C. Input errors occur when the following G codes are programmed in the compensation mode.

	Prohibited G codes
G codes producing input errors	G12, G13 (G17 to G19) G28, G29 G73, G74, G76, G77 G81 to G89 G92

* If a "reset operation" is performed in the compensation mode, compensation is cancelled and G40 remains.

D. Tool radius compensation C is applied to the movement path offset by tool length offset and tool position offset. However, in principle, avoid applying compensation C to the path using tool position offset for compensation of tool radius.

E. When programming G41, G42 and G40, G00 or G01 and an F code should be programmed in the same block or in a preceding block.

F. An input error occurs if a G code, G17 to G19 of plane designation for changing the compensation plane is programmed during compensation.

G. Program circle cutting (G12, G13), and canned cycles (G73, G74, G76, G77, G80 to G89) in the tool radius compensation cancel mode. Circle cutting and helical cutting incorporate tool radius compensating functions in themselves. Input error "024" occurs when they are programmed in the compensation mode.

H. Tool radius compensation C is also possible on circular interpolation by radius R designation.

I. Subprogram (M98, M99) can be programmed in the compensation mode.

J. Compensation is applied to the projection to the compensation plane designated by G17, G18 or G19 when simultaneous movement along three axes (four axes†) is programmed in compensation mode.

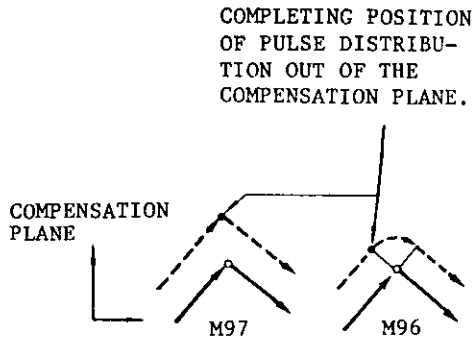

Fig. 2.55

K. Input error "046" occurs when circular interpolation is programmed out of the plane designated by G17, G18 or G19.

L. Offset position may be temporarily modified by programming a dummy block using addresses I, J, K.

```
(G42)
 .
 .
 .
N100  G01  X···  Y···  ;
N101       I···  J···  ;
N102       X···  ;
```

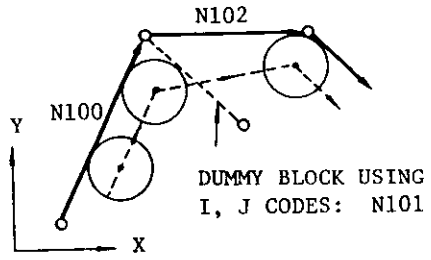

Fig. 2.56