EECS 373

Design of Microprocessor-Based Systems

Robert Dick
University of Michigan

Lecture 12: Memory and PCBs

16 February 2017

Slides inherited from Mark Brehob.

Outline

- Context and review
- Memory
- PCB design

Context and review

- ADC and DAC operation and error
  - Speed / accuracy / monotonicity trade-offs.
- Prototyping
  - Breadboards.
  - Soldering.
  - Wire wrap.
- Memory
  - Flash
  - Speed, functionality, reliability trade-offs.

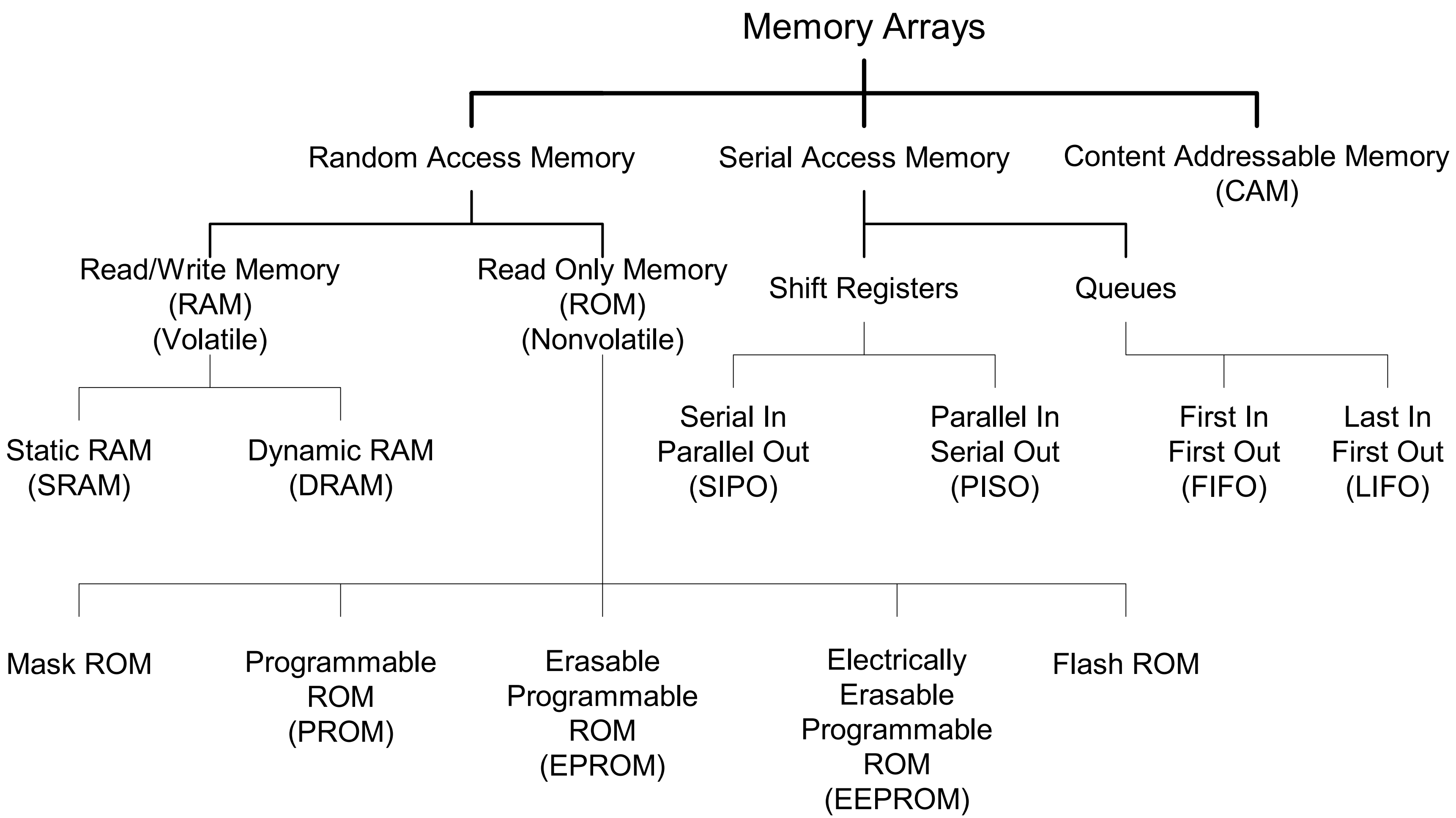
Outline

- ~~Context and review~~
- Memory
- PCB design

Memory: why?

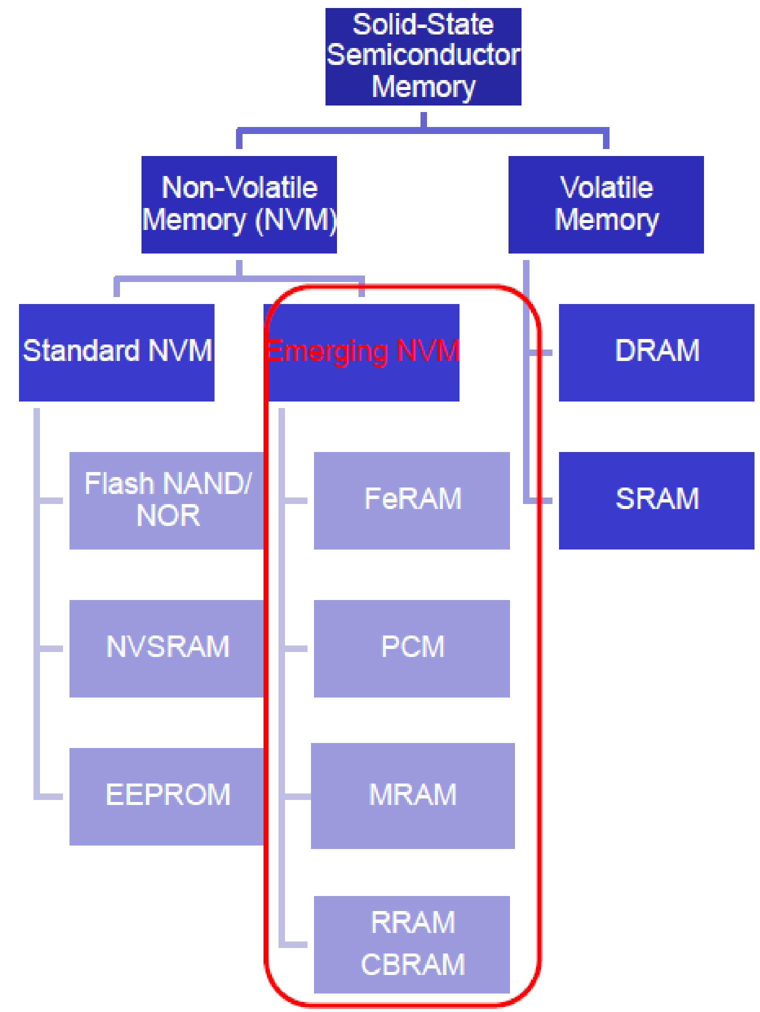
- You'll be dealing with this a lot in your projects.
- A little review now can save you trouble later.

Memory array types



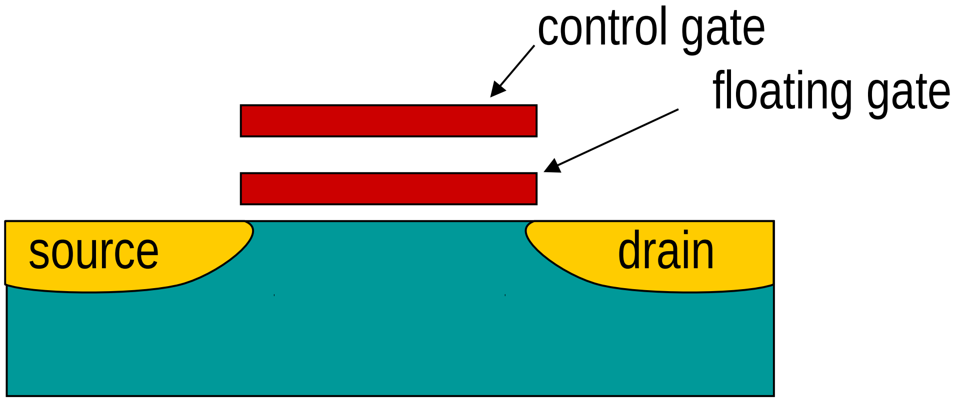
Nonvolatile memory types

- Ferroelectric RAM: polarization changed by electric field. DRAM/flash-like. Electical polarization instead of floating gate. Reading → current pulse on write? 150 ns. 10^{12} cycles.
- Phase change memory: amorphous (off) vs. crystalline (on). 5ns to 100 ns write. ns read. 10^{8} cycles.
- Magnetoresistive RAM. ns read. Slightly more for write. Many cycles.
- Programmable metallization cell: redox filament. 10s of ns read, write. 10^{10} cycles.
- Flash: ns read, ms write.
- EEPROM: ns read, ms write.
- SRAM/DRAM: ns read and write.



Floating gates

- Write: hot-electron injection or Fowler-Nordheim tunneling.
  - High voltage on control gate >> operating voltage
  - Electrons are trapped in the floating gate.
  - Will not discharge for many years.
- Erase? Fowler-Nordheim tunneling.



- Read by seeing whether it acts like a transistor or a wall.
- Tend to self-destruct after 10^5 write/erase cycles.

Outline

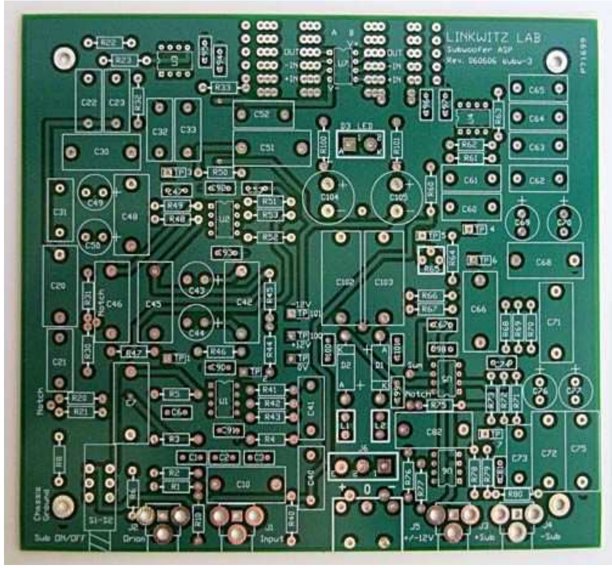
- ~~Context and review~~
- ~~Memory~~
- PCB design

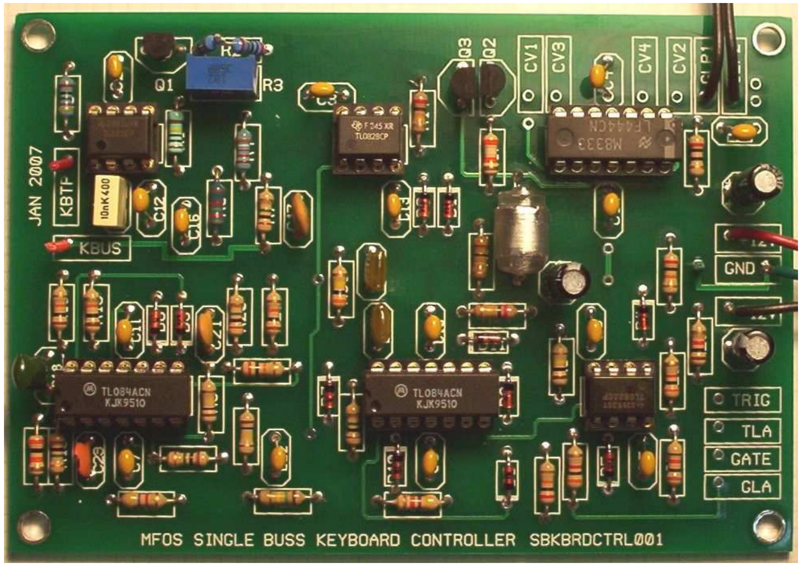
PCBs: why?

- Even if you aren't making one for your project, need to understand how they work for debugging / reverse engineering.

Printed circuit board design

- Physical support.
- Electrical connections.
  - Traces have restricted dimensionality.
  - Very thin, high resistance.
  - Holes/vias and pads.
  - Rework is hard.

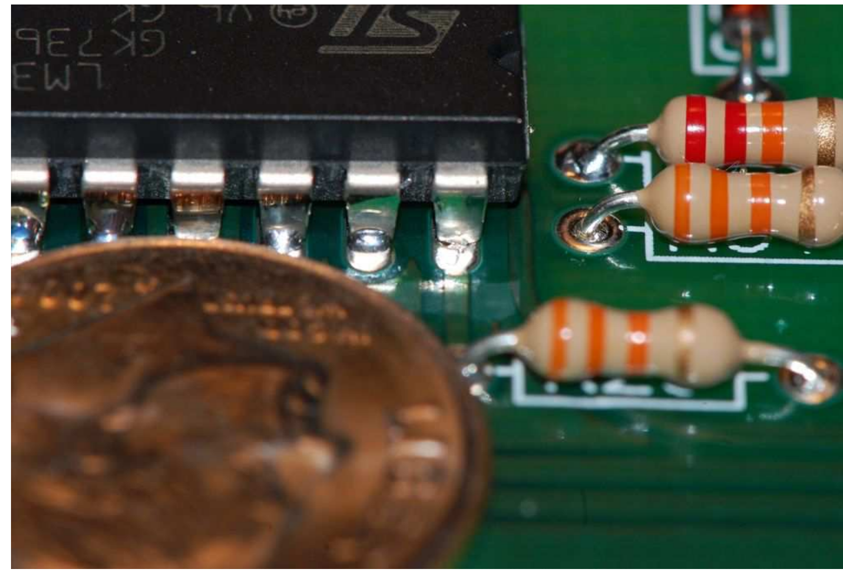


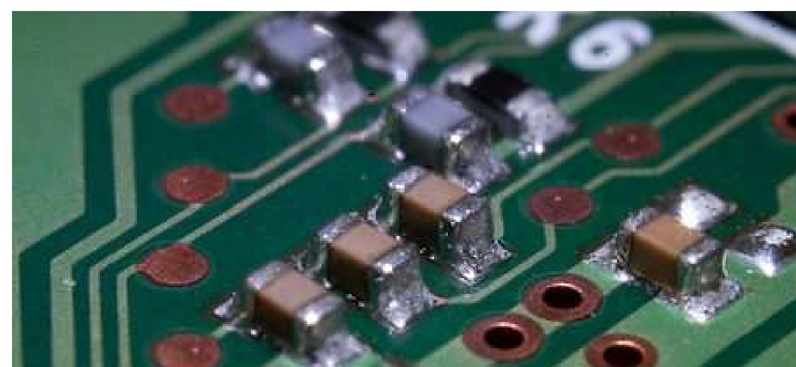


http://www.linkwitzlab.com/Pluto/supplies-subw.htm,
http://www.musicfromouterspace.com/analogsynth/SINGLEBUSSKEYBOARD2007/SINGLEBUSSKEYBOARD2007.php

Basic terminology

- Interconnects: traces.
- Inside of a given “layer” traces which cross are electrically connected.
- Can have multiple conductive layers by stacking/bonding boards.
- Through-hole: Having holes in the PCB designed to have pins put through the holes.
  - Contrast with surface mount where device goes on top.
- Surface mount.



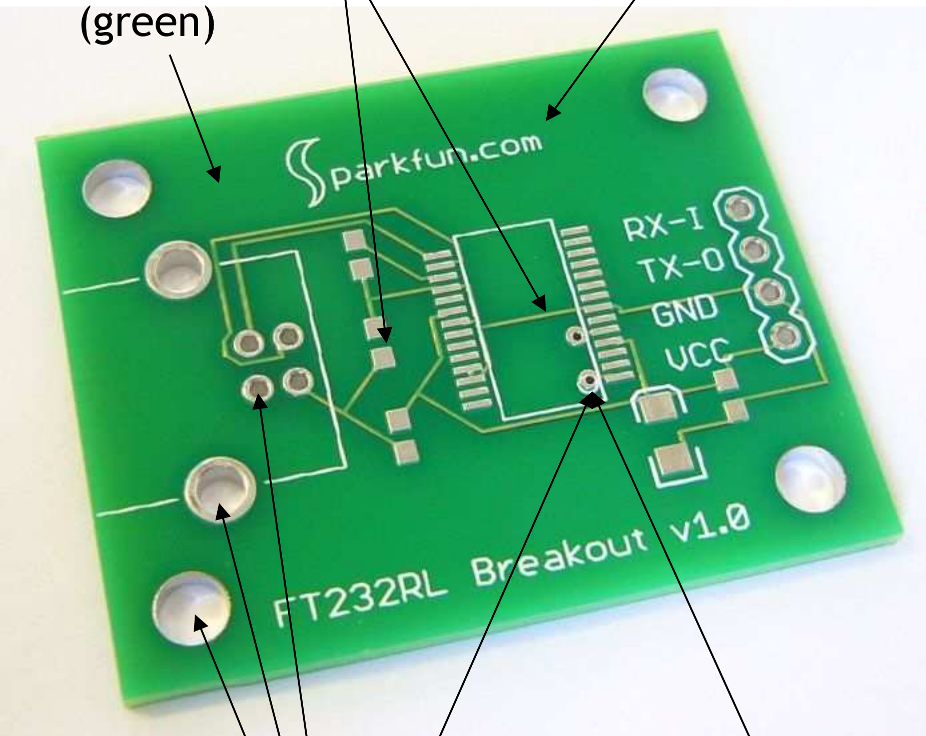


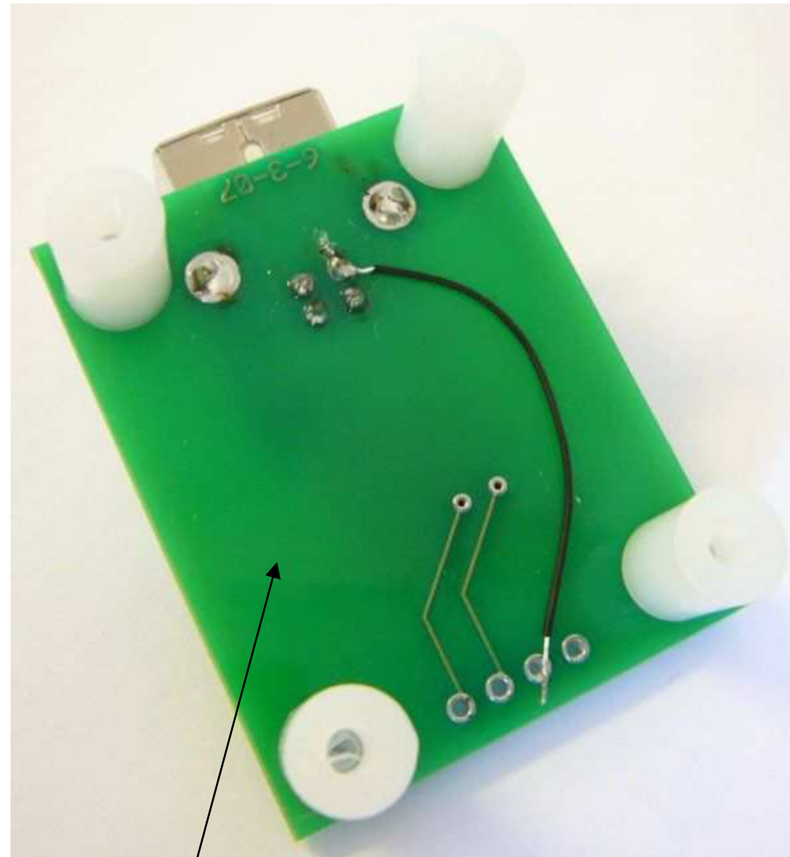
Parts of a PCB

Copper
(pads & traces)

Soldermask
(green)

Silkscreen
(white)





Drill holes

Via

Bottom side

Vias

- Sometimes you need to connect traces on layers.
  - Use a via: a plated through hole
    - Generally smaller than a through hole for a pin.

Clearances

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

The layered construction of a PCB: a six layer board

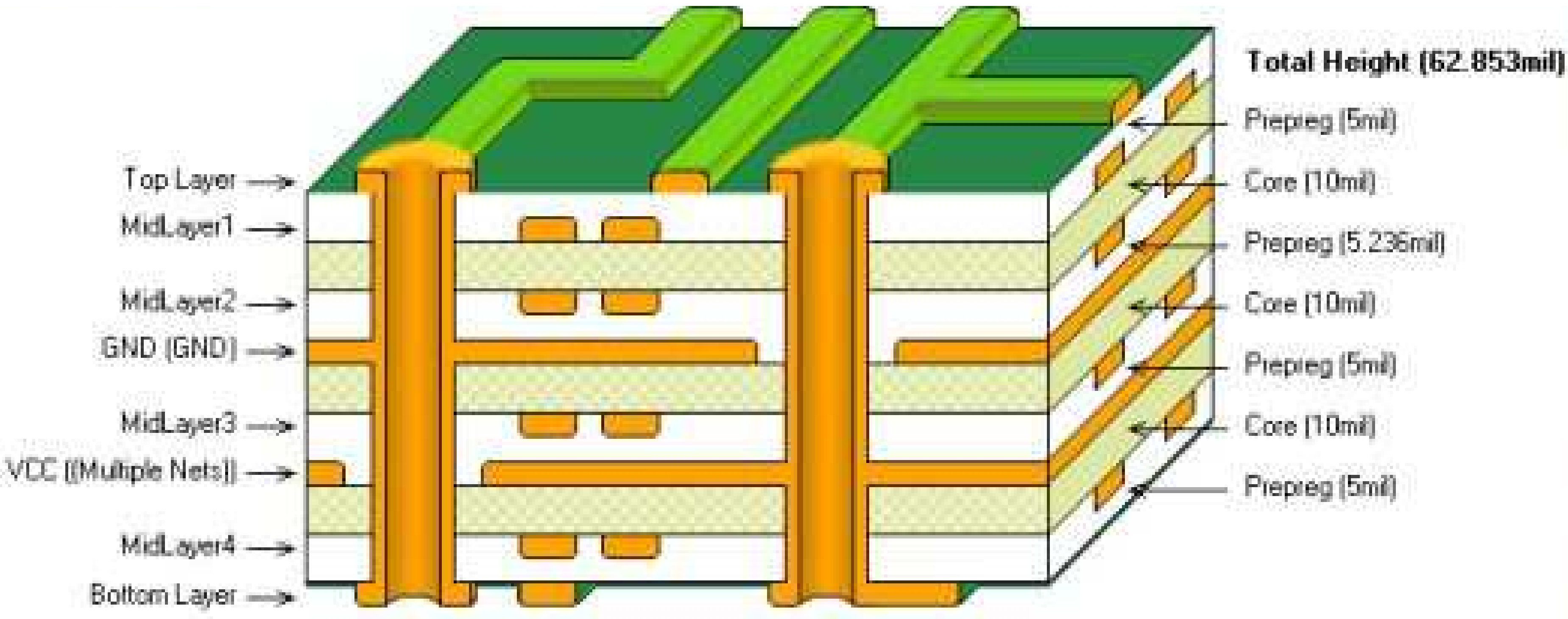


Figure from altium.com

What do do with layers?

- Mostly orthogonal routing layers.
- Ground planes to increase local power supply capacitance and minimize resistance.
- Power plane for similar reason.
- More layers → higher cost.

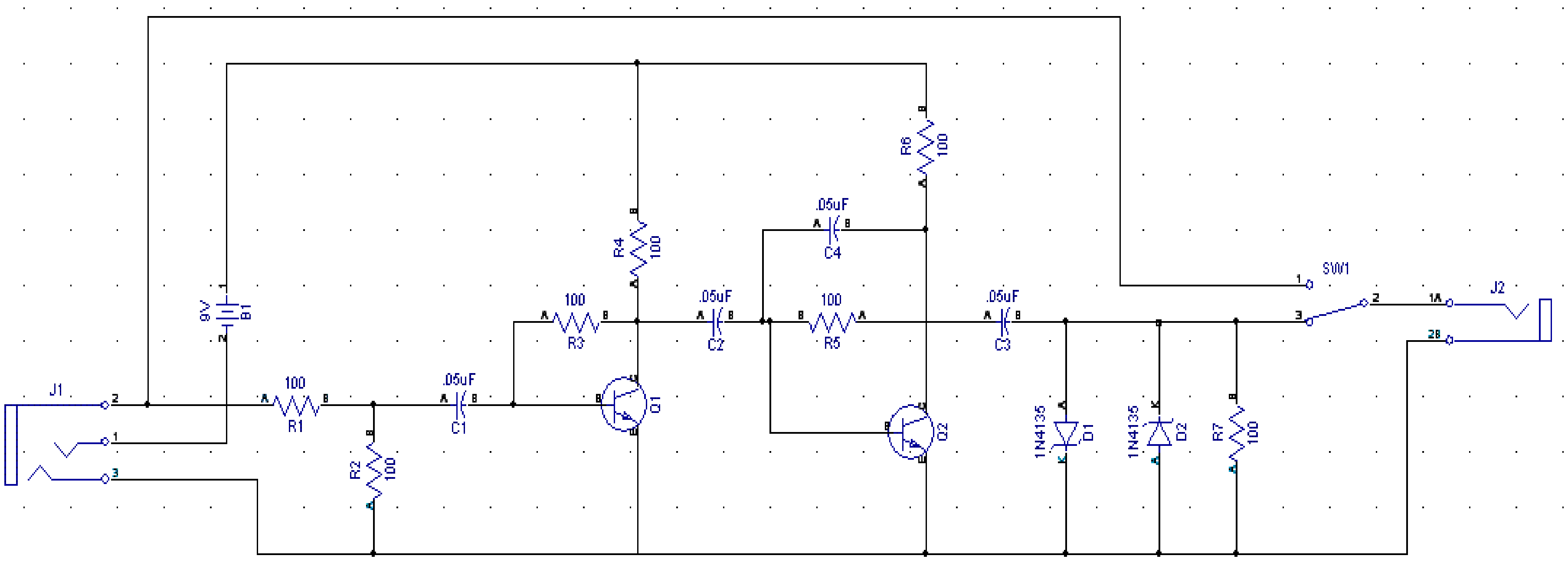
How to design PCB

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, although the schematic connectivity will influence layout.

Example schematic



Why a schematic?

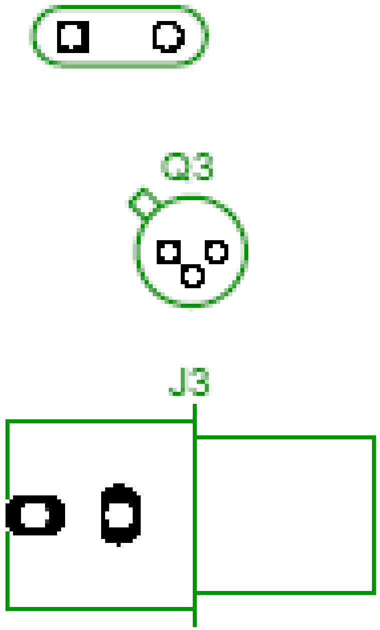
- Communication and formalization are main goals.
  - This is probably what your sketch on paper would look like.
  - You can find and fix bugs more easily here than the PCB layout.

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.
- Sometimes they screw up.

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.
  - 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.
- Rules vary by manufacturer.
- Units vary my manufacturer.

Issues of measure

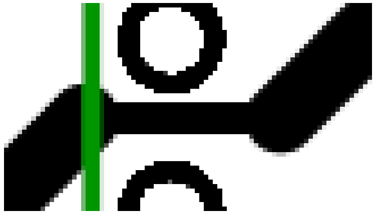
- PCB designers use odd terminology.
- A “thou” is a thousandth of an inch.
- A “mm” is a millimeter
- A “mil” is a thousandth of an inch.
  - Thou is generally preferred over mil to avoid confusion, but most tools/vendors use mil.

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.

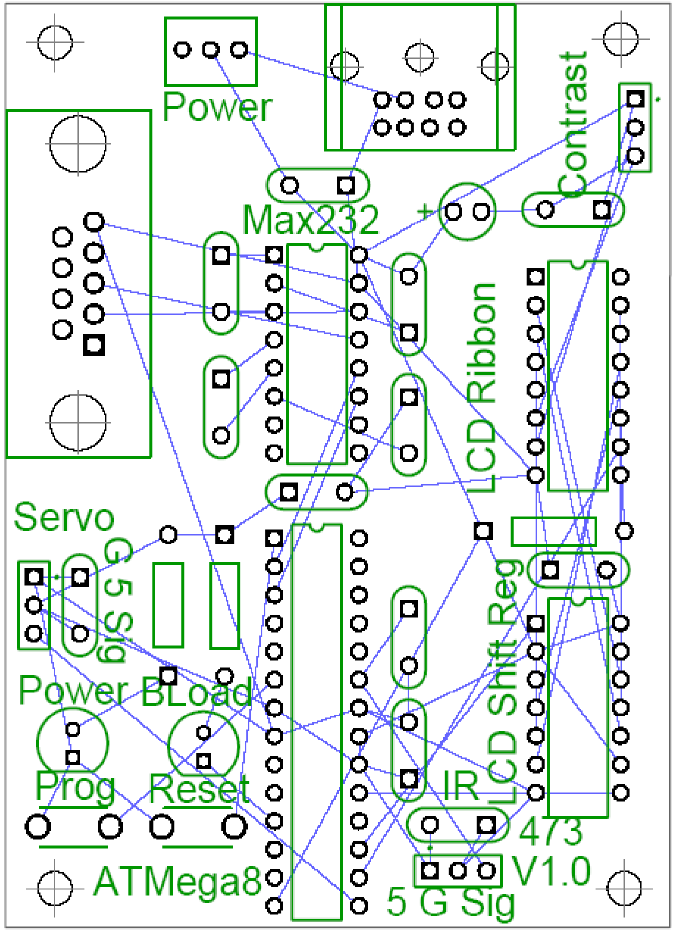
Trace width continued

- 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.



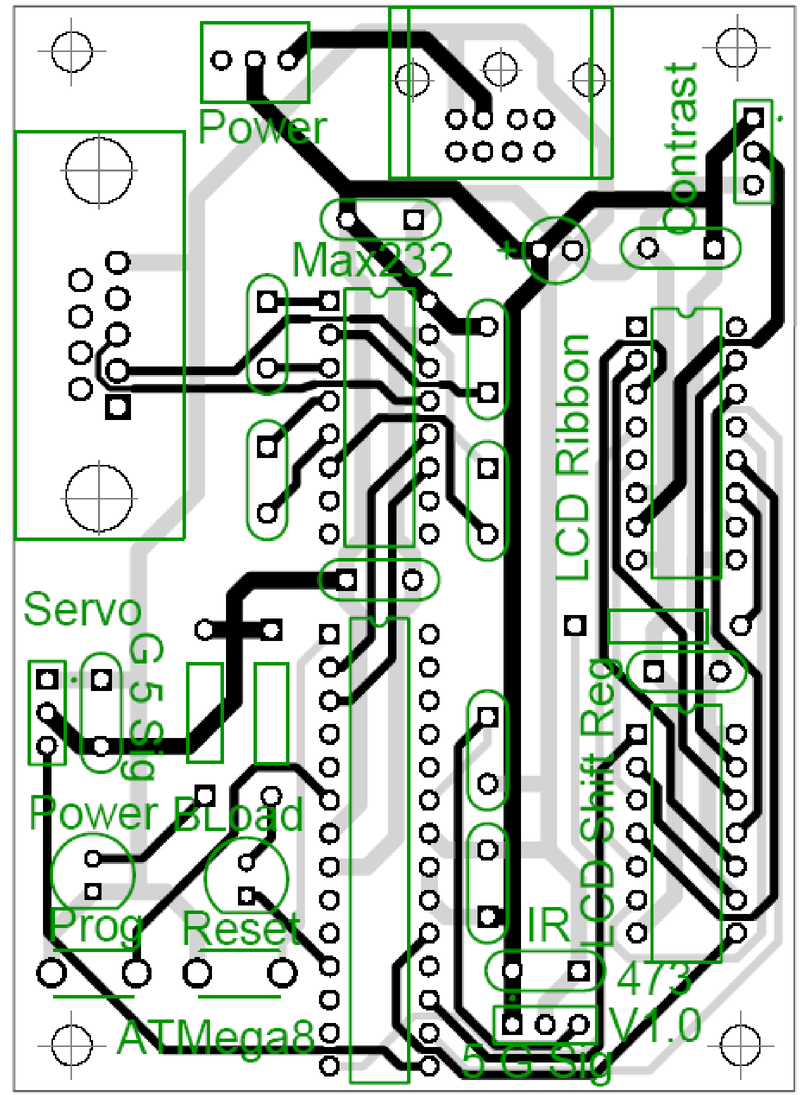
Rat's nest.

- Rat's nest shows device placements and connections but not routing.
  - Automatically generated for you.
  - Sometimes before placement, sometimes after.
  - Varies by tool.

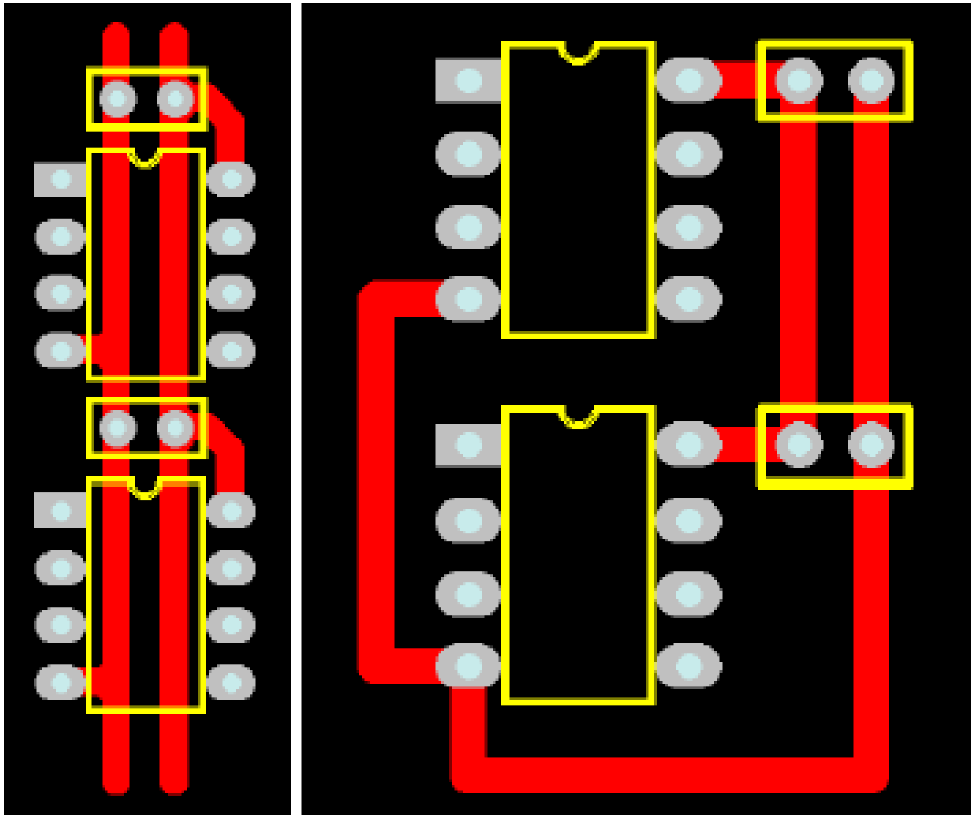


Routing for real

- 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 always as good as a person can do.
    - But much faster.
- Still generally need to do some (or all) of the routing by hand
  - Very, very tedious.



Routing quality

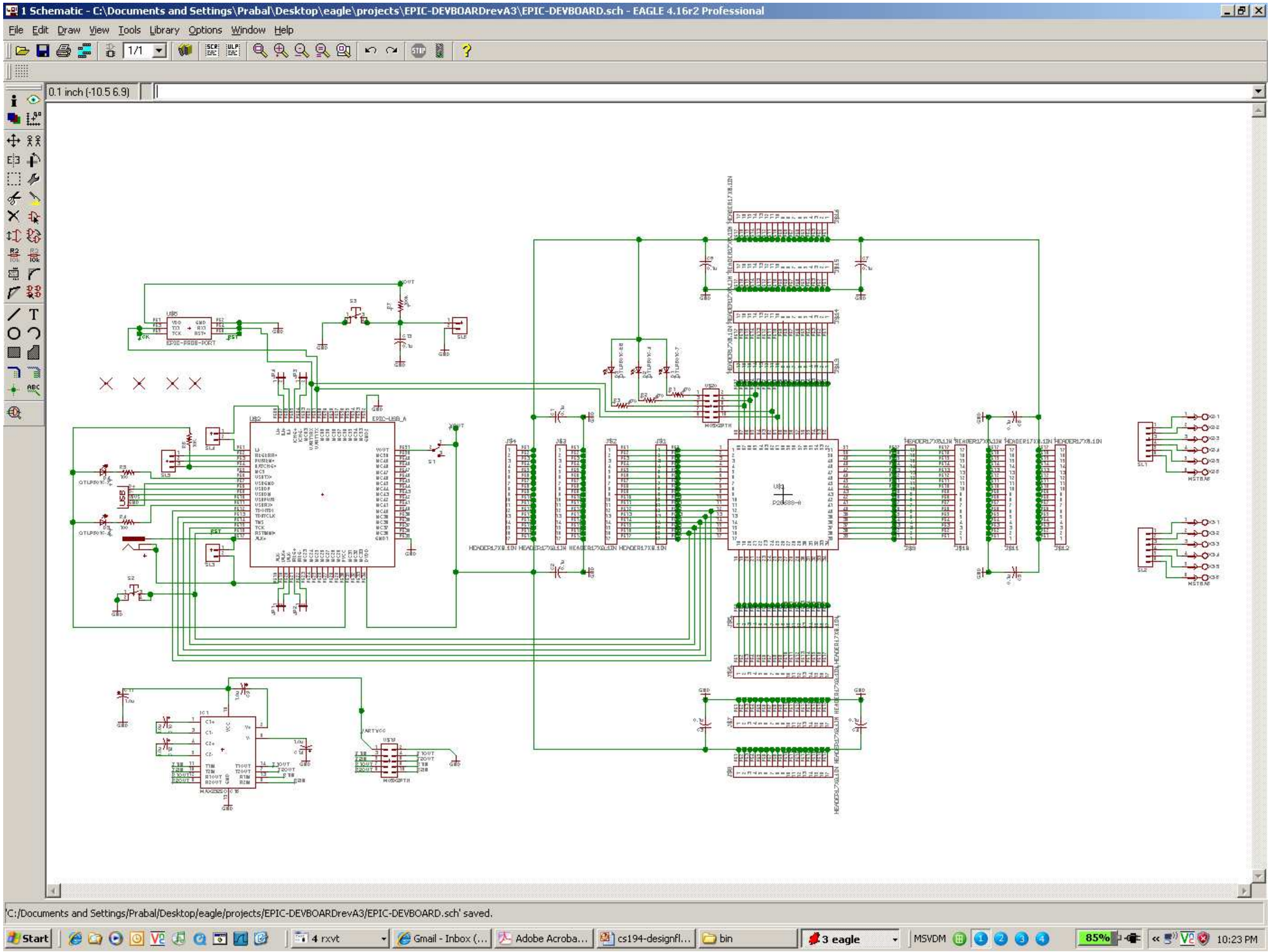


An example of GOOD power routing (Left) and BAD power routing (Right)

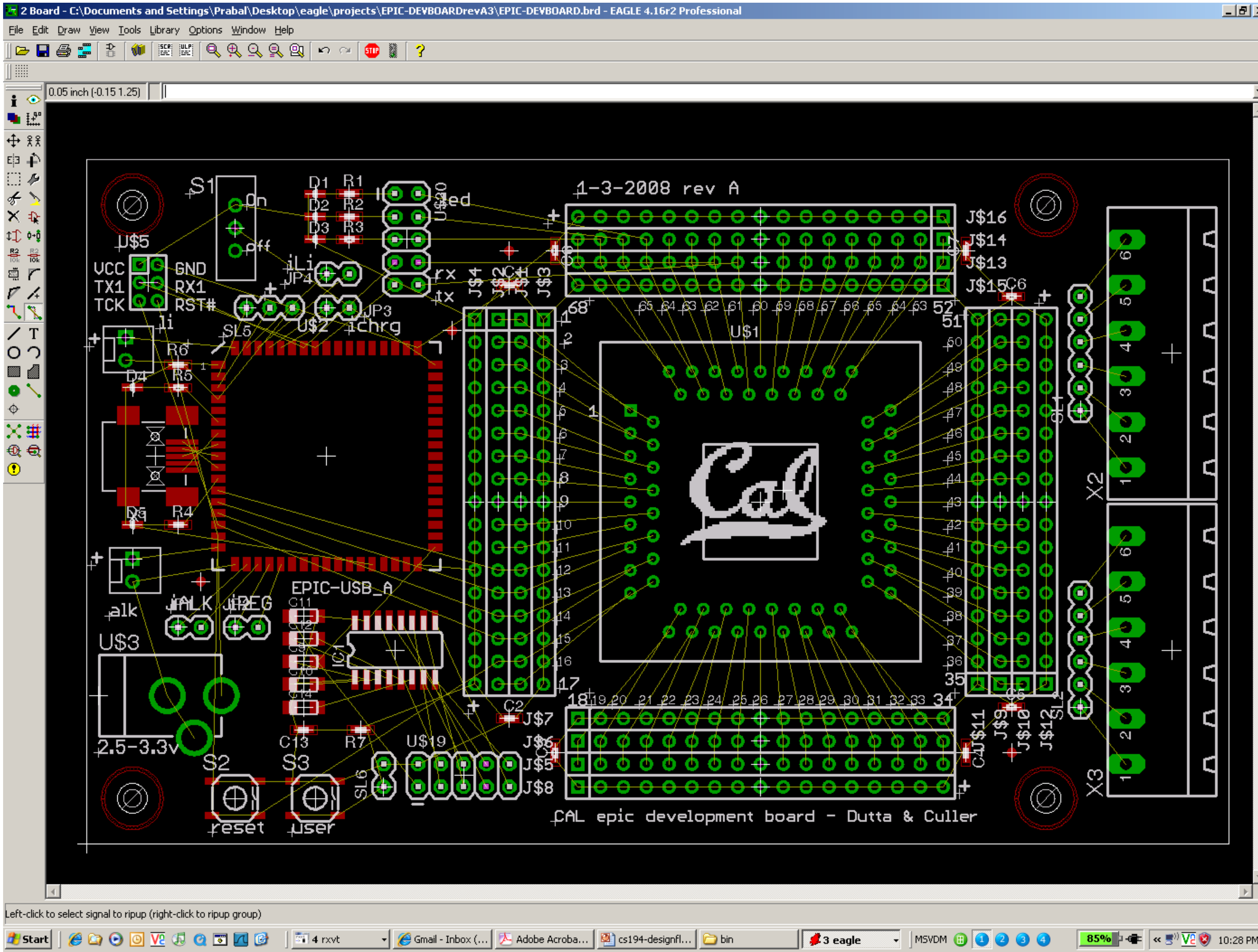
Step 4: Generate files

- Once the design is done, a set of files are generated.
  - Each file describes something different.
    - 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.
    - 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.

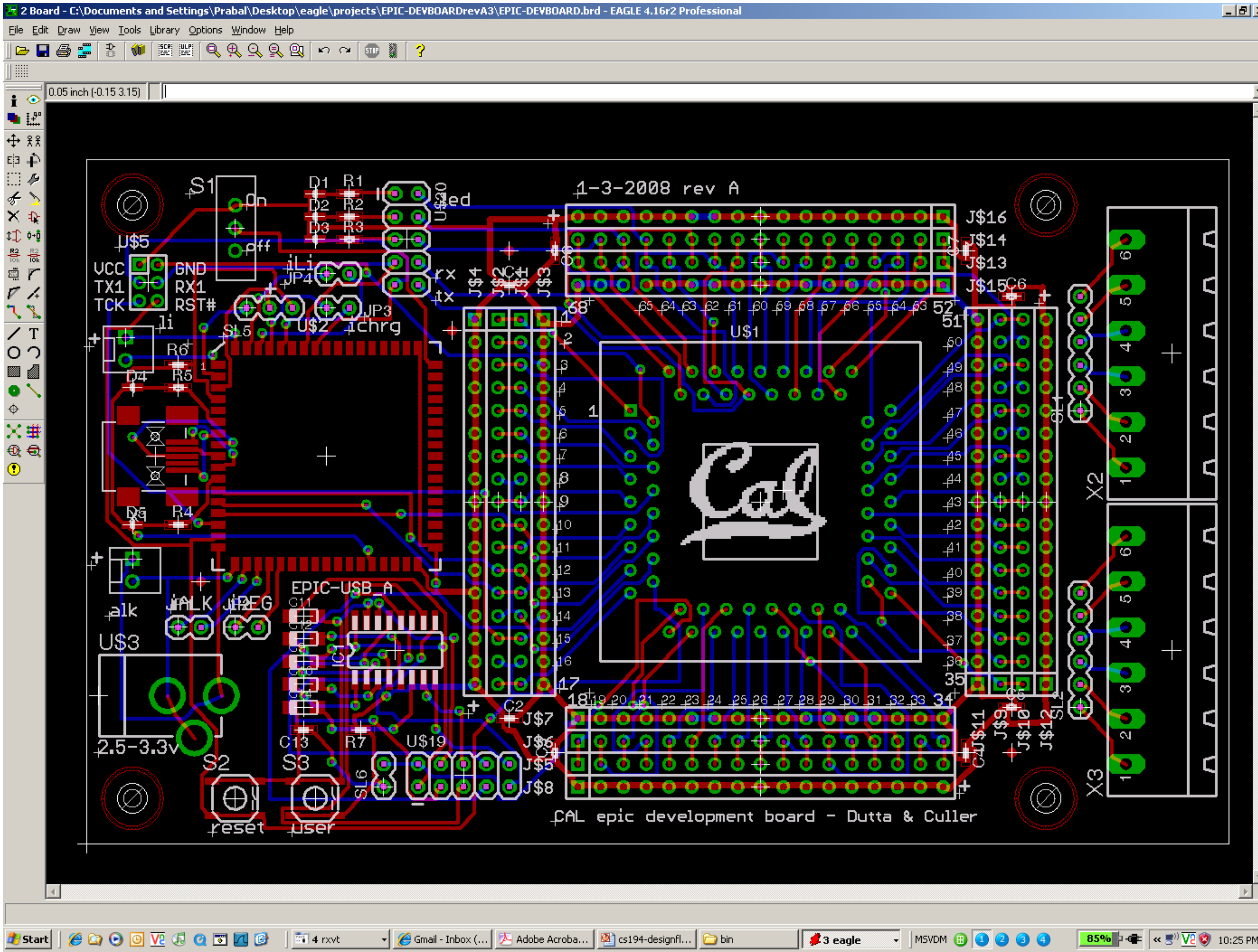
The schematic captures the logical circuit design



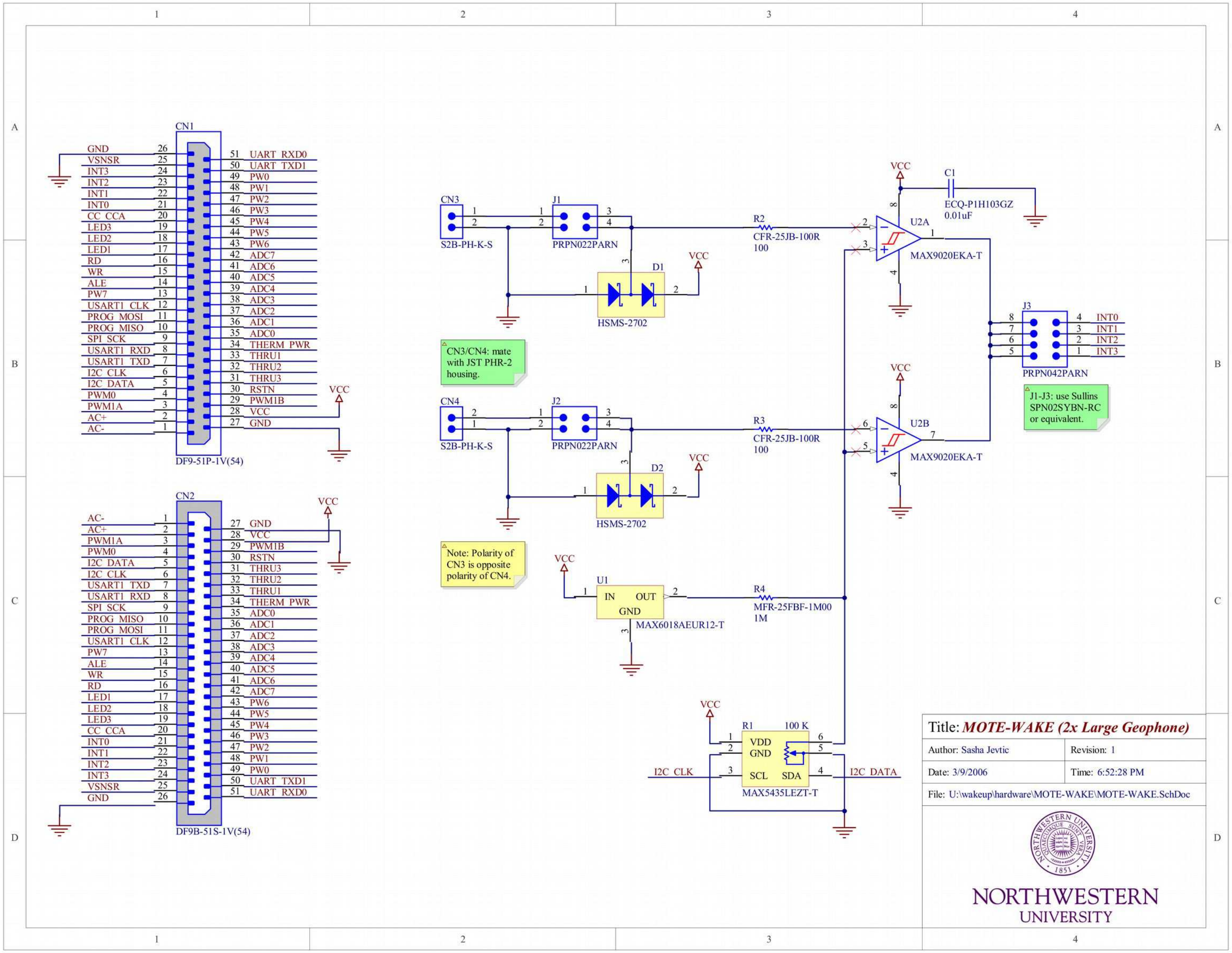
Floorplanning captures part locations



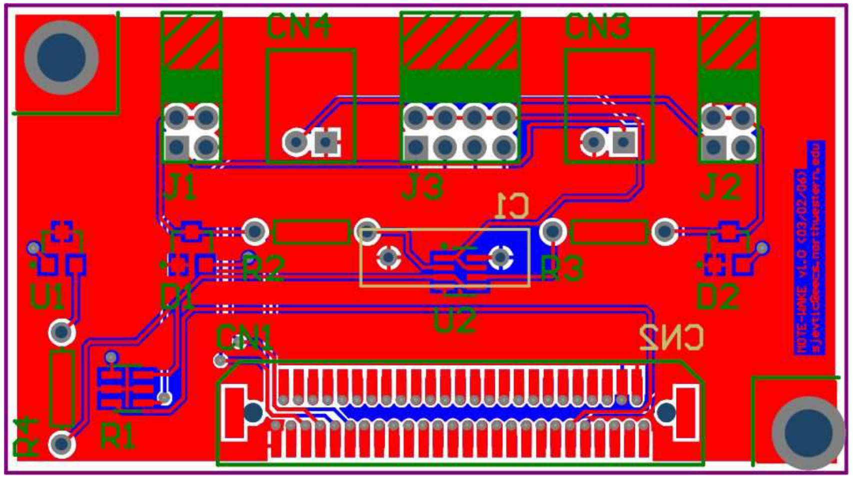
The auto-router places tracks on the board, saving time

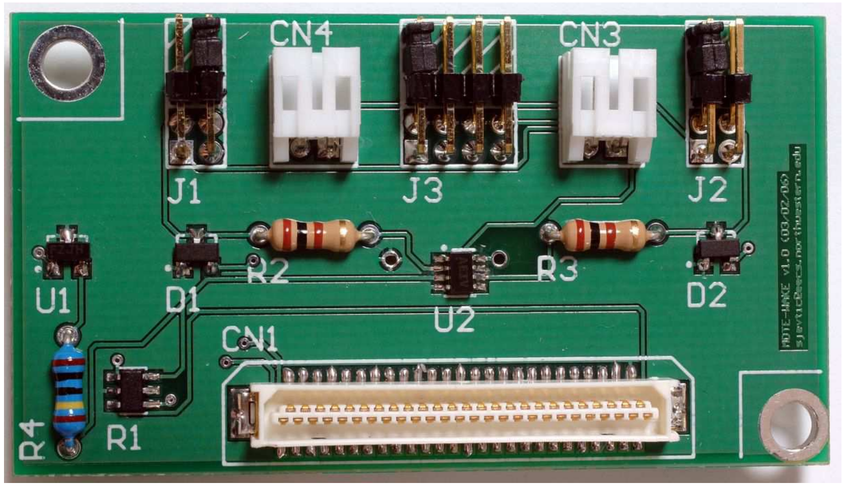


Another design, all the way to production



Another simple design, all the way to production





- Simple design that solved a hard problem.
- Deployed at many sites around U.S.

Not a simple design

- Note component density
- Can mount components on each side.
- Relationship between PCB layout, pinouts, and external components important.
  - LED.
  - Battery.
  - Others, e.g., big inductors.
- Form (and board shape) follows function.
- RF subsystem physical design tricky.

The layered construction of a PCB: A six layer board

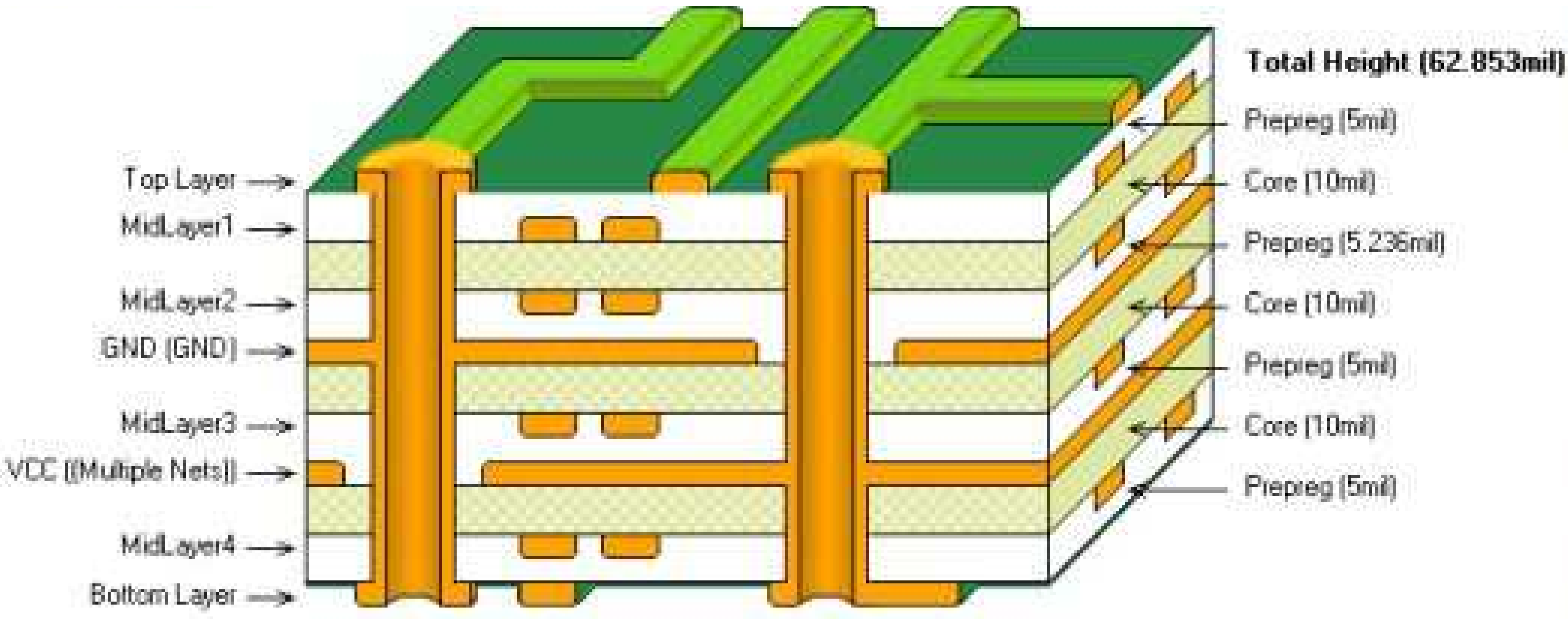


Figure from altium.com

Doesn't need to be expensive / complex

- Can CAD/CAM mill away solid Cu layer.
- Can use lithography.
  - Photoresist.
  - Mask (can print with laser printer).
  - Projector.
  - Etchant (many are dangerous to breathe and touch).
  - Safe way to dispose of Cu-containing solution.