2.9.20 Z-AXIS REFERENCE SURFACE OFFSET (G38)†

With this function, the tool is offset in the Z direction automatically for obtaining accurate dimension relative to a reference surface, with a touch sensor.

· G38 K··· F··· ;

With this command, the spindle moves in the Z direction at a speed F, and stops when the touch sensor gives out a contact signal. Then, d = k - g is calculated, and the spindle returns through g - d in rapid traverse, where k is a value specified by K and g is the distance from the start point to the contact point.
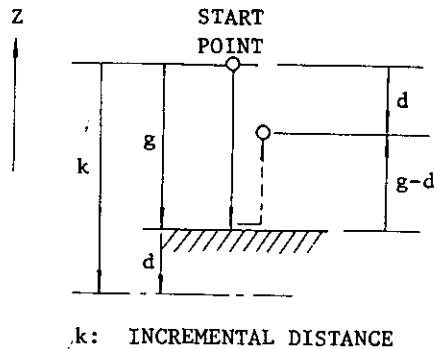

k: INCREMENTAL DISTANCE

Fig. 2.36

NOTES:

· If no contact signal is received during the travel through the specified incremental distance k, an error "087" is caused.

· When G38 is commanded in the tool compensation C mode, or in a canned cycle, this is regarded as an error "024."

EXAMPLE

M06 T11 ; ——— Selection of touch sensor

① G91 G00 Z-z ; —— Approach to the measuring point in Z direction

② G38 K-k F··· ; — Z reference surface compensation

③ Zz ; ————— Return to the offset position in the Z direction

G92 Z0 ; ——— Setting of Z coordinate origin at this point
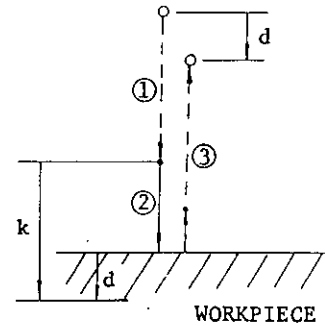

WORKPIECE

Fig. 2.37

2.9.21 TOOL RADIUS COMPENSATION C (G40, G41, G42)†

It is possible to specify the radius of the tool and to cause automatic tool path offset by this value. Store the offset value (tool radius value) in the offset value memory in advance by MDI, and program the tool offset number correspond to the tool radius value by a D code in the program.

1. Designation of compensation direction and of D code

Tool radius compensation C is programmed with G41, G42 and is cancelled by G40. G41 and G42 indicate the directions of tool offset with respect to the direction of movement.

G code of tool radius compensation C

G code	Group	Meaning
G40	07	Cancellation of tool radius compensation C
G41	07	Tool radius compensation C, left
G42	07	Tool radius compensation C, right

Note: When the power is turned on, G40 is effective.

Note that the directions of compensation (right, left) indicated above are reversed when the sign of the tool radius value in the offset memory designated by a D code is negative. Make sure to designate a D code in the block containing G41, G42 or in a preceding block. If D00 is commanded, tool radius will be regarded as "0."