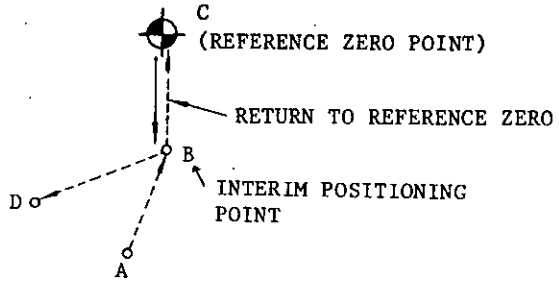
2.9.15 RETURN FROM REFERENCE ZERO (G29)†

This code is used to return the tool to its original position after return to reference zero by automatic return to reference zero, along the same path.

```
G28 Y··· Z··· ;        Point A → B → C
  .    Point B         (Reference zero point)
  .
  .
G29 Y··· Z··· ;        Point C → B → D
    Point D
```



Figl 2.29

When G29 is programmed, it is not necessary to consider the distance between point B and C in the program. Particularly when an incremental instruction is used, this is effective for returning tool to the original position, after returning to reference zero.

Movement of C → B and of B → D is made at rapid traverse rate simultaneously along three axes (simultaneously four axes†) by G29. However, in an axis for which a coordinate instruction was omitted, the tool will not move.

If G28 is programmed a number of times, the final coordinates of point B which the last G28 creates is effective for the move of G29.

EXAMPLE 1 (In the case of absolute input)

```
                          Interim point coordinates
                                X     Y     Z
N21 G90 ;
N22 G28 Z1000 Y2000 ; ⟶ (0, 2000, 1000)
N23 G28 X3000 ;       ⟶ (3000, 2000, 1000)
```

```
N24 G29 X-4000 Y-5000 ;
```

Equal to the two blocks =

```
G00 X3000 Y2000 ;
G00 X-4000 Y-5000 ;
```

EXAMPLE 2

```
N31 G91 ;
N32 G28 Z··· ;
N33 G28 X2000 Y4000 ;
N34 M06 ;
N35 G29 X4000 Y-4000 ;
 .
 .
 .
```

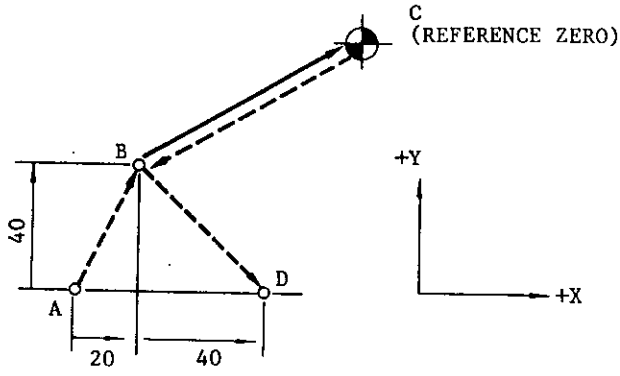


Fig. 2.30

NOTES:

- An input error "024" occurs if G29 is programmed in tool radius compensation mode (G41, G42) or during canned cycle mode (G73, G74, G76, G77, G81 to G89).
- An input error "059" occurs if G29 is given without execution of G28 after the control is turned on.
- In principle, cancel tool offset before programming G28 or G29. If they are programmed when offset is also effective, interim positioning point B will also be offset, and the tool passes point B'.

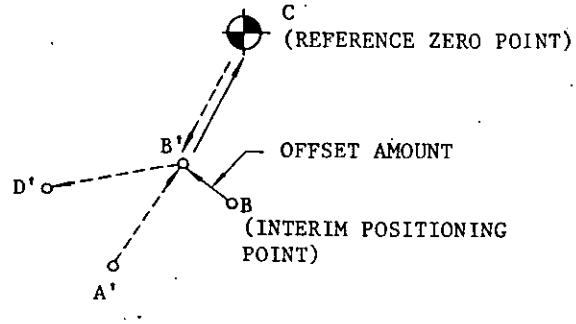


Fig. 2.31