

CNC SYSTEM

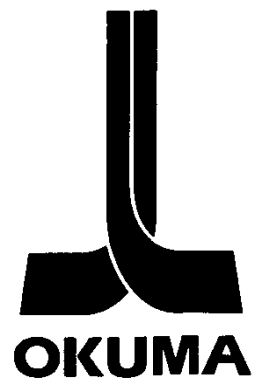
OSP-P300S/P300M/P300S-e/P300M-e

OSP-P300SA/P300MA/P300SA-e/P300MA-e

U-AXIS FUNCTION

INSTRUCTION MANUAL (2nd Edition)

Pub No. 6233-E-R1 (ME61-341-R2) Apr. 2017



INTRODUCTION

This book consists of several manuals and, therefore, it may contain specifications not selected for user's machine.

Please confirm your specification by definite specifications or equivalent document.

TABLE OF CONTENTS

SECTION 1	THREAD-CUTTING FUNCTION	1
1.	Outline	1
2.	Thread-Cutting Plane Designation.....	2
3.	G178 Thread-cutting Fixed Cycle	3
4.	G179 Thread-Cutting Fixed Cycle	4
5.	G33 Thread-Cutting (Non-Fixed Cycle)	5
6.	Thread-Cutting Compound Fixed Cycle (TRDL).....	6
7.	Thread-Cutting Compound Fixed Cycle (TRDT)	7
8.	Designation of Compound Fixed Cycle Cut Pattern	9
9.	Thread-Cutting Cycle 'Chamfering' Designation.....	11
10.	Thread-Cutting Cycle Operation	12
11.	Thread-Cutting 'Temporary Stop'	13
12.	Parameters	14
13.	Thread-Cutting Precautions	15
14.	List of Added Alarms.....	16
15.	List of Added Alarms.....	18
SECTION 2	TOOL GAUGING SYSTEM OF U-CENTER	19
1.	Outline	19
2.	Details of Specifications.....	19
2-1.	Tool Gauging Cycle	19
2-2.	U-axis Zero Offset Cycle.....	19
2-3.	Tool Breakage Detection Cycle	19
3.	Operation Flow	20
4.	General Flow of Tool Gauging Cycle	21
5.	Details of Individual Cycles	22
5-1.	Touch Sensor Zero Setting Cycle.....	22
5-2.	Tool Gauging Cycle	25
5-3.	Zero Offset Cycle.....	28
5-4.	Tool Breakage Detection Cycle	31
6.	Display of Measured Data on Crt.....	32
6-1.	Tool Gauging Cycle Example	32
6-2.	Tool Breakage Detection Cycle Example	32
6-3.	Others	32
7.	Precautions	33
8.	Program Examples	34
9.	Program List	35
10.	Alarm List.....	36

SECTION 3 U-AXIS DIAMETER INSTRUCTION	37
1. Outline	37
2. Function	37
SECTION 4 U-AXIS TORQUE LIMIT FUNCTION	39
1. Introduction	39
2. Caution	39
3. Parameters	39
SECTION 5 SECOND TOOL COMPENSATION FUNCTION	40
1. Outline	40
2. Specifications.....	41
3. Display	42
SECTION 6 NOSE RADIUS COMPENSATION	44
1. General Description of Nose Radius	44
2. Nose Radius Compensation and Tool Radius Compensation.....	45
3. Nose Radius Compensation Value and Position Number (P)	46
4. Setting Nose Radius Compensation Value and Position Number	47
5. Nose Radius Compensation Program	48
6. Display	49
7. Operation at Nose Radius Compensation ON.....	50
8. Tool Movement in Nose Radius Compensation Mode	54
9. Tool Movement when Nose Radius Compensation is Canceled.....	58
10. Changing Compensation Direction in Nose Radius Compensation Mode.....	61
11. Precautions on Nose Radius Compensation	64

SECTION 1 THREAD-CUTTING FUNCTION

1. Outline

In order to execute cutting of the designated lead thread, the spindle begins its cutting feed from its constant position in the usual manner. The cutting speed is controlled by a periodic pulse-count reading from the pulse generator attached to the spindle, and by the designated feed amount per 1 spindle revolution.

Thread-cutting can be executed using either the G01 'non-fixed cycle' method in which the tool nose path is designated separately at each block, or by the 'fixed cycle' and 'compound fixed cycle' methods in which operation occurs according to a pre-determined pattern.

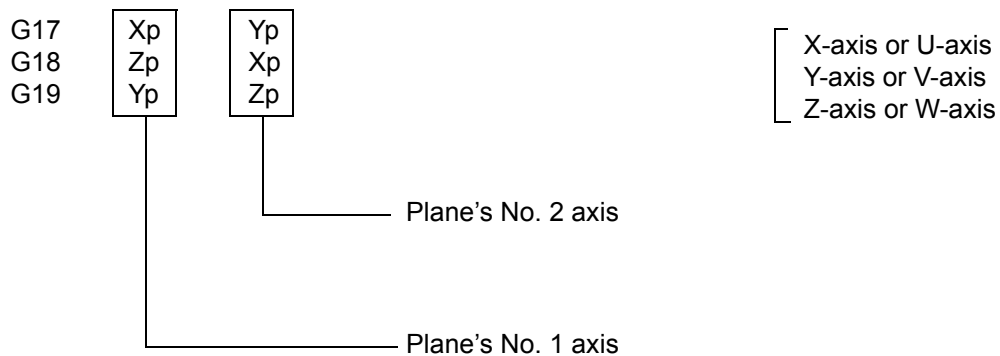
The thread-cutting function table is shown on the following page.

Table 1-1 Thread Cutting Function Table

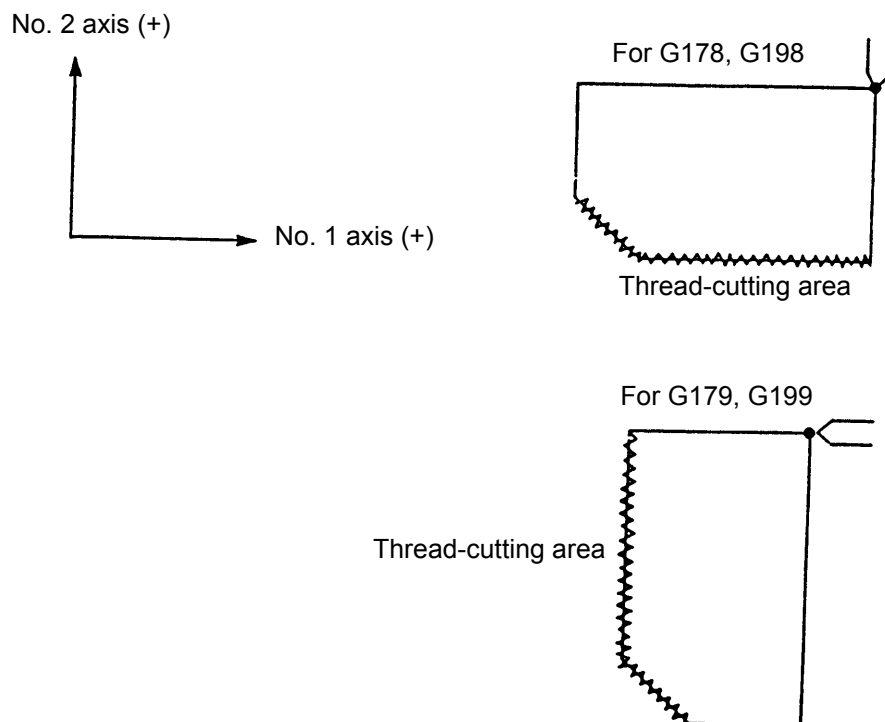
G Code	Cycle	Description	Temporary Stop Valid/ Invalid	Remarks
G178	Fixed cycle	Thread-cutting cycle for the No. 1 axis direction of the designated plane (2 axes).	Temporary stop is valid (execution of temporary stop cycle)	
G179	Fixed cycle	Thread-cutting cycle for the No. 2 axis direction of the designated plane (2 axes).	Temporary stop is valid (execution of temporary stop cycle)	
G33	Non-fixed cycle	Thread-cutting is executed at the designated pitch, from the present position to the designated position. (Pitch direction is specified by M-code.) Increasing pitch is possible when 'E' symbol is '+'.	Temporary stop is invalid	
G33	Non-fixed cycle	Thread-cutting is executed at the designated pitch, from the present position to the designated position. (Pitch direction is specified by M-code.) Decreasing pitch is possible when 'E' symbol is '-'.	Temporary stop is invalid	
TRDL	Compound fixed cycle	Thread-cutting 'compound fixed cycle' for the No. 1 axis direction of the designated plane (2 axes).	Temporary stop valid (execution of temporary stop cycle)	Applies only to 'M22' cutting mode (single-edge cutting)
TRDT	Compound fixed cycle	Thread-cutting 'compound fixed cycle' for the No. 2 axis direction of the designated plane (2 axes).	Temporary stop valid (execution of temporary stop cycle)	Applies only to 'M32' cutting mode (single-edge cutting)

2. Thread-Cutting Plane Designation

In thread-cutting 'fixed cycle' operations, the plane is designated by G-codes (G17, G18, G19).



Thread-cutting for the No. 1 axis direction of the plane designation by the above program is executed by the G178 and G198 cycles. Thread-cutting for the No. 2 axis direction of this plane is executed by the G179 and G199 cycles. The thread pitch for each axis direction is calculated and executed as specified.



3. G178 Thread-cutting Fixed Cycle

This is the thread-cutting cycle for the No. 1 axis direction of the plane.

(1) Command method

G178 Xp (Yp, Zp) Yp (Zp, Xp) $\left\{ \begin{array}{c} J(K, I) \\ R \end{array} \right\} I(J, K) L P E F$

Xp (Yp, Zp) : Coordinate value for thread-cutting end point.

Yp (Zp, Xp) : Thread cut dimension for each thread-cutting.

E : Pitch change amount per 1 pitch during variable pitch operation.

F : Thread pitch command ('F/P' pitch is adopted when 'P' command is used).

J (K, I) : Discrepancy between start and end points for tapered thread operation (incremental amount).

R : Taper angle (either 'J (K, I)' or 'R' designated) from No. 1 axis Xp (Yp, Zp) of plane.

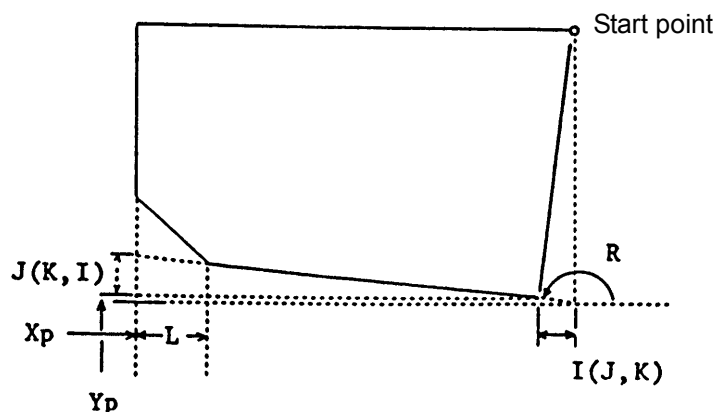
I (J, K) : Shift amount for plane's No. 1 axis Xp (Yp, Zp) when thread-cutting is started.

L : Thread-cutting distance (if not designated, distance will be 1 pitch when thread-cutting is started).

P : Number of thread cutting ridges within the distance designated by 'F' (if 'P' is not designated, 'P = 1' will be adopted).

Note

Increasing/decreasing pitch thread is possible by using the 'E' plus/minus symbols.



ME6101002

4. G179 Thread-Cutting Fixed Cycle

This is the thread-cutting cycle for the No. 2 axis direction of the plane.

(1) Command method

G178 Xp (Yp, Zp) Yp (Zp, Xp) $\left\{ \begin{array}{c} J(K, I) \\ R \end{array} \right\} I(J, K) L P E F$

Xp (Yp, Zp) : Thread cut dimension for each thread-cutting.

Yp (Zp, Xp) : Coordinate value of thread-cutting end point.

E : Pitch change amount per 1 pitch during variable pitch operation.

F : Thread pitch command ('F/P' pitch is adopted when 'P' command is used).

I (J, K) : Discrepancy between start and end points for tapered thread operation (incremental amount).

R : Taper angle (either 'I (J, K)' or 'R' designated) from the No. 1 axis Xp (Yp, Zp) of the plane.

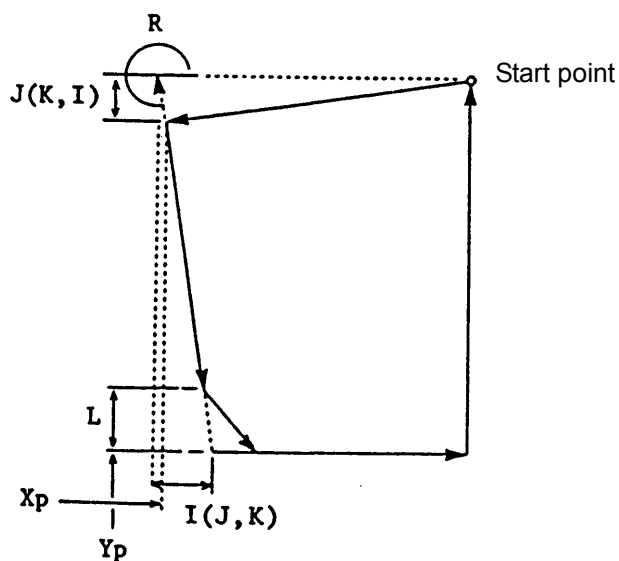
J (K, I) : Shift amount for the plane's No. 2 axis Yp (Zp, Xp) when thread-cutting is started.

L : Thread-cutting distance (If not designated, distance will be 1 pitch when thread-cutting is started).

P : Number of thread ridges within the distance designated by 'F' (If 'P' is not designated, 'P = 1' will be adopted).

Note

Increasing/decreasing pitch thread is possible by using the 'E' plus/minus symbols.



ME6101003

5. G33 Thread-Cutting (Non-Fixed Cycle)

(1) Command method

G33 Pp F E P

Pp : Absolute command (for G90, the thread's end point position at the coordinate system of the selected workpiece is designated).

For G91 (incremental), the thread length is designated.

F Thread pitch command

Specify the axis to which this thread pitch is to be applied at the parameter "Axis travel of pitch specified with F command." (For the procedure for setting this parameter, see "12. Parameters".) In this manual, all axes to which this thread pitch applies are called "pitch-applied axes."

Thread pitch of "pitch-applied axes"

[F command value]

Thread pitch of axes other than "pitch-applied axes"

$[F \text{ command value}] \times [\text{Travel of the axis}] / [\text{travel of "pitch-applied axis"}]$

E : Pitch change amount per 1 pitch of variable pitch thread operation.

Increasing/decreasing pitch thread is possible by using the 'E' plus/minus symbols.

P : Number of thread ridges within the distance designated by 'F' (If 'P' is not designated, 'P = 1' will be adopted).

6. Thread-Cutting Compound Fixed Cycle (TRDL)

This is the thread-cutting compound cycle for the No. 1 axis direction of the plane.

(1) Common method

TRDL (G198) $X_p (Y_p, Z_p) Y_p (Z_p, X_p) \left\{ \begin{matrix} J (K, I) \\ R \end{matrix} \right\} P E F H Q QA = QB = M$

$X_p (Y_p, Z_p)$: Coordinate value of thread-cutting end point.

$Y_p (Z_p, X_p)$: Thread-cutting 'last cut' dimension.

E : Pitch change amount per 1 pitch during variable pitch operation.

F : Thread pitch command ('F/P' pitch is adopted if 'P' command is used).

J (K, I) : Discrepancy between start and end points for tapered thread operation (incremental amount).

R : Taper angle (either 'J (K, I)' or 'R' is designated) from No. 1 axis $X_p (Y_p, Z_p)$ of the plane.

L : Thread-cutting distance (if not designated, distance will be 1 pitch when thread-cutting is started).

P : Number of thread ridges within the distance designated by 'F' (If 'P' is not designated, 'P = 1' will be adopted).

H : Tool nose angle ('0°' if designation is not as follows: $0^\circ \leq H < 180^\circ$)

Q : 1st cut amount

QA= : Finishing allowance (if not designated, 'QA = 0' is adopted, and a finishing cycle is not executed).

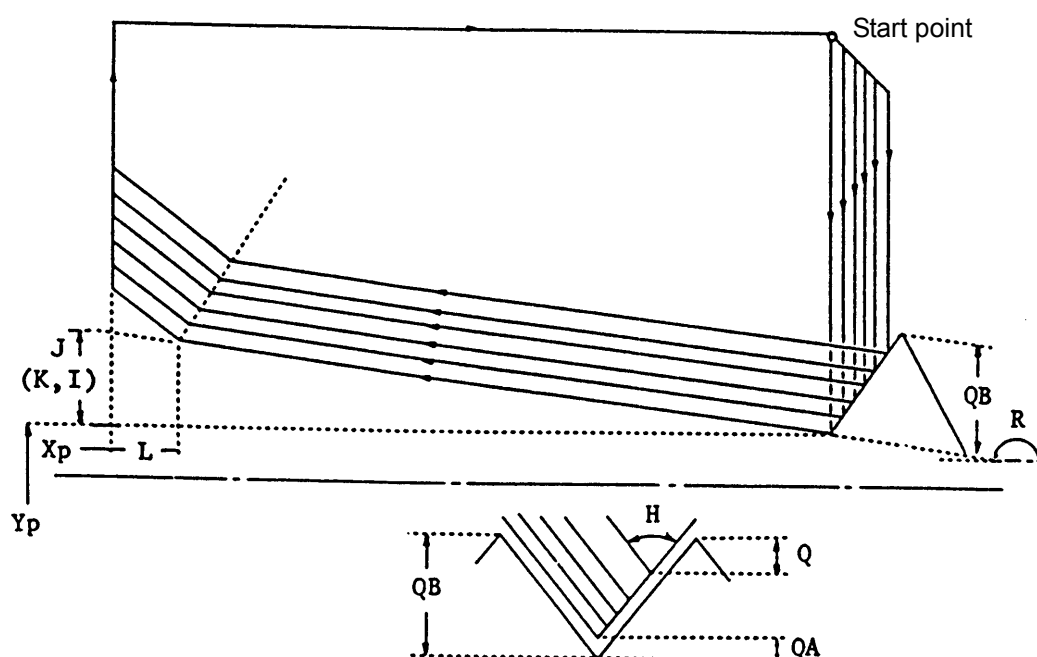
QB= : Discrepancy between thread's external pressure and root diameter.

M : Cutting pattern designation.

Note

Increasing/decreasing pitch thread is possible by using the 'E' plus/minus symbols.

(2) TRDL cycle diagram



7. Thread-Cutting Compound Fixed Cycle (TRDT)

This thread-cutting compound fixed cycle is used for the No. 2 axis direction of the plane.

(1) Command method

TRDT (G199) Xp (Yp, Zp) Yp (Zp, Xp) $\left\{ \begin{matrix} J (K, I) \\ R \end{matrix} \right\} L P E F H Q QA = QB = M$

Xp (Yp, Zp) : Thread-cutting 'last cut' dimension.

Yp (Zp, Xp) : Coordinate value of thread-cutting end point.

E : Pitch change amount per 1 pitch during variable pitch operation.

F : Thread pitch command ('F/P' pitch is adopted when 'P' command is used).

J (K, I) : Discrepancy between start and end points for tapered thread operation (incremental amount).

R : Taper angle (either 'I (J, K)' or 'R' is designated) from No. 1 axis Xp (Yp, Zp) of the plane.

L : Thread-cutting distance (If not designated, the distance will be 1 pitch when thread-cutting is started).

P : Number of thread ridges within the distance designated by 'F'.

H : Tool nose angle ('0°' if designation is not as follows: $0^\circ \leq H < 180^\circ$)

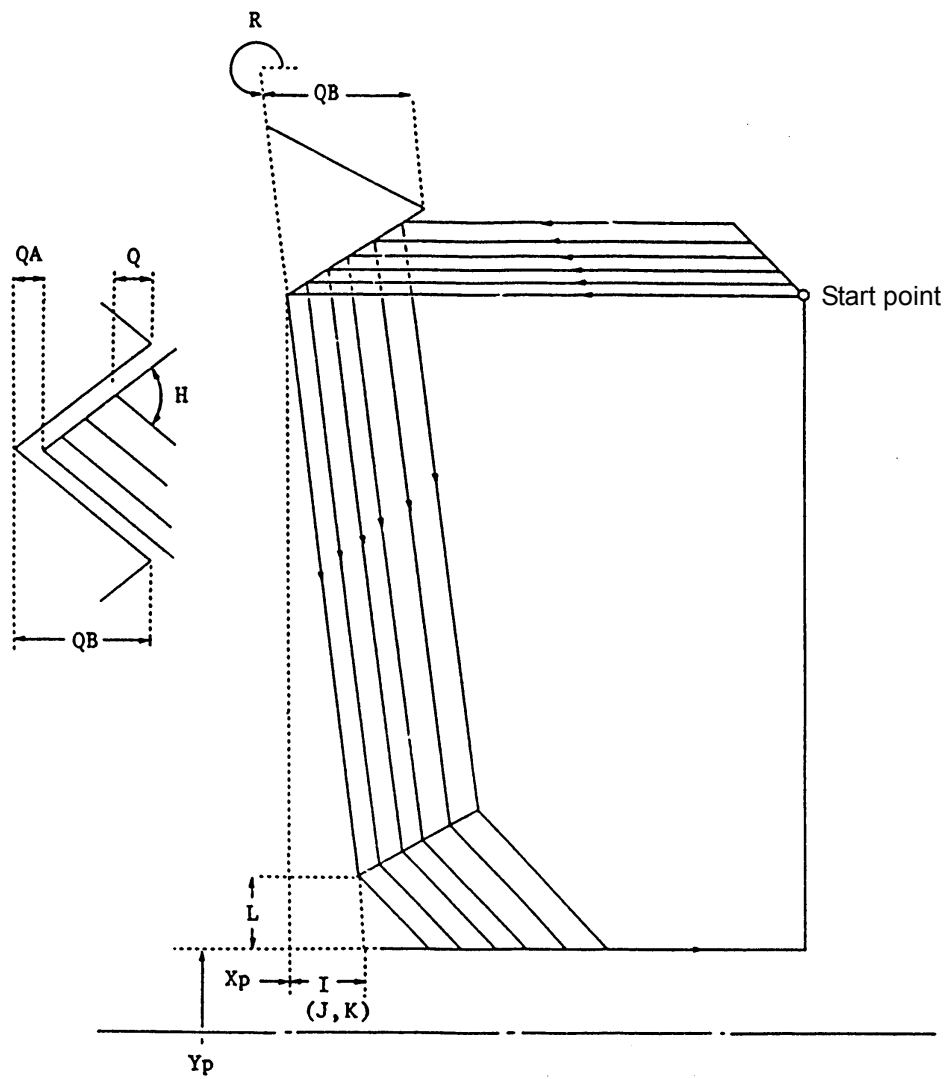
Q : 1st cut amount

QA= : Finishing allowance (If not designated, 'QA = 0' is adopted, and a finishing cycle is not executed).

QB= : Thread ridge height.

M : Cutting pattern designation.

(2) TRDT cycle diagram



ME6101005

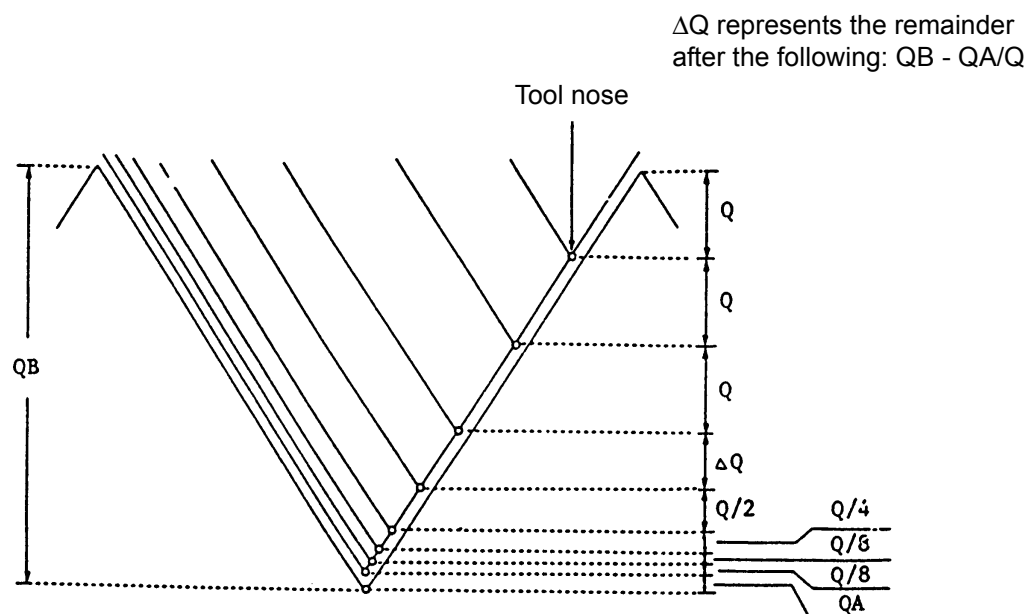
8. Designation of Compound Fixed Cycle Cut Pattern

Three types of compound fixed cycle cut patterns can be selected using M-codes. If the M-code is not designated, the system will adopt M294 as the code.

(1) Cut pattern 1 M294

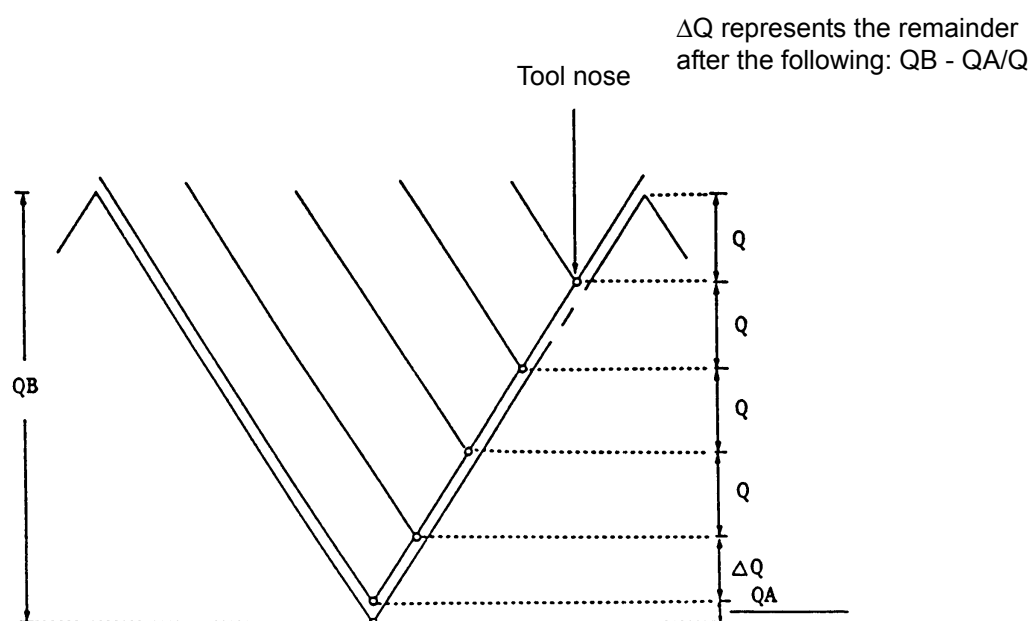
The 'Q' portions in the figure below represent the cut amount per cut. Beginning from one 'Q' portion prior to the 'QB-QA' point, the cutting amount changes as follows, and thread-cutting is executed: $Q/2 \rightarrow Q/4 \rightarrow Q/8 \rightarrow Q/8$

The QA portion represents the amount which is cut at the final finishing cycle. If QA is not designated, the finishing cycle will not occur.



(2) Cut pattern M295

Cutting is executed for the 'Q' portions shown below, up to the 'QA-QB' point. The QA portion represents the amount which is cut at the final finishing cycle. If QA is not designated, the finishing cycle will not occur.



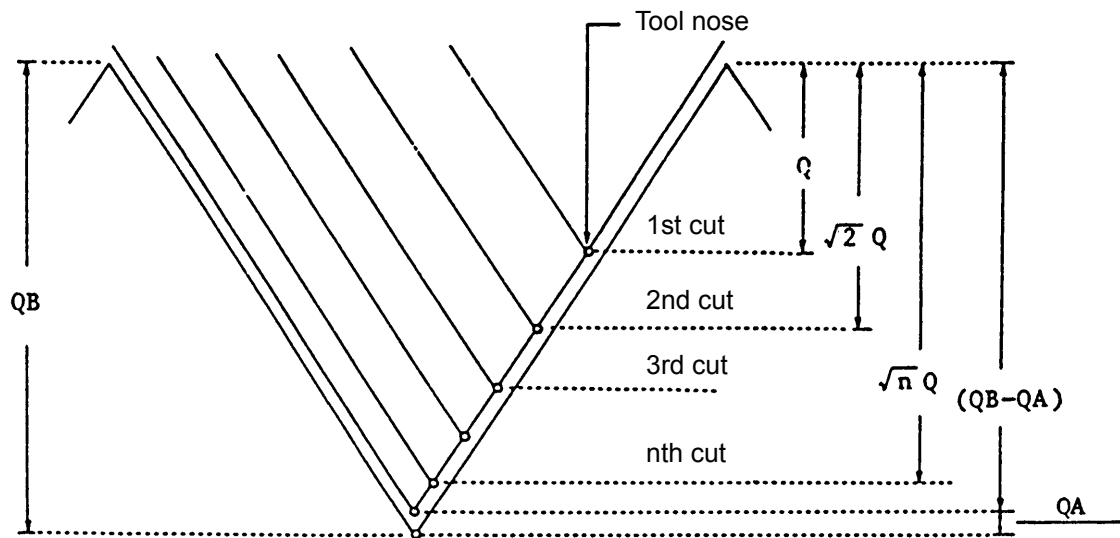
SECTION 1 THREAD-CUTTING FUNCTION

(3) Cut pattern 3 M296

Thread cutting is executed up to the 'QB-QA' point in the following manner:

1st cut amount is Q , 2nd cut amount is $\sqrt{2} Q$, nth cut amount is $\sqrt{n} Q$.

If the $(\sqrt{n} - \sqrt{n-1}) D$ becomes smaller than 'QA' during this operation, the cut amount per cut from that point will be 'QA'. 'QA' will also be the cut amount for the final finishing cycle. If 'QA' is not designated at this time, the finishing cycle will not occur.

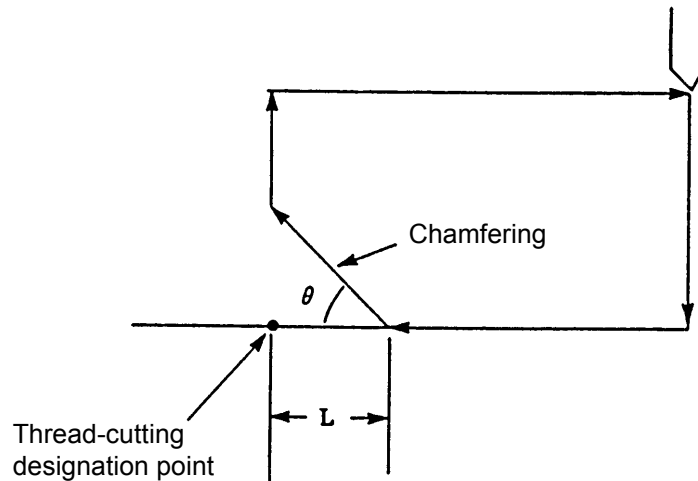


ME6101008

9. Thread-Cutting Cycle 'Chamfering' Designation

If chamfering is desired at the thread-cutting cycle, the M-codes shown below may be used.

M292.....Chambering OFF
 M293.....Chambering ON
 Chambering = Thread-cutting



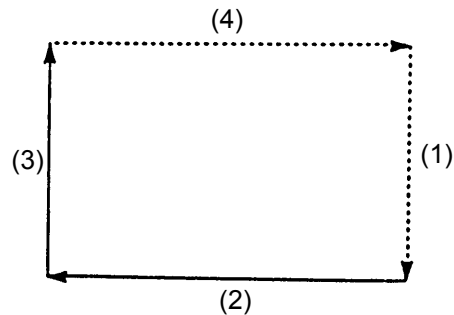
ME6101009

Note

- If 'L' is not designated, 1 pitch (F/P) is adopted.
- The run-off speed for chamfering is designated by the parameter setting. Therefore, the chamfering angle θ is determined by the thread-cutting feed rate, and by the run-off speed designated by the above parameter setting.

Optional parameter long word No. 58 Thread-cutting cycle's cutting feed unit quantity ($\mu\text{m}/8\text{ ms}$) Minimum: 10 to Maximum: 512

10. Thread-Cutting Cycle Operation



ME6101010

- (1) Rapid traverse positioning is executed to the thread-cutting start point.
- (2) Feed to the thread-cutting end point is executed by the 'F' command. If tapered cutting is being executed at this time, the thread-cutting direction axis operation will be according to the 'F' pitch command.
- (3) Operation speed is according to the parameter setting.
- (4) Rapid traverse positioning is executed back to the thread-cutting cycle's initial position.

Feed rate override during thread-cutting

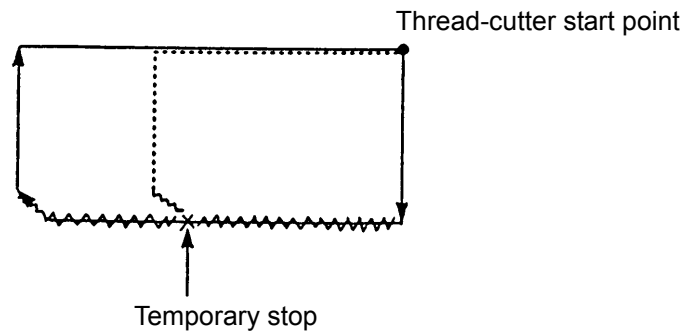
Feed rate override is invalid during thread-cutting, and system operation is 100%.

Spindle rpm override during thread-cutting

Spindle rpm override is invalid during thread-cutting, and system operation is 100%.

11. Thread-Cutting 'Temporary Stop'

- (1) For non-fixed cycle (G33):
The temporary stop function is invalid during G33 thread-cutting. If the TEMPORARY STOP button is pressed during the G33 cycle, the G33 cycle will be cancelled, and a single-block stop will be executed at the end of that block. Press the START button to reset the system.
- (2) For fixed cycle and compound fixed cycles:
If the TEMPORARY STOP button is pressed during a thread-cutting fixed cycle, the temporary stop cycle described below will be executed, and the thread-cutting cycle will return to its start point, at which time operation will be stopped. The system can be reset by pressing the START button.



ME6101011

Note

Temporary stop cycle:

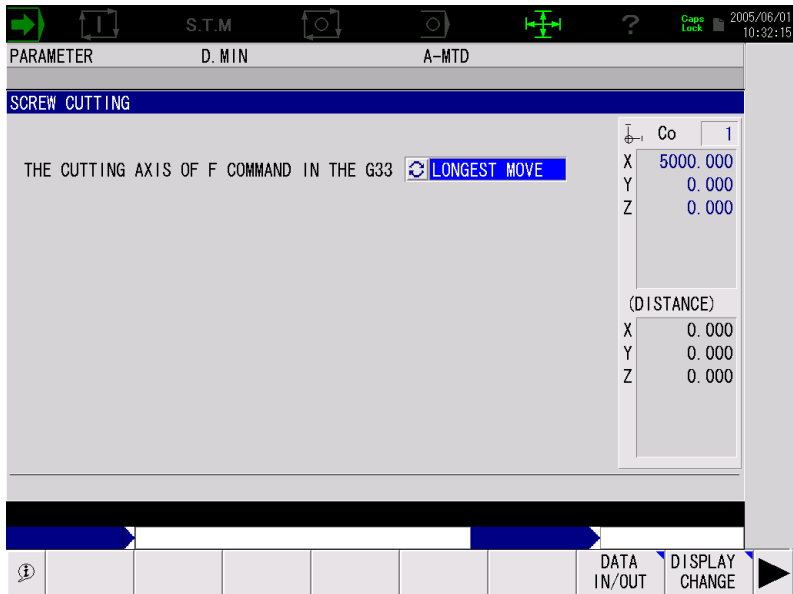
When the TEMPORARY STOP button is pressed, the thread-cutting process is interrupted by a 45° chamfering operation which is started.

After this, the cutting axis returns to the thread-cutting start point. Finally, the thread pitch feed axis returns to the thread-cutting start point.

- (3) If the TEMPORARY STOP button is pressed during dry run operation (for non-fixed cycle, fixed cycle, or compound fixed cycle), feed will be stopped at that point. The system can be reset by pressing the START button.

12. Parameters

Set the thread cutting parameters on the thread cutting function screen by selecting the parameter setting mode.



ME6101012

THE CUTTING AXIS OF F COMMAND IN THE G33 (pitch-applied axis)

Specify the axis to which the thread pitch F is applied (pitch-applied axis) in G33 mode.

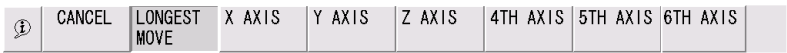
INITIAL VALUE	LONGEST MOVE
---------------	--------------

When the LONGEST MOVE is selected, the axis with the longest travel distance in the block where G33 is specified is regarded as the pitch-applied axis.

This parameter can be changed in the following procedure:

Procedure : _____

- 1 Select [F1] (MENU) from the function menu.
- 2 Select the function key of the axis that you want to set as a pitch-applied axis from the pop-up function menu.



ME6101013

Note

It is possible to select 4th, 5th, or even 6th axis regardless of the machine specifications. However, if the selected axis is not found during axis movement in G33 mode, the following alarm occurs in program execution:

Alarm B 2253 Data word: axis command

13. Thread-Cutting Precautions

- (1) If thread-cutting feed is unconditional, a feed command will be executed at each revolution of the spindle. Therefore, be sure to designate the 'F' command when changing from an individual portion feed mode to thread-cutting.
- (2) When the 'F' feed command is designated, the actual feed (mm/min value) of the axis will be limited by the feed rate clamp. If the designated feed rate exceeds the feed rate clamp value, the designated feed rate will be replaced by the feed rate clamp value, and operation will be executed accordingly. When this occurs, a warning message will be displayed.
- (3) A rotational axis command cannot be executed while in the thread-cutting mode (alarm will be activated).
(Alarm B 2359 Data word: rotating axis command)
- (4) An 'F1' shift is prohibited while in the thread-cutting mode (alarm will be activated).
(Alarm B 2244 Data word: 'F')
- (5) During the thread-cutting operation, 'machine lock' and 'STM lock' operation will according to the virtual RPM setting of the NC optional parameter word No. 9.
- (6) G33, G178, and G179 belong to the same group as G00, G01, G02, G03, G60. The G00, G01, G03, G60, etc., commands are required when the thread-cutting mode is cancelled.
- (7) An alarm will be activated if G178, G179, TRDL, TRDT are executed during coordinate conversion operations (parallel/rotational coordinate shift, copy, figure enlargement/reduction).
(Alarm B 2263 Data word: G code)
- (8) The 'drilling' fixed cycle (G73 - G76, G81 - G89) mode, and the 'thread-cutting' fixed cycle (G178, G179, TRDL, TRDT) mode must not be designated simultaneously. If so designated, operation cannot be guaranteed.
- (9) Do not execute a G22 (program full stroke limit) change during the thread-cutting fixed cycle (G178, G179, TRDL, TRDT). If executed, operation cannot be guaranteed.
- (10) In the thread-cutting fixed cycle (G178, G179, TRDL, TRDT) mode, high-speed drawing or 3D real simulation such as return search is not possible. In the thread-cutting mode, a series of operation is drawn in cutting feed since the operation is executed in cutting feed.

14. List of Added Alarms

Alarm A

1300 Thread feed

'Feed' became a negative value during the thread-cutting operation.
(Unsuitable for 'F' thread pitch command and 'E' variable pitch command.)

Alarm B

2537 Fixed cycle: Thread cycle

[Code]

- 1-> Unsuitable for taper command of thread-cutting fixed cycle.
- 2-> Unsuitable for shift amount designation of thread-cutting fixed cycle.
- 3-> Calculation error during pattern check at thread-cutting cycle.

2538 Fixed cycle: 'E'

[Code]

- 2-> 'Feed' became a negative value during the thread-cutting calculation.
(Unsuitable for 'F' thread pitch command and 'E' variable pitch command.)
- Other-> Command value is not within the 5000000 to -5000000 range (code is hexadecimal of command value).

2540 Compound fixed cycle: mode

[Code]

- 1-> Command was executed at other than normal mode (drilling fixed cycle, cutter-R comp., 3-D tool offset, area machining).
- 2-> Command was executed during coordinate calculation function, or during compound fixed cycle operation.

2541 Compound fixed cycle: Q illegal order

[Code]

- FFFFFFFF->There was no 'Q' command
- Other-> Command value is not within the following range:
 $99999999 \geq \text{command value} > 0$ (code is hexadecimal of command value.)

2542 Compound fixed cycle: QB illegal order

[Code]

- FFFFFFFF->There was no 'QB' command
- Other-> Command value is not within the following range:
 $99999999 \geq \text{command value} > 0$ (code is hexadecimal of command value.)

2543 Compound fixed cycle: H illegal order

One of the following occurred:
'TAN (H/2) < 0' or 'TAN (H/2) = $\pm \infty$ ' (code is the mantissa of TAN (H/2).)

2544 Compound fixed cycle: QA illegal order

Command value was not within the following range:
 $99999999 \geq \text{command value} \geq 0$ (code is hexadecimal of command value.)

2545 Compound fixed cycle: F illegal order**[Code]**

FFFFFFFF->There was no 'F' command

Other-> Command value is not within the following range:
 $99999999 \geq \text{command value} > 0$ (code is hexadecimal of command value.)

2546 Compound fixed cycle: axis command

Axis was not designated for the thread-cutting plane.

G17: X(U), Y(V) were not designated.

G18: Z(W), X(U) were not designated.

G17: Y(V), Z(W) were not designated.

(code axis Nos.: 1:X, 2:Y, 3:Z, 4:4th, 5:5th, 6:6th)

2547 Compound fixed cycle: cycle start point

Compound fixed cycle pattern was unsuitable.

- If cutting end point < start point:
Alarm will be activated when the start point is \leq (cutting end point + thread ridge height).
 - If cutting end point \geq start point:
Alarm will be activated when the start point is \geq (cutting end point + thread ridge height).
- There are no codes.

2548 Compound fixed cycle: QB-QA less than Q (M294)

When M294 is designated, alarm will be activated when the rough machining depth (QB-QA) is less than Q.

There are no codes.

2549 Compound fixed cycle: I, J, K illegal order

When taper angle 'R' command is designated, alarm will be activated when the taper thread start/end point discrepancy 'J' (K, I) command occurs.

[Code]

1-> 'I' command is superfluous.

2-> 'J' command is superfluous.

3-> 'K' command is superfluous.

2550 Compound fixed cycle: angle

Alarm is activated when TAN(R) is $[\infty]$ at TRDL.

Alarm is activated when COT(R) is $[\infty]$ at TRDT.

2551 Compound fixed cycle: QA greater than QB

Alarm is activated when QA is greater or equal to QB.

There are no codes.

2536 Fixed cycle, Compound fixed cycle: 'L'

Alarm is activated when command value is not as follows:

$99999999 \geq \text{command value} \geq 0$ (code is hexadecimal of command value.)

15. List of Added Alarms

Alarm B

2263 Data word: G code

[Code]

- F -> G178, G179, TRDL or TRDT was specified during the execution of the commands below.
G51 (enlarge/reduce), G10 (parallel/rotation shift of coordinate), G236 (copy), G68 (slant face coordinate conversion)

2277 Fixed cycle: 'R'

[Code]

- 1 -> When designating commands with G178/G179, taper angle command R and the difference J (K, I) between the start and end of taper were specified simultaneously.
3 -> At the execution of G178/G179, calculation error took place in analyzing the taper angle command R.

2247 Data word: 'P'

[Code]

- 2 -> When designating commands with G178/G179, a P command value is smaller than or equal to "0".
3 -> When designating commands with G178/G179, a P command value is larger than "99999999".

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

1. Outline

The system is especially designed for carrying out the tool gauging cycle for U-centers*. Tool tip whose position is adjusted through the control of U-axis is brought into contact with the touch sensor mounted on the machine, using the movement of Y-axis. By this operation, the dimension from the spindle center to the tool tip is automatically measured.

The specifications of the system are largely classified into the following three cycles:

- (1) Tool gauging cycle
- (2) U-axis zero offset cycle
- (3) Tool breakage detection cycle

*: U-center: Cutting tools whose radius is adjustable by controlling U-axis

2. Details of Specifications

2-1. Tool Gauging Cycle

This cycle measures the U-center radius, i.e., the dimension from the spindle center to the tool tip automatically.

Tool tip position data is easily known without using the tool presetter in the following cases:

- (1) When a new toolholder is set.
- (2) When a new replaceable insert holder is set.
- (3) When a new replaceable insert is set.

The system also has the following features:

- (4) Tool tip position, after adjusted using the U-axis control, is checked.
- (5) Distance between the spindle center and the tool tip point can be checked even when the U-axis zero point is set at other than the spindle center.

2-2. U-axis Zero Offset Cycle

Zero offset is executed after the completion of the tool gauging cycle as explained above in an arbitrary coordinate system so that the spindle center will come to the program zero of the U-axis. In other words, if zero offset is executed in the current coordinate system, the dimension between the spindle center and the tool tip calculated by the tool gauging cycle will be the actual position of the U-axis.

2-3. Tool Breakage Detection Cycle

If actual radius of the cutting tool measured does not fall within allowable range from the expected dimension, tool breakage alarm will occur and the control stops axis movements.

This cycle is used in the following cases:

- (1) Detection of chipping of the replaceable inserts
- (2) Erroneous setting of toolholders

3. Operation Flow

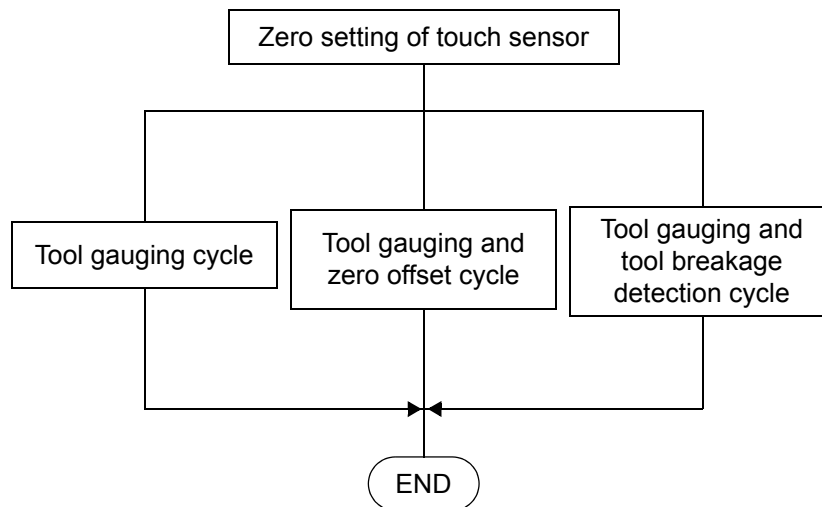


Fig.2-1 Operation Flow

- (1) Zero setting of the touch sensor is usually carried out only once when the machine is installed. However, high gauging accuracy can be maintained if zero setting is carried out regularly. Zero offset cycle executed just before the execution of the gauging cycle will eliminate thermal influence on Y-axis to ensure high accuracy in gauging.
- (2) The type of gauging cycles -- tool gauging cycle, tool gauging cycle with zero offset cycle, and tool gauging cycle with tool breakage detection cycle -- is selected by the data assigned to the specific variable.

Note

Since the tool gauging cycle is carried out using the touch sensor mounted on the table, tool tip must be set in the toolholder so that it is directed in the plus (+) direction of the U-axis. With this setting, it is directed in the -Y direction when the spindle is orient stopped.

4. General Flow of Tool Gauging Cycle

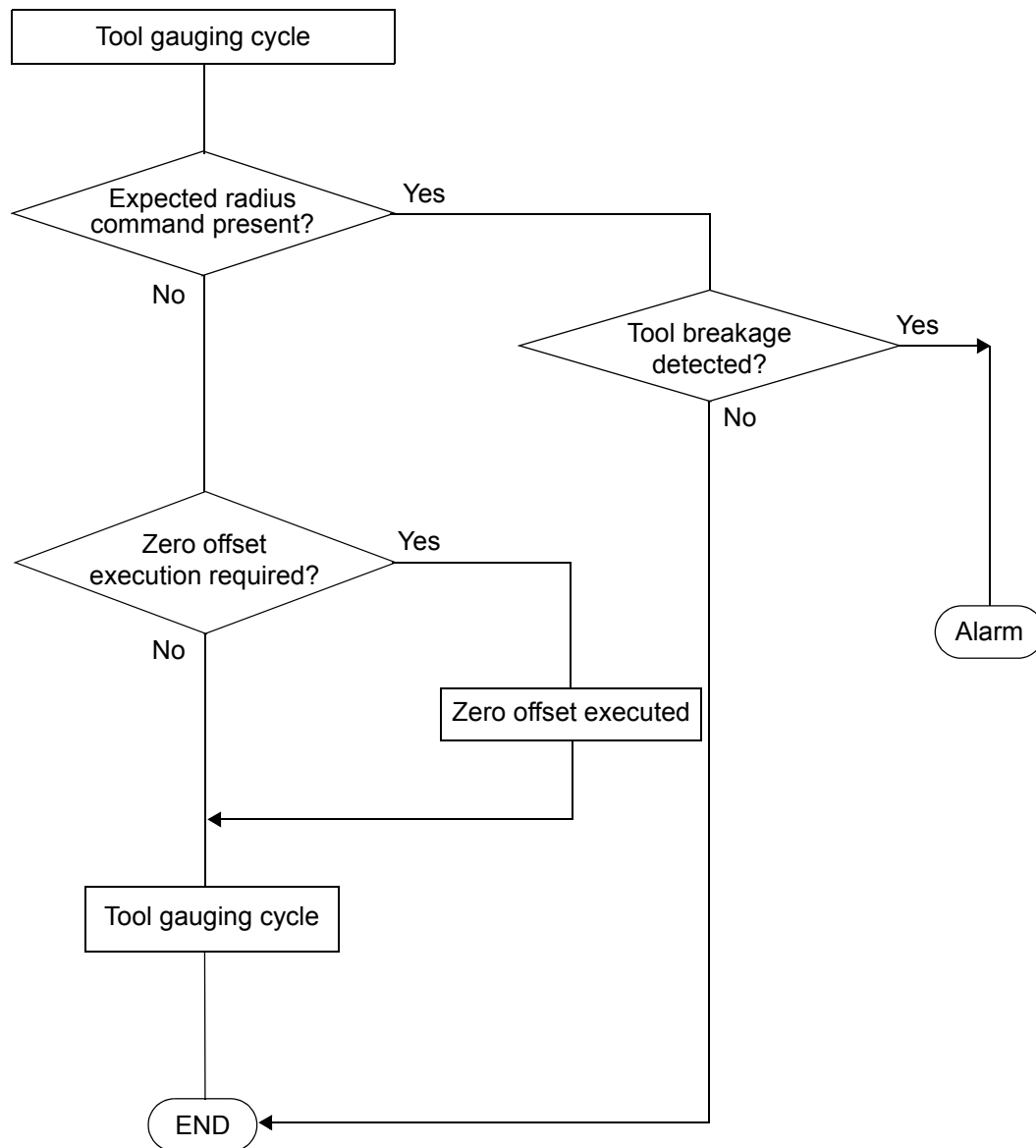


Fig.2-2 General Flow of Tool Gauging Cycle

5. Details of Individual Cycles

5-1. Touch Sensor Zero Setting Cycle

Zero Setting of the touch sensor is carried out in the following steps:

- (1) First set a small tool in the spindle and manually align the small tool center with the touch sensor center (in X-axis direction)

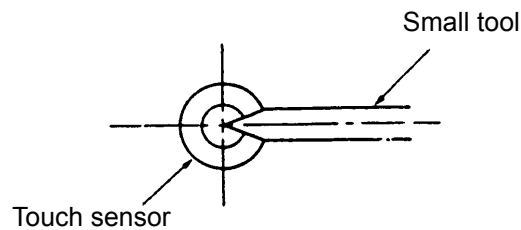


Fig.2-3 Manual Alignment of Small Tool Center with Touch Sensor Center

ME6102001

- (2) Change the small tool with the standard tool manually without moving the X-axis, then bring the standard tool manually until its front edge comes within 10 mm from the touch sensor. In this setting, make sure that the front edge is correctly aligned with the touch sensor center. This alignment adjustment is easily performed by actually bringing the standard tool into contact with the touch sensor and adjusting the alignment. Then, feed the standard tool in Y-axis direction to retract from the touch sensor.

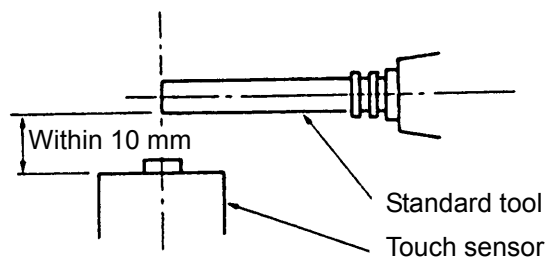


Fig.2-4 Aligning Standard Tool with Touch Sensor

ME6102002

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

- (3) Select the automatic mode and run the program indicated below:
 CALL OO30 PAXI=#97H PY = radius of standard tool
 (PLI = length of standard tool)
 M2

Note

For variables PY and PLI, set values in the unit currently selected. For PY, set the value actually measured. Note that the result of the gauging cycle is greatly dependent on the accuracy of PY data.

The Cycle executed is explained below:

- The Z-axis moves up to the positive travel limit at a rapid feedrate.
- Spindle orientation is carried out.
- The Z-axis moves to the touch sensor position, where the standard tool has been manually placed beforehand.

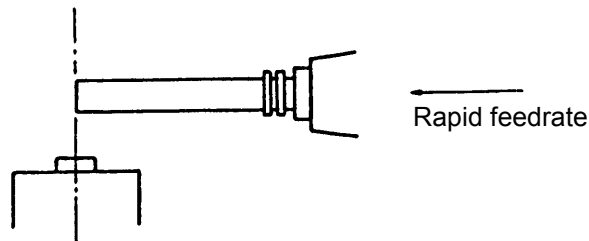


Fig.2-5 Approach of Standard Tool to Touch Sensor

ME6102003

- The Y-axis is fed to the touch sensor at a middle approach speed (F1000). Axis feed stops when the standard tool contacts the touch sensor.

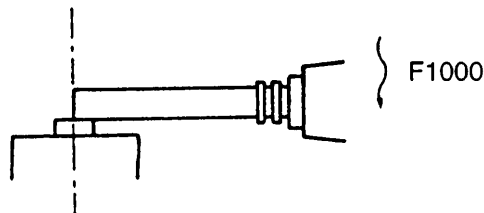


Fig.2-6 Y-axis Approach to Touch Sensor

ME6102004

- The Y-axis returns by 0.5 mm from the contact point.

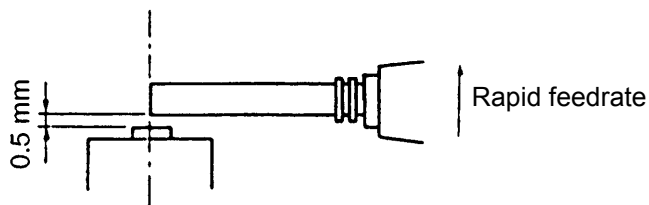


Fig.2-7 Y-axis Return from Touch Sensor

ME6102005

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

- f) The Y-axis is fed again to the touch sensor at a slow approach speed (F10). Axis feed stops when the standard tool contacts the touch sensor.

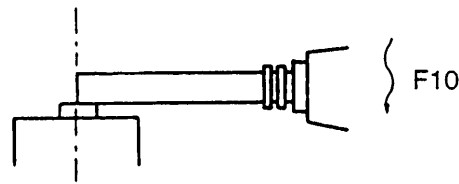


Fig.2-8 Slow Y-axis Approach to Touch Sensor

ME6102006

- g) From the position where the Y-axis movement stops, the touch sensor contact position is calculated and position data of X, Y and Z axes are saved to variables VSZO*[4].
- h) The Y-axis returns to the position, where the standard tool is initially located in step c).

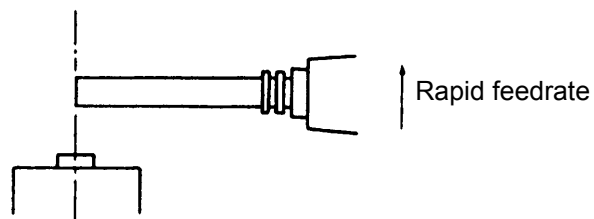


Fig.2-9 Rapid Retraction of Y-axis

ME6102007

- i) The Z-axis returns to the positive travel end at a rapid feedrate.

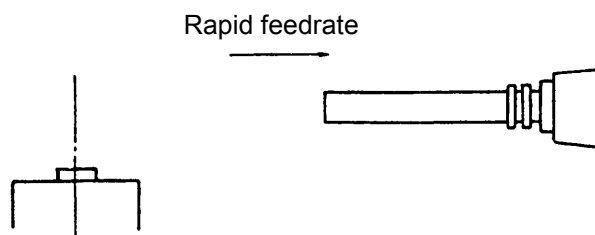


Fig.2-10 Rapid Retraction of Z-axis

ME6102008

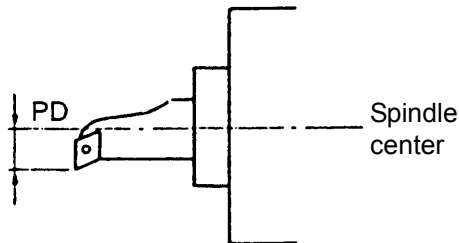
Note

The work coordinate system #4 is specially used for the touch sensor for side gauging. Therefore, neither setting nor referencing is possible in the zero set mode.

5-2. Tool Gauging Cycle

Tool tip position of the U-center is measured using the Y-axis movement of the machine. For this gauging cycle, it is necessary to set the tool length offset value of the U-center to the H number identical to the actual tool number.

- (1) Programming
CALL OO40 PSET = 1 (PD = ∞)



PSET = 1 ... To be specified always
PD When expected radius is set:

- The first approach speed to the touch sensor is increased.
- If the measured value is greater or smaller than the expected value PD by approximately 1 mm, it causes an alarm and axis movement stops.

Fig.2-11 Expected Radius

ME6102009

- (2) Cycle
The cycle operations executed when the program above is run in the automatic mode is explained below:

- The Z-axis moves to the positive travel end at a rapid feedrate.
- Spindle orientation is carried out.
(U-center tool tip is oriented in the Y-axis direction.)
- The following positioning is carried out simultaneously:
X-axis... Touch sensor position
Y-axis... $150 + 10$ mm in the negative direction from the touch sensor position
- Z-axis is fed at a rapid feedrate so that the U-center tool tip comes to the center of the touch sensor. In this positioning, the tool length offset value is taken into account. That is, the Z-axis is shifted by the amount set in the tool offset memory of the same number as the tool number of U-center.

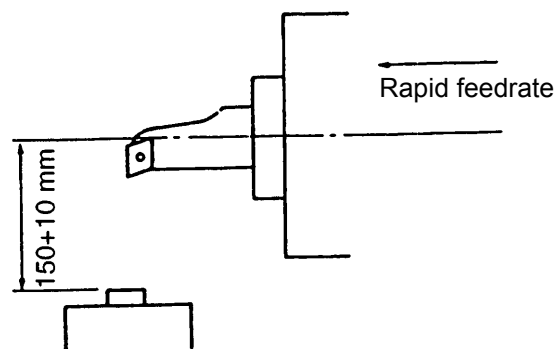


Fig.2-12 Positioning of U-Center above Touch Sensor

ME6102010

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

- e) If a PD command has been specified, the Y-axis moves to the position (PD+10) mm in the positive direction from the touch sensor position A F4000. If the U-center comes into contact with the touch sensor, an alarm occurs (2305 ALARM-B UNTENDED: gauging impossible 33).

Note that this movement occurs only when a numerical value is set at PD.

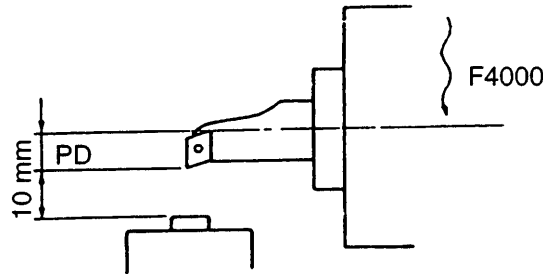


Fig.2-13 Y-axis Movement when PD Command is Specified

ME6102011

- f) The Y-axis moves up to a point just before the negative travel end at a middle approach speed (F1000).

If the PD command has been specified, Y-axis moves toward the point 10 mm beyond the expected sensor contact point. If such a point is beyond the negative travel end, the target point for this movement is changed to the point just before the negative travel end.

Y-axis movement stops as soon as the tool tip comes into contact with the touch sensor.

If no contact is detected during this movement, an alarm occurs (2305 ALARM-B UNTENDED: gauging impossible 34).

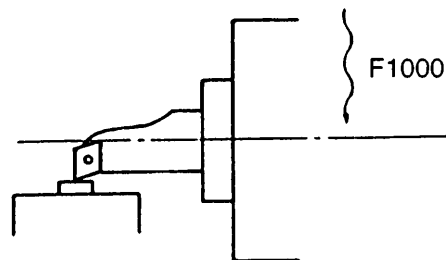


Fig.2-14 Y-axis Middle Speed Approach to Touch Sensor

ME6102012

- g) The Y-axis returns by 0.5 mm from the contact point at a rapid feedrate.

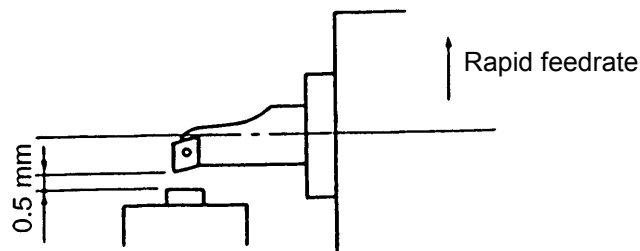


Fig.2-15 Y-axis Retraction from Contact Point

ME6102013

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

- h) The Y-axis again moves toward the touch sensor at a slow approach speed (F10). In this approach, the target point is set at a point 1 mm beyond the sensor contact point. Axis movement stops as soon as the tool tip comes into contact with the sensor. If the PD command has not been specified, step jumps to step j).

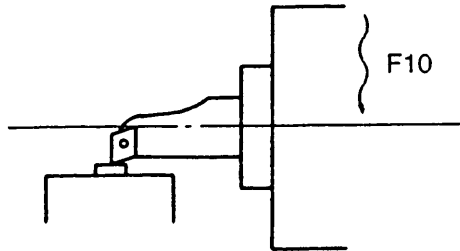


Fig.2-16 Y-axis Slow Speed Approach to Touch Sensor

ME6102014

- i) The U-center radius measured by the cycle above and the expected radius set at PD are compared. If difference is greater than 1 mm, an alarm occurs (ALARM B 2307 UNTENDED: off centerline).
- j) The Y-axis returns to the position, where the standard tool is initially located in step d).
- k) The Z-axis moves to the positive travel end in a rapid feedrate.

Note

If the measured value is positive, it indicates that the cutting tool is at the ID cutting position (tool tip orientation is opposite to the spindle center).

If the measured value is negative, it indicates that the cutting tool is at the OD cutting position (tool tip orientation is in the spindle center side).

5-3. Zero Offset Cycle

Zero offset of U-axis is executed when the tool gauging cycle explained in 5-2 is executed so that the value (distance between the spindle center and the tool tip point) measured in the cycle will be the actual value of the U-axis.

(1) Programming I

CALL OO40 PHN = oo (PD = oo POST = oo PDE = oo)

PHN = Zero offset work coordinate system of U-axis (alarm if omitted)

PD = Expected radius of U-center tool

PDE = Permissible error range from expected radius PD (default: 1 mm)

POST = Amount to be added in zero offset (default: 0)

(2) Cycle

The axis movements in the zero offset cycle are identical to the cycle explained in 5-2.

a) The zero offset is carried out between the steps i) and j) in 5-2.

The program zero of the coordinate system designated by PHN is shifted so that the measured value (= U-center radius) is identical to the actual position value of the U-axis.

b) When POST = oo is designated, the program zero of the coordinate system designated by PHN is shifted so that actual position value of the U-axis will be the current actual position value added with "oo".

Example 1: CALL OO40 PHN = 1

Measured value in tool gauging cycle	25.023
Actual position value on H1 coordinate system	25.000

└─ Zero offset is executed so that the actual position value on the H1 coordinate system will be 25.023.

Example 2: CALL OO40 PHN = 1 POST = 0.1

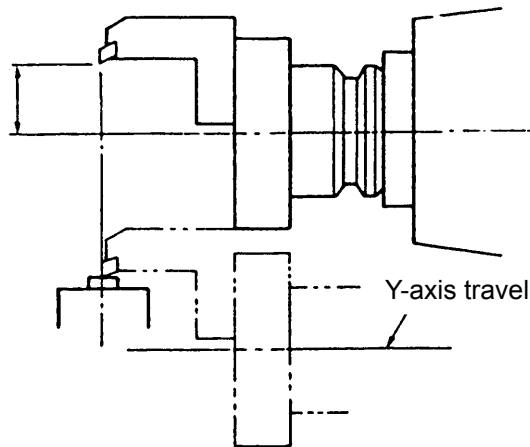
Measured value in tool gauging cycle	25.023
Actual position value on H1 coordinate system	25.000

└─ Zero offset is executed so that the actual position value on the H1 coordinate system will be 25.123.

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

(3) Programming II

In the Y-axis travel range, you can measure the OD tool edge position by directly touching the tool edge with the touch sensor.



If the gauging stroke exceeds the Y-axis travel range, indirectly measure the edge position in the method described below and set the zero position of the tool.

Fig.2-17 Direct Measurement of ID Using Touch Sensor

ME6102015

CALL OO40 PHN = 00 POST = 00 PRVS = 1 PTOF = 00
 PHN = U-axis zero offset work coordinate system (alarm if omitted)
 POST = Amount to be added in zero offset (default: 0)
 PRVS = Indirect gauging mode (default: direct gauging mode)
 PTOF = Incremental amount in reference to tool offset amount of U-center (default: 0)
 Note that this setting is effective only when PRVS is set at "1".

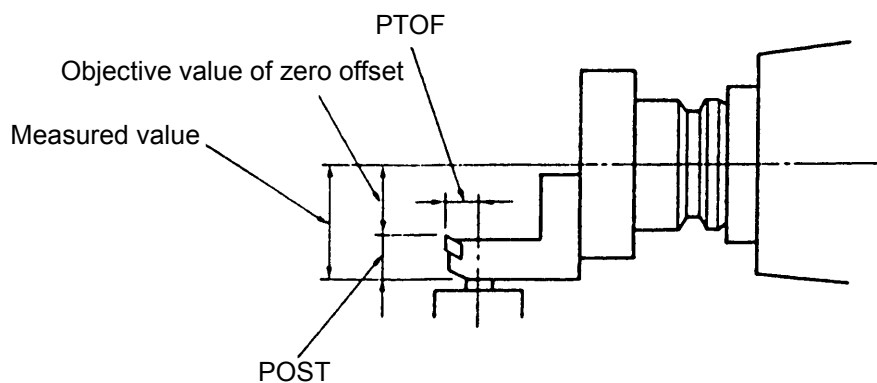


Fig.2-18 Relationship of Parameters Used in Zero Offset Operation

ME6102016

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

Example 3: CALL OO40 PHN = 1 POST = 38.501 PRVS = 1 PTOF = -25

If PRVS = 1 is set, the spindle rotates 180° automatically after spindle orientation.

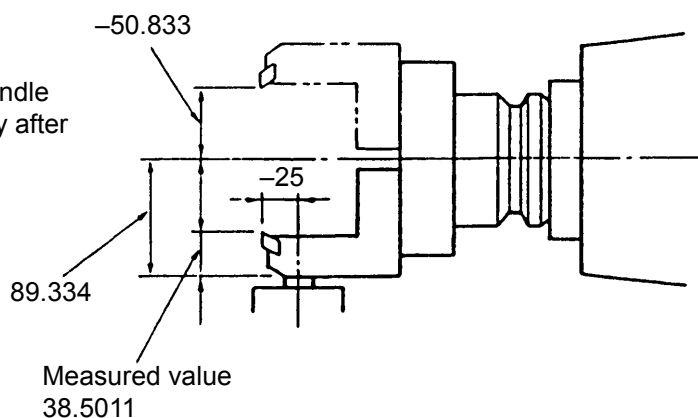


Fig.2-19 Example of Zero Offset

ME6102017

As illustrated above, the back face of the toolholder is brought into contact with the touch sensor and the dimension between the spindle center and the toolholder back face is automatically measured (89.334).

Based on the measured value and the POST data (38.501), dimension from the spindle center to the tool tip is calculated ($50.833 = 89.334 - 38.501$). Then zero offset is executed so that the actual position value will be "-50.833".

Note that the POST value must be measured in advance.

Example 4: N1 CALL OO40 PSET = 1 PRVS = 1 PTOF = -25
N2 CALL OO40 PSET = 1

Measured value calculated from N177.084

Measured value calculated from N2-38.583

POST data 38.501 is calculated from the values indicated above: $77.084 - 38.583 = 38.501$

Note

The POST data cannot be measured precisely disregarding of the measurement methods whether other device is used or the touch sensor on the machine is used. Therefore, the zero offset value obtained in the manner as in Example 3 may be used only as a rough value.

- (4) Parameters
 NC optional parameter bits

No.	bit	Description
74	5	<input type="checkbox"/> The U-axis is indicated in radius and used in the U-axis zero offset cycle.
		<input checked="" type="checkbox"/> The U-axis is indicated in diameter and used in the U-axis zero offset cycle. Even if this parameter is checked, all commands will be executed with the radius value except that POST (Amount to be added in zero offset (default: 0)) is a diameter command.

5-4. Tool Breakage Detection Cycle

If the measured value does not fall within the permissible range from the expected radius, the tool broken alarm occurs and the machine stops.

(1) Programming

CALL OO40 PSET = 1 PD = 000 PDE = 000

PSET = 1 To be set always

PD = Expected radius of U-center tool To be set always

PDE = Permissible error range from expected radius PD (default: 1 mm)

(2) Cycle

The axis movements in the tool breakage detection cycle are identical to the cycle as explained in 5-2.

The U-center tool radius measured by the movement of Y-axis is compared with the radius designated by PD and if the difference is greater than the permissible error set at PDE, the tool broken alarm occurs.

$| \text{Measured Value} - \text{PD} | > \text{PDE}$ An alarm occurs

Example 5: CALL OO40 PSET = 1 PD = 30.123 PDE = 0.1

Measured value: 30.003

Tool broken alarm occurs since $| 30.003 - 30.123 | = 0.12 > 0.1$

6. Display of Measured Data on Crt

After the execution of the tool gauging cycle or the tool breakage detection cycle, the results of the gauging cycle are displayed at the PERSONAL screen on the CRT. Example of display is explained below:

6-1. Tool Gauging Cycle Example

In the tool gauging cycle, the length of the U-center tool from the spindle center to the tool tip point displayed.

CUTTER RADIUS OF U-CENTER 35.123

6-2. Tool Breakage Detection Cycle Example

In the tool breakage detection cycle, the length of U-center tool between the spindle center and the tool tip point and judgement result on tool breakage are displayed.

(1) OK judgement

CUTTER RADIUS OF U-CENTER 35.123

TOOL BREAKAGE DETECTION: TOOL MANAGEMENT NO. 3 DECISION OK

(2) When tool is judged broken

CUTTER RADIUS OF U-CENTER 35.567

TOOL BREAKAGE DETECTION: TOOL MANAGEMENT NO. 3

ALARM TOOL BREAKAGE 0.123 [AMOUNT OF TOOL BREAKAGE (U)]

| Expected value – Measured value |

6-3. Others

When gauging cycle is not completed satisfactorily since the tool tip failed to contact the sensor, etc. the following message is displayed.

TOOL BREAKAGE DETECTION: TOOL MANAGEMENT NO. 3

ALARM IMPOSSIBLE TO MEASURE

7. Precautions

- (1) If execution of a subprogram is attempted under the following conditions, the 'gauging impossible' alarm will occur:
 - a) Machine coordinate system is selected.
 - b) Mirror image is ON.
 - c) Manual shift amount is not "0".
- (2) Subprogram cannot be executed in the machine lock condition.
- (3) Rotation and shift of coordinate system, enlargement/reduction of coordinate are all cancelled when a subprogram is executed.
Therefore, if such conversion is required after the completion of a subprogram it is necessary to specify them again.
- (4) Cutting tools and tip holder must be set so that the tip is directed in the +U direction.
- (5) Before executing the gauging cycle subprogram OO40,
 - a) input the tool length offset value for U-center tool in advance. The offset number (H number) must be identical to the active tool number, and
 - b) make sure that the zero point has been established with the zero offset setting cycle OO30.
- (6) The second approach speed F10 to the touch sensor is not influenced by the setting of the cutting feedrate override dial.
- (7) The measured value obtained using the touch sensor is the U-center tool position while it is stationary. This position will be changed during cutting due to deflection, etc. depending on the cutting conditions. Therefore, if a hole with strict tolerance is to be machined using the U-center tool, it is necessary to compensate for the U-center tool position after carrying out trial cutting.
- (8) Nesting level in calling sub routines from a subprogram is two. Therefore, allowable sub routine nesting level is six.

8. Program Examples

(1) General Programming

N1	T1						: T1: U-center tool
N2	M6						
N3	T3						: T3: Next tool
N4	G15	H1					
N5	G0	U20	S200				: U-axis movement (U-axis is at the home position when M6 command is executed.)
N6	CALL	OO40	PHN = 1				: U-axis gauging cycle, result is saved in H1. Zero offset
N7	G0	X100	Y100	G56	H1		
N8		U25.005					
		:					
		:					
		:					
N30	G0	Z300	M5				
N31	CALL	OO40	PSET = 1	PD = 25.005	PDE = 0.1		: Tool breakage detection cycle
N32	M6						: When M6 is executed, the U-axis is automatically returned to the home position.

(2) Zero Setting of Touch Sensor is Programmed and Executed before Execution of gauging Cycle (It is necessary to set the zero position in advance.)

N1	T4						: Standard tool
N2	M6						
N3	T1						: U-center tool
N4	G16	H54	G0	X0	Y100		: X-axis positioning at touch sensor position
N5	G16	H54	Z0				: Z-axis positioning at touch sensor position
N6	G16	H54	Y34				: Y-axis positioning (9 mm in front of touch sensor position)
N7	CALL	OO30	PAXI = #97H	PY = 24.995			: Touch sensor zero setting cycle
N8	M6						
N9	T3						
N10	G15	H1					
N11	G0	U20	S200				: U-axis movement
N12	CALL	OO40	PHN = 1				: U-axis gauging cycle, result is saved in H1. Zero offset
N13	G0	X100	Y100	G56	HI		
N14		U25.005					
		:					
		:					
		:					

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

9. Program List

Table 2-1 Program List

Contents of Subprogram	Subprogram Name	Variables always Used	Variables Used a Needed
Zero Setting of Touch Sensor	OO30	PAXI = #97H Zero setting of X, Y and Z axes PY: Radius of standard tool	PLI: Tool length of standard tool (default: 200 mm)
Tool Gauging Cycle	OO40	PSET = 1	PD: Expected radius PDE: Permissible error range (default: 1 mm) PRS: Spindle orientation position PRVS = 1 Indirect gauging mode PTOF: Incremental amount in reference to tool offset amount (effective only when PRVS = 1)
Zero Offset Cycle	OO40	PHN: Coordinate system no. for which zero offset is executed	PD: Expected radius PDE: Permissible error range (default: 1 mm) POST: Value to be added to zero offset PRS: Spindle orientation position PRVS = 1 Indirect gauging mode PTOF: Incremental amount in reference to tool offset amount (effective only when PRVS = 1)
Tool Breakage Detection Cycle	OO40	PSET = 1 PD: Expected radius	PDE: Permissible error range (default: 1 mm) PRS: Spindle orientation position

SECTION 2 TOOL GAUGING SYSTEM OF U-CENTER

10. Alarm List

Table 2-2 Alarm List

Alarm Name	Cause and Measures to Take
UNTENDED: gauging impossible (ALARM B 2305)	<p>The code indicating the cause of alarm is displayed at the end of the alarm message.</p> <p>2305 ALARM B UNTENDED: gauging impossible **</p> <p style="text-align: right;">↑ Cause code</p> <p>Measures to take for individual cause are summarized in "Table 2-3".</p>
UNTENDED: command missing (ALARM B 2308)	<p>Cause: Variables to be specified are not specified.</p> <p>Measures to take: Change the program conforming to the programming rule.</p>
UNTENDED: off centerline (ALARM B 2307)	<p>Cause: Difference between the value commanded by PD and the U-center radius measured is greater than the permissible error range specified by PDE.</p> <p>Measures to take: Observe the U-center tip. Is it chipped or are chips accumulating on it? If the tip has been chipped, replace it. If chips are accumulating on the tip, remove chips.</p>

Table 2-3 Cause of Impossible Measurement Alarm and Measures to Take

Cause Code	Cause and Measures to Take
1	<p>Cause: Subroutine is called out from the machine coordinate system (H0)</p> <p>Measures to take: Call the subroutine only after selecting a work coordinate system.</p>
2	<p>Cause: Mirror image is active.</p> <p>Measures to take: Cancel the mirror image.</p>
3	<p>Cause: Manual shift amount (sum) is not zeroed.</p> <p>Measures to take: Clear the manual shift amount.</p>
33	<p>Cause: In the OO40 cycle, contact to the touch sensor is detected in the sequence in which contact must not be detected.</p> <p>Measures to take: Check PD value; also check the sensor.</p>
34	<p>Cause: In the OO40 cycle, contact to the touch sensor is not detected in the sequence in which contact must be detected.</p> <p>Measures to take: Check PD value; also check the sensor.</p>

SECTION 3 U-AXIS DIAMETER INSTRUCTION

1. Outline

When the U-axis is provided to the machining center to meet needs where dimensions indicated on drawings are diameters and it is convenient to use diameters for measuring dimensions, diameter commands are used as the U-axis commands.

2. Function

(1) The diameter commands are usable in the following command blocks.

- G00
- G01
- G02/G03 (I, J and K are radius commands.)
- Helical cutting
- Angle command (AG command)
- Unidirectional positioning (G60)
- Exact stop check
- Skip function (G31)
- Programmable mirror image (G62)
- Tool length offset (Tool length offset data for the diameter instruction axis is set with the diameter command value.)
- Cutter radius compensation (The override function is included. Use the radius command for specifying the cutter radius compensation value.)
- 3-D tool offset (I, J and K tool offset values are set by radius commands.)
- Parallel or rotary movement of coordinates (local coordinates system)
- Copy function
- Drawing enlargement/reduction function
- Fixed cycle
- Home position

(2) If diameter axis commands are used for the following instructions, alarms will occur.

- Area machining
 - Coordinates calculation
- Alarm B: 2579 Diameter instruction axis command
 A diameter instruction axis is specified for a function to which use of diameter instruction axis commands is prohibited.
- Index: Axis
- Code: 1 Area machining
 2 Coordinates calculation

SECTION 3 U-AXIS DIAMETER INSTRUCTION

- (3) Use the radius command for the following data settings:
- Tool data setting:
 - Cutter radius compensation value
 - Parameter setting:
 - NC optional parameter long word
 - G73 cycle relief
 - G83 cycle relief
 - Arc check data
 - Cutter radius compensation vector check data
 - Deceleration distance at end point of cutter radius compensation corner override
 - Deceleration distance at start point of cutter radius compensation corner override
 - Vector size for 3-D tool offset
 - Computation processing data for circumference of circle containing the small arc at cutter radius compensation
 - Relief before spindle orientation in G76, 87 cycles
- (4) For the following data settings, the diameter command is used when the subject axis is a diameter instruction axis.
- Tool data setting:
 - Tool length offset
 - Home position offset setting
 - Parameter settings:
 - System parameters
 - Travel end limit
 - Thread pitch compensation range
 - Thread pitch compensation interval
 - In-position width
 - HP in-position width
 - Location of home position
 - Origin of machine coordinates system
 - Encoder offset value
 - Barrier
 - Inductosyn offset value
 - User parameters
 - Programmable travel limit
 - G60 overtravel distance
 - Backlash
 - Thread pitch compensation
- (5) Others
- Line graphics and animation by diameter instruction axis are not supported.
 - Automation (auto gauging, tool offset and tool breakage detection) are not supported.

SECTION 4 U-AXIS TORQUE LIMIT FUNCTION

1. Introduction

The torque limit function limits the spindle torque for protecting the U-axis attachment.

The torque limit control is always effective.

Set a value at the following parameter with a percentage of the spindle torque.

2. Caution

(1) When the parameter is set to 100 (%), no torque limit control is performed.

(2) No torque limit control is performed without change of default parameter setting (0%).

3. Parameters

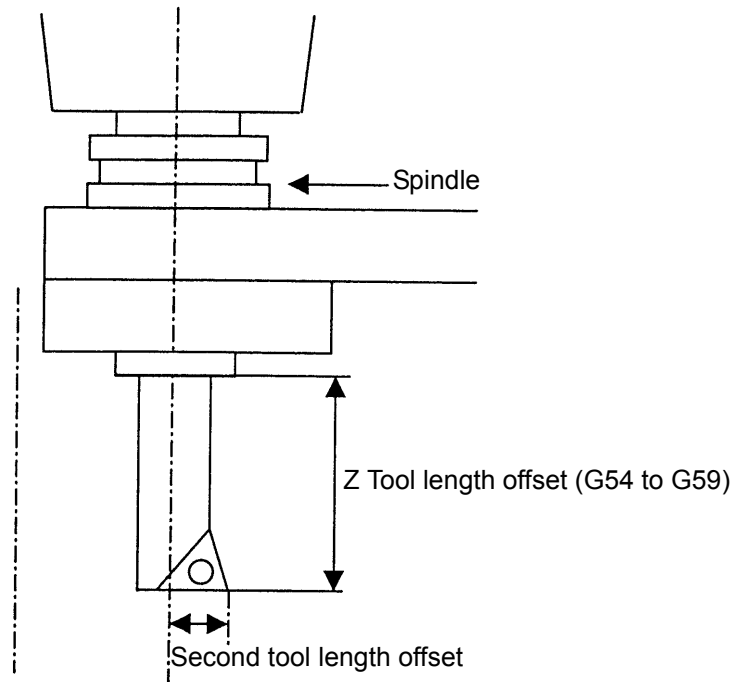
- NC optional parameter (word) No. 64

Key command	Maximum	Minimum	Default	Setting unit
SET/ADD	100	1	0	%

SECTION 5 SECOND TOOL COMPENSATION FUNCTION

1. Outline

A U-center tool requires compensation to be made both in the Z-axis direction (spindle direction) and the bite direction (U-axis direction). Compensation in the bite direction is enabled using the second tool compensation function.



ME6105001

Second tool length offset and conventional tool length offset are independent functions. These two tool length offset functions can be used together.

2. Specifications

Command form

Second tool length offset ON/OFF command form (G188 and G189 belong to group No. 69.)

Second tool length offset ON

G189 PH**

P: Second tool length offset axis specification

- The U-axis will be specified when no axis is specified, or an alarm will occur if two or more axes are specified.
- Set the address name on the axis name.
- When the turning cut function (optional) is available, the X-axis will be specified if no axis is specified.

H: Second tool length offset number

- Set compensation amount data with the same compensation number as that of the previous tool length offset.
- The number of the second tool length offset previously executed will be specified if this number is omitted.
- The number will be cleared by resetting the NC.

Second tool length offset OFF

G188

Note

- 1) When second tool length offset and G15 and G53 - G59 are commanded in the same block, H will be considered H of the G code given a higher priority. G15 > G53 to G59 > G188, G189
- 2) "2253 Alarm B Data word: axis command FFFFFFFF" occurs because of any of the following causes when G189 is commanded:
 - a) An axis other than the 2nd axis was specified in second tool length offset axis specification.
G189 X1Y1H1
 - b) The axis name address data specified for second tool length axis specification was not 1.
G189 X5 H1
 - c) The turning axis was specified for second tool length offset axis specification.
G189 A1 H1
- 3) Second tool length offset will be canceled with second tool length offset OFF (G188) or NC reset (including M02 and M30) power OFF. (The second tool length offset axis will also be canceled.) Second tool length offset will not be canceled with G53.
- 4) The second tool length offset number will be cleared (H00) with NC reset (including M02, M30) power OFF. It will not be cleared with second tool length offset OFF. The H number held will be set if H is not specified in the next G189 command.

3. Display

The second tool length offset number is displayed on the following screens:

- (1) Each page of actual position screen
- (2) Program display screen (main, schedule, MDI)
- (3) Block data screen (current, next, next after next, 3 pages ahead)

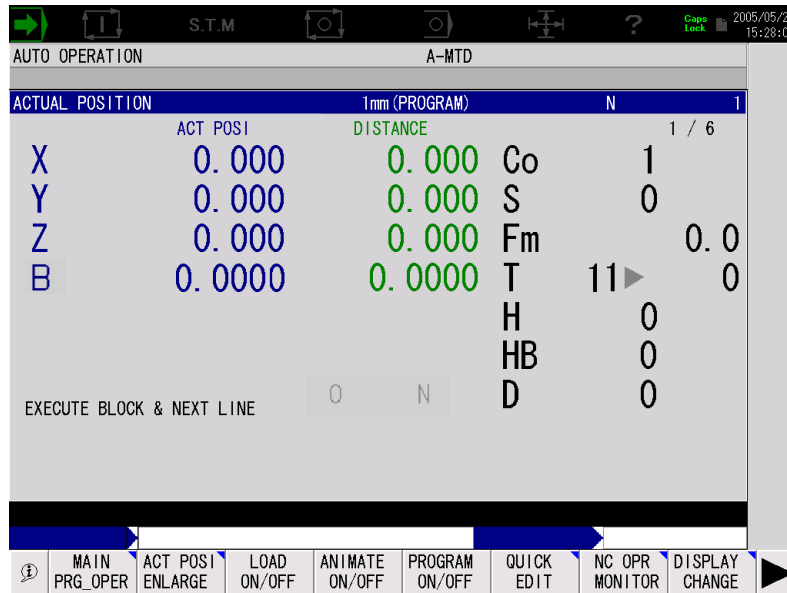


Fig.5-1 Each page of actual position screen

SECTION 5 SECOND TOOL COMPENSATION FUNCTION

ME6105002

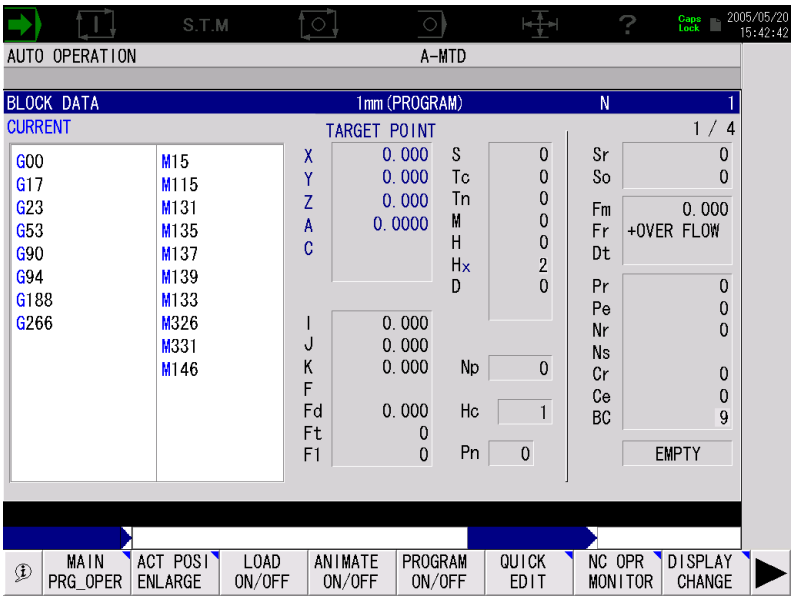


Fig.5-2 Block data display screen (current, next, next after next, next after next after next)

ME6105003

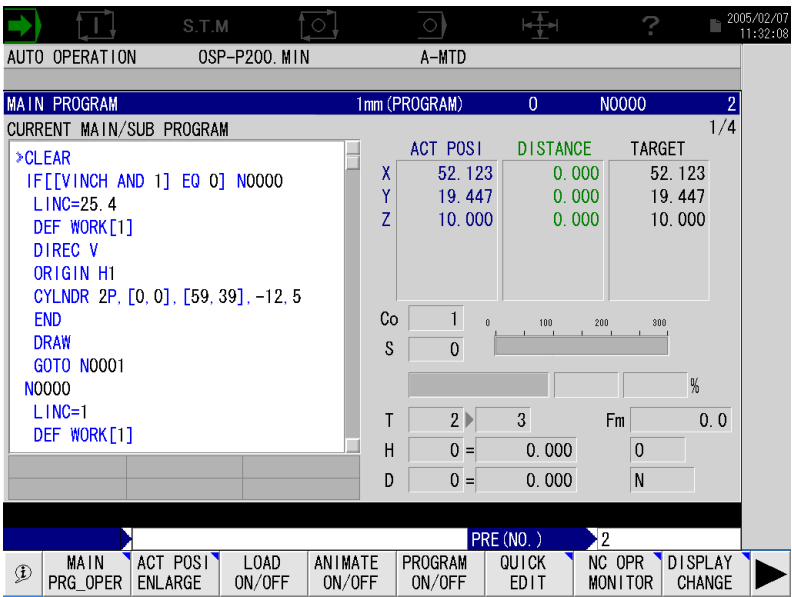


Fig.5-3 Program display screen (main schedule, MDI)

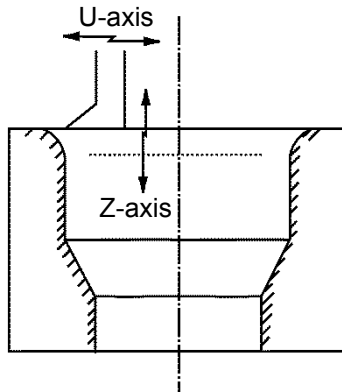
ME6105004

SECTION 6 NOSE RADIUS COMPENSATION

1. General Description of Nose Radius

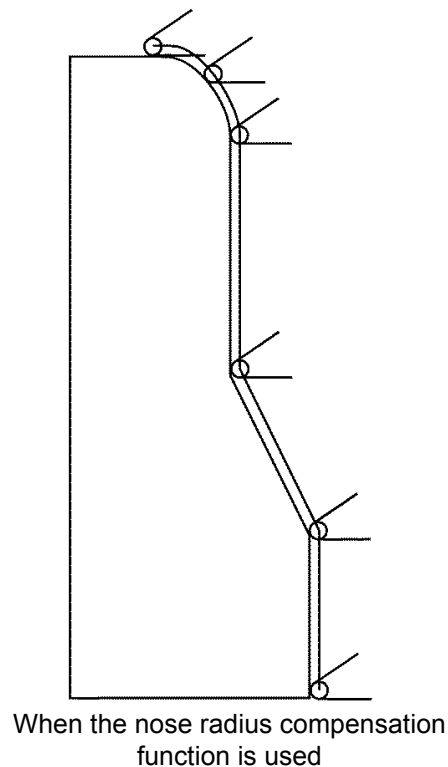
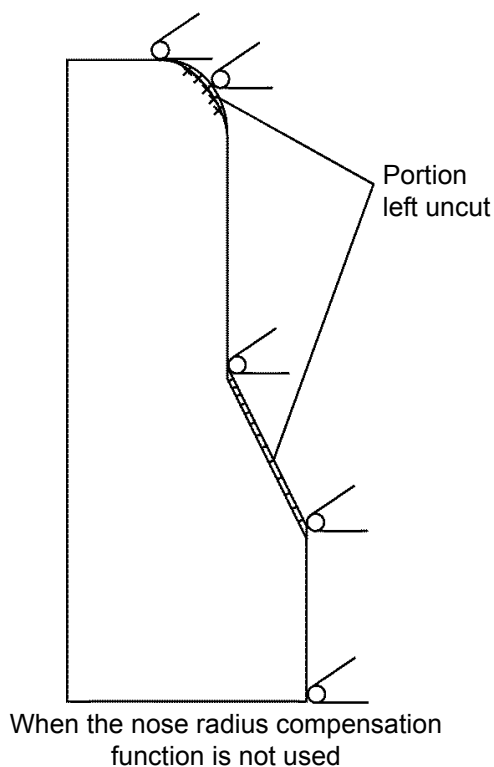
On a machining center, if the U-axis is implemented to the U-center, turning operation is enabled using a tool mounted to the U-center.

In this case, the tool tip is given a tip radius, which causes an error between the commanded shape and the machined shape. The nose radius compensation function automatically compensates for this error caused by this tip radius using a simple program.



If the hatching portion is machined with the U-axis and the Z-axis as shown in the left figure, an error occurs as shown in the left figure shown below. The right figure below shows the movement of the tool when the nose radius compensation function is used.

ME6106001



ME6106002

2. **Nose Radius Compensation and Tool Radius Compensation**

The basic operation of nose radius compensation and that of tool radius compensation are the same. The nose radius or tool radius will be automatically compensated when the operator executes the program according to the shape to be machined. Difference between nose radius compensation and tool radius compensation is described below:

The nose radius compensation compensates for the radius by setting the nose radius center position with the offset amount and the position number to the imaginary tool tip.

In the mean time, the tool radius compensation compensates for the radius by setting the tool radius as the offset amount to the cutter center (cutter turning center).

3. Nose Radius Compensation Value and Position Number (P)

Specify the nose radius compensation value and the direction of the nose radius (position number) as shown below:

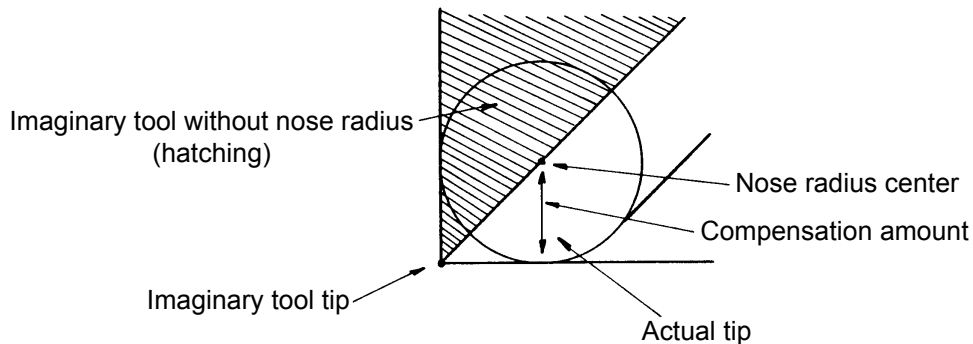


Fig.6-1 Imaginary tool tip and compensation amount

ME6106003

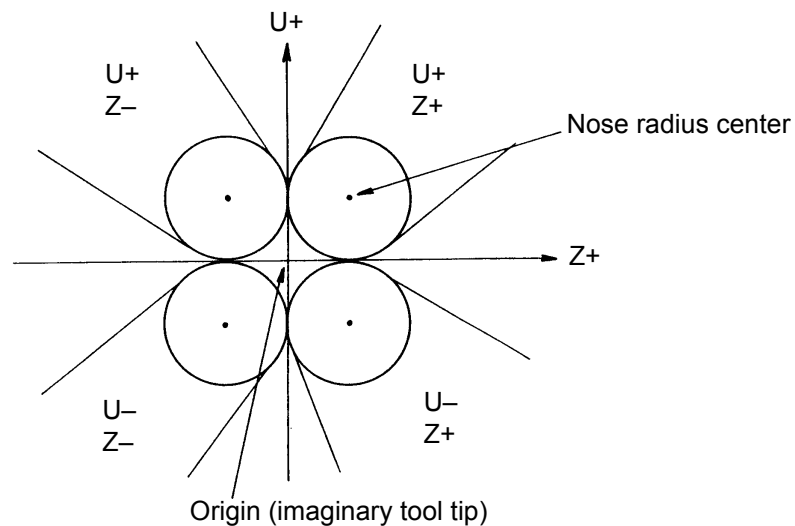


Fig.6-2 Compensation value codes based on nose radius center position

ME6106004

Determine the number (P number) for each nose radius center position to identify the direction.

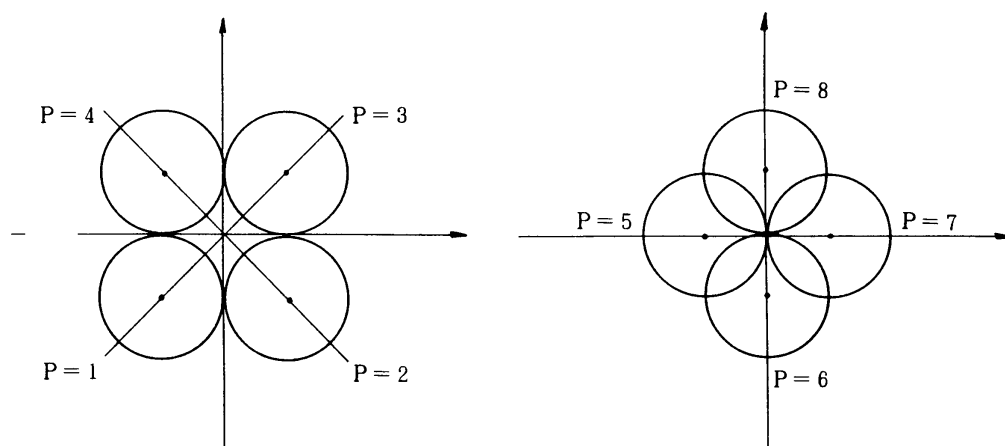


Fig.6-3 P numbers by nose radius center position

ME6106005

When $P = 0$ or 9 , the imaginary tool tip and the nose radius center are located in the same position.

4. Setting Nose Radius Compensation Value and Position Number

The tool radius compensation amount and the nose radius compensation amount must be the same data. Set the nose radius compensation amount and the position number for nose radius center setting on the setting screen shown below.

Setting procedure

Procedure :

- 1 Select tool data setting mode.
- 2 Have the tool length offset and tool radius compensation screen shown below displayed using the ITEM keys for tool data display.
- 3 Using the page key, have the screen for the desired nose radius compensation number displayed.
- 4 Using the cursor keys, move the cursor to the data setting position for the nose radius compensation number.
- 5 Set the desired nose radius compensation amount.
[The setting of the nose radius compensation amount is completed.]
- 6 Move the cursor right to the position number setting position (the position of P).
- 7 Set a value between 0 and 9 for P data.
For values from 0 to 9, see "3. Nose Radius Compensation Value and Position Number (P)".

TOOL DATA			
RAK432H. MIN A-MTD			
TOOL OFFSET/COMPENSATION			
TOOL LENGTH OFFSET (H--)		CUTTER R COMP (D--)	
NO.	NO.	NO.	P NO.
1	1.500	11	16.500
2	3.000	12	18.000
3	4.500	13	19.500
4	6.000	14	21.000
5	7.500	15	22.500
6	9.000	16	24.000
7	10.500	17	25.500
8	12.000	18	27.000
9	13.500	19	28.500
10	1.000	20	2.000

[SET UNIT: 1mm]

USE PAGE KEY TO CHANGE THE OFFSET/COMP. NO.

PRE (SET) 10

SET ADD CAL FIND ITEM ↑ ITEM ↓ DISPLAY CHANGE

Fig.6-4 Nose radius compensation and P-corresponding tool data setting screen

ME6106006

Note

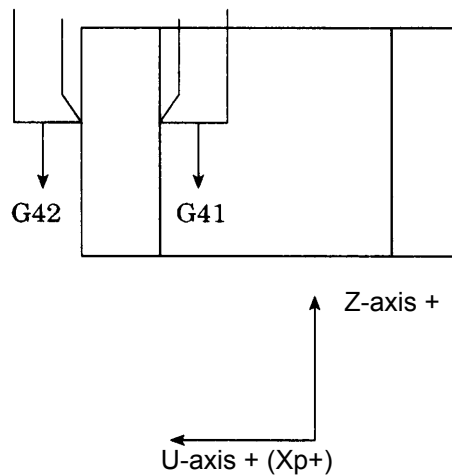
When the position number (P) is 0 or 9, it will be treated as regular tool radius compensation data.

5. Nose Radius Compensation Program

The nose radius compensation program is the very same as the tool radius compensation program.

- (1) Specifying (Selecting) a plane

G17 : Xp Yp plane command	XpX-axis or U-axis
G18 : Zp Xp plane command	YpY-axis or V-axis
G19 : Yp Zp plane command	ZpZ-axis or W-axis
- (2) G codes used for nose radius compensation
 - G40 : Canceling nose radius compensation
 - G41 : Left cutting (The center of the tool passes on the left side of the machined face to the relative motion direction of the tool.)
 - G42 : Right cutting (The center of the tool passes on the right side of the machined face to the relative motion direction of the tool.)



ME6106007

- (3) Command form

G17	G41 (G42)	Xp_	Yp_	D_
[G18	G41 (G42)	Zp_	Xp_	D_]]
[G19	G41 (G42)	Yp_	Zp_	D_]]

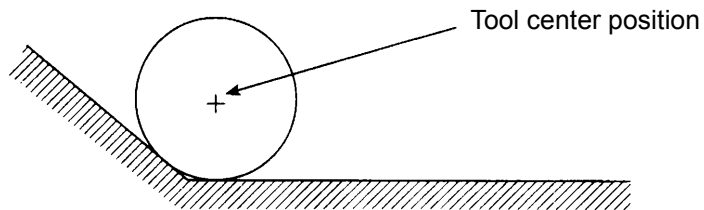
Note

D is the nose radius compensation number.

6. Display

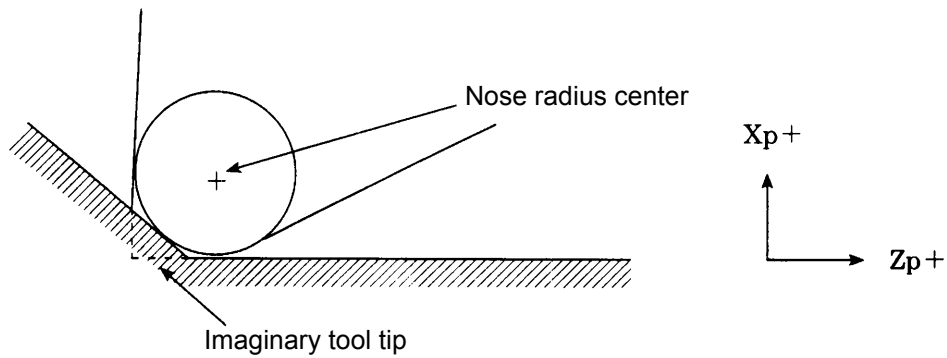
The tool center (vertical center) is displayed for the actual position when tool radius compensation is executed. However, the imaginary tip position is displayed when nose radius compensation is performed.

- (1) In tool radius compensation
The tool center position is displayed.



ME6106008

- (2) In nose radius compensation (when $P = 3$)
The imaginary tool tip is displayed.



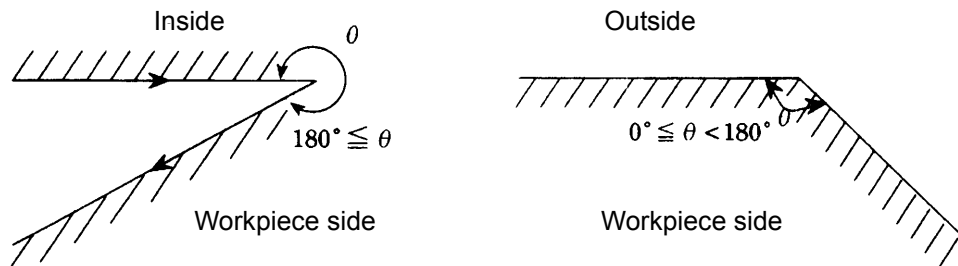
ME6106009

7. Operation at Nose Radius Compensation ON

The program below is described with regard to the G17 plane set when the power is turned ON. The G18 and G19 planes conform to this description.

“Inside” and “Outside” are defined as follows.

The angle of intersection of the move command, which is more than 180 degrees when measured on the workpiece side, “Inside” and the angle, which is between 0 degrees and 180 degrees when measured on the workpiece side, is “Outside”.



ME6106010

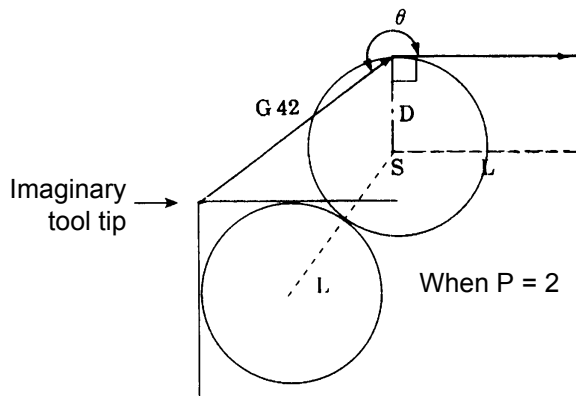
Description of symbols used in the figures

- S : Single block stop point
- L : Linear movement
- C : Circular movement
- T : Tangent of arc
- D : Nose radius compensation amount
- θ : Workpiece side angle
- CP : Cross point. The cross point of the programmed paths or of the tangents for an arc drawn by the tangent moved parallel by the compensation amount.
- : Program command path
- · · → : Tool center path (nose radius center route)
- · · · : Auxiliary line

SECTION 6 NOSE RADIUS COMPENSATION

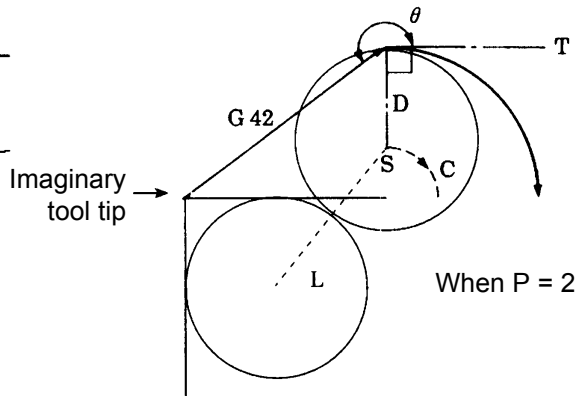
Movement of the tool in nose radius compensation approach(1) Inside cutting $\theta \geq 180$ degrees

a) Straight line - straight line



ME6106011

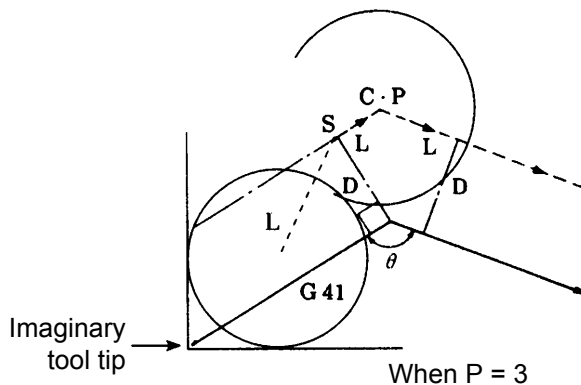
b) Straight line - arc



ME6106012

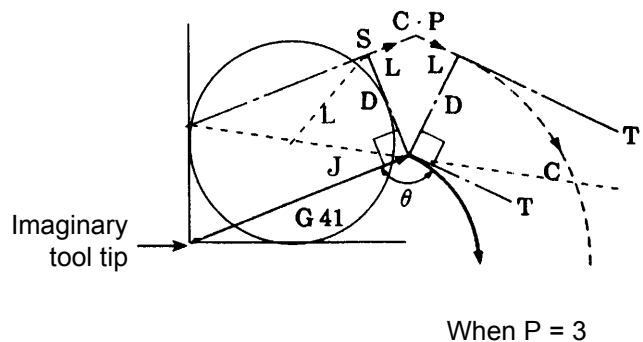
(2) Outside obtuse angle $90 \text{ degrees} \leq \theta < 180$ degrees

a) Straight line - straight line



ME6106013

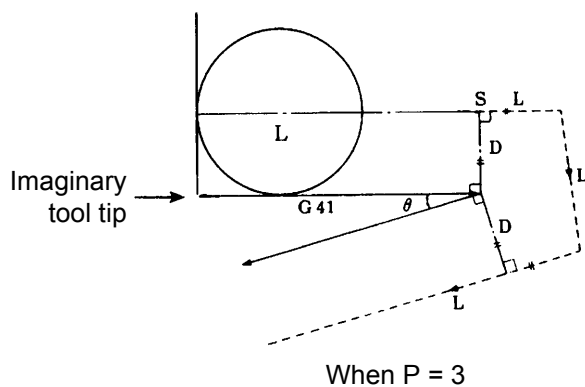
b) Straight line - arc



ME6106014

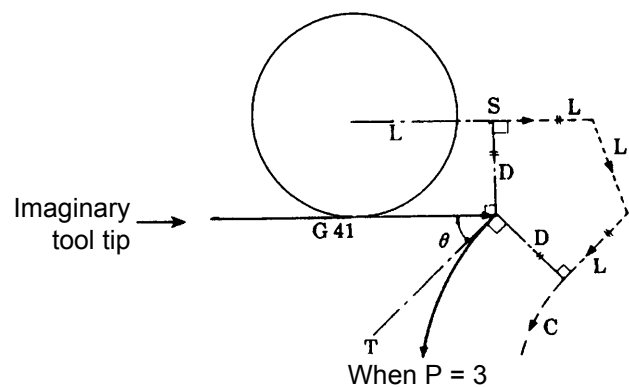
(3) Outside acute angle $\theta < 90$ degrees

a) Straight line - straight line



ME6106015

b) Straight line - arc



ME6106016

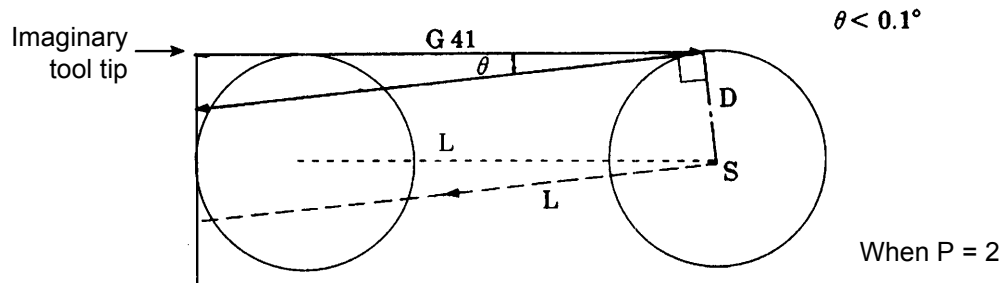
SECTION 6 NOSE RADIUS COMPENSATION

Exceptional case

Outside cutting at an acute angle within 0.1 degrees is considered inside cutting as shown below:

a) Straight line - straight line

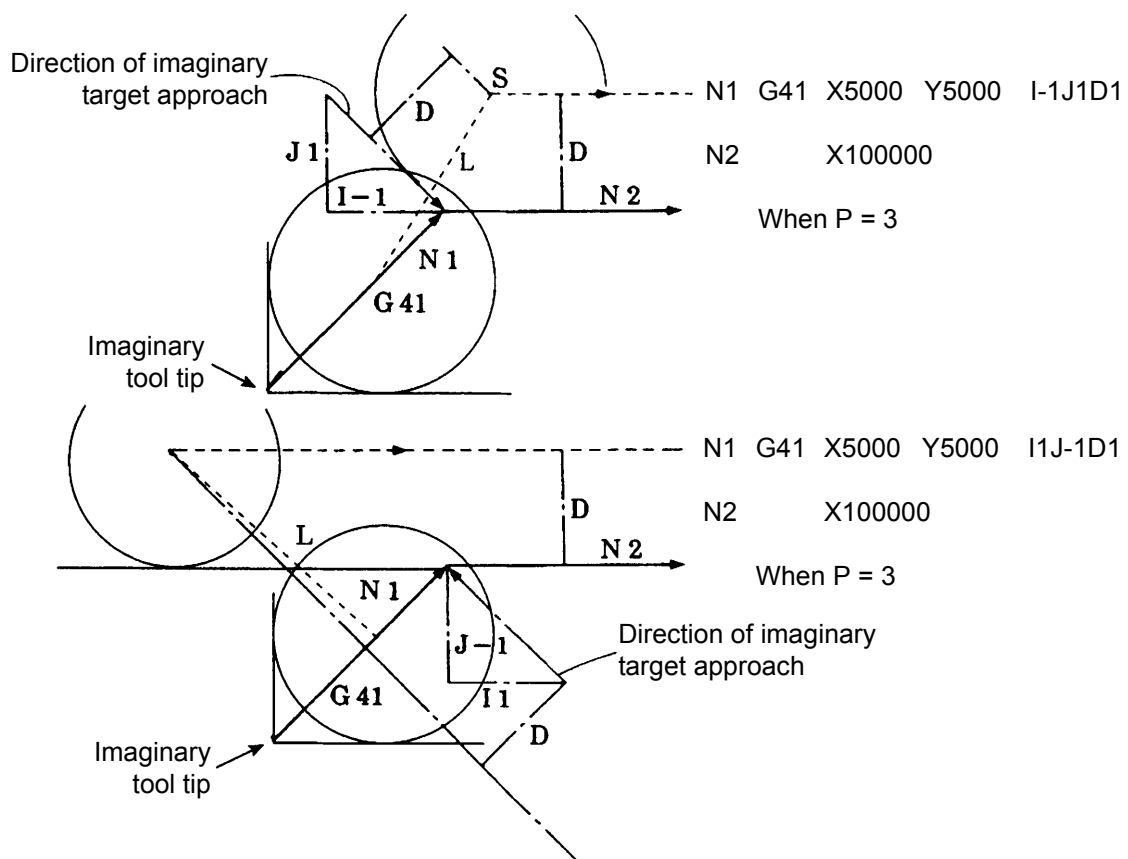
b) Straight line - arc



ME6106017

(4) When there is a direction of imaginary target approach

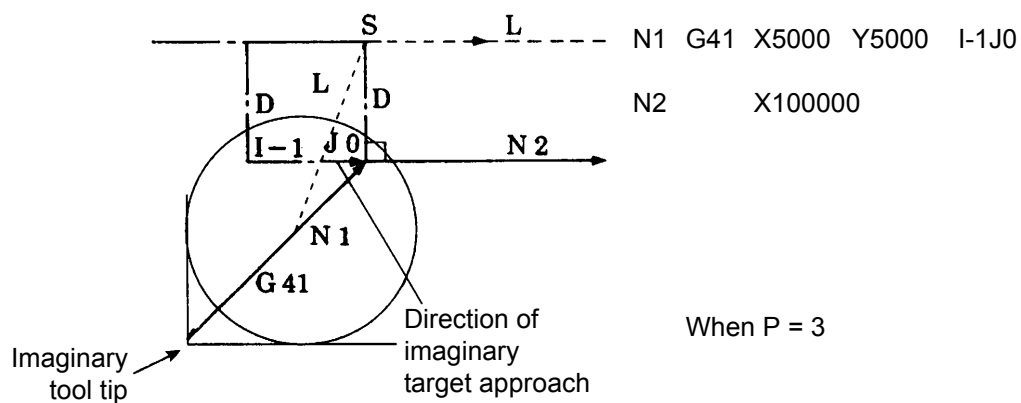
When a direction of imaginary target approach belonging to one of the offset planes I₋, J₋, and K₋ is commanded in the block where nose radius compensation begins, the tool moves from the direction I₋, J₋ (G17) to the commanded point in the block as if it is commanded. Exercise care in this case because the cross point is always to be calculated, irrespective of whether inside cutting or outside cutting is performed.



ME6106018

SECTION 6 NOSE RADIUS COMPENSATION

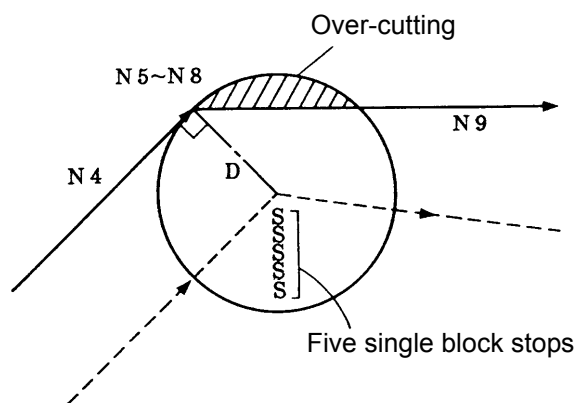
When there is no cross point, the tool will be positioned to the position shifted vertically by the compensation amount from the command value of G41.



ME6106019

8. Tool Movement in Nose Radius Compensation Mode

This section describes the movement of the tool after the machine enters nose radius compensation mode until nose radius compensation mode is canceled. The four commands G00, G01, G02, and G03 are used to specify the nose radius compensation movements. Commands without plane axis movement are permitted in up to three consecutive blocks in nose radius compensation mode. The processing shown below takes place when there is a command without plane axis movement in four or more consecutive blocks or when the plane axis movement is 0 even in a block, resulting in over-cutting or insufficient cutting. Do not give a command in this manner.

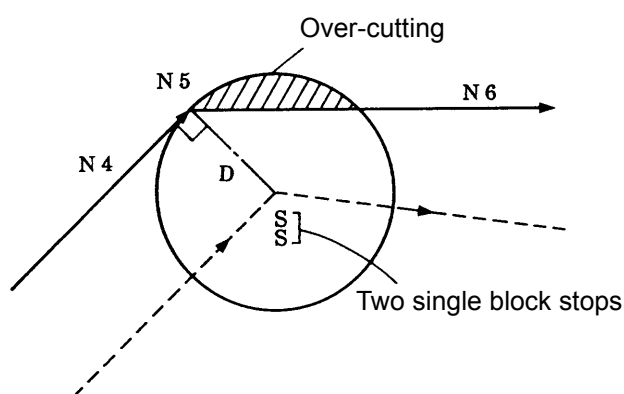


```

:
:
:
:
N4 X5000 Y5000
N5 X5000
N6 F1000
N7 M01
N8 G04 F50
N9 X100000
:
:
:

```

ME6106020



```

:
:
:
:
N4 G91 X5000 Y5000
N5 X0
N6 X5000
:
:
:

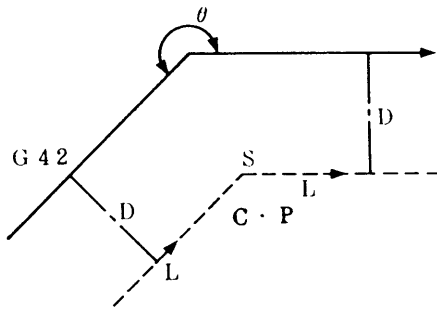
```

ME6106021

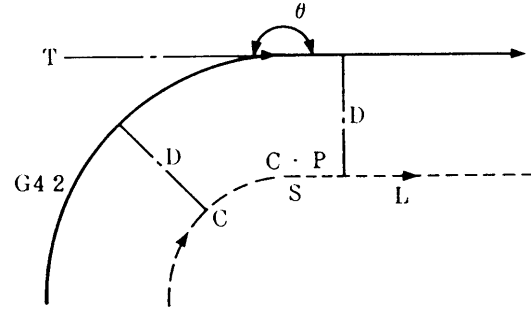
SECTION 6 NOSE RADIUS COMPENSATION

(1) Inside cutting $180 \text{ degrees} \leq \theta$

a) Straight line - straight line



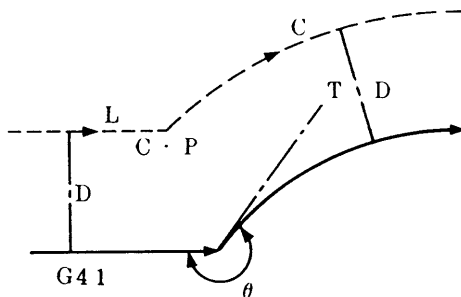
b) Arc - straight line



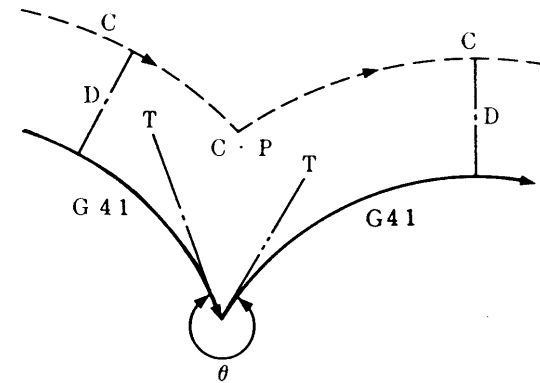
ME6106022

ME6106023

c) Straight line - arc



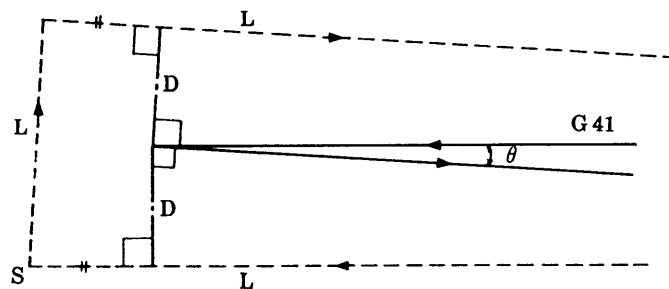
d) Arc - arc



ME6106024

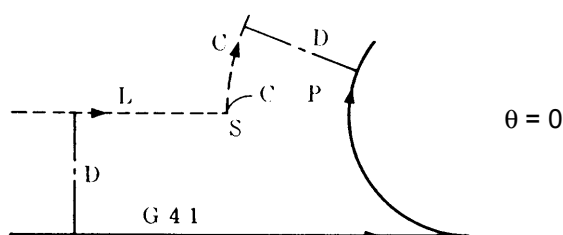
ME6106025

When performing inside cutting at an angle within 0.1 degrees for a straight line-straight line configuration, the actual configuration significantly deviates from the command value with which the cross point is regularly calculated. The same processing as outside acutely angled cutting described later will therefore be performed.

Straight line - straight line $\theta < 0.1 \text{ degrees}$ 

ME6106026

This processing is performed only for a straight line - straight line configuration, and calculate a cross point as regularly in other cases.

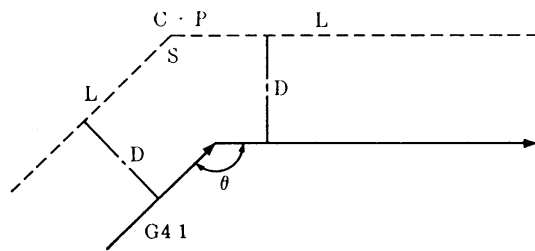


ME6106027

SECTION 6 NOSE RADIUS COMPENSATION

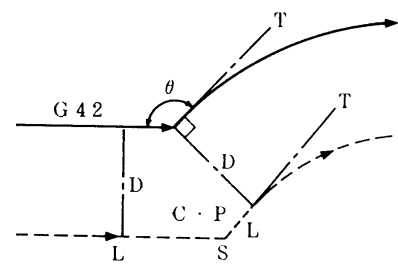
(2) Outside obtusely angled cutting $90 \text{ degrees} \leq \theta < 180 \text{ degrees}$

a) Straight line - straight line



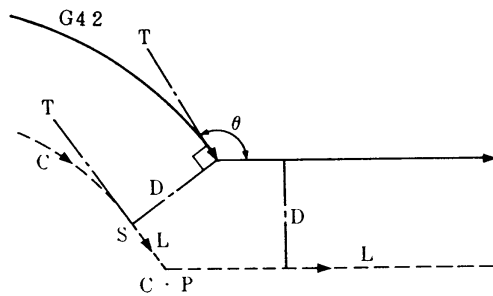
ME6106028

b) Straight line - arc



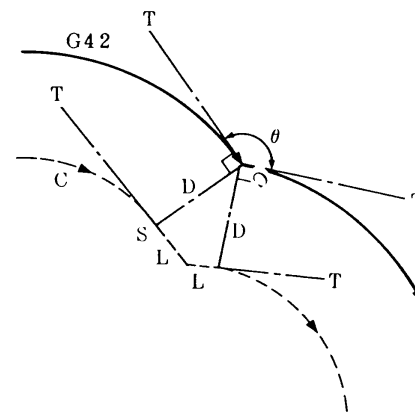
ME6106029

c) Arc - straight line



ME6106030

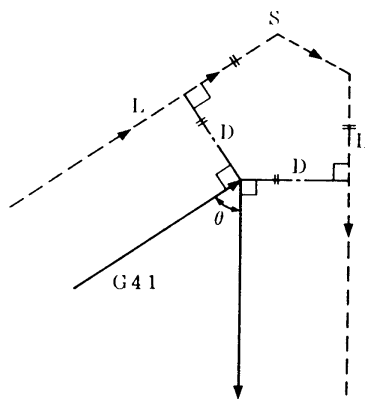
d) Arc - arc



ME6106031

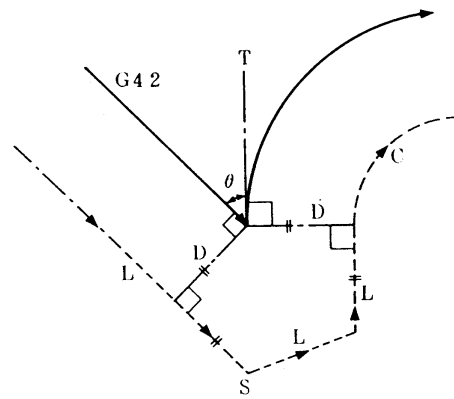
(3) Outside acutely angled cutting $\theta < 90 \text{ degrees}$

a) Straight line - straight line



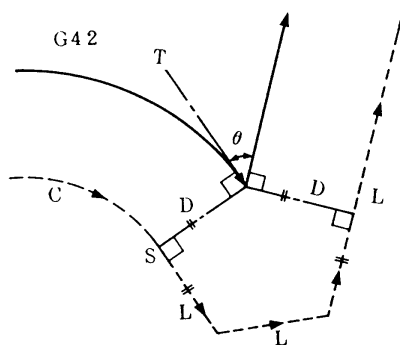
ME6106032

b) Straight line - arc



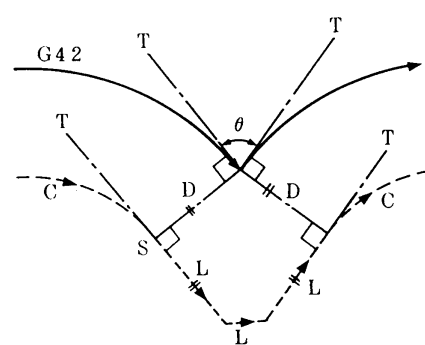
ME6106033

c) Arc - straight line



ME6106034

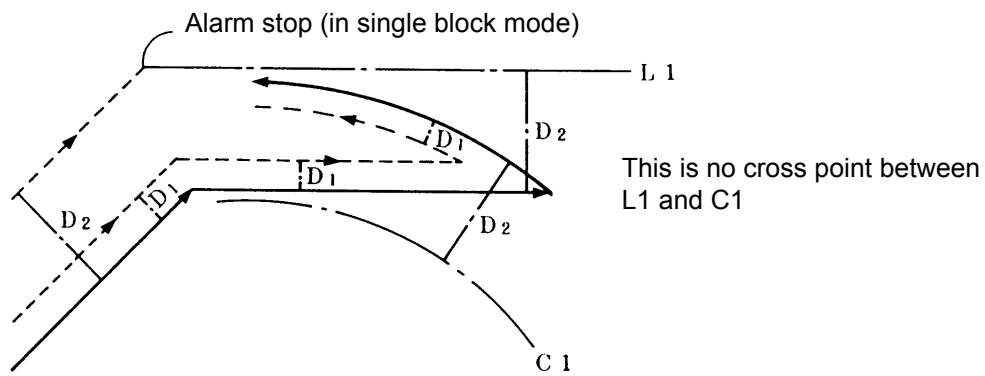
d) Arc - arc



ME6106035

SECTION 6 NOSE RADIUS COMPENSATION

- (4) When a cross point is not calculated in inside cutting



ME6106036

No cross point may exist when the compensation amount is large even though there is a cross point when the compensation amount is small as shown in the above figure. In this case, an alarm will be issued in the preceding block in single block mode or in a block several blocks ahead of the block without a cross point, and the operation will come to a stop.

9. Tool Movement when Nose Radius Compensation is Canceled

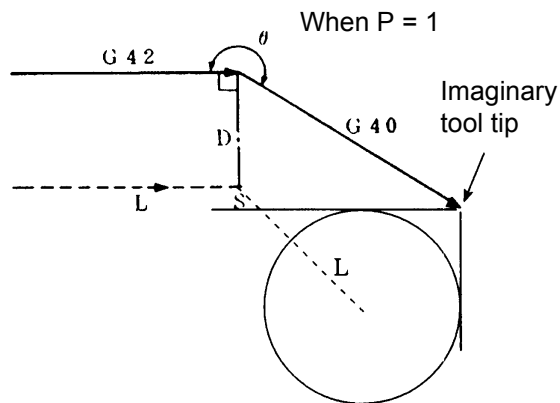
Nose radius compensation cancel mode becomes activated when the following command is issued in nose radius compensation mode:

Command form G40 [G00 (G01) Xp_ Yp_]

Do not use modes other than G00 and G01 for the movement mode for cancellation.

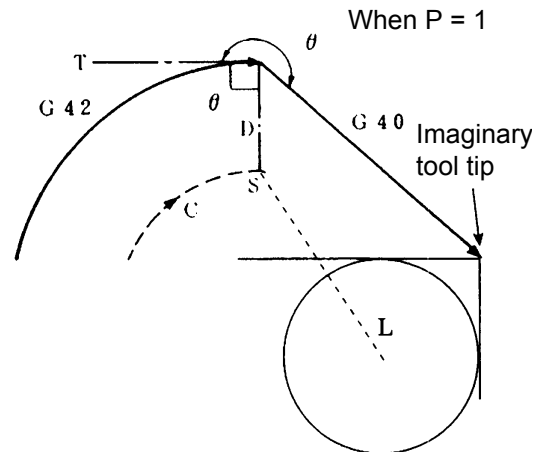
(1) Inside cutting $180 \text{ degrees} \leq \theta$

a) Straight line - straight line



ME6106037

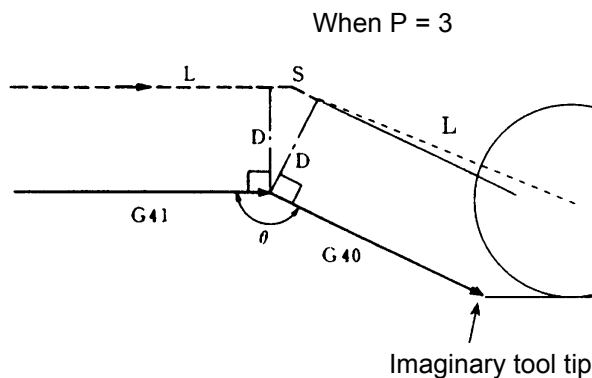
b) Arc - straight line



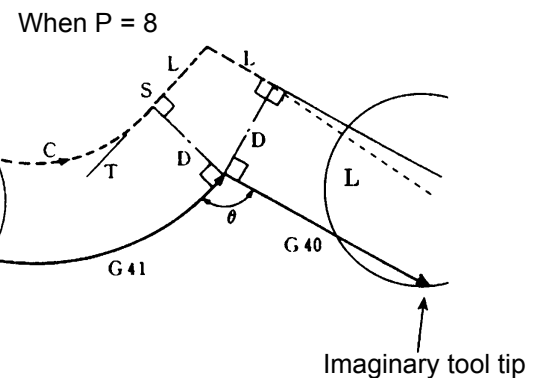
ME6106038

(2) Outside obtusely angled cutting $90 \text{ degrees} \leq \theta < 180 \text{ degrees}$

a) Straight line - straight line



b) Arc - straight line



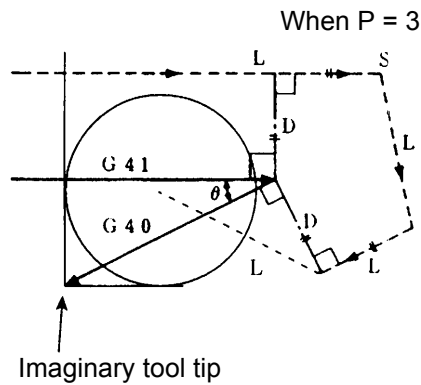
ME6106039

SECTION 6 NOSE RADIUS COMPENSATION

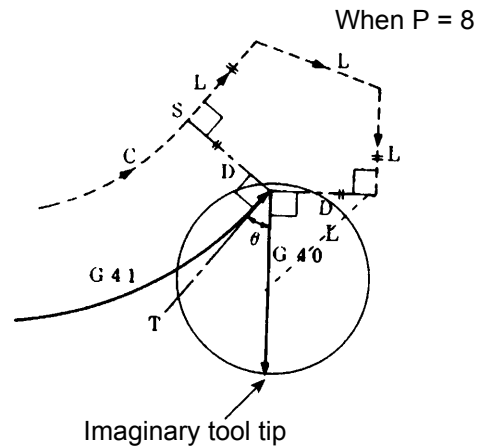
(3) Outside acutely angled cutting $\theta < 90$ degrees

a) Straight line - straight line

b) Arc - straight line



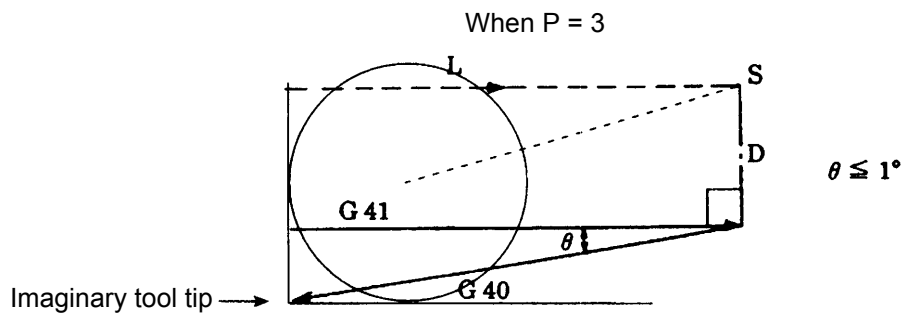
ME6106040



ME6106041

Exceptional case

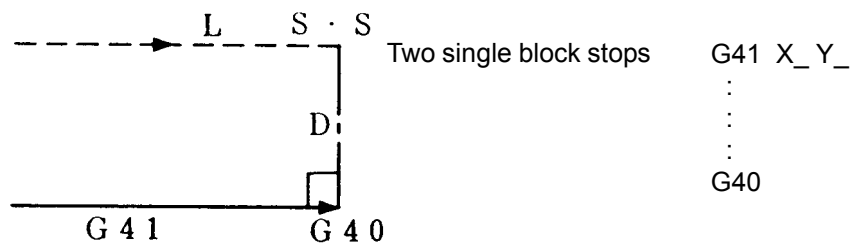
Outside cutting at an acute angle within 0.1 degrees considered inside cutting as shown below:



ME6106042

(4) G40 independent command

- Straight line



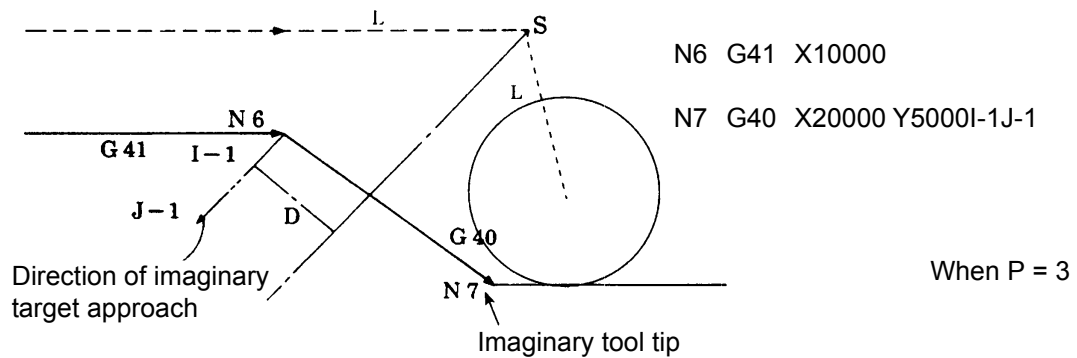
ME6106043

When G40 is independently commanded, the tool will be positioned to the position shifted vertically by the compensation amount from the command value of the preceding block.

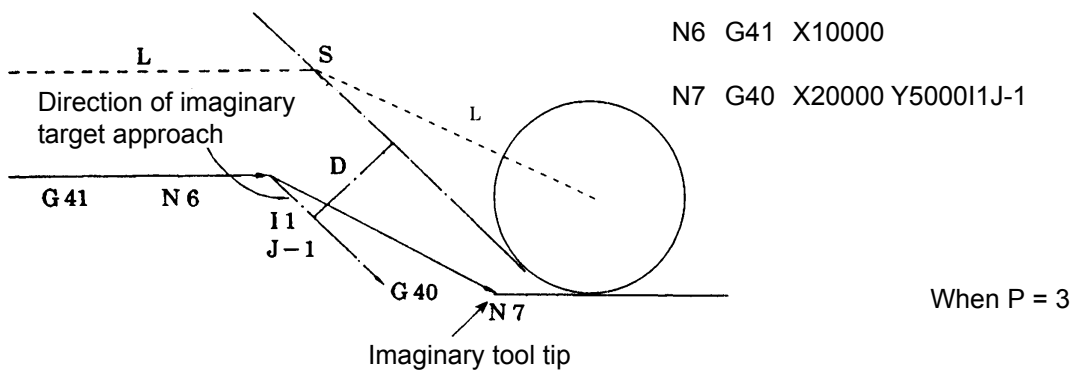
SECTION 6 NOSE RADIUS COMPENSATION

(5) When there is a direction of imaginary target approach

When a direction of imaginary target approach belonging to one of the offset planes $I_$, $J_$, and $K_$ is commanded in the same block as G40, the tool moves in the direction $I_$, $J_$ (G17) from the end point of the preceding block as if it is commanded. Exercise care in this case because the cross point is always to be calculated, irrespective of whether inside cutting or outside cutting is performed.

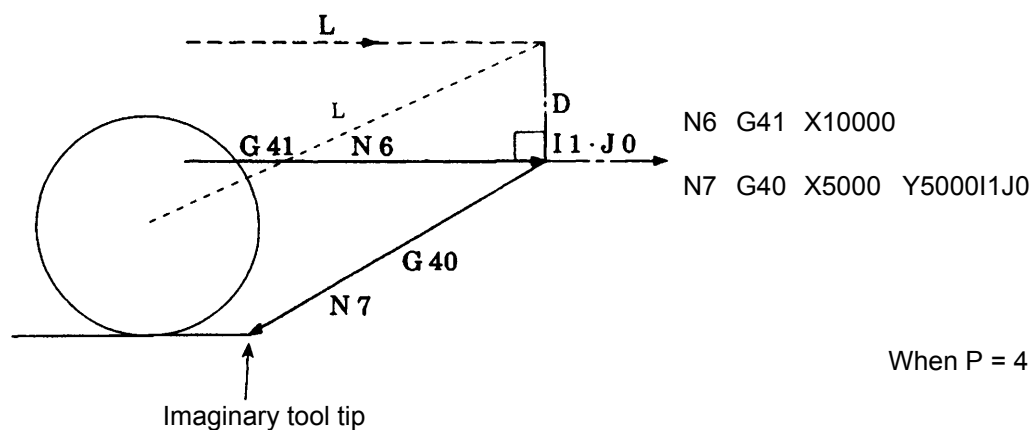


ME6106044



ME6106045

When there is no cross point, the tool will be positioned to the position shifted vertically by the compensation amount from the end point of the preceding block of G40.



ME6106046

10. Changing Compensation Direction in Nose Radius Compensation Mode

The compensation direction can be changed even in compensation mode by executing G41 or G42 or reversing the positive or negative sign of the compensation amount.

Offset amount sign G code	+	-
G41	Offset to left (left side cutting)	Offset to right (right side cutting)
G42	Offset to right (right side cutting)	Offset to left (left side cutting)

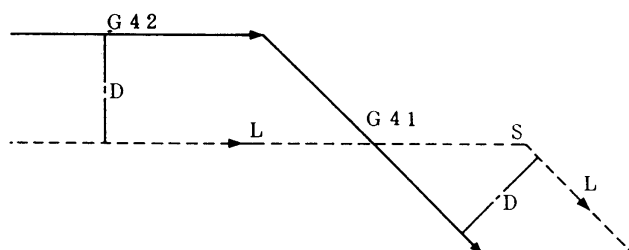
Execution conditions

Mode	Command	Straight line - straight line	Straight line - arc	Arc - straight line	Arc - arc
G41	G41	Invalid (when the offset amount sign is not changed)			
G42	G42				
G41	G42	Executed		An alarm is issued when there is no cross point.	
G42	G41				

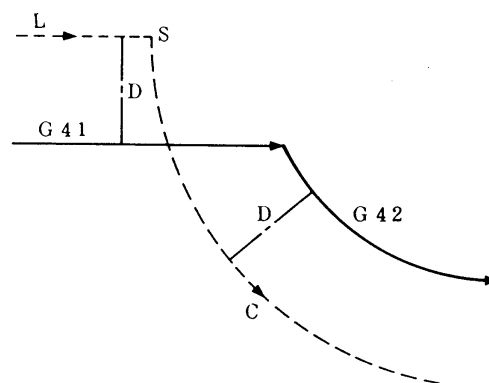
The compensation direction is changeable, irrespective of whether inside cutting or outside cutting is performed. In the following description, the compensation amount is positive.

(1) When there is a cross point

a) Straight line - straight line



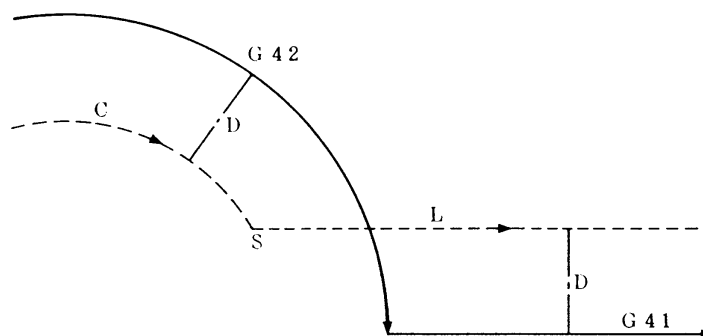
b) Straight line - arc



ME6106047

ME6106048

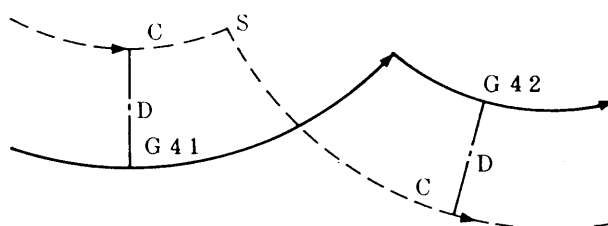
c) Arc - straight line



ME6106049

SECTION 6 NOSE RADIUS COMPENSATION

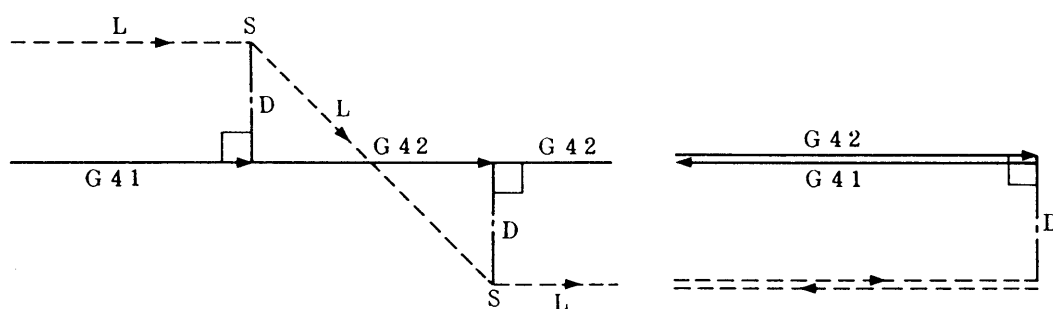
d) Arc - arc



ME6106050

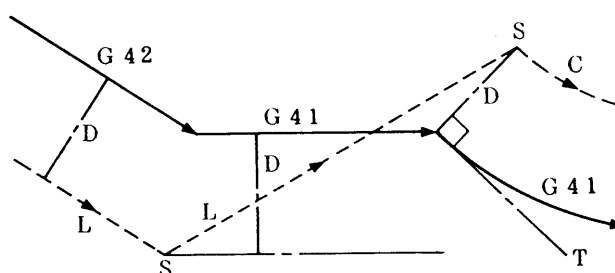
(2) When there is no cross point

a) Straight line - straight line



ME6106051

b) Straight line - arc

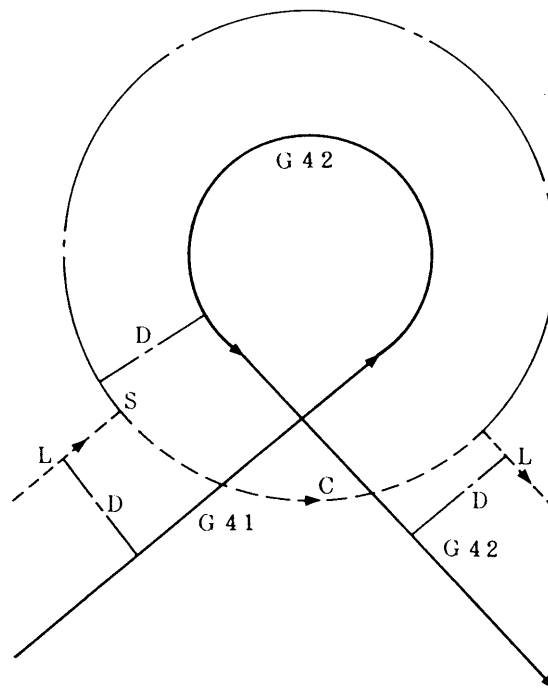


ME6106052

SECTION 6 NOSE RADIUS COMPENSATION

- (3) When an arc forms an overlapping circle

When an arc forms an overlapping circle as a result of the change of the compensation direction, the tool passes along a short arc as shown below. In this case, give a command by dividing the arc.



ME6106053

11. Precautions on Nose Radius Compensation

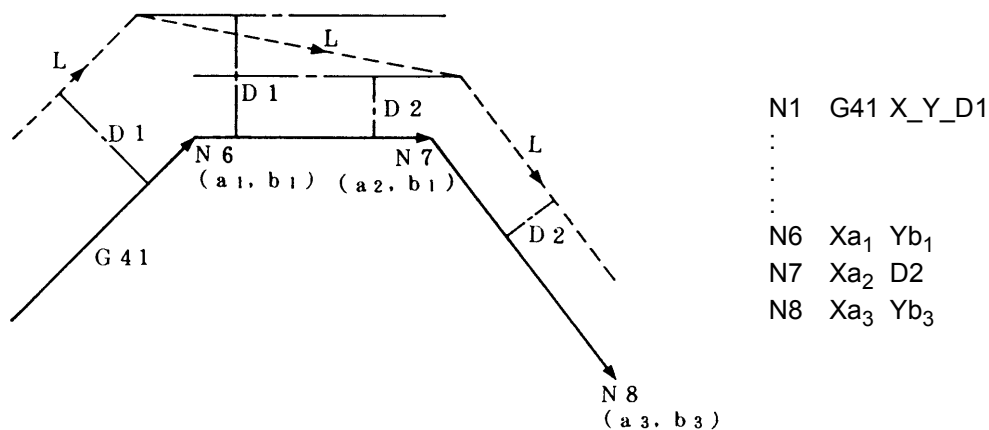
(1) Nose radius compensation amount command

Command a number for the compensation amount with the D command. Command it in the same block as G41 and G42, but the D command previously commanded will be used when it is not omitted.

The number can be set between D00 and D50 as a standard setting or between D00 and D100, D200, and D300 as an optional setting. The compensation amount D00 is 0. Set the compensation amount in tool data setting mode.

(2) Changing the compensation amount

When the compensation amount is changed in compensation mode, it will become valid at the end of the block.



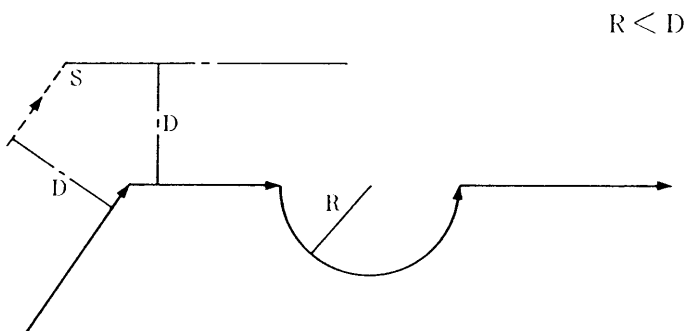
ME6106054

(3) Actual position display

The imaginary cutter tip is displayed as the actual position.

(4) When cutting the inside of an arc that is smaller than the nose radius

Alarm stop (in single block mode)



ME6106055

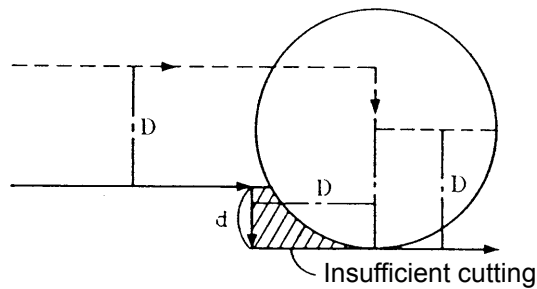
In this case, cutting is impossible, and an alarm stop occurs. However, the tool stops in a block several blocks ahead of the block when single block mode is not selected, or in the second block from the block when single block mode is selected.

SECTION 6 NOSE RADIUS COMPENSATION

(5) Insufficient cutting

This problem occurs when a step smaller than the nose radius is machined.

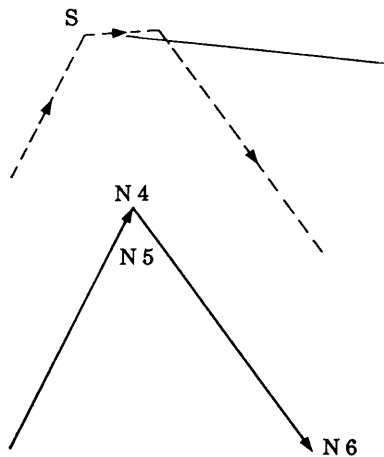
$$d < D$$



ME6106056

(6) Precautions when cutting a corner

- a) In outside cutting, the corner is cut along a polygonal tool path. The movement mode and the feedrate at the corner conform to those commanded in the next block. However, the tool moves as if G01 is selected when the next mode is G02 or G03.



This movement follows the command of N5 (F800)

N4 X_Y_ F500

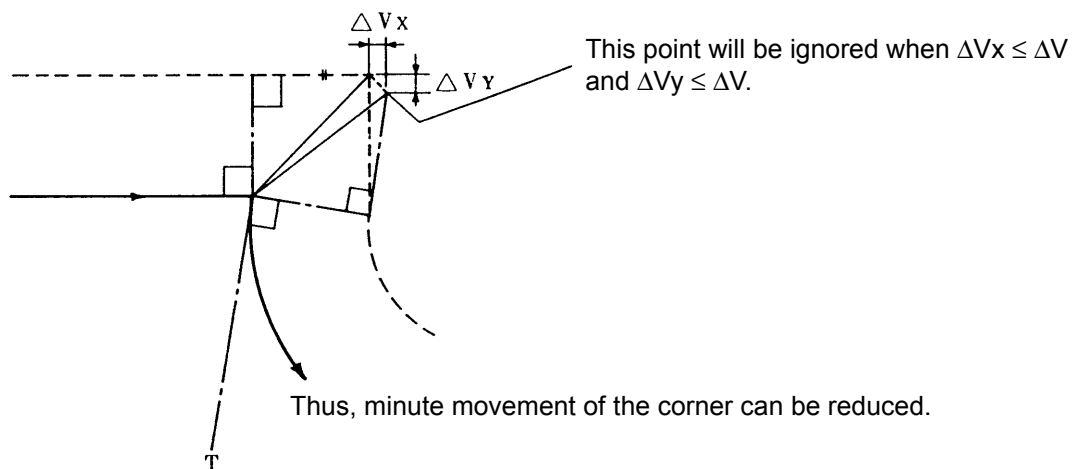
N5 Z_ F800

N6 X_Y_

Z-axis movement is carried out at the points.

ME6106057

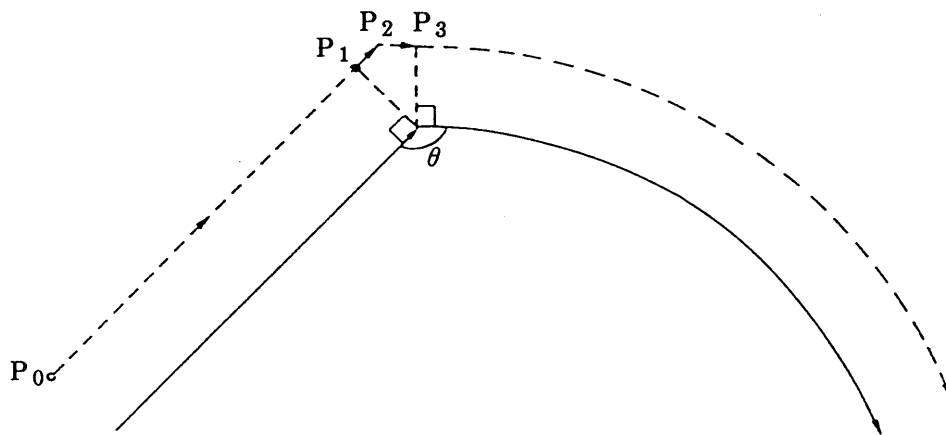
- b) When the movement of the corner is very small as shown in the figure below, the rear movement point will be ignored when $\Delta V_x \leq \Delta V$ and $\Delta V_y \leq \Delta V$. Set the value of this ΔV with NC optional parameter long word No. 4. The initial value is 0.050 mm.



ME6106058

SECTION 6 NOSE RADIUS COMPENSATION

c) This processing does not take place when the next block is forms a full circle.



ME6106059

In the above figure:

$P_0 - P_1 - P_2$ Straight line

$P_2 - P_3$ Straight line

From P_3 Full circle

However, if the movement from P_2 to P_3 is neglected in this program in minute movement processing:

$P_0 - P_1 - P_2$ Straight line

$P_2 - P_3$ Arc

The program neglects the full circle will be neglected and forms an arc from P_2 to P_3 .

(7) Interference

[Supplement]

Interference refers to excessive cutting of the tool into the workpiece. The NC checks it in advance. The NC judges that excessive cutting (interference) will occur when: the difference between the movement direction in the program and the movement direction resulting from radius compensation is between 90 degrees and 270 degrees.

As a result, cutting may be considered to be interference in this check even if the tool does not interfere with the workpiece, or interference may not be detected in this check.

When a corner is cut along a polygonal tool path, an interference check is performed at the last point of the last corner and at the first point of the next corner, assuming that the points are P_1, P_2, P_3 , and P_4 and P_5, P_6, P_7 , and P_8 (up to four points are formed at a corner). If interference is detected at a point, the point will be ignored and the next points will be checked with each other. When no interference is detected halfway, no interference check will be performed at the subsequent points. The movement mode in this case is straight line movement. If the block is an arc, the tool moves in G1 mode.

If the tool interferes with the workpiece after all the points have been checked, an interference alarm occurs, and this last point is not ignored. Consequently, over-cutting may be performed in single block mode.

Representative examples are given on the pages that follow to describe the above.

-
- The diagram illustrates the geometry of a particle's path in a magnetic field. It features a large solid circle and a smaller dashed circle centered at the origin. Several points are labeled: P 1, P 2, P 7, P 8, P 4, P 5, P 3, and P 6. Arrows indicate the direction of motion along the paths. Labels C, T, N, L, S, D, and arrows represent various parameters and directions.

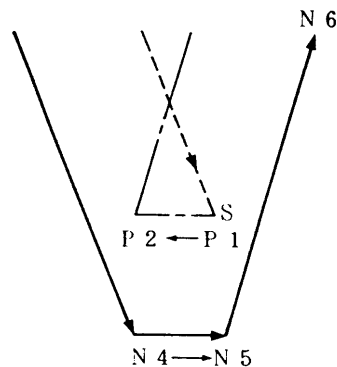
ME6106060

-

In the above figure, the movement direction of N4 - N5 and those of P4 - P5, P3 - P6, and P2 - P7 are checked and ignored. However, the tool makes a straight line movement (G01) to P1 - P8 because the direction of P1 - P8 is correct.

SECTION 6 NOSE RADIUS COMPENSATION

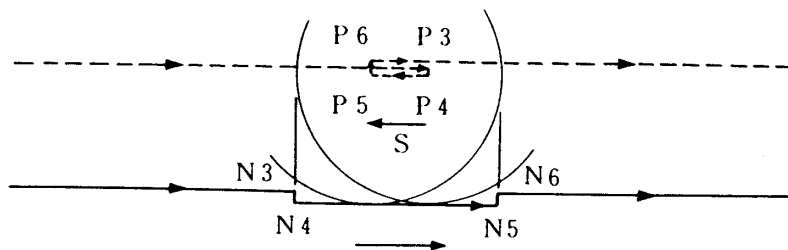
- c) When an alarm occurs in an interference check



ME6106062

In the above figure, each corner has only one point, and P1 is not ignored. An alarm stop occurs after the tool moves from P1 in single block mode or in a block several blocks ahead of the block when the mode is not single block mode.

- d) Non-interference which is regarded as interference



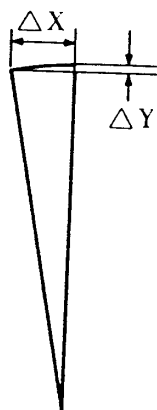
ME6106063

Although the tool does not interfere with the workpiece when N4 - N5 is smaller than the tool diameter, an interference alarm occurs because the movement direction of N4 - N5 is opposite to that of P4 - P5.

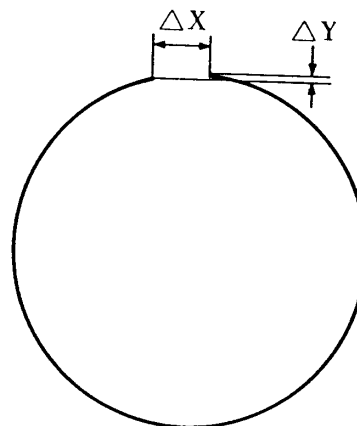
- e) Minute arc and quasi-full circle

A minute arc is defined as an arc whose horizontal and vertical distances from the starting point to the end point are smaller than NC optional parameter long word No. 9. A quasi-full circle is defined as an arc whose horizontal and vertical distances at the break are smaller than NC optional parameter long word No. 9.

Minute arc



Quasi-full circle



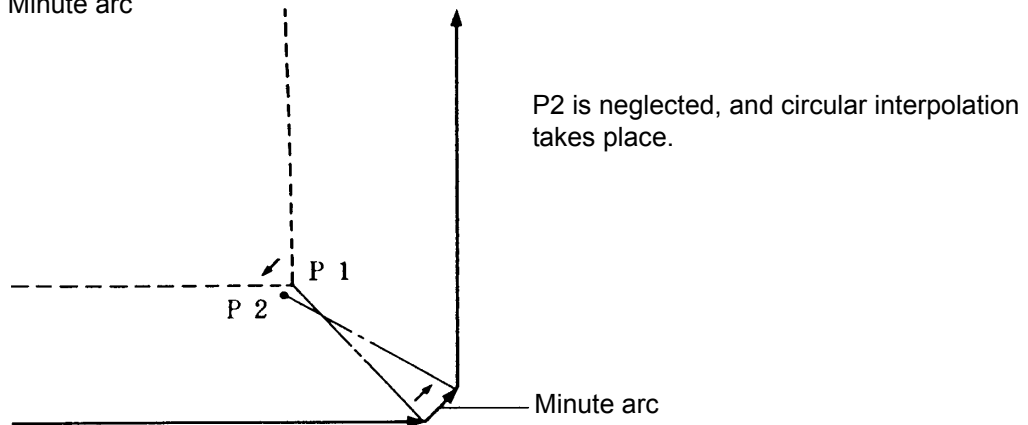
ME6106064

When $\Delta V_x \leq \Delta V$ and $\Delta V_y \leq \Delta V$, the arcs are considered as shown on the preceding page (however, $\Delta V = \text{NC optional parameter long word No. 9}$).

SECTION 6 NOSE RADIUS COMPENSATION

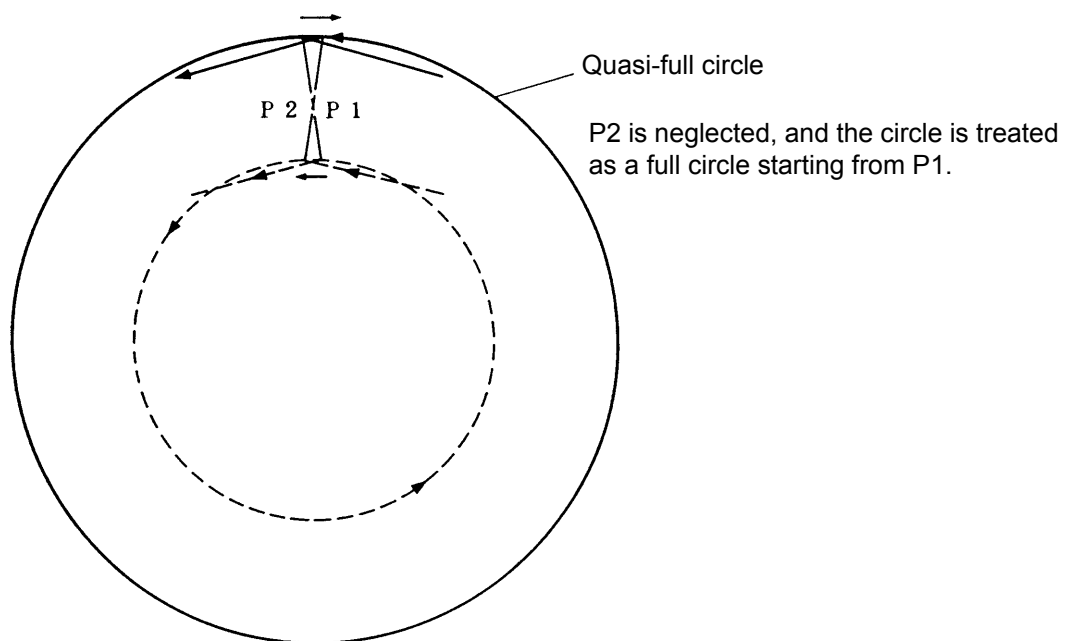
In the case of these arcs, an interference check is specially treated. Interference may be detected although the tool does not interfere with the workpiece due to an operational error inside. With regard to minute arcs and quasi-full circles, an interference, which is detected in an interference check, is not treated as an interference but as an operational error. Additionally, the end point of a minute arc is neglected and treated as a point, thus skipping the circular movement. The end of a quasi-full circle is also neglected, and the circle is treated as a full circle from the starting point.

Minute arc



ME6106065

Quasi-full circle

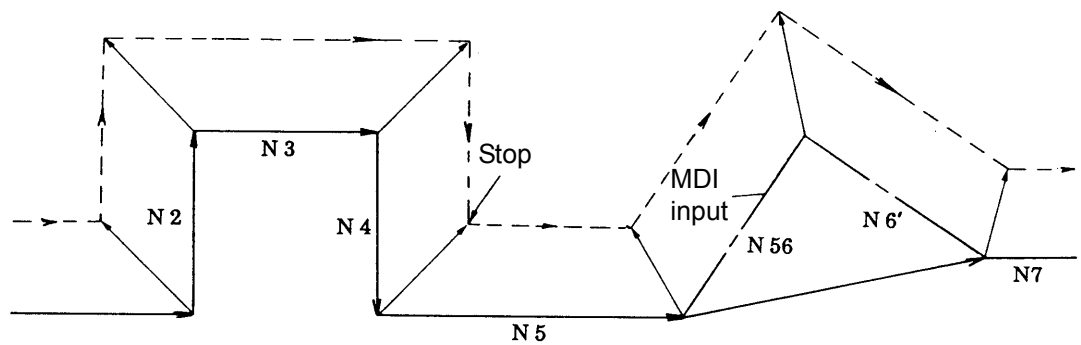


ME6106066

SECTION 6 NOSE RADIUS COMPENSATION

(8) To commands input from the MDI

- a) When the mode is changed from MDI mode to nose radius compensation mode or when nose radius compensation mode is selected, a block with movement is not executed immediately even if it is commanded. It will be executed until the next block with movement is commanded or four consecutive blocks with movement are commanded.
- b) In single block OFF during automatic operation, the operation is executed before the block, which is read into the buffer memory (the line with ">>" in the program display), when the mode is changed to MDI mode and comes to a stop. The command input from the MDI is read into the next block with ">>", and nose radius compensation is executed.



ME6106067

- c) If the mode is changed to MDI mode during execution of block N1, the operation is executed to N4 and comes to a stop when the program display is as shown in Fig.1. The program display changes to that shown in Fig.2.

```

      ⋮
      ⋮
↑ N1  X10
  N2      Y30
  N3  X30
  N4      Y-30
» N5  X50
  N6  X80 Y10
  N7  X100
      ⋮
      ⋮

```

Fig.1

```

      ⋮
      ⋮
  N1  X10
  N2      Y30
  N3  X30
↑ N4      Y-30
» N5  X50
  N6  X80 Y10
  N7  X100
      ⋮
      ⋮

```

Fig.2

ME6106068

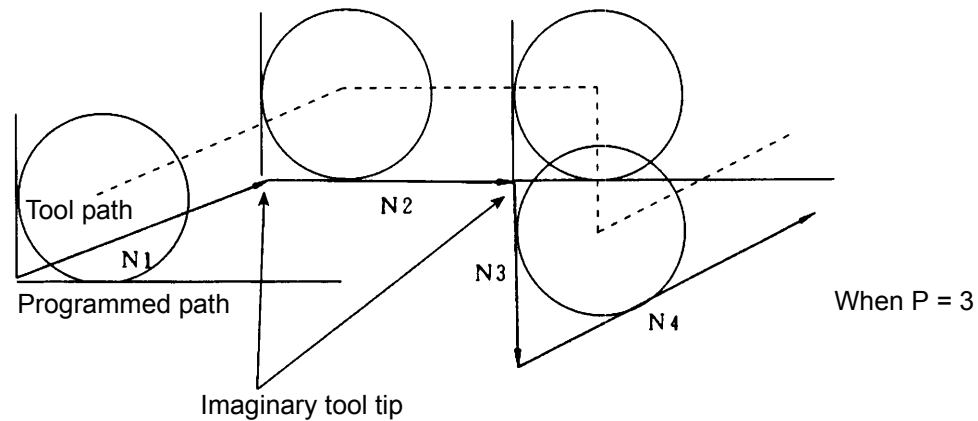
In this case, when block N56 is input from the MDI and starts, it comes to a stop after N5 is executed. N56, N6', and N7 will be executed in sequence by changing the mode to automatic operation mode again and pressing the CYCLE START key.

SECTION 6 NOSE RADIUS COMPENSATION

(9) Tool movement when the nose radius compensation offset amount is 0

a) Startup

When G41 or G42 is commanded in cancel mode, the mode is changed to offset mode, and the startup operation begins at an offset amount of 0, but nose radius compensation does not take place. However, the same operation as described in "11. (2) Changing the compensation amount" is performed when the offset number is changed to another number whose offset amount is not 0 in offset mode.

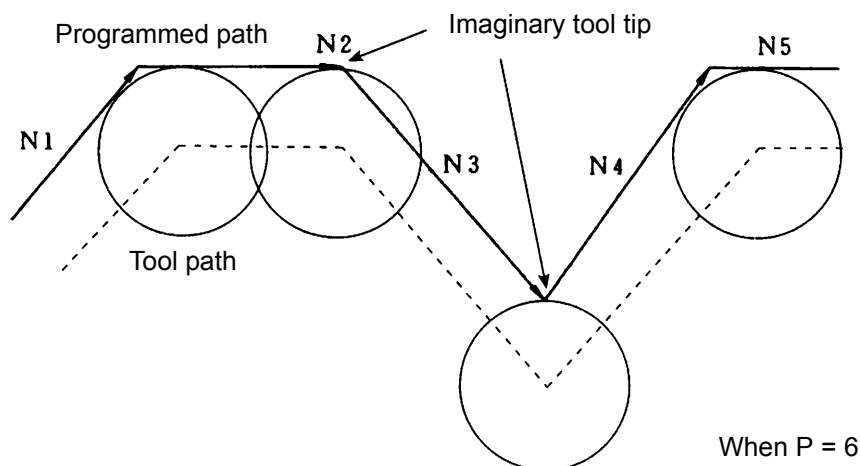


ME6106069

N1	G41	X_	Y_	D01	Offset amount of D01 = 0
N2		X_			
N3			Y_	D02	Offset amount of D02 ≠ 0
N4		X_	Y_		

b) In offset mode

The offset operation is not canceled or the mode is not changed to cancel mode even if the offset number is changed to another number whose offset amount is 0 in offset mode. The same operation as described in "11. (2) Changing the compensation amount" is performed. The same operation takes place even if the offset number is changed again to another number whose offset amount is not 0.



ME6106070

N1	G41	X_	Y_		Offset amount of D01 = 0
N2		X_		D01	
N3		X_	Y_		Offset amount of D02 ≠ 0
N4		X_	Y_	D02	
N3		X_			

LIST OF PUBLICATIONS

Publication No.	Date	Edition
6233-E	Jan. 2013	1st
6233-E-R1	Apr. 2017	2nd

This manual may be at variance with the actual product due to specification or design changes.

Please also note that specifications are subject to change without notice.

If you require clarification or further explanation of any point in this manual, please contact your OKUMA representative.