Printed Circuits Board (PCB) Design

ELC 4438: Embedded Systems Design
Spring 2016

What is a PCB?

A Mechanical Solution to an Electrical Problem
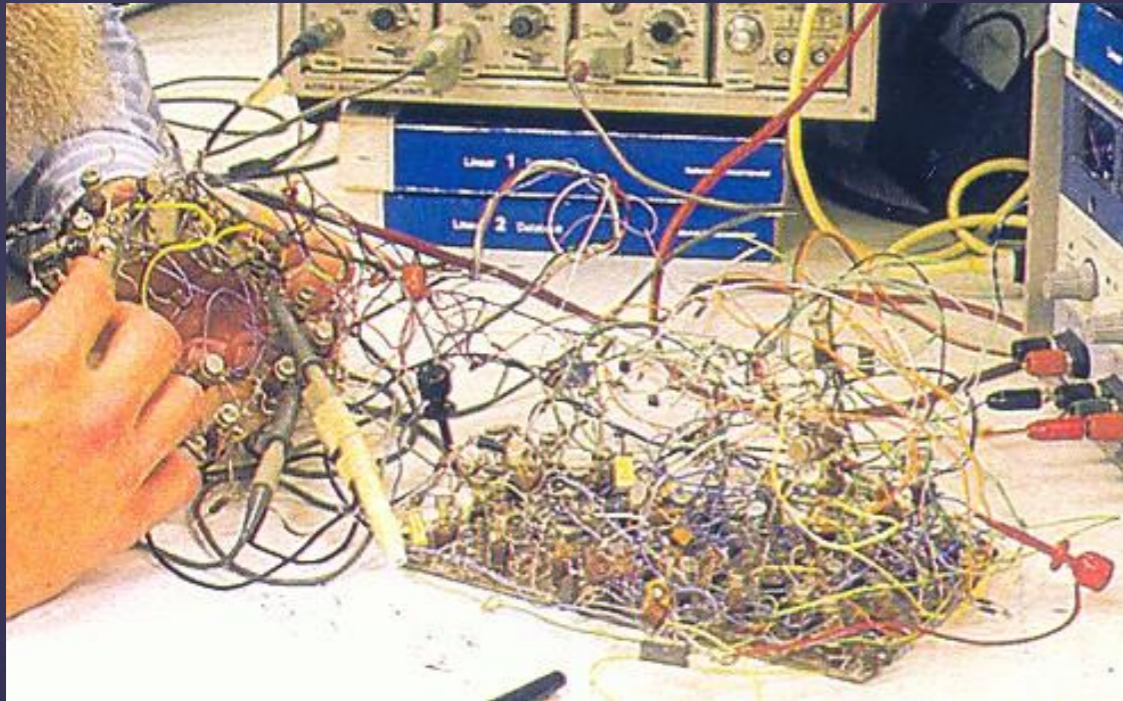

Photo from hephaestusaudio.com

Photo from www.sparkfun.com

What is a PCB?

Layers, Traces, Planes, and Vias ...
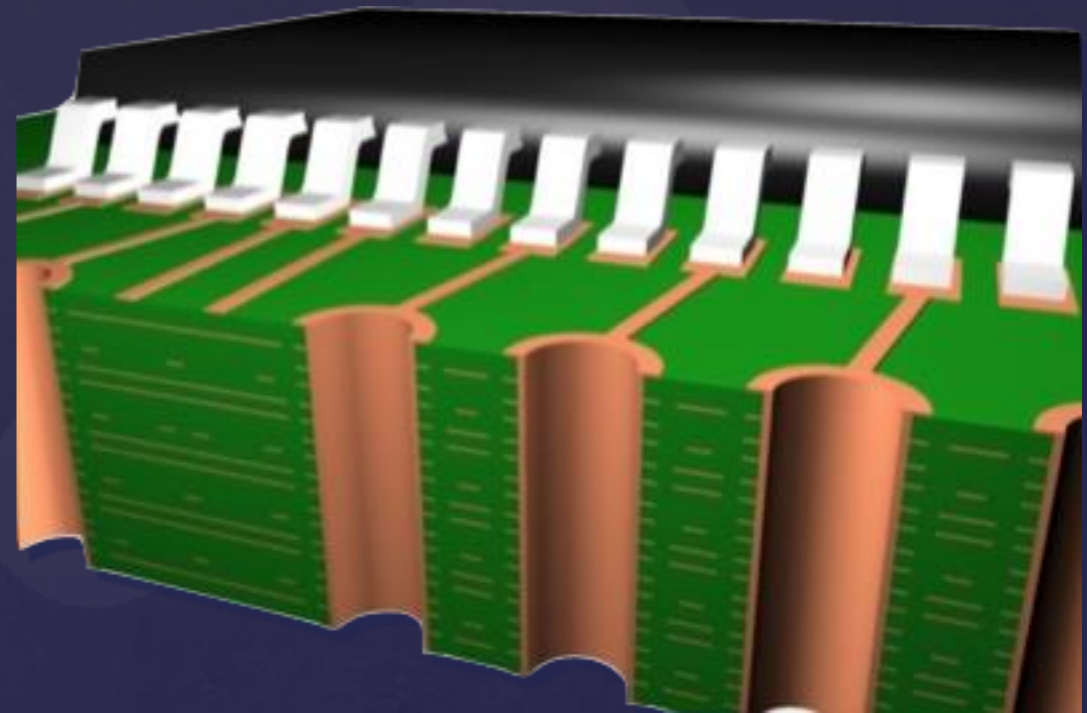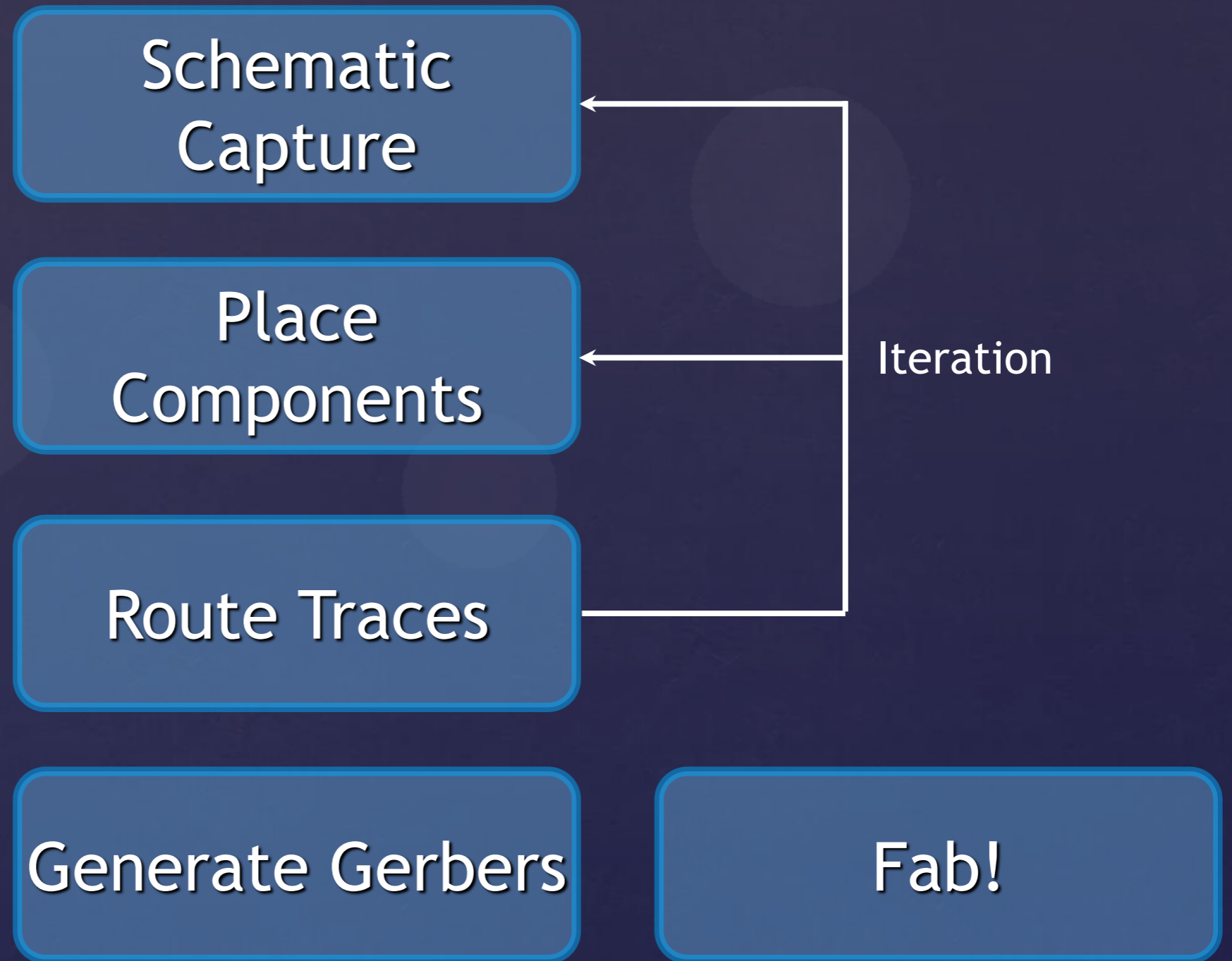

Image from www.elkosoft.com

What is a PCB?

Essentially, your job is this …

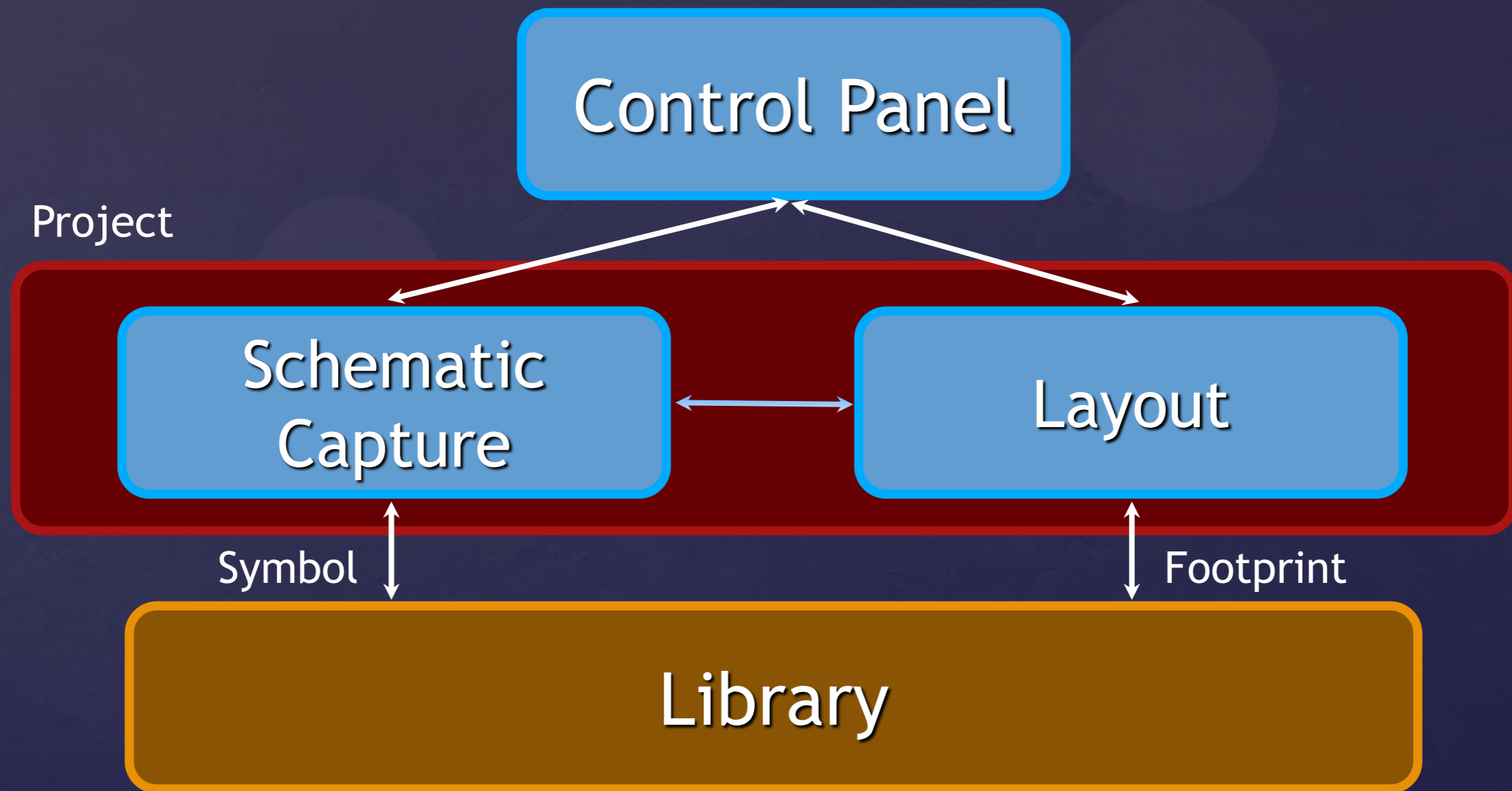
Image from www.gettyimages.com

The PCB Design Process

From Vision To Reality

Schematic Capture

Place Components

Route Traces

Iteration

Generate Gerbers

Fab!

PCB Design Software

A CAD Tool For Designing a Board

PCB Design Software

Symbol + Footprint + Device = Component

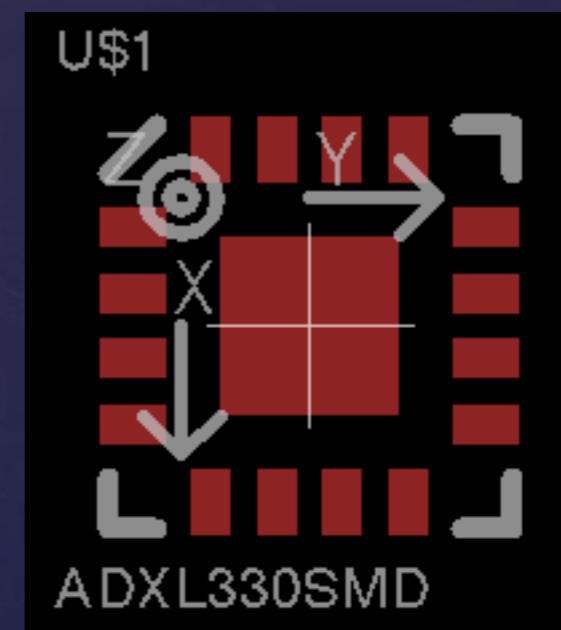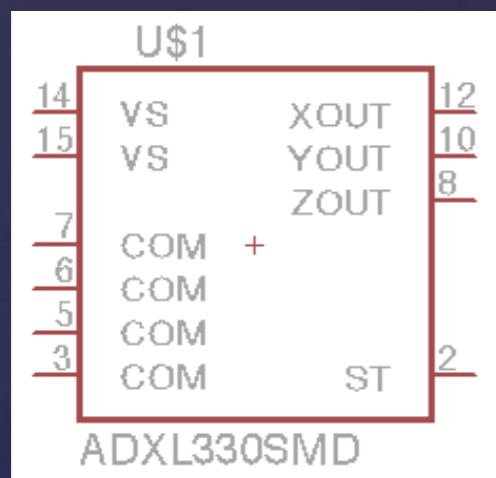| Symbol	Device	Footprint
Sym	*Dev*	*Pac*

PCB Design
Stuff I Need To Get Started

- A Plan
    - Block Diagram
    - Component Selection
    - Connections and Testing Considerations
    - Power and Performance Considerations
- A New Project
- Access to Libraries

PCB Design

It's all about the planning.

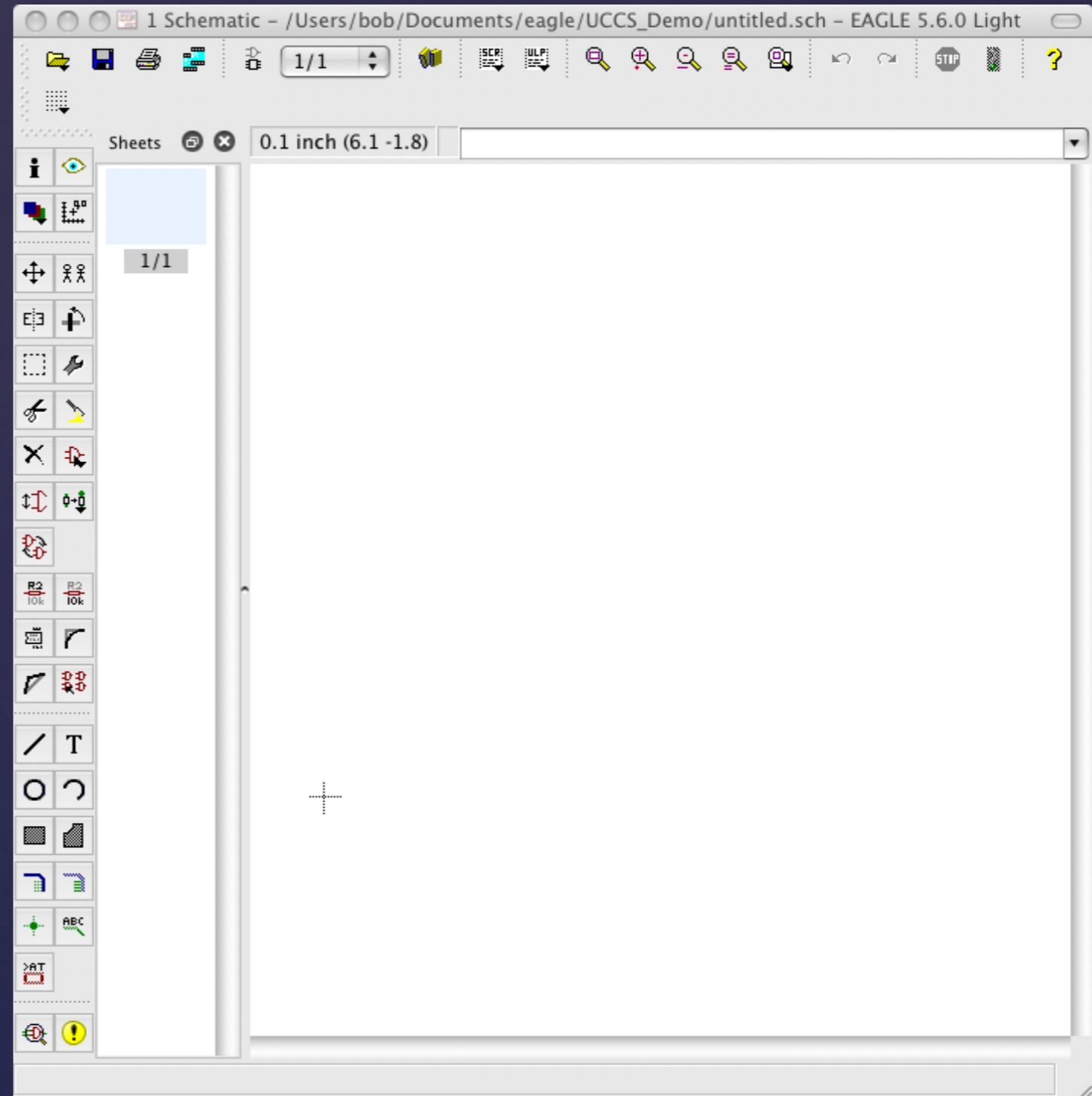| Components	Connections	Power & Performance
Passives: 0805, 0603, etc.	Mechanical Connections	Power Ratings
IC Packages: QFN, TQFP, etc.	Bus Connections	Battery Performance
Libraries	PC Connections	High-Speed / High-Sensitivity

Schematic Capture

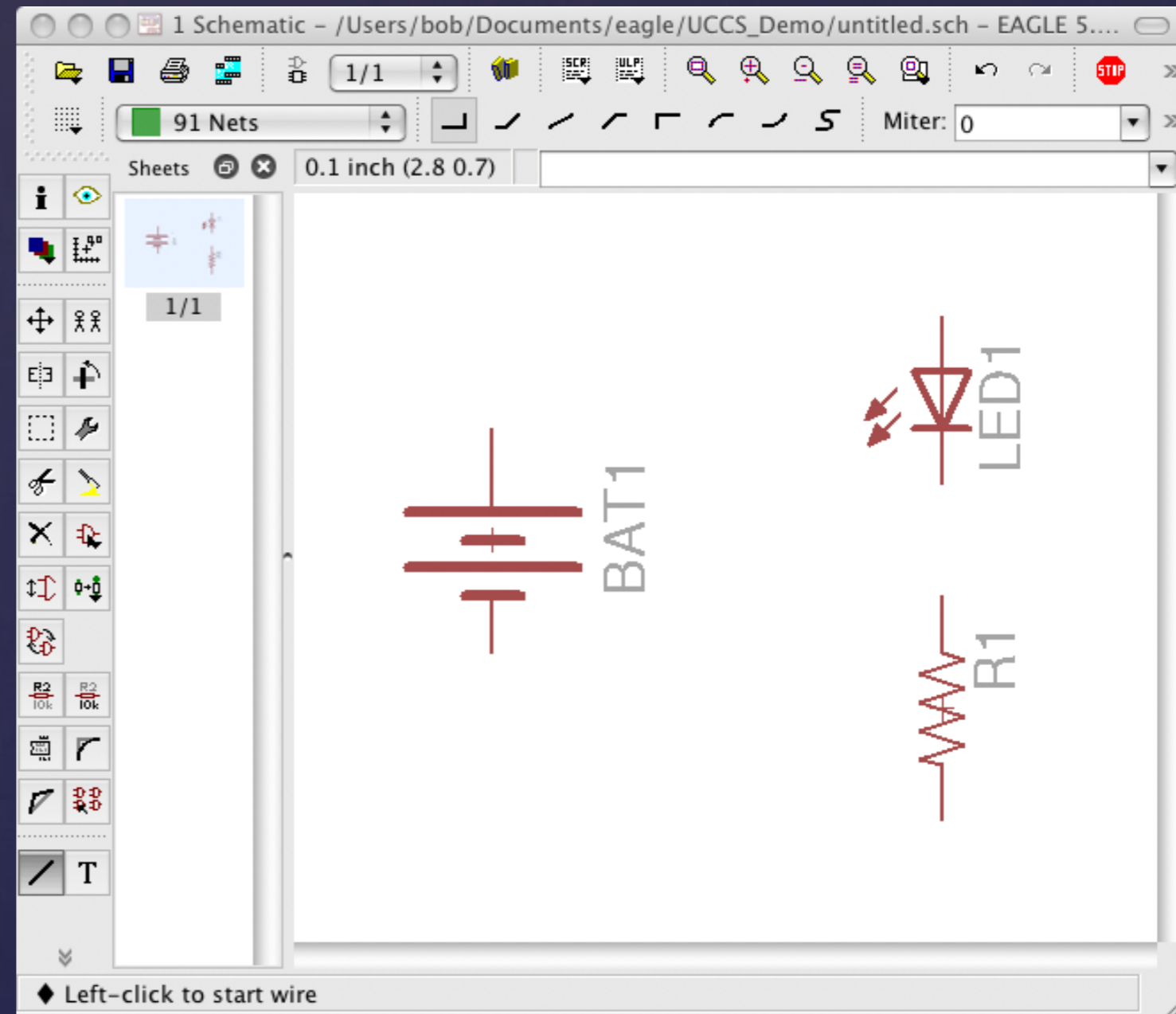## Creating a New Schematic

- `File -> New -> Schematic`

- Keep a grid.

10

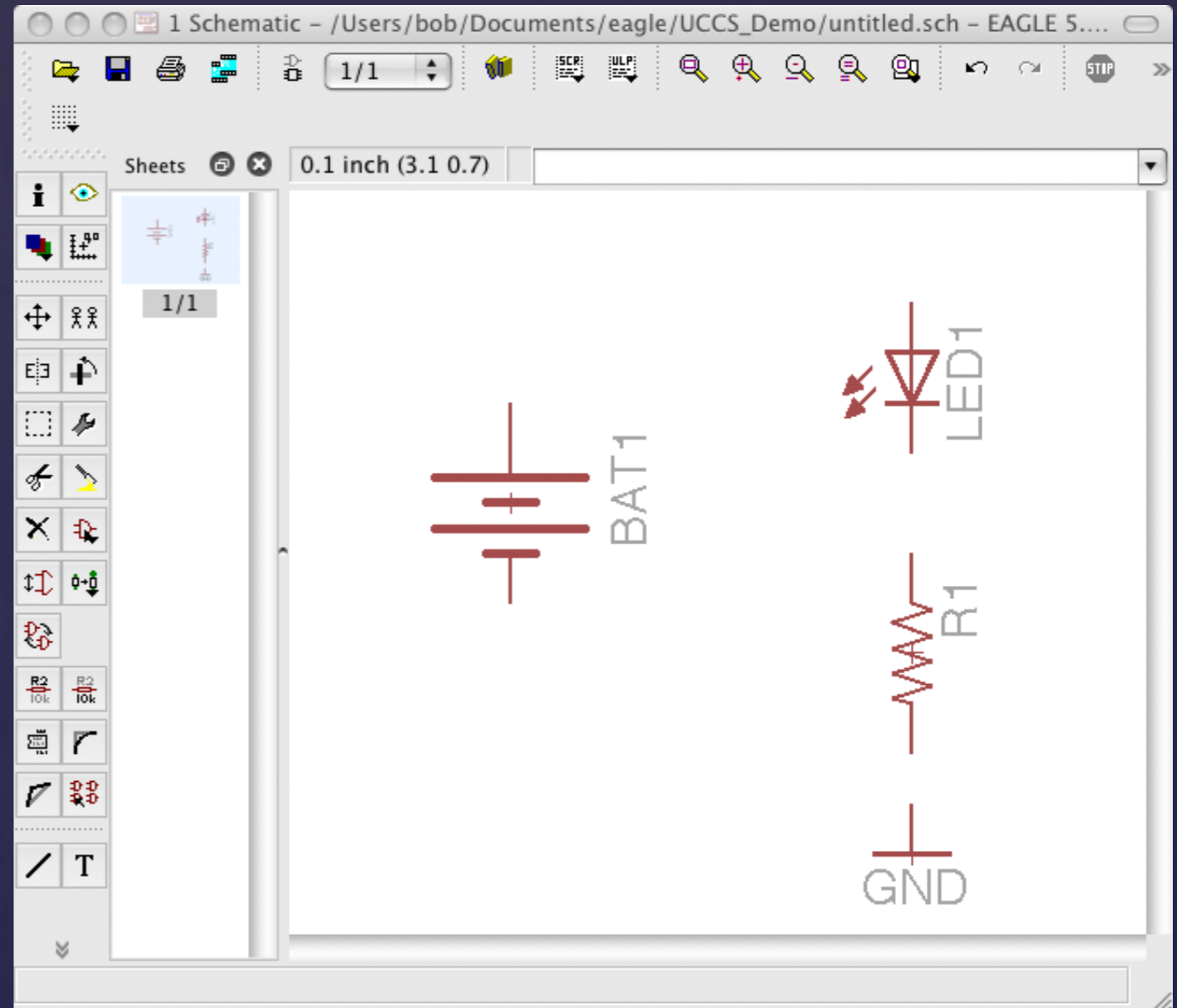# Schematic Capture

Adding Components

- Click the Add icon

- Find the component in the library

- Set values

- Don't see the library? Try Use -> Library

11

Schematic Capture

Adding Global Symbols

- Click the Add icon

- Find the global symbol in the library (supply1)

- Place as if it were a component

Schematic Capture

Adding Traces (aka Wires)

- Click the Wire icon

13
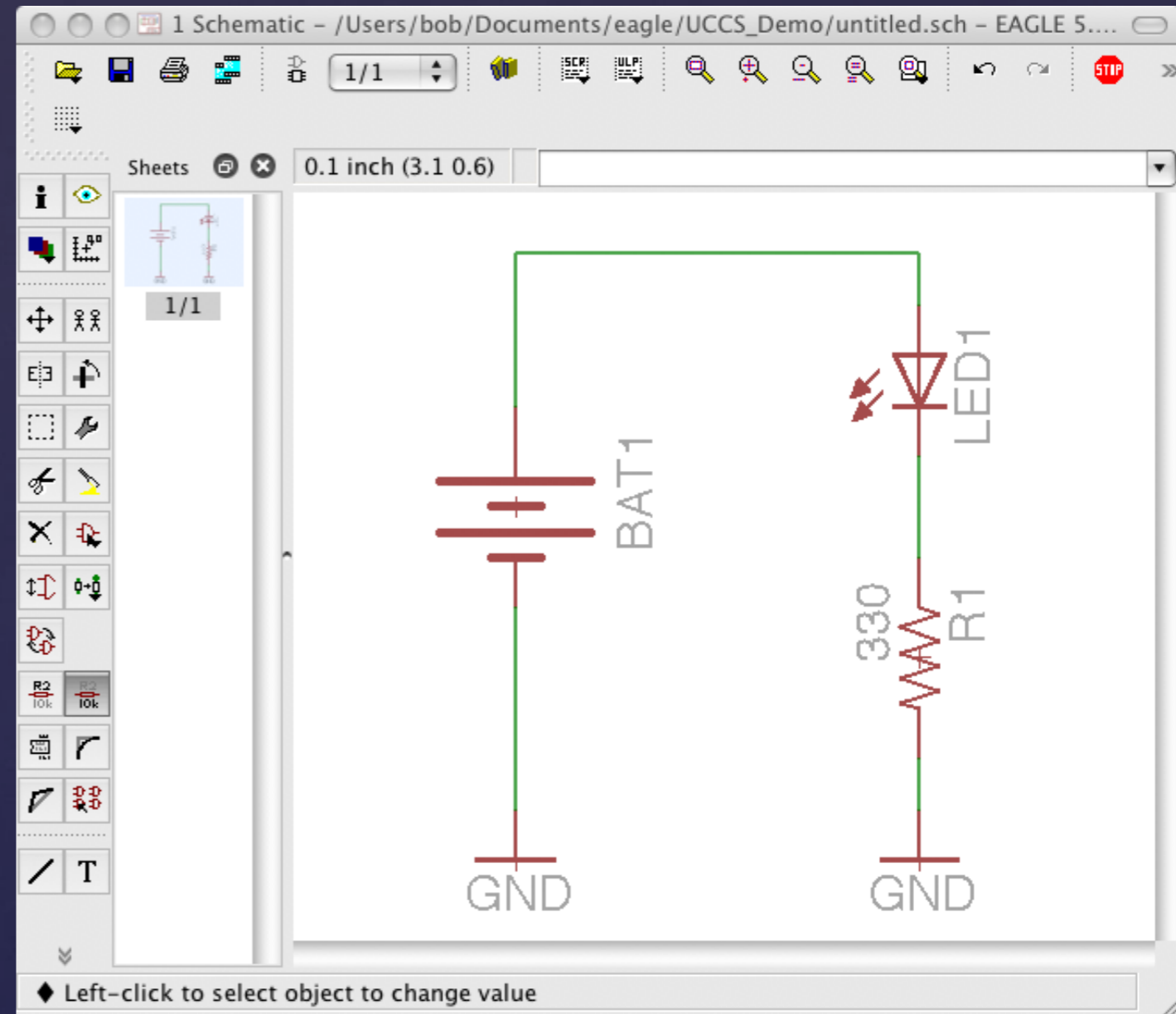
Schematic Capture

Some General Tips

- Avoid changing the grid unless absolutely necessary.

- Use multiple "sheets".

- Schematic drives the layout …

- Groups, Info, and Layers icons are useful.

- Pan, zoom easy to use.

- Really care about that "net"? Label it.

- Keep it clean - don't be lazy.

- Document!

Place Components

But before you do …
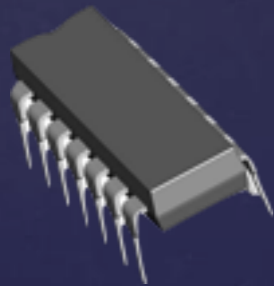
- Passives - 0805 means 0.08" by 0.05"

- Connectors - beware of physical fit

- Actives - lots of packaging alternatives

- Give thought to testability and troubleshooting

Thru-Hole (TH)	Surface Mount (SMT)
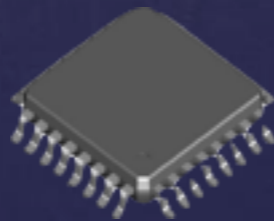
DIP   LQFP, QFP, TQFP   QFN   BGA

15
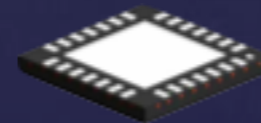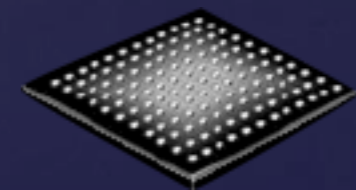
Place Components
Creating a new board design

- From the schematic, choose `File ->` `Switch to Board.`

- "Create From Schematic"

- Board window automatically opens.
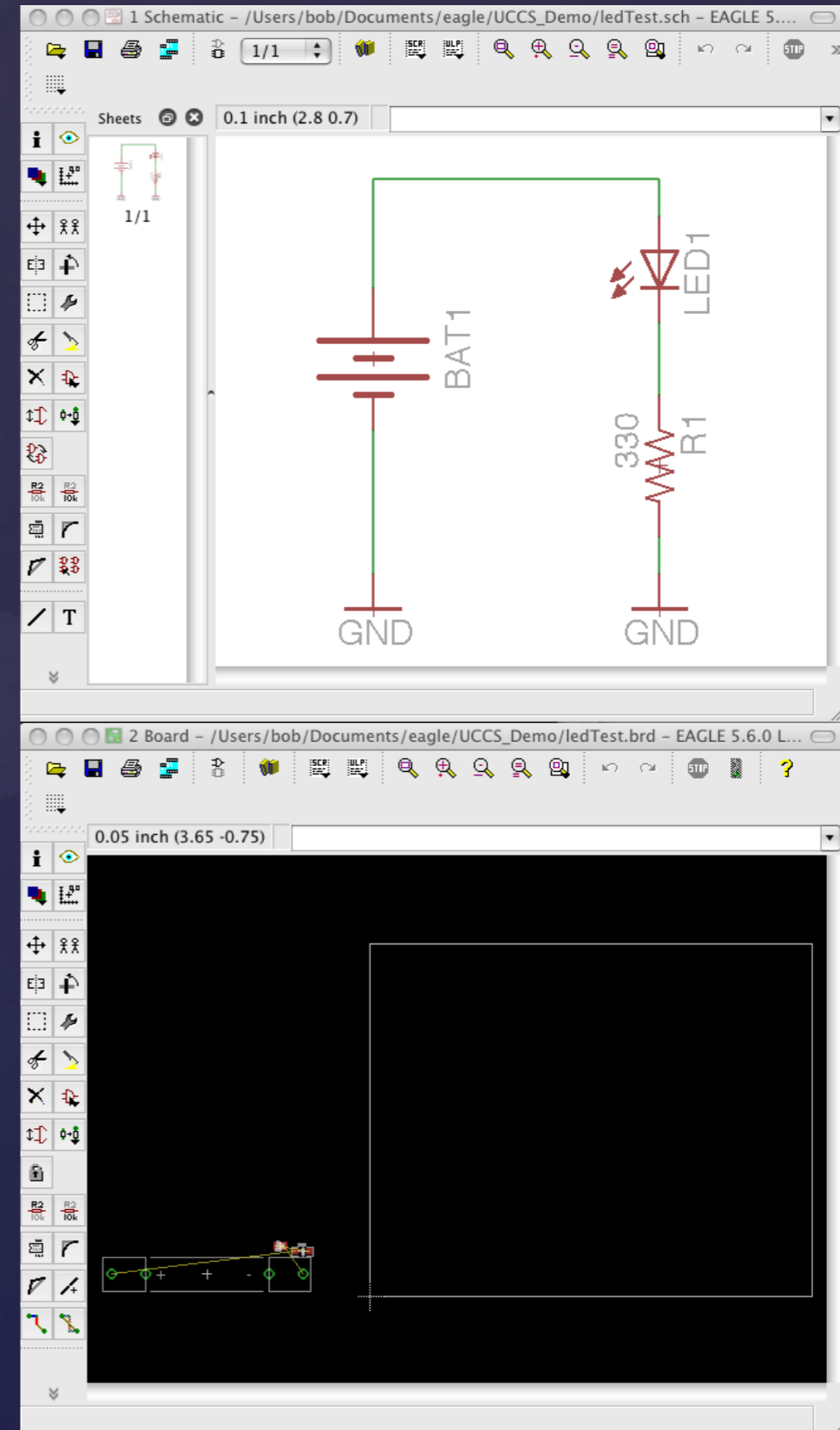
Place Components

Some Common Tasks

- Right-click on top of component, select Move to reposition.

- While moving, right-click to Rotate.

- Use "Group" wisely.

- Right-click to Mirror (change to other side of board)

- Use Move to change outline

17
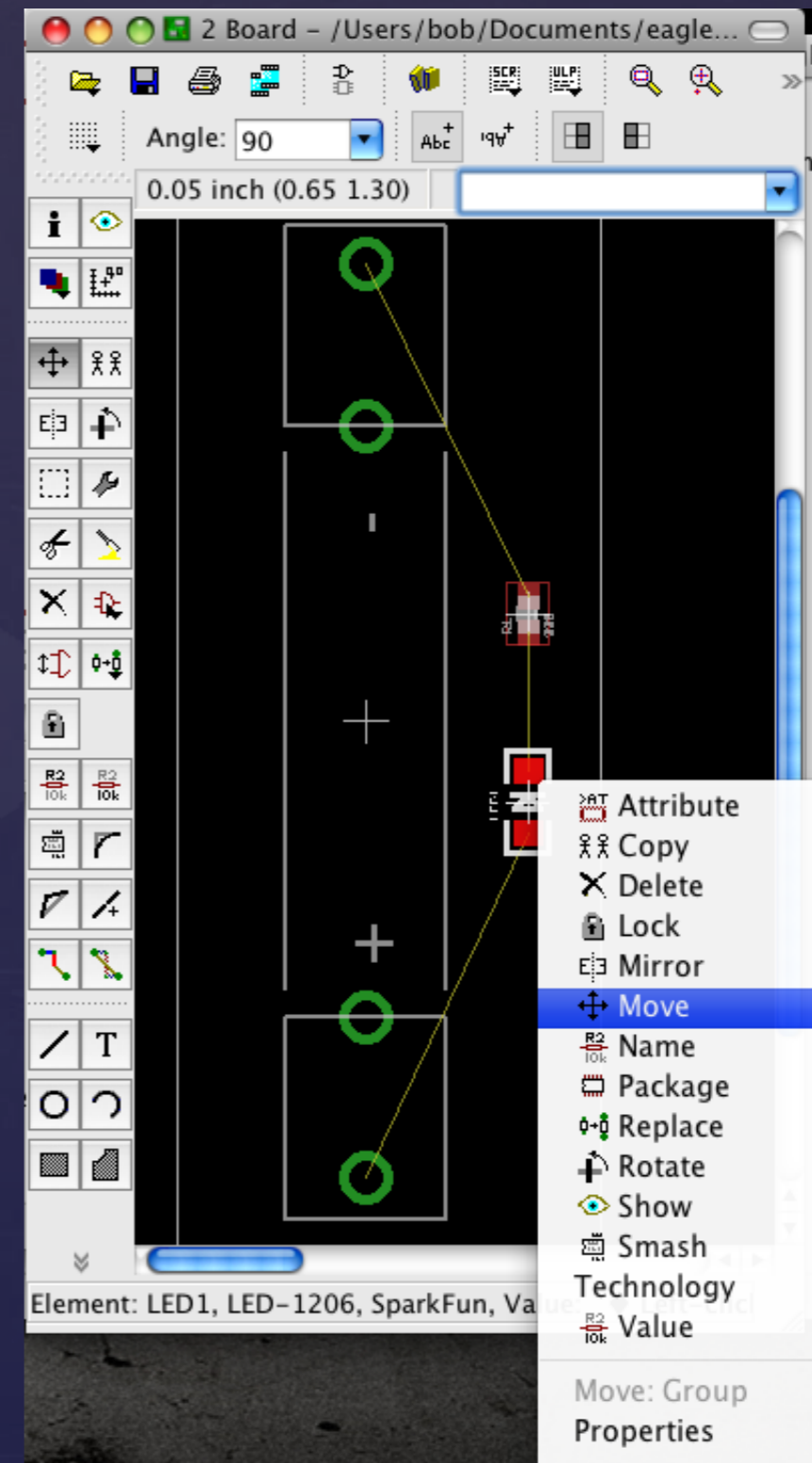
Place Components
Thinking ahead

- Take your time.

- Untangle flight lines.

- Think about planes.

- Consider testability.

- Mechanical Fit?

- Hard stuff first.

- Keep silkscreen.

- Think hard about components on 2-sides.

18
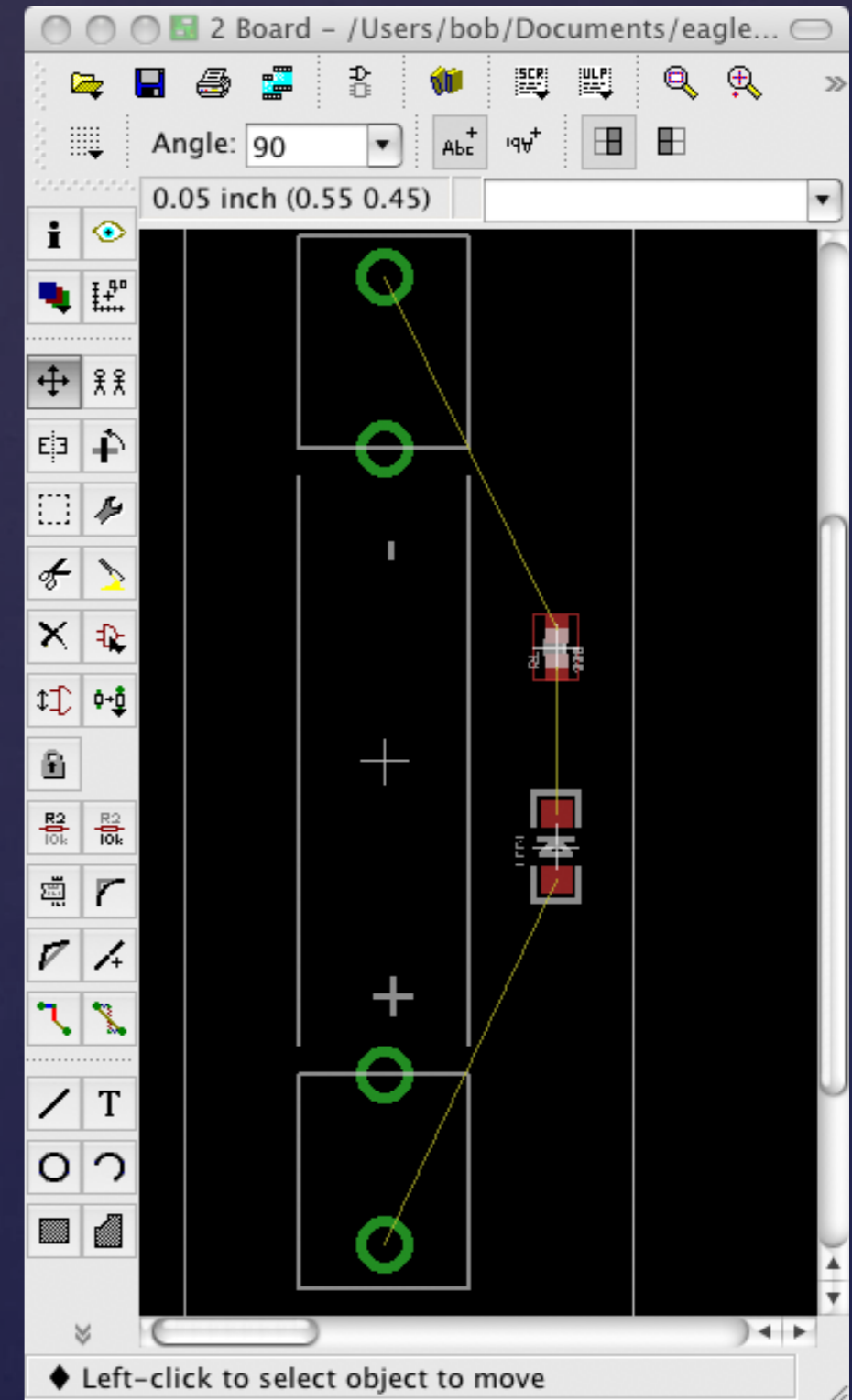
Place Components
Some General Tips
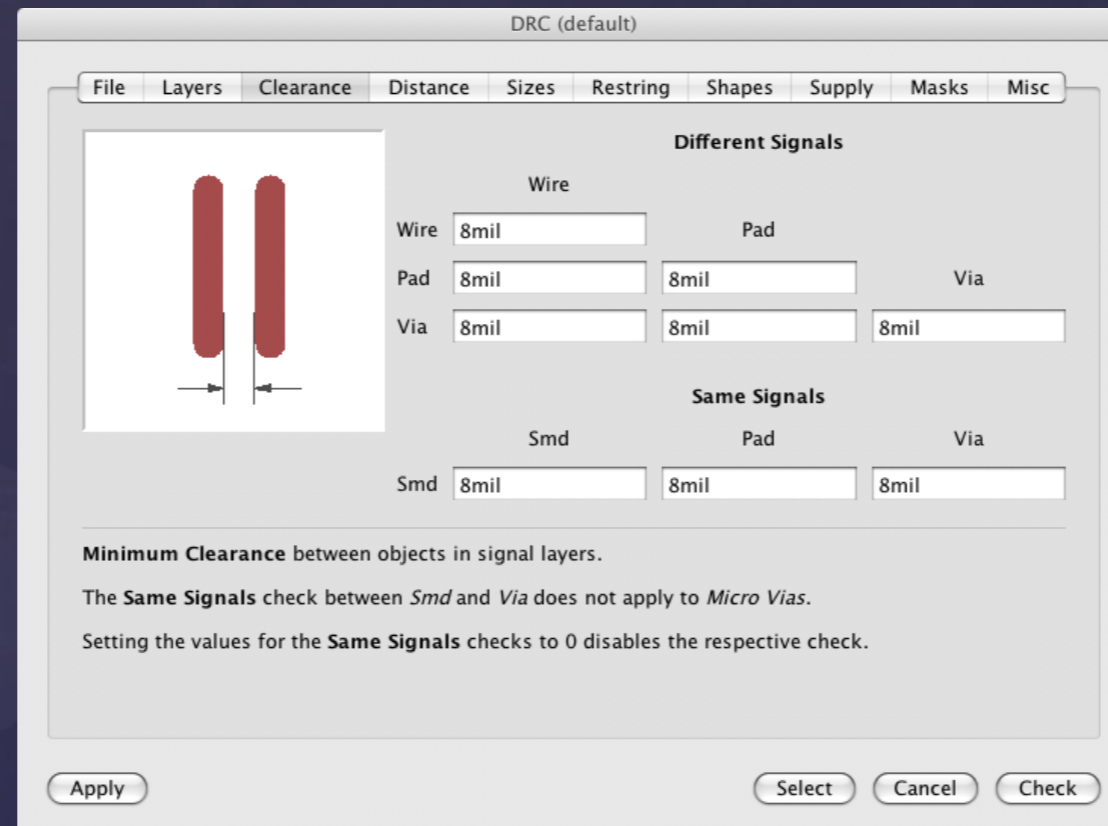
- Are your Design Rules setup correctly?

- Run the Design Rule Check (DRC) often.

- Run the Electrical Rule Check (ERC) often.

- Save a snapshot of your placement file.
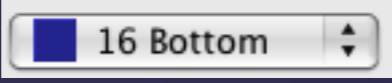
Route Traces

Adding "Waaaahrs" To Your Layout

- You want to *route*, not *wire*.

- You want to *ripup*, not *delete* nets.

- Are you on the top or the bottom?

- Vias are on top *and* bottom. Easy to add.

- DRC often!

- Iterate like mad.

20
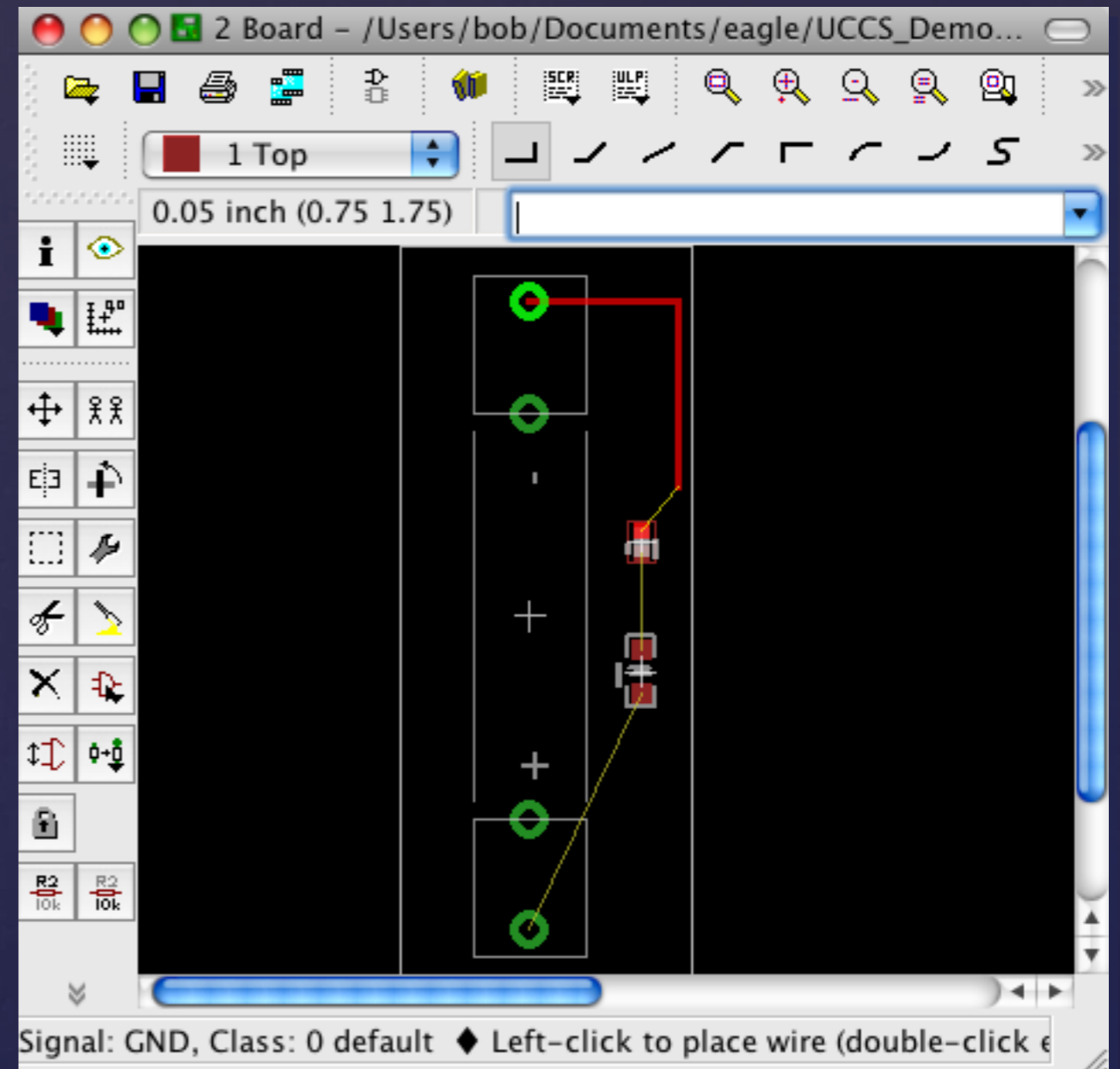
Route Traces
Some General Tips

- Two layers? Have an x-layer and a y-layer.

- Don't forget mounting holes, other nice things.

- Add testability features.

21

Generating Gerbers
Some General Tips

3 CAM Processor – /Applications/EAGLE/cam/AdvancedCircuitsALL.cam – EAGLE 5.6.0 Light

| Paste | Drill | Component side | Solder side | Silk screen CMP | Solder stop mask CMP | Solder stop mask SOL |

Job

Section: Solder stop mask SOL

Prompt:

Output

Device: GERBER_RS274X

File: ./gerber/%N.sts

Offset

X: 0inch

Y: 0inch

Style

☐ Mirror
☐ Rotate
☐ Upside down
☑ pos. Coord
☐ Quickplot
☑ Optimize
☑ Fill pads

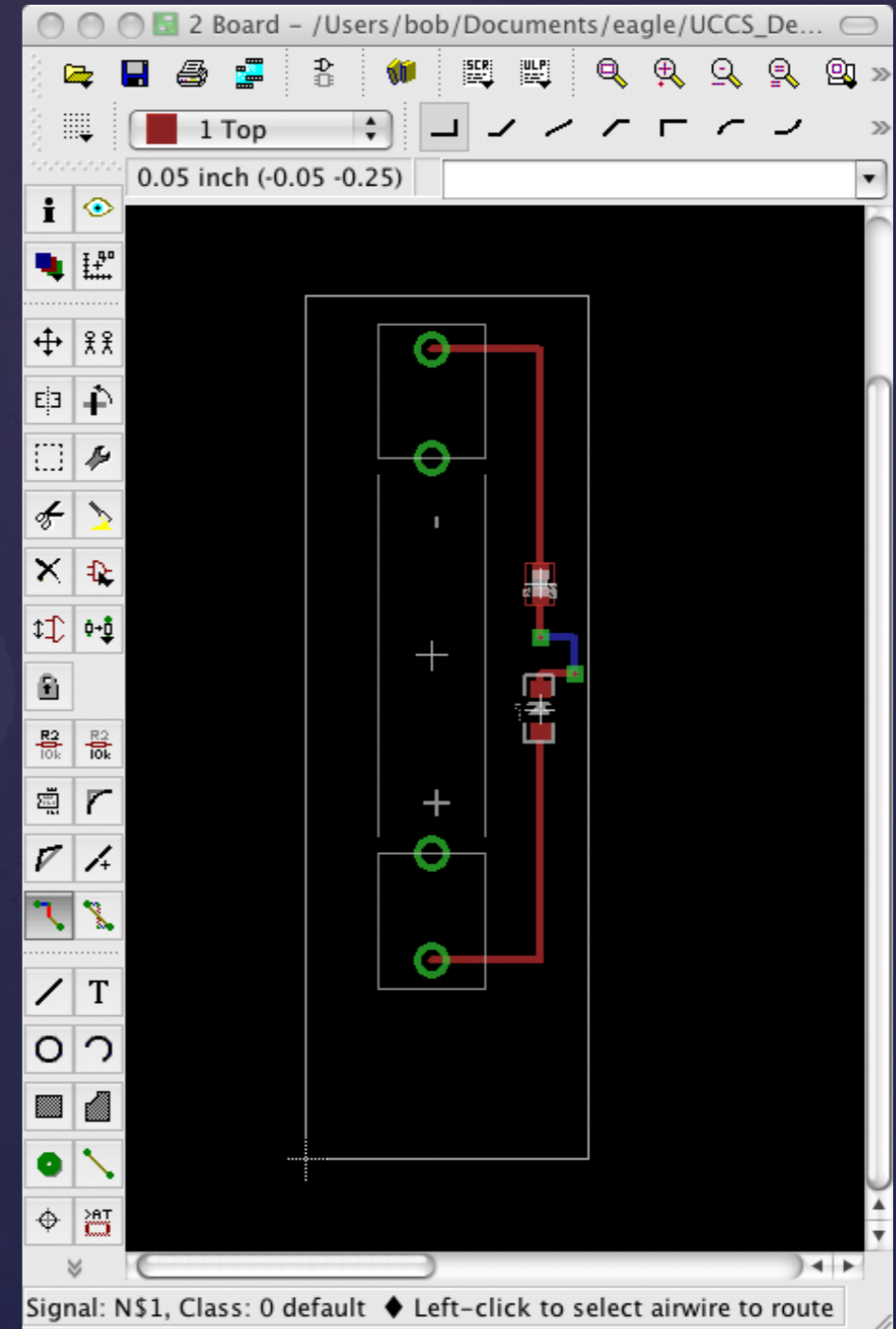| Nr	Layer
1	Top
16	Bottom
17	Pads
18	Vias
19	Unrouted
20	Dimension
21	tPlace
22	bPlace
23	tOrigins
24	bOrigins
25	tNames
26	bNames
27	tValues
28	bValues
29	tStop
30	bStop
31	tCream
32	bCream
33	tFinish
34	bFinish
35	tGlue
36	bGlue

Process Job   Process Section   Description   Add   Del

/Users/bob/Documents/eagle/KST_RPS/kst_rps_revB.brd

- Using Advanced Circuits? Have I got a script for you …
- Generate critical layers.

Generating Gerbers
Sanity Checking The Result
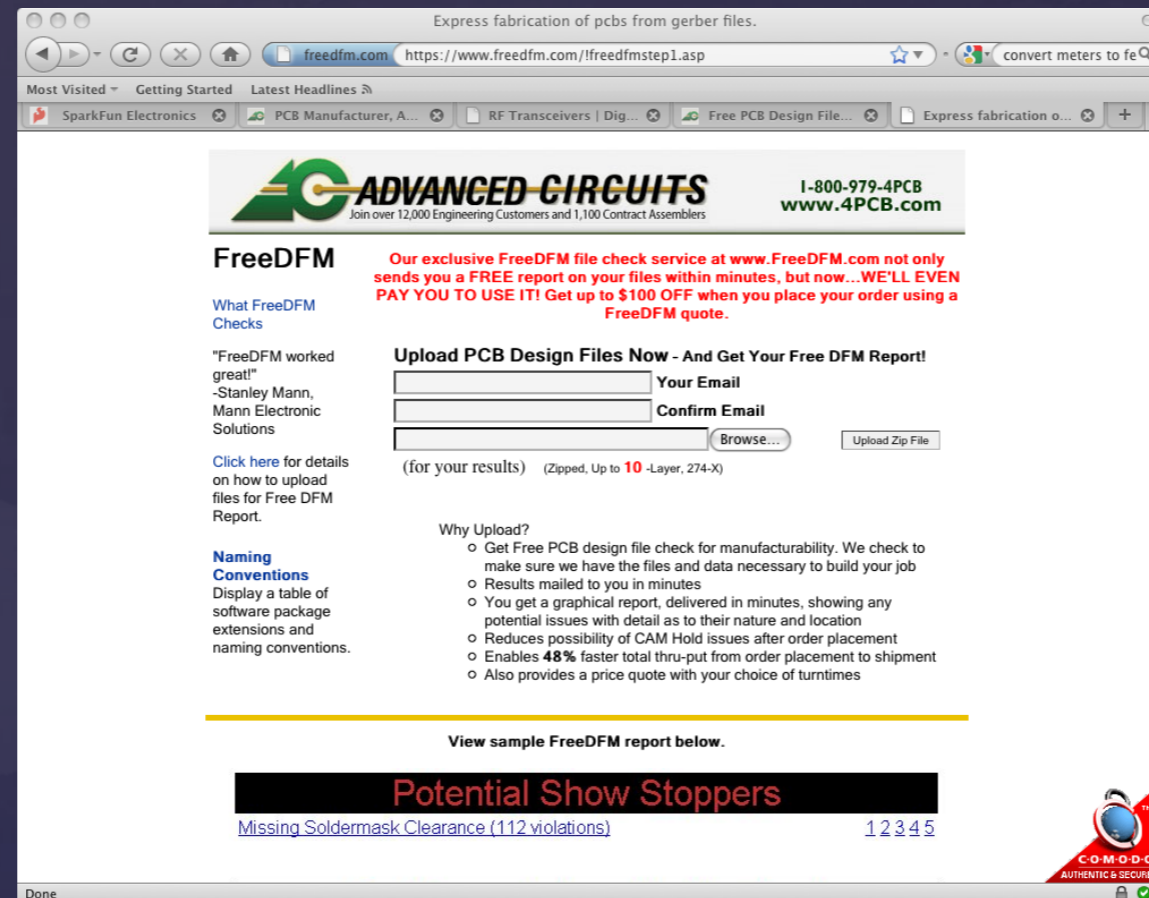
freedfm.com

- You'll need an account with Advanced Circuits.

- Email will be sent confirming design rule checking.

- Advanced Circuits even gives you a discount.

- Squash any errors, review all warnings. Look for open vias, closed vias, etc.

Generating Gerbers
Reviewing The Result

- ViewMate - Free Gerber Viewer for Microsoft Windows

- Review planes, drill holes, and "obvious" defects. Not usually any surprises here.

- Biggest error - not exporting the proper layers.

Fab
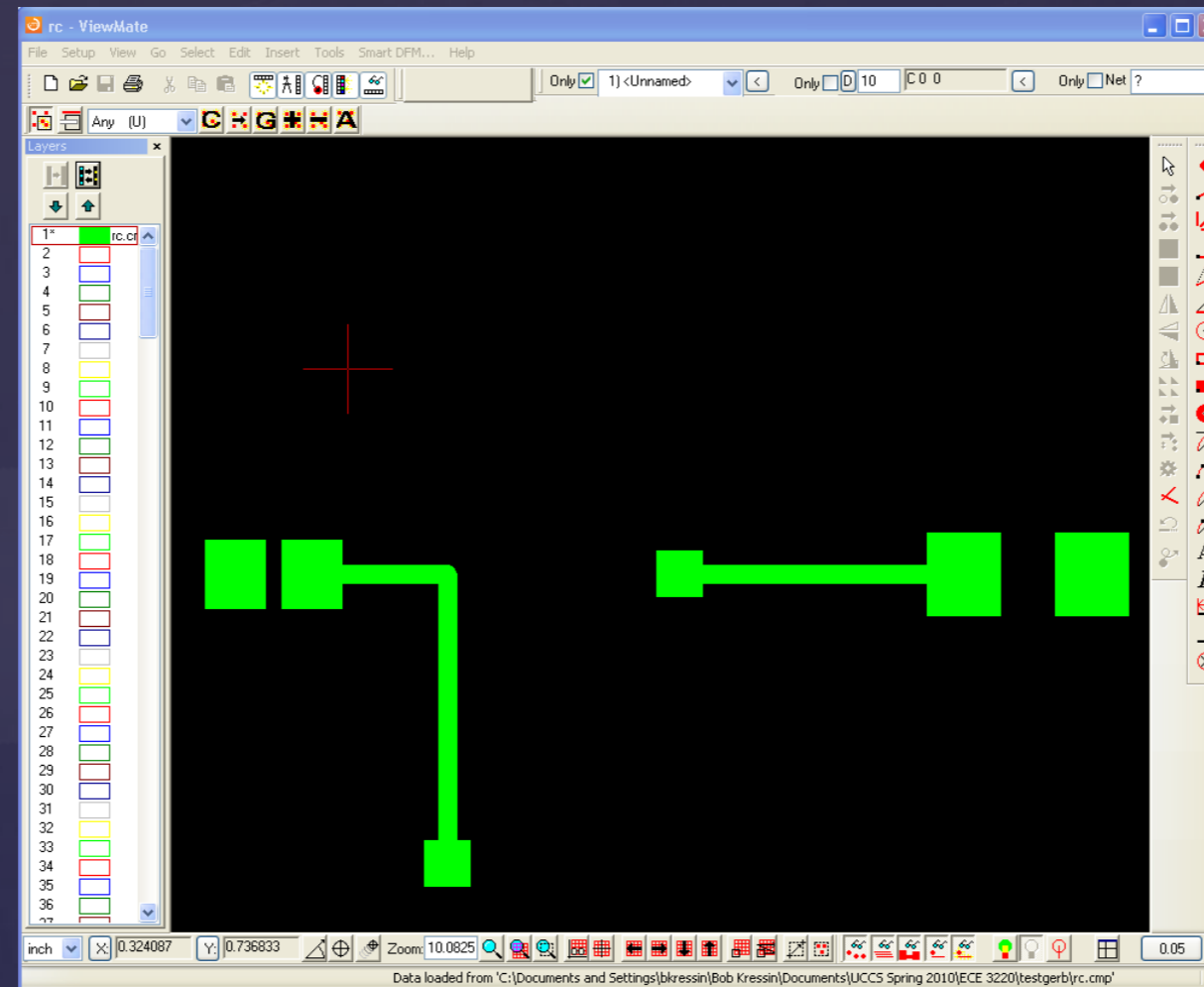## Before you hit "the button" …

- Generate a Parts List (aka "The BOM") using Export -> Partlist.

- Generate a Pin List (aka "The Netlist") using Export -> Netlist.

- Sanity check both.

```
000                            BOM
Partlist

Exported from ledTest.sch at 10/16/09 7:23 AM

EAGLE Version 5.6.0 Copyright (c) 1988-2009 CadSoft

Part      Value          Device           Package      Library  Sheet

BAT1                     BATTERYAAA       BATTERY-AAA  SparkFun 1
LED1                     LED1206          LED-1206     SparkFun 1
R1        330            RESISTOR0603-RES 0603-RES     SparkFun 1
```

```
000                            pinlist
Pinlist

Exported from ledTest.sch at 10/16/09 7:28 AM

EAGLE Version 5.6.0 Copyright (c) 1988-2009 CadSoft

Part      Pad      Pin      Dir      Net

BAT1      GND@1    -        Pwr      GND
          PWR@1    +        Pwr      N$1

LED1      A        A        Pas      N$1
          C        C        Pas      N$2

R1        1        1        Pas      N$2
          2        2        Pas      GND
```

Fab
Some General Tips

- Do you have all of your components? Lead time?

- Find out what file format your PCB Vendor wants.

- Advanced Circuits - GERBER_RS_274X

- Consider routing it yourself.

- Consider barebonespcb.com. Really cheap.

- Sleep on it.

The PCB Design Process

A Summary of a Lot of Information