Introduction to Printed Circuit Board (PCB) Design

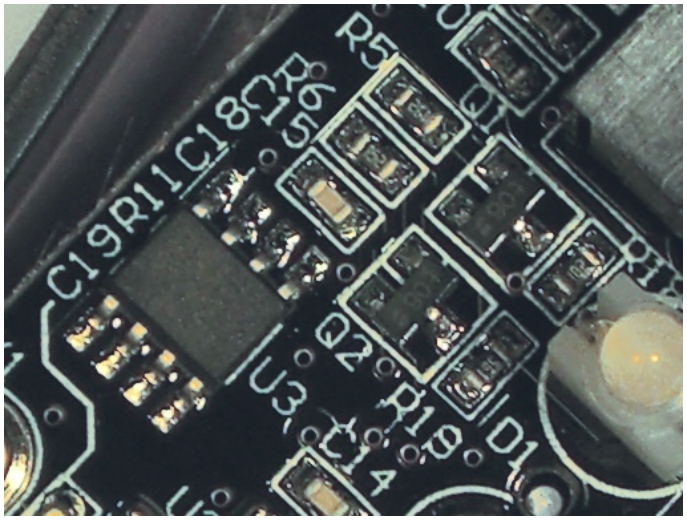
Mark Colton

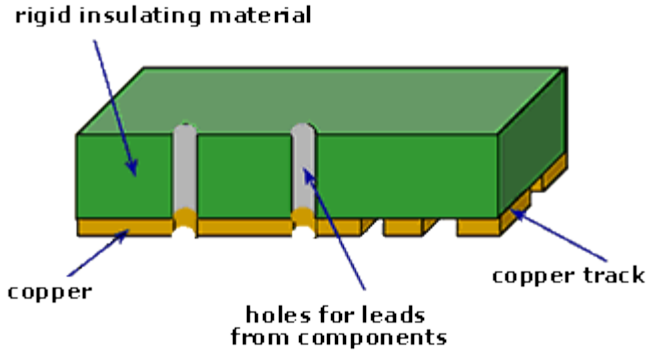
Department of Mechanical Engineering

Brigham Young University

Printed Circuit Boards (PCBs)

- Board for supporting and connecting electronic components
- Most modern circuits are implemented on PCBs
- Alternating layers of insulator and copper
- “Printed” refers to the process of removing copper to form “traces” or “tracks”

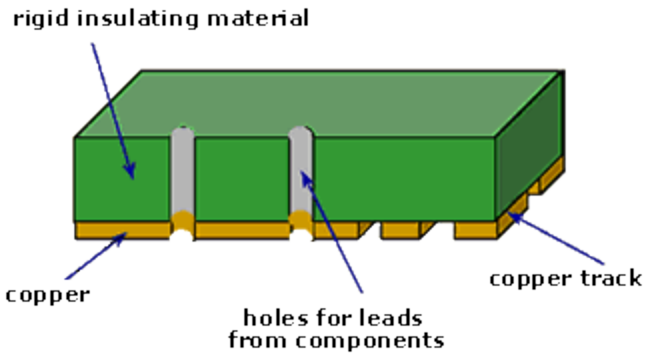




http://www.ami.ac.uk/courses/ami4809_pcd/unit_01/

Single-Sided PCBs

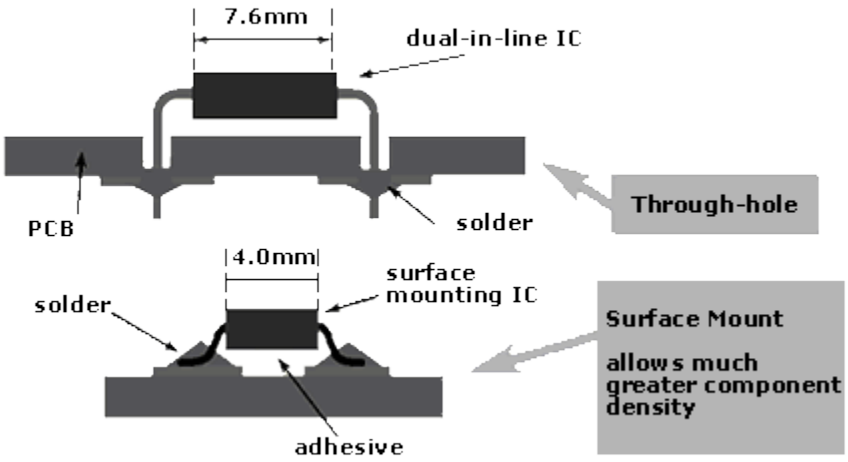
- All copper is on one side of the board
- Components typically reside on the insulated side of the board
- Leads pass through “through holes” or “vias” to the copper



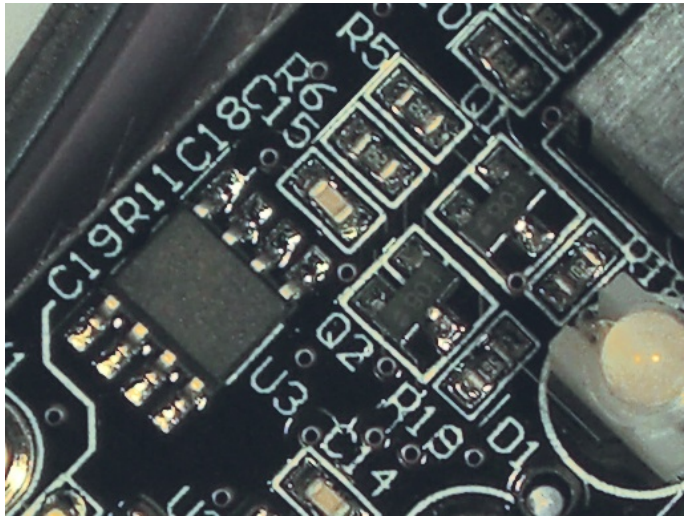
http://www.ami.ac.uk/courses/ami4809_pcd/unit_01/

Through-Hole vs. Surface Mount

- Components come in a huge variety of packaging
- Roughly divided into *through-hole* and *surface mount*
- Surface mount technology (SMT) is rapidly replacing through-hole
- Advantages of SMT:
  - Smaller components
  - Higher component density
  - Lower cost
  - Fewer holes
  - Simpler automated assembly
  - Facilitates multilayer boards
  - Etc.

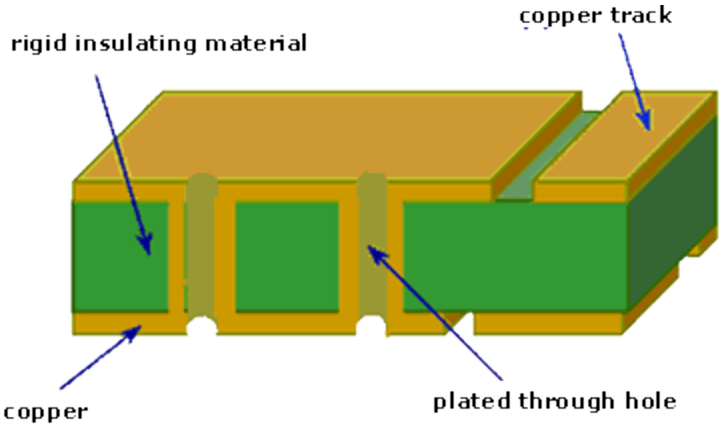


http://www.ami.ac.uk/courses/ami4809_pcd/unit_01/

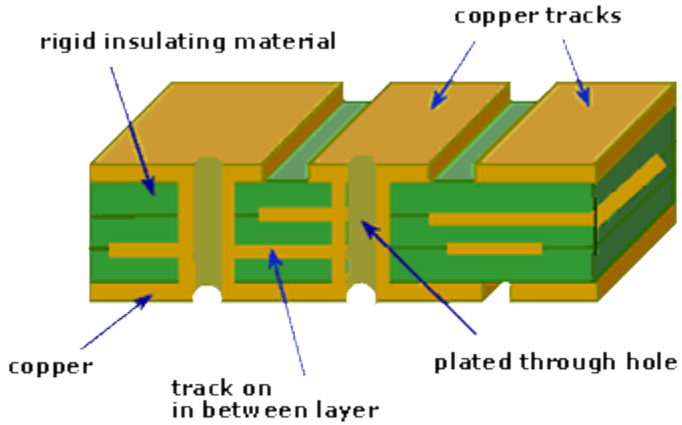


Multilayer PCBs

- More complicated circuits often require more than one layer (up to 32!)
- Intermediate layers connect components or provide power and ground planes
- Leads pass through “through holes” or “vias” to the various layers
- Plated vias are convenient for connecting layers



http://www.ami.ac.uk/courses/ami4809_pcd/unit_01/



http://www.ami.ac.uk/courses/ami4809_pcd/unit_01/

Pros & Cons of PCBs

- Pros
  - Robust
  - Repeatable (manufacturing & assembly)
  - High circuit performance
  - Mounting options
  - Design for noise
  - Consistent circuit performance
  - New: Rapid prototyping options
- Cons
  - Difficult to modify
  - Cost for one-shot devices

Many engineers are going straight to PCBs for prototypes, bypassing breadboards entirely!

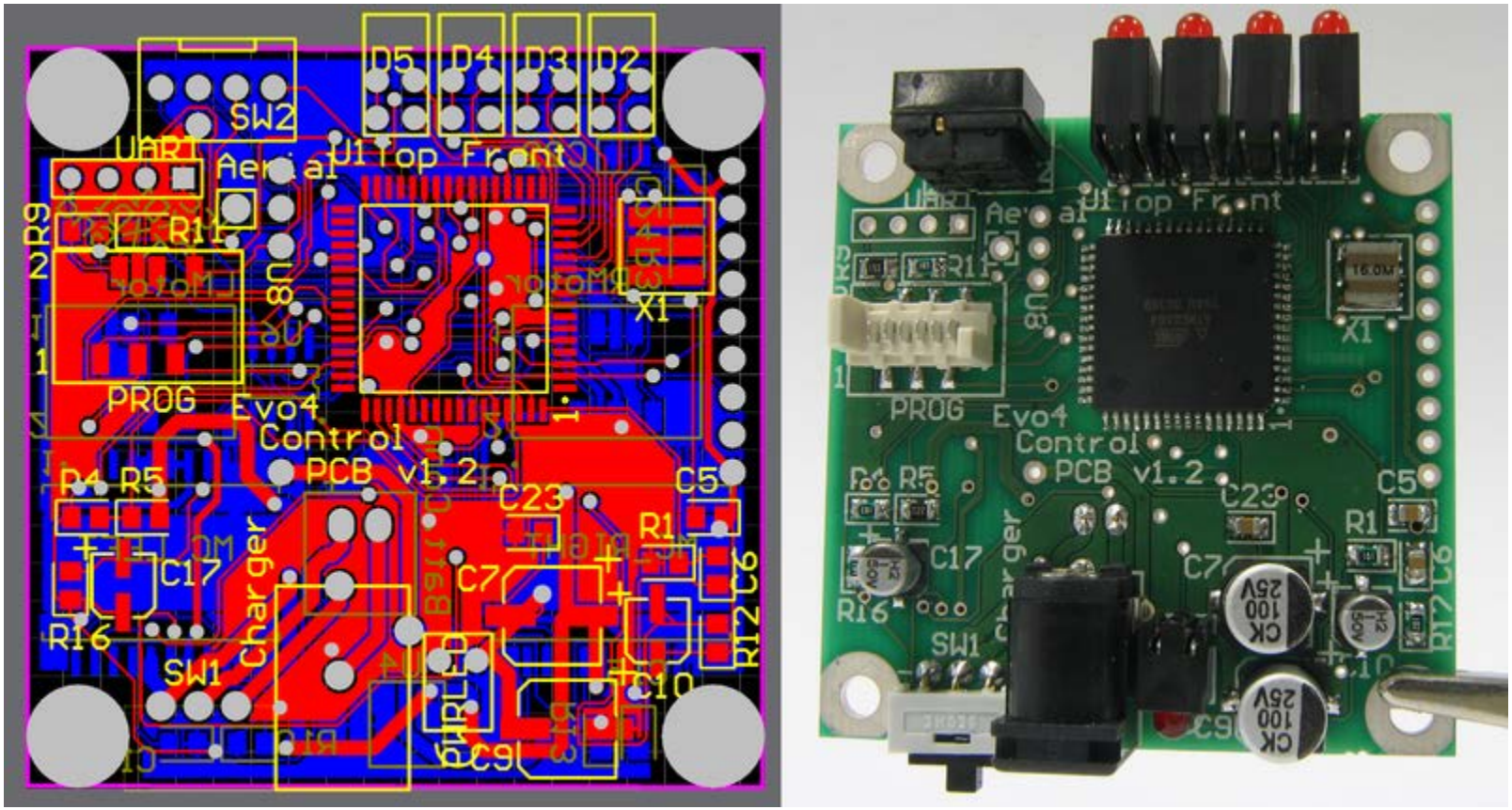
PCB Design

- Design circuit
- “Spec” (select) components
- “Capture” schematic (computer-based design)
- Create PCB footprint
- Place components on footprint
- Create routes/traces/tracks
- Add other layers
  - Silkscreen (component outlines, designators, text)
  - Solder mask/resist (polymer coating to prevent solder bridges between pins)
- Check and re-check
- Generate “Gerber” files
- Send to manufacturer
- Check and re-check
- Populate
- Test
- Iterate

Manufacturing Options

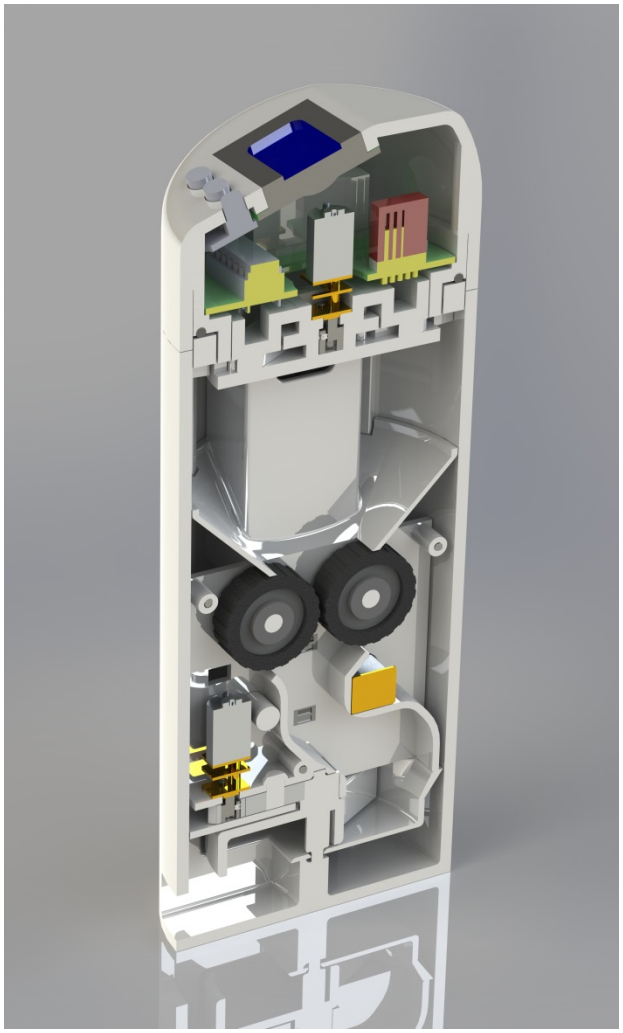
- Milling
  - Copper removed mechanically
  - Minimum size of traces and spaces limited by mill size and resolution
  - Not appropriate for large production runs
- Etching
  - Copper removed chemically using masks
  - Used for production boards and some prototypes

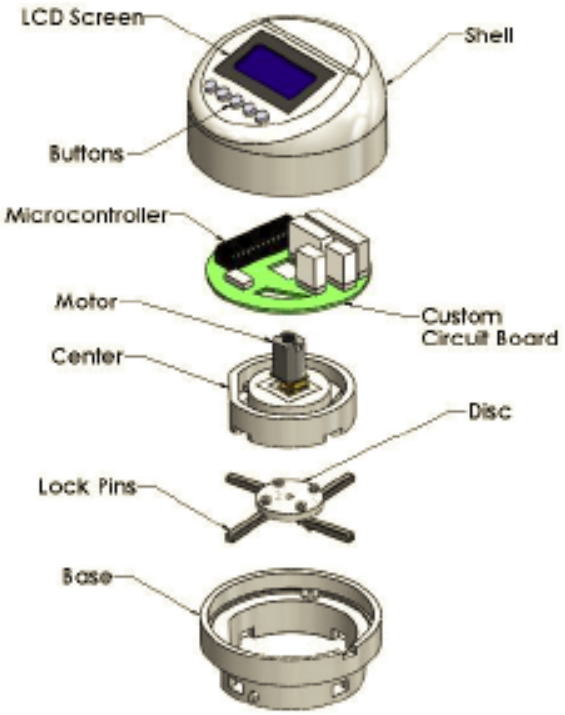
An Example PCB



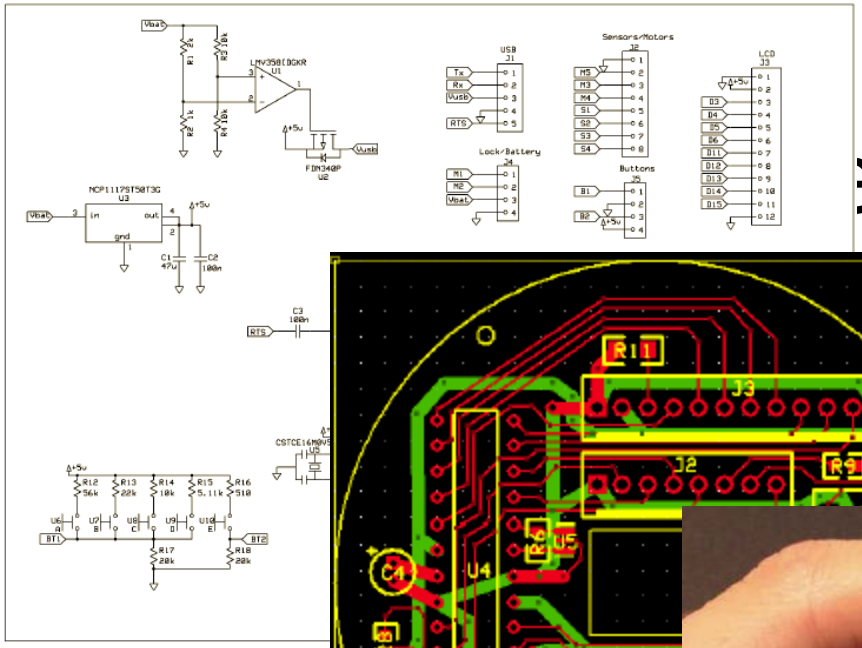
http://en.wikipedia.org/wiki/Printed_circuit_board

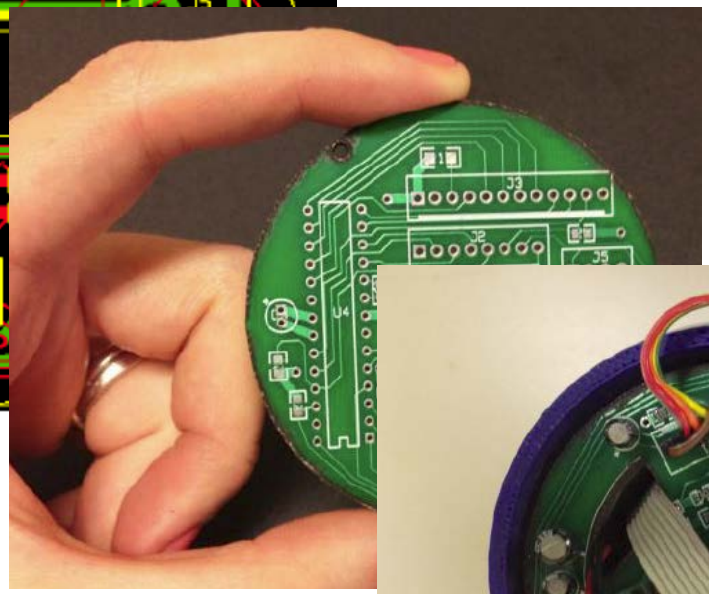
MedVault

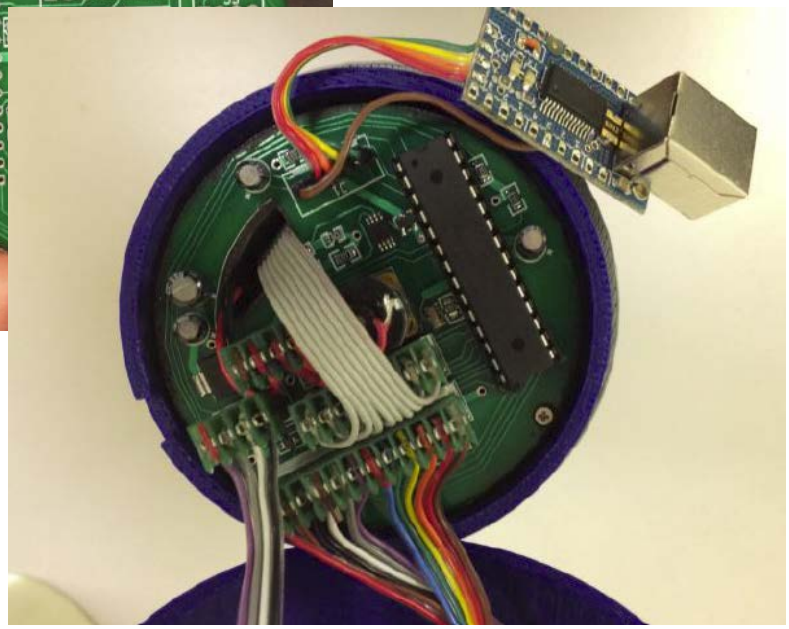



LCD Screen
Shell
Buttons
Microcontroller
Motor
Custom
Circuit Board
Center
Disc
Lock Pins
Base

edVault


Vbat
LMV358IDGKR
U1
USB
J1
Tx
Rx
Vusb
RTS
Sensors/Motors
J2
M5
M3
M4
S1
S2
S3
S4
LCD
J3
Lock/Battery
J4
M1
M2
Vbat
Buttons
J6
B1
B2
NCP1117ST50T3G
U3
in
out
gnd
C1
47u
C2
100n
C3
100n
CSTCE16M0V5
U5
R1
R2
R3
R4
R5
R6
R7
R8
R9
R10
R11
R12
56k
R13
22k
R14
10k
R15
5.11k
R16
510
R17
20k
R18
20k
BT1
BT2
J1
J2
J3
J4
J5
U1
C4



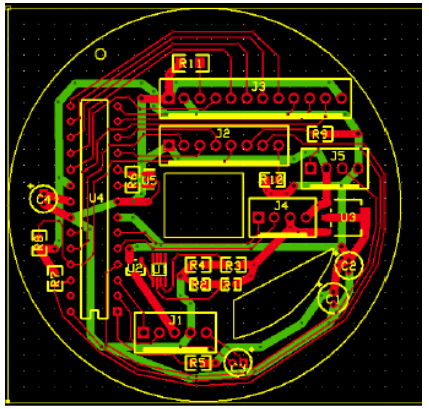


Tips/Tricks/Rules

- Trace width
  - Thin traces have higher resistance -> carry less current, greater voltage drops, and generate more heat
  - Use thicker traces
  - Can specify down to 4 mil
  - Inner layers need greater thickness
- Trace length
  - Longer traces are like antennas and capacitors
  - Minimize trace length where possible

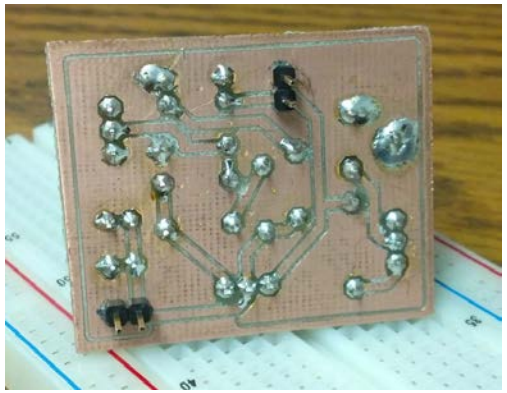
Tips/Tricks/Rules

- Component placement and routing
  - Use a snap grid
  - Put all components onto the board before arranging them
  - Route critical tracks first
  - Divide and place components into functional building blocks where possible
  - Use 45 degree angles
  - Leave unused copper in place
  - Ground unused copper



Tips/Tricks/Rules

- Ground planes
  - For multilayer boards, make one layer a ground plane (can do the same for a power plane)
  - For single layer boards, can leave as much material as possible for a ground plane



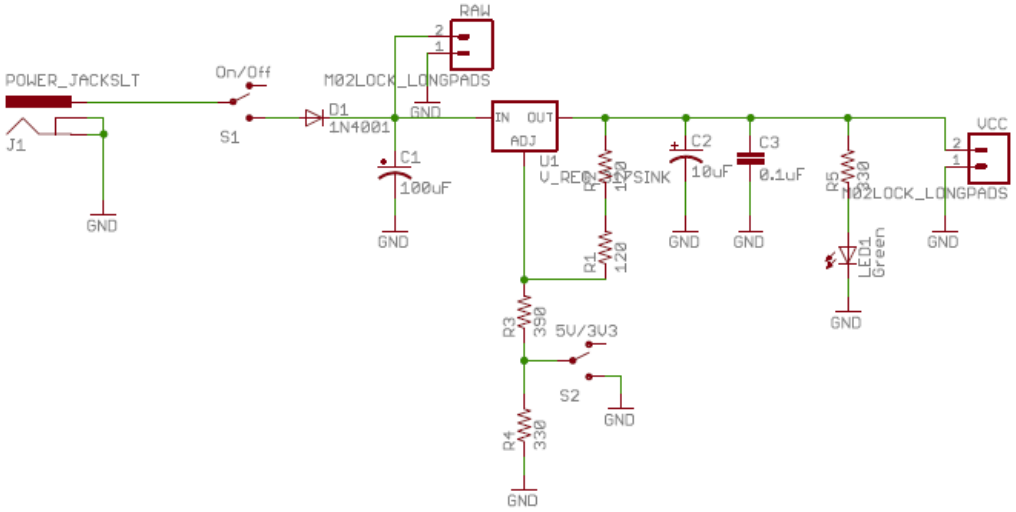
EAGLE

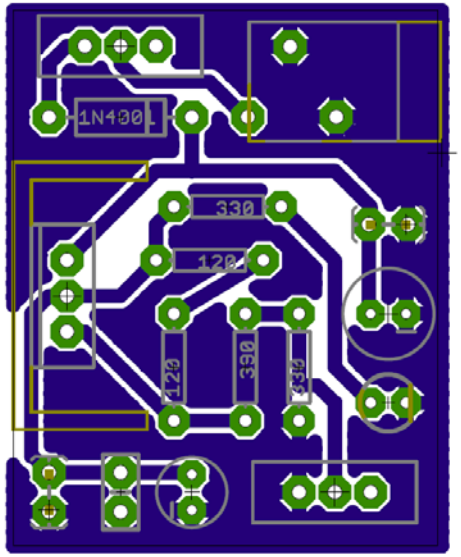
- A schematic capture and PCB design software
- FREE!
- Can download and import components from manufacturers and vendors
  - Get the right “footprint”
- Follow the tutorials in Lab 1
- Use again in Lab 1 to design a power supply

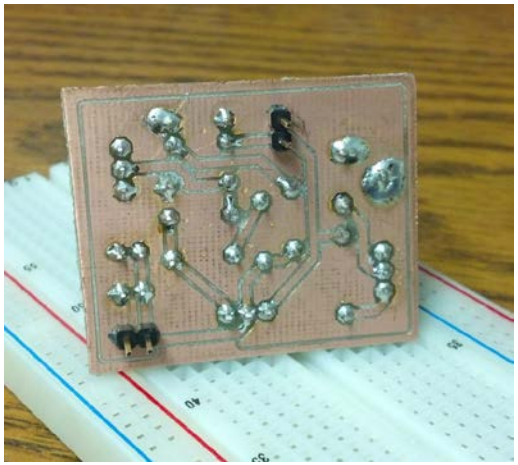
Lab 1

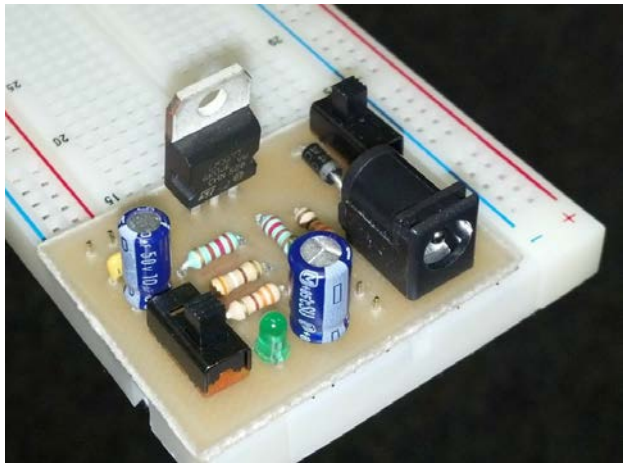
- You will design a power supply, using an LM317 voltage regulator, that outputs 3.3 V or 5 V (switchable) (awesome and useful!)
- But first:
  - Complete two unrelated tutorials related to PCB design using EAGLE software
  - Read through the tutorial on power supply design
  - Understand the LM317 data sheet
  - Generate a preliminary design for your voltage regulator circuit
  - Look at available parts on Learning Suite

Lab 1 Example & Demo





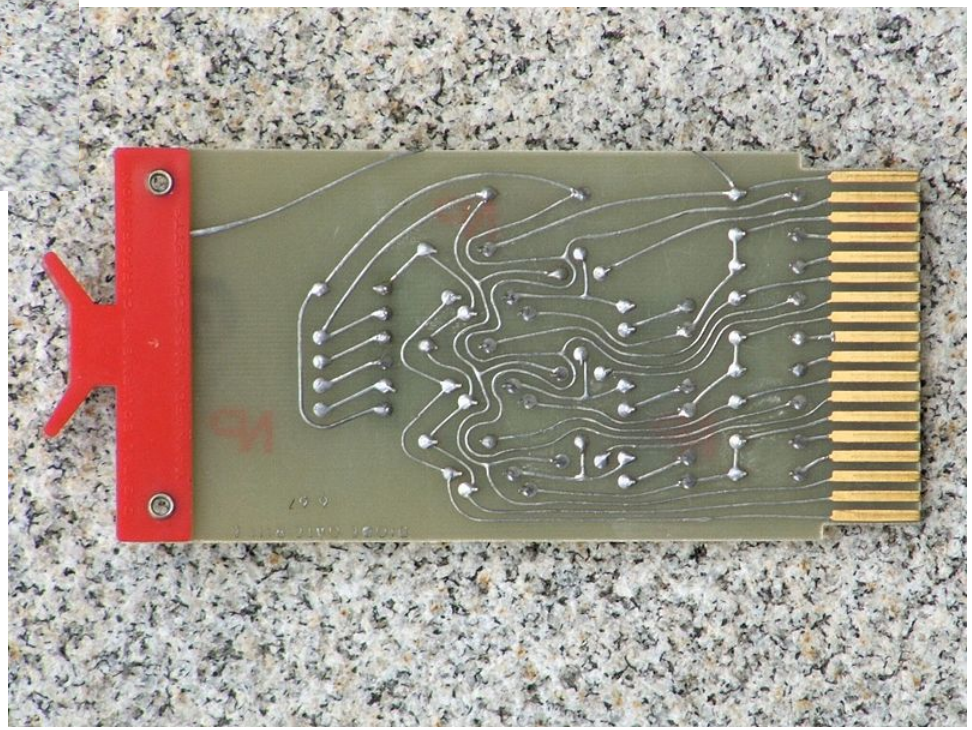




Our Circuit Boards

- Milled in EE shop
- 10 mil trace/space (you'll sometimes see this listed as 10/10)
- Non-plated vias
- 15 mil drill

http://en.wikipedia.org/wiki/File:S111FlipChipTop.jpg



http://en.wikipedia.org/wiki/File:S111FlipChipBack.jpg