EECS 373

PCB design, and Power Integrity
Today

• Introduction to PCB design
  – Overview and definitions
  – Design steps

• Power issues
  – Circuits review
  – 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,
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.
PCBs – basic terminology

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 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
Today

- Introduction to PCB design
  - Overview and definitions
  - Design steps

- Power issues
  - Circuits review
  - Power integrity
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
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 mill to avoid confusion, but most tools/vendors use mill.
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.

- 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)
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.
Floorplanning captures the desired part locations.
The auto-router places tracks on the board, saving time
The layered construction of a PCB: A six layer board

Figure from altium.com
Today

• Introduction to PCB design
  – Overview and definitions
  – Design steps

• Power issues
  – Circuits review
  – Power integrity
Power 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.
Today

• Introduction to PCB design
  – Overview and definitions
  – Design steps

• Power issues
  – Circuits review
  – Power integrity
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)

\[ W = \frac{J}{s} = \frac{N \cdot m}{s} = \frac{kg \cdot m^2}{s^3} \]

• Power (as opposed to energy), in-and-of itself is important in embedded system design.
  – For example there may be a cap 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.
What do we mean by Power?

- **Max Power**: Artificial code generating max CPU activity
- **Worst-case App Trace**: Practical applications worst-case
- **Thermal Power**: Running average of worst-case app power over a time period corresponding to thermal time constant
- **Average Power**: Long-term average of typical apps (minutes)
- **Transient Power**: Variability in power consumption for supply net
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

**Power** is the height of the curve.

*Lower power design could simply be slower*

**Energy** is the area under the curve.

*Total energy needed to complete operation*
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. It 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
A look at impedance
(with capacitors, inductors and resistors vs. frequency)

Notice the log scales!
Today

• Introduction to PCB design
  – Overview and definitions
  – Design steps

• Power issues
  – Circuits review
  – Power integrity
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.

- ESR is Effective Series Resistance
- ESL is Effective Series Inductance
- Ceff is the effective capacitance.
  - How does quantity effect these values?
- Obviously impedence 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
**Given the previous table..**

### Decoupling Impedance vs Frequency

- **Z(pup)**
- **Z(tant)**
- **Z(1uF)**
- **Z(0.1uF)**
- **Z(0.01uF)**
- **Z(pcb)**
- **ZT**
- **Z(LICA)**

---

**Power Integrity**

<table>
<thead>
<tr>
<th>Device</th>
<th>Quantity</th>
<th>Cap</th>
<th>ESR</th>
<th>ESL</th>
<th>Cell</th>
<th>ESR</th>
<th>ESL</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC/DC Converter</td>
<td>1</td>
<td>1.00E-04</td>
<td>7.00E-07</td>
<td>1.00E-04</td>
<td>7.00E-07</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tantalum Capacitors</td>
<td>2</td>
<td>1.00E-06</td>
<td>3.00E-02</td>
<td>7.00E-10</td>
<td>2.00E-03</td>
<td>1.50E-02</td>
<td>3.50E-10</td>
</tr>
<tr>
<td>0603 Ceramic Caps.</td>
<td>8</td>
<td>1.00E-07</td>
<td>6.00E-02</td>
<td>6.00E-10</td>
<td>8.00E-03</td>
<td>7.50E-03</td>
<td>7.50E-11</td>
</tr>
<tr>
<td>0603 Ceramic Caps.</td>
<td>8</td>
<td>1.00E-08</td>
<td>9.00E-02</td>
<td>5.00E-10</td>
<td>8.00E-09</td>
<td>1.13E-02</td>
<td>6.25E-11</td>
</tr>
<tr>
<td>0603 Ceramic Caps.</td>
<td>10</td>
<td>1.00E-09</td>
<td>1.50E-01</td>
<td>5.00E-10</td>
<td>1.00E-08</td>
<td>1.50E-02</td>
<td>5.00E-11</td>
</tr>
<tr>
<td>PC Board</td>
<td>1</td>
<td>8.50E-08</td>
<td>8.50E-08</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Removing the PCB...

Decoupling Impedance vs Frequency

Impedance vs Frequency

Power Integrity
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
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.
Much material taken from others:

  - Very nice tutorial/overview
  - Seems to have strong viewpoint
  - Some definitions taken verbatim.
- Wikipedia
- Dr. Dutta
- And others where noted.
Exam scores

- 9: 8710
- 8: 998776543311
- 7: 997666443300
- 6: 9977443
- 5: 776444
- 4: 8
- 3: 8