Review

- Misc project-related applications and examples
- Prototyping
Outline

- Memory
- PCB design
Memory: why?

- You'll be dealing with this a lot in your career.
- Technologies will change.
Nonvolatile memory types

- Flash: ~ms. 10E6 V/cm. Around 1E6 write cycles.
- EEPROM: ~ms.
- SRAM/DRAM: ~ns.
- Spin-Transfer Torque Magnetic Memory: Recently entered commercial production. Compared to Flash, high endurance (10E12 write cycles), low or no leakage, fewer masks, write energy of 120 fJ/bit.

- Others still at / just left research stage.
  - Spin-Orbit Torque.
  - Voltage Controlled Magnetic Anisotropy (6 fJ/bit write energy).
  - Bi-Polar Non-filamentory OxRAM; Conducting Bridge Memory.
  - Macromolecular (Polymer) Memory.
  - Ferroelectric FET.
  - Ferroelectric Tunnel Junction.
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 100,000 writes/erasures.
Outline

- 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 size.
  – Thin, high resistance.
  – Holes/vias and pads.
  – Rework is hard.

http://www.linkwitzlab.com/Pluto/supplies-subw.htm,
Basic terminology

• Interconnects: traces.
• Traces that touch on the same layer are electrically connected.
• Multilayer common: stack.
• Through-hole: for pins.
• Surface mount.
Parts of a PCB

- Copper (pads & traces)
- Soldermask / board G10 (green or brown)
- Silkscreen (white)
- Drill holes
- Via
- Bottom side
Vias

- Connect traces on layers.
- Use a via: plated-through hole
- Generally smaller than a through hole for a pin.
Clearances

• Space between the traces, other traces, and plated holes.
• Meet manufacturer requirement.
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.
  • Increase power supply capacitance
  • Minimize resistance.
  • Some shielding effect.
• Power plane for similar reasons.
• More layers $\rightarrow$ higher cost.
How to design PCB

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

- Show devices and connections.
  - May consider pinouts.
- Layout follows functionality and connectivity, not physical structure.
Example schematic
Purposes of schematic

• Communication and formalization.
• Bug hunting.
• Synthesis.
Step 2: Place parts

• Place patterns on board.
  – No component overlaps on same side.
  – Leave room for traces.
• An art.
• Some tools help.
• Sometimes they fail.
Patterns

- Trace/component sizes.
- Hole positions.
- Each device has a pattern.
- Many are standard.
- Some aren’t: create own.
Step 3: Route interconnect

• Route: connection among devices.
  – Multiple traces.

• Design rules.
  – Minimum trace width.
  – Minimum traces–hole spacing.
  – Minimum hole–hole.

• Rules vary by manufacturer.

• Units vary by 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

• Trace width minimum of 6-10 thou common.
  – Finer at a price.
• Guidelines to control R and temperature:
  – 50 thou min for power/ground.
  – 25 thou min.
  – 10 cm trace >= 10 thou wide at 1 amp.
  – 5 amps >= 110 thou.
Trace width continued

• Wide traces hard to route.
• Necking down sometimes acceptable.
• Consider series resistances.
Rat’s nest

• Device placements and connections.
• Automatically generated.
Routing for real

• Autorouter
  – May seem disorganized.
  – Quick.
  – Often worse than manual.
• Some nets fail.
• Do them manually.
Routing quality

An example of GOOD power routing (Left) and BAD power routing (Right)
Step 4: Generate files

- Different layers/stages.
  - Copper on a given layer.
  - Silkscreen.
  - Solder mask.
- Gerber format common.
  - Human-readable (barely) ASCII.
  - Commands like draw and fill.
- Drill files in Excellon.
  - Human-readable (barely) ASCII.
  - Hole locations and diameters.
- Archive and send all files to PCB house.
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
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.
Done.