When circular interpolation (G02, G03) is to be programmed, usually, the plane of interpolation should be specified in advance with G17, G18 or G19.

G17: XY plane or X4 plane†

G18: ZX plane or Z4 plane†

G19: YZ plane or Y4 plane†

In addition to the plane of circular interpolation, these G codes specify planes for tool radius compensation (G41, G42). If no selection is made to the contrary, XY plane (G17) is selected automatically immediately after the switching of the power supply.

The end point of the circular arc may be specified by G90 or G91 respectively in absolute or incremental values. However, the center of the circle is always programmed in incremental values from the start point, irrespective of G90 or G91.

EXAMPLE

G17 G90 G03 X1500 Y4000 I-3000 J-1000 F150 ;

(a) Absolute command with (G90)

G17 G91 G03 X-4000 Y2000 I-3000 J-1000 F150 ;

(b) Incremental command

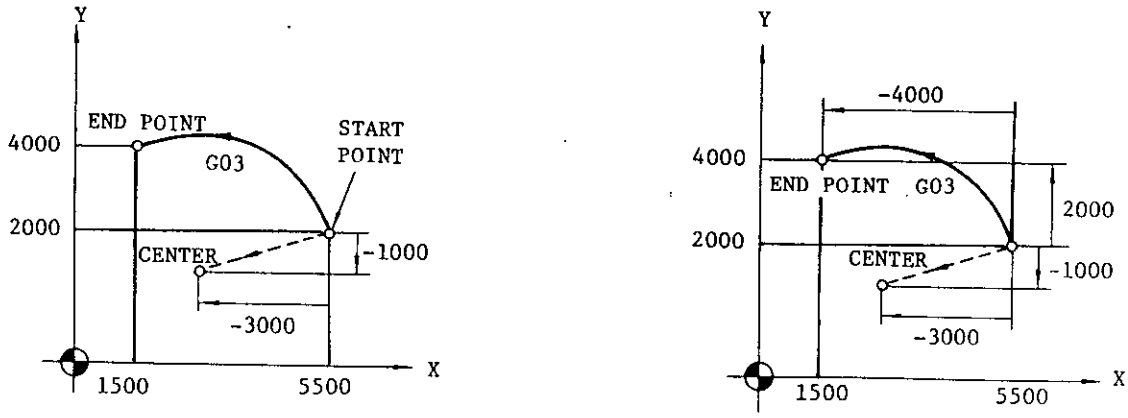

Fig. 2.14

Instead of the coordinates I, J, and K of the center of the circle, the radius can be directly specified with an R command. This is called circular interpolation with radius R designation mode.

G17 G02 X··· Y··· R±··· F··· ;

In this case,

when R > 0, a circular arc with the center angle less than 180°, and
when R < 0, a circular arc with the center angle larger than 180° are specified.

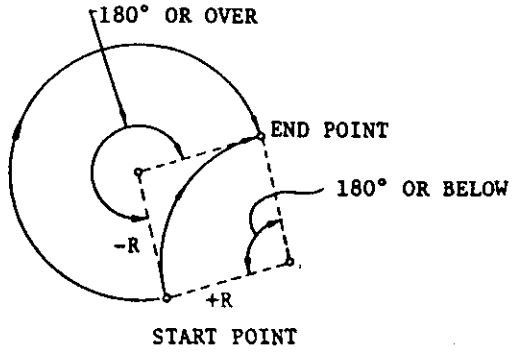

Fig. 2.15