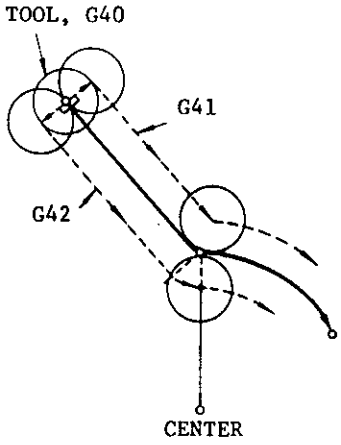
EXAMPLE B

(c) G17 G01 F··· ;

G41(G42) D··· ;

X··· Y··· ;

G02 X··· Y··· J··· ;



(d) G17 G01 F··· ;

G41(G42) D··· ;

G02 X··· Y··· J··· ;

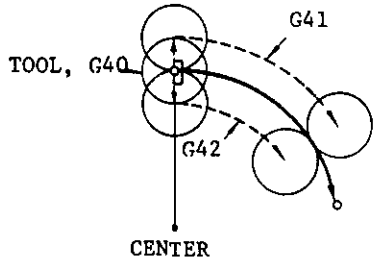


Fig. 2.40

Pay attention to the fact that offset is made on the normal line to the program line determined by the block after G41(G42) at the start point in all of the examples (a) to (d) above. When the movement on the compensation plane is not programmed in the block after G41(G42), the next one block is read ahead and the compensation start with the block. The blocks without move command can be programmed continuously up to two. Input error occurs if move commands on the compensation plane are not programmed in more than three blocks.

When compensation entry is programmed in the G00 mode, positioning movement is made independently by each axis to the offset point. Take care not to make the tool interfere with the workpiece.

4. Movement in compensation mode

When after the tool radius compensation is programmed by G41, G42, the tool moves along the offset path until the instruction G40 is given.

As calculation of the path is automatically made by the control, designate only the shape of the workpiece in the program. The tool path is controlled as follows depending on the angle between blocks.

A. Inside corner (180° or less): Intersection computing type

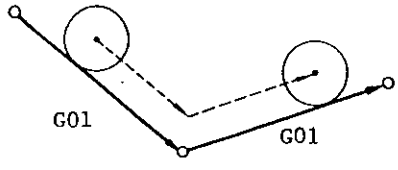


Fig. 2.41