

## G Codes

G codes are used in part programming to define machine motion, canned cycles, and special machine functions.

Code	Description
G00 -----	Point to point positioning at maximum speed
G01 -----	Linear positioning at current feedrate
G02 -----	Circular positioning clockwise at current feedrate
G03 -----	Circular positioning counter-clockwise at current feedrate
G04 -----	Dwell
	<i>Used in conjunction with an "F" value</i>
G09 -----	In position contouring One Shot
	<i>Produces a complete machine stop for one single block</i>
G40 -----	Cutter compensation off
	<i>Used in conjunction with D00</i>
G41 -----	Cutter compensation to left
	<i>Used in conjunction with "D" value</i>
G42 -----	Cutter compensation to right
	<i>Used in conjunction with "D" value</i>
G54 -----	Activate punch offset ( <i>modal</i> )
G55 -----	Activate torch offset ( <i>modal</i> )
G56 -----	Activate tap offset ( <i>optional equipment</i> )
G61 -----	In position contouring ON ( <i>modal</i> )
G64 -----	Continuous path mode ( <i>modal</i> )
	<i>Normal operating condition</i>
G70 -or-INCHMODE-----	Inch mode
G71 -or-METRICMODE--	Metric mode
G90 -----	Absolute programming ( <i>modal</i> )
G91 -----	Incremental programming ( <i>modal</i> )
G160 -or-YVUNSYNC----	Parks the V-axis
	<i>(V-axis must be parked to torch, drill, or tap)</i>
G161 -or-YVSYNC-----	Re-synchronizes the Y and V axes
	<i>(Y and V must be synchronized to punch)</i>
G175 -or-REPOSITION--	Reposition command