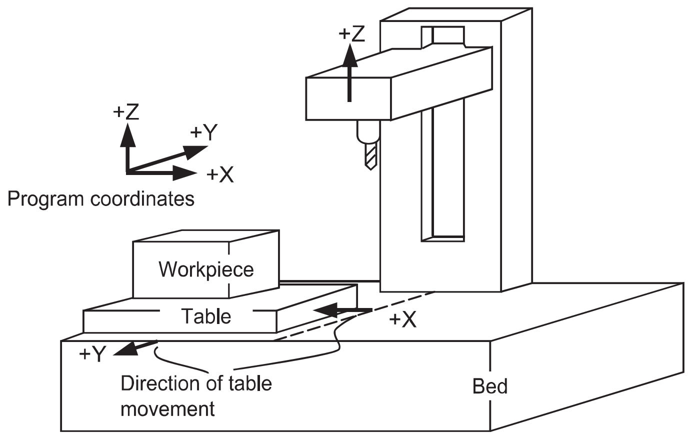
1.1 Coordinate Words and Control Axes

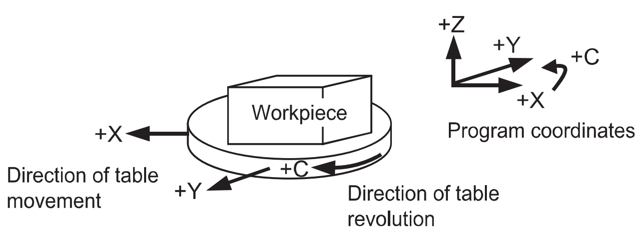
Function and purpose

The number of control axes is set to "3" in the standard specifications; however, up to eight axes can be controlled if an additional axis is added. To specify each machining direction, use alphabetical coordinate words that are predefined appropriately.

X-Y table



X-Y and rotating table



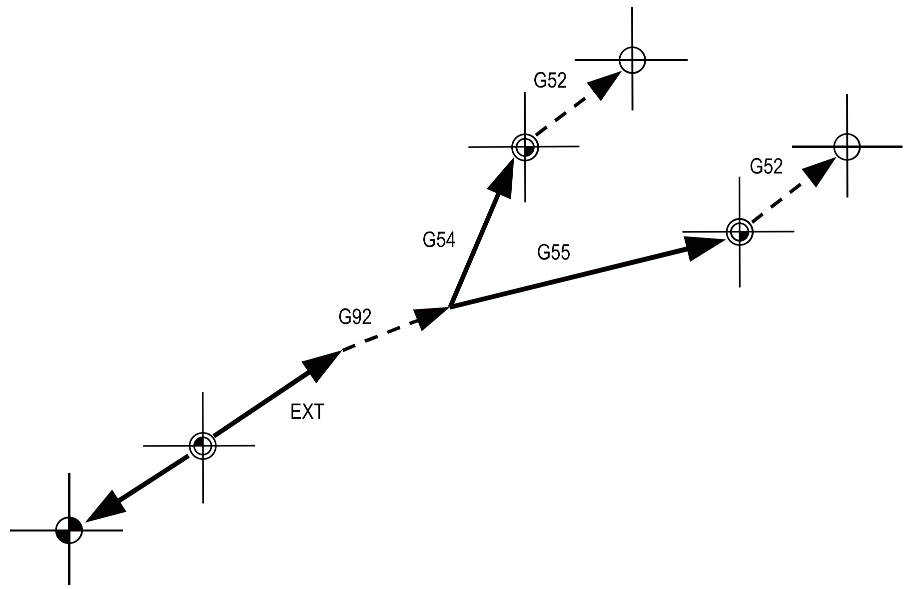
1.2 Coordinate Systems and Coordinate Zero Point Symbols

Reference position:
A specific position to establish coordinate systems and change tools

Basic machine coordinate zero point:
A position specific to machine

Workpiece coordinate zero points (G54 to G59)
A coordinate zero point used for workpiece machining

The basic machine coordinate system is the coordinate system that expresses the position (tool change position, stroke end position, etc.) that is specific to the machine.
Workpiece coordinate systems are used for workpiece machining.
Upon completion of the dog-type reference position return, the parameters are referred and the basic machine coordinate system and workpiece coordinate systems (G54 to G59) are automatically set.
The offset of the basic machine coordinate zero point and reference position is set by a parameter. (Normally, set by MTB)
Workpiece coordinate systems can be set with coordinate systems setting functions, workpiece coordinate offset measurement (additional specification), and etc.



Reference position

Basic machine coordinate zero point

Workpiece coordinate zero points

Local coordinate zero point

Offset set by a parameter

Offset set by a program
("0" is set when turning the power ON)

G52 Local coordinate system offset (*1)
G54 Workpiece coordinate (G54) system offset (*1)
G55 Workpiece coordinate (G55) system offset
G92 G92 Coordinate system shift
EXT External workpiece coordinate offset

(*1) G52 offset is independently possessed by G 54 to G59 respectively.

The local coordinate systems (G52) are valid on the coordinate systems designated by workpiece coordinate systems 1 to 6.
Using the G92 command, the basic machine coordinate system can be shifted and made into a hypothetical machine coordinate system. At the same time, workpiece coordinate systems 1 to 6 are also shifted.

6.1 Positioning (Rapid Traverse); G00

Function and purpose

This command is accompanied by coordinate words and performs high-speed positioning of a tool, from the present point (start point) to the end point specified by the coordinate words.

Command format

Positioning (Rapid Traverse)

G00 X__ Y__ Z__α__ ,I__ ,F__;

X, Y, Z, α	Coordinate values. (α is the additional axis.) An absolute position or incremental position is indicated based on the state of G90/G91 at that time.
,I	In-position width. (1 to 999999) This address is valid only in the commanded block. A block that does not contain this address will follow the parameter "#1193 inpos" settings. For details, refer to "7.12 Deceleration Check".
,F	Specifies the rapid traverse rate of the movement initiated by a G00 command, the movement in the G00 mode, and the movement during the fixed cycle for drilling. The range is equal to the range of the feed per minute F command (mm/min, inch/min) in the G01 mode. Switching inch/mm is invalid for rotary axes. For details, refer to "7.1.2 G00 Feedrate Command (,F Command)".

The command addresses are valid for all additional axes.

Detailed description

(1) The rapid traverse speed varies depending on the MTB specifications (parameter "#2001 rapid").
When the "G00 feedrate designation (,F command)" function is enabled and an ",F" command is included in the same block as for the G00 command, positioning is carried out at the feedrate specified by the ",F" command. If this function is invalid or an ",F" command is not designated, positioning is carried out at the feedrate specified in parameter "#2001 rapid".

(2) G00 command belongs to the 01 group and is modal. When G00 command is successively issued, the following blocks can be specified only by the coordinate words.

(3) In the G00 mode, acceleration and deceleration are always carried out at the start point and end point of the block. Before advancing to the next block, a commanded deceleration or an in-position check is conducted at the end point to confirm that the movement is completed for all the moving axes in each part system.

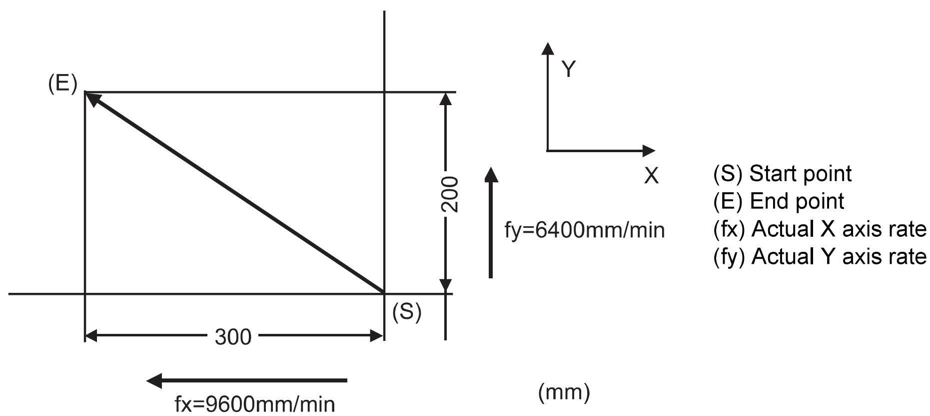
(4) G functions (G72 to G89) in the 09 group are canceled (G80) by the G00 command.

> ⚠ **CAUTION**
>
> ⚠ **The commands with "no value after G" will be handled as "G00".**

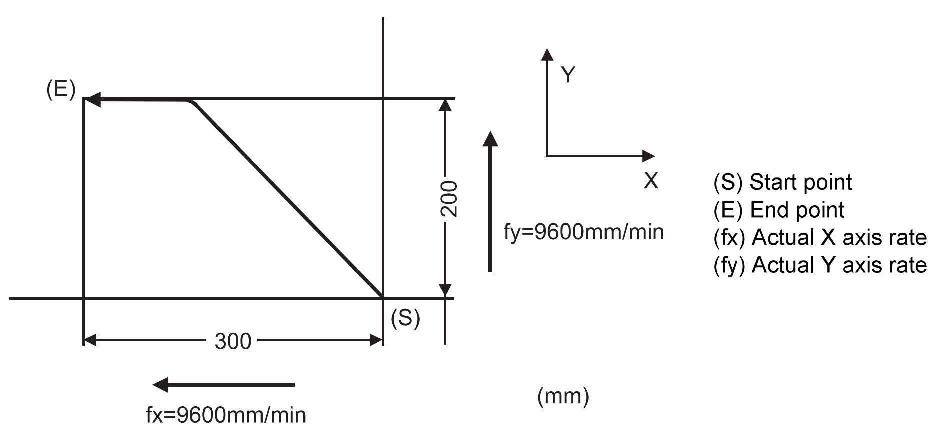
Tool path

Whether the tool moves along a linear or non-linear path varies depending on the MTB specifications (parameter "#1086 G0Intp").
The positioning time does not change according to the path.

(1) Linear path (When parameter "#1086 G0Intp" is set to "0")
In the positioning process, a tool follows the shortest path that connects the start point and the end point. The positioning speed is automatically calculated so that the shortest distribution time is obtained in order that the commanded speeds for each axis do not exceed the rapid traverse rate.
When, for instance, the X axis and Y axis rapid traverse rates are both 9600 mm/min and when programmed as follows, the tool will follow the path shown in the figure below.
G91 G00 X-300000 Y200000; (With an input setting unit of 0.001 mm)



(2) Non-linear path (When parameter "#1086 G0Intp" is set to "1")
In positioning, the tool will move along the path from the start point to the end point at the rapid traverse rate of each axis.
When, for instance, the X axis and Y axis rapid traverse rates are both 9600 mm/min and when programmed as follows, the tool will follow the path shown in the figure below.
G91 G00 X-300000 Y200000; (With an input setting unit of 0.001 mm)



6.2 Linear Interpolation; G01

Function and purpose

This command is accompanied by coordinate words and a feedrate command. It makes the tool move (interpolate) linearly from its current position to the end point specified by the coordinate words at the speed specified by address F. In this case, the feedrate specified by address F always acts as a linear speed in the tool nose center advance direction.

Command format

Linear interpolation

G01 X__ Y__ Z__ α__ F__ ,I__ ;

X,Y,Z,α	Coordinate values. (α is the additional axis.) An absolute position or incremental position is indicated based on the state of G90/G91 at that time.
F	Feedrate (mm/min or °/min)
,I	In-position width. (1 to 999999) This address is valid only in the commanded block. A block that does not contain this address will follow the parameter "#1193 inpos" settings.

Detailed description

(1) G01 command is a modal command in the 01 group. When G01 command is successively issued, the following blocks can be specified only by the coordinate words. If there is no command, a program error (P62) will occur.

(2) The feedrate for a rotary axis is commanded by °/min (decimal point position unit). (F300=300°/min)

(3) The G functions (G72 to G89) in the 09 group are cancelled (G80) by the G01 command.

Programmable in-position width command for linear interpolation

This command commands the in-position width for the linear interpolation command from the machining program.

G01 X_ Y_ Z_ F_ ,I_ ;	
X,Y,Z	Linear interpolation coordinate value of each axis
F	Feedrate
,I	In-position width

The commanded in-position width is valid in the linear interpolation command only when carrying out deceleration check.

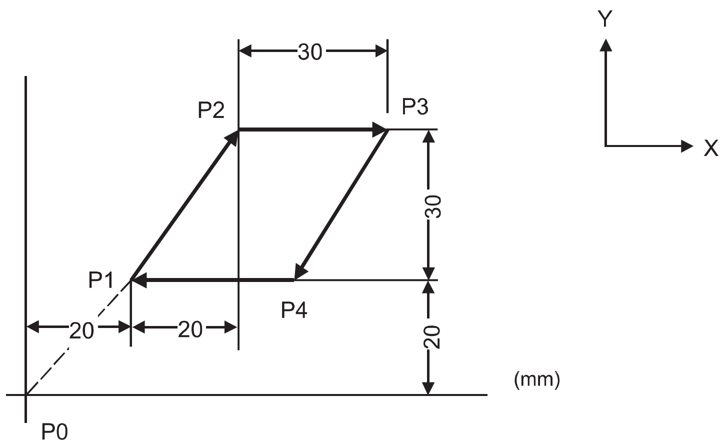
- When the error detection switch is ON.
- When G09 (exact stop check) is commanded in the same block.
- When G61 (exact stop check mode) is selected.

Note

(1) Refer to section "6.1 Positioning (Rapid Traverse); G00" for details on the in-position check operation.

Program example

(Example) Cutting in the sequence of P1 -> P2 -> P3 -> P4 -> P1 at 300mm/min feedrate.
However, P0 -> P1 is for tool positioning.



G91 G00 X20. Y20. ;	P0 -> P1
G01 X20. Y30. F300 ;	P1 -> P2
X30. ;	P2 -> P3
X-20. Y-30. ;	P3 -> P4
X-30. ;	P4 -> P1

15.1 Corner Chamfering I/Corner Rounding I

Function and purpose

Chamfering at any angle or corner rounding is performed automatically by adding ",C_" or ",R_" to the end of the block to be commanded first among those command blocks which shape the corner with lines only.

15.1.1 Corner Chamfering I ; G01 X_ Y_ ,C

Function and purpose

This chamfers a corner by connecting the both side of the hypothetical corner which would appear as if chamfering is not performed, by the amount commanded by ",C_".

Command format

```
N100 G01 X__ Y__ ,C__ ;
N200 G01 X__ Y__ ;
```

,C	Length up to chamfering starting point or end point from hypothetical corner

Corner chamfering is performed at the point where N100 and N200 intersect.

Detailed description

(1) The start point of the block following the corner chamfering is the hypothetical corner intersection point.

(2) If there are multiple or duplicate corner chamfering commands in a same block, the last command will be valid.

(3) When both the corner chamfer and corner rounding commands exist in the same block, the latter command is valid.

(4) Tool compensation is calculated for the shape which has already been subjected to corner chamfering.

(5) When the block following a command with corner chamfering does not contain a linear command, a corner chamfering/corner rounding II command will be executed.

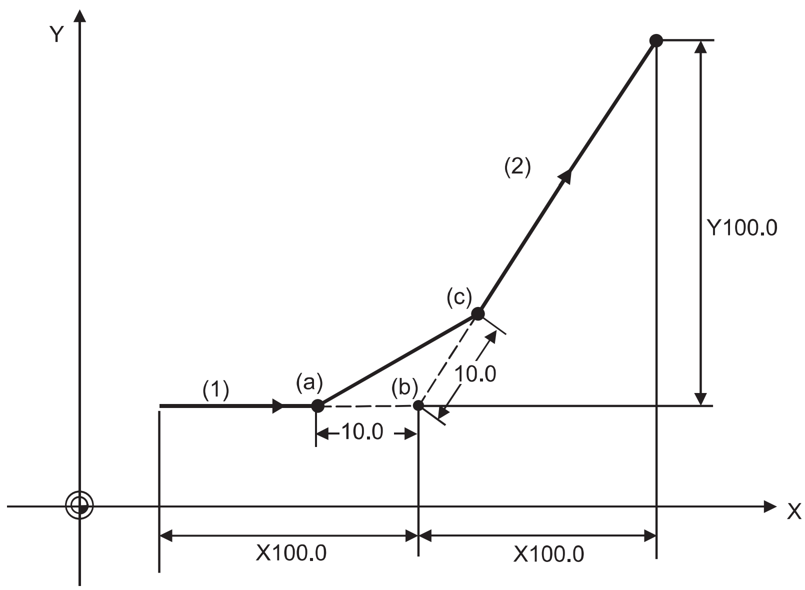
(6) Program error (P383) will occur when the movement amount in the corner chamfering block is less than the chamfering amount.

(7) Program error (P384) will occur when the movement amount in the block following the corner chamfering block is less than the chamfering amount.

(8) Program error (P382) will occur when a movement command is not issued in the block following the corner chamfering I command.

Program example

(1) G91 G01 X100. ,C10.;
(2) X100. Y100.;



(a) Chamfering start point
(b) Hypothetical corner intersection point
(c) Chamfering end point

15.1.2 Corner Rounding I ; G01 X_ Y_ ,R_

Function and purpose

The hypothetical corner, which would exist if the corner were not to be rounded, is rounded with an arc that has a radius commanded by ",R_" only when configured of linear lines.

Command format

```
N100 G01 X__ Y__ ,R__ ;
N200 G01 X__  Y__ ;
```

,R	Arc radius of corner rounding

Corner rounding is performed at the point where N100 and N200 intersect.

Detailed description

(1) The start point of the block following the corner rounding is the hypothetical corner intersection point.

(2) When both corner chamfering and corner rounding are commanded in the same block, the latter command will be valid.

(3) Tool compensation is calculated for the shape which has already been subjected to corner rounding.

(4) When the block following a command with corner rounding does not contain a linear command, a corner chamfering/corner rounding II command will be executed.

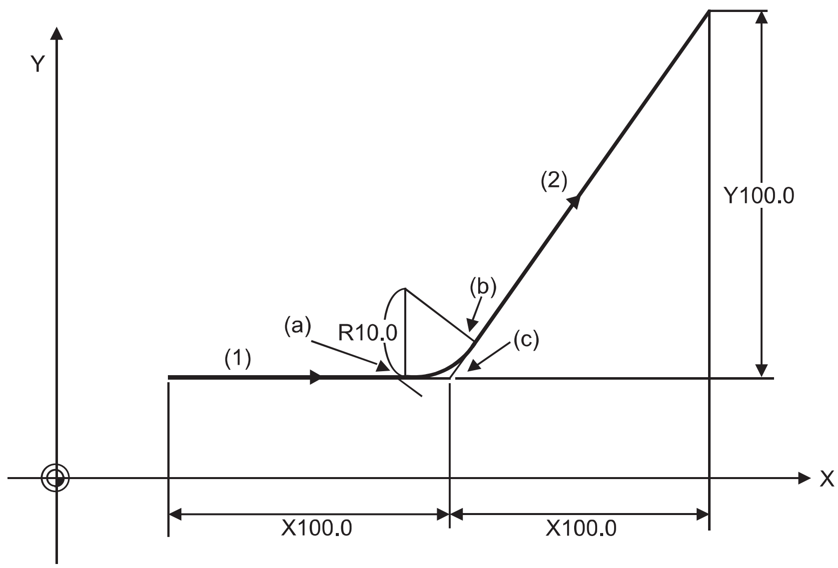
(5) Program error (P383) will occur when the movement amount in the corner rounding block is less than the R value.

(6) Program error (P384) will occur when the movement amount in the block following the corner rounding block is less than the R value.

(7) Program error (P382) will occur if a movement command is not issued in the block following the corner rounding.

Program example

(1) G91 G01 X100. ,R10.;
(2) X100. Y100.;



(a) Corner rounding start point

(b) Corner rounding end point

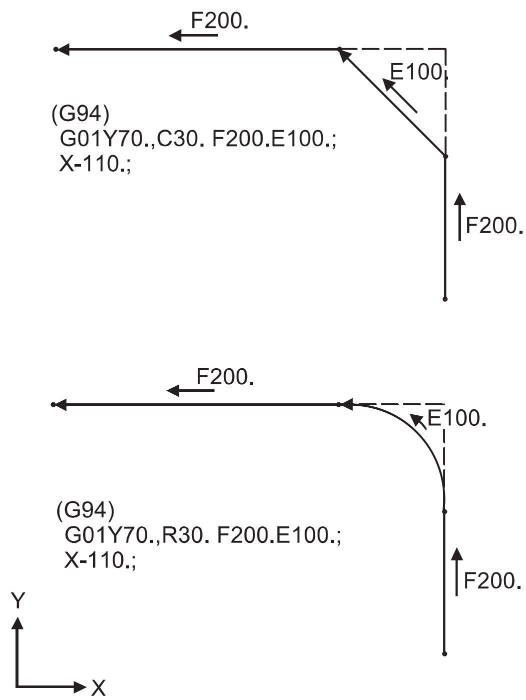
(c) Hypothetical corner intersection point

15.1.3 Corner Chamfering Expansion/Corner Rounding Expansion

Function and purpose

Using an E command, the feedrate can be designated for the corner chamfering and corner rounding section.
In this way, the corner section can be cut into a correct shape.

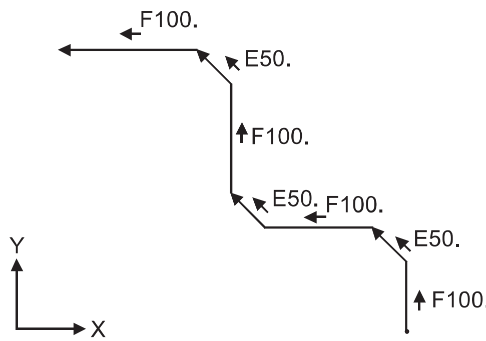
Example



Detailed description

(1) The E command is modal. It is also valid for the feed in the next corner chamfering/corner rounding section.

Example

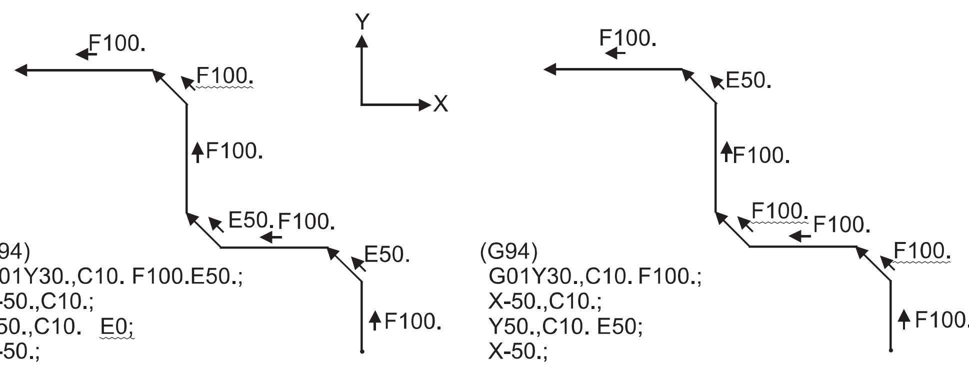



```
(G94)
 G01Y30.,C10. F100.E50.;
 X-50.,C10.;
 Y50.,C10.;
 X-50.;
```

(2) E command modal has separate asynchronous feedrate modal and synchronous feedrate modal functions. Which one is validated depends on the asynchronous/synchronous mode (G94/G95).

(3) When the E command is 0, or when there has not been an E command up to now, the corner chamfering/corner rounding section feedrate will be the same as the F command feedrate.

Example




```
(G94)
 G01Y30.,C10. F100.E50.;
 X-50.,C10.;
 Y50.,C10.   E0;
 X-50.;
```

```
(G94)
 G01Y30.,C10. F100.;
 X-50.,C10.;
 Y50.,C10. E50;
 X-50.;
```

(4) E command modal is not cleared even if the reset button is pressed.
It is cleared when the power is turned OFF. (In the same manner as F commands.)

(5) All E commands except those shown below are at the corner chamfering/corner rounding section feedrate.
- E commands during thread cutting modal
- E commands during thread cutting cycle modal

6.3 Circular Interpolation; G02, G03

Function and purpose

These commands serve to move the tool along a circular.

Command format

Circular interpolation : Clockwise (CW)

G02 X__ Y__ I__ J__ F__ ;

Circular interpolation : Counterclockwise (CCW)

G03 X__ Y__ I__ J__ F__ ;

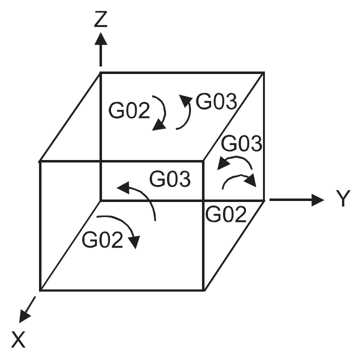
X,Y	Arc end point coordinates
I,J	Arc center coordinates
F	Feedrate

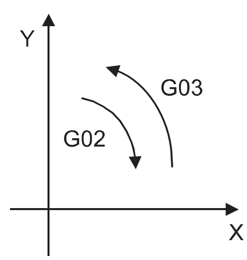
Detailed description

(1) For the arc command, the arc end point coordinates are assigned with addresses X, Y (or Z, or parallel axis X, Y, Z), and the arc center coordinate value is assigned with addresses I, J (or K).
Either an absolute value or incremental value can be used for the arc end point coordinate value command, but the arc center coordinate value must always be commanded with an incremental value from the start point.
The arc center coordinate must be commanded in the input setting unit. Caution is required for the arc command of an axis for which the input command unit differs. Command with a decimal point to avoid confusion.

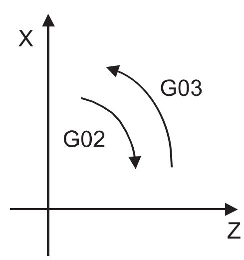
(2) G02 (G03) is a modal command of the 01 group. When G02 (G03) command is issued continuously, the next block and after can be commanded with only coordinate words.
The circular rotation direction is distinguished by G02 and G03.
G02 CW (Clockwise)
G03 CCW (Counterclockwise)

(3) Select the XY plane, ZX plane or YZ plane to draw an arc on it, using the plane selection G code.

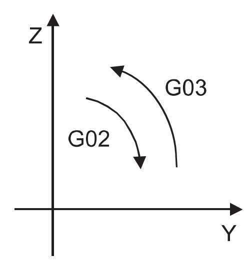




G17(X-Y) plane



G18(Z-X) plane



G19(Y-Z) plane

(4) An arc which extends for more than one quadrant can be executed with a single block command.

(5) The following information is needed for circular interpolation.

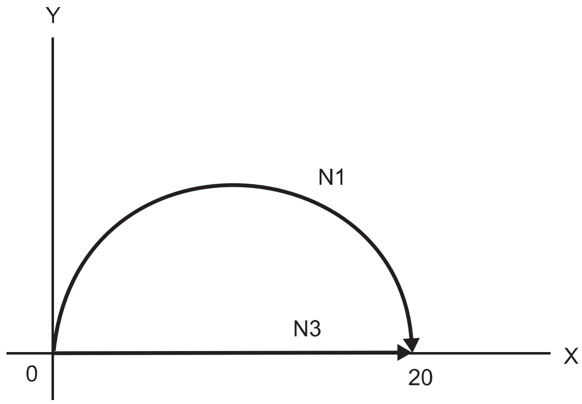
(a) Plane selection	Is there an arc parallel to one of the XY, ZX or YZ planes?
(b) Rotation direction	Clockwise (G02) or counterclockwise (G03)
(c) Arc end point coordinates	Set by addresses X, Y, Z.
(d) Arc center coordinates	Set by addresses I, J, K. (incremental value commands)
(e) Feedrate	Set by address F

(6) If an R specification and I, K specification are given at the same time in the same block, the circular command with the R specification takes precedence.

Change into linear interpolation command

Program error (P33) will occur in general use when the center and radius are not designated at circular command. Depending on the MTB specifications, the linear interpolation can be carried out up to the end point coordinates only in a block with no center coordinates or radius specified (parameter "#11029 Arc to G1 no Cent"). However, a modal is the circular modal.
This function is not applied to a circular command by a geometric function.

(Example) #11029 = "1"

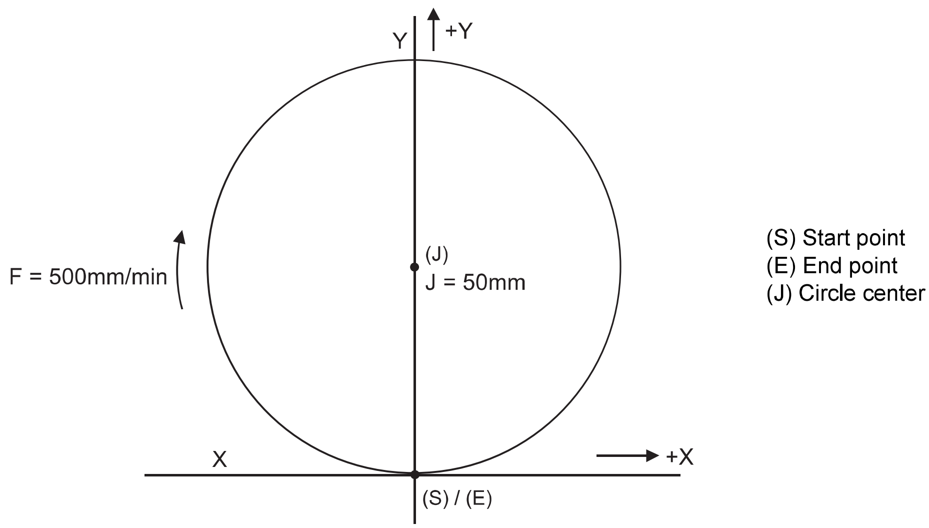



```
G90 X0 Y0 ;
N1 G02 X20. I10. F500 ;     ... (a)
N2 G00 X0 ;
N3 G02 X20. F500 ;          ... (b)
M02 ;
```

(a) The circular interpolation (G02) is executed because there is a center command.
(b) The linear interpolation (G01) is executed because there is no center and radius command.

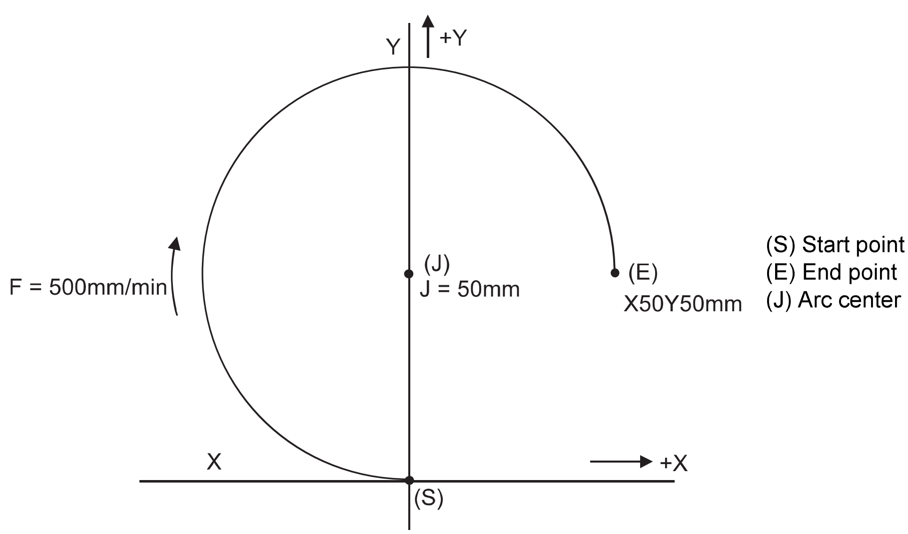
Program example

(Example 1)



G02 J50. F500;	Circle command

(Example 2)



G91 G02 X50. Y50. J50. F500;	3/4 command

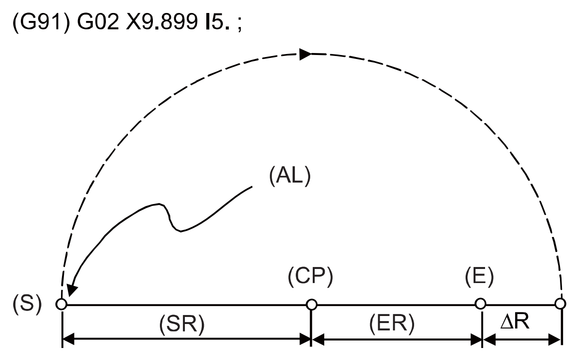
Precautions

(1) The terms "clockwise" (G02) and "counterclockwise" (G03) used for circular operations are defined as a case where, in a right-hand coordinate system, the negative direction is viewed from the positive direction of the co-ordinate axis which is at right angles to the plane in question.

(2) If all the end point coordinates are omitted or the end point is at the same position as the start point, commanding the center using I, J and K is the same as commanding a 360°arc (perfect circle).

(3) The following occurs when the start and end point radius do not match in a circular command :

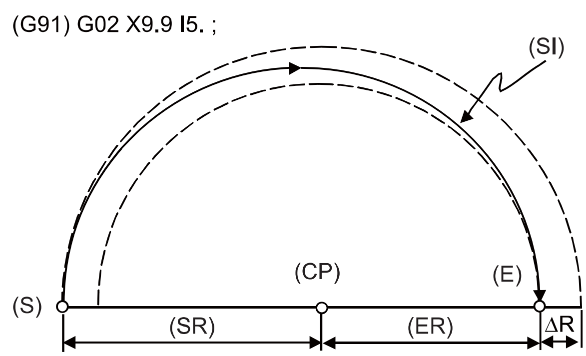
(a) Program error (P70) results at the circular start point when error ΔR is greater than parameter "#1084 RadErr".



#1084 RadErr parameter value 0.100
Start point radius=5.000
End point radius=4.899
Error ΔR =0.101

(S) Start point
(CP) Center point
(E) End point
(SR) Start point radius
(ER) End point radius
(AL) Alarm stop

(b) Spiral interpolation in the direction of the commanded end point will be conducted when error ΔR is less than the parameter value.

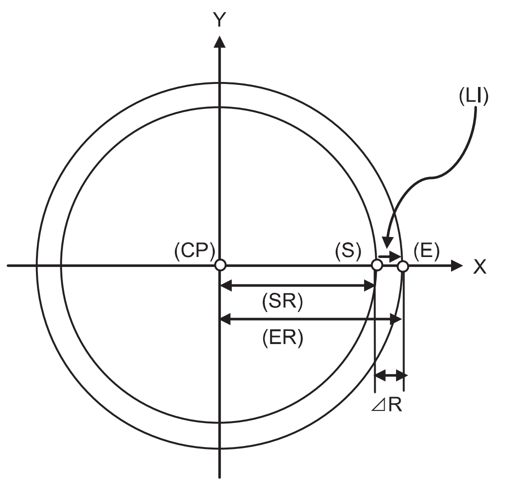
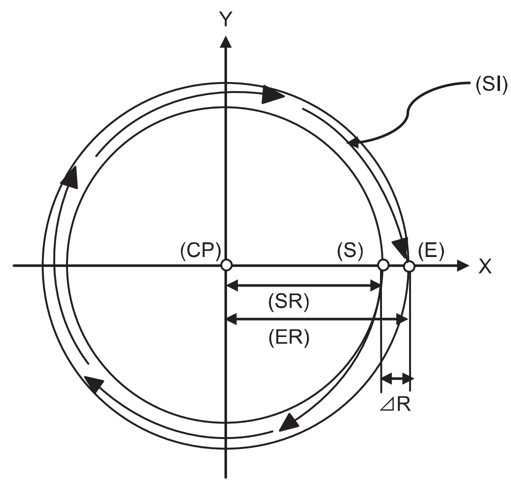


#1084 RadErr parameter value 0.100
Start point radius=5.000
End point radius=4.900
Error ΔR =0.100

(S) Start point
(CP) Center point
(E) End point
(SR) Start point radius
(ER) End point radius
(SI) Spiral interpolation

Also, if "#1084 RadErr" is set to "0", "0.1" is assumed to set.

(c) If the start point radius differs from the end point radius but if the start point angle does not differ from the end point angle, the linear interpolation or spiral interpolation is selected depending on the MTB specifications (parameter "#1278 ext14/bit7").

#1278 ext14/bit = 0 **Linear interpolation**	**#1278 ext14/bit7 = 1** **Spiral interpolation**
G90 G00 X10. Y0.; G02 X10.01 Y0. I-10.01;	G90 G00 X10. Y0.; G02 X10.01 Y0. I-10.01;
	
(CP) Center point (S) Start point (E) End point	(SR) Start point radius (ER) End point radius (LI) Linear interpolation (SI) Spiral interpolation

6.4 R Specification Circular Interpolation; G02, G03

Function and purpose

Along with the conventional circular interpolation commands based on the circular center coordinate (I, J, K) designation, these commands can also be issued by directly designating the circular radius R.

Command format

R specification circular interpolation Clockwise (CW)

```
G02  X__ Y__ R__ F__ ;
```

R specification circular interpolation Counterclockwise (CCW)

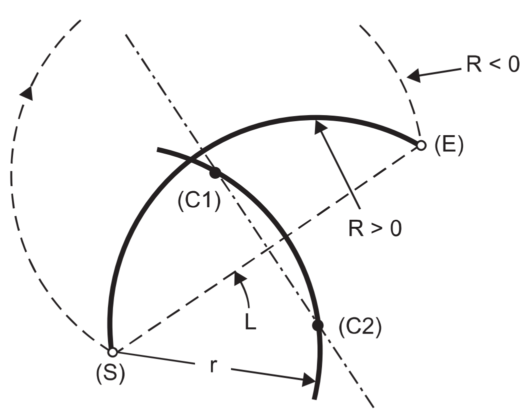
```
G03  X__ Y__ R__ F__ ;
```

X	X axis end point coordinate
Y	Y axis end point coordinate
R	Circular radius
F	Feedrate

The arc radius must be commanded in the input setting unit. Caution is required for the arc command of an axis for which the input command unit (#1015 cunit) differs. Command with a decimal point to avoid confusion.
A maximum of 6 digits before decimal point can be specified for the radius.

Detailed description

The circular center is on the bisector line which is perpendicular to the line connecting the start and end points of the circular. The point, where the circular with the specified radius whose start point is the center intersects the perpendicular bisector line, serves as the center coordinates of the circular command.
If the R sign of the commanded program is plus, the circular is smaller than a semicircular; if it is minus, the circular is larger than a semicircular.



(S) Start point
(E) End point
(C1): Arc center if R<0
(C2): Arc center if R>0

The following condition must be met with an R-specified arc interpolation command:

$\frac{L}{2 \cdot r} \leqq 1$ When (L/2 - r) > (parameter : #1084 RadErr), an alarm will occur.

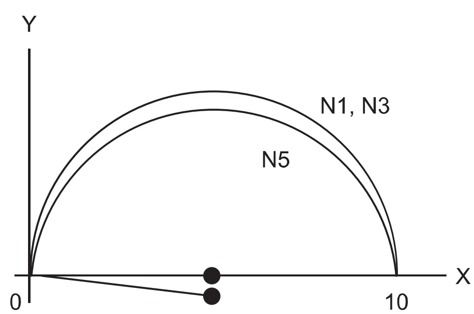
Where L is the line from the start point to the end point. If an R specification and I, J, (K) specification are given at the same time in the same block, the circular command with the R specification takes precedence. In the case of a full-circle command (where the start and end points coincide), an R specification circular command will be completed immediately even if it is issued and no operation will be executed. An I, J, (K) specification circular command should therefore be used in such a case.
The plane selection command is the same as the I, J, or K specification circular command.

Circular center coordinate compensation

When "the error margin between the segment connecting the start and end points" and "the commanded radius × 2" is less than the setting value because the required semicircle is not obtained by calculation error in R specification circular interpolation, "the midpoint of the segment connecting the start and end points" is compensated for as the circular center.
The setting value depends on the MTB specifications (parameter "#11028 Tolerance Arc Cent" (Tolerable correction value of arc center error)).

(Example) #11028 = "0.000 (mm)"

Setting value	Tolerance value
Setting value < 0	0 (Center error will not be interpolated)
Setting value = 0	2×minimum setting increment
Setting value > 0	Setting value




```
G90 X0 Y0 ;
N1 G02 X10. R5.000;
N2 G00 X0;
N3 G02 X10. R5.001;     ...(a)
N4 G00 X0;
N5 G02 X10. R5.002;     ...(b)
N6 G00 X0;
M02 ;
```

(a) Compensate the center coordinate: Same as N1 path
(b) Do not compensate the center coordinate: Inside path a little than N1

Calculation error margin compensation allowance value: 0.002 mm
Segment connecting the start and end points: 10.000
N3: Radius × 2 = 10.002 "Error 0.002 -> Compensate"
N5: Radius × 2 = 10.004 "Error 0.004 -> Do not compensate"

Program example

(Example 1)

G02 Xx1 Yy1 Rr1 Ff1 ;	XY plane R-specified arc

(Example 2)

G03 Zz1 Xx1 Rr1 Ff1 ;	R specification circular on Z-X plane

(Example 3)

G02 Xx1 Yy1 Ii1 Jj1 Rr1 Ff1 ;	XY plane R-specified arc (When the R specification and I, J, (K) specification are contained in the same block, the circular command with the R specification takes precedence.)

(Example 4)

G17 G02 Ii1 Jj1 Rr1 Ff1 ;	XY plane This is an R-specified arc, but as this is a circle command, it will be completed immediately.

Precautions

(1) In the case of a full-circle command (where the start and end points coincide), an R specification circular command will be completed immediately even if it is issued and no operation will be executed. An I, J, K specification circular command should therefore be used in such a case.

(2) If an R specification and I, K specification are given at the same time in the same block, the circular command with the R specification takes precedence.

6.10 Circular Cutting; G12,G13

Function and purpose

Circular cutting starts the tool from the center of the circle, and cuts the inner circumference of the circle. The tool continues cutting while drawing a circle and returns to the center position.

Command format

Circular cutting Clockwise (CW)

G12 I__ D__ F__ ;

Circular cutting Counterclockwise (CCW)

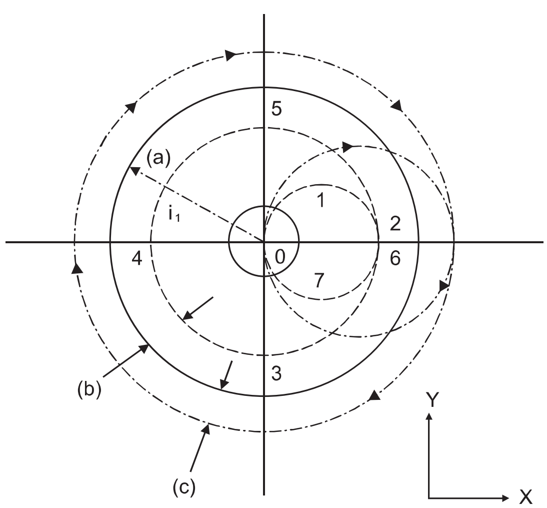
G13 I__ D__ F__ ;

I	Radius of circle (incremental value), the sign is ignored
D	Offset No. (The offset No. and offset data are not displayed on the setting and display unit.)
F	Feedrate

Detailed description

(1) The sign + for the offset amount indicates reduction, and - indicates enlargement.

(2) The circle cutting is executed on the plane G17, G18 or G19 currently selected.



————— Compensation amount sign +   —·—·—·— Compensation amount sign -

(a) Circle radius   (b) d1 offset amount +

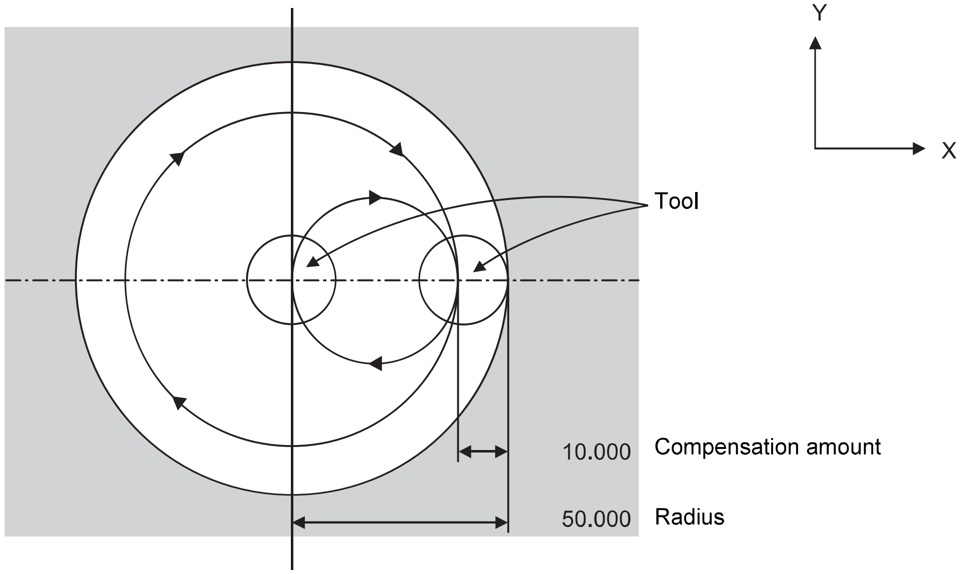
(c) d1 offset amount -

For G12 (tool center path) 0->1->2->3->4->5->6->7->0

For G13 (tool center path) 0->7->6->5->4->3->2->1->0

Program example

(Example 1) G12 I50.000 D01 F100 ; When compensation amount is +10.000mm



Precautions

(1) If the offset No. "D" is not issued or if the offset No. is illegal, the program error (P170) will occur.

(2) If [Radius (I) - offset amount] is 0 or negative, the program error (P223) will occur.

(3) If G12 or G13 is commanded during radius compensation (G41, G42), the radius compensation will be validated on the path after compensated with the D, commanded with G12 or G13.

(4) If an address not included in the format is commanded in the same block as G12 and G13, the program error (P32) will occur.
But when the parameter "#11034 Circular cutting command address check type" is set to "1", it operates as follows;

(a) Program error will not occur except for an "H" command.

(b) Only "D","F","I" and "M","S","T","B" will be valid.

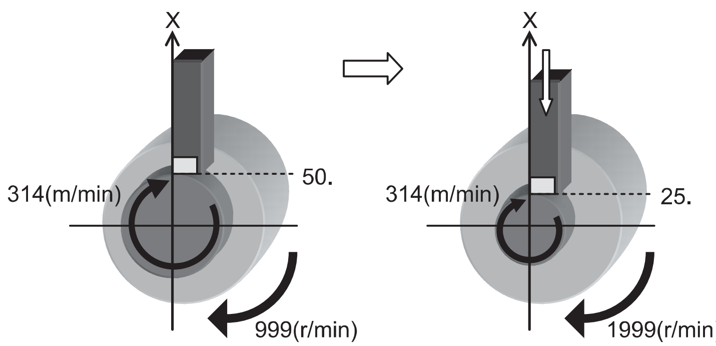
10.2 Constant Surface Speed Control; G96, G97

Function and purpose

This function adjusts the spindle rotation speed (constant surface speed control) in accordance with the movement of the tool nose point so that the cutting point always remains at the constant speed (constant surface speed).
Using this function for processes such as a cutting-off process is effective in terms of machining time, tool life, etc.
Note that when the tool nose point is moving to the workpiece zero point, the rotation may be at the maximum rotation speed defined in the machine specifications; this is dangerous. Be sure to specify the maximum clamp rotation speed with the spindle clamp speed setting command (G92/G50).

Constant surface speed control at constant surface speed command G96 S314 m/min

Workpiece diameter: 50 mm (Radius value) — Workpiece diameter: 25mm (Radius value)



To keep the surface speed constant, this function obtains and automatically adjusts the spindle rotation speed in accordance with the movement of the tool nose point.
In the example above, to keep the surface speed (314 (m/min)) constant, the rotation speed is changed from 999 (r/min) to 1999 (r/min) with changes of the workpiece radius (50mm → 25mm).

Spindle rotation speed (r/min) = Surface speed (m/min) / Workpiece surface (m/r)

Surface speed (m/min)	Workpiece surface (m/r)
G96 command value	Automatically calculated from the workpiece zero point and tool nose position

Note

(1) When the surface speed constant control is commanded under Inch system, the error of the spindle rotation speed specification depends on the MTB specifications (parameter "#1255 set27/bit0").

Command format

Constant surface speed ON

G96 S__ P__ ;	
S	Surface speed (-99999999 to 99999999 (m/min), -99999999 to 99999999 (feet/min))
P	Constant surface speed control axis 0 to n (n: Number of axes that can be controlled in the part system with G96 commanded)

Note

(1) The S command is handled as the absolute value (the sign is ignored).

(2) If the value of the S command exceeds the allowable range, a program error will occur (P35).

(3) If the value of the P command exceeds the allowable range, a program error will occur (P133).

Constant surface speed cancel

G97 S__ ;	
S	Spindle rotation speed (-99999999 to 99999999 (r/min))

Note

(1) The S command is handled as the absolute value (the sign is ignored).

10.3 Spindle Clamp Speed Setting; G92

Function and purpose

The maximum clamp rotation speed of the spindle can be assigned by address S following G92 and the minimum clamp rotation speed by address Q.
Use this command when the spindle speed needs to be limited depending on the workpiece to be machined, the chuck to be mounted on the spindle and the tool specifications, etc.

Command format

Spindle clamp speed setting

G92 S__ Q__ ;

S	Maximum clamp rotation speed
Q	Minimum clamp rotation speed

Detailed description

(1) Besides this command, parameters can be used to set the rotation speed range up to 4 stages in 1 r/min units to accommodate gear selection between the spindle and spindle motor. The lowest upper limit and highest lower limit are valid among the rotation speed ranges based on the parameters and based on "G92 S_ Q_;".

(2) Whether to carry out rotation speed clamp only in the constant surface speed mode or even when the constant surface speed is canceled depends on the MTB specifications (parameters "#1146 Sclamp" and "#1227 aux11/bit5").

<Note>

⬩G92S command and rotation speed clamp operation

<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Sclamp=0</th><th colspan="2">Sclamp=1</th></tr>
<tr><th>aux11/bit5=0</th><th>aux11/bit5=1</th><th>aux11/bit5=0</th><th>aux11/bit5=1</th></tr>
<tr><td rowspan="2">Command</td><td>In G96</td><td colspan="2">Rotation speed clamp command</td><td colspan="2">Rotation speed clamp command</td></tr>
<tr><td>In G97</td><td colspan="2">Spindle rotation speed command</td><td colspan="2">Rotation speed clamp command</td></tr>
<tr><td rowspan="2">Operation</td><td>In G96</td><td colspan="2">Rotation speed clamp execution</td><td colspan="2">Rotation speed clamp execution</td></tr>
<tr><td>In G97</td><td colspan="2">No rotation speed clamp</td><td>Rotation speed clamp command</td><td>No rotation speed clamp</td></tr>
</table>

⬩The address Q following the G92 command is handled as the spindle speed clamp command regardless of the constant surface mode.

(3) The command value of the spindle clamp rotation speed will be cleared by modal reset (reset 2 or reset & rewind). Note that the modal is retained if the parameter "#1210 RstGmd / bit19" is ON.
It is set to "0" during power ON.

5.1 Position Command Methods ; G90,G91

Function and purpose

By using the G90 and G91 commands, it is possible to execute the next coordinate commands using absolute values or incremental values.
The R-designated circle radius and the center of the circle determined by I, J, K are always incremental value commands.

Command format

G90/G91 X__ Y__ Z__ α__ ;

G90	Absolute command
G91	Incremental command
X,Y,Z,α	Coordinate values (α is the additional axis.)

Detailed description

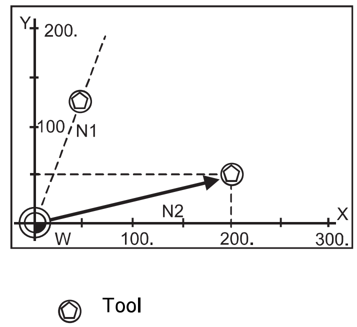
(1) Regardless of the current position, in the absolute value mode, it is possible to move to the position of the workpiece coordinate system that was designated in the program.

N1 G90 G00 X0 Y0 ;

In the incremental value mode, the current position is the start point (0), and the movement is made only the value determined by the program, and is expressed as an incremental value.

N2 G90 G01 X200. Y50. F100 ;
N2 G91 G01 X200. Y50. F100 ;

Using the command from the 0 point in the workpiece coordinate system, it becomes the same coordinate command value in either the absolute value mode or the incremental value mode.



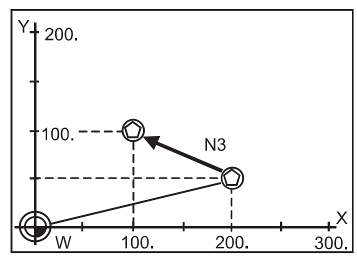
(2) For the next block, the last G90/G91 command that was given becomes the modal.

(G90) N3 X100. Y100. ;

The axis moves to the workpiece coordinate system X = 100.mm and Y = 100.mm position.

(G91) N3 X-100. Y50. ;

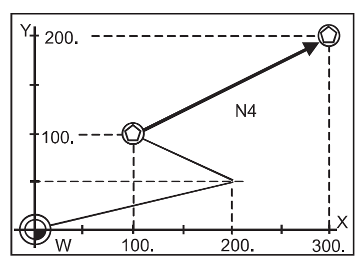
The X axis moves to -100.mm and the Y axis to +50.0mm as an incremental value, and as a result X moves to 100.mm and Y to 100.mm.



(3) Since multiple commands can be issued in the same block, it is possible to command specific addresses as either absolute values or incremental values.

N4 G90 X300. G91 Y100. ;

The X axis is treated in the absolute value mode, and with G90 is moved to the workpiece coordinate system 300.mm position. The Y axis is moved +100.mm with G91. As a result, Y moves to the 200.mm position. In terms of the next block, G91 remains as the modal and becomes the incremental value mode.



(4) When the power is turned ON, it is possible to select whether you want absolute value commands or incremental value commands with the #1073 I_Absm parameter.

(5) Even when commanding with the manual data input (MDI), it will be treated as a modal from that block.

6.5 Plane Selection; G17, G18, G19

Function and purpose

The plane to which the movement of the tool during the circle interpolation (including helical cutting) and tool radius compensation command belongs is selected.
If the 3 basic axes and the parallel axes corresponding to these basic axes are entered as parameters, the commands can select the plane composed of any 2 axes which are not parallel axes. If a rotary axis is entered as a parallel axis, the commands can select the plane containing the rotary axis.
These commands are used to select following planes:

- Plane that executes circular interpolation (including helical cutting)
- Plane that executes tool radius compensation
- Used to select a plane that executes fixed cycle positioning.

Command format

G17 ;	**X-Y plane selection**

G18 ;	**Z-X plane selection**

G19 ;	**Y-Z plane selection**

X, Y and Z indicate each coordinate axis or the parallel axis.

Detailed description

Parameter entry

	#1026-1028 Basic axis I, J, K	**#1029-1031 Flat axis I, J, K**
I	X	U
J	Y	
K	Z	V

Table 1 Examples of plane selection parameter entry

As shown in the above example, the basic axis and its parallel axis can be registered.
The basic axis can be an axis other than X, Y and Z.
Axes that are not registered are irrelevant to the plane selection.

12.3 Tool Radius Compensation; G38,G39/G40/G41,G42

Function and purpose

This function compensates the radius of the tool. The compensation can be done in the random vector direction by the radius amount of the tool selected with the G command (G38 to G42) and the D command.
When using tool nose radius compensation, refer to "12.4 Tool Nose Radius Compensation (for Machining Center System)".

Command format

G40 X__Y__;	**Tool radius compensation cancel**

G41 X__Y__ D__;	**Tool radius compensation (Left)**

G42 X__Y__ D__;	**Tool radius compensation (Right)**

G38 I__J__;	**Change or hold of compensation vector (Can be commanded only during the radius compensation mode.)**

G39 X__Y__;	**Corner changeover (Can be commanded only during the radius compensation mode.)**

Detailed description

The number of sets for the compensation differ according to machine specification. (The No. of sets is the total of the tool length offset, tool position offset and tool radius compensation sets.)
The H command is ignored during the tool radius compensation, and only the D command is valid.
The compensation will be executed within the plane designated with the plane selection G code or axis address 2 axis, and axes other than those included in the designated plane and the axes parallel to the designated plane will not be affected. Refer to the section on plane selection for details on selecting the plane with the G code.

12.3.1 Tool Radius Compensation Operation

Detailed description

Tool radius compensation cancel mode

The tool radius compensation cancel mode is established by any of the following conditions.

(1) After the power has been switched on
(2) After the reset button on the setting and display unit has been pressed
(3) After the M02 or M30 command with reset function has been executed
(4) After the compensation cancel command (G40) is issued

The compensation vectors are zero in the compensation cancel mode, and the tool nose point path coincides with the programmed path.
Programs including tool radius compensation must be terminated in the compensation cancel mode.

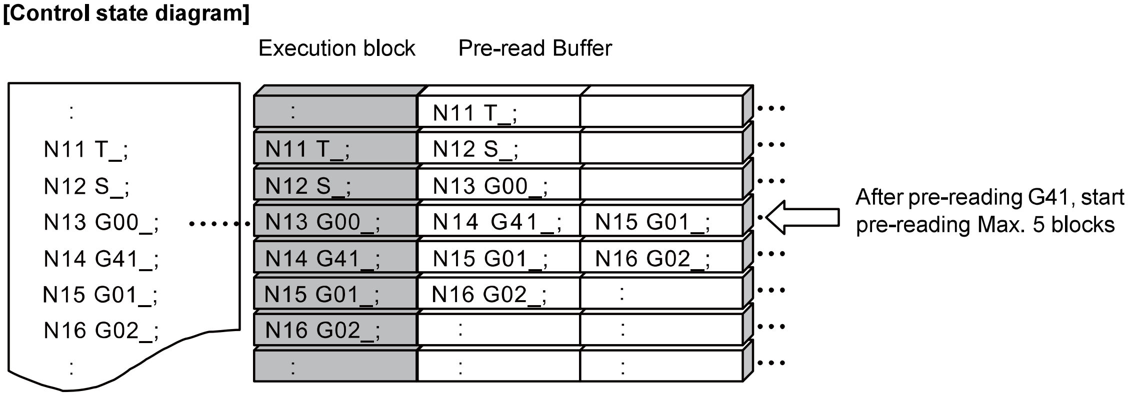
Tool radius compensation start (startup)

Tool radius compensation starts when all the following conditions are met in the compensation cancel mode.

(1) The movement command is issued after G41 or G42.
(2) The tool radius compensation offset No. is 0 < D <= max. offset No.
(3) The movement command of positioning (G00) or linear interpolation (G01) is issued.

Whether in continuous or single block operation, compensation always starts after reading three blocks, or if the three blocks do not contain any movement command, up to five continuous blocks will be pre-read.
In compensation mode, too, up to 5 blocks are pre-read and the compensation is arithmetically processed.

[Control state diagram]



There are two ways of starting the compensation operation: type A and type B.
The type depends on the setting of the parameter "#8157 Radius comp type B".
This type is used in common with the compensation cancel type.

12.2 Tool Length Compensation/Cancel; G43, G44 / G49

Function and purpose

The end position of the movement command for each axis can be compensated for by the preset amount when this command is issued. A continuity can be applied to the program by setting the actual deviation from the tool length value decided during programming as the compensation amount using this function.

Command format

Tool length compensation start

G43 Zz Hh ; (+ direction)

G44 Zz Hh ; (- direction)

Tool length compensation cancel

G49 Zz ;

Detailed description

Tool length compensation movement amount

The movement amount is calculated with the following expressions when the G43 or G44 tool length compensation command or G49 tool length compensation cancel command is issued.

	Z axis movement amount	Operation
G43 Zz Hh1;	z +(lh1)	Compensation in + direction by tool compensation amount
G44 Zz Hh1;	z -(lh1)	Compensation in - direction by tool compensation amount
G49 Zz;	z -(+)(lh1)	Compensation amount cancel

lh1; Compensation amount for compensation No. h1

Regardless of the absolute value command or incremental value command, the actual end point will be the point compensated for by the compensation amount designated for the programmed movement command end point coordinate value.

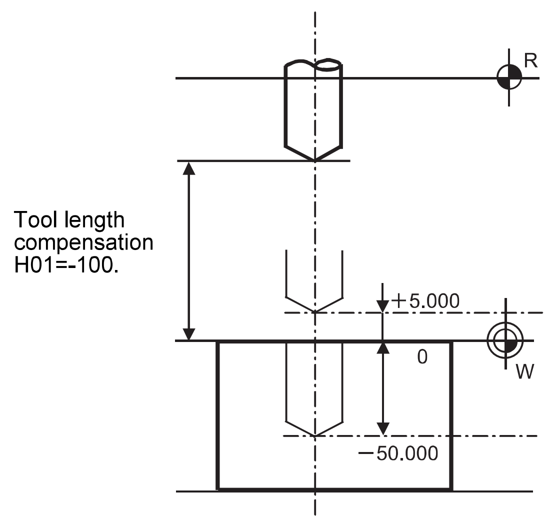
The G49 (tool length compensation cancel) mode is entered when the power is turned ON or when M02 has been executed.

(Example 1) For absolute value command H01 = -100000

```
N1 G28 Z0 T01 M06 ;
N2 G90 G92 Z0 ;
N3 G43 Z5000 H01 ;
N4 G01 Z-50000 F500 ;
```

(Example 2) For incremental value command H01 =-100000

```
N1 G28 Z0 T01 M06 ;
N2 G91 G92 Z0 ;
N3 G43 Z5000 H01 ;
N4 G01 Z-55000 F500 ;
```

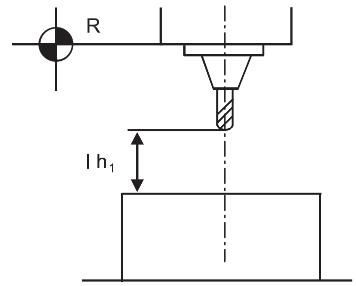


Compensation No.

(1) The compensation amount differs according to the compensation type. The following example shows a case in which "G43 Hh1;" is commanded.

Type I

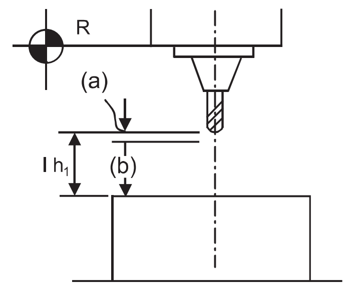
The compensation amount lh1 commanded with compensation No. h1 will be applied commonly regardless of the tool length compensation amount, tool radius compensation amount, shape compensation amount or wear compensation amount.



Type II

The compensation amount lh1 commanded with compensation No. h1 is as follows.

lh1: Shape compensation (b) + wear compensation amount (a)



Type III

The compensation amount lh1 commanded with compensation No. h1 is as follows. (Refer to the figure of type II.)
lh1: Tool length compensation amount in Z axis direction (b) + Wear compensation amount in Z axis report (a)

(2) The valid range of the compensation No. will differ according to the specifications (No. of compensation sets).

(3) If the commanded compensation No. exceeds the specification range, the program error (P170) will occur.

(4) Tool length cancel will be applied when H0 is designated.

(5) The compensation No. commanded in the same block as G43 or G44 will be valid for the following modals.

(Example 3)

G43 Zz1 Hh1 ;	Tool length compensation is executed with h1.
:	
G45 Xx1 Yy1 Hh6 ;	
:	
G49 Zz2 ;	Tool length compensation is canceled.
:	
G43 Zz2 ;	Tool length compensation is re-executed with h1.
:	

(6) If G43 is commanded in the G43 modal, a compensation of the difference between the compensation No. data will be executed.
(Example 4)

G43 Zz1 Hh1 ;	The axis moves by "z1 + (lh1)".
:	
G43 Zz2 Hh2 ;	The axis moves by "z2 + (lh2-lh1)".
:	

The same applies for the G44 command in the G44 modal.

Axis valid for tool length compensation

(1) When parameter "#1080 Dril_Z" is set to "1", the tool length compensation is always applied to the Z axis.

(2) When parameter "#1080 Dril_Z" is set to "0", the axis will depend on the axis address commanded in the same block as G43. The order of priority is shown below.
Zp > Yp > Xp
(Example 5)

G43 Xx1 Hh1;	+ compensation to X axis
:	
G49 Xx2 ;	
:	
G44 Yy1Hh2;	- compensation to Y axis
:	
G49 Yy2 ;	
:	
G43 αα1 Hh3;	+ compensation to additional axis
:	
G49 αα1 ;	
:	
G43 Xx3Yy3Zz3 ;	Compensation is applied on Z axis.
:	
G49 ;	

The handling of the additional axis will follow the parameters "#1029 aux_I" to "1031 aux_K" settings.
If the tool length compensation is commanded for the rotary axis, set the rotary axis name for one of the parallel axes.

(3) If H (compensation No.) is not designated in the same block as G43, the Z axis will be valid.
(Example 6)

G43 Hh1 ;	Compensation and cancel to Z axis
:	
G49 ;	
:	

Movement during other commands in tool length compensation modal

(1) If reference position return is executed with G28 and manual operation, the tool length compensation will be canceled when the reference position return is completed.
(Example 7)

G43 Zz1 Hh1 ;	
:	
G28 Zz2 ;	Canceled when reference position is reached. (Same as G49)
:	
G43 Zz2Hh2 ;	
:	
G49 G28Zz2 ;	The tool length compensation will be included when positioning the intermediate point. Canceled when reference position is reached.

(2) The movement is commanded to the G53 machine coordinate system, the axis will move to the machine position without tool compensation amount.
When the G54 to G59 workpiece coordinate system is returned to, the position returned to will be the coordinates shifted by the tool compensation amount.