2.9.21 TOOL RADIUS COMPENSATION C (G40, G41, G42)† (CONT'D)

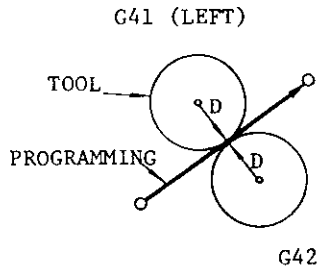


Fig. 2.38

Switching between G41 and G42 can be made in the compensation mode. Details will be given in item 5 below.

2. Designation of compensating plane

The plane in which tool radius compensation is made is designated by G17, G18, G19. They are G codes of 02 group. The XY plane (G17) is in effect at the time power is turned on.

G codes for designation of planes

G code	Group	Meaning
G17	02	XY plane
G18	02	ZX plane
G19	02	YZ plane

Note: When the power is turned on, G17 is effective.

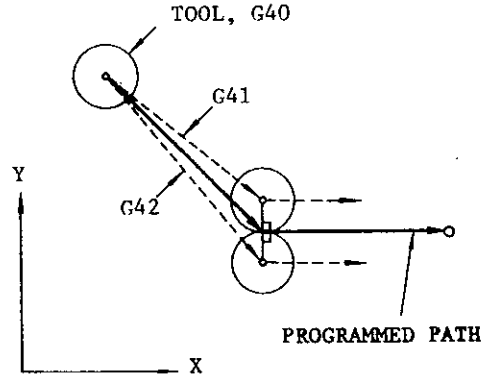
Make sure to designate a G code for plane designation in the same block as that of G41, G42 or in a preceding block. Plane designation cannot be made in a compensation mode. It is not possible to apply tool radius compensation in a plane including the fourth axis†.

3. Method of entry into compensation mode

When G41(G42) is programmed, the tool moves to an offset position with the distance equal to the radius. The offset position is on the normal line at the start point of the block immediately after G41(G42). If no coordinate instruction is programmed in the block of G41(G42), movement is made by the offset value only. Because G41(G42) accompanies a movement, it is necessary to program G00 or G01 for a G code in group A. An input error (alarm code "026) occurs if a G code other than G00, G01 is programmed.

EXAMPLE A

(a) G17 G01 F··· ;
G41(G42) D··· X··· Y··· ;
X··· ;



(b) G17 G01 F··· ;
G41(G42) D··· X··· Y··· ;
G02 X··· Y··· J··· ;

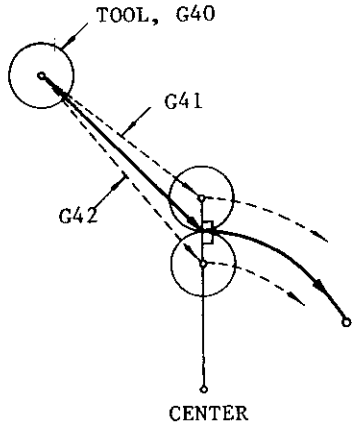


Fig. 2.39