2.9.2 POSITIONING (G00, G06)

G00 X··· Y··· Z··· (α+...) ;
(where α = A, B, C, U, V, or W)

With this command, the tool is sent to the specified position in rapid traverse motions along the 3 axes (4 axes+) simultaneously. If any of the coordinate positions is not specified, the machine does not move along that coordinate axis.

The rapid traverse rate for the respective axes are inherent to the machine tool. Refer to the machine tool builder's manual.

Motions in the respective axis directions are independent each other, and therefore, the resultant tool path is not necessarily straight. When programming tool positioning commands, take care to avoid the possibility of tool and workpiece interference.

EXAMPLE

G01 X4000 Y4000 Z4000 ;
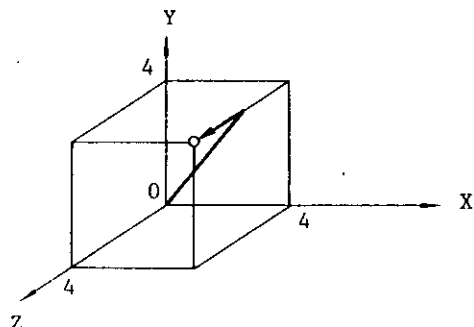
Rapid traverse rate

X axis: 8 m/min
Y axis: 8 m/min
Z axis: 4 m/min

Fig. 2.11

G00 is a modal G code belonging to the 01 group.

Error detect OFF positioning (G06)

G06 X··· Y··· Z··· (α+...) ;

With this command, the same positioning motions are initiated as with a G00 command, with the following exceptions.

· After the completion of the positioning motion with G06 block, the program advances to the next block in the ERROR DETECT OFF mode (Note). Therefore, the tool path at the corner is rounded.
G06 is a non modal G code belonging to the * group, and therefore, it is effective only in which it is programmed.

NOTE

· G00 commands position the tool in the ERROR DETECT ON mode, which means that the program advances to the next block only after the servo lag pulses are decreased below the permissible level, and this is detected by the control. With this command, therefore the corner of the workpiece is machined sharp.

· With the ERROR DETECT OFF mode commanded by G06, the program advances to the next block immediately after the completion of pulse distribution.

2.9.3 LINEAR INTERPOLATION (G01)

G01 X··· Y··· Z··· (α+...) F··· ;

where α = A, B, C, U, V, or W

With this command, the tool is moved simultaneously in the three (four+) axial directions resulting in a linear motion. When a certain axis is missing in the command, the tool does not move in the axial direction of that axis.
Feedrate is specified by an F code the feedrate in the component axial directions are so controlled that the resultant feedrate becomes the specified feedrate.

$$F = \sqrt{Fx^2 + Fy^2 + Fz^2 + F\alpha^2}$$

(where Fx, Fy··· are feedrate in the X, Y··· directions.)

The end point can be programmed either in absolute coordinates or in incremental values with G90 or G91 respectively. (Refer to 2.9.31 Absolute/Incremental Programming (G90, G91)).

If no F code is given in the block containing the G01 or in preceding blocks, the block constitutes an error "030."

EXAMPLE

G01 X4000 Y4000 Z4000 F100 ;

100 mm/min
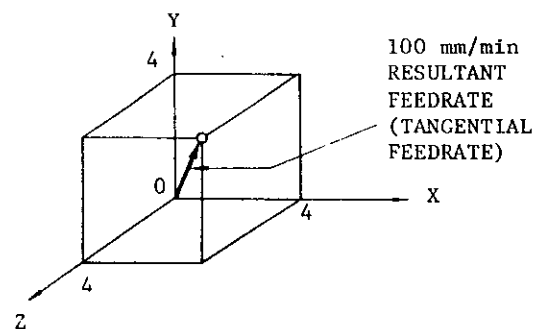RESULTANT
FEEDRATE
(TANGENTIAL
FEEDRATE)

Fig. 2.12

Where the optional 4th axis is a rotary axis (A, B or C), for the same F code, the feedrates in the basic three axis directions (X, Y and Z), and the rotary axis feedrate are as indicated.