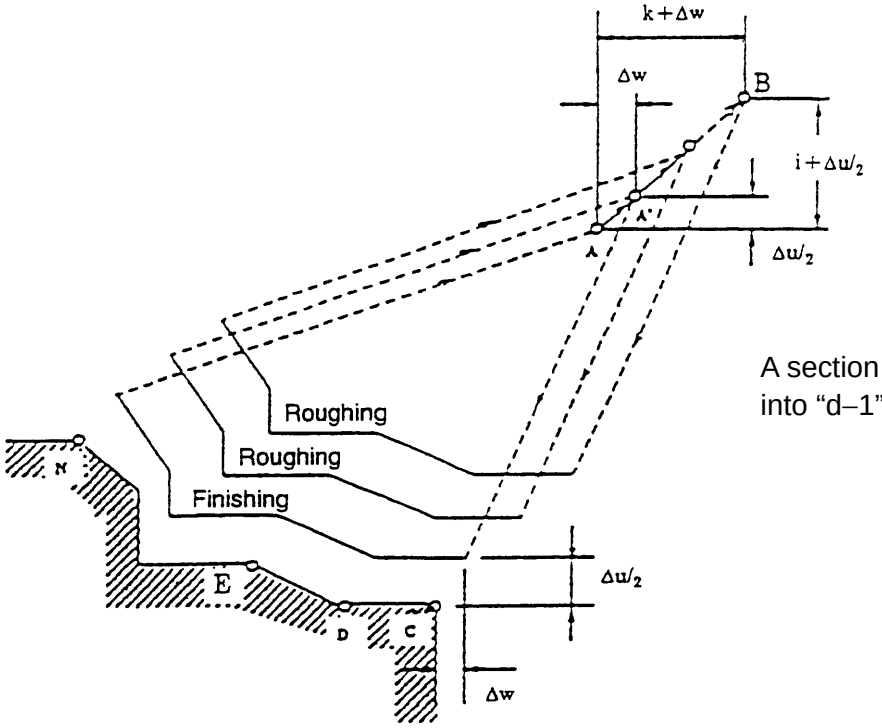
2) Execution of rough finishing cycle

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

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

3. G73 Closed loop cutting cycle (Option)

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



A section from A' to B is divided into "d–1".

Pattern Specified by Tape

Point A →Point C →point D →Point E ..... →point N

Create a tape command as follows:

•G73P (ns) Q (nf) I±__ K±__ U±__ W±__ D__ F__ S__ ;

(ns)

N.....
.....
(nf)
N.....

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

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

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

I : X directional all roughing allowance (Δi) ..... Radius designation

K : Z directional all roughing allowance (Δk)

U : X directional finishing allowance (ΔU) ..... Diameter designation

W : Z-directional finishing allowance (ΔW)

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

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

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

4. G70 Finishing cycle

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

N __ G70 P (ns) Q (nf) ;

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

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

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

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

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

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

  G71P100Q200...;

  N100...

  ...

  ...

  N200

  G71P300Q400...;

  N300

  ...

  ...

  N400

  ...

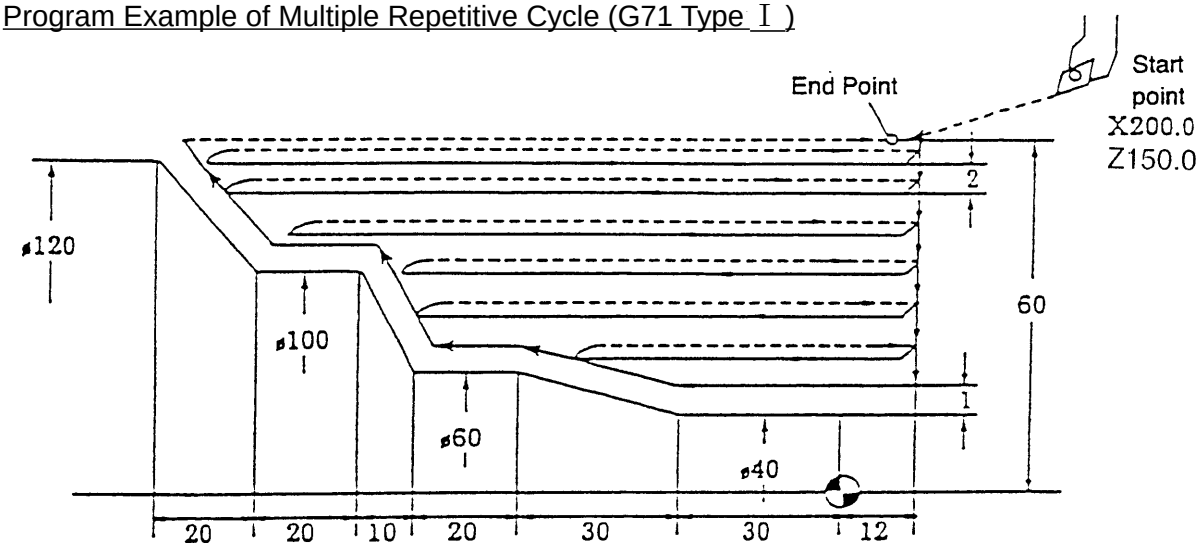
  ...

  G70P100Q200; Up to 3 is executed without search.

  G70P300Q400; Over 4 is executed after searching.

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

Program Example of Multiple Repetitive Cycle (G71 Type Ⅰ)



N100 (OD-R)

N101 T0100;

N102 G97 S240 M08;

N103 G00 X120.0 Z10.0 M03;

N104 G96 S120;

N105 G71 P106 Q112 U2.0 W2.0 D2.0 F0.3;

N106 G00 X40.0 F0.15;

N107 G01 Z–30.0;

N108 X60.0 Z–60.0 ;

N109 Z–80.0;

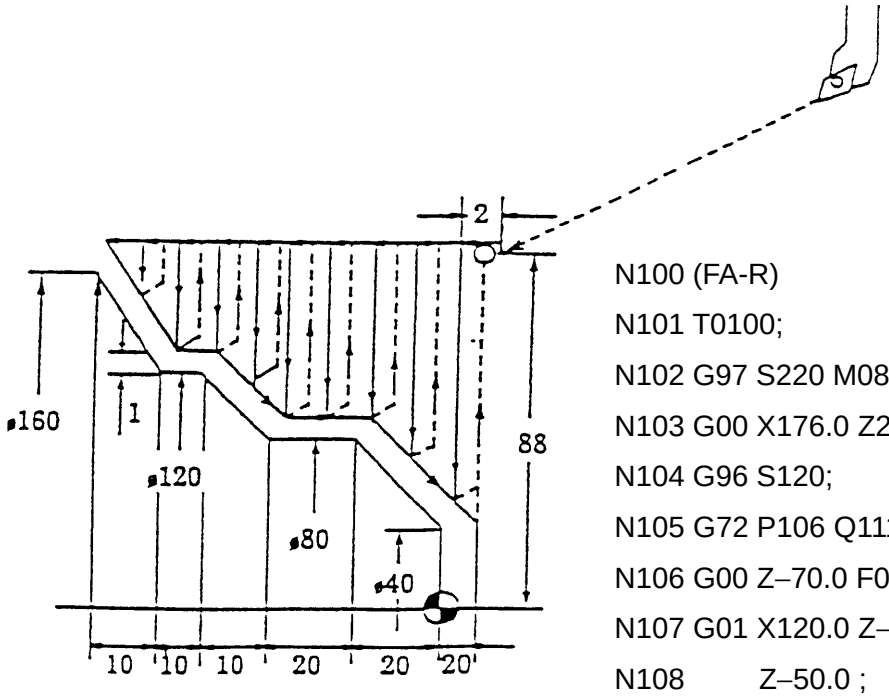
N110 X100.0 Z–90.0;

N111 Z–110.0;

N112 X120.0 Z–130.0;

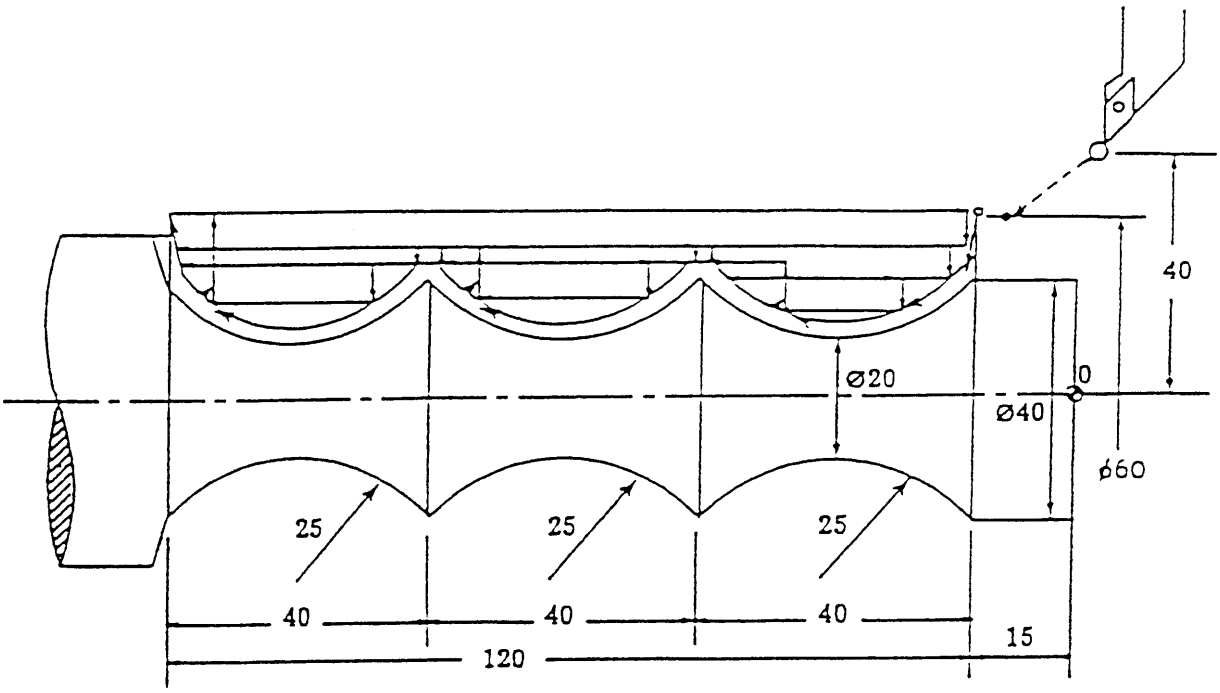
N113 G00 G97 X200.0 Z150.0 S500;

Program Example of Multiple Repetitive Cycle(G72)




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

Program Example of Compound Canned Cycle (G71 Type II)



N010 T0300;

N011 G97 S1650 M08;

N012 G00 X60.0 Z–15.0 M03;

N013 G71 P014 Q018 U0.5 D5.0 F0.3;

N014 G01 X40.0 W0 F0.15;

N015 G02 Z–55.0 R25.0;

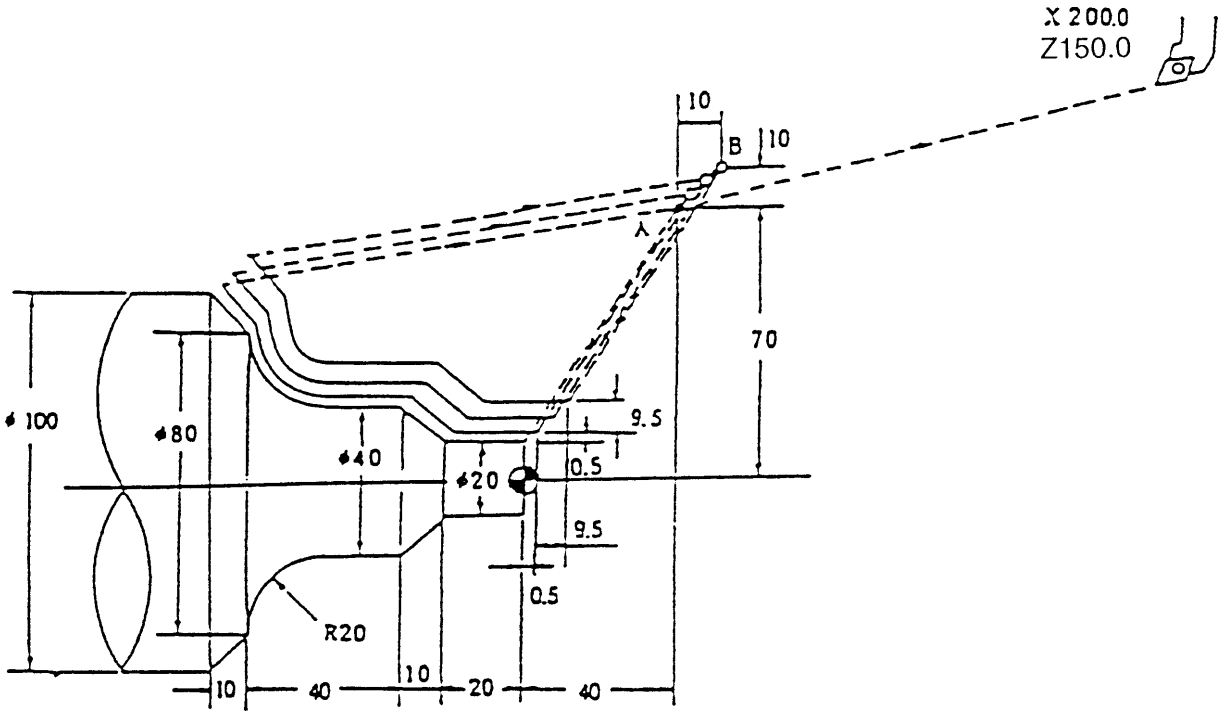
N016 Z–95.0 R25.0;

N017 Z–135.0 R25.0;

N018 G01 X60.0;

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

Program Example of Multiple repetitive Cycle (G73)



N101 T0100;

N102 G97 S200 M08;

N103 G00 X140.0 Z40.0 M03;

N104 G96 S120;

N105 G73 P106 Q112 I9.5 K9.5 U1.0 W0.5 D3 F0.3;

N106 G00 X20.0 Z0;

N107 G01 Z–20.0; F0.15 S150;

N108 X40.0 Z–30.0;

N109 Z–50.0;

N110 G02 X80.0 Z–70.0 R20.0;

N111 G01 X100.0 Z–80.0;

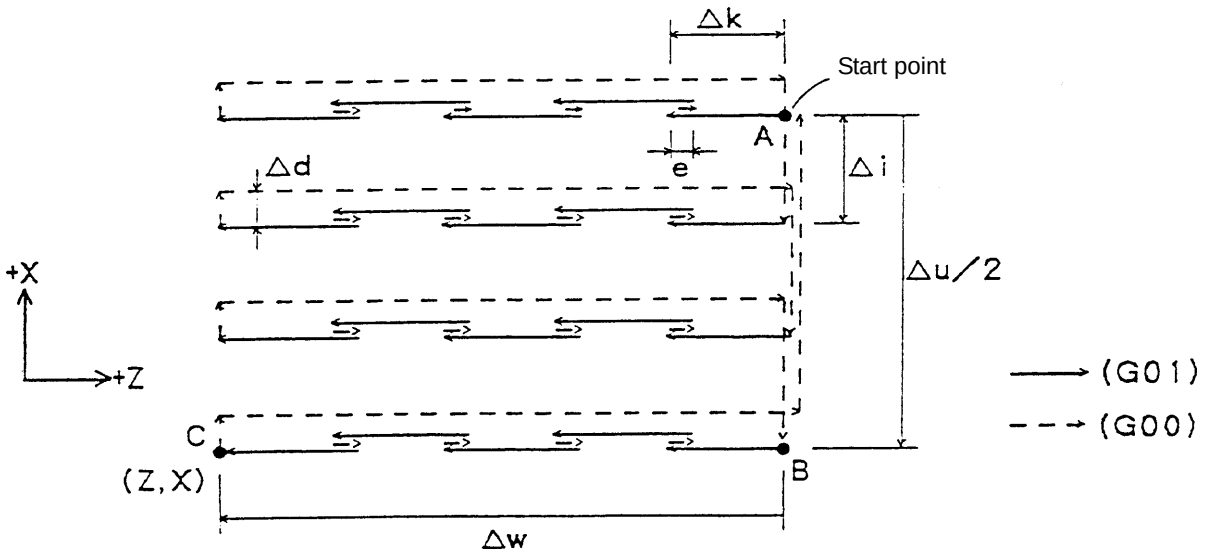
N112 X105.0;

N113 G00 G97 X200.0 Z150.0 S500

N114 M01;

5. G74 End face cutting-off cycle

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



G74 X(U)__ Z(W)__ I__ K__ D__ F__ R__ ;

X : Point C

U : A→C X-direction incremental amount (Δu)

Z : Point C

W : A→C Z-direction incremental amount (Δw)

I : One-time cutting amount in X-direction (absolute value) (Δi)

K : One-time cutting amount in Z-direction (absolute value) (Δk)

D : Tool relief amount at the cutting bottom (Δd)

F : Feed rate

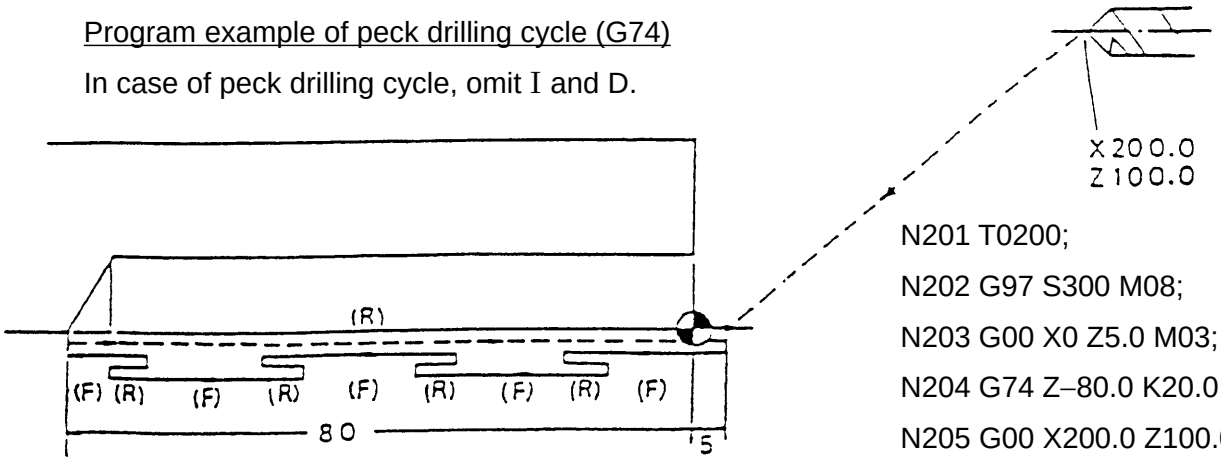
e : Return amount (parameter No.5147)

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

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

Program example of peck drilling cycle (G74)

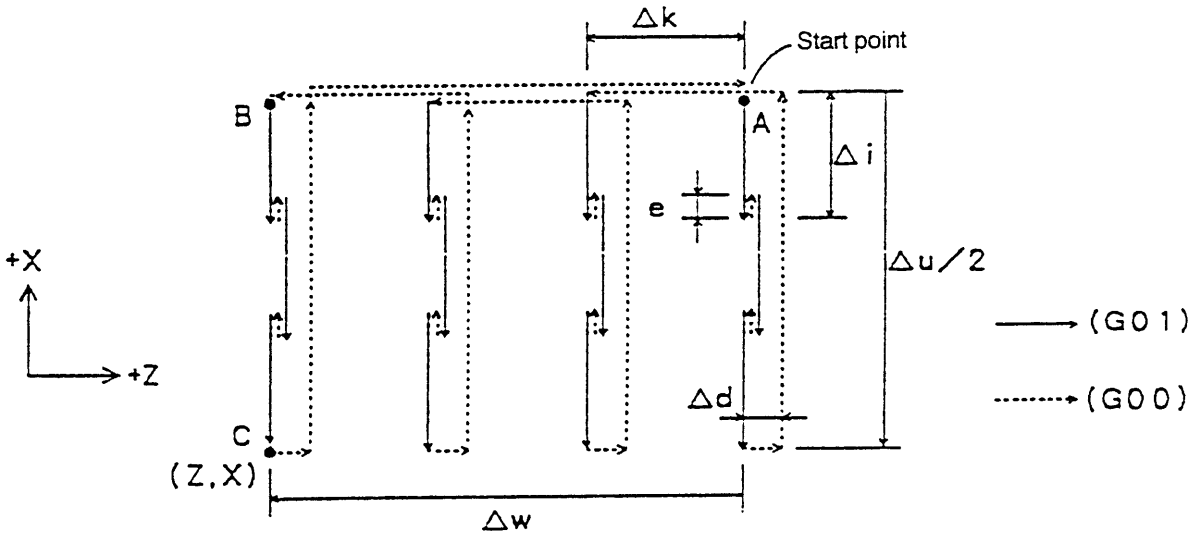
In case of peck drilling cycle, omit I and D.




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

6. Outside diameter cutting-off cycle

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



G75 X(U)__ Z(W)__ I__ K__ D__ F__ R__ ;

X : Point C

U : A→C X-direction incremental amount (Δu)

Z : Point C

W : A→C Z-direction incremental amount (Δw)

I : One-time cutting amount in X-direction (absolute value) (Δi)

K : One-time cutting amount in Z-direction (absolute value) (Δk)

D : Tool relief amount at the cutting bottom (Δd)

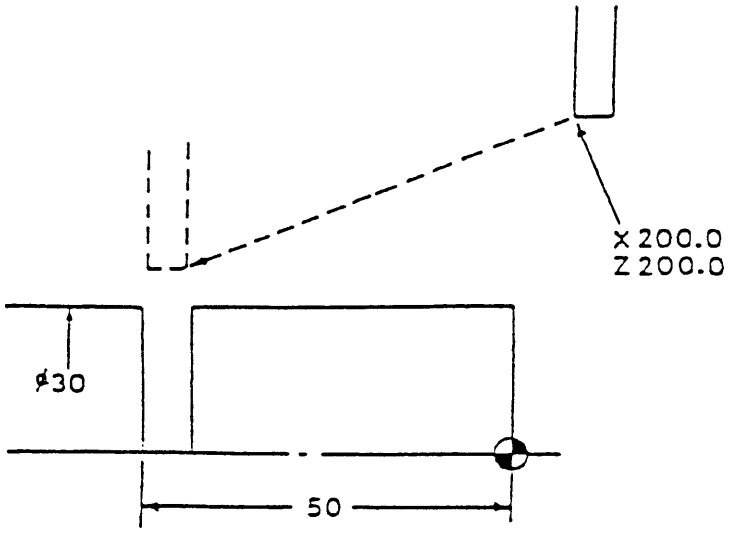
F : Feed rate

e : Return amount (parameter No.5147)

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

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

Program example of OD grooving cycle (G75)




```
N1101 T1100;
N1102 G97 S700 M08;
N1103 G00 X35.0 Z–50.0 M03;
N1104 G96 S80;
N1105 G75 X–1.0 I5.0 F0.15;
N1106 G00 G97 X200.0 Z200.0 S500;
N1107       M01;
```

Precautions for Multiple Repetitive Cycles(G70-G76)

(1) In the blocks where multiple repetitive cycles are to be specified, you must correctly specify necessary parameters, such as P, Q, X, Z, U, W, I, K, D and A, block by block.

(2) In the G71, G72 and G73 blocks with the sequence number specified with P, you must specify G00 or G01 of the group 01 without fail.

(3) It is not allowed to give a G70-G76 command by MDI.

(4) You cannot specify an M98/M99 command in the block where G70, G71, G72 or G73 was specified, and between the blocks of the sequence numbers specified with P and Q of G70, G71, G72, or G73.

(5) G codes which can be specified in the blocks excluding the sequence numbers specified with P and Q of G70, G71, G72 and G73 are G00, G01, G02, G03, G04, G09, G61, G64, G96, G97, G98, G99 and G196.

(6) Although you can make manual operation intervene in the multiple repetitive cycle (G70-G76) halfway its execution by suspending it, when resuming the multiple repetitive cycle, be sure to return the machine to the position where manual operation intervened.

If you resume the multiple repetitive cycle without returning the machine, a machine stroke by manual operation is not added to an absolute value even if the manual absolute switch is turned on, and thereafter, the machine functions with a shift of the stroke by manual operation.

(7) When executing G70, G71, G72 or G73, the sequence number-for a G70 block and those specified with P and Q should not be the same as those in the memory.

(8) In a G71 command, the X coordinate value of the start point must be equal to that of the end point of a finish shape.

(9) In a G72 command, the Z coordinate value of the start point must be equal to that of the end point of a finish shape.

(10) Do not command the program which contains chambering of rounding at the end of last moving command of finishing pattern block group with P and Q of G70, G71, G72, G73.

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

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

•Threading code and lead programmable range

Specify a lead with a numerical value following F.

G code	Description
G32	Threading
G92	Canned cycle for threading
G76	Multiple repetitive cycle for threading

Lead range

Command address	Metric input	Inch input
F	mm/rev	inch/rev
E	0.00001 ~999.99999	0.000001 ~99.999999

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

G32 Z_ E14.0

Limitation of spindle speed

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

$P \leqq \frac{5000}{N}$ P : Lead or pitch(mm)

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

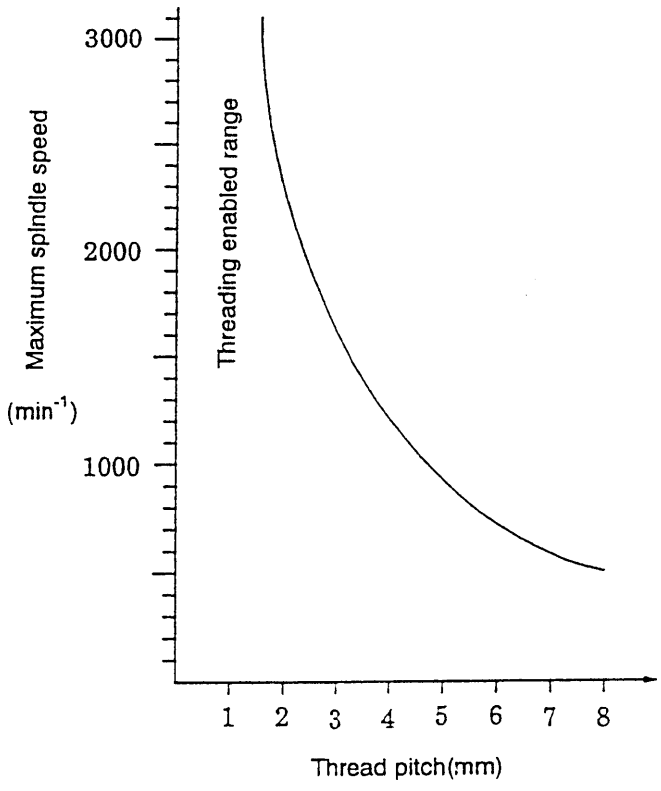
(Example)

When a thread pitch is 3 mm, a spindle speed is; (min^{-1})

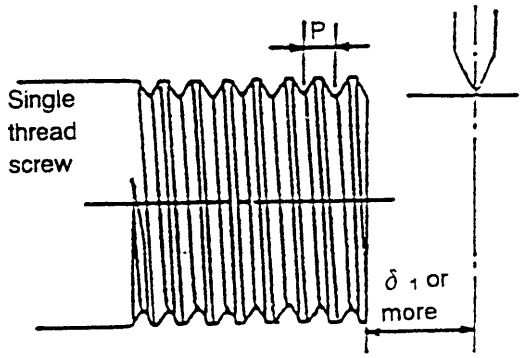
$N \leqq \frac{5000}{P} = \frac{5000}{3} = 1666\ (min^{-1})$

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

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

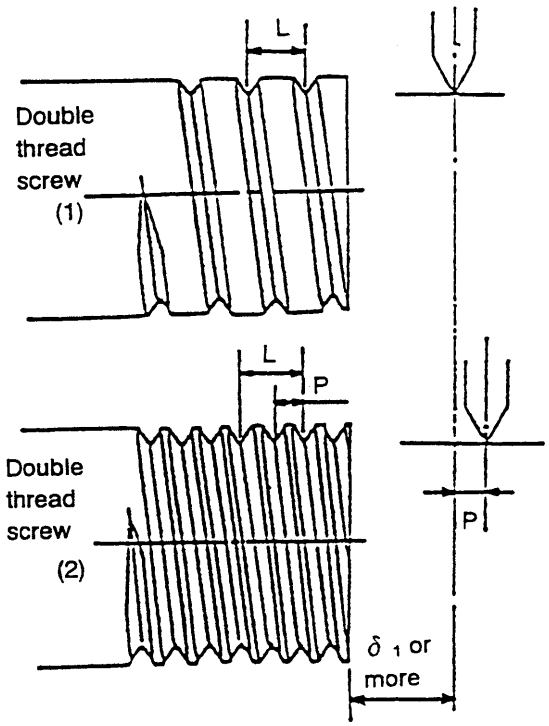


1. Cutting the single thread screw



For a single thread screw, cut at a threading feed rate of P mm/rev from an arbitrary position by δ_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 δ_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

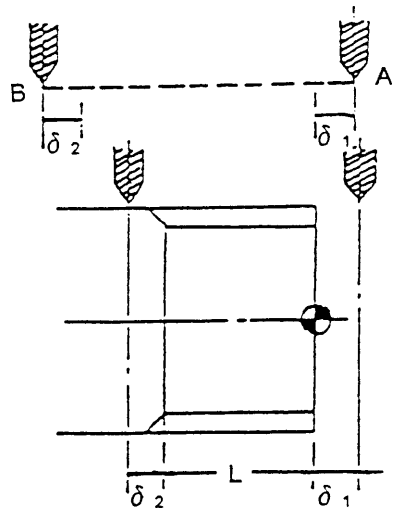
$\iota = n \cdot P \quad P = \frac{\iota}{n}$ (ι: Thread lead, P : Pitch, α: Twisting angle, n : No. of threads, β: Lead angle)

$\alpha + \beta = 90° \quad \tan\beta = \frac{\iota}{2\pi r} \quad \cot\alpha = \frac{\iota}{2\pi r}$

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

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

<Incomplete thread>



When cutting the thread from the point A to the point B, it causes shorter leads(pitches) of δ_1 and δ_2 at the cutting start point A due to acceleration and at the cutting end point B due to deceleration, respectively.
Therefore, when obtaining an effective thread length "1", a threading length of "$1+\delta_1+\delta_2$" is required.

<How to determine δ_1>

Obtain δ_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 δ_2>

Obtain δ_2 from the spindle speed and thread lead (pitch) used for threading, using the following formula:

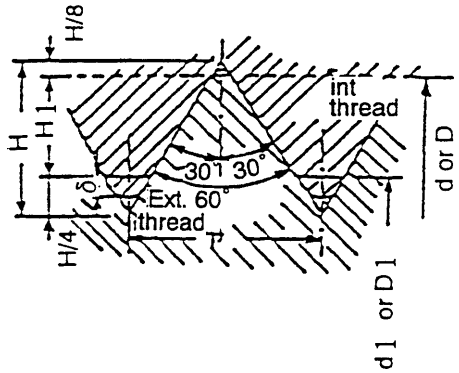
$$\delta_2 \text{ (mm)} = 0.00042 \times R(\text{min}^{-1}) \times L(\text{mm})$$

Example) When cutting a JIS Class-1 thread with a pitch of 1.5 at a spindle speed of 800min^{-1}

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

***Note**) As mentioned above, δ_1 and δ_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:

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

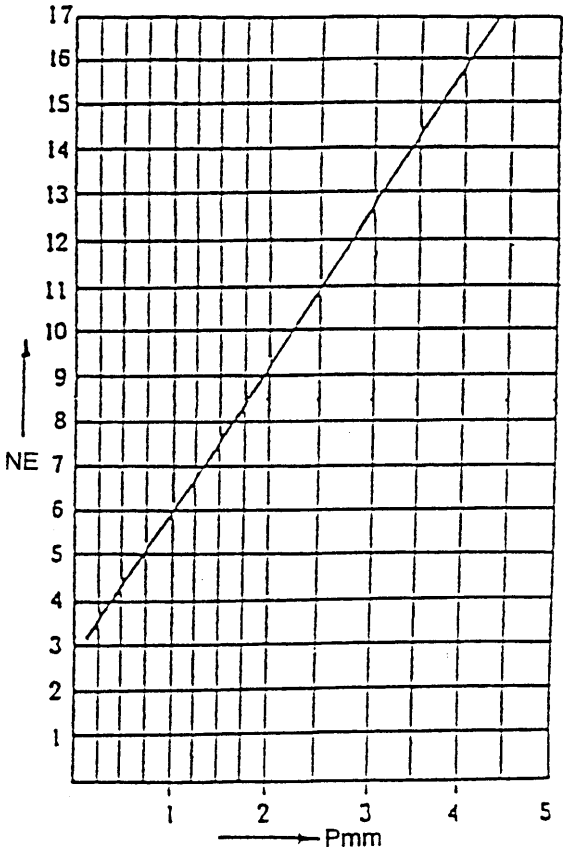
$= 3.3P + 2.5$

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

NE : No. of cutting times

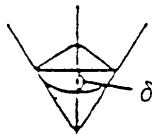
K : Constant based on various conditions (assumed to be 3.3 in this case)

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 δ and a relief cutting part to be an arc (nose R);

$$\left.\begin{array}{l}\delta = \dfrac{1}{4}H - R = \dfrac{1}{4}P\cos 30^\circ - R \quad \text{external thread} \\ \delta = \dfrac{1}{8}H - R = \dfrac{1}{8}P\cos 30^\circ - R \quad \text{internal thread}\end{array}\right\} \quad \text{..... (1)}$$



A maximum allowable value for R is;

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

external thread　　internal thread

Example) When P = 2.5, a maximum nose R is; Rmax (external thread) = 0.36, Rmax (internal thread) = 0.18

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

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

When R0.1 is used for the internal thread,

$\delta = 0.17$, 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_1	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
$d + H_1$ 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	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW	ΔX(n)/ ΔW
*1	0.45\ (0.641\)	0.45\ (0.541\)	0.5\ (0.722\)	0.5\ (0.605\)	0.55\ (0.729\)	0.55\ (0.679\)	0.6\ (0.821\)	0.6\ (0.751\)	0.65\ (0.900\)	0.65\ (0.815\)	0.7\ (1.035\)	0.7\ (0.989\)	0.75\ (1.171\)	0.75\ (0.027\)	0.8\ (1.226\)	0.8\ (1.128\)
2	0.878\ 0.124	0.765\ 0.091	1.02\ 0.150	0.856\ 0.103	1.031\ 0.139	0.96\ 0.118	1.162\ 0.162	1.066\ 0.135	1.273\ 0.180	1.153\ 0.145	1.414\ 0.206	1.398\ 0.202	1.657\ 0.122	1.453\ 0.203	1.734\ 0.270	1.595\ 0.230
3	1.075\ 0.057	0.937\ 0.05	1.25 \ 0.066	1.049\ 0.056	1.261\ 0.067	1.177\ 0.063	1.423\ 0.075	1.305\ 0.069	1.559\ 0.082	1.412\ 0.075	1.793\ 0.095	1.712\ 0.091	2.03\ 0.107	1.779\ 0.194	2.124\ 0.112	1.954\ 0.103
4	1.242\ 0.048	1.082\ 0.042	1.444\ 0.056	1.211\ 0.047	1.457\ 0.057	1.358\ 0.052	1.643\ 0.064	1.507\ 0.058	1.800\ 0.070	1.631\ 0.063	2.071\ 0.080	1.977\ 0.096	2.363\ 0.090	2.055\ 0.88	2.453\ 0.095	2.256\ 0.087
5	1.282\	1.122\	1.614\ 0.049	1.354\ 0.041	1.628\ 0.049	1.519\ 0.046	1.837\ 0.056	1.685\ 0.051	2.013\ 0.061	1.823\ 0.055	2.315\ 0.070	2.211\ 0.067	2.62\ 0.08	2.297\ 0.07	2.742\ 0.083	2.523\ 0.077
6	1.322\	1.162\	1.654\	1.394\	1.784\ 0.045	1.664\ 0.042	2.013\ 0.051	1.846\ 0.046	2.205\ 0.055	1.897\ 0.050	2.536\ 0.064	2.422\ 0.061	2.87\ 0.072	2.517\ 0.063	3.004\ 0.075	2.763\ 0.070
7			1.674\	1.434\	1.824\	1.704\	2.174\ 0.046	1.994\ 0.043	2.381\ 0.051	2.157\ 0.046	2.739\ 0.058	2.616\ 0.056	3.1\ 0.066	2.718\ 0.058	3.245\ 0.069	2.985\ 0.014
8					1.864\	1.744\	2.214\	2.034\	2.546\ 0.047	2.306\ 0.043	2.928\ 0.054	2.796\ 0.052	3.314\ 0.062	2.906\ 0.054	3.469\ 0.064	3.191\ 0.060
9							2.254\	2.074\	2.586\	2.346\	3.106\ 0.051	2.966\ 0.043	3.515\ 0.058	3.082\ 0.051	3.673\ 0.861	3.385\ 0.056
10									2.626\	2.386\	3.146\	3.006\	3.705\ 0.055	3.249\ 0.048	3.898\ 0.057	3.568\ 0.053
11											3.186\	3.046\	3.866\ 0.046	3.408\ 0.045	4.067\ 0.054	3.742\ 0.050
12													3.906\	3.448\	4.248\ 0.052	3.908\ 0.048
13													3.946\	3.488\	4.422\ 0.050	4.068\ 0.046
14															4.462\	4.108\
15															4.502\	4.148\

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

When Cutting Straight (External Thread)

G	X	Z	F	Remarks
G00	X...	Z...		
G92	X9.55	ZΔΔ.ΔΔ	F1.0	d–ΔX(1)= 10 – 0.45 = 9.55
	X9.12			d–ΔX(2)= 10 – 0.878 = 9.122
	X8.92			d–ΔX(3)= 10 – 1.075 = 8.925
	X8.76			d–ΔX(4)= 10 – 1.242 = 8.758
	X8.72			d–ΔX(5)= 10 – 1.282 = 8.718
	X8.68			d–ΔX(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+ΔX(1)= 8.8 + 0.45 = 9.25
	X9.57			d+ΔX(2)= 8.8 + 0.765 = 9.565
	X9.73			d+ΔX(3)= 8.8 + 0.937 = 9.737
	X9.88			d+ΔX(4)= 8.8 + 1.082 = 9.882
	X9.92			d+ΔX(5)= 8.8 + 1.122 = 9.922
	X9.96			d+ΔX(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–ΔX(1)= 10 – 0.45 = 9.55
G01 or G00		W–0.12		ΔW= 0.124 0.12
G92	X9.12	ZΔΔ.ΔΔ		d–ΔX(2)= 10 – 0.878 = 9.122
G01 or G00		W–0.06		ΔW= 0.057 0.06
G92	X8.92	ZΔΔ.ΔΔ		d–ΔX(3)= 10 – 1.075 = 8.925
G01 or G00		W–0.05		ΔW= 0.048 0.05
G92	X8.76	ZΔΔ.ΔΔ		d–ΔX(4)= 10 – 1.242 = 8.758
	X8.72			d–ΔX(5)= 10 – 1.282 = 8.718
	X8.68			d–ΔX(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+ΔX(1)= 8.8 + 0.45 = 9.25
G01 or G00		W–0.09		ΔW= 0.091 0.09
G92	X9.57	ZΔΔ.ΔΔ		d+ΔX(2)= 8.8 + 0.765 = 9.565
G01 or G00		W–0.05		ΔW = 0.05
G92	X9.73	ZΔΔ.ΔΔ		d+ΔX(3)= 8.8 + 0.937 = 9.737
G01 or G00		W–0.04		ΔW = 0.042
G92	X9.88	ZΔΔ.ΔΔ		d+ΔX(4)= 8.8 + 1.082 = 9.882
	X9.92			d+ΔX(5)= 8.8 + 1.122 = 9.922
	X9.96			d+ΔX(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	ZΔΔ.ΔΔ	F1.0	d–DX(1)=10–0.45=9.55
G01 or G00		W–0.12		ΔW=0.124 0.12
G92	X9.12	ZΔΔ.ΔΔ		d–ΔX(2)=10–0.818=9.122
G01 or G00		W(+)0.06		ΔW=0.057 0.06
G92	X8.92	ZΔΔ.ΔΔ		d–ΔX(3)=10–1.075=8.925
G01 or G00		W–0.05		ΔW=0.048 0.05
G92	X8.76	ZΔΔ.ΔΔ		d–ΔX(4)=10–1.082=8.758
	X8.72			d–ΔX(5)=10–1.282=8.718
	X8.68			d–ΔX(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	ZΔΔ.ΔΔ	F1.0	d+ΔX(1)=8.8+0.45=9.25
G01 or G00		W–0.09		ΔW=0.091 0.09
G92	X9.57	ZΔΔ.ΔΔ		d+ΔX(2)=8.8+0.765=9.565
G01 or G00		W(+)0.05		ΔW=0.05
G92	X9.73	ZΔΔ.ΔΔ		d+ΔX(3)=8.8+0.937=9.737
G01 or G00		W–0.04		ΔW=0.042
G92	X9.88	ZΔΔ.ΔΔ		d+ΔX(4)=8.8+1.082=9.882
	X9.92			d+ΔX(5)=8.8+1.122=9.922
	X9.96			d+ΔX(6)=8.8+1.162=9.962
G00	X...	Z...		

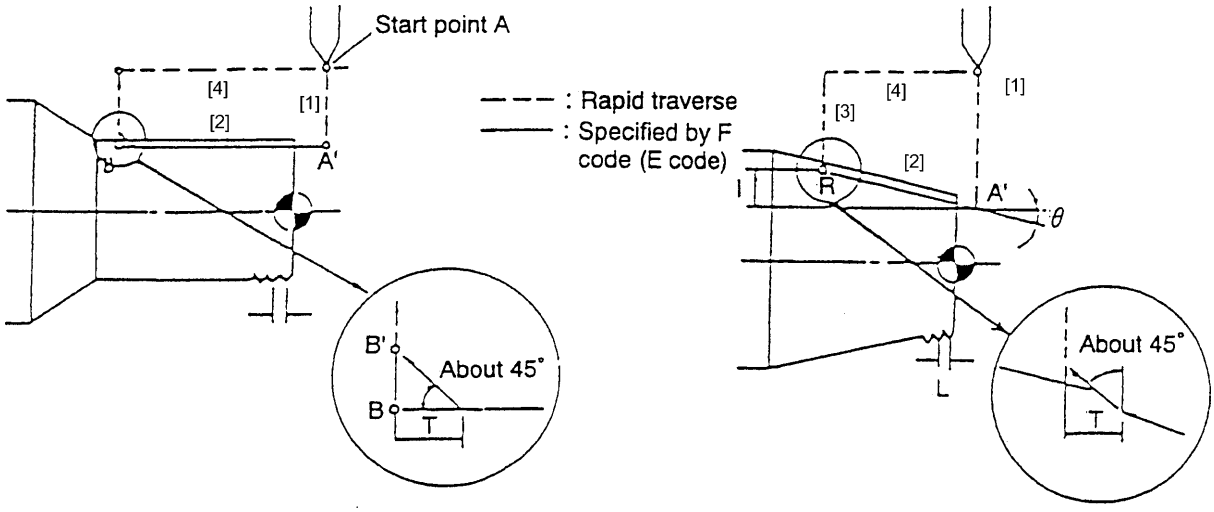
Thread chamfering

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

1. M functions for chambering selection

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

M24 ..... chambering OFF



Details of thread chamfering Details of thread chamfering

A range of chambering value r of thread cutting is 0~12.7L (L is a lead of thread) and any valve of 0.1L increment can be selected by parameter No.5111. Normally, it has been set to 1.0 L .

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

Normally, it is set at 45°.

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 chambering width is set at parameter No. 5111.

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

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

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

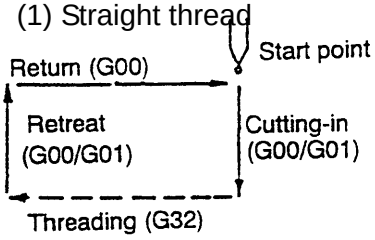
1. G32 Threading

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

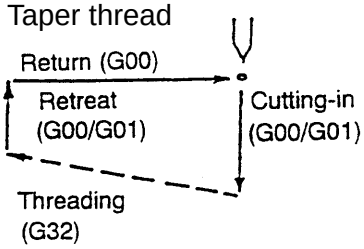
G32 does not allow cycle operation. Therefore, blocks before and after threading require programs for cutting retreat and return.

• Program cutting retreat and return with G00 or G01.

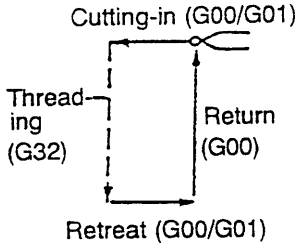
(1) Straight thread



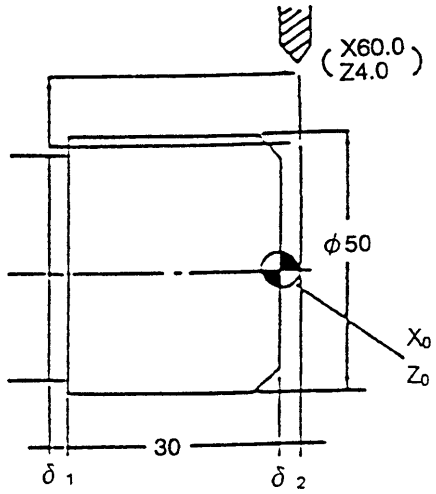
(2) Taper thread



(3) Face thread



(1) Example of straight threading



Lead : 4mm
δ 1=4mm
δ 2=2mm

M50 × P4.0 thread shown in the left figure

```
N901      T0900
N902 G97  S540 M08
N903 G00  X60.0 Z4.0 M03        • To the start point
N904      X49.12                • Cutting-in
N905 G32  Z–32.0 F4.0...(W–36.0) • Threading
N906 G00  X60.0                 • Retreat
N907      Z4.0                  • Return
N908      X48.39                • Cutting-in
X909 G32  Z–32.0...(W–36.0)     • Threading
N910 G00  X60.0                 • Retreat
N911      Z4.0                  • Return
N912      X48.03                • Cutting-in
N913 G32  Z–32.0...(W–36.0)     • Threading
N914 G00  .....                 • Retreat
. .       .....
. .       .....
N965 G32  Z–32.0(W–36.0)        • Threading
N966 G00  X60.0                 • Retreat
N967      X200.0 Z200.0 M09     • To the index point
N968      M01
```

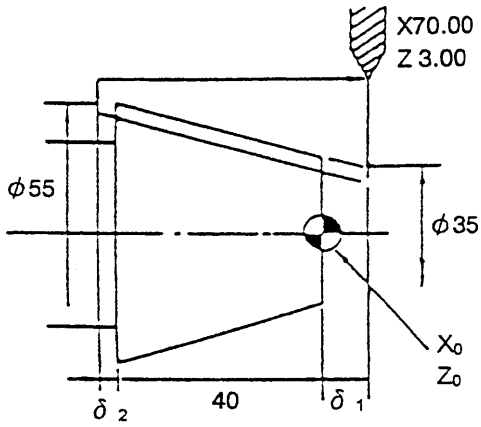
* This example cuts a thread from an outer diameter of 50.0 mm.

• For the threading depth and number of threading times, refer to the number of threading list.

• U... and W... within parentheses specify strokes (incremental programming) from a threading start point to an end point.

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

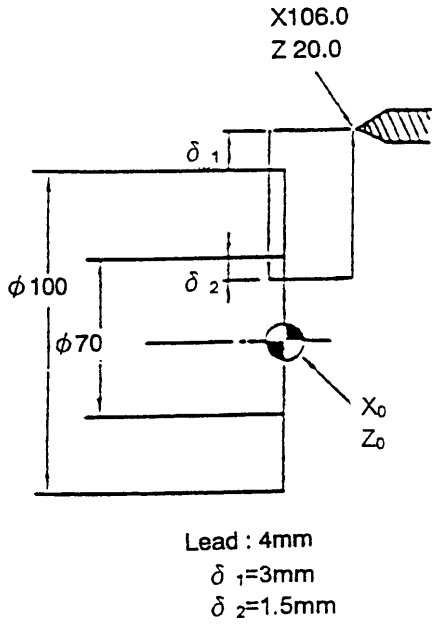
(2) Example of taper threading Program example for ϕ35/55 taper threading shown in the left figure.



Lead : 2mm
δ 1=3mm
δ 2=2mm

```
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
X909 G32    X53.96 Z–42.0...(U20.0 W–45.0)
N910 G00    X70.0
N911        Z3.0
N912        X33.72
N913 G32    Z53.72 Z–42.0...(U20.0 W–45.0)
N914 G00 .....
. .     .....
. .     .....
N940 G32    X52.83 Z–42.0...(U20.0 W–45.0)
N941 G00    X70.0
N942        X200.0 Z200.0 M09
N943        M01
```

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




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

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

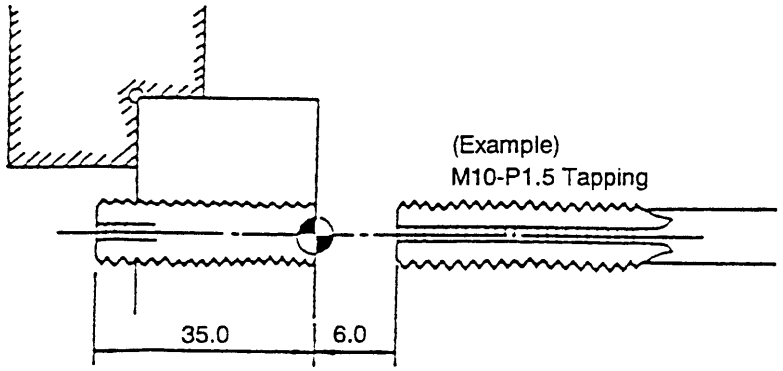
Command values in this case specify strokes from the threading start point to the threading end point.

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

2. G32 Tapping

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

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



Program Example:

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

• N605 G32 Z–35.0 F1.5 M05

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

• N607 G32 Z10.0 M04

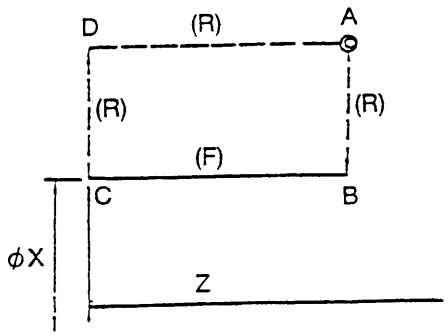
The spindle runs in the reverse direction, and then, the Z axis moves to a position of +10 mm. To retreat the tap, the safer, the bigger a command value for the Z-axis position is.

• When tapping, use a special purpose taper.

3. G92 Threading Cycle

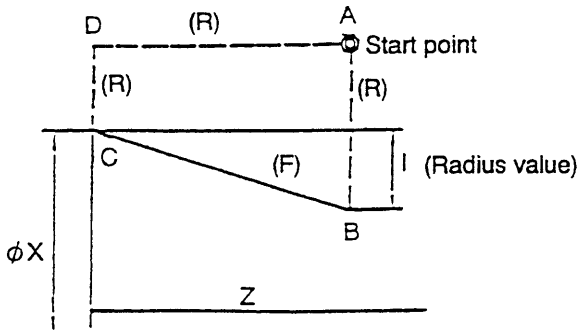
From a threading start point, four actions of cutting-in, threading, retreat and return to the start point can be specified in one block as one cycle.

(1) Straight thread



G92 X··· Z··· F···

(2) Taper thread

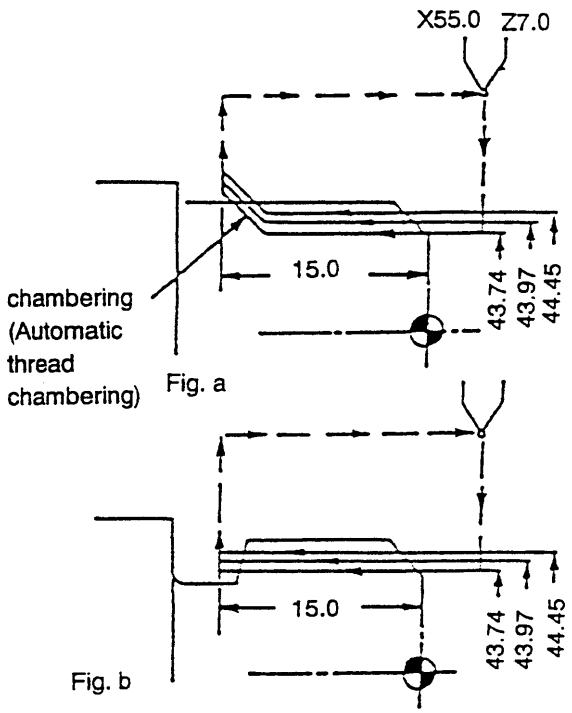


G92 X··· I±··· F ···

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

R : Rapid traverse
F : Threading

(1) Example of straight threading Program example for M45-P1.5 threading (left figure)



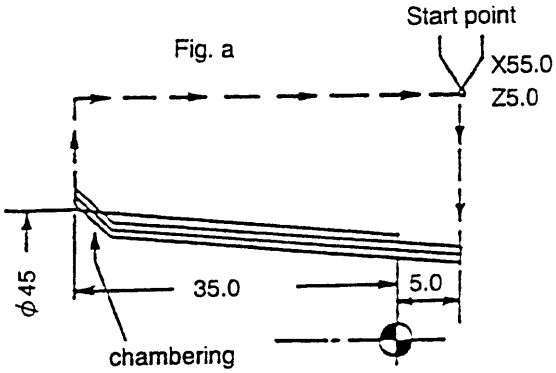

```
      N901      T0900
      N902G97   S565 M08
      N903 G00  X55.0 Z7.0 M03
*     N904      M23
      N905 G92  X44.45 Z–15.0 F1.5
      N906      X43.97
      N907      X43.74
      N908      X43.54
      X909      X43.37
      N910      X43.22
      N911      X43.18
      N912      X43.14
* 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

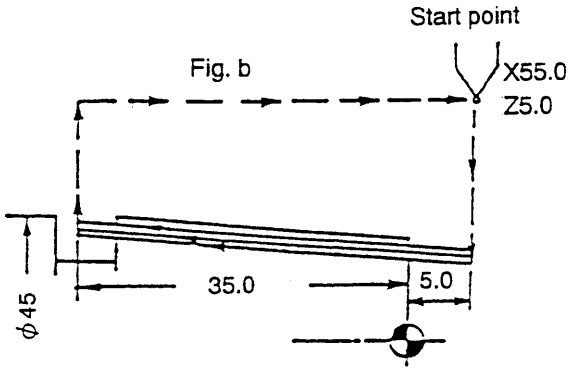
N905 G92 X44.45 Z–15.0 F1.5

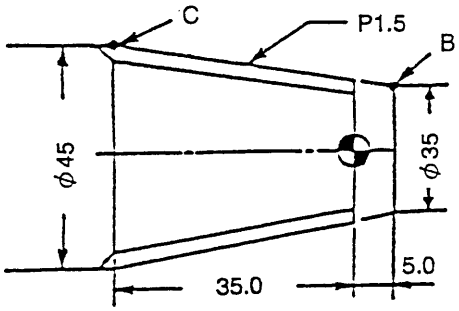
- G92: Threading cycle command
- X44.45: Cutting diameter dimension for 1st threading
- Z–15.0: Threading end position in the longitudinal direction
- F1.5: Thread pitch or lead

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

(2) Example of taper threading







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

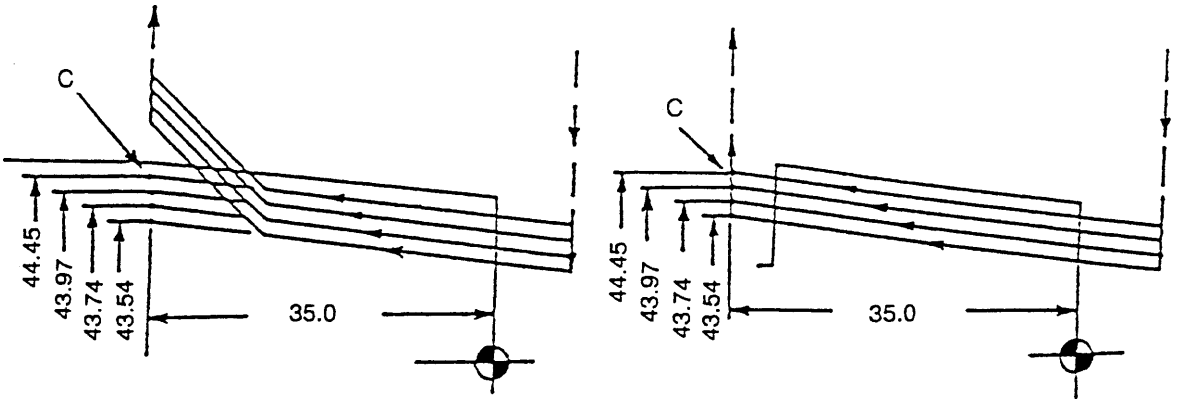
$$I = \frac{45-40}{2} = 2.5\text{mm}$$

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

Therefore; I = –2.5

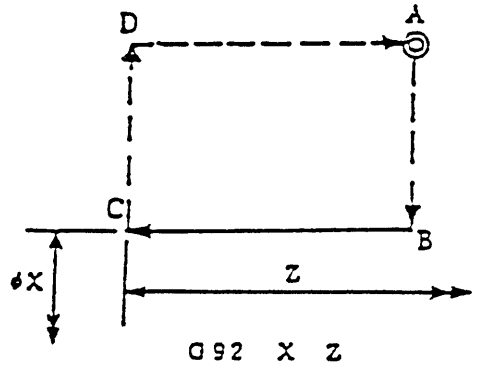
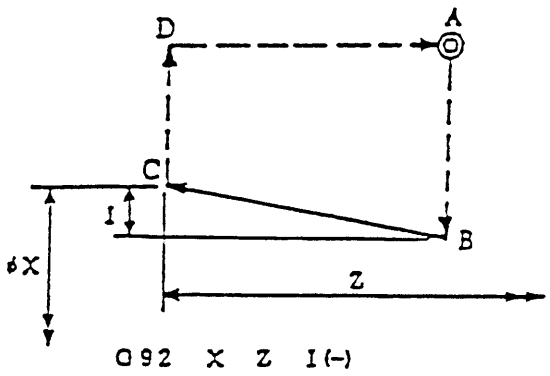
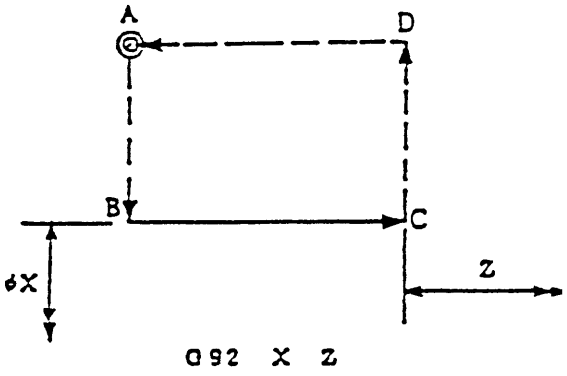
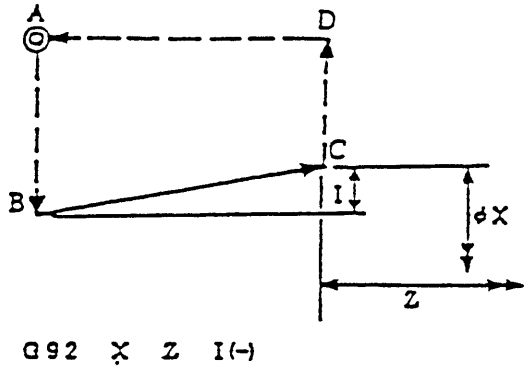
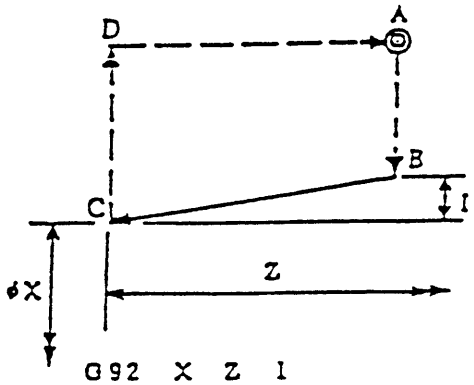
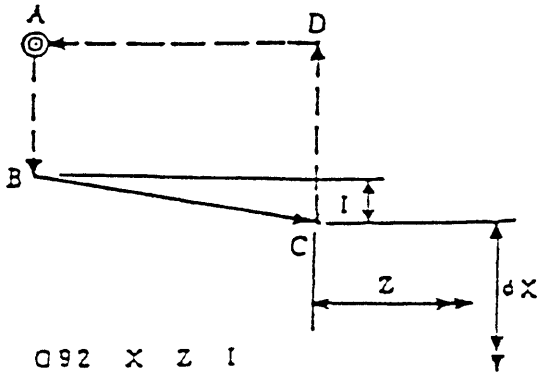
```
  N901       T0900
  N902 G97   S500 M08
  N903 G00   X55.0 Z5.0 M03
* N904       M23
  N905 G92   X44.45 Z–35.0 I–2.5 F1.5...(W–40.0)
  N906       X43.97
  N907       X43.74
  N908       X43.54
  X909       X43.37
  N910       X43.22
  N911       X43.18
  N912       X43.14
* N913       M24
  N914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

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

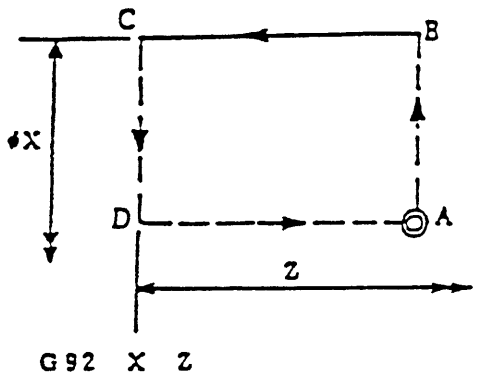
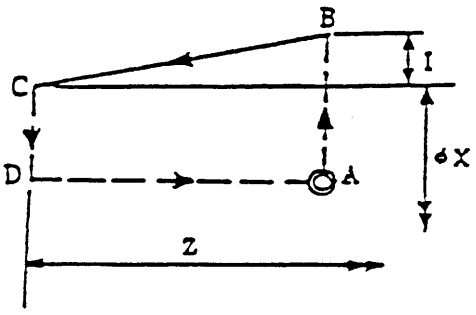
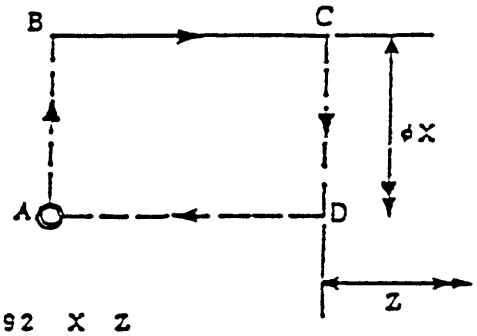
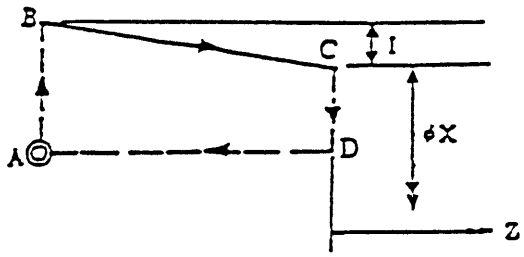
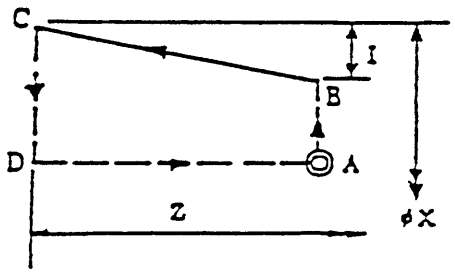
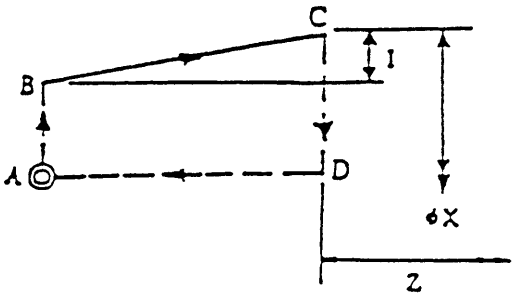


Point C: A position of end point of threading on the extend line of taper.

G92 Cycles

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

G92 Cycles

	Straight thread	Taper thread
I D (1)	 G92 X Z	 G92 X Z I
I D (2)	 G92 X Z	 G92 X Z I
I D (3)		 G92 X Z I(-)
I D (4)		 G92 X Z I(-)

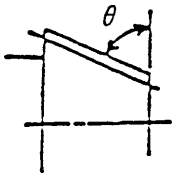
Note)

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 $\theta \geqq 45°$, a load of Z-axis direction cut by 4mm.

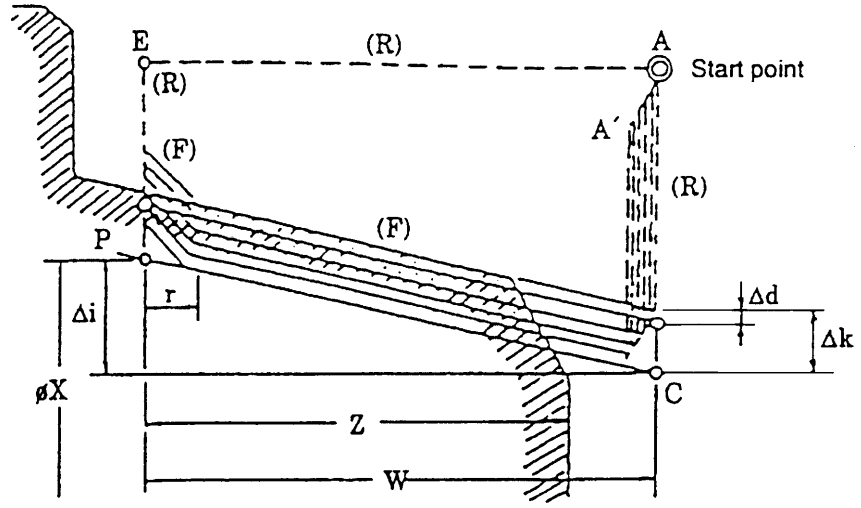
   When $\theta < 45°$, a load of X-axis direction cut by 4mm.

   Therefore, when $\theta=30°$, a lead of Z-axis direction becomes $4 \times \tan 30°$ 2.31mm.

6. Tool nose radius compensation is not allowed in threading.
7. The lengths δ_1 and δ_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.)

4. G76 Thread cutting cycle

A thread cycle shown in the figure below is performed by the following command:



G76 X (u) ±__Z (w) ±__ I±__K_D (H)_F__A__P__Q__;

I : When the radius of the thread portion is even, the value of "I" = 0, then straight thread is cut. (Δi)

K : Height of thread (The distance in X direction is designated by the value of radius in the command.) (ΔK)

D : The depth of 1st cut (To be designated by the value of the radius in the command.) (Δd)

H : No. of cutting times

F : Lead of thread

A : Angle of tool tip (no decimal point for crest angle).... Any angle within the range of 0°~120°, with 1°increment, can be selected. If the value omitted, the angle is regarded as 0°. (Parameter No.6217)

P : Cutting method (To designate P1~P4. Omission or P0 is regarded as P1.)

Q : Shift amount of the start angle of thread cutting .... Used for cutting multiple threads.

r : Thread finish (Chamfering): Parameter No.6204
When the value of lead is "L", the value of "r" can be selected within the range of 0.1L~12.7L, with 0.1L increment. (The standard setting value is 1.0L.)

The value of thread finish angle can also be selected within the range of 1°~89°, with 1° increment. Usually, thread finish is performed at 45°. In the thread cutting cycle illustrated above, the command by F code is applicable between C and P only, other sections being performed by rapid traverse. In the case of the illustrated cycle, the sign of the incremental amount is as follows.

U,W : Negative (decided by the direction of the path C→P)

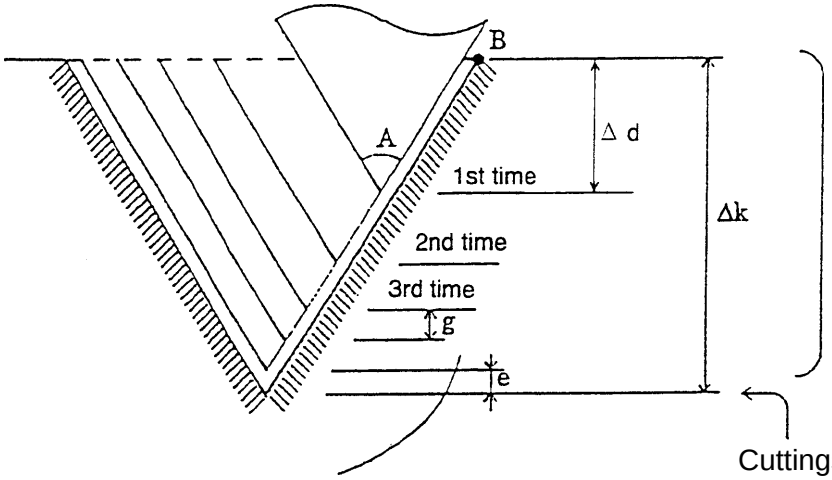
I : Negative (decided by the direction of the path A→C)

K : Positive (always positive)

D : Positive (always positive)

Cutting method

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



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

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

(Finishing)

Parameter No.5149

(In case of 1st cutting amount <g)

1st cutting amount ..... Δd

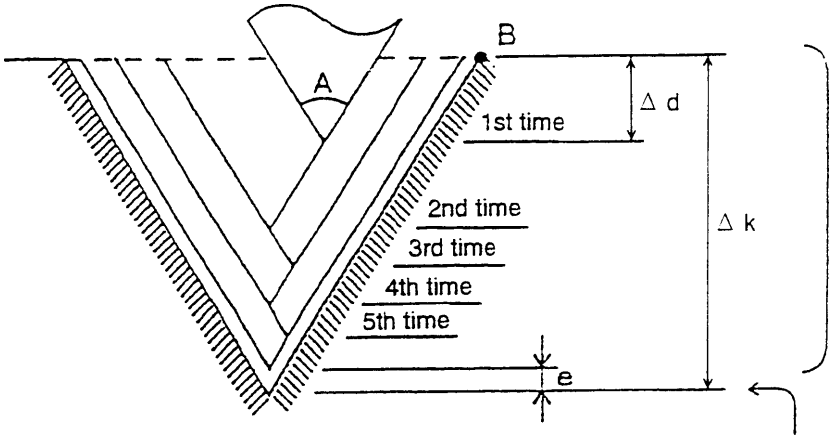
2nd cutting amount ..... $\Delta d\sqrt{2}$

3rd cutting amount ..... $\Delta d\sqrt{3}$

4th cutting amount ..... $\Delta d\sqrt{4}$

5th cutting amount ..... $\Delta d\sqrt{5}$

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



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

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

(Finishing)

1st cutting amount ..... $\Delta d \cdot \frac{\sqrt{2}}{2}$

2nd cutting amount ..... $\Delta d\sqrt{2}$

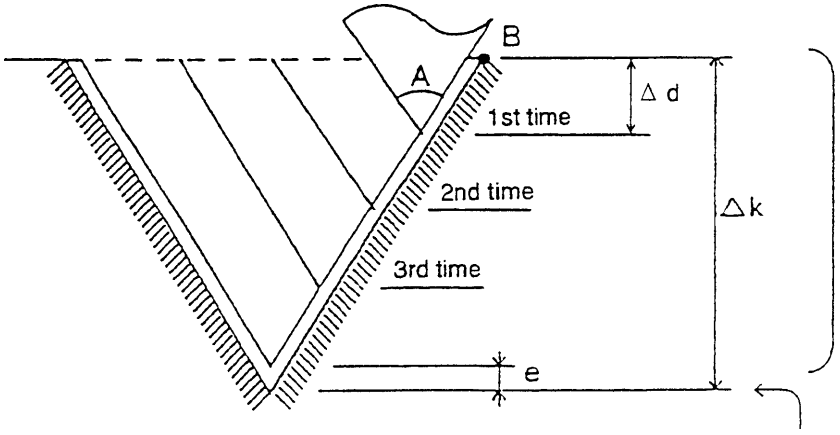
3rd cutting amount ..... $\Delta d \cdot \frac{\sqrt{2}+\sqrt{4}}{2}$

4th cutting amount ..... $\Delta d\sqrt{4}$

5th cutting amount ..... $\Delta d \cdot \frac{\sqrt{4}+\sqrt{6}}{2}$

5th cutting amount ..... $\Delta d\sqrt{6}$

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



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

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

(Finishing)

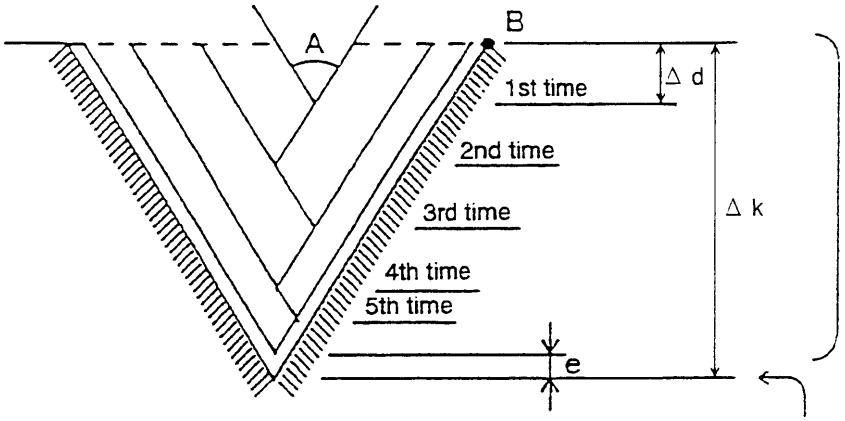
1st cutting amount ..... Δd

2nd cutting amount ..... $\Delta d \cdot 2$

3rd cutting amount ..... $\Delta d \cdot 3$

4th cutting amount ..... $\Delta d \cdot 4$

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



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

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

(Finishing)

1st cutting amount ..... Δd

2nd cutting amount ..... Δd • 2

3rd cutting amount ..... Δd • 3

4th cutting amount ..... Δd • 4

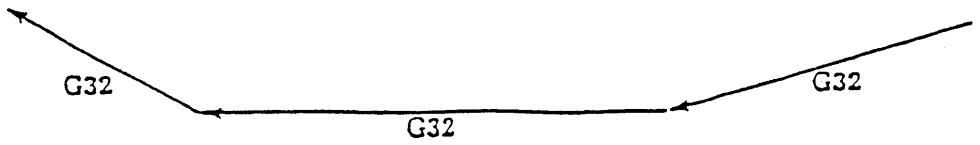
Note)

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

2-3-21 Continuous Thread Cutting

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

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



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

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

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

(1) Form of command

G34 α_ β_ F_ K_ ;

α, β : Any axis

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

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

Command range of K is as follows:

Metric command : ±0.001 ~ ±99999.999 (mm/rev)

Inch command : ±0.0001 ~ ±9999.9999 (inch/rev)

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

(2) Program example

Straight thread cutting with variable lead

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

G34 W−10. F2.0 K0.3 ;

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

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

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

Command

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

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

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

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

Least increment 1°

Commanding range $0 \leqq Q \leqq 360°$

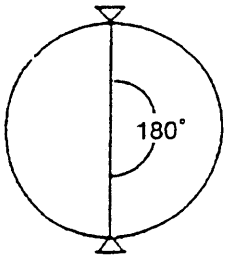
Don't enter a decimal point in Q command.

Note *1) Unit of shifted angle can be chosen by parameter setting. 1° or 0.001°.*
But when 0.001° chosen, actual resolving power is 0.088°.
2) When macro parameter is used in Q, choose 0.001° as unit of Q.

Number of thread of multi-thread and command

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

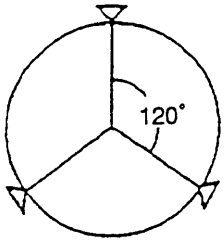
Double thread screw
Cut-in start point



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

(a) 2 thread screw

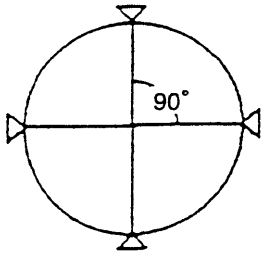
Three thread screw
Cut-in start point



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

(b) 3 thread screw

Four thread screw
Cut-in start point



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

(c) 4 thread screw

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

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

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

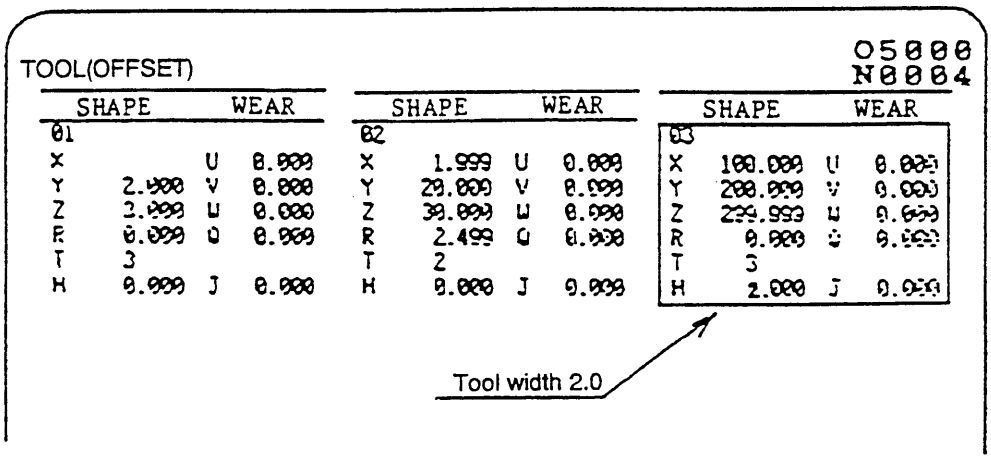
G150 Groove compensation, OFF

G151 Groove compensation for end surface, ON

G152 Groove compensation for OD/ID, ON

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

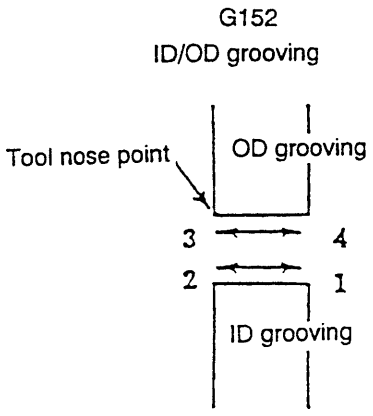
In case of using grooving tool on T03 with tool width 2.0mm.

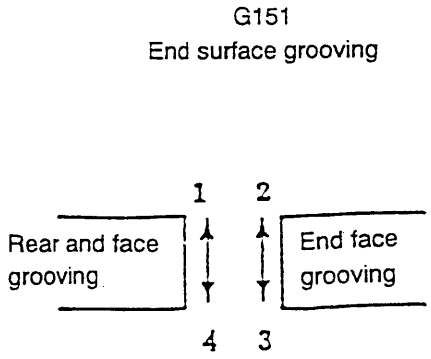


If groove width compensation (G151, G152) is commanded, tool point will be shifted as follows.

Tool width compensation is canceled by G150 or T command.

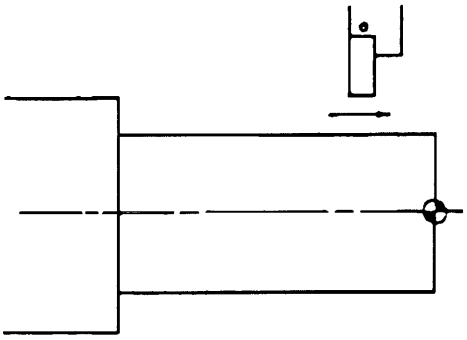
Refer to “3. Tool nose R compensation automatic calculating function” about tool nose point.

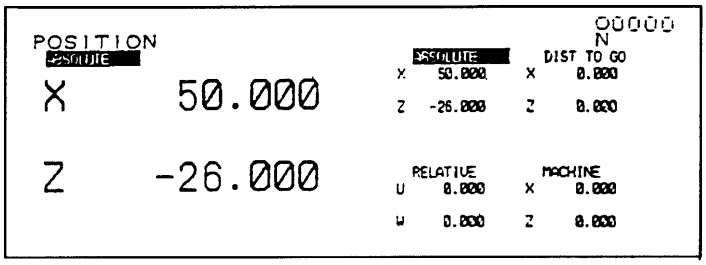




(Example 1) In case of 0D grooving tool (tool nose point 3)

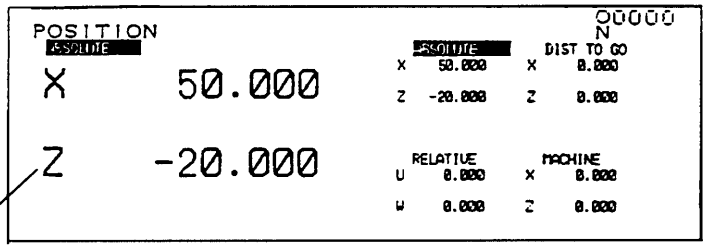
Tool width 6.0





G152 ON

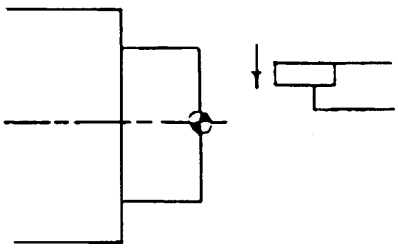
- The coordinate system of the Z-axis is shifted by the tool width amount.
- The tool nose point is shifted internally from 3 to 4.

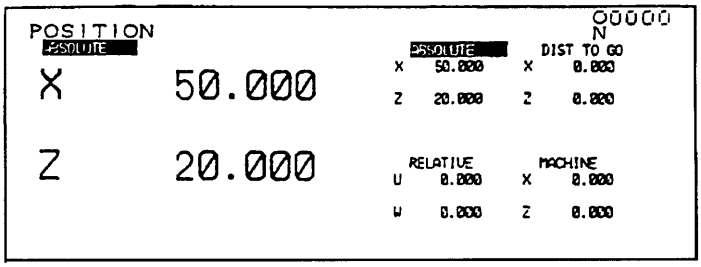


Coordinate system changes

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

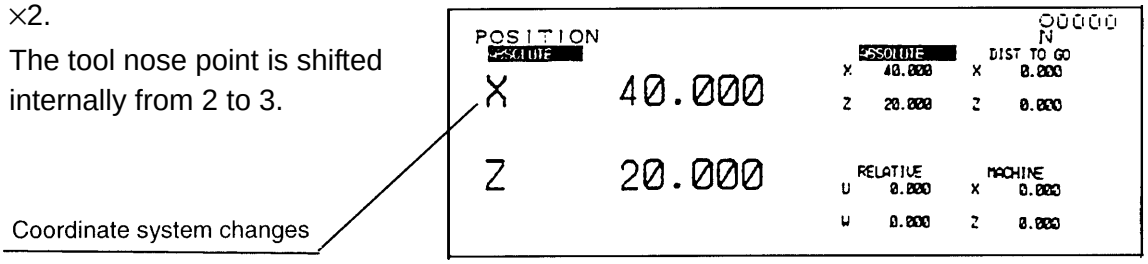
Tool width 5.0





G151 ON

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



Note *1) Except when the tip point is at 1~4, alarm is produced.*

2) With G151/G152 are continuously commanded in a program, the current correction is canceled and a new one is applied.

3) With a spindle shift command given simultaneously with G150~G150, an alarm is produced.

4) Correction is canceled on resetting while in correction, where, however, no change takes place in the coordinate axes.

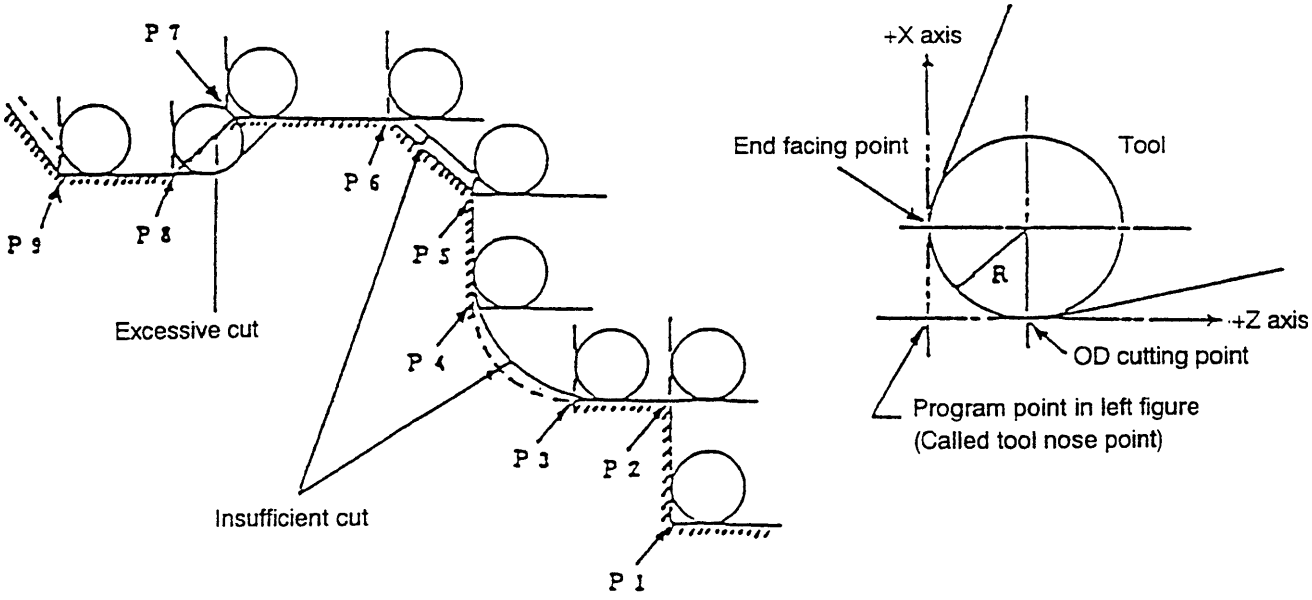
3. AUTOMATIC CALCULATING FUNCTION OF TOOL NOSE RADIUS COMPENSATION

3-1 Outline

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

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

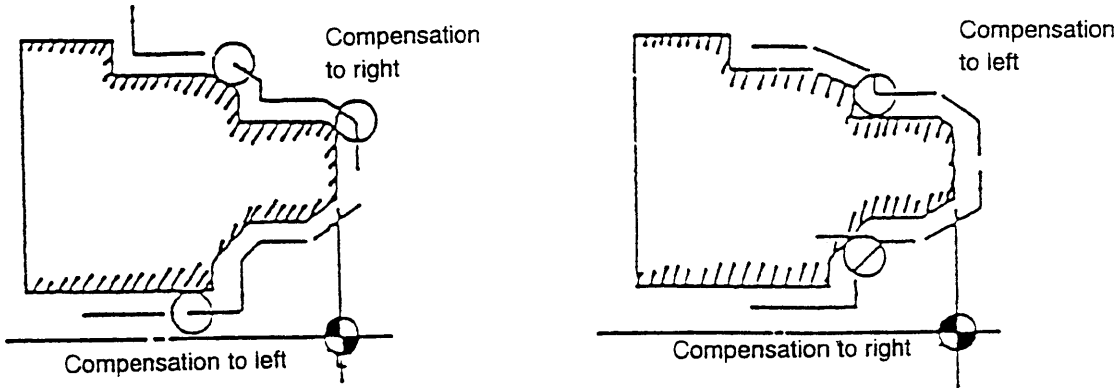
Tool nose radius compensation automatic calculating function makes operation processing done inside the NC unit, automatically controls the tool nose and prevents insufficient and excessive cutting.



P1 through p9 are program points.

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

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



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

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

These are set in the tool offset screen.

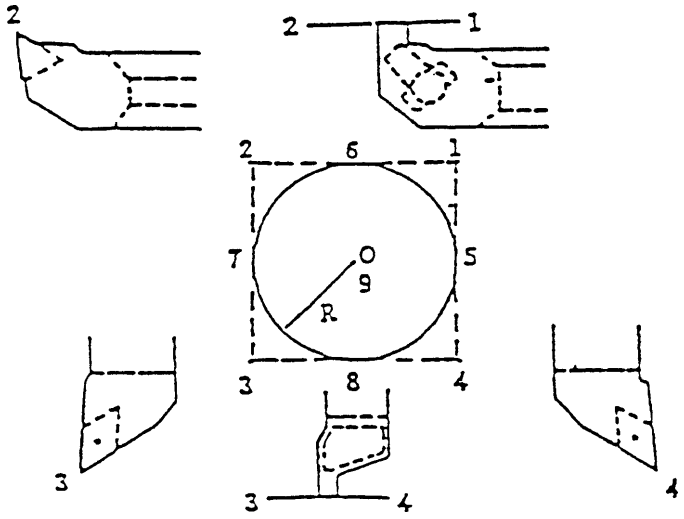
1. Tool tip point (refer to the lower sketch) ... Input at the T_ of tool offset screen.
2. Size of nose R ..... Input at the R_ of tool offset screen.

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

Tool tip point

A programmed point in nose R is called like this.

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



3-3 Three Conditions of Nose Radius Compensation

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

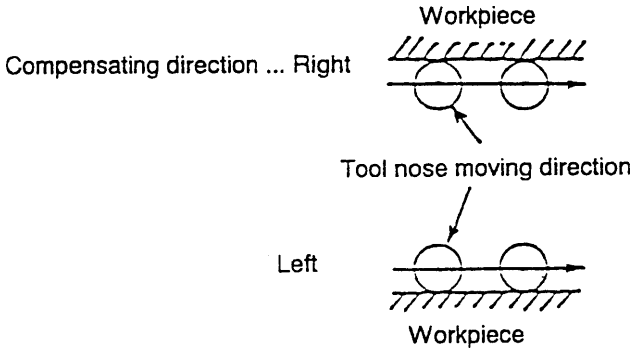
These are divided into three states and each block is called as follows:

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

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

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

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



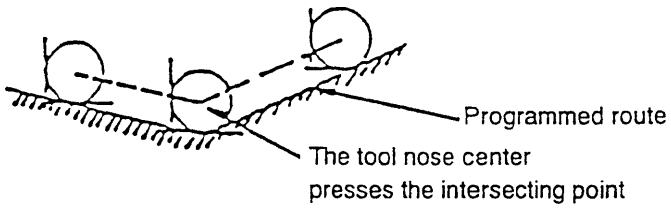
○ : Follows the compensating direction in a preceding block (because the compensating direction cannot be determined).

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

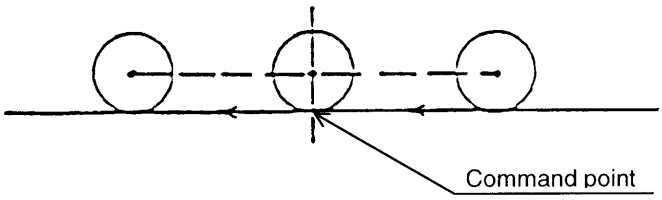
Moving direction / Nose point	→	↗	↑	↖	←	↙	↓	↘
1	Right	○	Left	Left	Left	○	Right	Right
2	Right	Right	Right	○	Left	Left	Left	○
3	Left	○	Right	Right	Right	○	Left	Left
4	Left	Left	Left	○	Right	Right	Right	○
5	○	Left	Left	Left	○	Right	Right	Right
6	Right	Right	○	Left	Left	Left	○	Right
7	○	Right	Right	Right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

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

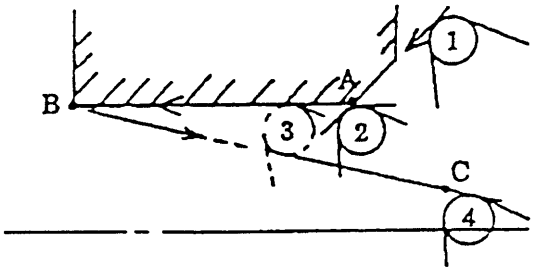
a) When a tangent angle is 180°or less (inner corner), an intersecting point is operated and the tool nose center moves to that intersecting point.



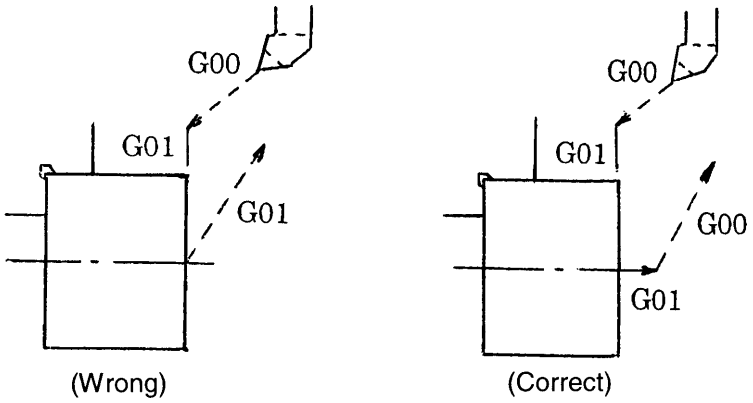
b) When the tangent angle is 180°, the tool nose center comes on the normal of a command point.



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



In case of the path A → B → 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° with the Z axis.)

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

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

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

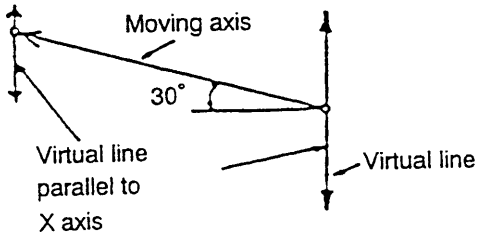
[3] Determine the compensating direction (right or left) to the virtual line.

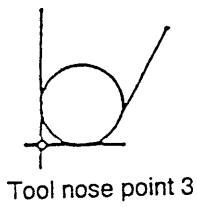
[4] Calculate the intersecting point.

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

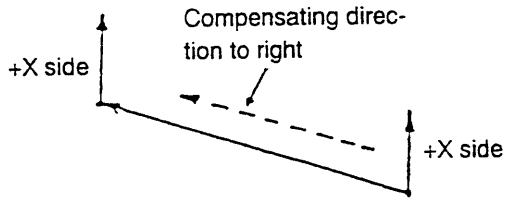
Example 1) For the tool nose point 3

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

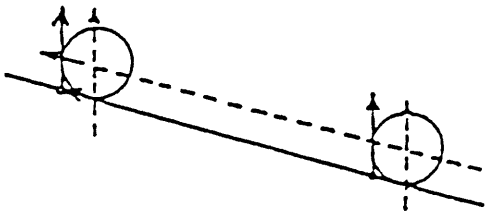


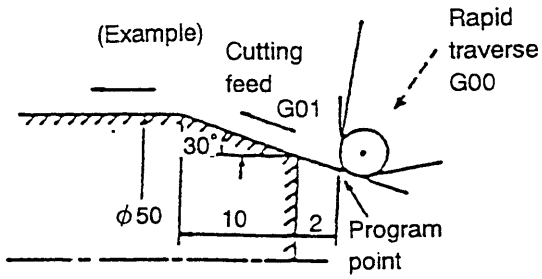


[2] Determine a virtual line direction in the same direction as the compensating direction of the moving axis (+X side because the compensating direction is to the right).



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

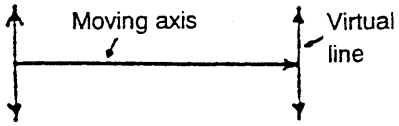


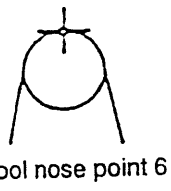



```
N100 T0100 M40 ;
N101 G97 S1060 M08 ;
N102 X36.14 Z2.0 M03 ;
N103 G01 G96 X50.0 Z-10.0 F0.2
S120 ;
N104 Z-XXXX ;
```

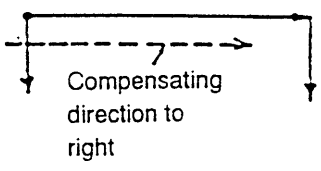
Example 2) For the tool nose point 6

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

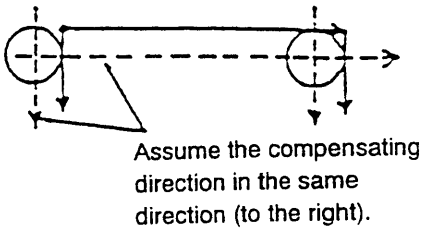




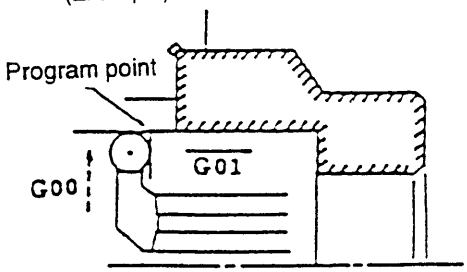
[2] Determine a virtual line direction to the -X side, because the compensating direction of the moving axis is to the right.



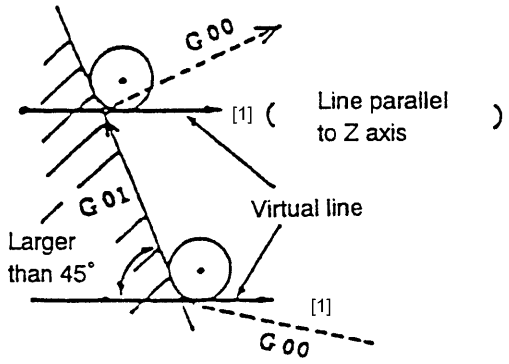
*[3] Since the compensating direction of the virtual line cannot be determined, assume it in the same direction as the compensating direction of the moving axis.

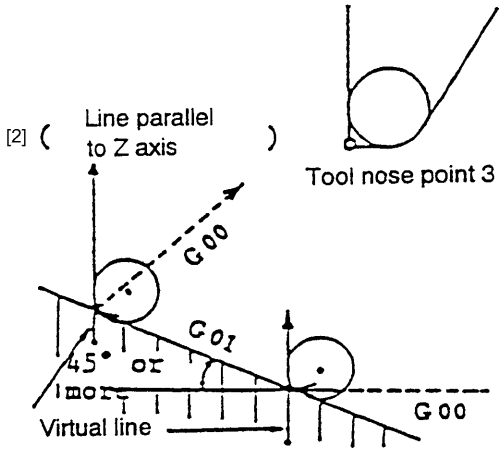


(Example)

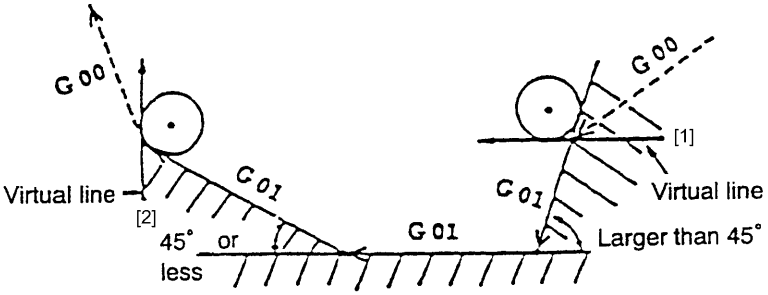


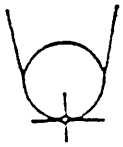
Example 3) A. For the tool nose point 3





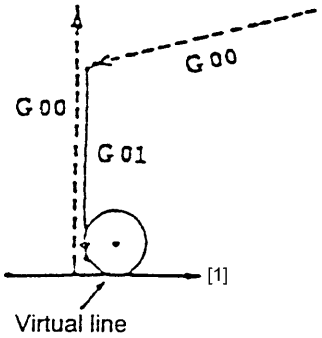
B. For the tool nose point 8

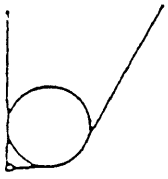




Tool nose point 8

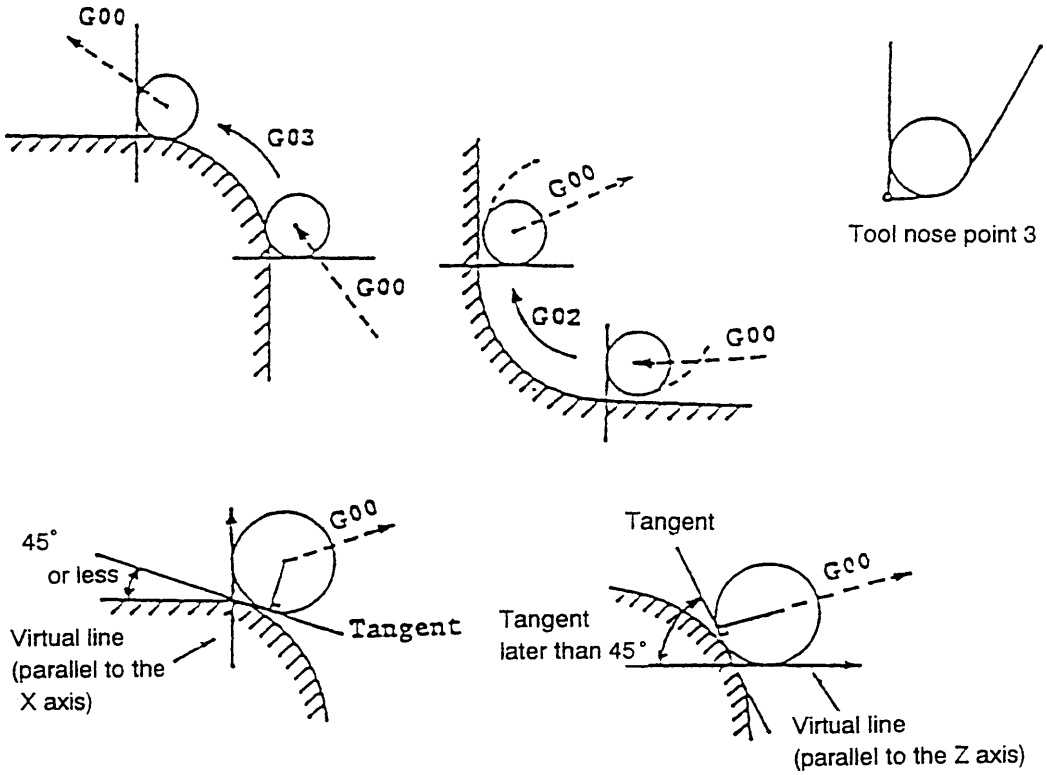
C. For the tool nose point 3 in grooving (when returning only a single axis)





Tool nose point 3

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

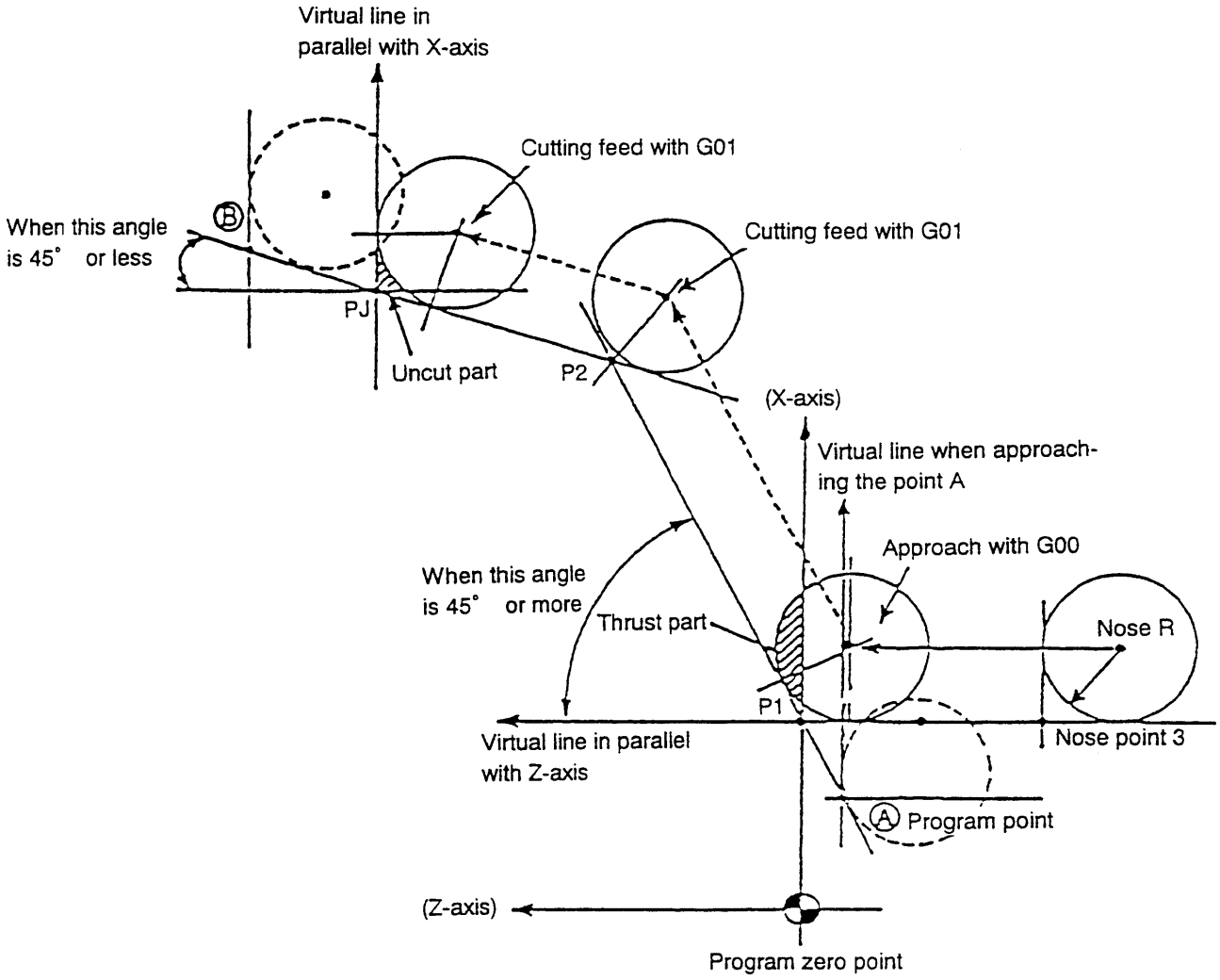


When commanding either of the following modes in the status of compensation currently the compensation is canceled.

[1] Axial travel is performed in the plane by G00.

[2] Coordinate system setting by T command.

3-4 Caution Point of Approach to Workpiece



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

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

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

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

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

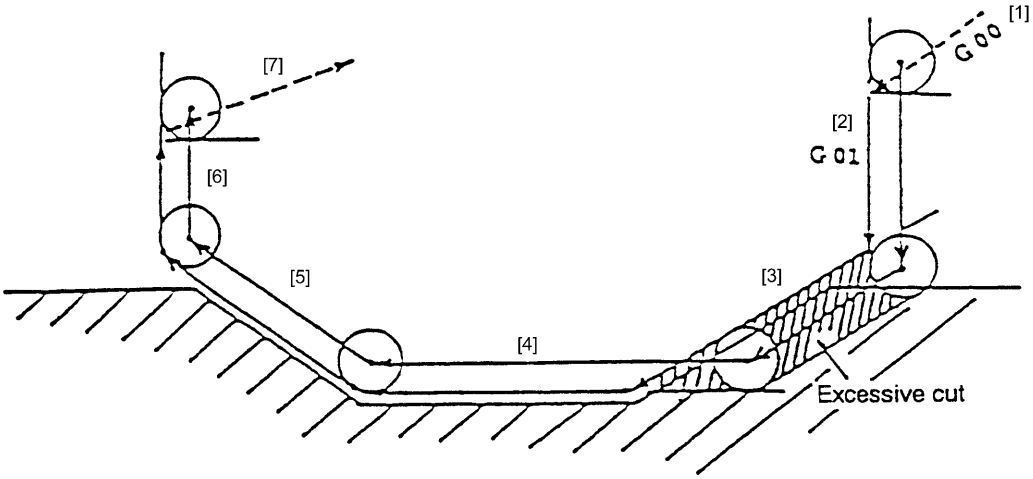
In indenting, there is no particular problem for finishing. In roughing, however, specify a compensation direction with the following G codes:

G141 Tool nose radius compensation direction to left

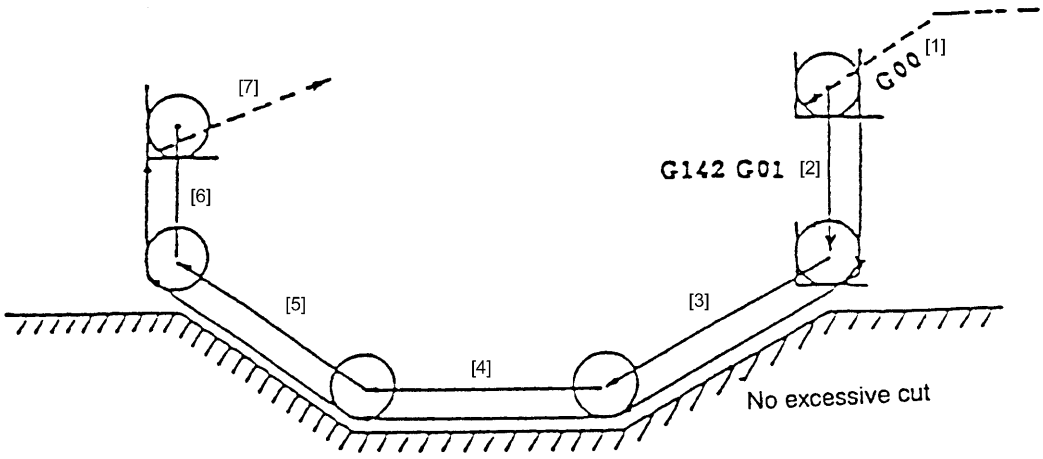
G142 Tool nose radius compensation direction to right

Effective designated one block only.

Example 1) For the tool nose point 3



Example 2) For the tool nose point 3



Program

Example 1

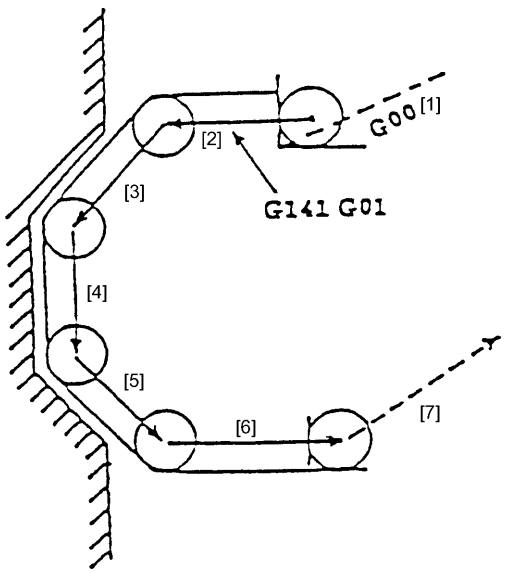
[1]	G00	X_	Z_	
[2]	G01	X_		F • • •
[3]		X_	Z_	
[4]			Z_	
[5]		X_	Z_	
[6]		X_		
[7]	G00	X_	Z_	

Example 2

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

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

Example 3)




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

Overall Precautions

1. An error will result if you specify a tool move inside an arc smaller than a tool nose radius or a groove width up to 2 times or less of the tool nose radius while executing automatic tool nose R compensation.
2. Tool nose R compensation is not performed by data input operation.

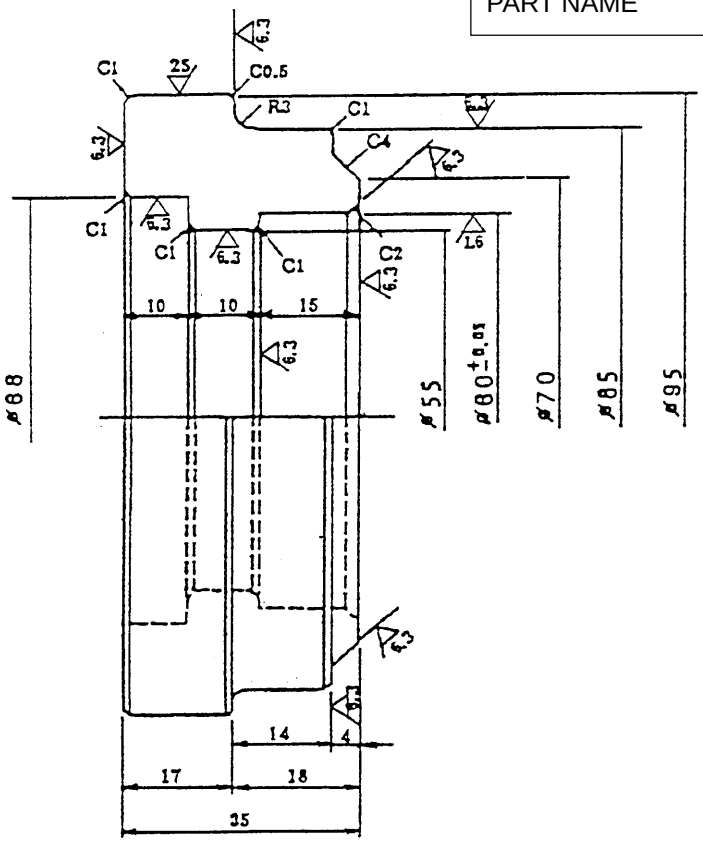
   It is available only by a program command.
3. During tool nose R compensation, if you continuously specify 3 or more blocks which do not have any move commands, compensation will be temporarily canceled.
4. In case of axis move command and T code is placed in the same block during tool nose R compensation, canceled the previous coordinate system after completion of movement then perform new coordinate system setting.
5. In the start-up block, you cannot specify a moving direction for which the compensation direction cannot be determined. If specified, excessive or insufficient cutting may be caused.

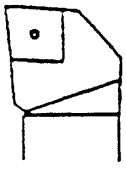
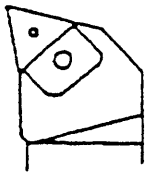
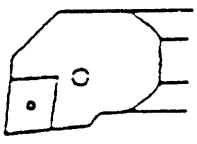
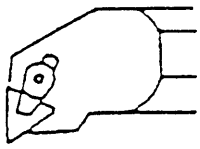
4. PROGRAM EXAMPLE (NC PROGRAM)

4-1 Chuck Work

4-1-1 Machining Drawing

EXAMPLE		SEIKI	CNC LATHE
PROCESS : 2ND	NC UNIT		TOOL LAYOUT SHEET
		PART NAME	
			MATERIAL S48C

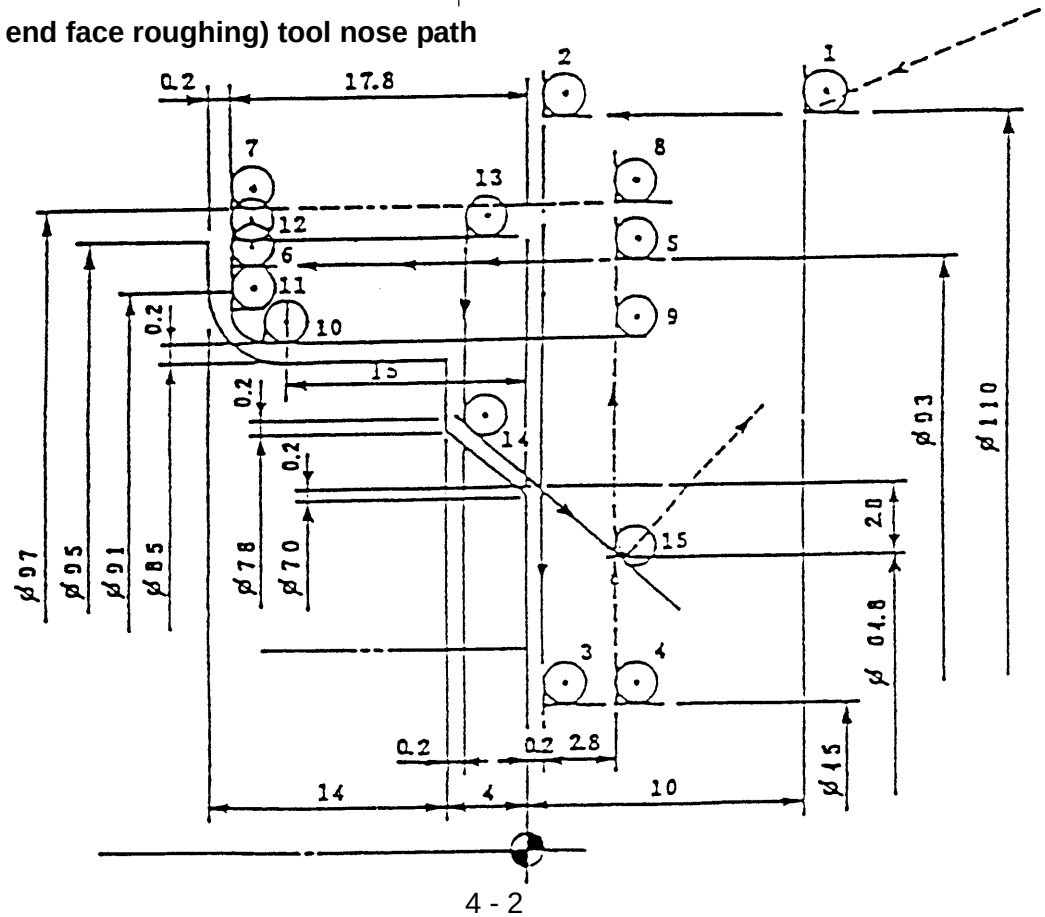


T1	T3	T5	T7	T9
				
T2	T4	T6	T8	T10
				

4-1-2 Chuck Work Program

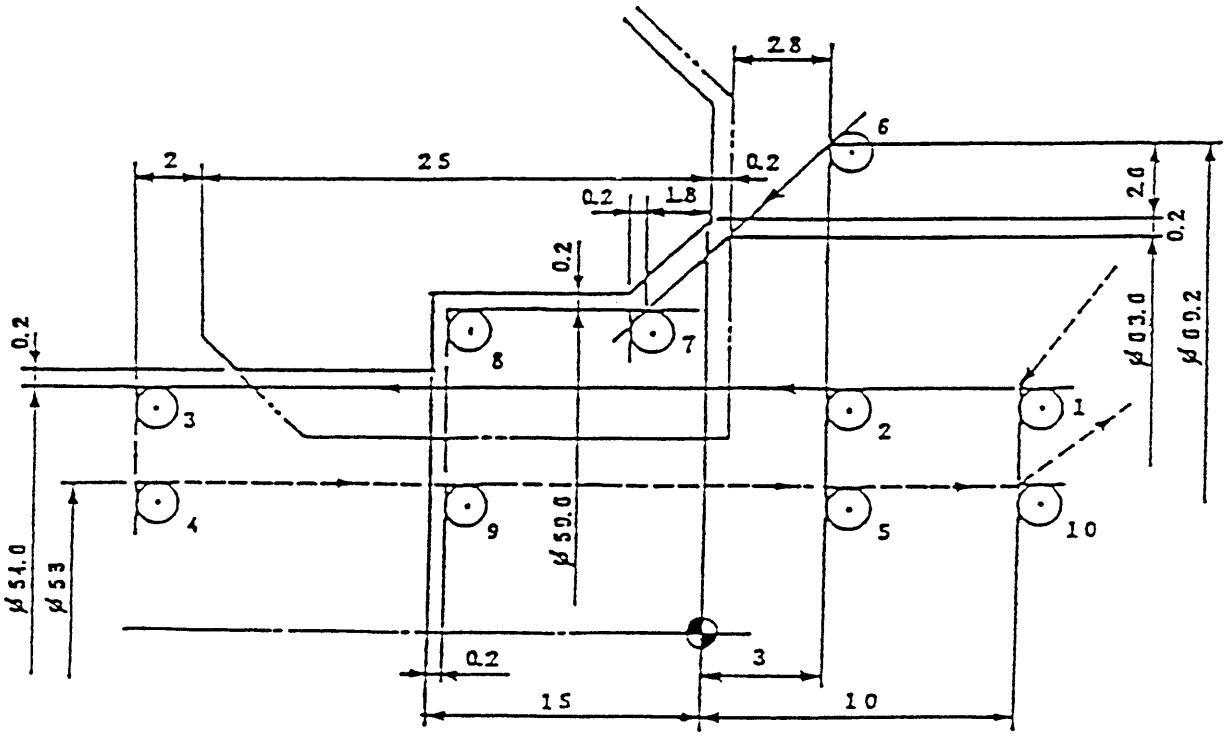
Programming	Description
O0052	Program No. Be sure to provide it.
N1 G28 U0	Automatic reference point return (X axis)
N2 G28 W0 T0100	Automatic reference point return (Z axis) Setting of T01 coordinate system
N3 G50 S2000	Maximum spindle speed clamp (2,000 rpm)
N4 G00 X200.0 Z200.0	Move to the index position.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

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



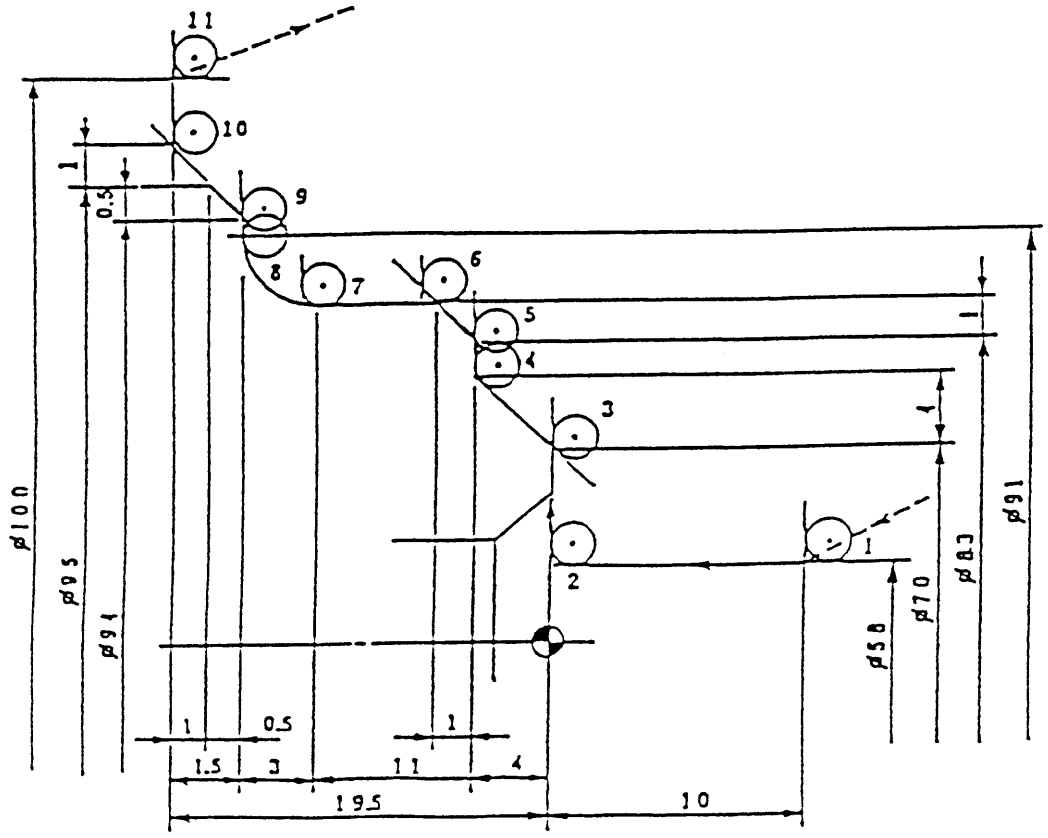
```
N401 T0400 M40
N402 G97 S650 M08
N403 G00 X54.6 Z10.0 M03
N404 Z3.0
N405 G01 Z–27.0 F0.4
N406 X53.0
N407 G00 Z3.0
N408 X69.2
N409 G01 X59.6 Z–1.8 F0.3
N410 Z–14.8 F0.4
N411 X53.0
N412 G00 Z10.0
N413 X260.0 Z100.0
N414 M01
```

2. T04 (ID roughing) tool nose route



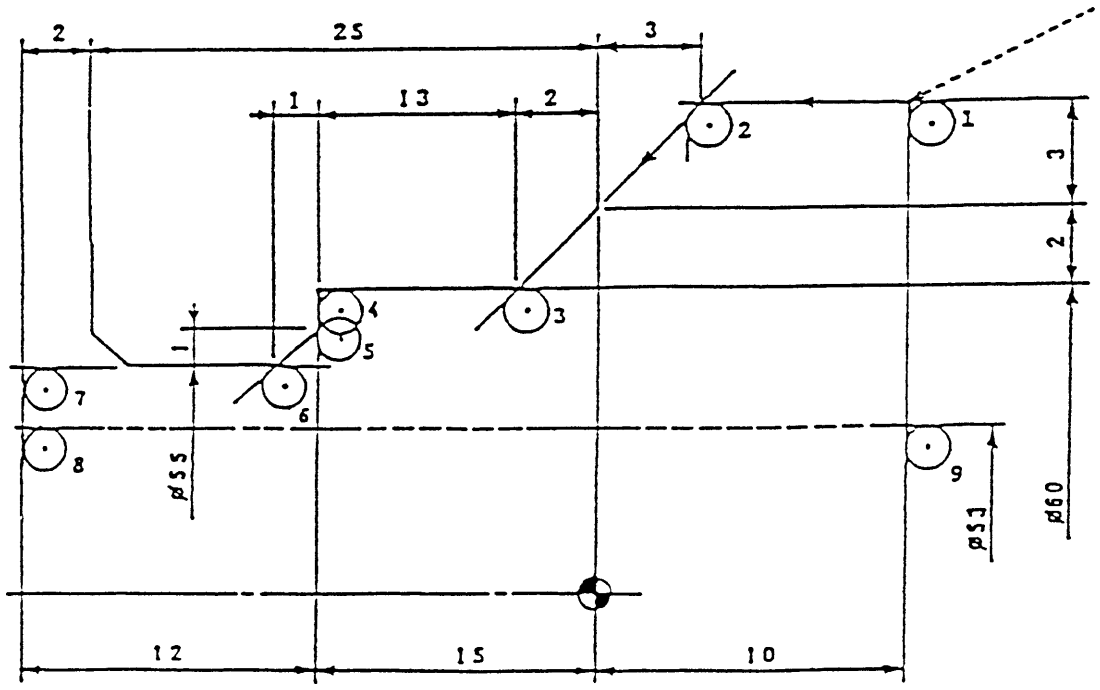
```
N701 T0700 M41
N702 G97 S1100 M08
N703 G00 X58.0 Z10.0 M03
N704 G01 G96 Z0 F1.5 S200
N705 X70.0 F0.2
N706 X78.0 Z–4.0
N707 X83.0
N708 X85.0 Z–5.0
N709 Z–15.0
N710 G02 X91.0 Z–18.0 R3.0 F0.15
N711 G01 X94.0
N712 X97.0 Z–19.5
N713 X100.0
N714 G00 G97 X200.0 Z200.0 S650
N715 M01
```

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

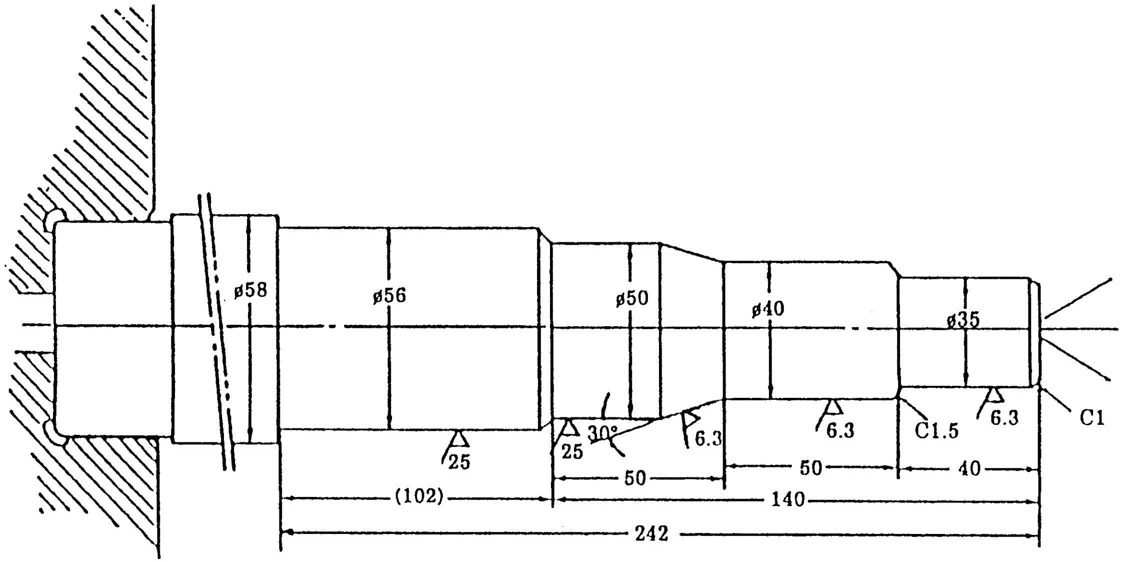


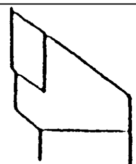
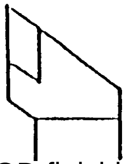
```
N801 T0800 M41
N802 G97 S1000 M08
N803 G00 X70.0 Z10.0 M03
N804 G01 G96 Z3.0 F1.5 S200
N805 X60.0 Z–2.0 F0.2
N806 Z–15.0 F0.15
N807 X57.0 F0.2
N808 X55.0 Z–16.0
N809 Z–27.0
N810 X53.0
N811 G00 Z10.0 M09
N812 G97 X260.0 Z100.0 S1200 M05
N813 M01

N6   G28 U0 W0 T0100      Automatic reference point return (X and Z axes)
N7   M30                  Program end & rewind
%                         Be sure insert a stop code
```

4. T08 (ID finishing) tool nose route



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



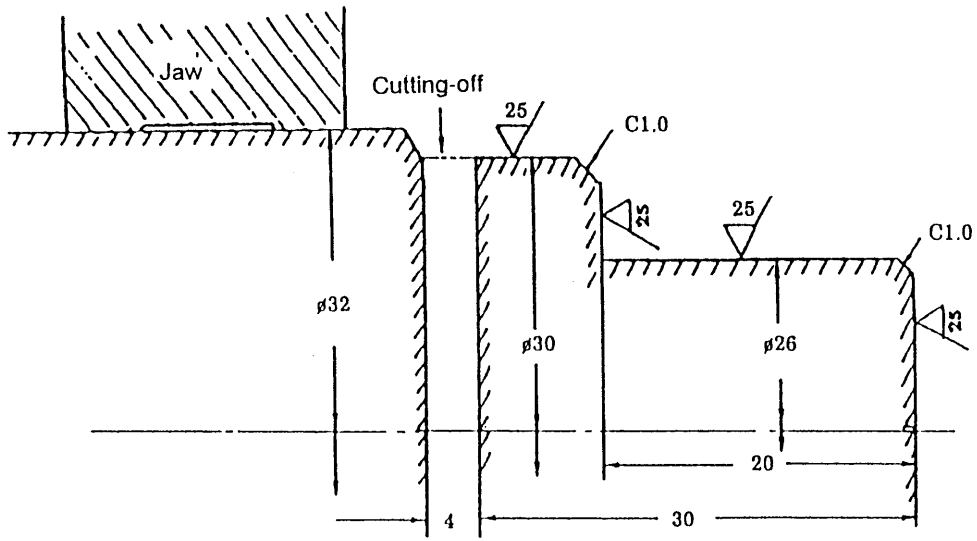
T1	T3	T5	T7	T9
	 OD roughing		 OD finishing	
T2	**T4**	**T6**	**T8**	**T10**

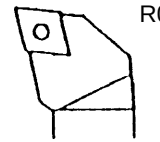
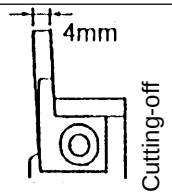
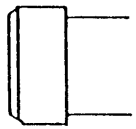
4-2-2 Center Work Program

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

N326 X37.4	
N327 X42.4 Z–42.3	
N328 G00 X50.0	
N329 G97 X200.0 Z10.0 S825	Canceling the constant surface speed
N330 M01	
OD finishing	
N701 T0700 M40	Selecting the turret face No.7
N702 G97 S1350 M08	
N703 G00 X210.0 Z2.0 M03	
N704 X40.0	
N705 G96 S170	Constant surface speed
N706 X29.0	Approach to the cutting position
N707 G01 X35.0 Z–1.0 F0.15	Machining
N708 Z–40.0	
N709 X37.0	
N710 X40.0 Z–41.5	
N711 Z–90.0	
N712 X50.0 Z–98.66	
N713 Z–140.0 F0.2	
N714 X54.0	
N715 X56.0 Z–141.0	
N716 Z–242.0	
N717 X 65.0	
N718 G00 G97 X200.0 Z10.0 S835	Canceling the constant surface speed
N719 M01	
N6 G28 U0 W0 T0300	Automatic reference point return
N7 M30	End of program & rewind
%	

BAR WORK EXAMPLE		SEIKI	CNC LATHE		
PROCESS :	NC UNIT		TOOL LAYOUT SHEET		MATERIAL S48C-D



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

4-3-2 Bar Work Program

O005	
N1 G28 U0	
N2 G28 W0 T1000	
N3 G50 S2000	
N4 G00 X200.0 Z200.0	
N5 M01	
Material sizing	
N1001 T1000 M40	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 X200.0 Z200.0	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	

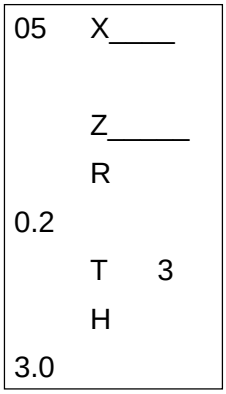
N112 Z–36.0	
N113 X34.4 Z–38.0	
N114 G00 X40.0	
N115 Z3.0	
N116 X18.0	OD finishing
N117 G01 X26.0 Z–1.0 F0.3	
N118 Z–20.0	
N119 X28.0	
N120 X30.0 Z–21.0	
N121 Z–35.0	
N122 G00 X40.0	
N123 G97 X200.0 Z200.0 S955	
N124 M01	
Cutting-off	
N901 T0900 M40	Selecting the turret face No.9
N902 G97 S795 M08	
N903 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 MG97 X200.0 Z200.0 S795 M05	Canceling the constant surface speed
N910 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

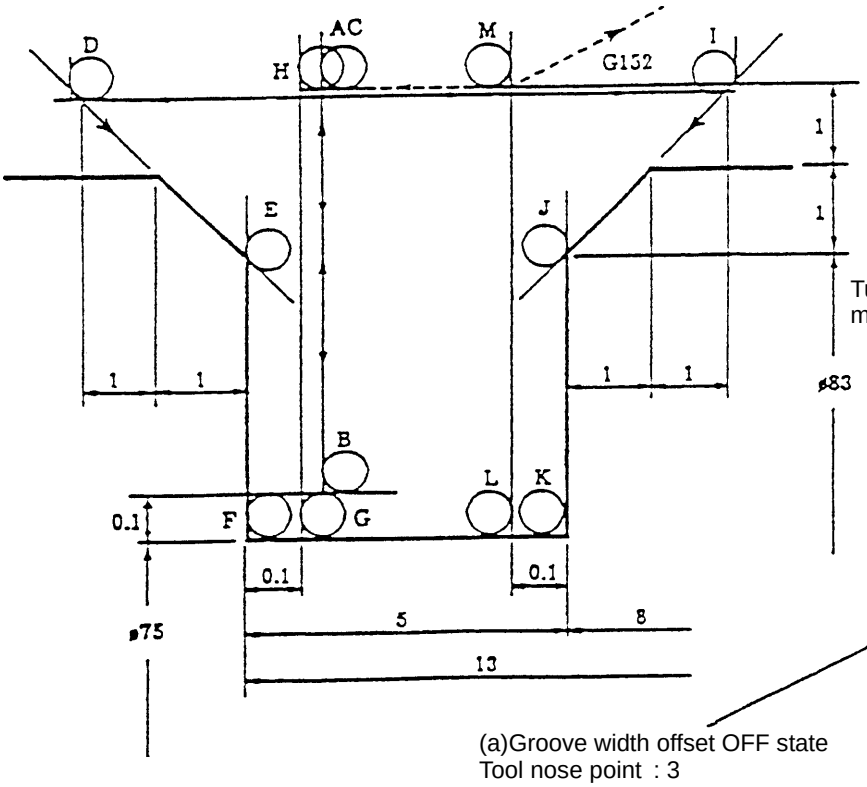
4-4-1 OD Grooving

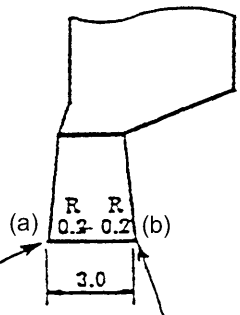
Programming		Description
	N501 T0500 M40	No.5 turret face calling
	N502 G97 S360 M08	
	N503 G150	Groove width offset OFF
	N504 G00 X87.0 Z10.0 M03	
	N505 G01 G96 Z−12.0 F5.0 S100	
A→B	N506 X75.2 F0.1	
B→C	N507 X87.0 F5.0	
C→D	N508 Z−15.0	
D→E	N509 X83.0 Z−13.0 F0.1	
E→F	N510 X75.0	
F→G	N511 Z−12.9	
G→H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

Tool offset



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





(a)Groove width offset OFF state
Tool nose point : 3
Nose R : 0.2
Nose width : Non

(b)Groove width offset ON state
(G152)
Nose width : 0.3

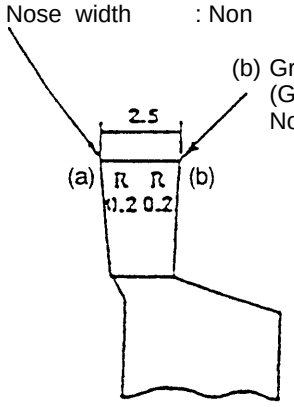
4-4-2 ID Grooving

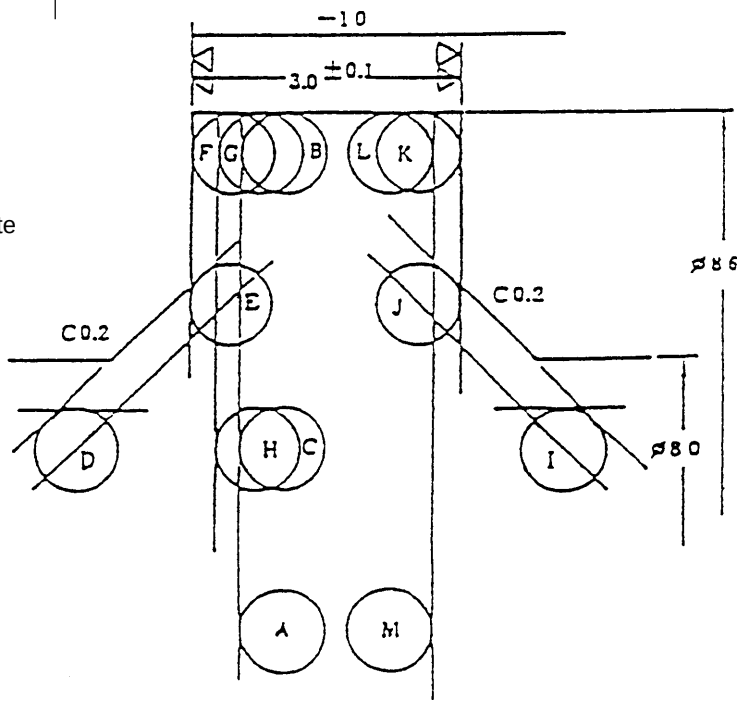
Programming	Description
N601 T0600 M40	No.6 turret face calling
N602 G97 S400 M08	
N603 G150	Groove width offset OFF
N604 G00 X78.0 Y0 Z10.0 M0.3	Tool offset
N605 G01 G96 Z–9.75 F5.0 S100	06 X____ Z____ R 0.2 T 2 H 2.5
A→B N606 X86.0 F0.1	
B→C N607 X79.0 F1.0	
C→D N608 Z–10.7	
D→E N609 X80.4 Z–10.0 F0.1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

	N613 G152	Groove width offset ON.
		Change to a program point “b”
H→I	N614 Z–6.3	
I→J	N615 X80.4 Z–7.0	
J→K	N616 X86.0 F0.1	
K→L	N617 Z–7.2	
L→M	N618 X78.0 F1.0	
	N619 G00 Z10.0	
	N620 G150	Groove width offset OFF
	N621 G00 G97 X260.0 Z100.0 S400	
	N622 M01	

(a) Groove width offset OFF state
Tool nose point : 2
Nose R : 0.2
Nose width : Non

(b) Groove width offset ON state
(G152)
Nose width : 2.5



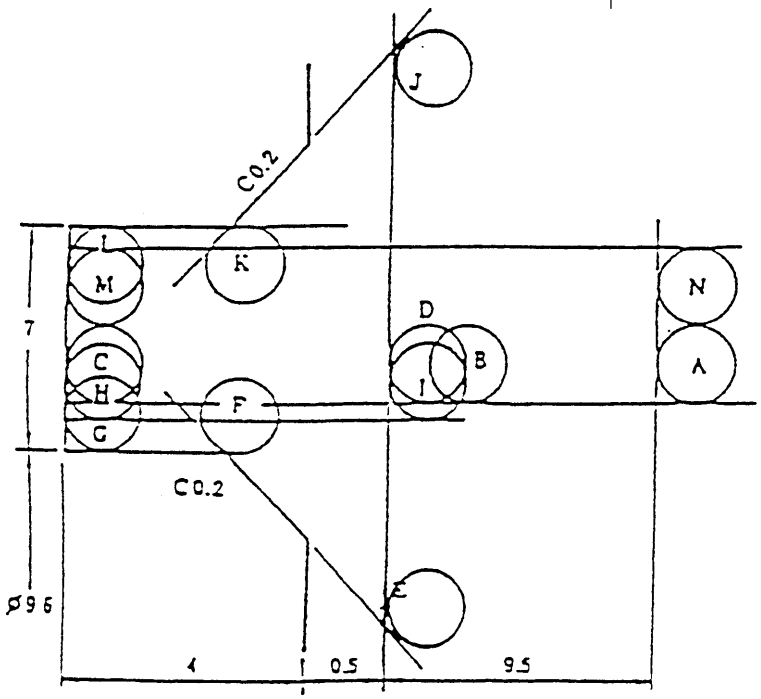


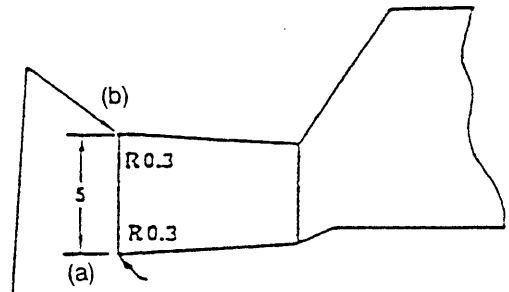
4-4-3 End Face Grooving

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

Tool offset

03	X____	
	Z____	
	R	0.2
	T	3
	H	5.0



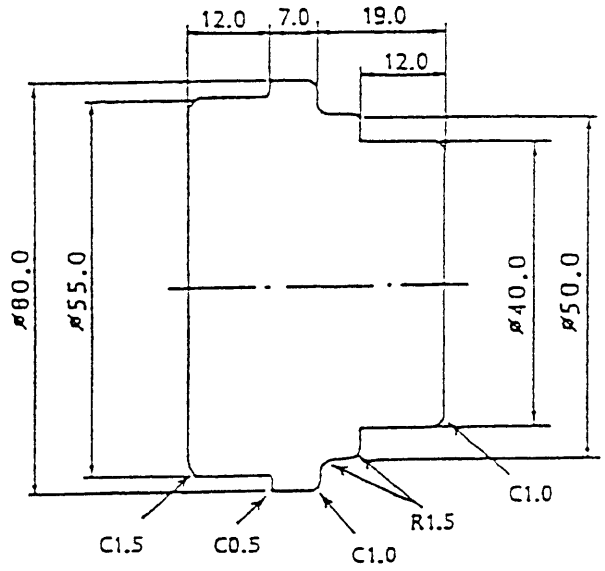


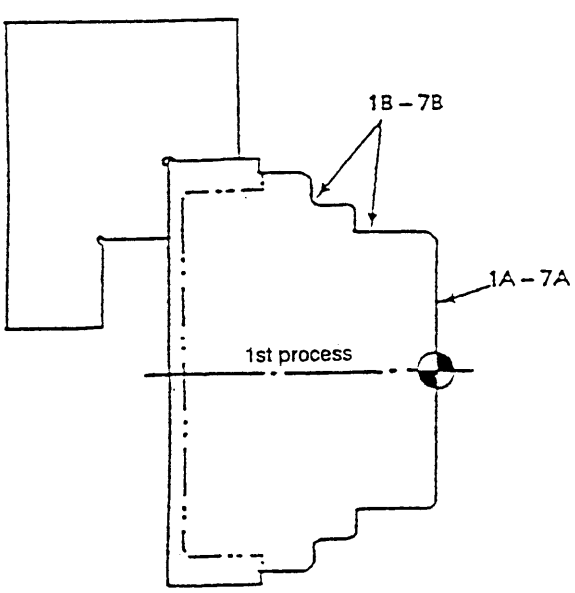
(a) Groove width offset OFF state
Tool nose point : 3
Nose R : 0.3
Nose width : Non

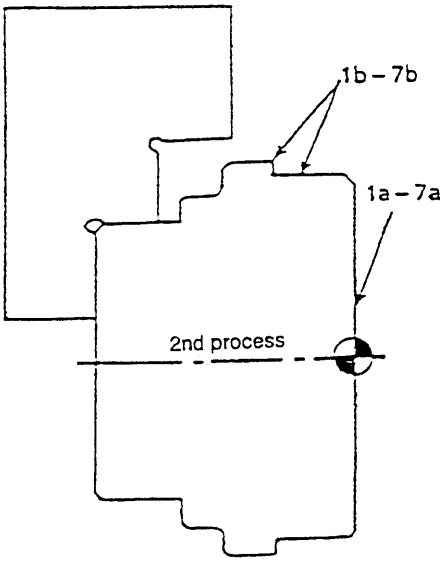
(b) Groove width offset ON state (G151)
Nose width : 5.0

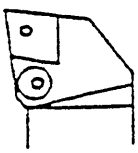
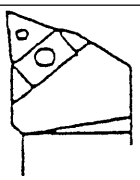
4-5 1st and 2nd Process Continuous Machining Method

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







T1	T3	T5	T7	T9
R0.8  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  G00  X200.0  Z175.0
N6  M01

N100     (OD-R)
N101     T0100 M40
N102     G97  S545
N103     G00  X70.0     Z10.0  M03
N104     G01  Z0.2 F1.5 M08
M105     G96  X–1.2 F0.2   S120
N106     ..
N...
N...     M01
N700     (OD-F)
N701     T0700 M40
N702     G97 S.... M08
N703     G00 X...    Z...M03
N704     ..
N...
N...     G00 G97 X200.0  Z175.0
N...     M01

N8  G28  U0      W0

N9  M00  ← Turning over the workpiece.

         (2nd process)
N11  G28  U0     W0
N12  G54  Z–12.0      * (Difference from the finishing
N13  G50  S1800         end face of 1st process.)
N14  G00  X200.0  Z175.0
N15  M01

N5100    (OD-R)
N5101    T0100 M40
N5102    G97  S545
N5103    G00  X...    Z...M03
N...     M08
N...
N...     G30  U0      W0
N...     M01
N5700    (OD-F)
N5701    T0700 M40
N5702    G97  S...M08
N5703    X...     Z...M03
N5704    ..
N...
N...     G00  X200.0  Z175.0
N...     M01
N20  G54  Z0          Reference point shift cancel.
N21  G28  U0     W0
N22  M30
%
```

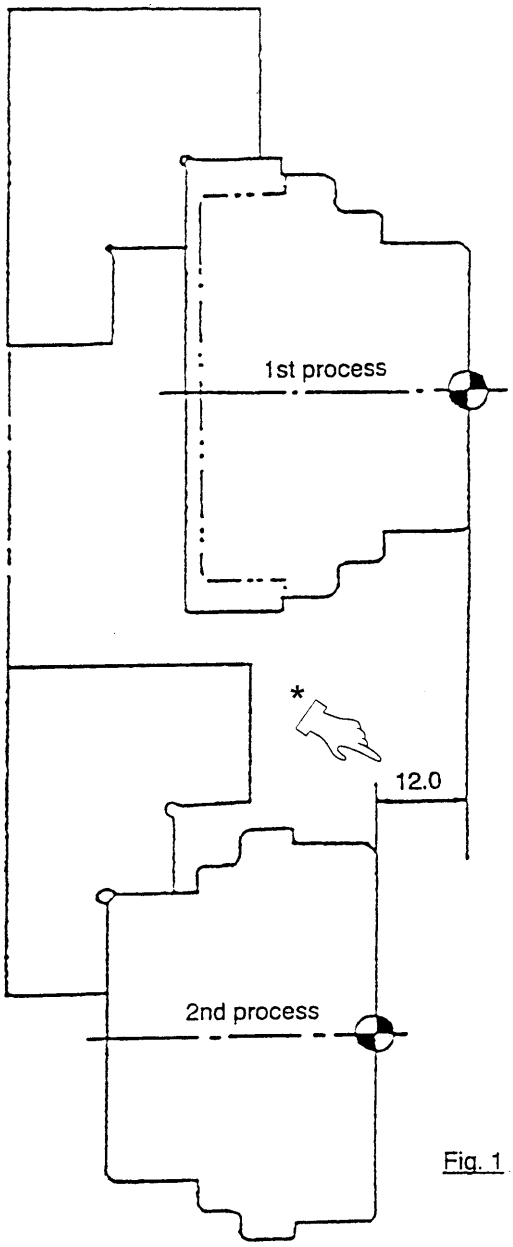


Fig. 1

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

4-5-2 Machining Method by Subprogram Calling

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

*** Main program***

02222 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
N3 G54 Z0
N4 G56 S1800
N5 G00 X200.0 Z175.0
N6 M01
N100 (OD-R)
N101 T0100 M40
N102 G97 S545
N103 G00 X70.0 Z10.0 M03
N104 G01 Z0.2 F1.5 M08
N105 G96 X–1.2 F0.2 S120
N106 Z...
N...
N...
N... M01
N700 (OD-F)
N701 T0700 M40
N702 G97 S....M08
N703 G00 X... Z...M..
N704 ..
N...
N...
N... G00 G97 X200.0 Z175.0
N... M01
N8 G28 U0 W0
/N9 M99
N10 M30
%

Reference point shift cancel.

(Program for 2nd process)
O0002 (OP-2)
N20 G28 U0 W0
N21 G54 Z–12.0
N22 G50 S1800
N23 G00 X200.0 Z175.0
N24 M01
N5100 (OD-R)
N5101 T0100 M40
N5102 G97 S545
N5103 G00 X... Z...M03
N... M08
N...
N... G00 X200.0 Z175.0
N... M01
N5700 (OD-F)
N5701 T0700 M40
N5702 G97 S...M08
N5703 X... Z...M03
N5704 ..
N...
N...
N... G00 X200.0 Z175.0
N... M01
N26 G54 Z0
N27 G28 U0 W0
/N28 M99
N29 M30
%

* (Difference from the finishing end face of 1st process.)

Reference point shift cancel.

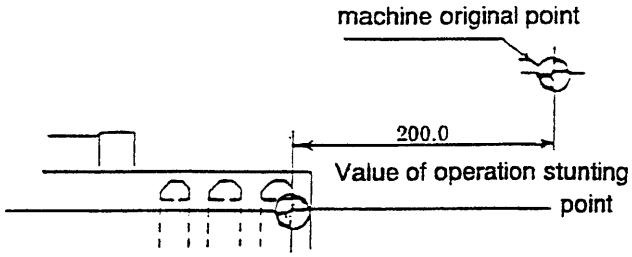
4-6 Operation Example of Many Short Length Works

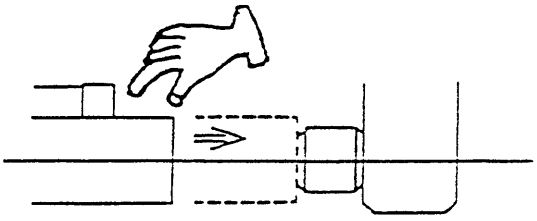
```
O1111 (Main program)
N1  G28  U0
N2  G28  W0
N3  G10  P00  Z200.0
N4  T1000
N5  G50  S2000
N6  G00  X200.0  Z50.0
N7  M01
N1000  T1000  M40
N1001  G00  Z1.0
N1002  X0
N1003  M00
N1004  G00  X200.0  Z50.0
N1005  M01
N8  M98  P2222  L3
N9  G28  U0  M09
N10  G28  W0  M05
N11  M30
```

Operation starting point

Positioning of works (manual)

Call of operation program and setting





```
02222  (Sub-program)
N100  T0100  M40
N101  G96  S120  M08
N102  G00  X...  Z...M03
```

〜

Operation program

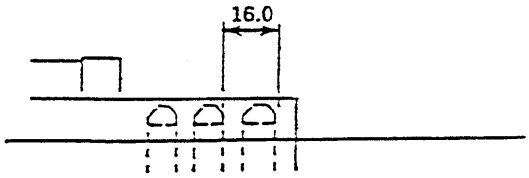
〜

Cutting-in program

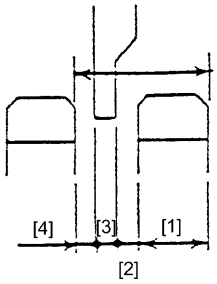
〜

```
N...  M01
N12  G10  P00  W16.0
N13  M99
```

Shifted amount



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.*



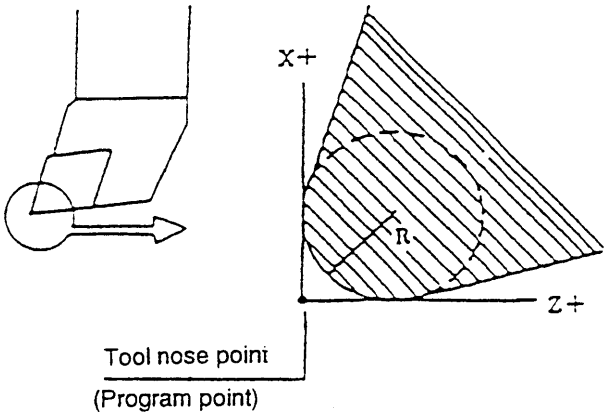
5. REFERENCE MATERIALS

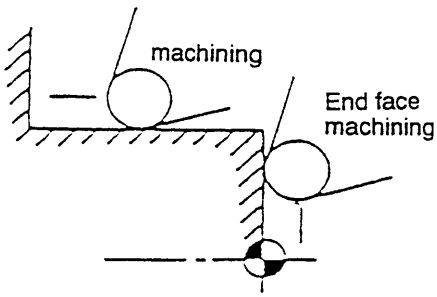
5-1 How to Calculate the Tool Nose Radius Compensation Amount Without Using the Tool Nose Radius Compensation Function

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

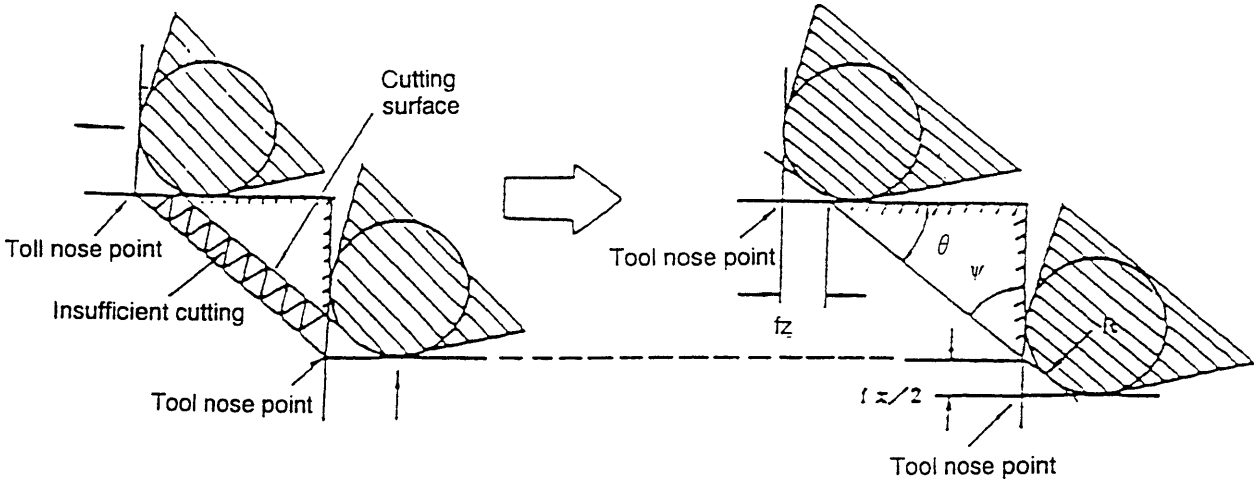
A tool nose has roundness called nose R. When cutting an outer diameter, inner diameter or end face in parallel with an axis, it can be cut as per the drawing, even if the tool nose is programmed as one point (tool nose point).





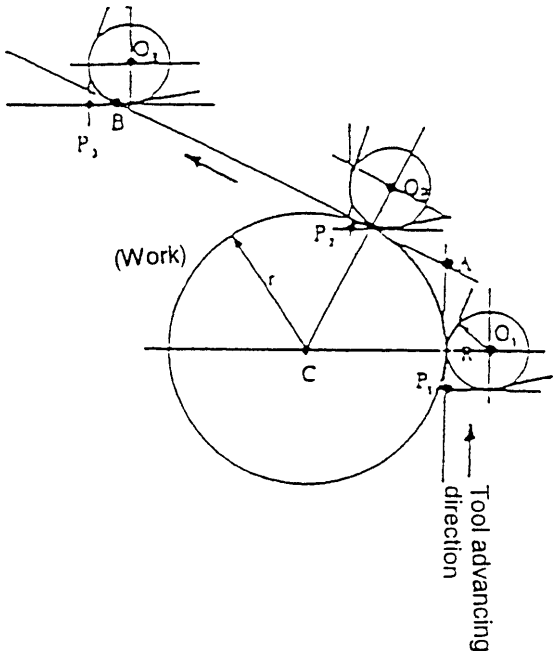
It can be cut according to the drawing even program by tool nose point.

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 (fx, fz in lower sketch) by manual calculation.



2. Calculating procedure of tool nose position

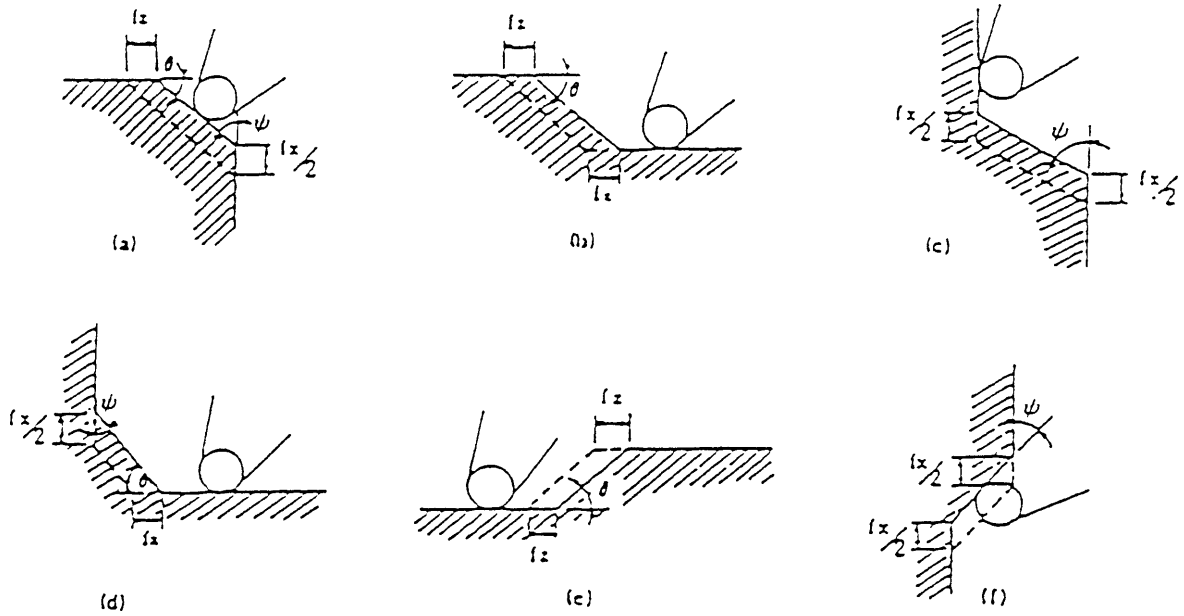
1. Calculate the coordinate values of the intersecting points of a straight line and those of the center of a circular are. (in the above-mentioned figure, coordinate values of the points A, B and C)
2. Calculate the center coordinate values of the nose R to each intersecting point or contact point,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

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

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

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

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

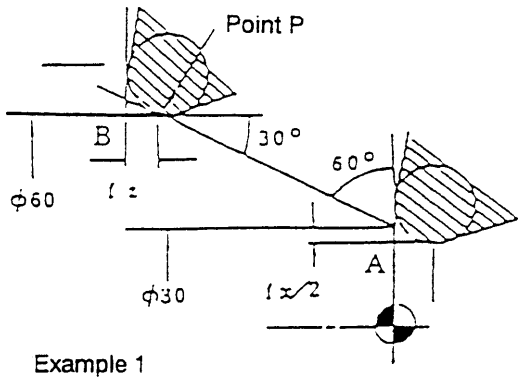
$$\text{(e) } fz = R\left(1 + \tan\frac{\theta}{2}\right)$$

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

Tool nose R (radius) / Angle		0.2	0.4	0.5	0.8	1.0	1.2	1.6
θ mm 5°	fx	0.033	0.067	0.084	0.134	0.167	0.201	0.268
	fz	0.191	0.383	0.478	0.765	0.956	1.148	1.530
10°	fx	0.064	0.129	0.161	0.257	0.322	0.386	0.515
	fx	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.

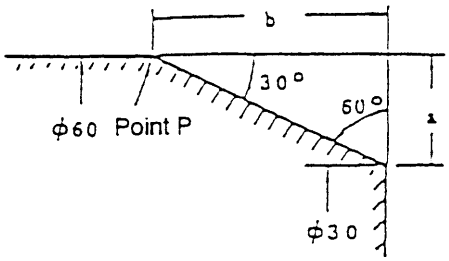
4. Example of tool nose radius compensation amount calculation in chamfering and taper cutting



Example 1

When the tool is located at the positions A and B in the above figure, the tool nose radius compensation amount (fx, fz) 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 ;

$b = \tan 60° \times a$

$= 1.732 \times 15 = 25.98$

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

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

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

$= 2\times0.8\left(1 - \tan\frac{60^\circ}{2}\right)$

$= 2\times0.8\,(1-\tan30^\circ)$

$= 2\times0.8\,(1-0.57735)$

$= 2\times(0.42265)$

$= 2\times0.338$

$= 0.676$

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

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

$= 0.8\times(1-\tan15^\circ)$

$= 0.8\times(1-0.268)$

$= 0.8\times0.732$

$= 0.5856$

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

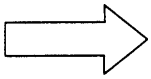
Tool nose point position of the tool A

$X = \phi30 - fx$

$= 30 - 0.676$

$= 29.234$

29.32



Coordinate value

X29.32

Z0

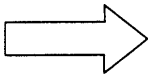
Tool nose point position of the tool B

$Z = P - fz$

$= -25.98 - 0.5856$

$= -26.5656$

-26.57



Coordinate value

X60.0

Z–26.57

(4) Program example

G00 X..... Z ..... T□□ΔΔ

G01 Z0 F .....

X29.32 ← A

X60.0 Z–26.57 ← B

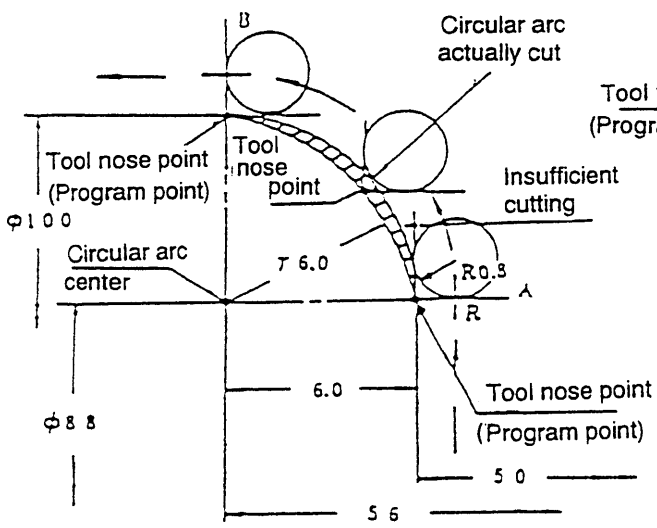
Z–.....

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

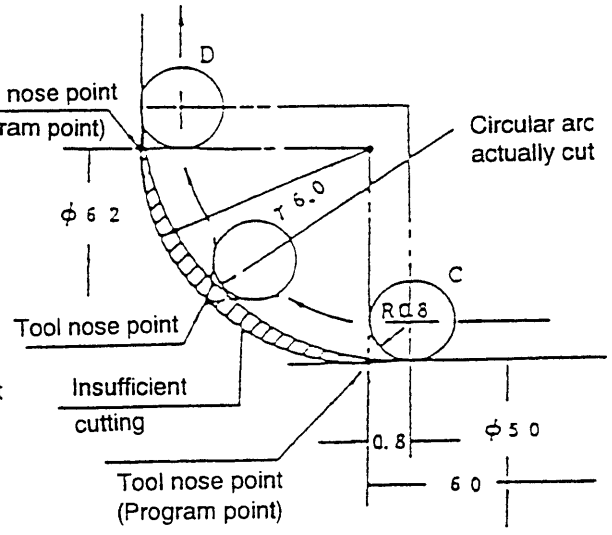
(1) Program example without considering tool nose R compensation amount In circular cutting, a tool cuts a workpiece along its circular are “r” with the nose R being in contact with the arc.

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

Exampli 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;

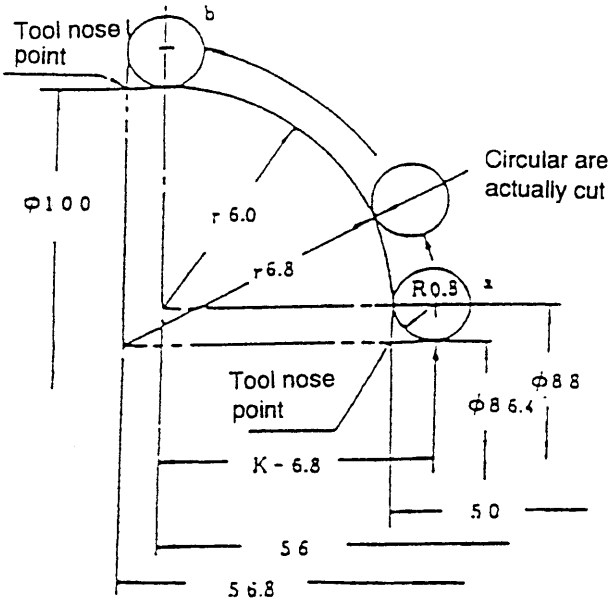
To prevent insufficient cutting of a concave circular arc;

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.

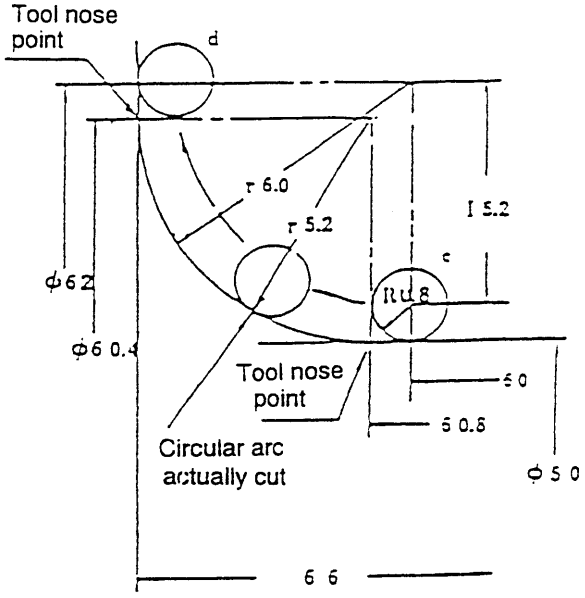
For the convex circular arc, command a circular arc “r” larger by a tool nose radius.

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

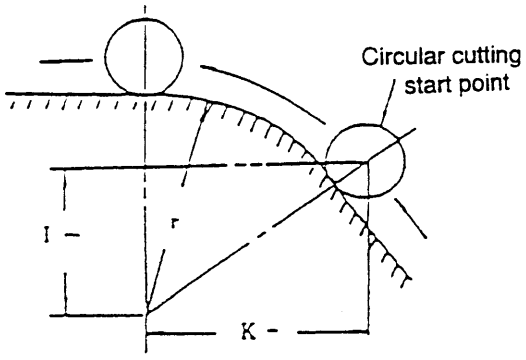
G01 X–50.0 F0.2

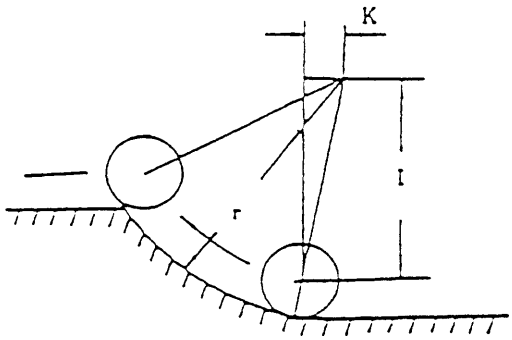
Z–60.8 ← c

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.





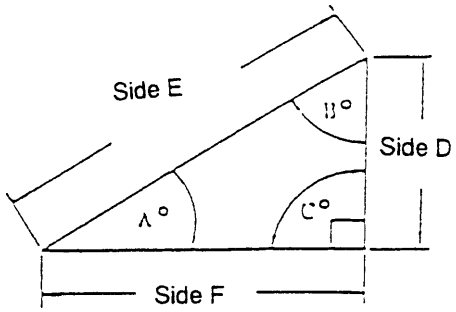
G03 X__ Z__ I–__ K–

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.



$\sin A° = \frac{D}{E}$

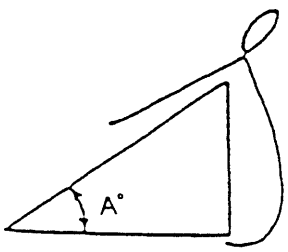
$\cos A° = \frac{F}{E}$

$\tan A° = \frac{D}{F}$

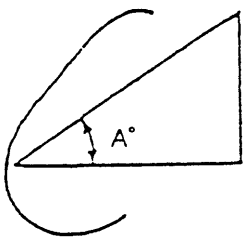
Bottom of the stream can be seen after a water mill stops.

Formulas (for right triangle) $A°+B°+C°=180°$

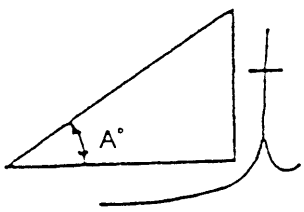
Side and angle given	Formula obtaining side or angle	
Angle “A” and side “D”	$E = \frac{D}{\sin A°}$	$F = \frac{D}{\tan A°}$
Angle “A” and side “E”	$D = E \times \sin A°$	$F = E \times \cos A°$
Angle “A” and side “F”	$D = F \times \tan A°$	$E = \frac{F}{\cos A°}$
Angle “B” and side “D”	$E = \frac{D}{\cos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



Sine



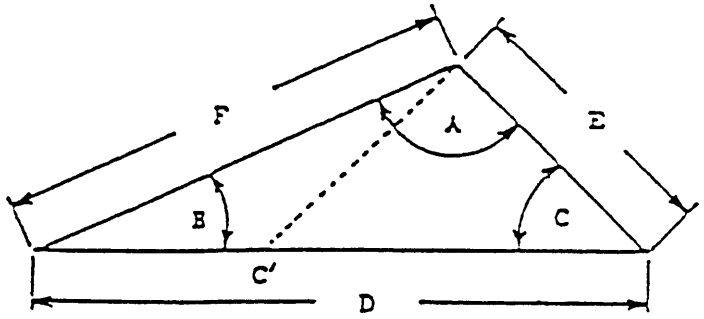
Cosine



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:



$A°+B°+C°=180°$

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

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

(2) When 2 sides (E and F) and an angle (A°) between them are known;

$$D = \sqrt{E2 + F2 - 2 \times E \times F \times \text{coas}A°}$$

(3) When 2 sides (E and F) and 1 opposite angle (B°) are known;

$$\sin C° = \frac{F}{E} \times \sin B° \qquad A° = 180° - B° - C°$$

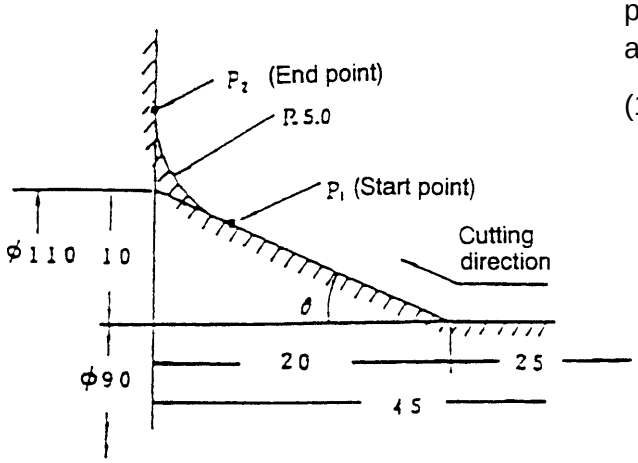
***Note**) Pay attention to existence of double solution as c' for a solution of c.*

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

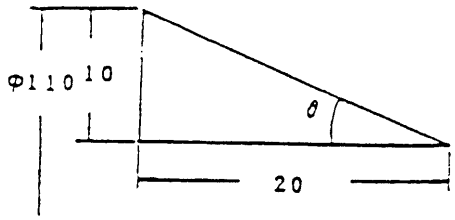
$$E = \frac{\sin B°}{\sin A°} \times D \qquad F = \frac{\sin C°}{\sin A°} \times D$$

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.



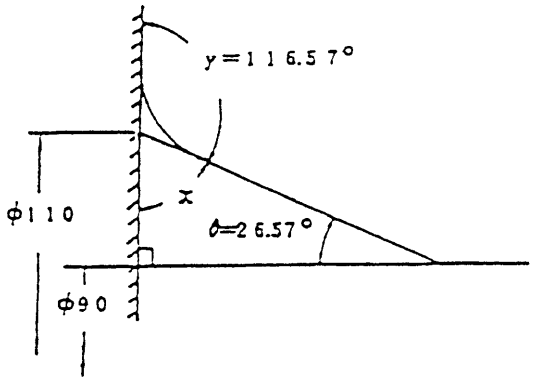
(1) Obtain the taper angle "θ" in the left figure.



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

$$\theta = \tan^{-1} 0.5 = 26.57°$$

(2) Obtain the following angles "x" and "y" from "θ".

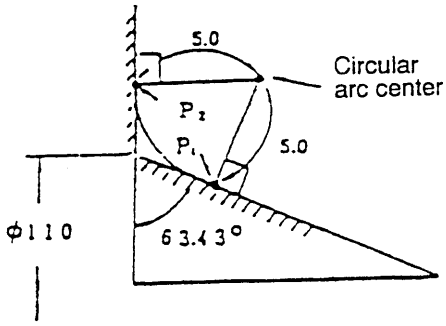


$x = 90° - 26.57° = 63.43°$

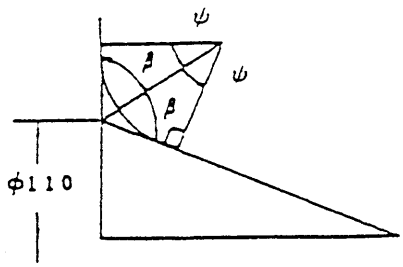
$y = 180° - x$

$y = 180° - 63.43° = 116.57°$

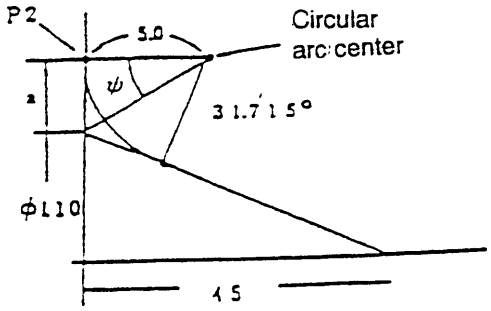
(3) From the start point and the end point of the circular arc to the center of the circular arc, draw lines which are as long as a radius of 5.0.



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



(5) Obtain the length of the side "a".



a = tan 31.715° × 5.0 = 3.08987

a 3.09

β = 116.57°÷2=58.285°

ψ = 90°–58.285°=31.715°

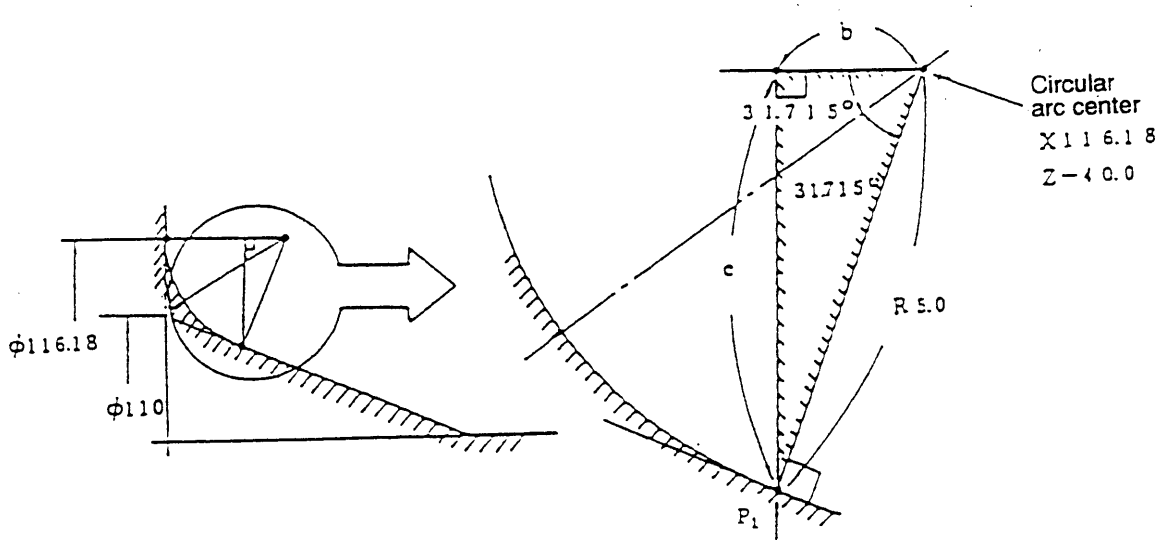
(6) The position of the end point (P_2) of the circular arc is ; X of P_2 = 3.09×2+ϕ110=ϕ116.18

PointP_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”

b = 5.0×cos (31.715°+31.715°)

= 5.0×cos63.43°

= 5.0×0.47729

= 2.24

Length of the side “c”

c = 5.0×sin (31.715°+31.715°)

= 5.0×sin63.43°

= 5.0×0.8943

= 4.47

(9) Based on the calculations on the left, obtain the position of the circular arc start point (P_1) from the circular arc center.

X of P_1 = φ116.18–(C×2)

= φ116.18–(4.47×2)

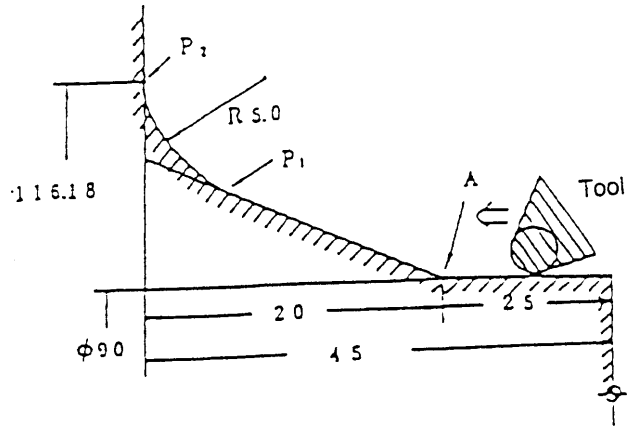
= φ107.24

Z 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

<table>
<tr><th>Classification</th><th>Calculation formula</th><th>Remarks</th></tr>
<tr><td rowspan="3">Spindle</td><td>Cutting speed “V” $V = \frac{\pi \bullet D \bullet N}{1000}$</td><td rowspan="6">V. Cutting speed (m/min)
N. Spindle speed (rpm)
π. Number π(3.1416)
D. Workpiece diameter (mm)
F. Feed per revolution (mm/rev)
f. Feed per minute (mm/min)
L. Total cutting length (depth)
T. Time</td></tr>
<tr><td>Spindle speed “N” $N = \frac{V \bullet 1000}{\pi \bullet D}$</td></tr>
<tr><td>Tool nose Position “ϕD” $D = \frac{V \bullet 1000}{\pi \bullet N}$</td></tr>
<tr><td rowspan="3">Feed</td><td>Max. cutting feed “F $F = \frac{5000}{N}$</td></tr>
<tr><td>Approach feed rate “F” $F \leqq \frac{2000}{N}$</td></tr>
<tr><td>Feed rate per minute “f” $f = F \times N$</td></tr>
<tr><td>Machining time</td><td>Cutting time “T” $T = \frac{\pi \bullet D \bullet L}{V \bullet F \bullet 1000} = \frac{L}{N \bullet F}$</td></tr>
<tr><td rowspan="5">Thread cutting</td><td>Thread lead $L = n \times P$</td><td rowspan="5">L. Thread lead
P. Thread pitch
n. No. of threads
N. Spindle speed
δ_1. Incomplete thread area (before machining)
δ_2. Incomplete thread area (after machining)
K_1, K_2. : constant

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

6. SPECIFICATIONS OF C-AXIS CONTROL (SEIKI-SEICOS Σ21L)

6-1 Outline

The spindle can be controlled by the feed motor. It enables the spindle to position precisely, and it enables X and Z axis, and the spindle to interpolate.

The name of axis is called C-axis.

This instruction manual describes the C-axis, and as to the SEIKI-SEICOS Σ21L Standard functions, refer to the separated instruction manual.

6-2 Standard Specifications

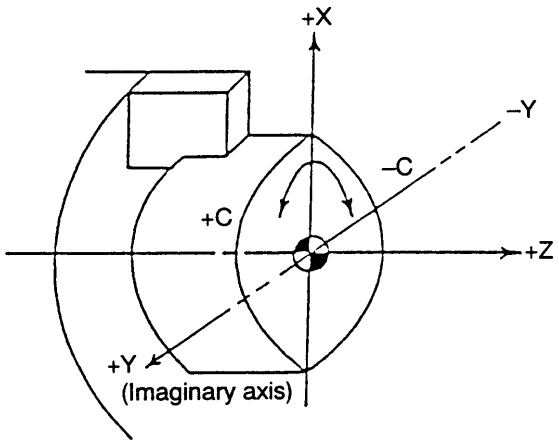
1. Controlled axis C
2. Simultaneous controllable axis ..... 3 axis, X, Z and C
3. Least input incremental unit 0.001deg
4. Least input traveling unit ..... Refer to the instruction manual.
5. Maximum commandable value ..... ± 99999.999deg
6. Decimal point input ..... Available (decimal point position is of degree unit)
7. Rapid traverse rate ..... Refer to the instruction manual.
8. Cutting feed rate ..... Refer to the instruction manual.
   (F and E functions)
9. Automatic acceleration/ deceleration ..... Linear acceleration/ deceleration ..... (Positioning, manual feed)
   Exponential acceleration/deceleration ..... (Cutting feed)
10. Feed rate override ..... Rapid traverse override, cutting feed rate override and feed override cancel
11. Absolute/incremental programming ..... C: Absolute programming
    H: Incremental programming
12. Coordinate system setting ..... G50 C … ;
13. Positioning ..... G00 C … ;
14. Exact stop ..... G09 C … ;
15. Linear interpolation ..... G01 C … ;
    (Simultaneously 3 axis interpolation X-Z-C)
16. Circular interpolation ..... Circular interpolation with C-axis is impossible

17. Reference point return ..... G27: Reference point return check
    G28: Reference point return
    In the reference point return, rotating axis processing is performed. (The reference point return is completed within 360°)
    G29: Return from the reference point
18. Feed per minute/feed per revolution ..... G98: Feed per minute (mm/min)
    G99: Feed per revolution (mm/REV) (synchronized with that of rotating tool spindle)
    Note) *In this case, it is necessary to set PC on the rotating tool spindle, and of its feedback pulse to be 4069P/REV.*
19. Tool position offset ..... Tool offset for C-axis is not available.
20. Backlash compensation ..... 32767 pulse MAX
21. Manual feed ..... Rapid traverse (RAPID)/Manual jog feed (JOG)/Manual handle feed (STEP)
22. Manual reference point return ..... Processed by the high speed and reduction LS using type rotating shaft.
23. Operation support switch ..... Single block
    Block skip
    Dry run (processed through coverting the value of mm/min designated by X and Z into that of deg/min.)
    Machine lock
    Manual absolute (fixed to “ON” for C-axis as well as X and Z axes.)
24. Feed hold (Halt) ..... Effective
25. Buffering function ..... Effective
26. Inch/Metric conversion ..... Regarding C-axis, both inch and mm system is 0.001°.
27. The second reference point return ..... Possible
28. Automatic coordinate system setting ..... At the time of manual reference point return, coordinate system is set automatically.
29. Work coordinate system shift ..... Set it by work shift screen.
30. Stored stroke limit ..... Ineffective for C-axis
31. Chamfer/Corner R ..... Impossible for C-axis
32. Return to machining ..... Automatic return to the position changed over to the interrupting point manual mode available for C-axis as well.

6-3 Program

6-3-1 Coordinate Axis

The C-axis is included in the ordinary cutting coordinate system. Each coordinate axis and signs are defined as follows.



As a matter of fact the Y-axis not exists, however, prepare a program as if imaginary Y-axis exists.

6-3-2 Plane Selection of G17, G18, G19

Designate a plane executing circular interpolation and tool radius compensation etc. by G17, C18 or G19.

A plane should be designated either turning or machining by rotating tool.

Command form

G17 $X_P\_Y_P\_$; $X_P\_Y_P$; plane

G18 $Z_P\_X_P\_$; $Z_P\_X_P$; plane

G19 $Y_P\_Z_P\_$; $Y_P\_Z_P$; plane

Provided that, X_P: X-axis or it's parallel axis.

Y_P: Y-axis or it's parallel axis.

Z_P: Z-axis or it's parallel axis.

Select G18 (A plane of $Z_P\_X_P$) at normal turning, G19 (A plane of $X_P\_Y_P$) at machining from direction of Z-axis.

Since a condition at power on is G18, plane selection must be done by direction of machining.

(Note) 1. *If a plane is not fixed at a block commanded by G17, G18 or G19, it becomes an alarm.*

6-3-3 Miscellaneous Function for Rotating Tool (M Code)

In case of hole machining, you can use these codes to specify start, stop and reverse rotation of the tool.

M13 Rotating tool connection + Rotating tool forward rotation
M14 Rotating tool connection + Rotating tool reverse rotation
M15 Rotating tool stop + Spindle positioning
(In case of simple spindle stop, it is available the same spindle stop code M05 for the turning spindle. Never fail to use M05 when stopping the rotating tool in tapping process.)
M40 C-axis connection release
(Release is available by the low-speed range selection code of the cutting spindle.)
M41 C-axis connection release
(Release is available by the high-speed range selection code of the cutting spindle.)
M43 C-axis connection (Incl. spindle indexing)
M44 Rotating tool connection
M45 Rotating tool connection release

Note: *1) For C-axis specifications, the hole machining canned cycles are added both to X-axis and Z-axis.*
2) H is used to specify C-axis incremental.
3) Execute the reference point return of the C-axis after connection of the C-axis.

6-3-4 Fixed Cycle for Hole Making G80~G89, G831, G841, G861

With this function, machining cycle such as drilling, tapping or boring can be commanded by one block.

Furthermore, in case of making the same hole repeatedly, just command hole position and it is very effective to simplify a program.

(1) Command form

[G198 / G199] G_X_Z_R_D_Q_P_C(H)_L_F_E_ ;

G198/G199 : Return point

G_ : G code for fixed cycle (G81~G89, G831, G841, G861)

X_ : Hole position (Note 3)

Z_ : Position of Z point (Note 3)

R_ : Position of R point (Coordinate value of diameter)

D_ : Position of D point (Incremental diametral command from R point)

Q_ : Cutting depth of G831 or C83 or shift amount of G861, always radius value.

P_ : Dwelling time

C_ : Rotating angle of C-axis

L_ : Number of times of repetition

F_ : Cutting feedrate

E_ : Cutting feedrate

(Note 1) *If omit a number of times of repetition (L), it deems as L=1. If L=0 is commanded, move to hole making position but hole making is not executed.*

(Note 2) *I, J or K is used at G83, G831 or G861 as well.*

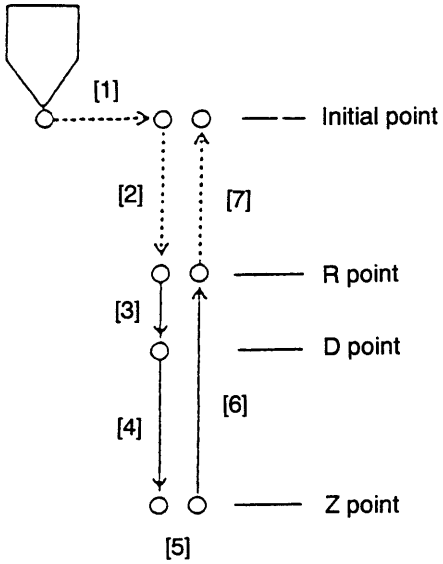
(Note 3) *Command a position of Z point by an address of axis of hole making axis. Command a position of hole making by an address of axis other than hole making axis.*

(Note 4) *R and Z points, P, Q, I, J and K etc. are modal during fixed cycle.*

(Note 5) *Incremental command on B-axis can't be used during fixed cycle of hole making.*

(2) Machining cycle

Machining cycle of fixed cycle consists of following motion [1] ~ [7] generally.



[1] Positioning to hole making position
[2] Rapid traverse up to R point
[3] Cutting feed to D point (Feedrate E)
[4] Machining of hole up to Z point (Feedrate F)
[5] Motion at Z point
[6] Return to R point
[7] Rapid traverse to initial point

Provided, ·····→ Rapid traverse
——→ Cutting feed

(Note 1) *Motion between [4] and [6] does not stop by single block.*

(Note 2) *If E is omitted, moving section by feedrate E moves by feedrate F.*

(Note 3) *If omit D, motion [3] is ignored.*

(Note 4) *Command E at tapping cycle G84 or G841, it becomes feedrate of motion [6].*

(3) Return point

Return point of fixed cycle command following G code.

G198 Initial point level return

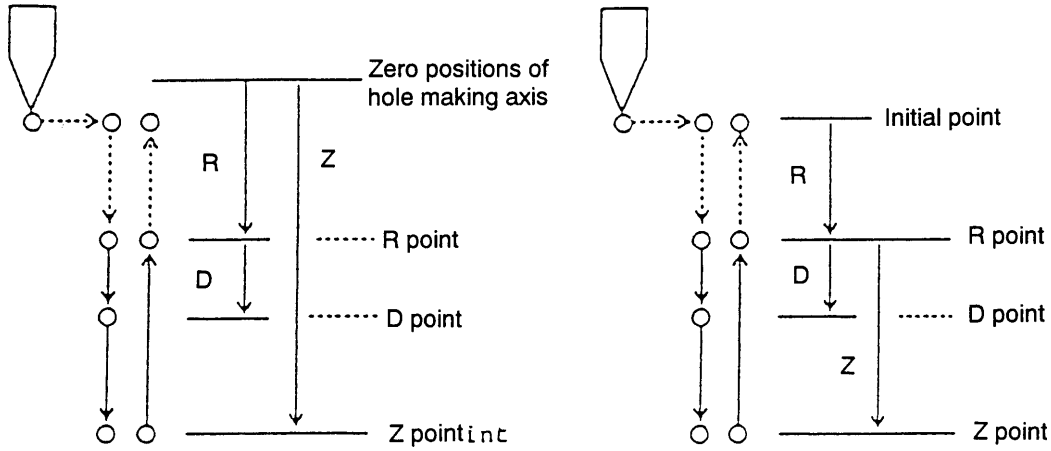
G199 R point level return

(Note) *An initial point is a position of hole making axis at time of fixed cycle mode from a condition of fixed cycle cancel.*

(4)“R point”, “Z point” and “D point”

R and Z points are available both absolute and incremental command, however, D point is commanded by incremental always.

[Absolute] [Incremental]



(Note) *D point is incremental position from R point and at the time of machining from diametral direction, indicates by diametral value.*
Incremental command of the B axis by “D” is not available during fixed cycle.

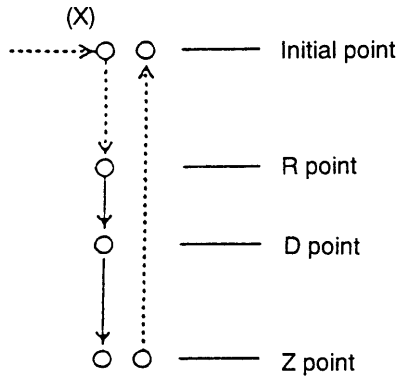
(5) Explanation of motion of fixed cycle

In this explanation of motion of fixed cycle, positioning axis hole making position is X and hole making axis is Z.

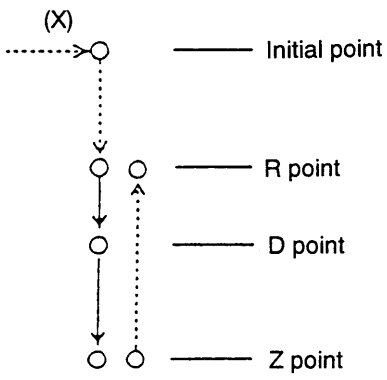
(a) G81 (Drilling, Spot drilling)

[G198 / G199] G81 X_Z_R_D_C(H)_L_F_E_ ;

[G198]



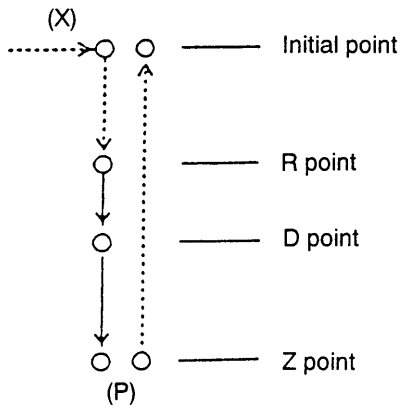
[G199]



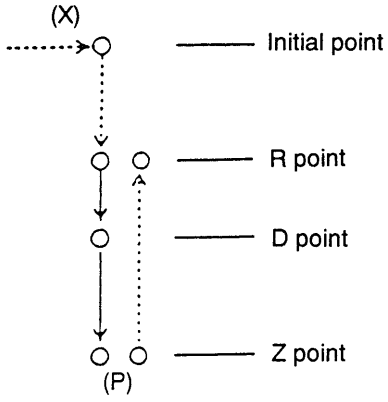
(b) G82 (Drilling, Counter boring)

[G198 / G199] G82 X_Z_R_D_P_C(H)_L_F_E_ ;

[G198]



[G199]

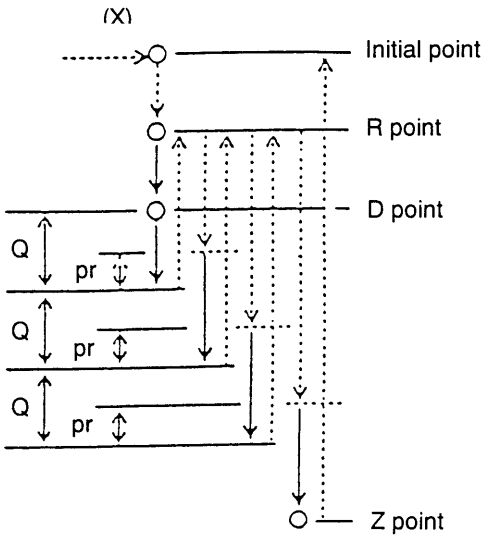


(P): Dwell

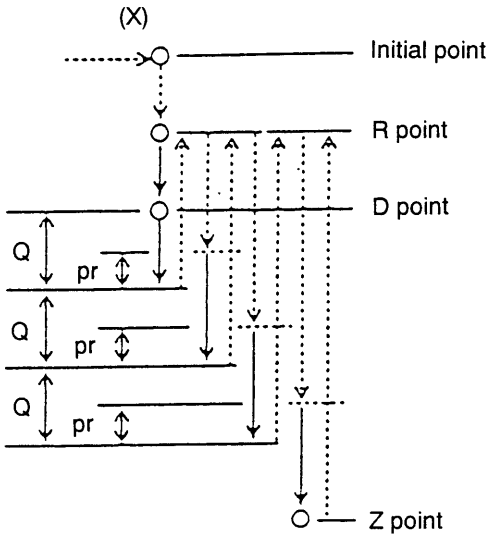
(c) G83 (Deep hole drilling)

[G198 / G199] G83 X_Z_R_D_Q_C(H)_L_F_E_ ;

[G198]



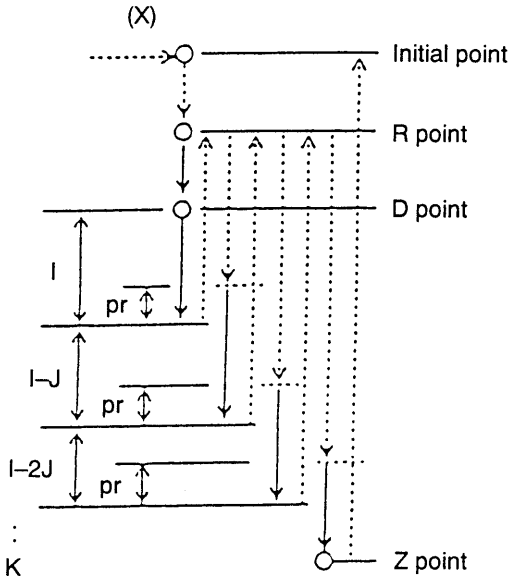
[G199]



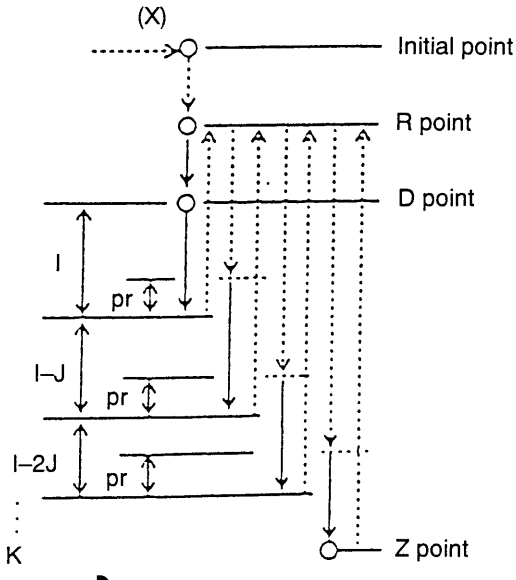
Set a clearance amount Pr on the parameter No.6222.

[G198 / G199] G83 X_Z_R_D_I_J_K_C(H)_L_F_E_ ;

[G198]



[G199]



Provided; I : Initial value of cutting amount (Positive value)
J : Subtruction value after the second time (Positive value)
K : Final value of cutting amount (Positive value)

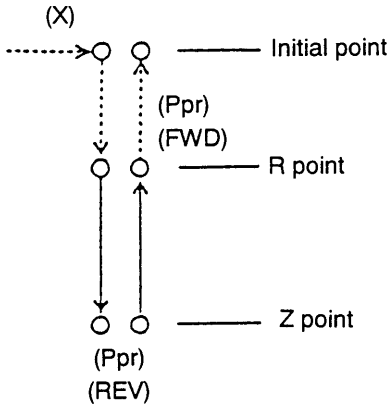
} Always radius value

(Note) *In case of existence of command Q before command a variable pitch by I, J or K, command Q as zero.*

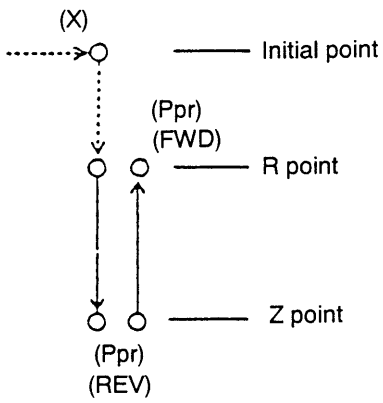
(d) G84 (Tapping)

[G198 / G199] G84 X_Z_R_P_C(H)_L_F_E_ ;

[G198]



[G199]



(FWD) : Forward rotation of tool
(REV) : Reverse rotation of tool
(Ppr) : Dwell (Parameter setting)

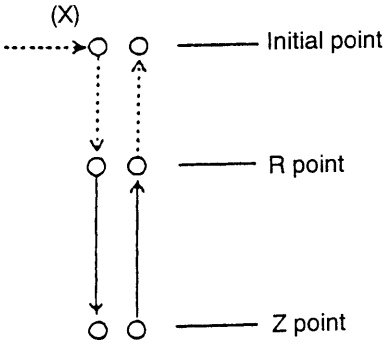
(Note 1) *A dwell by command P can be ineffective by parameter setting.*

(Note 2) *Feed hold and feedrate override are ineffective while cutting.*

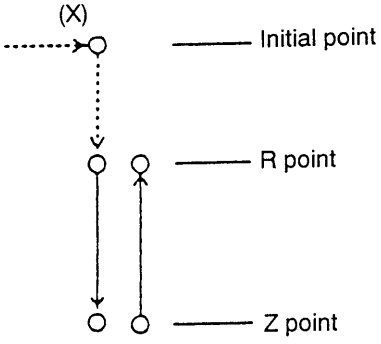
(e) G85 (Boring)

[G198 / G199] G85 X_Z_R_C(H)_L_F_ ;

[G198]



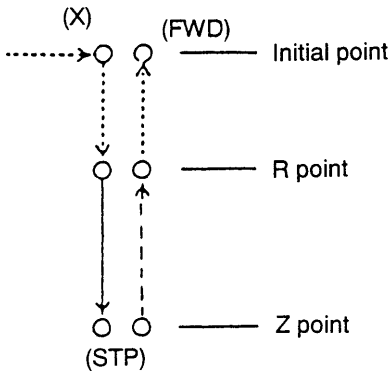
[G199]



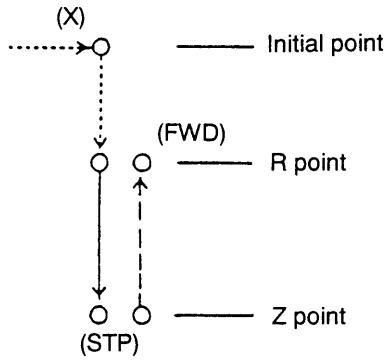
(f) G86 (Boring)

[G198 / G199] G86 X_Z_R_C(H)_L_F_ ;

[G198]



[G199]

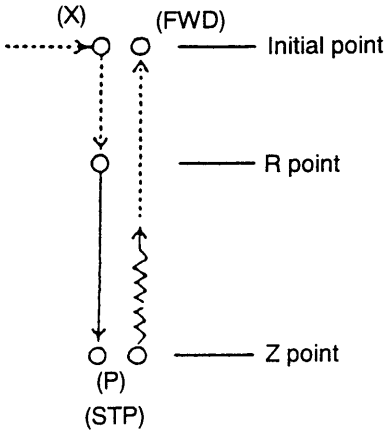


(FWD) : Forward rotation of tool
(STP) : Stop of tool

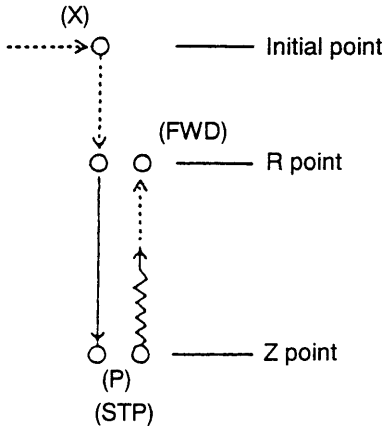
(g) G88 (Boring)

[G198 / G199] G88 X_Z_R_P_C(H)_L_F_ ;

[G198]



[G199]



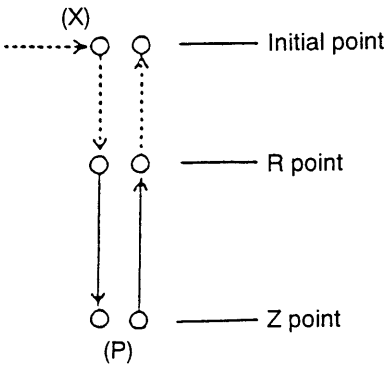
(P) : Dwell
(FWD) : Forward rotation of tool
(STP) : Stop of tool
∿∿∿→ : Manual feed

(Note) *A tool reaches to Z point and stop a rotation of tool after dwell, it becomes single block stop condition automatically. If select a manual mode, manual feed is available.*
If select an automatic mode then press "START" button, restart an automatic operation.

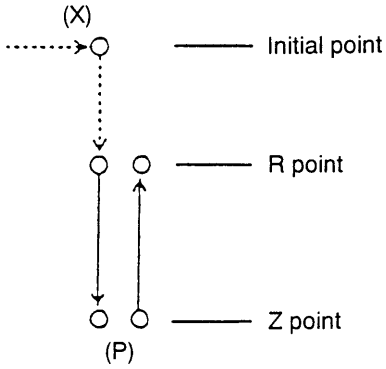
(h) G89 (Boring)

[G198 / G199] G89 X_Z_R_P_C(H)_L_F_ ;

[G198]



[G199]

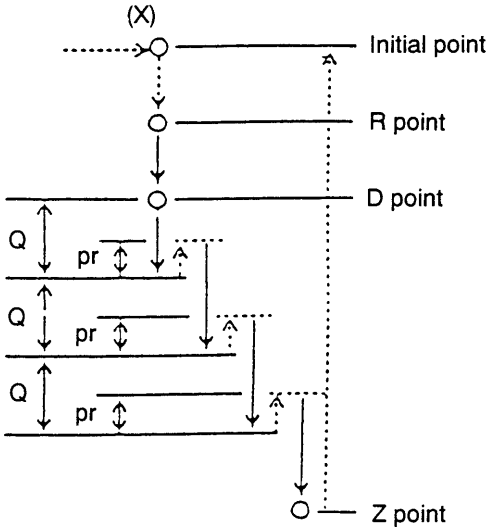


(P) : Dwell

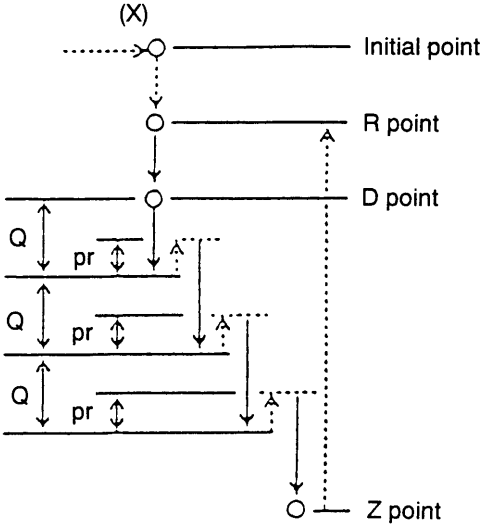
(i) G831 (High speed deep hole drilling)

[G198 / G199] G831 X_Z_R_D_Q_C(H)_L_F_E_ ;

[G198]



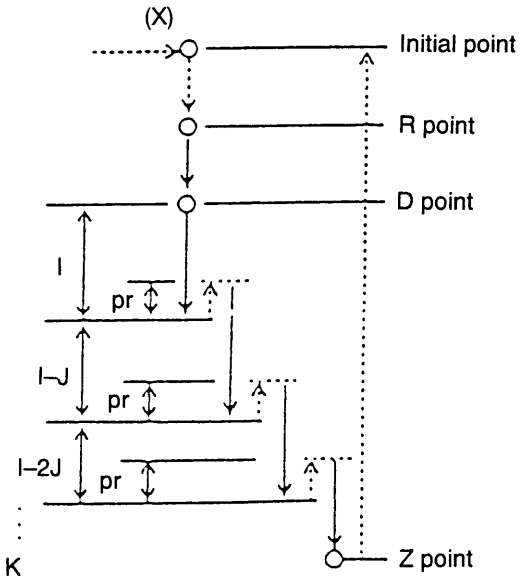
[G199]



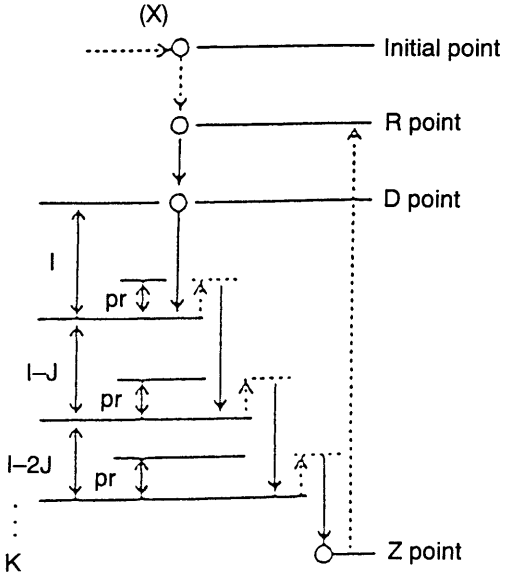
Set a clearance amount Pr on the parameter No.6222.

[G198 / G199] G831 X_Z_R_D_I_J_K_C(H)_L_F_E_ ;

[G198]



[G199]



Provided; I : Initial value of cutting amount (Positive value)
J : Subtraction value after the second time (Positive value)
K : Final value of cutting amount (Positive value)

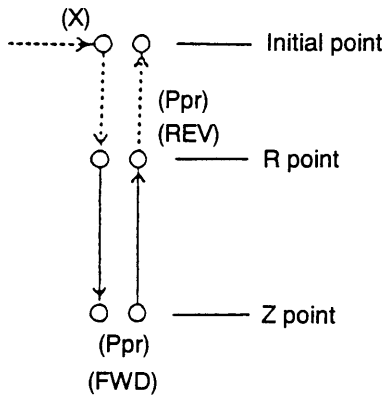
} Always radius value

(Note) *In case of existence of command Q before command a variable pitch by I, J or K, command Q as zero.*

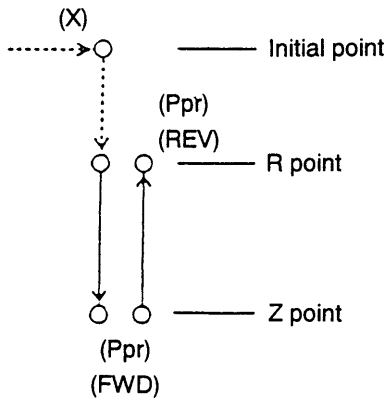
(j) G841 (Reverse tapping)

[G198 / G199] G841 X_Z_R_P_C(H)_L_F_E_ ;

[G198]



[G199]



(FWD) : Forward rotation of tool
(REV) : Reverse rotation of tool
(Ppr) : Dwell (Parameter setting)

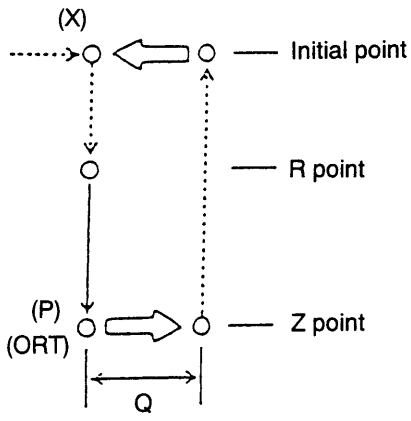
(Note 1) *A dwell by command P can be ineffective by parameter setting.*

(Note 2) *Feed hold and feedrate override are ineffective while cutting.*

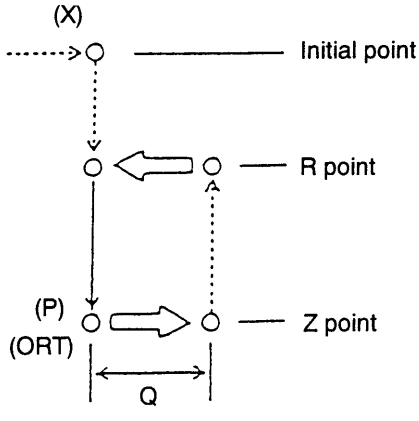
(k) G861 (Fine boring)

[G198 / G199] G861 X_Z_R_P_Q_C(H)_L_F_ ;

[G198]



[G199]



(P) : Dwell
(ORT) : Tool orientation and stop
⇨ : Shift (Rapid traverse)

(Note) *Set a shift amount Q at the parameter No.6212.*

(6) Precautions

(1) When single block is ON, stop at the end point of motion [1] [2], [3] and [7]. In this case a feed hold lamp light at the end point of motion [1], [2], [3] and the end point of motion [7] if remain the number of times of repetition.
A cycle motion between [4] and [6] other a tapping cycle (G84, G841) can be stopped on single block by setting of parameter.

(2) If the “FEED HOLD” button is pressed at the motion [4] ~ [6] of G84 or G841, a feed hold button lights immediately and a motion stops after execution of motion continuously up to [7].

(3) A feedrate override is fixed at 100% during a motion [4] ~ [6] of G84 or G841.
Effective or ineffective of dry run can be selected by setting of parameter.

(4) Q, P, I, J and K should command in the block contains axis command. P, Q, I, J and K are not handled as a data of fixed cycle in the other blocks.
Also, P, Q, I, J and K are not handled as a data of fixed cycle in the block commanded by G code of 00 group except G09.

(5) A fixed cycle mode will be cancelled by commanding G80 or G code of 01 group such as G00, G01 etc.

(6) If M or S is commanded in a block of fixed cycle command, it issues the first motion [1], positioning to hole making position.
If number of times of repetition (L) is commanded, M and S issues at the first time only.

(7) Command by positive value for the numerals of P, Q, I, J, K, L, F or E etc.

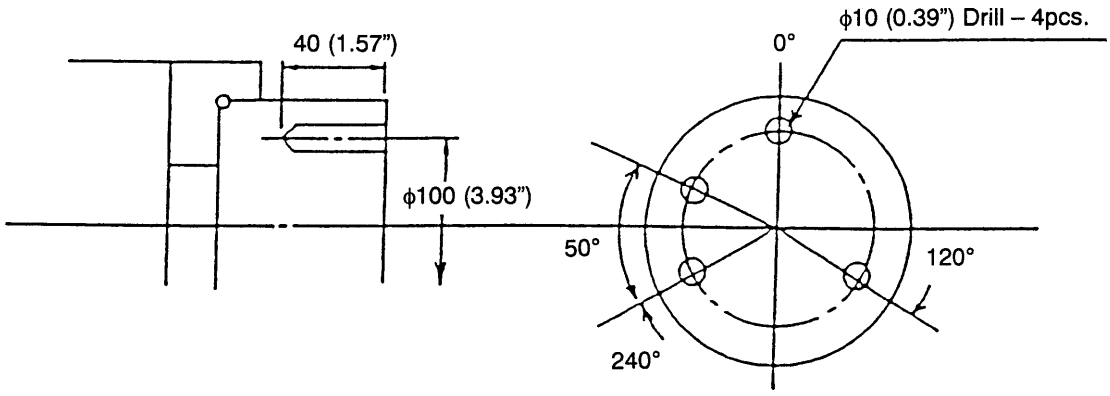
(8) Command Q, I, J and K by radius designation always.

(9) R and D are diamtral designation.

(10) When G17~G19 going to be changed, execute it after a fixed cycle is cancelled.

6-3-5 Program Example

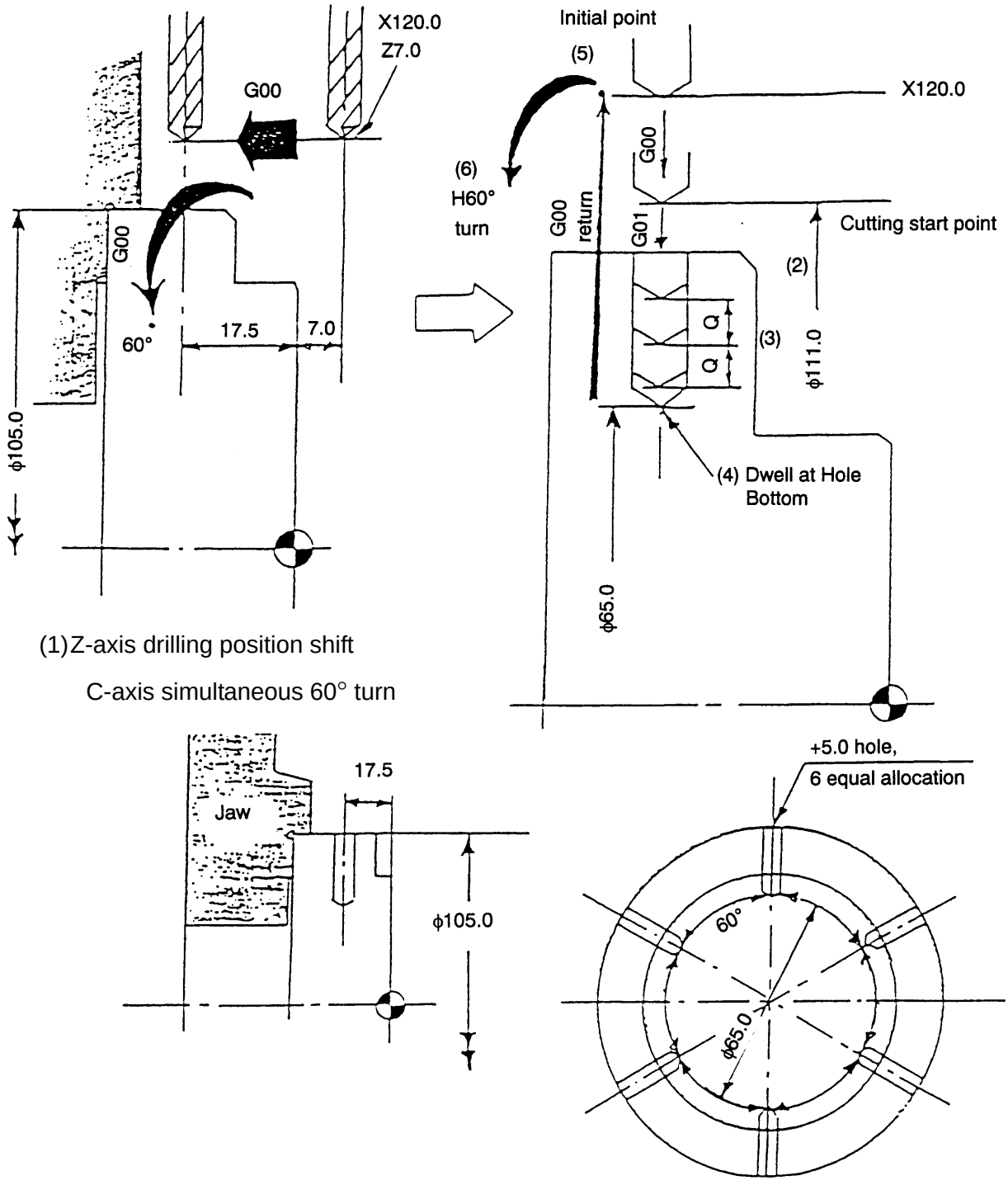
Example 1: Drilling (Z-axis Rotating Tool)



OΔΔΔΔ
G28 U0
G28 W0
G18 ..... X-Z plane designation
⌇
Turning
⌇
N600
T0600 M40 ..... 6th turret face selection•Spindle low speed sideselection
C-axis connection release
G17 ..... X-Z plane designation
M43 ..... C-axis connection
G28 H0 ..... C-axis zero return
G50 C0 ..... C-axis coordinate setting
G97 S800 M08 ..... Rotating tool rotating speed command. Coolant ON
G00 X100.0 Z10.0 C0 M13 ..... Axial travel + Rotating tool forward rotation start
Z2.0
G01 G98 Z–40.0 F120
G00 Z2.0
C120.0 ..... C-axis absolute command
G01 Z–40.0
G00 Z2.0
C240.0
G01 Z–40.0

```
G00 Z2.0
    H50.0 ..................................  C-axis incremental command
G01 Z–40.0
G00 Z2.0 M09
G00 X200.0 Z200.0 M05 .....  Return to index position + Rotating tool rotation stop
G28 H0 .....................................  C-axis zero return
M45 .....................................  Rotating tool connection release
M01
 .
 .
 .
 .
```

Example 2: Drilling (X-axis Rotating Tool)



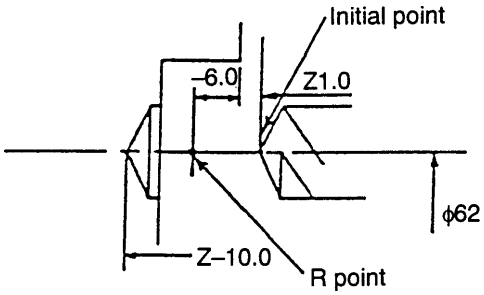
(1) Z-axis drilling position shift

C-axis simultaneous 60° turn

```
N400
T0400 M40
G19
G23 ..... Stored stroke 2 turned off
G97 S100 M05
M43 ..... C-axis connection
G28 U0 H0 ..... X-axis and C-axis zero return
G50 C0 ..... C-axis coordinate setting
G98 S1270 M08 ..... Rotating tool spindle speed (Feed: mm/min)
G00 X120.0 Z7.0 M13 ..... Rotating tool forward rotation
G198 ..... Initial point return command
G83 Z−17.5 H60.0 X65.0 R110.0 P0.5 Q3.0 L6 F127 } X-axis peck drilling cycle
G80 ..... Peck drilling cycle canceled
G28 U0 H0 M09
G00 Z200.0 M05
M45 ..... Rotating tool connection release
G99 M40 ..... C-axis connection release (Feed: mm/rev)
G22 ..... Stored stroke 2 turned on
M01
```

Example 3: Drilling (Z-axis Rotating Tool)

```
N1000 (D6.5 – DRL)
T1000 M40
G17 ..... X-Y plane designation
M43 ..... C-axis connection
G28 H0 ..... C-axis zero return
G50 C0 ..... C-axis coordinate setting
G97 S1500 M08
G00 X62.0 Z5.0 M13 ..... Rotating tool forward rotation
G98 G01 Z1.0 F5000 ..... Feed per minute (mm/min)
G198 ..... Initial point return
G81 Z–10.0 H60.0 R–6.0 P1.0 L6 F130
G80 ..... Fixed cycle cancel
G00 Z5.0
G99 M40 ..... C-axis connection release
G00 X200.0 Z200.0 M05
M45 ..... Rotating tool connection release
M01
```



Example 4: Drilling and Tapping (Z-axis Rotating Tool)

N400 (D4.2 – DRL)

T0400 M40

G17

M43

G28 H0

G50 C0

G97 S2000 M08

G00 X62.0 Z15.0 M13

G98 G01 Z10.0 F5000

G199

G83 Z–26.0 H60.0 R3.0 P1.0 Q4.0 L6 F200

G80

G00 Z5.0

G99 M40

G00 X200.0 Z200.0 M05

M45

M01

N800 (M5 * P0.8)

T0800 M40

G17

M43

G28 H0

G50 C0

G97 S300 M08

G00 X62.0 Z20.0 M13

G98 G01 Z5.0 F5000

G84 Z–8.0 H60.0 L6 F240

G80

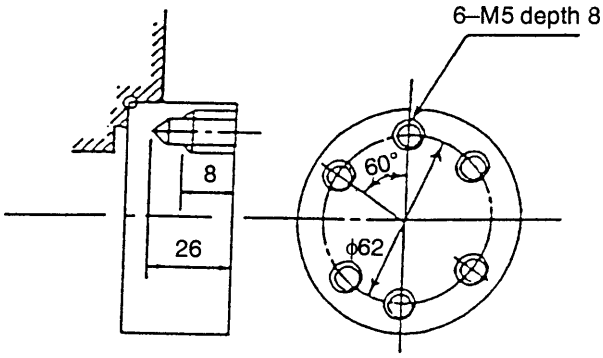
G00 Z5.0

G99 M40

G00 X200.0 Z200.0 M05

M45

M01



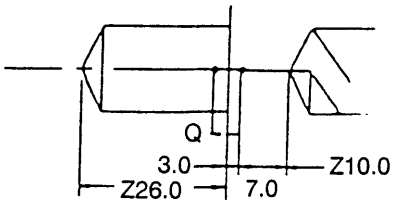
H60.0 → Angle

R3.0 → R point

P1.0 → Dwell

Q4.0 → Depth of cut

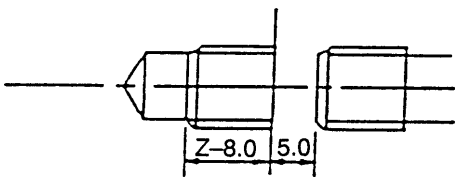
L6 → No. of repeats (6 equal allocation)



F240 → Spindle speed × Pitch

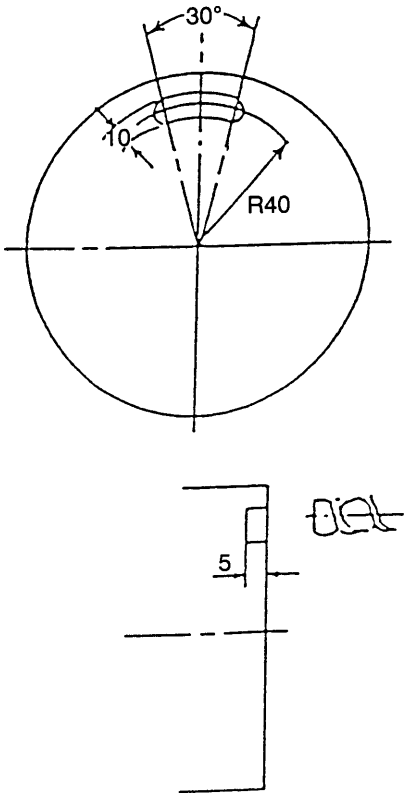
300min^{-1} × 0.8=240

(A feed rate should be reduced several percentage from a calculated value when a tension type tap holder is applied.)



Example 5: End-milling (Z-axis Rotating Tool)

```
N200 (D10.0 – MIL)
T0200 M40
G17
M43
G28 H0
G50 C0
G97 S500 M08
G00 X80.0 Z5.0 C–15.0 M13
G98 G01 Z1.0 F3000
Z–5.0 F25
C15.0 F50
G00 Z5.0
G99 M40
G00 X200.0 Z200.0 M05
M45
M01
```



6-4 Polar Coordinate Interpolation Function

6-4-1 Polar Coordinate Function

A workpiece can be machined into an arbitrary shape with the linear axis (X-axis) and rotary axis (C-axis).

If G121 is specified, polar coordinate interpolation is put into effect and a virtual coordinate system is set assuming the zero point of the absolute coordinate system as that on the X-C plane.

A polar coordinate interpolation is executed on this plane.

If G121 is specified, a current C-axis position is assumed as “0”. Therefore, it is necessary to return the C-axis to the program zero point before specifying G121.

Polar coordinate interpolation allows cutting by G01, G02 or G03. A feedrate is oo mm/min.

Either absolute programming (X, C) or incremental programming (U, H) Is available.

Polar coordinate interpolation allows selection of diameter/radius designation for X-axis and C-axis commands.

In case of execution of circular interpolation (G02, G03), designation of radius of arc performs by the address R.

Polar coordinate interpolation is executed over a shape after tool radius compensation.

However, command G120 and G121 at the cancel mode of tool radius compensation (G40).

G120 command turns off the polar coordinate interpolation function.

6-4-2 G Function

There are limit of commandable G code during G121 mode.

Commandable G code	G00 G01 G02 G02 G03 G04
	G09 G40 G41 G42
	G65 G66 G67
	G98 G99

(Note) *1. Command the G120 or G121 in the individual block. If it is not individual, it becomes an alarm.*

2. A plane (any one of the G17, G18 or G19) before the G121 has commanded is cancelled once by a command of the G121 and returns by a command of the G120.

3. During a polar coordinate interpolation should be G98 (feed per minute).

4. During the G121 mode, the T or S command is not available.

5. The following functions are not available for a block during the G121 mode.

- *Program restart*
- *Block restart*
- *Return to the interrupted point during machining*
- *Manual intervention by the manual absolute ON*

6. Prepare a program the coordinate axis of the X should be corresponded to the real coordinate axis of the X.

7. The G00 command is available during the polar coordinate interpolation mode.

However, the end point only changes to the position of polar coordinate system and a path is not changed for the G00 block.

Also, a positioning system becomes a linear type and a compound travelling speed becomes a setting speed at the parameter (No.1468). A positioning system and compound travelling speed is the same for the axis have no relation with the polar coordinate interpolation.

Example) When G00 X100.0 G50.0 ; is commanded at the G121 mode becomes the same path which is commanded by the G01 X141.421 G45.0 ; at the G120 mode and a speed applies a parameter value and acceleration or deceleration speed becomes the same as the G00.

Also, a movement of rotary axis by G00 is a short cut (movement within ±180°) to reduce a machining time.

At the time of approach to a workpiece or retract from the workpiece, if the G00 is commanded during the polar coordinate interpolation mode, may be interfared to the workpiece.

Avoid a command of G00 at the polar coordinate interpolation mode if possible.

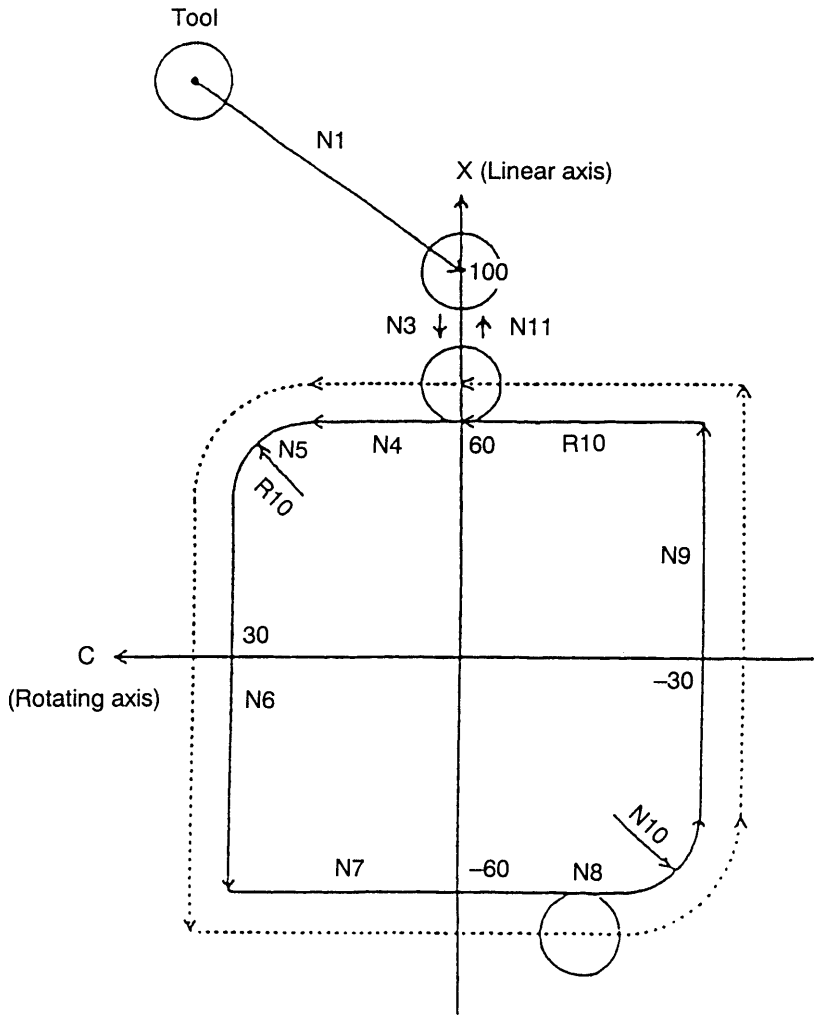
8. At the polar coordinate interpolation, a shape which is programmed by the rectangular coordinate changes a movement of a linear and rotary axes.

Therefore, a speed of rotary axis at the movement near the center of workpiece (reference point of coordinate) becomes large and it may beyond the limit speed of the machine.

In this case, clamp a speed of rotary axis at the setting speed of the parameter (No.1443 : clamping speed of cutting feedrate per each axis) and prevent a speed too fast.

6-4-3 Program Example (X-axis : Linear axis/C-axis : Rotating axis)

N1 G00 X100.0 C0 ;	Positioning to the start point
N2 G121 ;	Polar coordinate interpolation starts
N3 G42 G01 X60.0 F100 ;	(Tool radius compensation right side)
N4 C20.0 F60 ;	
N5 G03 X40.0 C30.0 R10.0 ;	
N6 G01 X–60.0 ;	
N7 C–20.0 ;	
N8 G03 X–40.0 C–30.0 R10.0 ;	
N9 G01 X60.0 ;	
N10 C0 ;	
N11 G40 X100.0 F100 ;	
N12 G121 ;	Polar coordinate interpolation cancel



6-5 G40, G41, G42, G140, G143, G145 Tool Radius Compensation Function

Generally, an imaginary tool nose point at 0 or 9 can not be applied a tool radius compensation, however, at the time of G143 mode, a tool radius compensation can be effective by G145 at an imaginary tool nose point 9.

However, a plane designation by G17, G18 or G19 must be set previously.

G140 : Cancel mode of automatic tool nose R compensation/tool radius compensation

G143 : Automatic tool nose R compensation effective mode (At the time of power on and reset, a control becomes this mode.)

G145 : Tool radius compensation effective mode

For an information of tool radius compensation, input a compensating amount of tool radius (radius value of tool) to R and an imaginary to 1 point to T of an applied tool compensating No.

In case of an imaginary tool point 9, approach by G140 mode and a tool radius compensation becomes effective by G145.

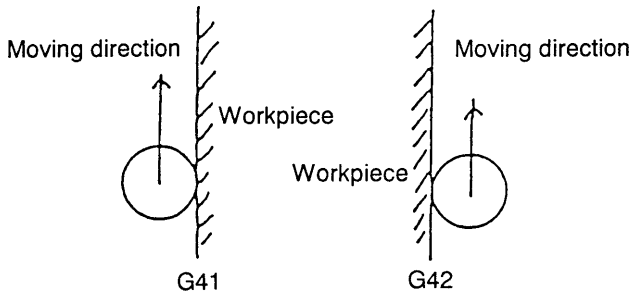
In case of a tool radius compensation effective, a tool radius compensating program is available by G40, G41 or G42.

6-5-1 Direction of Tool Radius Compensation

G40 : Tool radius compensation cancel (A tool moves on a path of program.)

G41 : Tool radius compensation, left side (A tool offsets of workpiece toward a moving direction of tool.)

G42 : Tool radius compensation, right side (A tool offsets right side of workpiece toward a moving direction of tool.)



6-5-2 Movement of Tool Radius Compensation

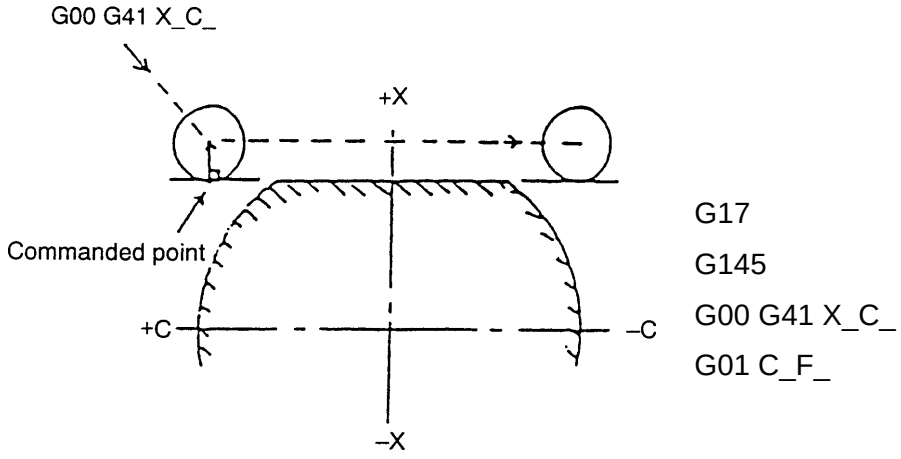
In case of execution of tool radius compensation, a program starts a status of compensation cancel (G40) and command a tool radius compensation mode (G41, G42) then completes after command a compensation cancel status again.

Divide it three conditions and each block calls as follows;

1. Start up
2. Tool radius compensation mode
3. Tool radius compensation cancel

1. Start up

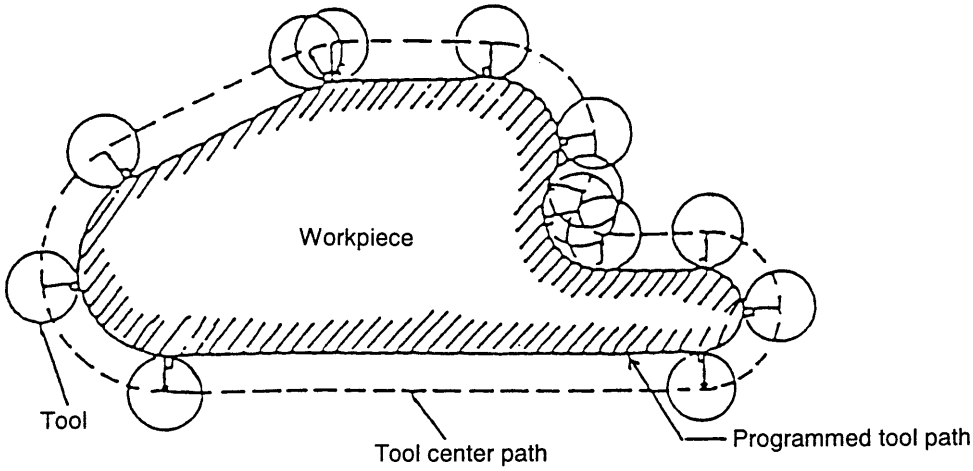
   1. A block changed over from the status of tool radius compensation cancel (G40) to the status of tool radius compensation mode (G41 or G42) is called as a block of start up.
   2. A center of tool moves to perpendicular position of start point of next move command at a start up block.



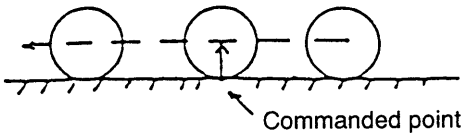
   3. Circular interpolation is not is not acceptable for a start up block. It should be executed either G00 or G01 mode.
   4. During tool radius compensation read three blocks in advance to find a stop position of movement. Therefore, block without move command such as M, S, T code or dwell etc. must not continue three blocks or more.

2. Tool radius compensation mode

During tool radius compensation mode, the tool moves so that the center of tool is located at the position perpendicular to the advance direction of the tool.

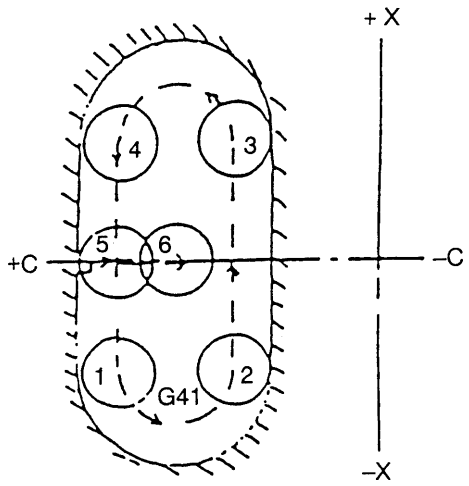


When tangent angle is 180°, the center of tool is located at the position perpendicular to the command point.



3. Tool radius compensation cancel

1. A block changed over from the status of tool radius compensation mode to the status of tool radius compensation cancel (G40) is named a cancel block.
2. When the compensation is completed, the tool moves so that the center of tool is located at the position perpendicular to the end point of the block immediately before the cancelled block.




```
G17
G145
G01 G41 C_F_
      :
      :
      :
G03 X_C_R_
G01 X_
G40 C_
```

3. In the cancelled block, the center of tool coincides with the command point.

(Note) *1. A plane designation should not change during tool radius compensation mode.*

2. In case of changing a direction of tool radius compensation during tool radius compensation, cancel a tool radius compensation once then execute a start up.

3. Inside compensation of smaller arc than tool radius can not machining because it generates over cut.

4. Execution of rotating radius command of arc is as follows;

(a) Command R_ when rotating angle is 0 ~ 180°.

*EX G02 X_C_R_*

(b) Put minus sign on a value of R if rotating angle is beyond 180° and less than 360°.

*EX G02 X_C_R_*

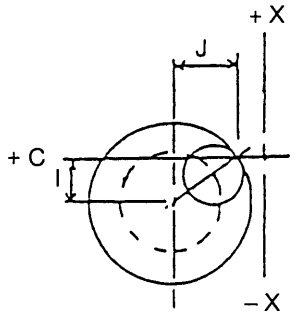
(c) Command by I, J or K instead of R if true circle cutting.

I : X component of a center position of rotation a view from start point of cutting.

K : C component of a center position of rotation a view from start point of cutting.

Put + or – sign either.

EX



G01 G41 X_C_F_

G03 I_J_F_

G01 G40 X_C_

5. During tool radius compensation mode, if command a block without movement three blocks or more, it generates insufficient or over cutting.

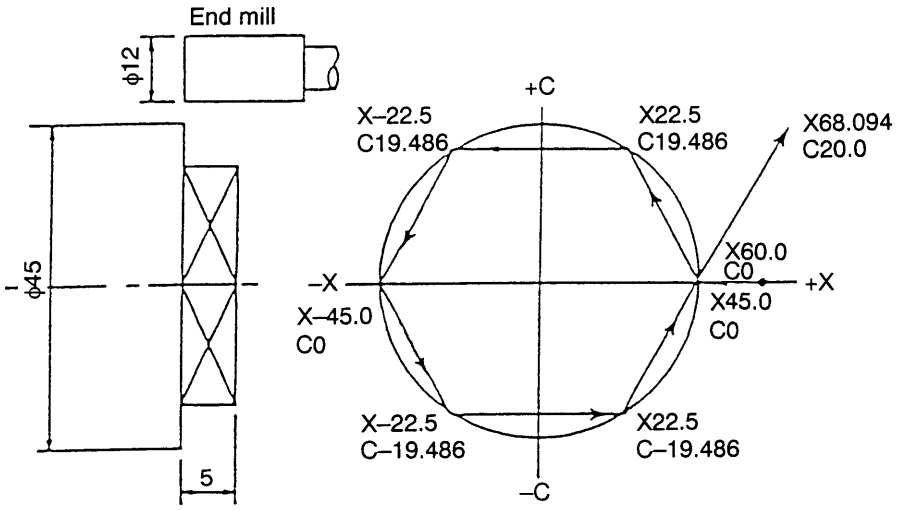
A block without move command should not continue three blocks or more.

6. Do not command the followings during tool radius compensation mode.

- *G31*
- *G37*
- *G53*
- *Fixed cycle by G code of 09 group*

6-6 Program Example (Polar Coordinate Interpolation, Tool Radius Compensation Function)

Example 1



N400

G28 U0

G28 W0

M43

G28 H0 ..... C-axis zero return

T0400

G17 G145 ..... X-Y plane designation, Tool radius compensation is effective.

G97 S800 M08

G00 X100.0 Z200.0 M13 ..... Rotating tool forward start

G01 G98 Z10.0 F2000

G01 Z–5.0 F1000

G121 ..... Polar coordinate interpolation ON

G01 X60.0 C0 F500

G42 X45.0 F80 ..... Tool radius compensation ON

X22.5 C19.486

X–22.5

X–45.0 C0

X–22.5 C–19.486

X22.5

X68.094 C20.0

G40 X100.0 F2000 ..... Tool radius compensation OFF

G120 ..... Polar coordinate interpolation OFF

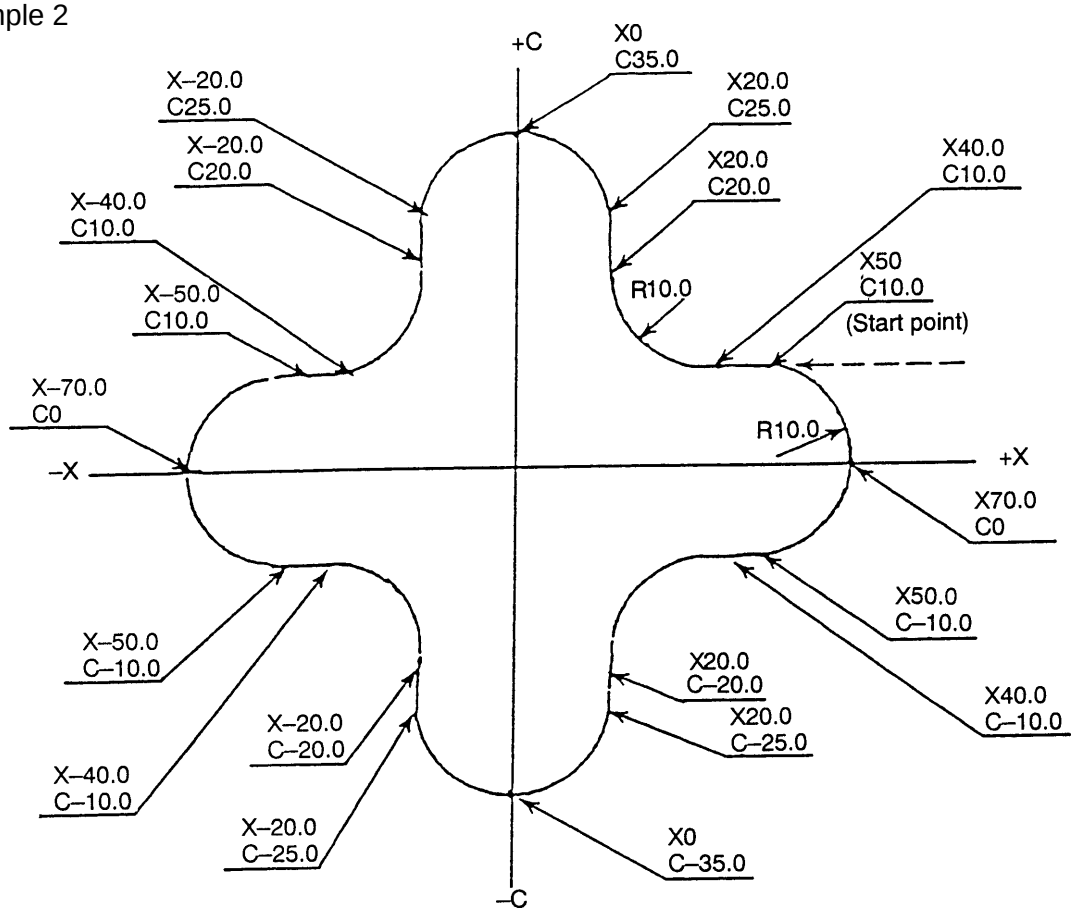
G99 M05

M45

G00 Z200.0

M01

Example 2




```
N1000
M43 ..... C-axis connection
G28 H0 ..... C-axis zero return
T1000
G17 G145
G97 S1000 M08
G00 X100.0 Z20.0 M13 ..... Rotating tool
G01 G98 Z1.0 F1000          forward start
G121
G01 C10.0
G42 X80.0 F300 ..... Tool radius
    X40.0 F100              compensation ON
G02 X20.0 C20.0 R10.0
G01 C25.0
G03 X–20.0 C25.0 R10.0
G01 C20.0
G02 X–40.0 C10.0 R10.0
G01 X–50.0
G03 X–50.0 C–10.0 R10.0
G01 X–40.0
G02 X–20.0 C–20.0 R10.0
G01 C–25.0
G03 X20.0 C–25.0 R10.0
G01 C–20.0
G02 X40.0 C10.0 R10.0
G01 X40.0
G03 X50.0 C10.0 R10.0
G01 X40.0
G00 Z20.0
G40 X100.0 C50.0 F2000 M09 ..... Tool radius
G120                                compensation
                                    cancel
    X200.0 M05
    M45
G99 Z200.0 M40
    M01
```

6-7 G824, G843 Direct Tapping

A direct tapping is performed with a spindle speed of rotating tool and feed rate of tapping axis synchronize perfectly, therefore, a floating tap holder is not required and a high accuracy tapping is available at high speed.

(1) Command form

[G842 / G843] [G198 / G199] [G98 / G99] X_C_Z_R_P_L_S_F(E)_ ;

G842	:	Forward direct tapping	
G843	:	Reverse direct tapping	
G198/G199	:	Return point	
X_/C_	:	Hole position (Note 2)	
Z_	:	Z point position (Note 2)	
R_	:	R point position	
P_	:	Dwell time	
L_	:	No. of repetition	
F_	:	G98 mode	Feed rate of tapping axis
		G99 mode	Pitch of tap
E_	:	No. of thread per inch (Effective at G99 mode only)	
S_	:	Spindle speed of rotating tool	

*(**Note** 1) If number of repetition (L) is omitted, it deems as L=1.*

If L=0 is commanded, it moves at hole position but tapping is not performed.

*(**Note** 2) Command a position of Z point by an axis address of tapping axis.*

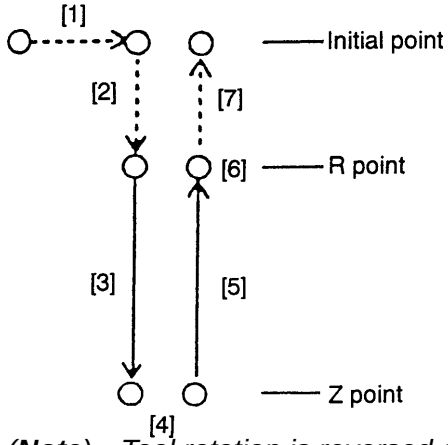
Command a hole position by an axis address other than tapping axis.

*(**Note** 3) R and Z points and P are mordal during a fixed cycle.*

(2) Machining cycle

A machining cycle of direct tapping consist motions from [1] to [7].

[G842]



••••• Rapid traverse, —— Cutting feed

[1] Positioning at hole position

[2] Rapid traverse to the R point

[3] Tapping to the Z point with forward rotation

[4] Dwell by the parameter setting

[5] Return to the R point with reverse rotation and stop the tool rotation

[6] Dwell by the parameter setting

[7] Rapid traverse to the initial point

*(**Note**) Tool rotation is reversed at the G843.*

(3) Designation of feed rate and pitch (F command)

At the direct tapping, the meaning of F command differs at the feed per minute mode (G98) and feed per revolution mode (G99).

Also, the E command is available instead of the F command at the G99 mode.

- G98 mode : The F shows a feed rate of tapping axis. (mm/min, inch/min)
- G99 mode : The F shows a pitch of tap. (mm, inch)
  The E shows a number of thread per inch. (thread/inch)

*(**Note** 1) A motion becomes a feed per minute even at the G99 mode.*

*(**Note** 2) The number of effective digits are same for the F and E.*

*(**Note** 3) If the F and E are commanded in the same block, the F becomes effective.*

*(**Note** 4) At the G98 mode, a pitch is determined by the F and S (Number of rotation of tool).*

$$\text{Pitch (min, inch)} = \frac{\text{F (mm/min, inch/min)}}{\text{S (min}^{-1}\text{)}}$$

(4) Magnification of returning speed

Returning speed of the direct tapping (Z point - R point) can be changed to the cutting feed rate (R point - Z point).

Set a magnification of returning speed to the cutting feed rate at the parameter. (unit 0.1).

$$\text{Returning speed} = \text{Cutting feed rate} \times \frac{\text{Parameter No.5211}}{100}$$

However, if zero is set at this parameter, a parameter value is deemed as 10 and the magnification becomes 1.

Normally, setting value is zero.

(5) Cancel of direct tapping

Command G80 or G code of 01 group (G00, G01, G02, G03 etc.).

However, do not command the other fixed cycle (G70, G71 etc.) in the same block of cancellation command.

(6) Precautions

(a) Command the direct tapping at the condition of cancellation of constant surface speed control (G97).

(b) Effective or ineffective can be selected for the dwell by the parameter setting.

Normally, it is set as effective.

(c) A feed rate override and spindle speed override are fixed at 100% while tapping.

However, effective or ineffective can be selected for the dry run by the parameter

setting.

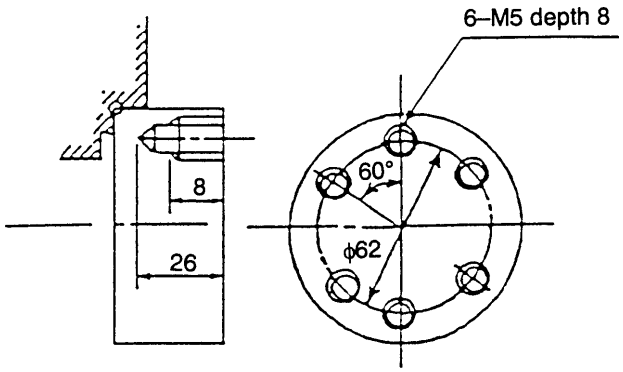
(d) When it performs at the single block, a tool stops at the initial point or R point.

(e) If the “Halt” button is pressed during the tapping, the halt lamp turns on immediately but the motion continues until the R point then stops.

(f) To cancel a direct tapping, command G80 or G codes of G01 group (G00, G01, G02 . . . .). However, do not command the other fixed cycle (G70, G71 etc.) in the same block of the cancellation command.

(g) When executing the direct tapping, command the spindle speed of the rotating tool at the immediately before or the same block of the G842 or G843 command.

(7) Program example



N800 (M5*P0.8)

T0800 M40

G17

M43

G28 H0

G50 C0

G97 S300 M08

G00 X62.0 Z20.0 M13

G99 G01 Z5.0 F5000

G842 G198 Z–8.0 H60.0 L6 F0.8

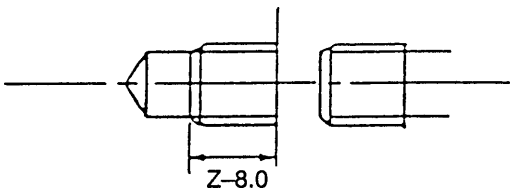
G80

G00 Z5.0 M05

M45

G00 X200.0 Z200.0

M01



6-8 G271 Cylindrical Interpolation

When commanding a traveling amount of linear axis and angle of rotary axis by a program command, a traveling amount of rotary axis commanded by an angle converts to a distance on the circumference internally. A distance on the circumference deems a traveling amount of linear axis on the circumference, therefore, the linear or circular interpolation with the other linear axis is available.

After interpolation, it convert reversly to the angle of rotary axis.

(1) Command form

G271 C Radius of cylinder ; Cylindrical interpolation ON

.

. Cylindrical interpolation mode

.

G271 G0 ; Cylindrical interpolation cancel

(C is a rotary axis.)

A rotating angle of the rotary axis is calculated reversely from the traveling amount on the circumference.

For example, if a traveling amount on the circumference at the cylinder with a radius = 50.0 is wanted to move by 100.0, find a rotating angle of the rotary axis by the following formula.

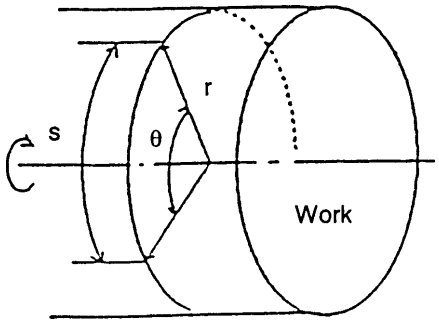
r : Radius of cylinder

θ : Rotating angle

s : Traveling amount on the circumference of cylinder

$$\text{Rotating angle} = \frac{360 \times s \text{ (Traveling amount on the circumference}}{2\pi \times r \text{ (Radius of cylinder)}}$$

$$= \frac{360 \times 100.0}{2\pi \times 50.0} = 114.591$$



(2) Feed rate

A feedrate during the cylindrical interpolation mode becomes a traveling speed of a tool on the outer diameter of the cylinder.

(3) Plane selection

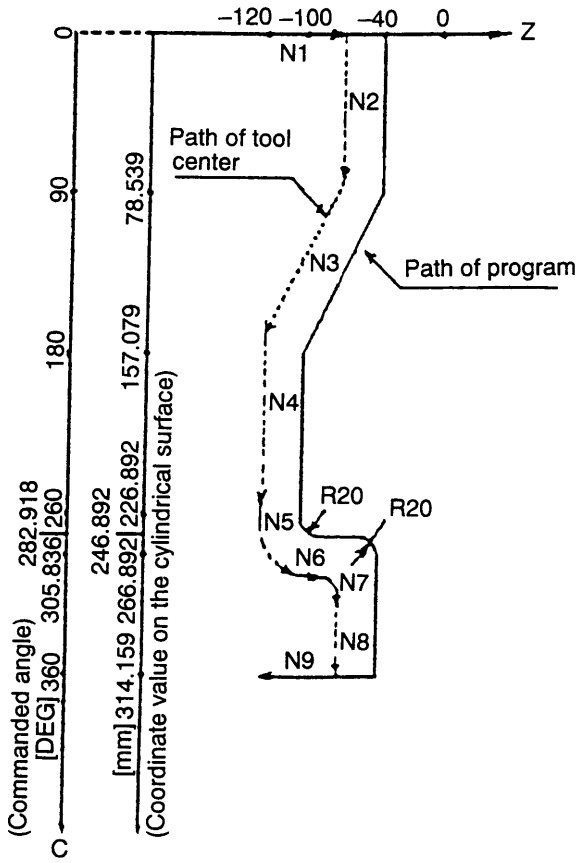
During the cylinder interpolation mode, the plane in which performs the cylindrical interpolation is determined as a horizontal axis is a linear axis (Z axis) and a vertical axis is a rotary axis (C axis). Therefore, since the plane for the cylindrical interpolation is determined by these parameters, the plane selection (G17 ~ G19) can not commanded during the cylindrical interpolation.

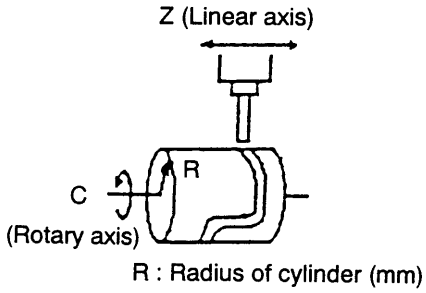
(4) Program example (X axis is a diametal designation)

(Select the C - Z plane by the parameter No. 3426 and 3427)

```
N400;
G28 U0;
G28 W0 M43;
G28 H0;
T0400;
G19 G98 M44;
G40 G80;
G50 C0;
G97 S600;
M145;
G00 X120.0 Z–120.0 C0 M13;
G271 C50.0;                          Cylindrical interpolation mode ON
                                       (Radius of cylinder = 50.0)
N1   G42 G01 Z–40.0 F500;
     G01 X100.0 F50;
N2   C90.0 F100;
N3   Z–100.0 C180.0;
N4   C260.0;                         Under cylindrical interpolation mode
N5   G03 Z–80.0 C282.918 R20.0;
N6   G01 Z–60.0;
N7   G02 W20.0 H22.918 R20.0;
N8   G01 C360.0;
     G00 X200.0;
N9   G40 G01 Z–120.0 F500;
     G271 C0;                        Cylindrical interpolation mode OFF
     G00 Z50.0 C0 M05;
     G143;
     M45;
     G18 G00 X200.0 Z200.0  M40;
     M01;
```

Unfolded drawing of cylindrical
surface with radius of cylinder is 50.0





(5) Precautions

(a) If a tool radius compensation is commanded, start up and cancel should be done during the cylindrical interpolation mode.

(b) The G271 command (G271 Cxx;) should be commanded in the block individually.

Also, if an axis command is missing after the G271 (G271; for example), It becomes an alarm.

(c) If an axis other than the axis which is set by the parameter No.7817 is commanded by the G271 command, it becomes an alarm.

(d) The following functions are not available.

Block restart

Return to the interrupted point of machining

Manual intervention by manual absolute ON

(e) If the following command is issued during the cylindrical interpolation mode, it becomes an alarm.

G17, G18, G19	Plane designation
G28, G30, G53	Machine coordinate system
Thhtt ; (T command), G54, G50	Work coordinate system
G70 ~ G76, C81 ~ G89, G831 ~ C861	
G90 ~ G94	Various fixed cycle
G31, G121, G232	Others

G00 (Restricted only when the rotary axis which performs the cylindrical interpolation has been commanded.)

(f) At the cylindrical interpolation mode, convert an angle of rotary axis to the distance on the circumference then convert reversely after interpolation.

At this time a conversion error generates slightly.

(g) By the above conversion error, if the circular interpolation for small radius is executed during the cylindrical interpolation mode, the circular interpolation alarm may occur, therefore, an attention is required when it applies. Also, a tool radius compensation alarm may occur at the tool radius compensation by the above reason.

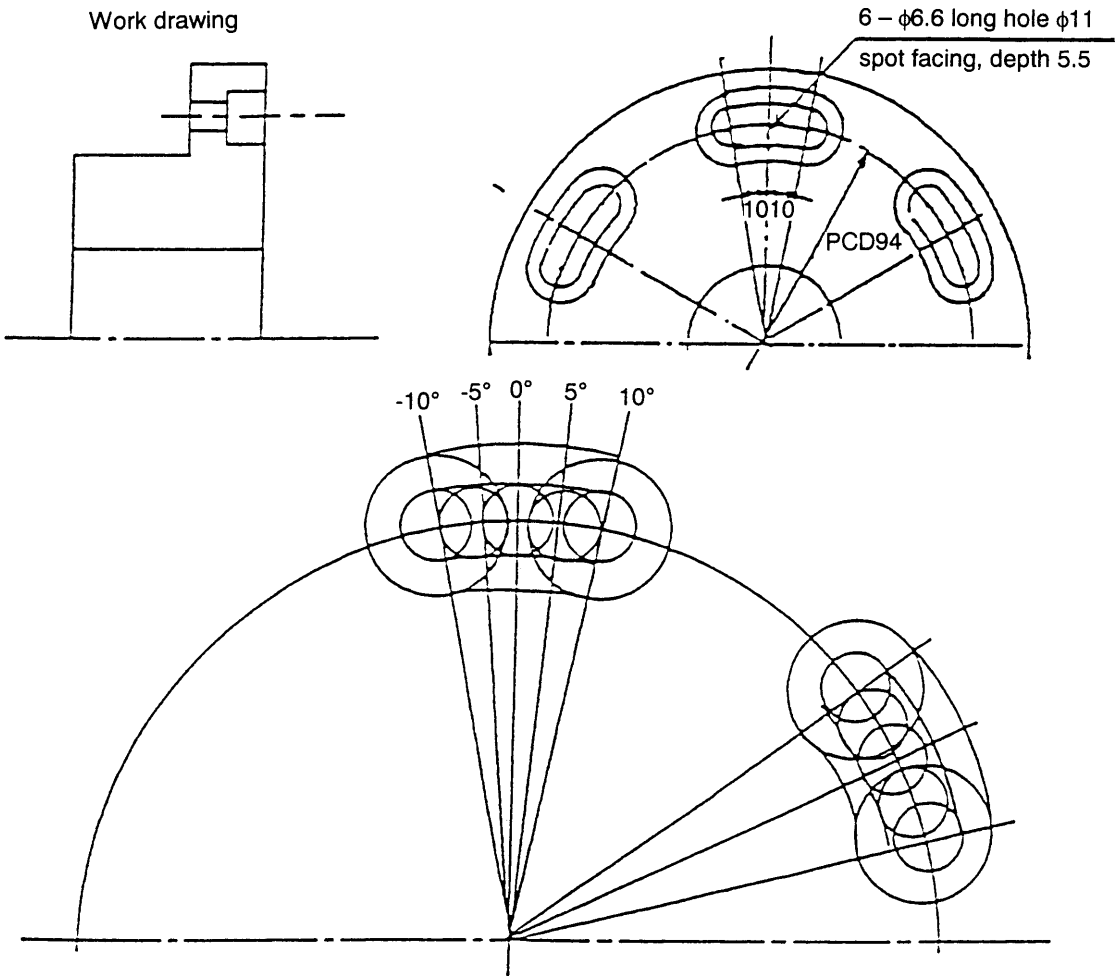
(h) If the cylindrical interpolation mode ON, G271 Cxx; ($C \neq 0$), is commanded again during the cylindrical interpolation mode, it becomes an alarm.

(i) A remaining traveling amount is shown a value which is traveling the outer diameter of the cylinder.

7. REFERENCE
(SPECIFICATIONS OF C-AXIS CONTROL)

7-1 How to Calculate C-axis Feed Rate for Long Hole Machining

Work drawing



1) C-axis feed rate (mm/min); No decimal point allowed

Arc length per 1°

$$\frac{D \times \pi}{360} = \frac{94 \times 3.14}{360} = 0.82\text{mm/deg}$$

D : Cutting diameter

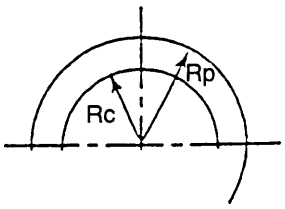
Feed rate per minute :

Feed rate ÷ Arc length per 1°

= F50mm/min ÷ 0.82 = 60.975deg/min

Where ; feed rate in normal cutting is taken as 50mm/min. Feed rate of C axis becomes a command of 61mm/min.

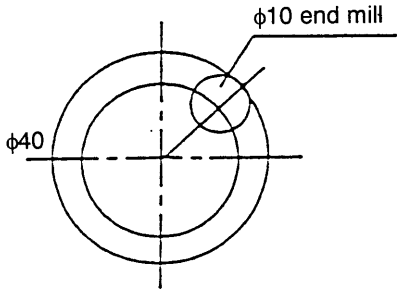
2) Feed rate inside/outside the arc



Rp radius of program path

Rc radius of center path of the cutter $F = F \times \frac{Rc}{Rp}$

Example

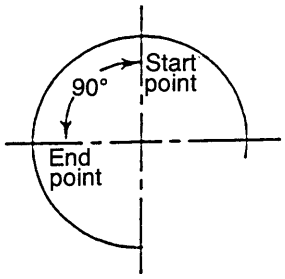


If the program path is F100

$F = 100 \times \frac{15}{20} = 75\text{mm/min}$

3) Feed rate of the rotating axis

Example Specify in deg/min.



Move at 300 deg/min for 90°.

Time $\frac{300}{90} = 0.3$ min.

4) Feed rate at linear interpolation including linear axis and rotating axis.

Tangent rate in Cartesian coordinate of the rotating axis (deg) and the linear axis (mm) is F.

Example G00 W-20.0 C40.0 F300

Supposing C-axis 40 deg is 40 mm, distribution time will be

$$\frac{\sqrt{20^2 + 40^2}}{300} = 0.15 \text{ min}$$

C-axis rate is

$$\frac{40 \text{ deg}}{0.15 \text{ min}} = 267 \text{ deg/min}$$

7-2 How to Calculate the Number of Rotation and Feed Rate of the Rotating Tool

1) The number of rotation of the rotating tool

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

N = Rotation per minute (min^{-1})
D = Diameter of the cutter (mm)
V = Cutting rate

Example) Rotation per minute when machining with D10.0 drill, V20

$$N = \frac{1000 \times 20}{3.14 \times 10} = 636.9 \qquad S = 637\ min^{-1}$$

2) Feed rate per minute

For end mill		F = Feed rate (mm/min)
or drill	$F = N \times f$	N = Rotation per minute (min^{-1}) f = 1 rotation feed (mm/rev)
For tap	$F = N \times P$	P = Pitch (mm)

Example) Feed rate with D10.0 drill, V20 and 1 rotation feed of 0.2.

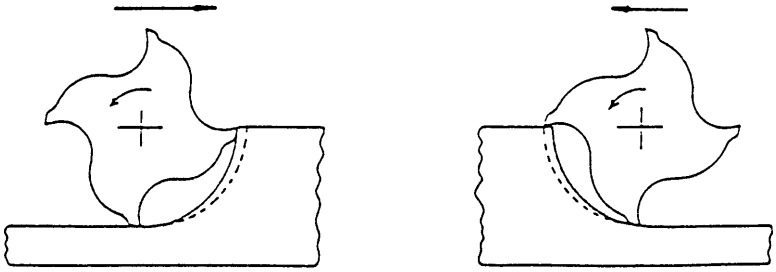
$F = 637 \times 0.2 = 127.4$ F = 127

Example) Feed rate when machining with M8 tap and the spindle rpm of 320.

$F = 320 \times 1.25 = 400$ F = 400

3) Up-cutting and Down-cutting

Up-cutting Down-cutting



	Up-cutting	Down-cutting
Merits	• Undercut is easily caused. • Finish surface roughness is good in wet cutting. • A finish surface is glossy and looks fine because it is rubbed by a tool nose. • More advantageous than down-cutting in scale or sand caught machining.	• A tool nose flank is worn out less and a tool life is longer. • A cutting resistance is low. • Finish surface roughness is superior in dry cutting.
Demerits	• Since a tool nose tends to slip, a flank is worn out more and a tool life is shorter. • It is necessary to firmly mount a workpiece. • A cutting resistance is high. • Finish surface roughness is inferior in dry cutting. • Burr tends to be caused at the end of a workpiece. • A hardened layer is caused for a hardenable material.	• Coolant has a less effect on finish surface roughness and may worsens it to the contrary. • A tool nose is likely to be damaged in scale or sand caught machining.

CNC LATHE
INSTRUCTION MANUAL
PROGRAMMING
SEIKI-SEICOS Σ10L/21L
Version 1.01

11-1999 First Edition
02-2000 Revision