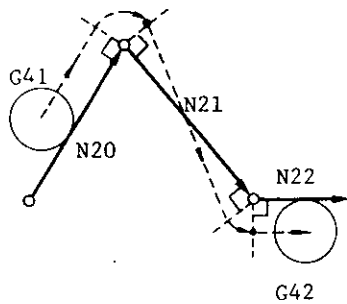
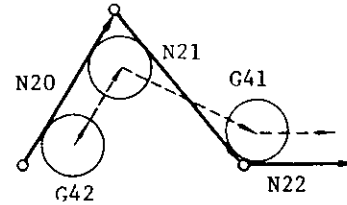


2.9.21 TOOL RADIUS COMPENSATION C  
(G40, G41, G42)<sup>†</sup> (CONT'D)



(a) G41 → G42  
(M96 mode)



(b) G42 → G41

Fig. 2.48

Note: The movement identical to what is shown above is also obtained when block N21 is split into two blocks as follows.

```
G42 (G41) ;
      X... Y... ;
```

7. Change of tool radius value in compensation mode

New D code commanded in the compensation mode is effective in the block next to the commanded block.

8. Method of cancellation of compensation

G40 is the command for cancelling tool radius compensation C and for positioning or feeding the tool to just programmed end point. In this case, the tool moves to a point on the normal line at the end point of the block immediately before the block containing G40.

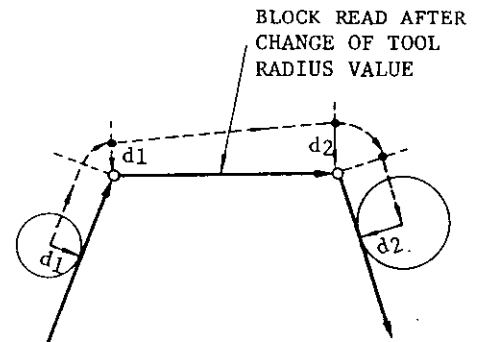


Fig. 2.49

Therefore, no portion will be left unmachined even when a cancellation with sharp angle is programmed. Because G40 accompanies cancelling movement, program it in the G00 or G01 mode like G41, G42. An input error "027" occurs if group A other than G00, G02 is used.