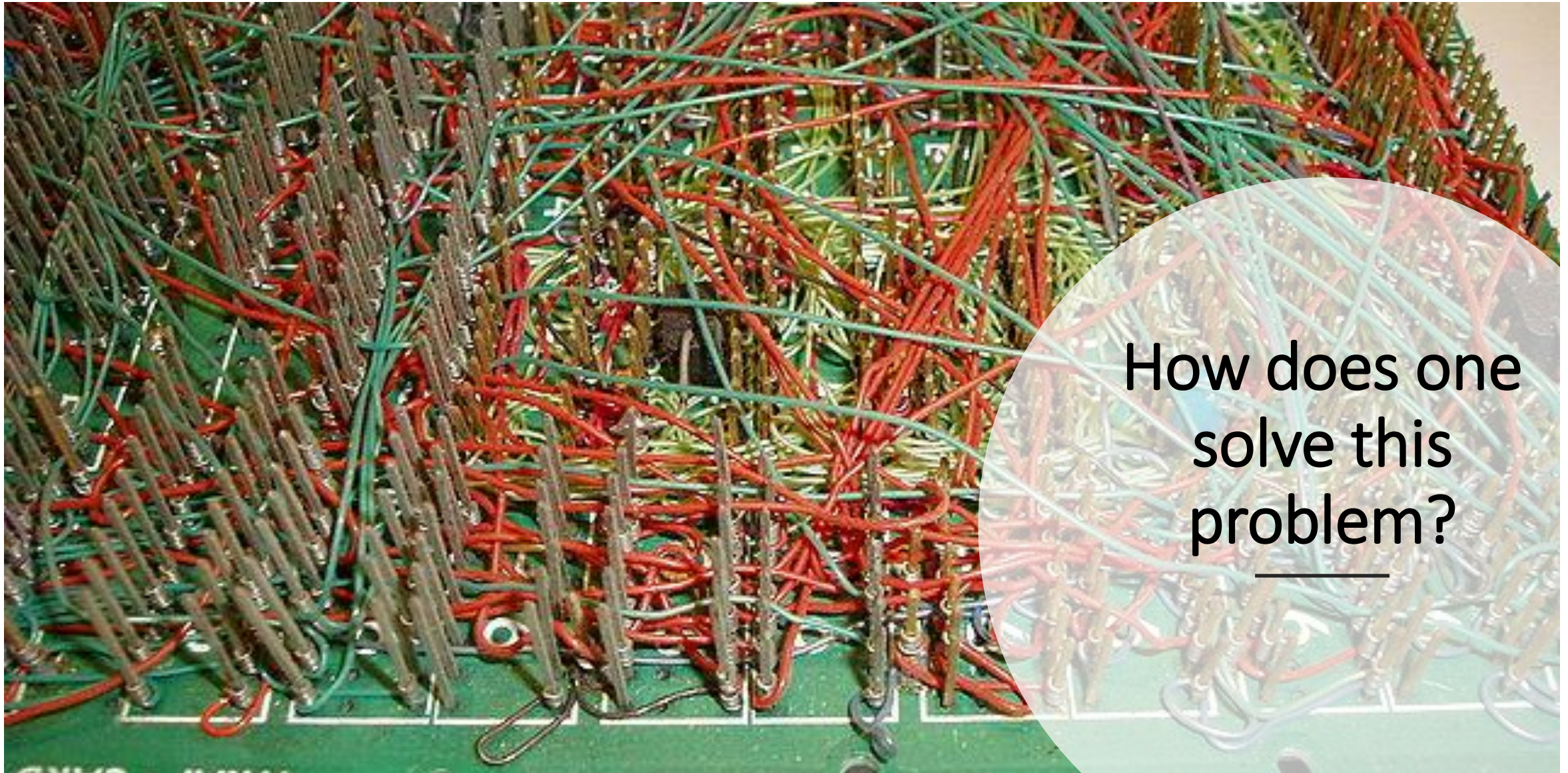




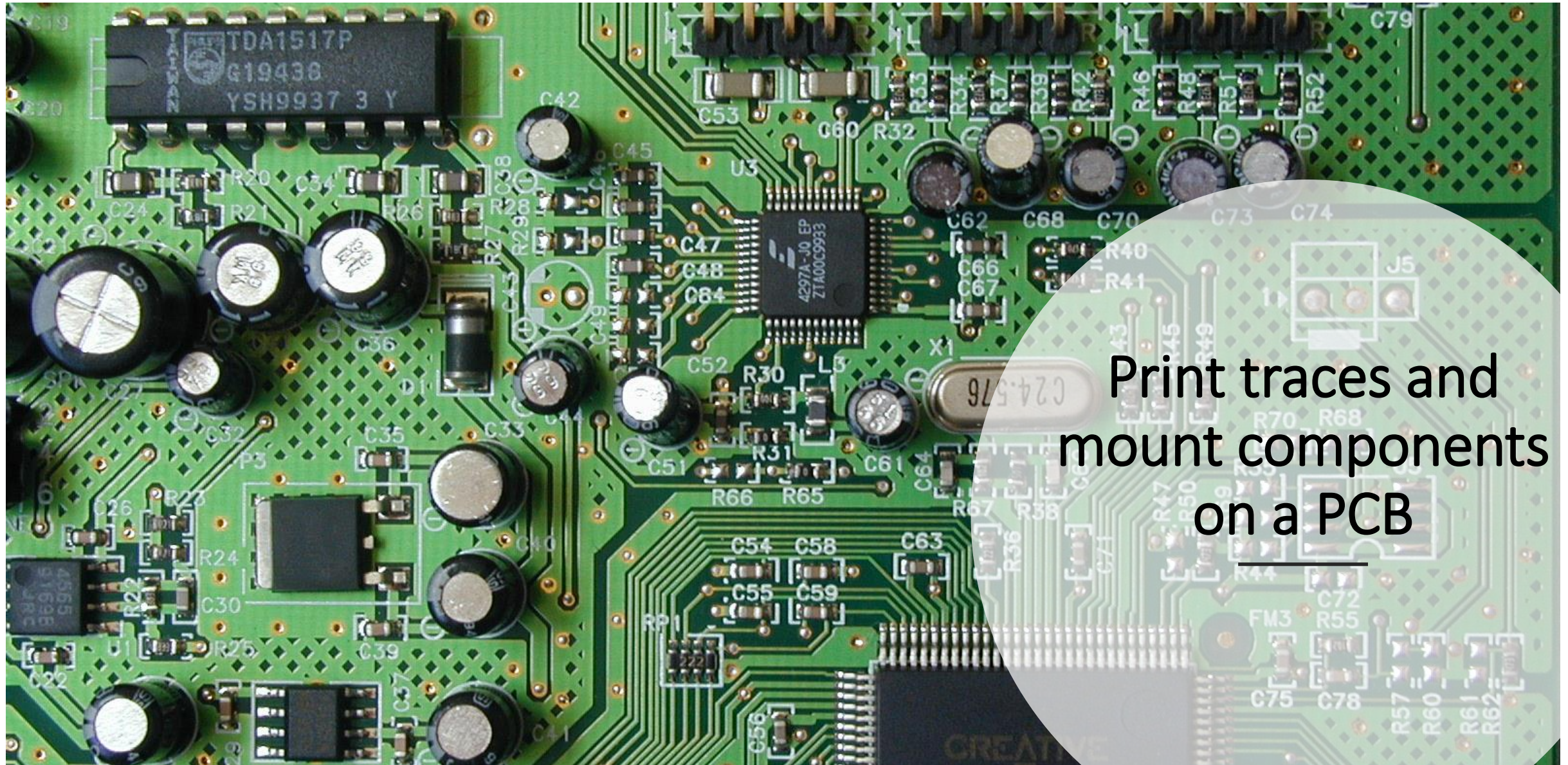
UCSB ENGINEERING
capstone

Introduction to Embedded Design and Printed Circuit Boards (PCB's)

By Ilan Ben-Yaacov

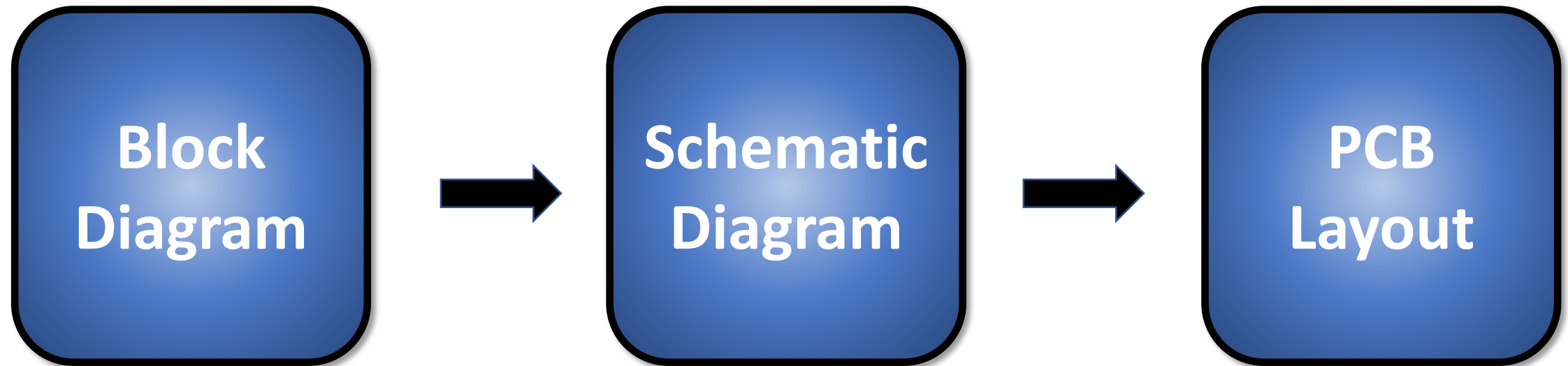


How does one
solve this
problem?

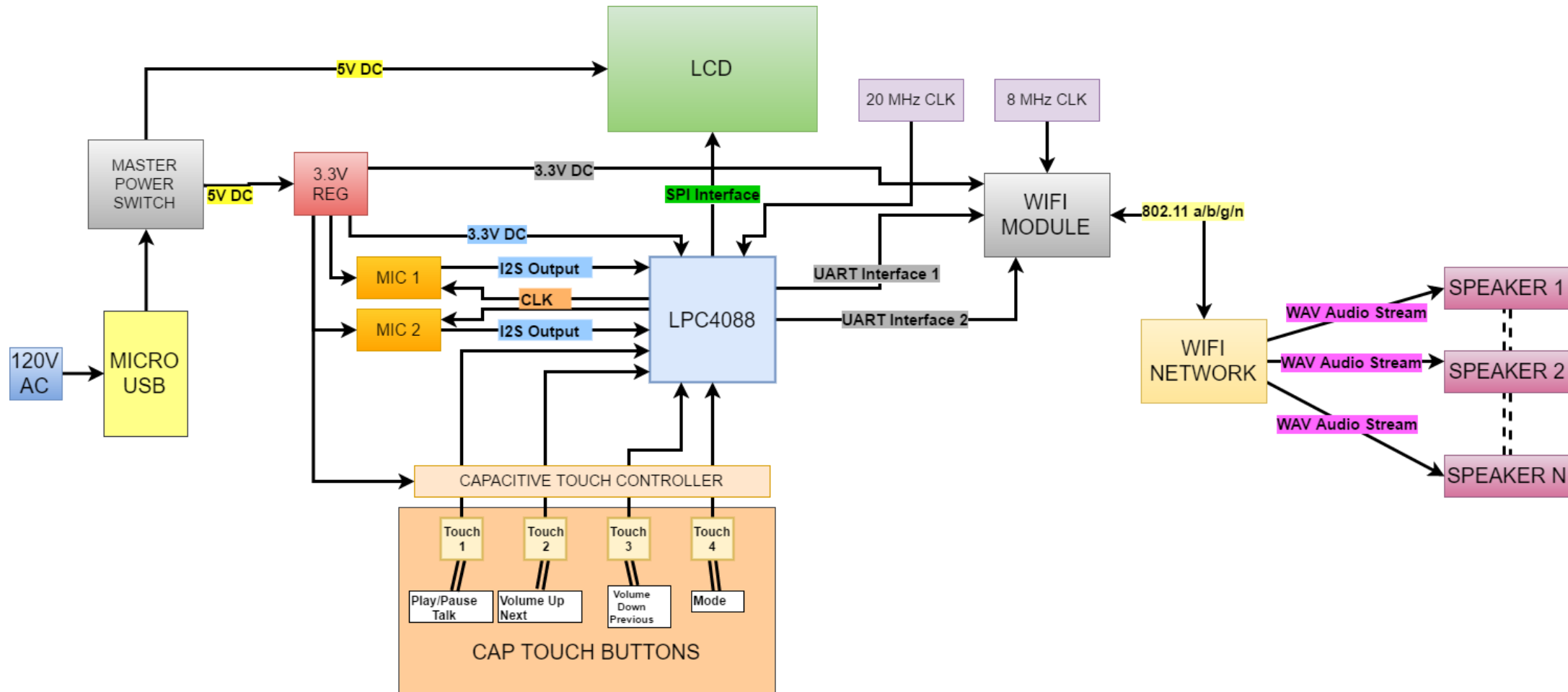


Print traces and
mount components
on a PCB

Embedded Design:



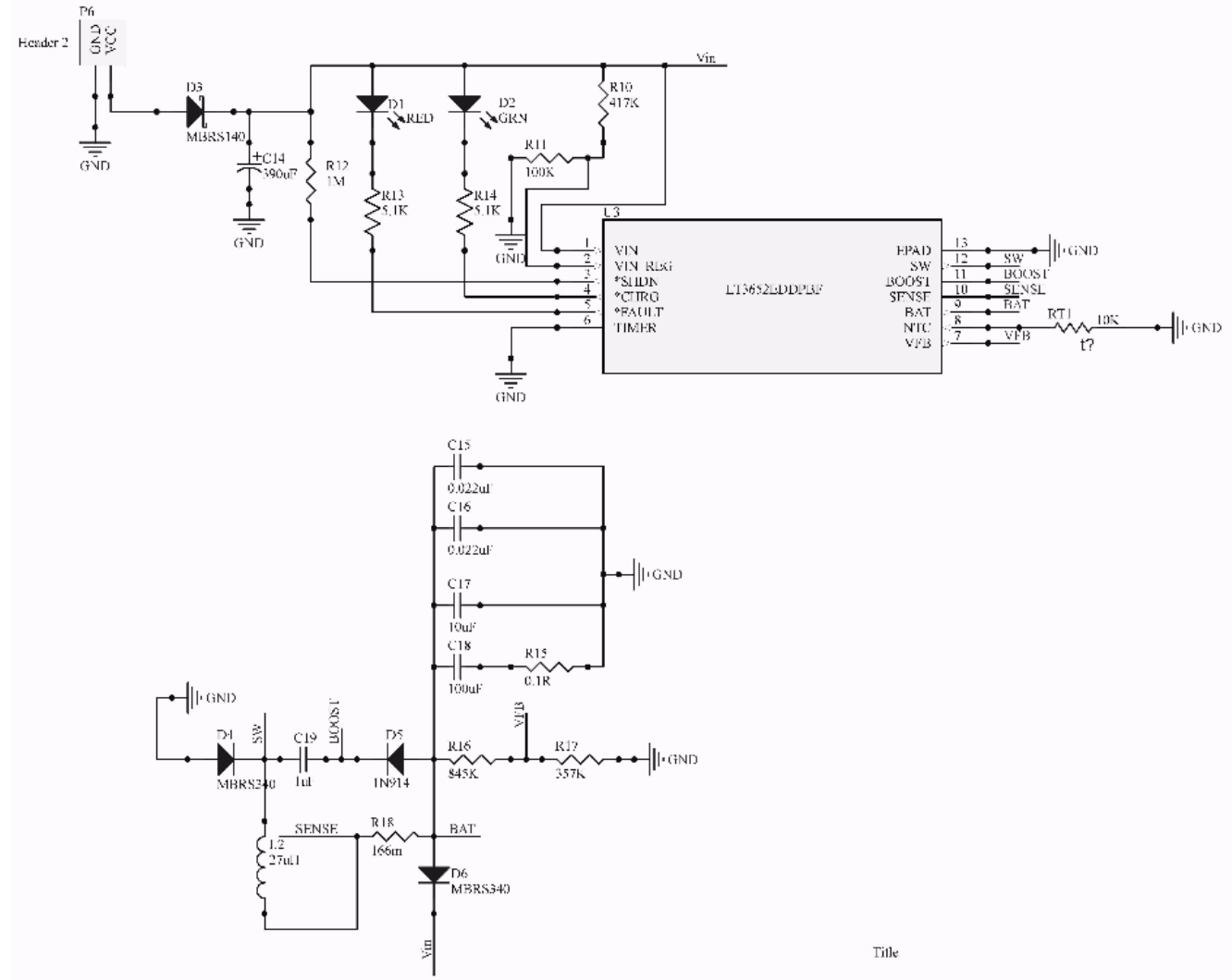
Step 1: Block Diagram



Once your block diagram is complete:

- Select components based on defined specs
- Prototype (development kits)
- Create a full schematic diagram

Step 2: Schematic Diagram

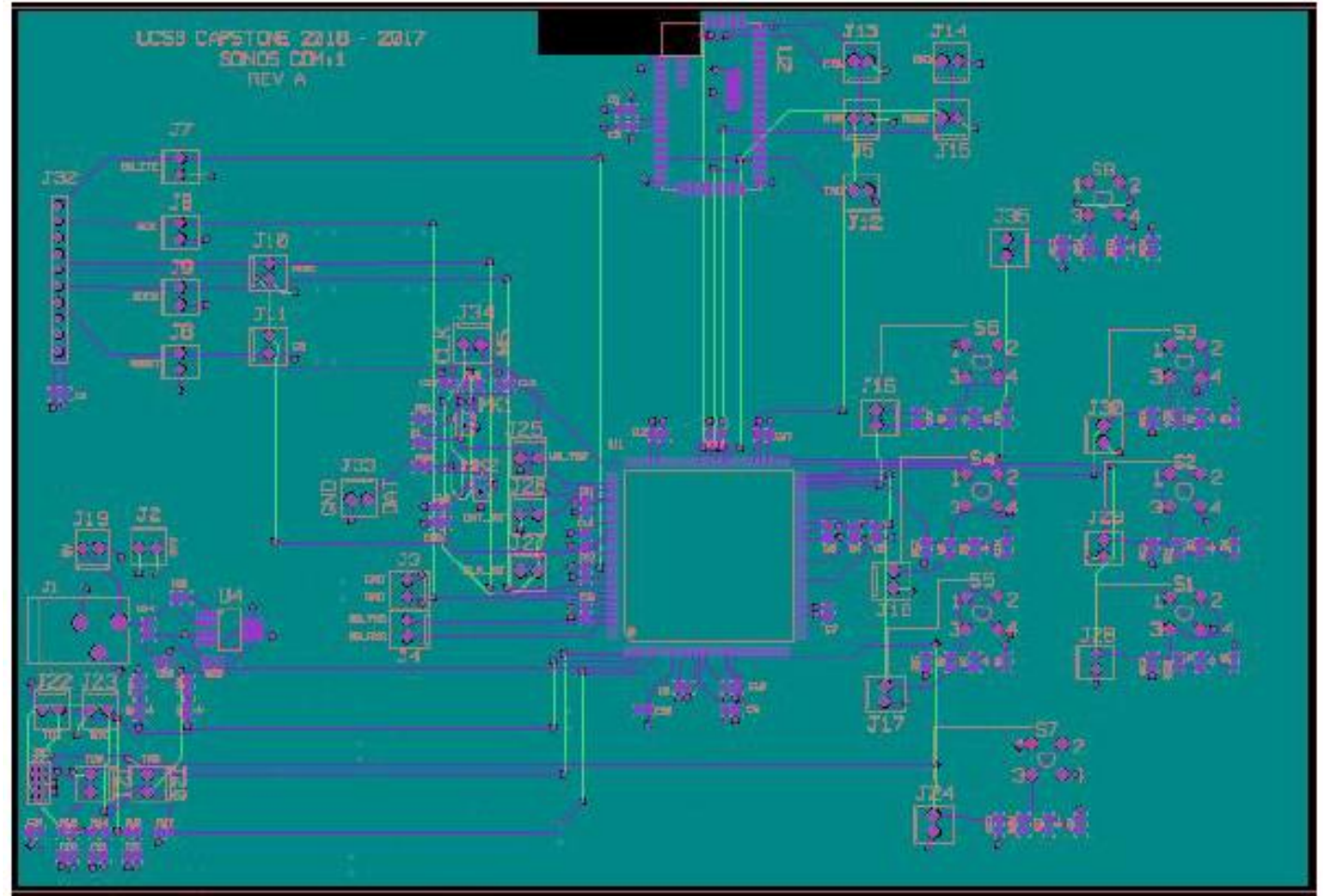


Title

Schematic Diagram:

- Collection of parts (represented by logic symbols) and their interconnections (represented by nets)
- Defines every node of every part as well as every interconnection between nodes
- Often used as primary view of a design
- Schematic diagram is mapped to and defines your PCB design

Step 3: PCB Layout



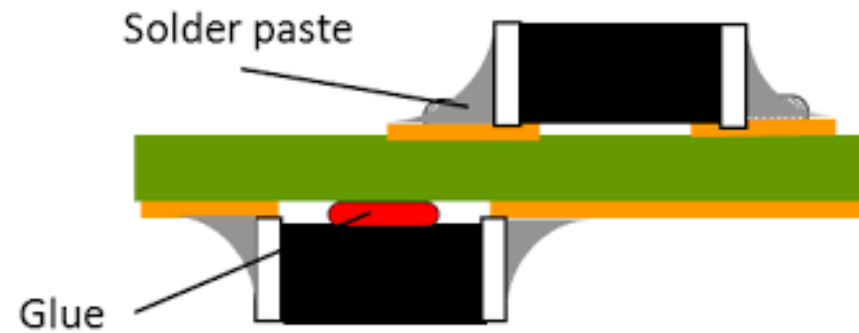
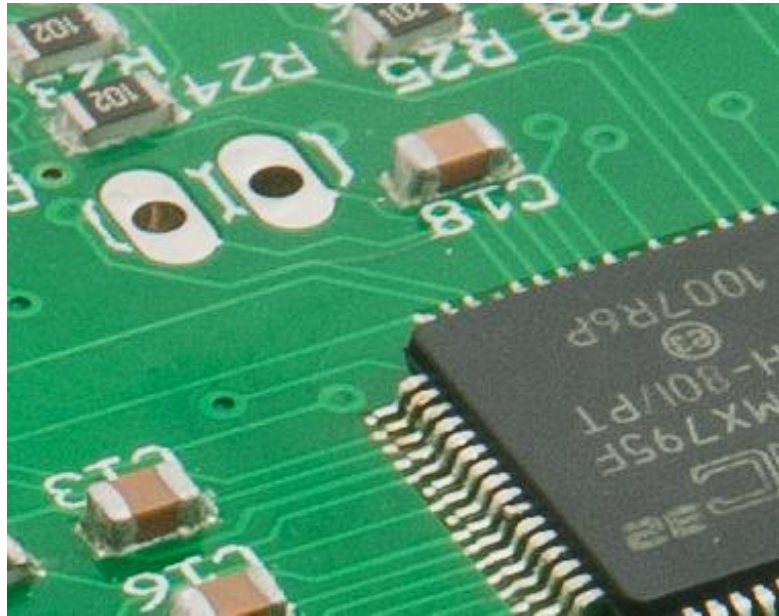
Embedded Design: Process Flow

- I. Schematic Design (i.e., schematic diagram)
- II. Schematic Capture
 - a. Create computer readable netlist
 - b. Create bill of materials (B.O.M.)
- III. Physical PCB Layout / Design (Mentor, Eagle,...)
- IV. Board Fab
- V. Assembly

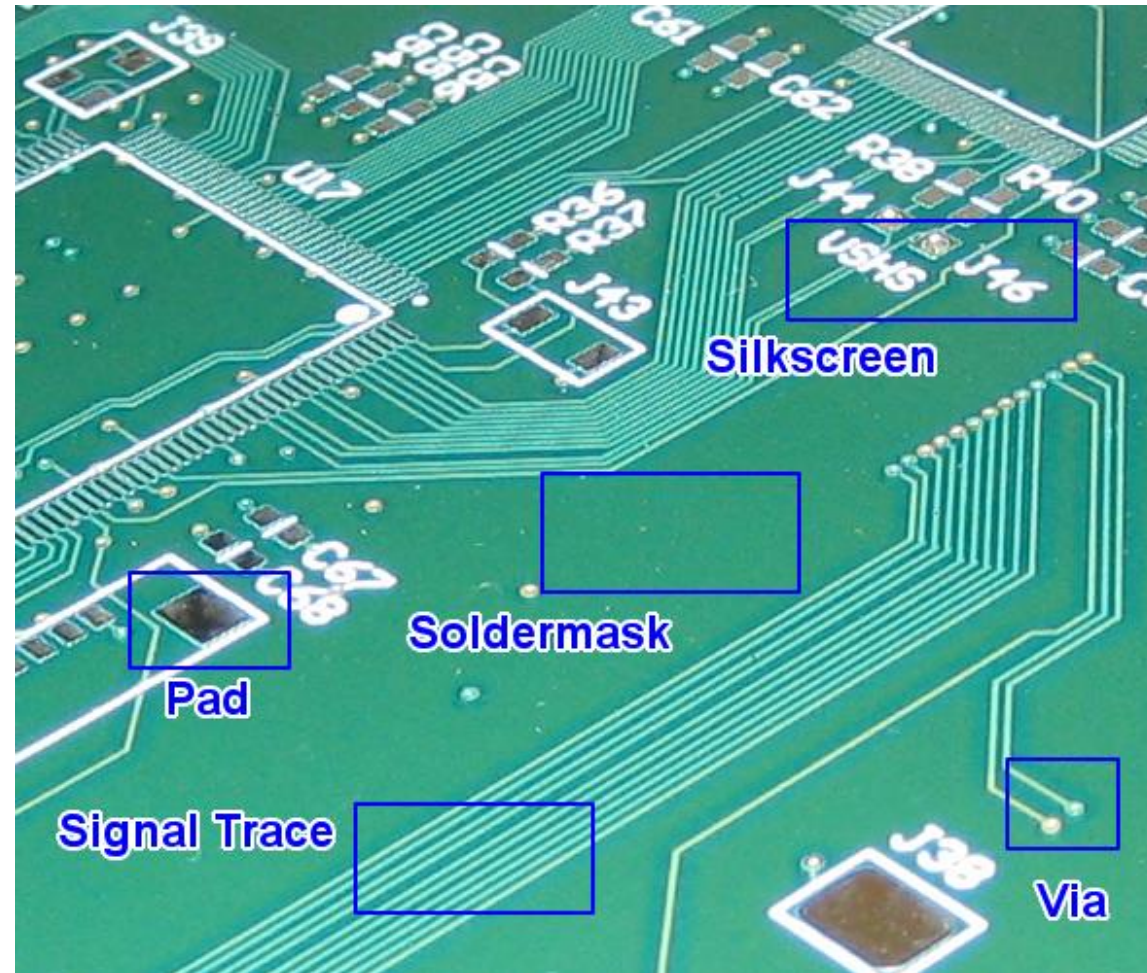
PCB Overview:

- PCB is like a layer cake of alternating layers laminated together with heat and adhesive
- Typical PCB includes one or more insulating (fiberglass layers) with metal (copper) layers in between and on either side
- Number of layers of a board (e.g., 2-layer board, 4-layer board,...) refers to the number of copper layers
- Copper layers are patterned to form traces, pads, etc.
- Surface mount (SMD) components soldered directly to pads

Surface Mount Components on a PCB:



- Pad
- Signal trace
- Soldermask
- Silkscreen
- Via



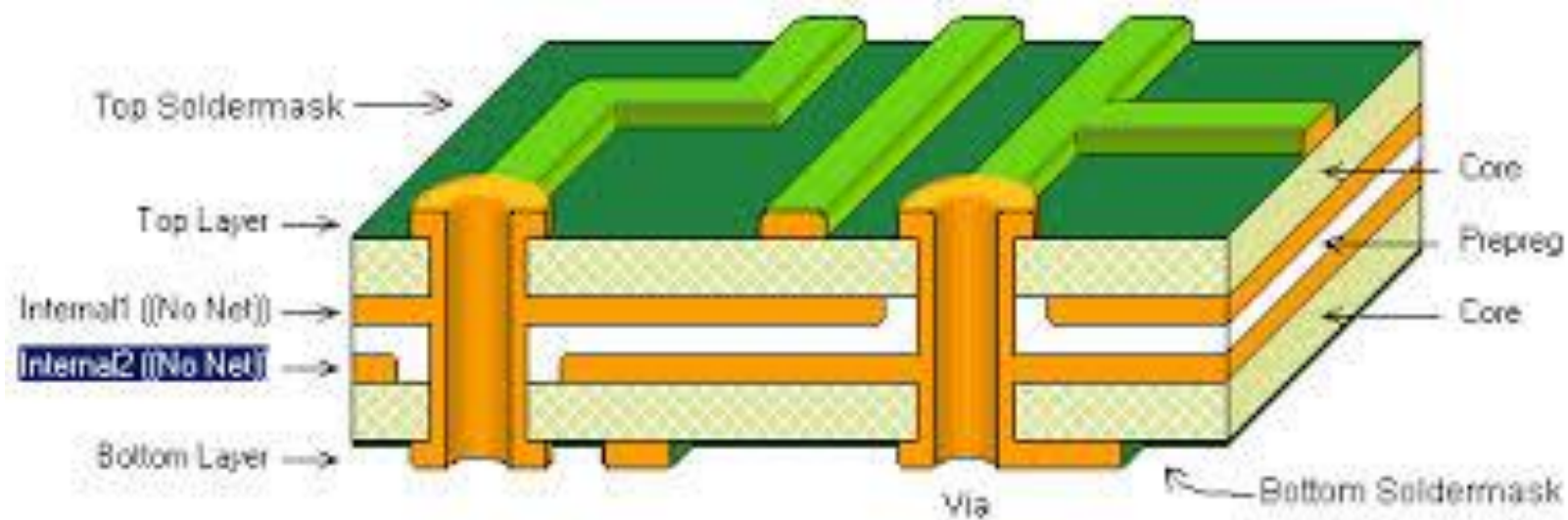
PCB Overview (cont.):

- Soldermask (green layer): Covers outer copper layers to prevent shorting of copper traces to other metal, solder, or conductive bits
- Silkscreen: Adds letters, numbers, and symbols to aid in assembly and identification of parts/regions
- Via holes: Allow for connections between layers or connect components to buried layers

4-Layer PCB Boards:

- 4-layer board technology
 - Top and bottom layers for signal routing
 - Middle (inner) layers for power and ground planes (inner planes typically are mostly solid copper)
- In cases where components are all mounted on top (one side), the top side is referred to as the “component side” and the bottom side is referred to as the “solder side”

Cross-sectional view along a via hole:



Multi-Layer PCB Construction:

- 6-layer board (at cross-section of through hole)
 - Made from 2 cores and 3 layers of prepreg (fiberglass cloth)
 - After stacking, panel is pressed and fused together
 - Any traces or copper features are embedded into the prepreg, making one final smooth, flat assembly

