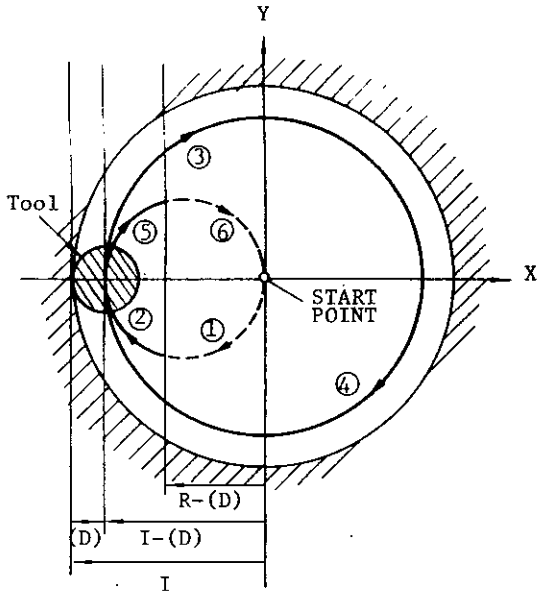
2.9.9 CIRCLE CUTTING (G12, G13)† (CONT'D)



Tool path

G12: ① → ② → ③ → ④ → ⑤ → ⑥

G13: ⑥ → ⑤ → ④ → ③ → ② → ①

(D) represents a set value of tool radius compensation.

G12: Clockwise(CW)

G13: Counterclockwise (CCW)

I: Radius of finished circle (incremental value with sign)

R: Rapid traverse section (incremental value with sign)

D: Tool radius compensation value

Fig. 2.23

· Combined designation of rapid traverse section, repeated circle and spiral circle.

Rapid traverse section, repeated circle designation and spiral circle can be commanded in combination as shown below.

G12(G13) I··· R··· (or J···) K··· Q···
L··· F··· ;

NOTES:

· Circle cutting is possible only on the XY plane.

· The tool speed in the rapid traverse section is set by parameter #6225.

· With a circle cutting command (G12, G13), the tool is offset for its radius compensation without the use of G41 or G42 (tool radius compensation). When using G12 or G13, cancel tool radius compensation with G40.

· In the explanation above, only motions in the + direction of X-axis is considered. With proper use of signs for I, J, K and D codes, motions in the - direction of X-axis (symmetrical with respect to Y-axis) can be commanded. In Fig. 2.23, signs of I, R and (D) are minus. However, cutting in the Y-axis direction is impossible.

· I, J, K, R, Q and L codes in circle cutting command are effective only in the block containing them.

· The radius I of finished circle and the rapid traverse section R are subject to the following restriction. When values not in conformity with the restriction are programmed, this is regarded as an error.

$$|R - d| < |I - d|$$

If R-d and I-d have different signs, this is also regarded as an error.

· When programming G12(G13), always specify a tool radius compensation number D. If this is not specified, the tool moves without radius compensation.

EXAMPLE

· G00 Z-4000 ;
G12 I5000 R4000 D10 F300 ; D10 = 10.0mm
G00 Z4000 ;

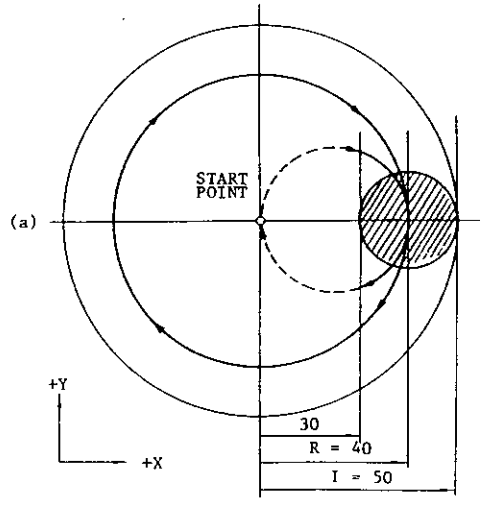


Fig. 2.24