

G Code Programming

- This is the most common type of programming.
- Fanuc, Haas, Mazak, Mitsubishi, Siemens all use some form of G code programming.
- Four Basic G codes G0 G1 G2 G3
- 3d Printers and CNC routers all use these four basic G codes.
- Can be written as G00 G01 G02 G03
- This extra zero is often called a leading zero. It is not needed and can be a waste of memory space.
- These four G codes are modal which means they stay in the control until changed.
- Only one of them can be active at any one time.

Basic G Codes

- Four Basic G codes G0 G1 G2 G3
- These four G codes are the same on all machines.
- Can be G00 G01 G02 G03
- G0 tells the machine to run all axis at maximum speed.
- It is important to note that a rapid move is not always a straight line.
- Rapid is used to quickly get the tool to the workpiece or away from the workpiece.
- You cannot machine with a rapid command.
- G0 X50. Y50 (Rapid to a position of X50. and Y50.)
- The motor for each axis is not necessarily the same power or maximum speed so you cannot be certain which axis will arrive first.
- A G0 can command X Y and Z all together.