Design iteration 2

System requirements

Your team's objective is to design, build, and program a fleet of wireless temperature monitoring probes which can be deployed across the Tufts Medford/Somerville campus.

The nodes must:

- Measure temperature at least every hour, and ensure that measurements have a timestamp accurate to within 5 minutes.
- Provide temperature measurements that accurately reflect the ambient air temperature within 2 degrees Farenheit.
- Report temperature data to an MQTT server on the Tufts campus network. We will decide on the exact protocol through class discussion.
- Be designed to be powered for at least 6 months without a connection to mains power.
- Be designed to withstand the weather for a year (heat, cold, rain, ice, wind, etc.)
- Have a complete BOM costing less than $15 each, assuming a quantity of 1000 nodes.

Since the goal of this course is to move beyond development kits and breadboards, the system should integrate a discrete microprocessor or microprocessor module onto your PCB instead of a standalone development board.

Conceptual design (due 3/15)

Your conceptual design should be a document which describes your design at a high level. You should explain what sensor(s) you will use to measure temperature, and how your system will be powered. Explain why you made the choice you did, particularly in comparison to some of the other alternatives.

Your conceptual design should include a section discussing the power requirements of your system:

- Estimate the power usage of your system, including time spent taking measurements, transmitting data, and asleep.
- What capacity battery do you need? How long will the system be able to run on a single battery charge?
- If you're using a solar panel or other energy harvesting system, how much power do you need to collect (i.e., what size solar panel do you need)?
- What power conversion circuitry will you need? You don't need to draw out the whole circuit (that's for next time!) but you should be clear about what the circuitry needs to do.

Preliminary design review (due Tuesday 3/26)

Your preliminary design should be a collection of documents which describe your design in detail and are (almost) ready to be manufactured.

- Completed schematic, showing all parts of the system (sensor, processor, power/charging, debug, etc).
- Complete bill of materials. This should be a spreadsheet with the following:
  - Description of part (e.g., 10k resistor)

  - Digi-Key part number
  - Quantity needed **per sensor node**
  - Cost/each assuming you're building 1000 nodes. (E.g., if you need five 0.1uF capacitors for each board, quote the price for 1 capacitor at quantity 5000.)
- PCB layout

For Tuesday's class, you should prepare a brief presentation to explain your design the to class. You should have 3 slides:

- System overview (a block diagram showing the main components of your system and how they fit together)
- Schematic diagram (this can be two or more slides if it's too small to see on one slide)
- PCB layout

You should be ready to answer (and ask) questions like:

- Why did you choose the parts you did?
- What are the main design-for-X features of the design? (design for test, design for debug, design for manufacturing, etc)
- What parts of the design are you uncertain about? What parts do you feel are the most risky or likely to have problems?

Put your KiCAD design, slides, and any other resources in your team's Google Drive folder.

Tapeout (due Friday 3/29)

You will build 3 boards for this revision; one for each team member. You should order your boards from either OSH Park or JLCPCB. JLC is cheaper and has more options, but is potentially more error prone (you have to upload Gerbers instead of KiCAD files) and overseas shipping/customs is less predictable.

Put your completed KiCAD files and BOM spreadsheet in your Google drive folder. Send me a Digi-Key "share cart" link with the parts you need in a quantity necessary to build 10 boards (we'll use the extra parts for iteration 3).

I will order stencils (no more squeezing solder paste out of the tube by hand!) for your boards using your KiCAD design.