Attendees: Erik, Jen, Rachel, Robby

Erik’s son got 2nd in the Pinewood Derby

3 (and a half?) PCB Board designs

* Looking at schematic- make it as readable as possible w/ a fram (printable on the page)
* Communication tool- the more info you put down the better for design review, keeps design decisions doc’d
* For easier communication, make a power block diagram in ppt
* Choose digital pins (UCA) to take as well?
* How is everything mechanically going to stack?
* Each capacitor should have its own via to a ground plane

Get a cam file (from Macrofab) to know which Eagle layers contribute to physical layers of pcb

Use Macrofab DRC to know what via size- hole size and annular size

* Generally used thicker traces power and ground 10 mils (don’t really need that much bc it’s low power, but might as well), kick them down to the plane, let those do the work
* As long as cap is located physically close to pin, they don’t need to be connected in literal sense, it can be connected to plane in local area
* For big power connector, add bigger cap 22, 0602 to all input powers
  + If board requests lots of power at very bigger, local cap has ability to withstand local needs
* Keep the number of different drill sizes to a minimum-- that’s what costs (having to change drill size)

Change all vias at once- just turn on via layer, group vias, use change tool

Thermally isolate the temp/hum sensor- negative space around the pcb

Put a via in under the sensor and name it the same as the pad

Be paranoid about putting debug hooks on everything, put probe points on i2c line

But it goes through a header anyway

Put a diode backwards on power header for all lines

Hang some LEDs on the gpios