**Basic Terminology**

PCB: Printed Circuit Board

Layers of connation: 2+ in powers of 2

Planes filled with copper traces, pours, etc

Stack up: How you layers are stacked on top of each other Eg. Signal, GND, PWR

Traces: Electrical connections from point to point

Plane areas: A copper pour on a layer, typically for power or GND

Split Planes: A plane split between several nets

Net: Global connection on the whole board

Vias: Electrical connection between planes

Typically thru the whole board but doesn’t necessarily mean that

Partial via is a via that does not go all the way through the board

Blind via: starts on an outer layer and ends on an inner layer

Buried via: starts on an inner layer and goes to another inner layer

Pads: Electrical connection landing pad for components.

Silk screens: Painted on lines for readability, component placement and reference designators

Reference designators: An indicator on the board indicating the component, usual a letter followed by a number

Important note: Always have a pin 1 indicator

Solder mask: a non-conductive mask that covers the copper traces on the board. Usually green.

Paste mask: Opening solder paste.

GERBER Files: The end generated CAM files used for printing the PCB

**Common Component Terms**

0603 and 0402: Typically but not limited to Resistors and capacitors

BGA: Ball Grid Array

QFP: Quad Flat Pack

SMT: Surface mount

Thru Hole