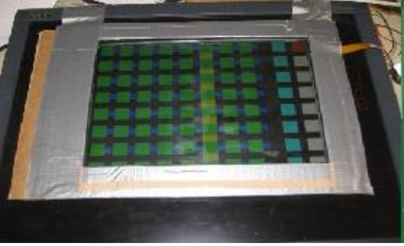
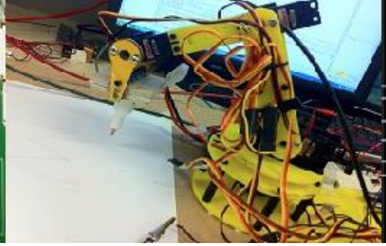
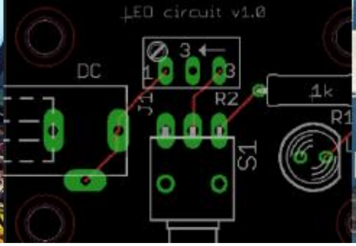


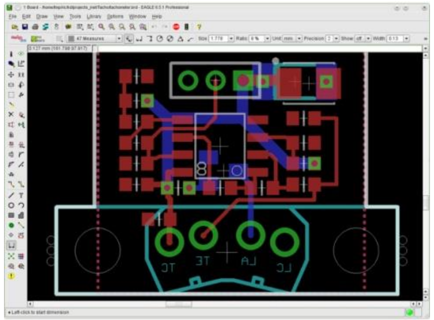


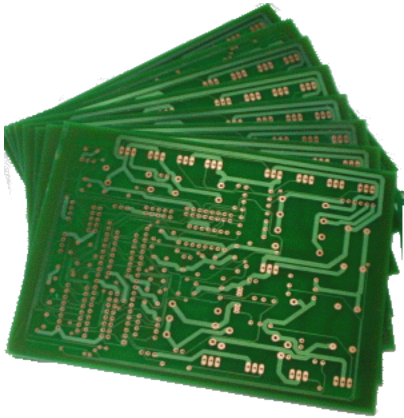


EECS 473
Advanced Embedded Systems

Lecture 8:

Projects, PCBs and Power integrity





Project status updates

- We will be having regular project status reports at the start of many classes starting Thursday
  - Half the groups will go each day.
  - Normally 60-90 seconds.
  - The purpose is to hear what others are doing, understand their issues, and maybe be able to provide help/ideas/part suggestions.
  - There will be about 4 of these total, each slightly different.

- This Thursday and next Tuesday the topic will be "elevator pitch"
  - You'll have ~2 minutes (longer than normal) to explain your idea.
    - Should include a bit of the why and the how.
    - Think of it as if you are trying to get a VC to fund your new start up.
      - Thursday: **GROUP 1: Infineon, Braille Notes, Lap-trigger, Robotics , Smart Handle, Maize Toss, Scoreboard.**
      - Tuesday: **GROUP 2: Robot Arm, TGFKARW, Key Oracle, Canbats, Transcriber Glasses, Chess, Solar Cell.**

What’s due project-wise?

- Final proposal
  - Submit proposal on this Thursday.
    - We *should* have feedback by Wednesday night to all groups.
    - <u>Be really really sure you’ve cleaned up your objectively demonstrable deliverables.</u>
- Written MS1 due on **Friday 10/13***
  - Need show objectively demonstrable parts to one of the GSIs.
    - The parts in bold in Appendix B.
    - Need to happen by Wednesday 10/18 in the afternoon
      - Note: Exam is that night.
  - These shouldn’t take more than an hour to write up.
    - *There will is a template on the website*.

*Date is a day later than stated in the project handout/schedule.

Other things upcoming

- Homework 1 coming out this week.
  - Due Wednesday 10/11
  - It will be a good review for the exam.
  - Won't be graded before the midterm.
- Midterm Wednesday 10/18 6-8pm
  - Expect 60-65% of grade to be short answer-type, the rest to be a design problem.
    - Generally people spend a bit more than 50% of the time on the design problem.
  - Old midterms posted.
    - Format largely the same.
      - You may notice some problems are often quite similar
- After the midterm the next non-project thing due is HW2 on 12/5!
  - So after the midterm it's all project other than lectures.

Ordering

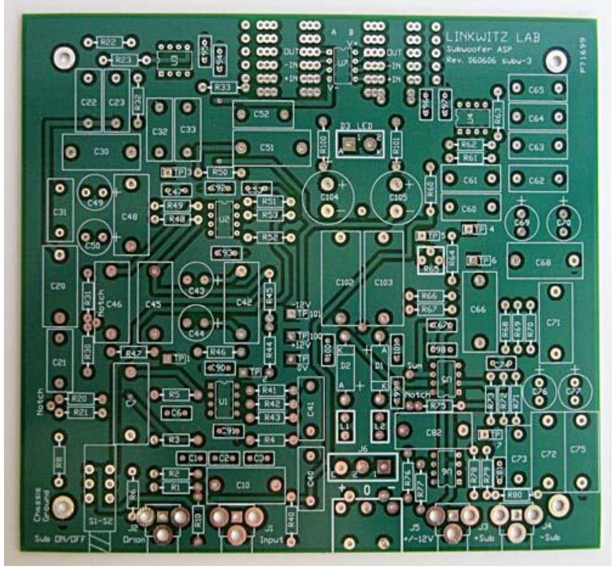
- At this point, you may start ordering things
  - Be aware that the budget is strict: $200/member.
    - Should not be buying things "off budget".
      - Staying on budget is part of the project.
  - Should probably only be ordering dev boards, sensors, etc.
- Two ways to order:
  - Preferred: Order on your own, get reimbursed.
    - Faster and more reliable.
    - Front your own money, if you lose receipts or order something really strange, could be a pain.
    - Talk with Cameron first. Use the sheet.
  - Order through the department
    - Slower, but don't need to front your own money.
    - See me if this is your plan.

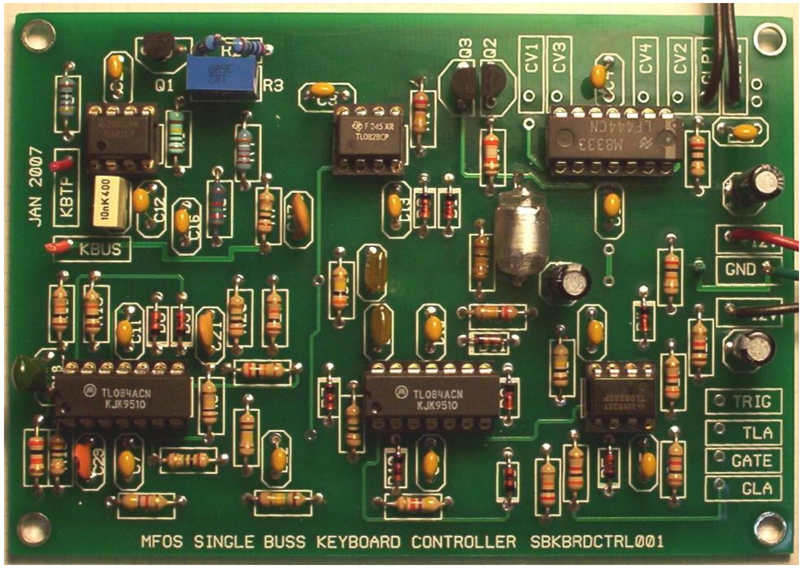
Now…

- Introduce PCBs
  - Terminology and concepts
- Do a review of basic circuits
  - Physics 240/EECS 215
- Discuss power integrity

So you want to make a Printed Circuit Board…

- At the end of the day a PCB is just a set of wires that connect components.
  - But there are some issues
    - The wires have restricted dimensionality
    - The wires are very thin
      - So high resistance (as conductors go)
    - The board needs to include holes (or pads) for the devices.
    - You can't easily change things once you build it.

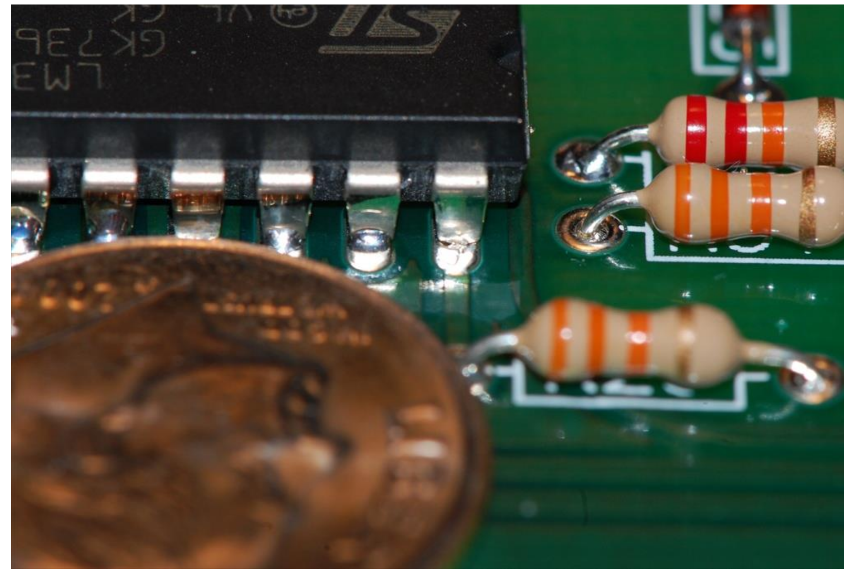


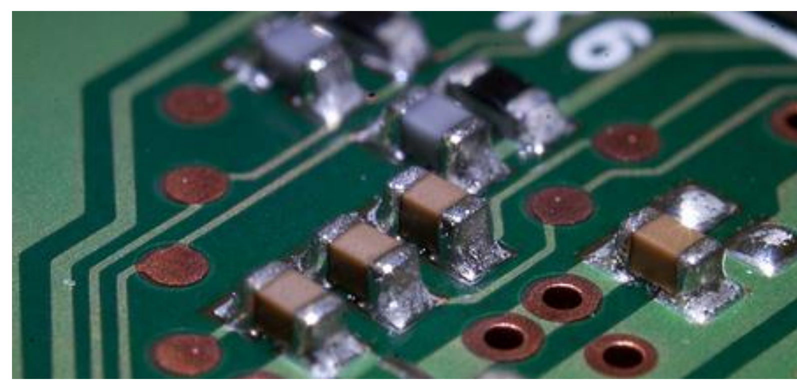


http://www.linkwitzlab.com/Pluto/supplies-subw.htm,
http://www.musicfromouterspace.com/analogsynth/SINGLEBUSSKEYBOARD2007/SINGLEBUSSKEYBOARD2007.php

Basic Terminology

- The wires you are laying out are called “traces” or “tracks”
- Inside of a given “layer” traces which cross are electrically connected.
  - If you have traces on both sides of the board, you are said to have two layers.
- Through-hole: Having holes in the PCB designed to have pins put through the hole
  - Contrast with surface mount where device goes on top.



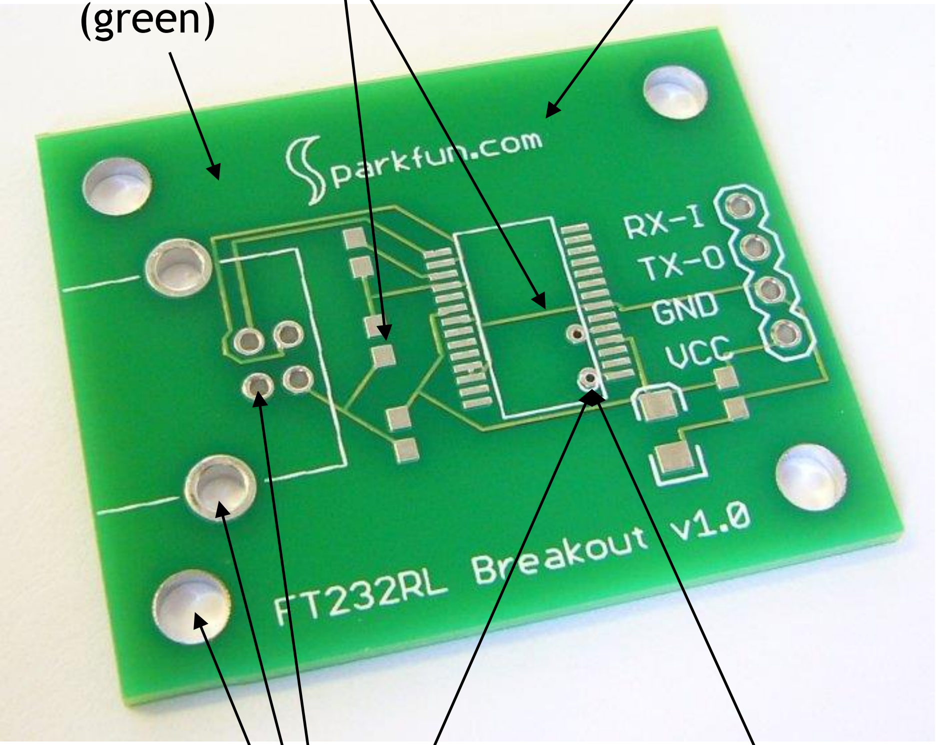


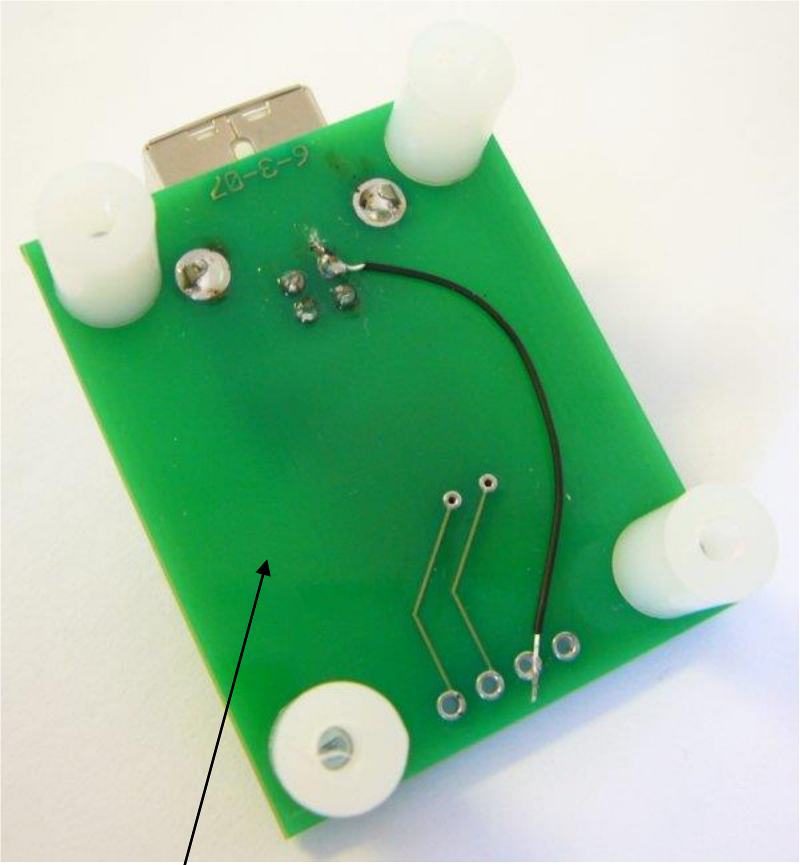
Parts of a PCB

Copper
(pads & traces)

Silkscreen
(white)

Soldermask
(green)





Drill holes

Via

Bottom side

Vias

- Sometimes you need to connect two traces on two different layers.
  - To do this we use a via.
  - It is just a a plated through hole
    - Generally smaller than a through hole for a part.

Clearances

- There will be space between the traces, plated holes and each other.
  - You need to meet the requirement of the manufacturer.

The layered construction of a PCB: A six layer board

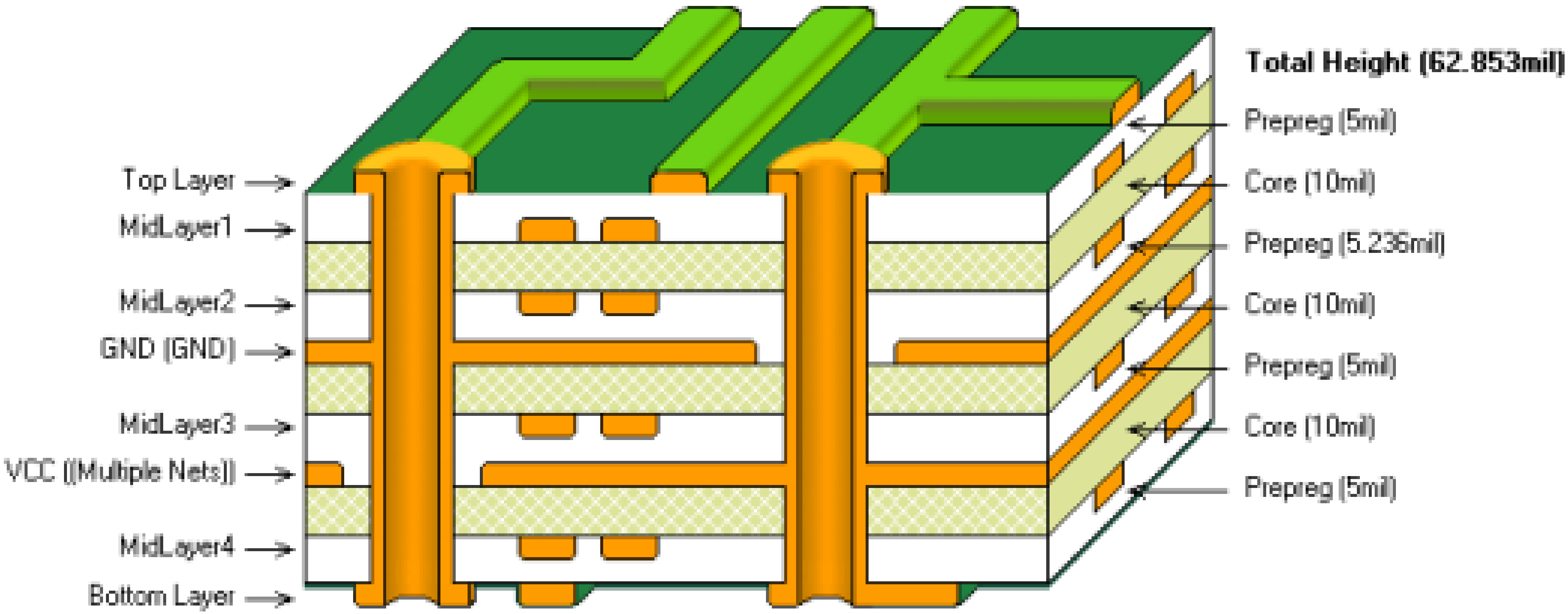


Figure from altium.com

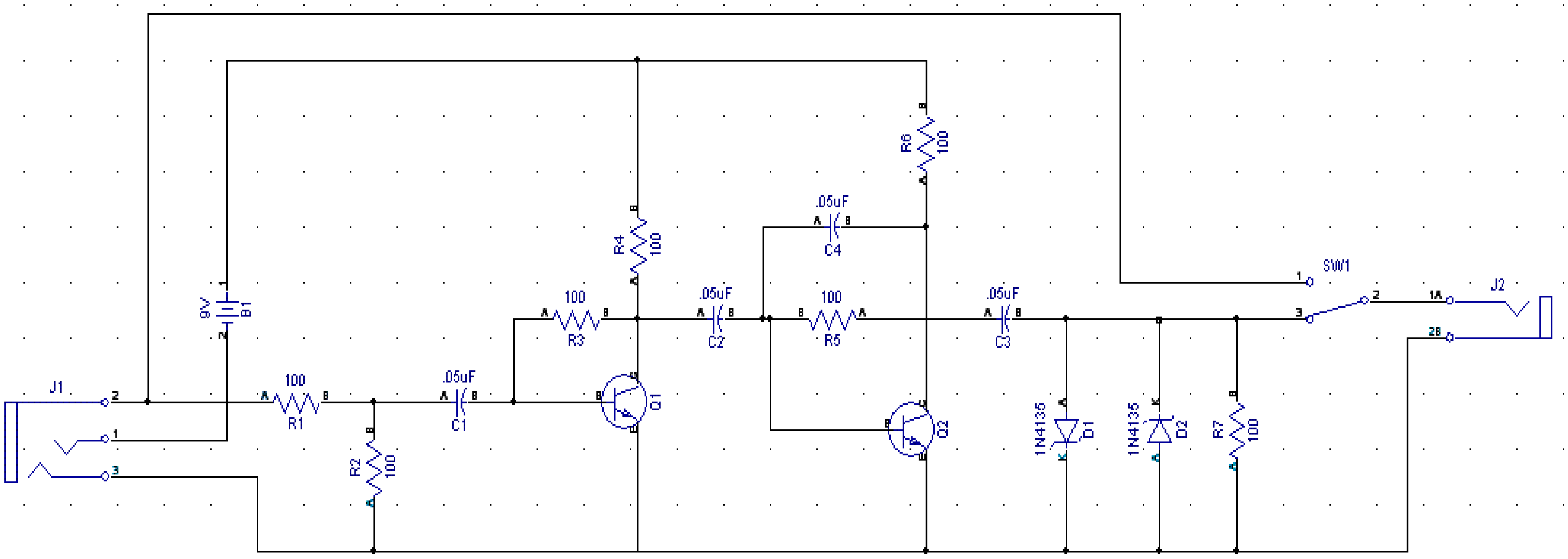
So, how do I design a PCB?

1. Create schematic
2. Place parts
3. Route interconnect
4. Generate files

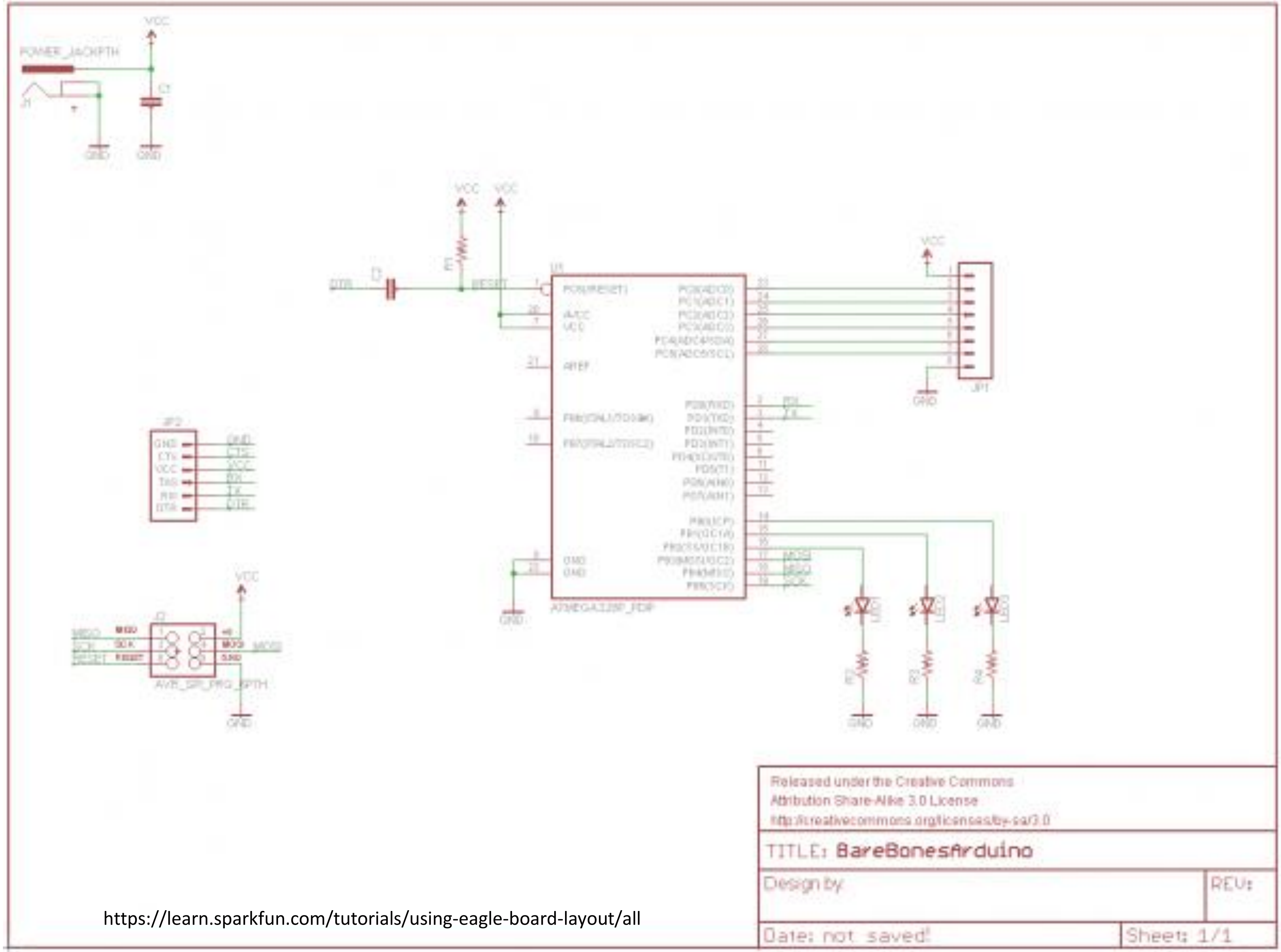
Step 1: Create schematic

- The first thing you want is something that looks like a textbook circuit diagram. It just shows the devices and how they are connected.
  - Sometimes you will worry about pinouts here (say when working with a microprocessor maybe)
  - But usually you don't
- **No notion of layout** belongs here!

Example schematic

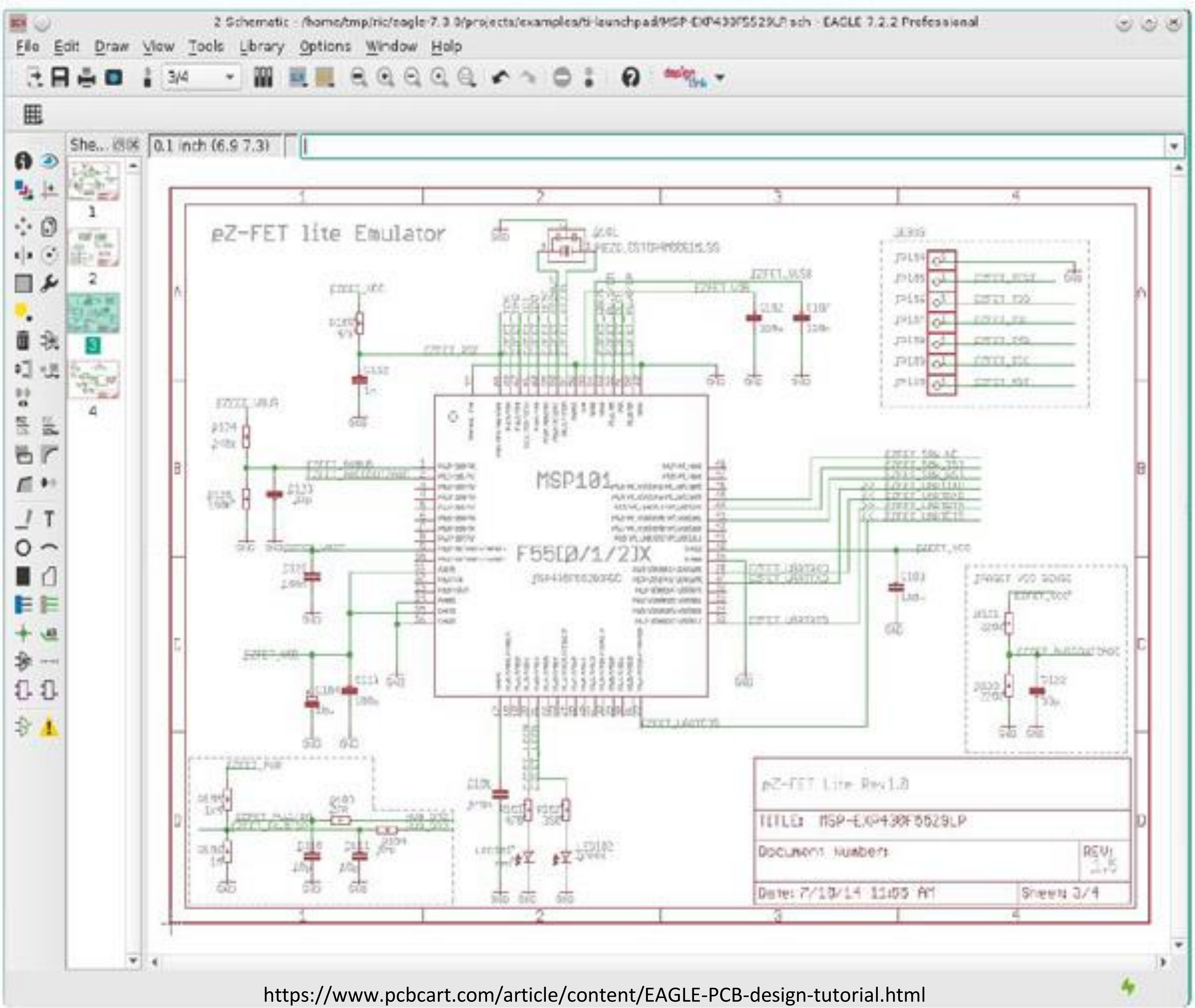


Another schematic



https://learn.sparkfun.com/tutorials/using-eagle-board-layout/all

One more



https://www.pcbcart.com/article/content/EAGLE-PCB-design-tutorial.html

Why a schematic?

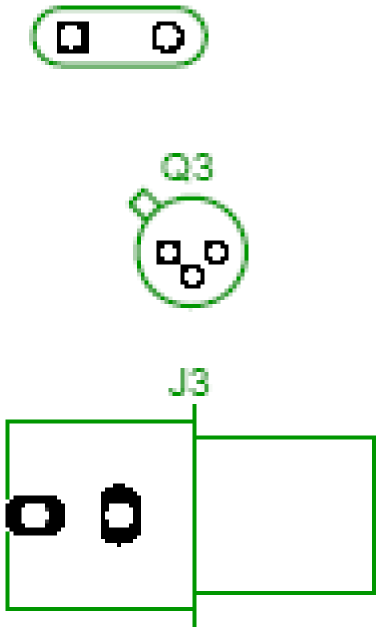
- In general it is drawn to be *readable*.
  - This is probably what your sketch on paper would look like.
  - You can find and fix bugs more easily here than the PCB layout.

Step 2: Place parts

- You need to place the patterns on the board.
  - You need to not overlap them to that the components can actually fit on the board.
  - You want to leave room for the traces to connect everything.
- This is *very* much an art form.
  - In fact you will find people who rant about "sloppy" or "unprofessional" placements.
- Some tools will do this for you. No one seems to like them.

Patterns

- Once you know what it is you want to build, you need to figure out how to lay it out on the board.
  - You need to know how big each piece is, and where the holes need to be placed.
- Each device has a pattern which shows exactly that.
  - You will occasionally need to create a pattern.



Step 3: Route interconnect

- A route is a connection between devices.
  - It may consist of multiple traces
- There are design rules which include:
  - Minimum trace width
  - Minimum spacing between traces and holes
  - Minimum spacing between holes and holes.
- These rules will vary by manufacturer.
  - Even better, ***units*** will vary by manufacturer!
  - Time for a brief aside…

Issues of measure

- PCB land uses some interesting terminology.
  - A “thou” is a thousandth of an inch.
  - A “mm” is a millimeter
  - A “mil” is a thousandth of an inch.
    - Thou is generally preferred over mil to avoid confusion, but most tools/vendors use mil.

Trace width

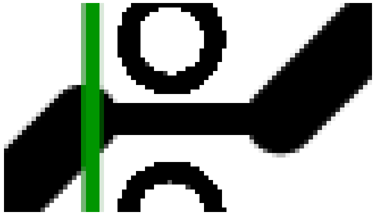
- In general most PCB manufactures seem to have trace-width minimums of 6-10 thous.
  - Most are willing to go smaller for a price.
- A rule of thumb is to use a 50 thou minimum for power/ground and 25 for everything else.
  - This is to drop the resistance of the traces.
  - In general you are worried about heat dissipation
- There are lots of guidelines for width/power but in general you are looking at:
  - A 10cm trace needs to be 10 thou wide if it will carry 1 amp.
  - 5 amps at 10cm would require 110 thou.

Why do you want wide traces?

- A narrow and long trace might create a fairly high resistance.
  - 6 thou, 4 cm long, standard thickness trace at 50 degrees C is about 0.14 Ohms.
    - Not a lot, but at 1 Amp, you're dropping 140mW on the wire and 0.14V.
      - Note: general recommendation lowest width for 1 Amp at 4cm is about 12 thou.
        - » That drops 70mW and 0.07V.

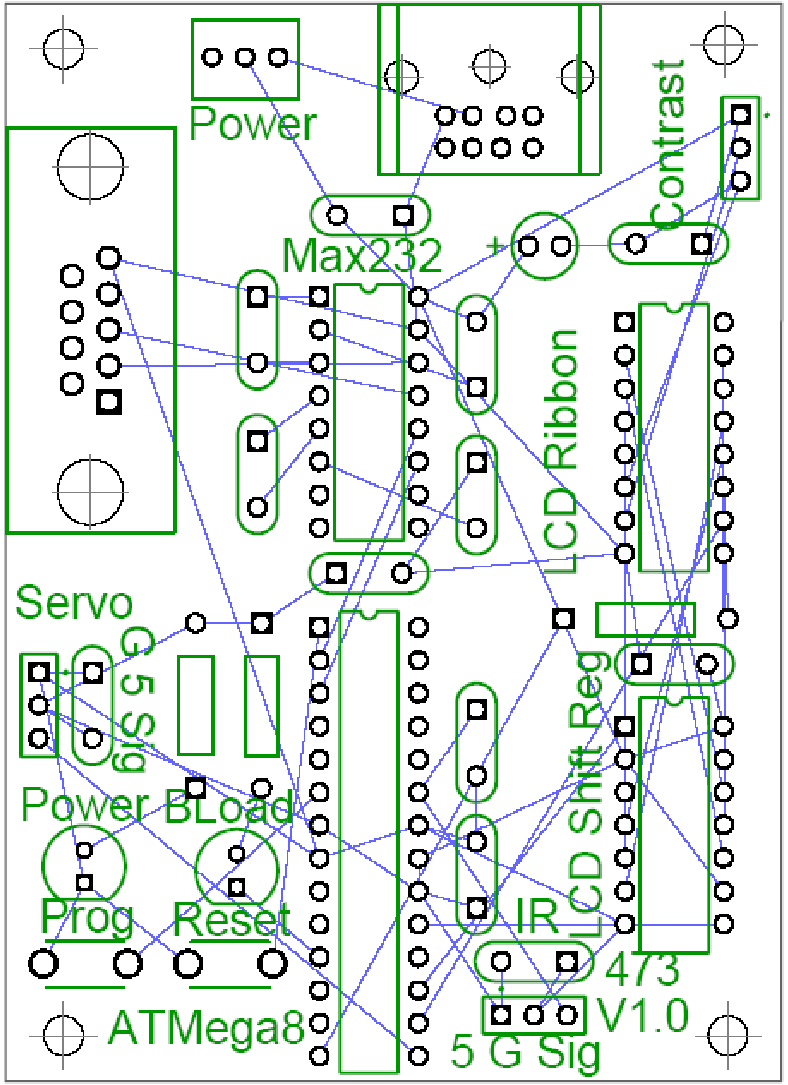
Trace width continued

- The *problem* with wide traces is that they are hard to route.
  - In particular you might wish to go between pins of a device.
- One solution is to be wide normally and “neck down” when you have to.
  - This is more reasonable than you think.
    - Think resistors in series.
    - But be careful of creating a “fuse” for high-current wires.



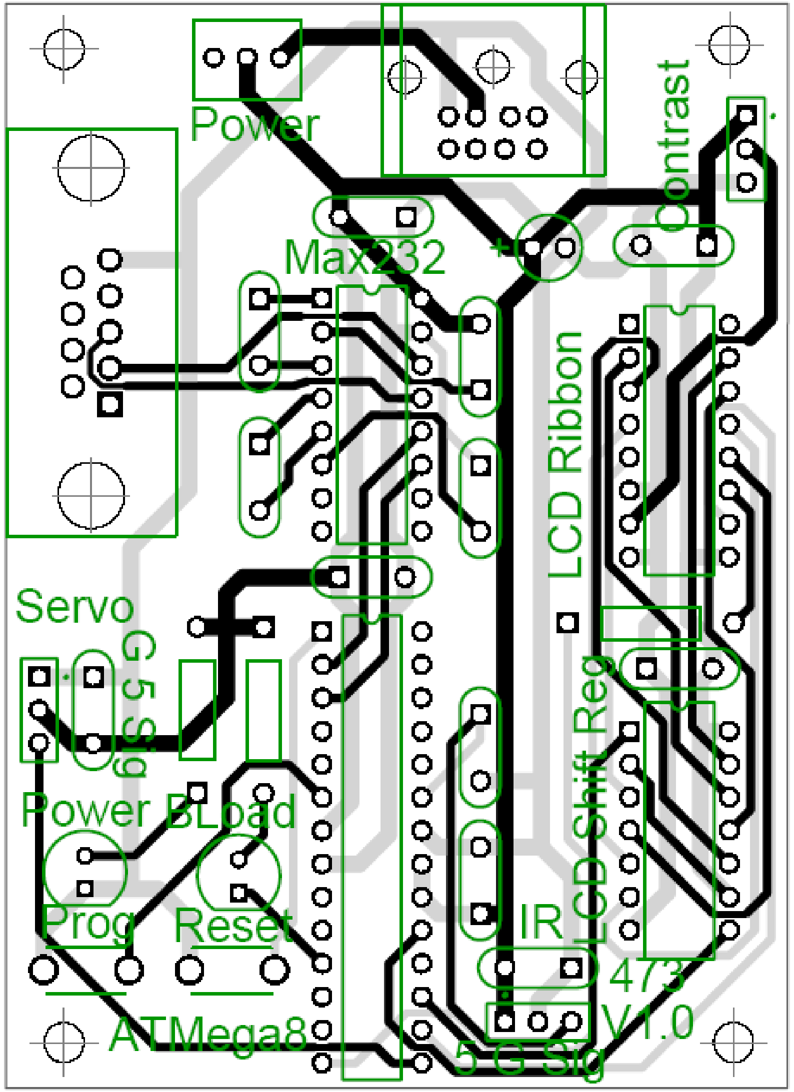
Rat's nest.

- A rat's nest shows the placement of the devices **and** the connections but not the routing
  - Automatically generated for you.
    - Sometimes before placement, sometimes after
      - Varies by tool.

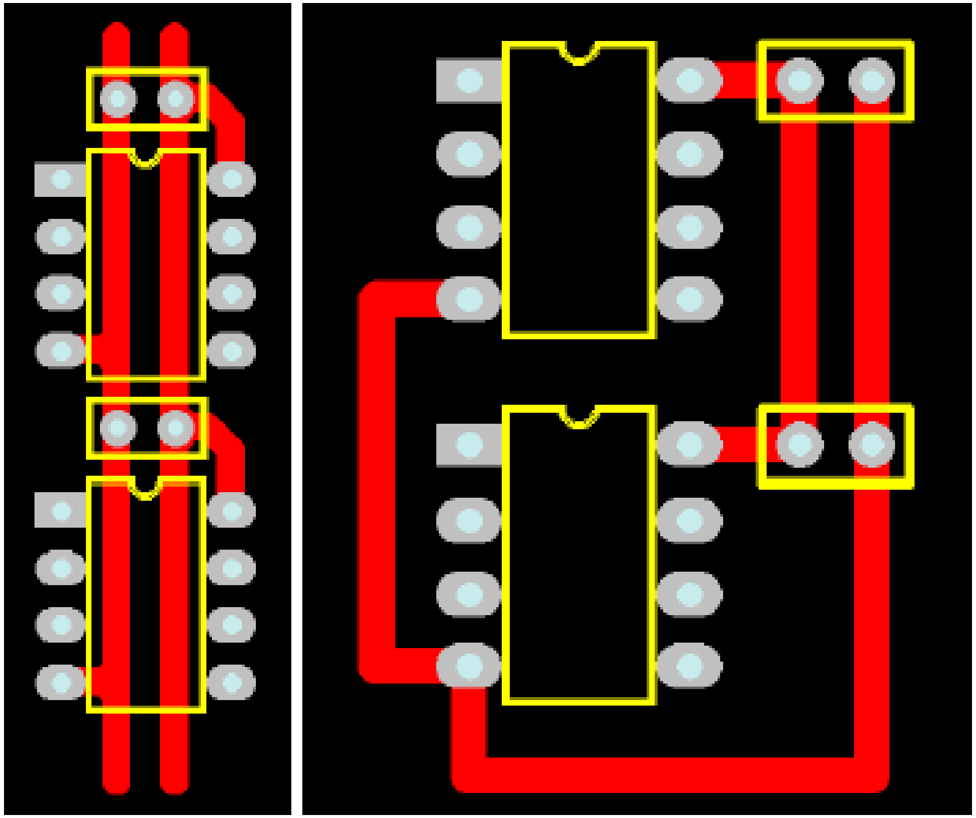


Routing for real

- You can use an autorouter to route your traces
  - Some people hate these as the design will be “ugly”
  - Saves a lot of time.
  - Oddly, not as good as a person can do.
    - But much faster.
- Still generally need to do some (or all) of the routing by hand
  - Very tedious…



Routing quality



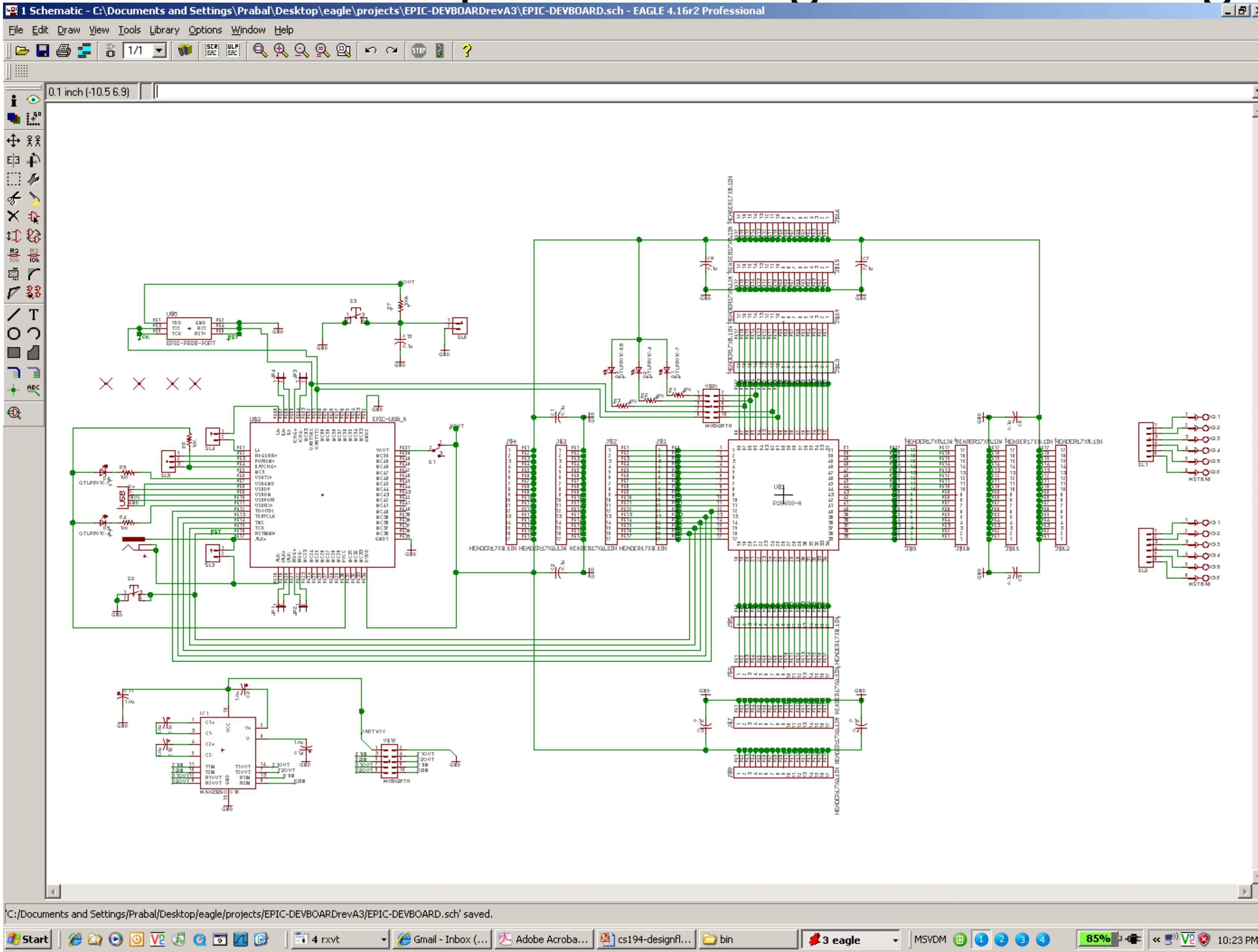
An example of GOOD power routing (Left) and BAD power routing (Right)

http://alternatezone.com/electronics/files/PCBDesignTutorialRevA.pdf

Step 4: Generate files

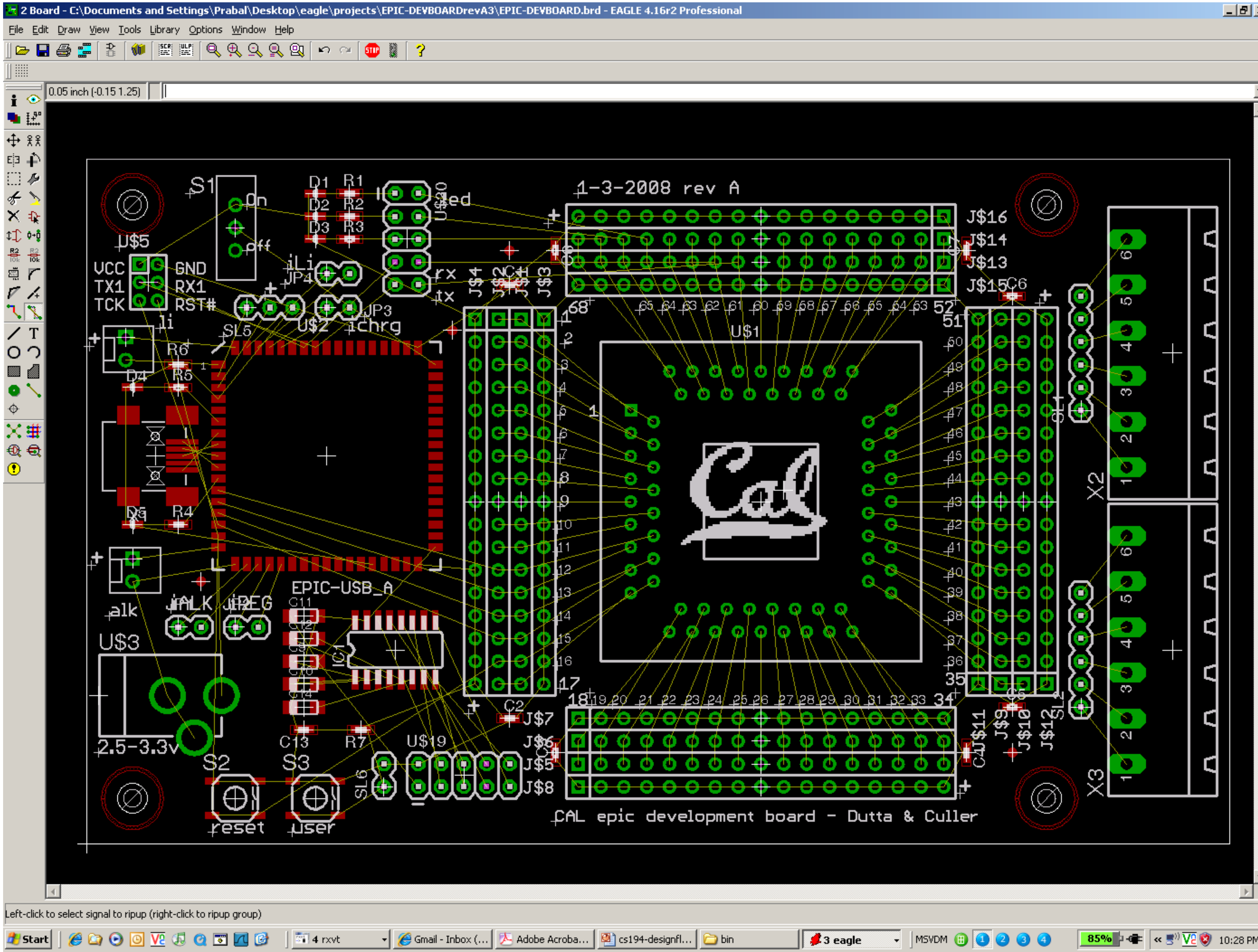
- Once the design is done, a set of files are generated.
  - Each file describes something different (e.g.)
    - Copper on a given layer
    - Silkscreen
    - Solder mask
  - Most files are in “Gerber” format
    - Human-readable (barely) ASCII format
    - Has commands like *draw* and *fill*.
  - Drill files are a different format called Excellon
    - Also human-readable (barely) ASCII with locations and diameters for the holes.
- Generally you zip all these files up and ship them as a single file to the PCB manufacturer.
  - Often a good idea to include the design file(s) too.

The schematic captures the logical circuit design



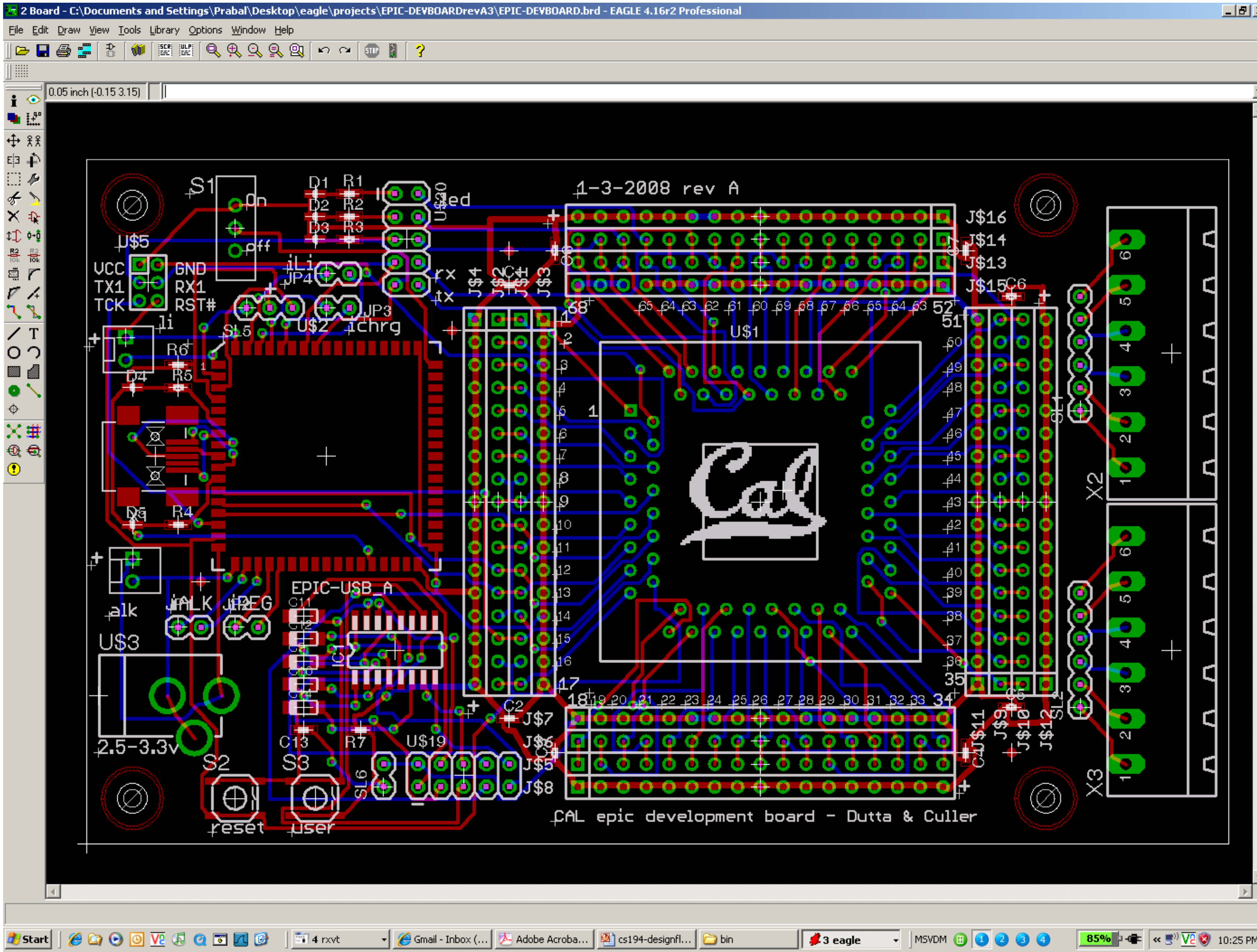
PCBs – closing example

Floorplanning captures the desired part locations

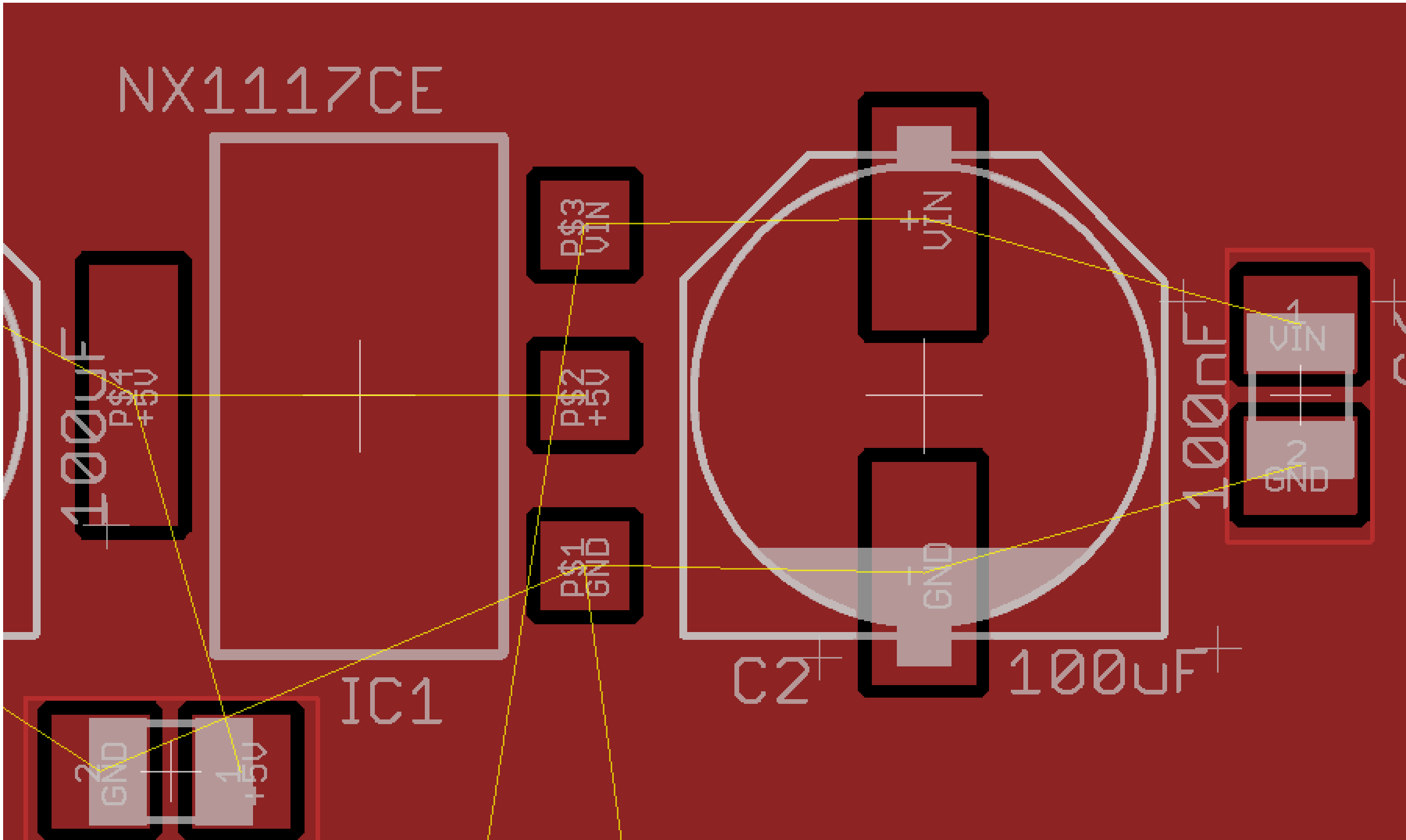


PCBs – closing example

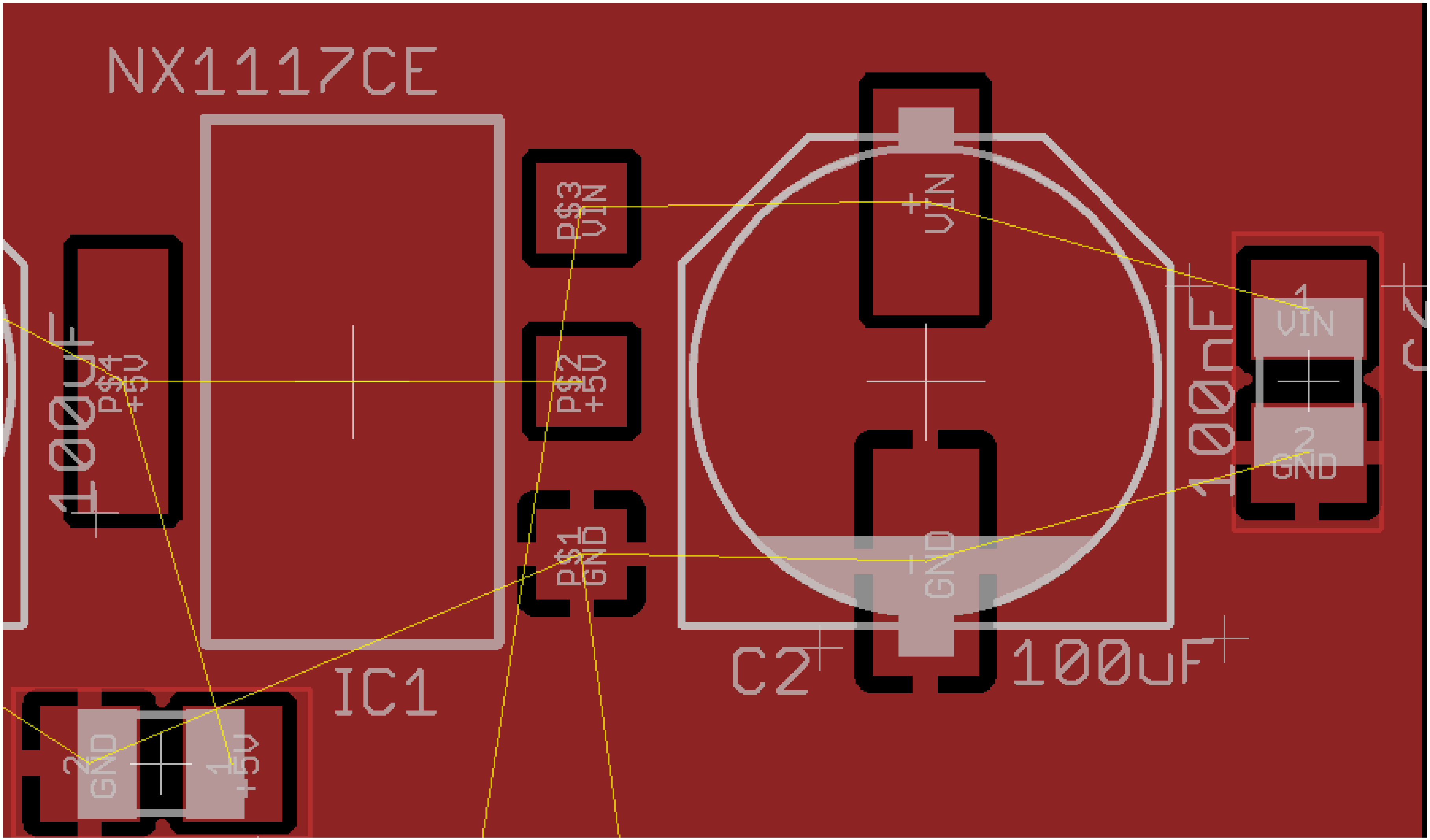
The auto-router places tracks on the board, saving time



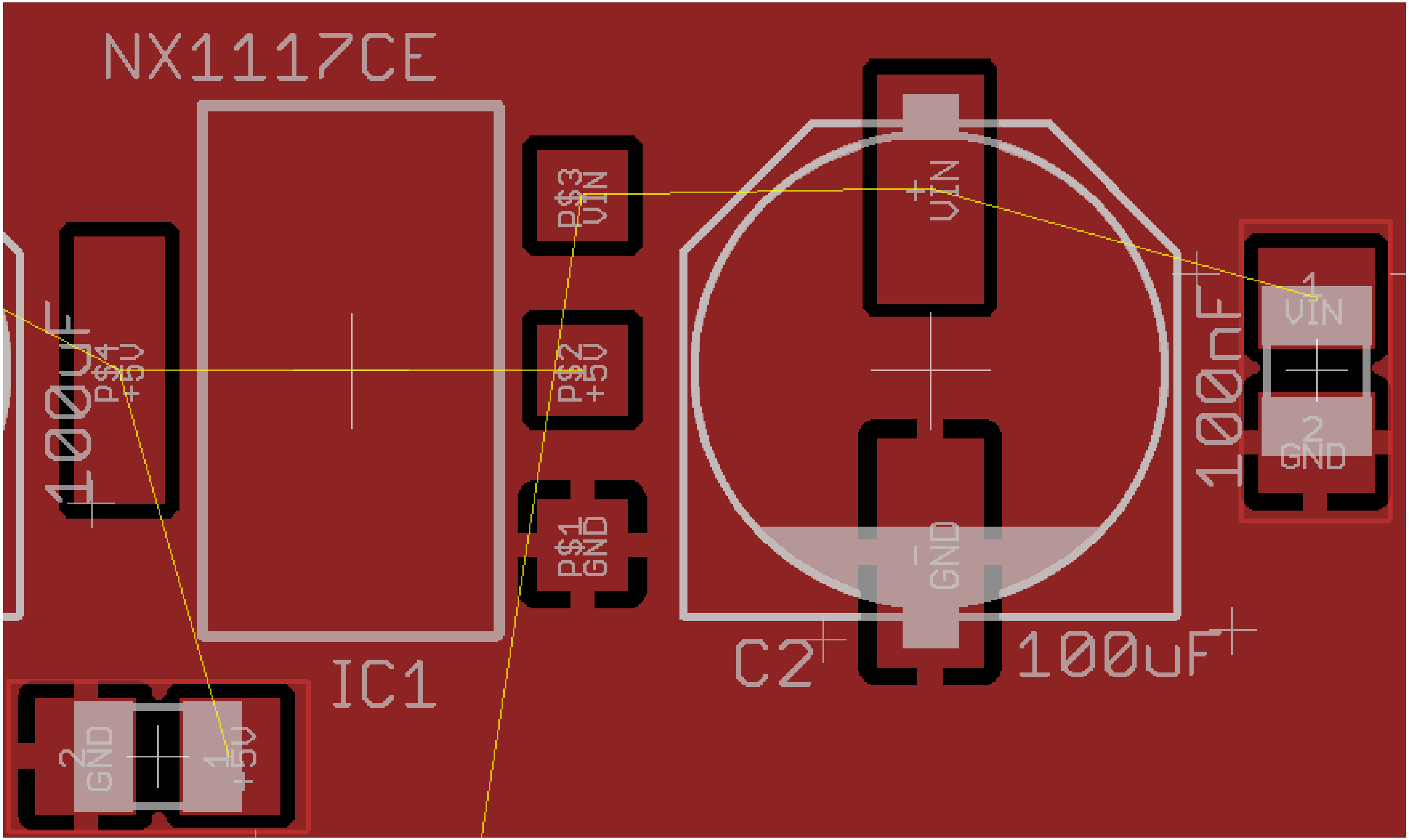
Example: Metal plane no name



Changed name of plane to GND

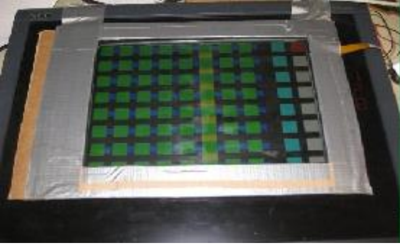


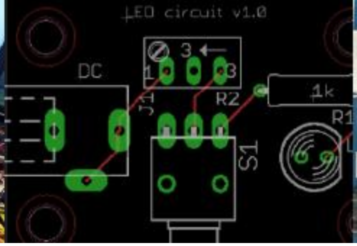
Ran “ratsnest” command



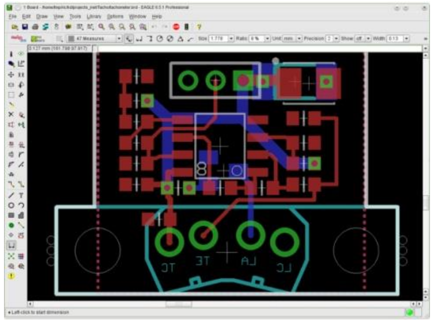
Much material taken from others:

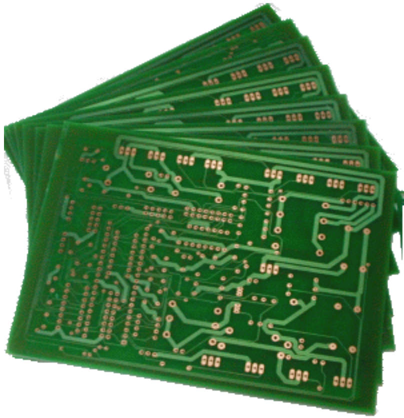
- http://alternatezone.com/electronics/files/PCBDesignTutorialRevA.pdf
  - **Very** nice tutorial/overview
  - Seems to have strong viewpoint
- http://www.goldengategraphics.com/pcgloss.htm
  - Some definitions taken verbatim.
- Dr. Prabal Dutta
- Wikipedia
- And others where noted.





Some electrical issues related to the building of PCBs





High-speed PCB design issues

- There are a lot of electrical issues to deal with when working with high-speed PCBs.
  - **Supplying power, storing energy and dissipating heat**
    - Power supplies, batteries, and heat sinks.
  - **Power Integrity (PI)**
    - We need to be sure that we keep the power and ground at approximately constant values.
  - **Signal Integrity (SI)**
    - We need to make sure data on the wires gets there.
  - **Electro-magnetic interference/compatibility (EMI/EMC)**
    - We need to watch out for generating radio-frequency noise
      - The FCC is a bit picky about this.
    - We don’t want RF noise to interfere with us.

Outline

- Background
  - Understanding power and energy
  - Current limits

- EECS 215 “review/introduction”
  - On capacitors, inductors, resistors and impedance

- Power integrity (PI)

Power!

- Electric power is the *rate* at which electric energy is transferred by an electric circuit. The SI unit of power is the Watt. (Wikipedia)

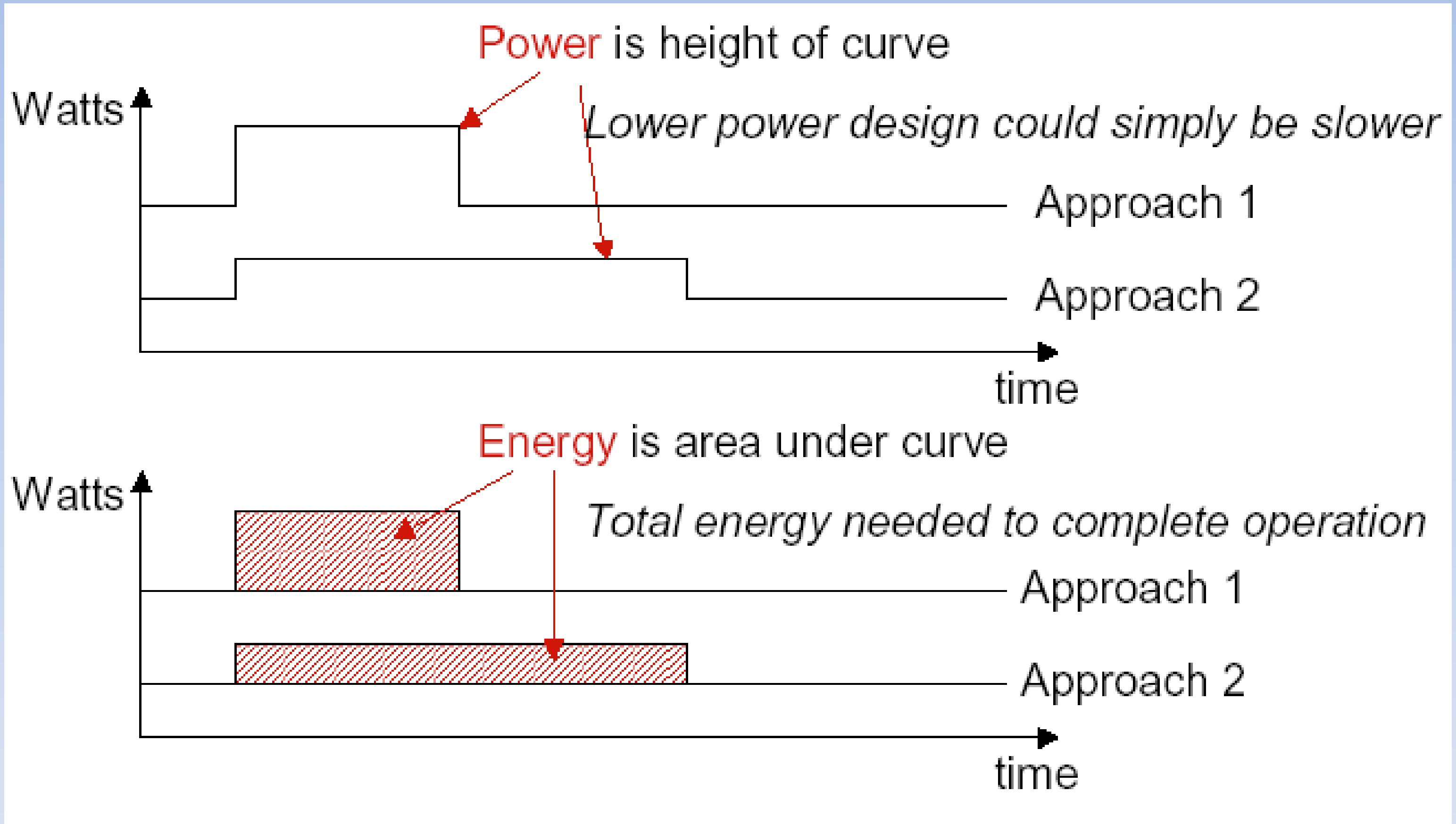
$$\mathrm{W} = \frac{\mathrm{J}}{\mathrm{s}} = \frac{\mathrm{N}\cdot\mathrm{m}}{\mathrm{s}} = \frac{\mathrm{kg}\cdot\mathrm{m}^2}{\mathrm{s}^3}$$

- Power (as opposed to energy), in-and-of itself *is* important in embedded system design.
  - For example there may be a limit on power draw from a given set of batteries.
    - That is, they can't supply energy at more than a given rate.
  - Melting issues are power issues
    - Admittedly over time.

Power vs. Energy

- Power consumption in Watts
  - Determines battery life in hours
  - Sets packaging limits
- Energy efficiency in Joules
  - Rate at which power is consumed over time
  - Energy = power * delay (Joules = Watts * seconds)
  - Lower energy number means less power to perform a computation at same frequency

Power vs. Energy

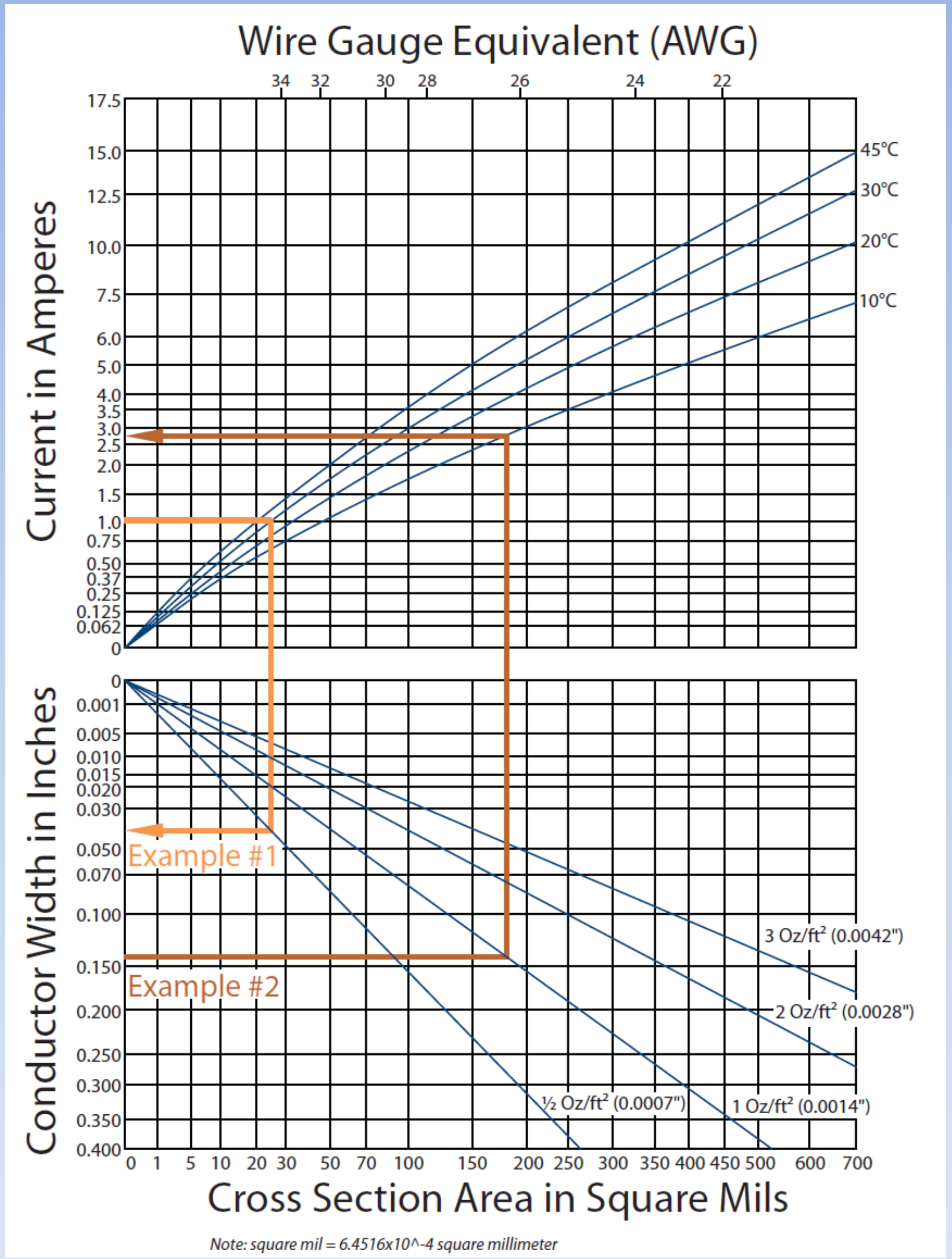


Current limits

- While most of you won't be doing high-power work, sometimes you do have high-power motors, boards, or other things.
  - If you push too much current through a trace, it can melt.
    - Need to worry about "thermal runaway"
  - There are lots of formulas, but let's do a chart version.
    - Assumes max temp you can deal with is 105 degrees C.

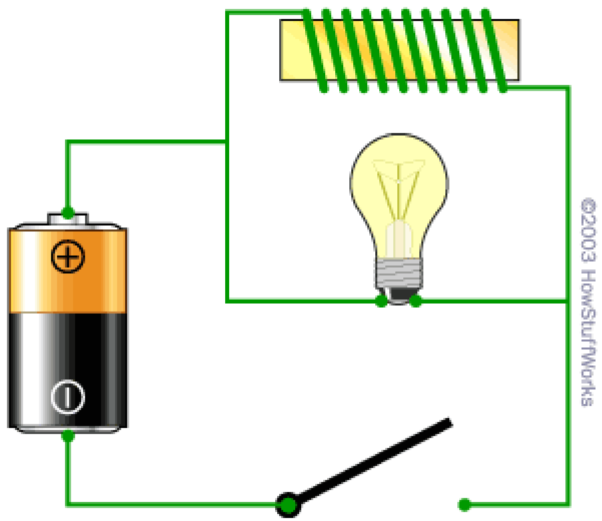
Minimum Trace width for a given current

- Example #1
  - Say you have 1 amp current and you don’t want to see a rise of more than 30 degrees C.
  - Your wire has a thickness of 0.0007 inches (fairly standard)
  - So you need a width of around 40 thou.
- What is Example #2?



Background issue #1:
Inductance

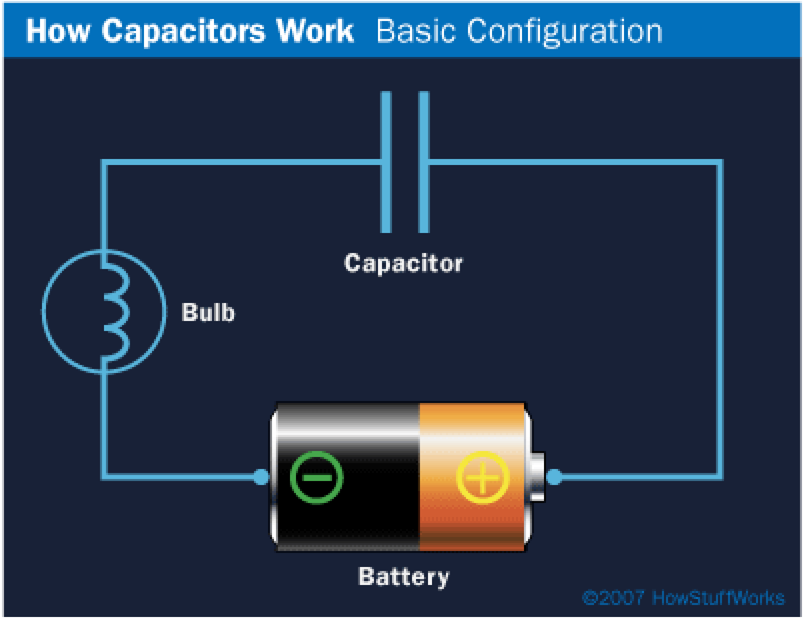
- An inductor “**resists the change in the flow of electrons**”



- The light bulb is a resistor. The wire in the coil has much lower resistance (it's just wire)
  - so what you would expect when you turn on the switch is for the bulb to glow very dimly.
- What happens instead is that when you close the switch, the bulb burns brightly and then gets dimmer.
  - And when you open the switch, the bulb burns very brightly and then quickly goes out.

http://electronics.howstuffworks.com/inductor1.htm

Background issue #2: **Capacitance**



- A capacitor resists the change of voltage
  - When you first connect the battery, bulb lights up and then dims
  - If you then remove the battery and replace with a wire the bulb will light again and then go out.

http://electronics.howstuffworks.com/capacitor1.htm

Background issue #3: **Impendence**

- Impedance (symbol Z) is a measure of the overall opposition of a circuit to current, in other words: how much the circuit **impedes** the flow of current.
  - It is like resistance, but it also takes into account the effects of capacitance and inductance. I
  - Impedance is measured in ohms.
  - Impedance is more complex than resistance because the effects of capacitance and inductance vary with the frequency of the current passing through the circuit and this means **impedance varies with frequency**!
    - The effect of resistance is constant regardless of frequency.

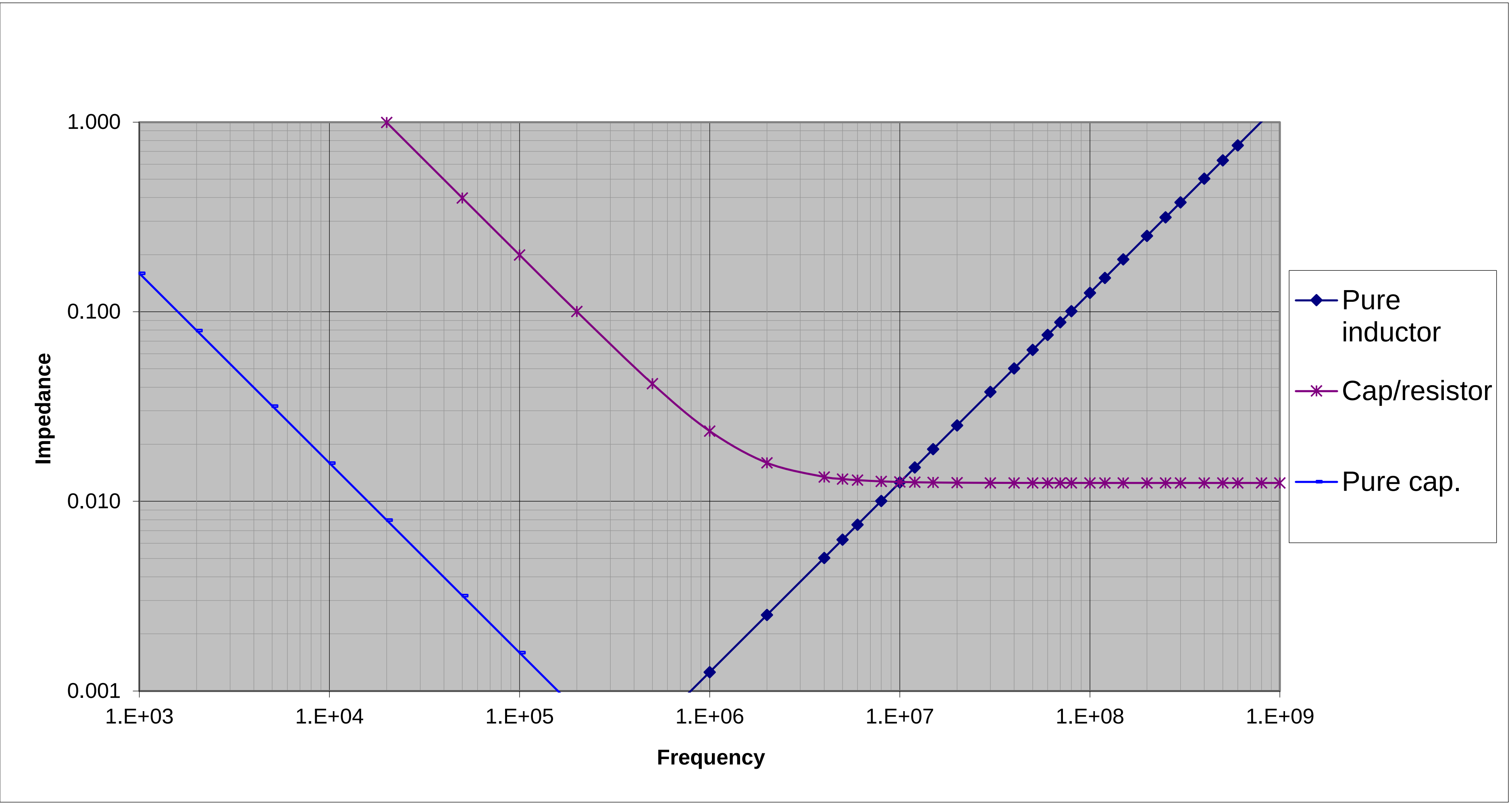
http://www.kpsec.freeuk.com/imped.htm

Impedance vs frequency

- Say you have a sine wave being driven over a device.
  - How does the impedance of each device vary with frequency?
- Resistors have an impedance that is independent of frequency
- Capacitors have a lower impedance as frequency goes up.
- Inductors have a higher impedance as frequency goes up.

A look at impedance

(with capacitors, inductors and resistors vs. frequency)



Notice the log scales!

Power Integrity

- In order to get digital electronics to work correctly, they need a minimum voltage differential.
  - If we get below that, the devices might
    - Be slow (and thus not meet setup times)
    - Lose state
    - Reset or halt
    - Just plain not work.
- Even a very (very) short “power droop” can cause the chip to die.
  - In my experience, this is a really common problem.
- Keeping power/ground constant and noise/droop free is “**Power Integrity**”

So?

- We need the Vcc/Ground differential to be fairly constant.
  - But rapid changes in the amount of current needed will cause the voltage to spike or droop due to inductance.
- We basically want a "no-pass" filter.
  - That is we don't want to see *any* signal on the Vcc/Ground lines.
  - The obvious thing?
    - "Add a capacitor"
      - That should keep the voltage constant, right?
    - The problem is we need to worry about a lot of frequencies AND capacitors aren't ideal.

Lots of frequencies

- Even fairly slow devices these days are *capable* of switching at very high frequencies.
  - Basically we get drivers that have rise and fall times capable of going 1GHz or so.
- This means we generally have to worry about frequencies from DC all the way to 1GHz.
  - Because our chip may be varying its draw at rates up to that fast.

Non-ideal devices.

			Each			Total		
Device		Quantity	Cap	ESR	ESL	Ceff	ESR	ESL
DC/DC Converter		1		1.00E-04	7.00E-07		1.00E-04	7.00E-07
Tantalum Capacitors		3	3.30E-04	3.00E-02	3.00E-09	9.90E-04	1.00E-02	1.00E-09
0603 Ceramic Caps.		2	1.00E-06	3.00E-02	7.00E-10	2.00E-06	1.50E-02	3.50E-10
0603 Ceramic Caps.		8	1.00E-07	6.00E-02	6.00E-10	8.00E-07	7.50E-03	7.50E-11
0603 Ceramic Caps.		8	1.00E-08	9.00E-02	5.00E-10	8.00E-08	1.13E-02	6.25E-11
0603 Ceramic Caps.		10	1.00E-09	1.50E-01	5.00E-10	1.00E-08	1.50E-02	5.00E-11
PC Board		1	8.50E-08			8.50E-08		

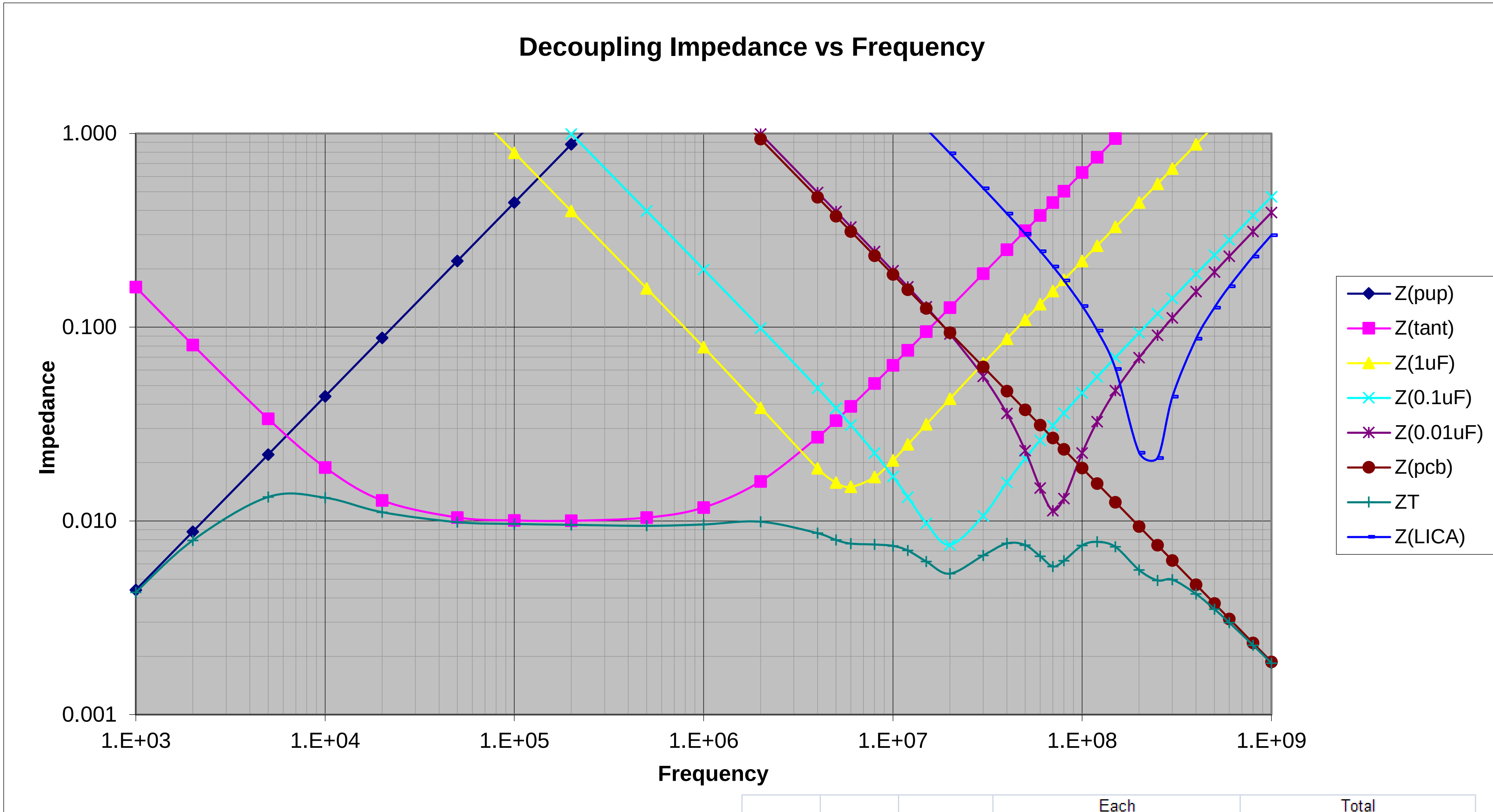
- ESR is Effective Series Resistance
- ESL is Effective Series Inductance
- Ceff is the effective capacitance.
  - How does quantity effect these values?
- Obviously impendence will be varying by frequency.

Other things can add to ESR/ESL

- Generally a bad solder job can make ESR/ESL worse.
- Packaging has an impact
  - wires have inductance so surface-mount packages preferred
- Pads can have an impact

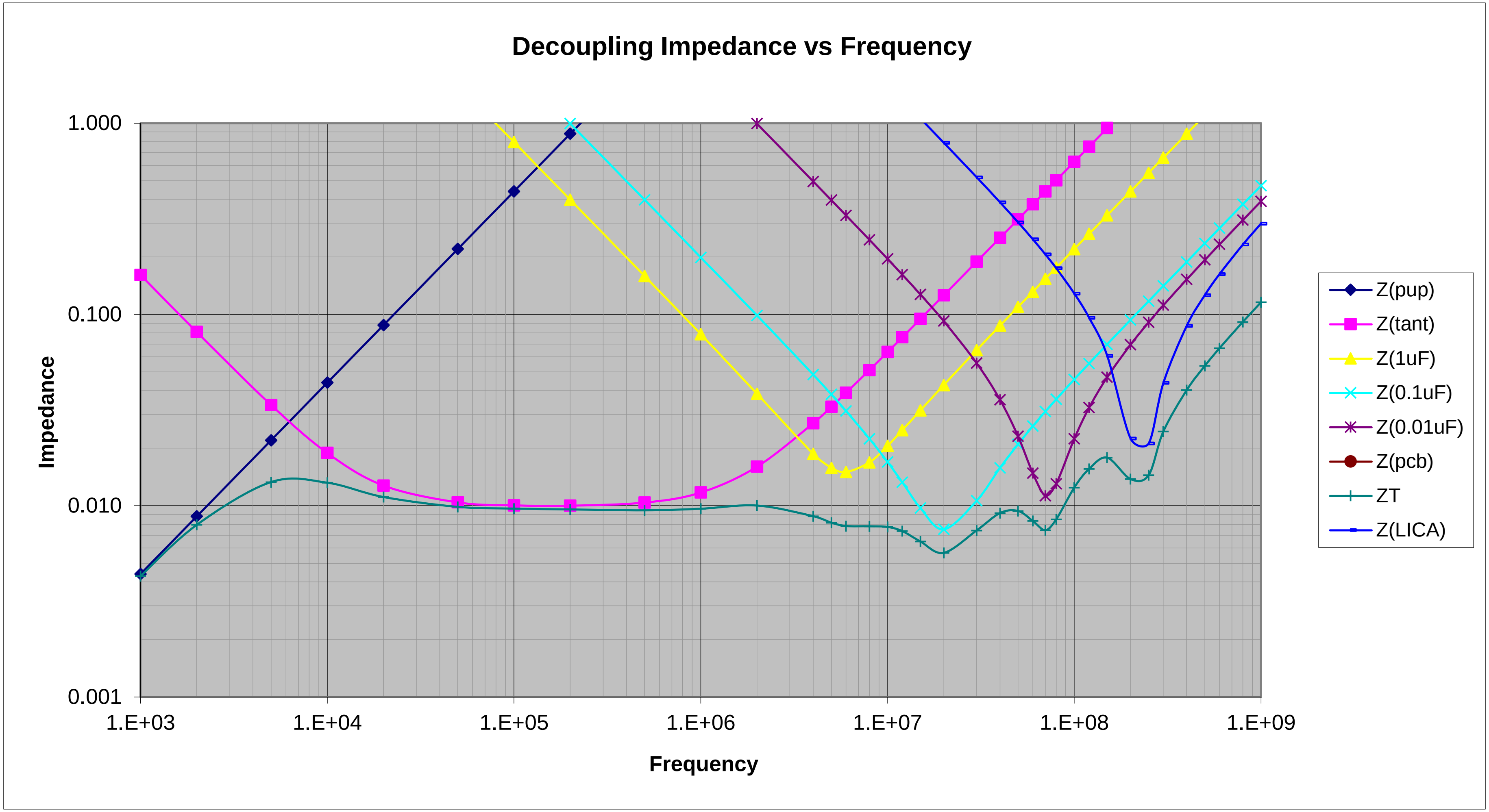
Power Integrity

Given the previous table..

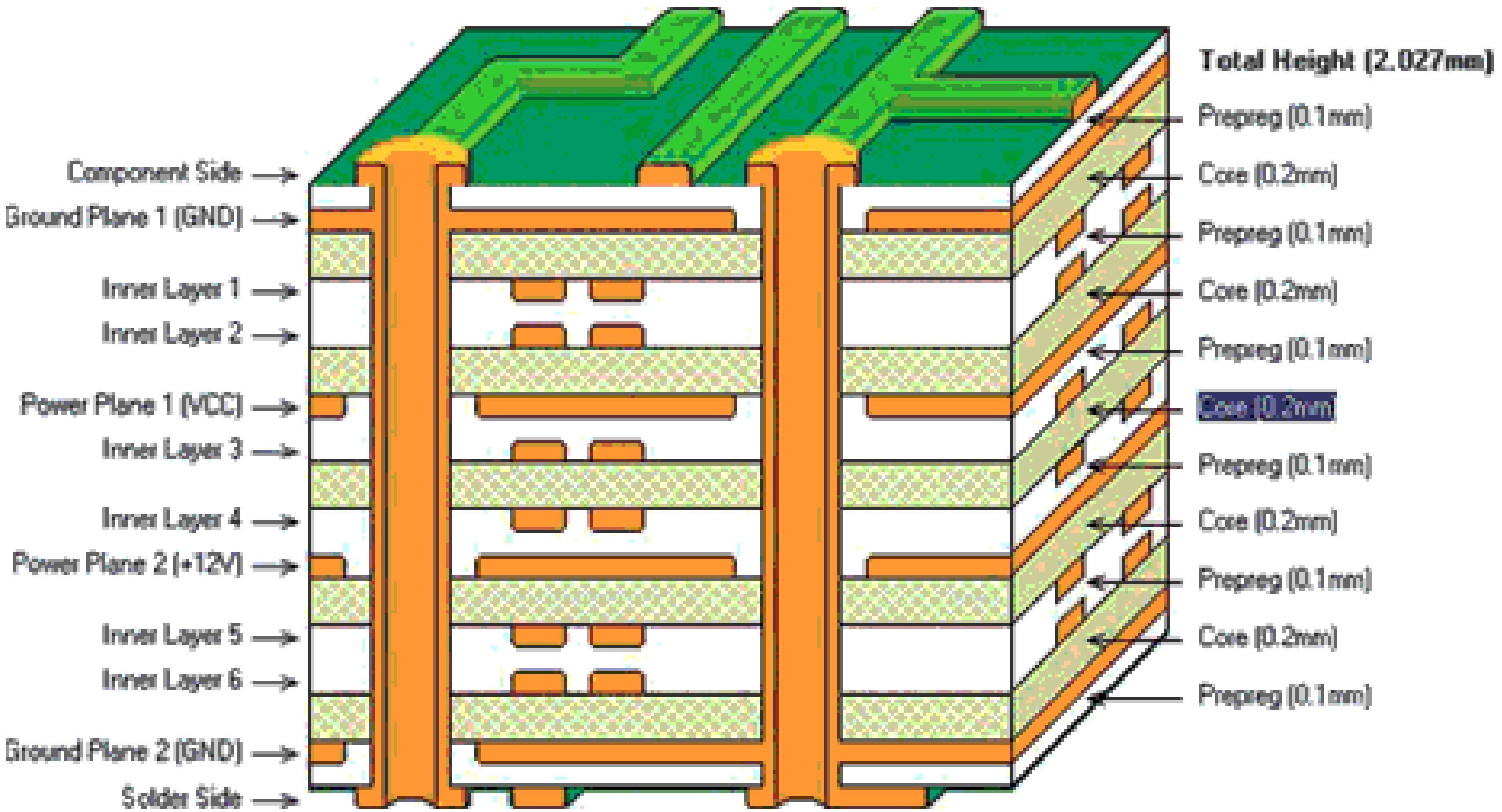


			Each			Total		
Device		Quantity	Cap	ESR	ESL	Ceff	ESR	ESL
DC/DC Converter		1		1.00E-04	7.00E-07		1.00E-04	7.00E-07
Tantalum Capacitors		3	3.30E-04	3.00E-02	3.00E-09	9.90E-04	1.00E-02	1.00E-09
0603 Ceramic Caps.		2	1.00E-06	3.00E-02	7.00E-10	2.00E-06	1.50E-02	3.50E-10
0603 Ceramic Caps.		8	1.00E-07	6.00E-02	6.00E-10	8.00E-07	7.50E-03	7.50E-11
0603 Ceramic Caps.		8	1.00E-08	9.00E-02	5.00E-10	8.00E-08	1.13E-02	6.25E-11
0603 Ceramic Caps.		10	1.00E-09	1.50E-01	5.00E-10	1.00E-08	1.50E-02	5.00E-11
PC Board		1	8.50E-08			8.50E-08		

Removing the PCB…

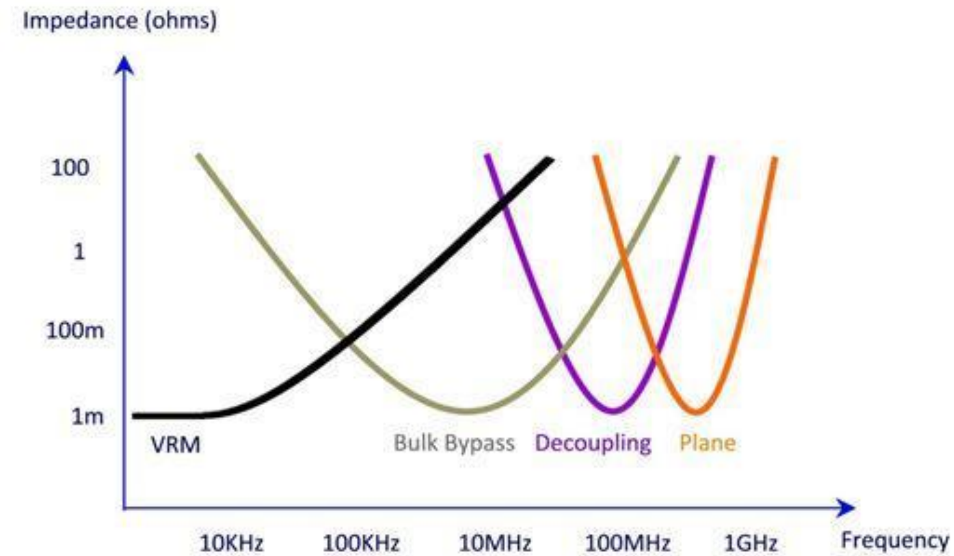


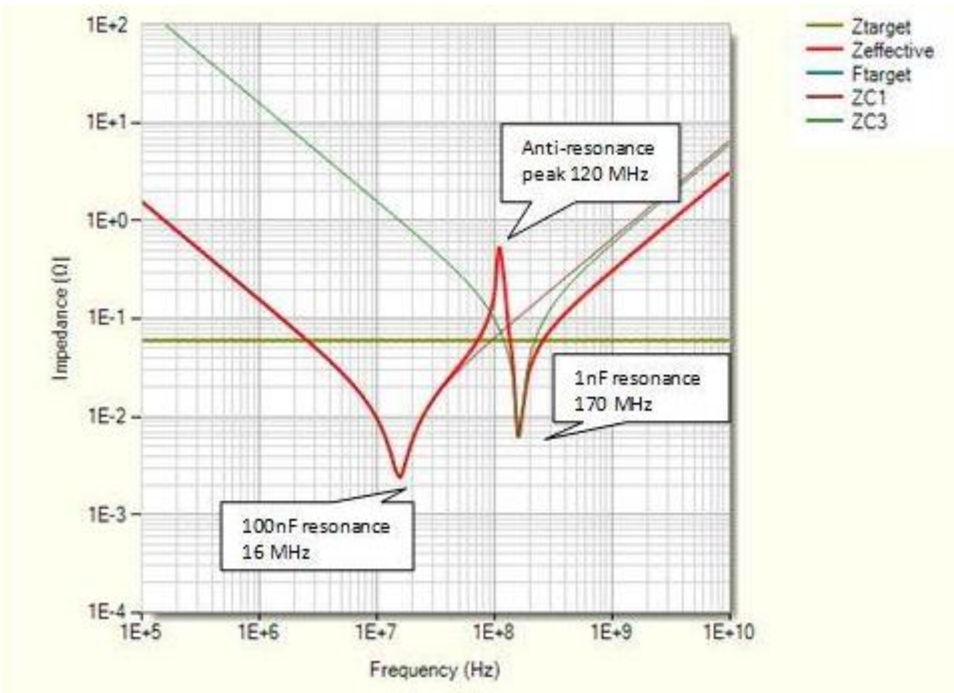
What is the PCB part?



But wait…

- VRM
  - Voltage regulator module
- bulk bypass (tantalum) and decoupling capacitors (ceramic).
  - These capacitors supply instantaneous current (at different frequencies) to the drivers until the VRM can respond.
- However sets of different capacitors cause problems!





http://www.pcbdesign007.com/pages/columns.cgi?artcatid=0&clmid=65&artid=85396&pg=3&_pf_=1

Power Integrity (PI) summary

- Power integrity is about keeping the Vcc/ground difference constant.
  - This is hard because the devices that sink power do so in "pulses" due to their own clocks
  - Need caps to keep value constant
    - But parasitic ESR/ESL cause problems
    - So lots of them==good
      - Reduce ESR/ESL
      - Increase capacitance.
  - But anti-resonance can cause problems!
    - Need Spice or other tools to model.
      - Will do a bit of this next time