

Fanuc G10 Programmable Offset

G10 Set Offsets (Group 00)

G10 allows the programmer to set offsets within the program. Using G10 replaces the manual entry of offsets (i.e. Tool length and diameter, and work coordinate offsets).

L – Selects offset category.

L2 Work coordinate origin for G52 and G54-G59

L10 Length offset amount (for H code)

L1 or L11 Tool wear offset amount (for H code)

L12 Diameter offset amount (for D code)

L13 Diameter wear offset amount (for D code)

L20 Auxiliary work coordinate origin for G110-G129

P – Selects a specific offset.

P1-P100 Used to reference D or H code offsets (L10-L13)

P0 G52 references work coordinate (L2)

P1-P6 G54-G59 references work coordinates (L2)

P1-P20 G110-G129 references auxiliary coordinates (L20)

P1-P99 G154 P1-P99 reference auxiliary coordinate (L20)

R Offset value or increment for length and diameter.

X Optional X-axis zero location.

Y Optional Y-axis zero location.

Z Optional Z-axis zero location.

A Optional A-axis zero location.

Programming Examples

G10 L2 P1 G91 X6.0 {Move coordinate G54 6.0 to the right};

G10 L20 P2 G90 X10. Y8. {Set work coordinate G111 to X10.0 ,Y8.0};

G10 L10 G90 P5 R2.5 {Set offset for Tool #5 to 2.5};

G10 L12 G90 P5 R.375 {Set diameter for Tool #5 to .375-};

G10 L20 P50 G90 X10. Y20. {Set work coordinate G154 P50 to X10. Y20.}