# Golriz Sedaghat

## Last week:

* Created the schematic for sensor board
* Worked on the PCB layout in free version of EAGLE CAD which only supports 2-layer PCB; and therefore, I ran into issues which stopped me from proceeding and completing the layout
* After meeting with Andrew, I revised the circuit based on the given feedback
* Purchased the EAGLE Hobbyist version of EAGLE right after meeting with Andrew
* Finally, Thursday, I received the license number of the EAGLE

## Next week:

* Learning about 4-layer PCB design in EAGLE
* Creating 4-layer PCB layout for the main board
* Creating PCB layout of the sensor boards, probably in two layers
* Possibly, meet or email the final version of the design and PCB layout to Andrew for the final approval
* Send the layouts to fab

## Issues:

* Learning 4-layer PCB design with EAGLE
* Completing our documentation