**Schematic and Board Layout Design Review Notes**

TEAM BEING REVIEWED: Group 14

REVIEWING TEAM: Group 7

DATE: 11/6/18

**Instructions:**

1. Reviewed team explains the concept of operation of the project.
2. Reviewer reads through schematic, layout, and BOM attempting to answer basic questions such as: what is the power supply? What is this IC and what does it do? Does the layout make sense?
3. Reviewer reads through schematic, layout, and BOM looking for best practices, such as: Are there enough bypass caps? Are the traces the right thickness?
   1. Reviewers are encouraged to use the Schematic and Layout Checklists that are posted on the D2L site.
4. One person on the reviewed team interacts and answers reviewer’s questions. A different person on the reviewed team takes notes, organized in the format below.

* **SCHEMATIC**
  + CRITICAL
    - Add 10uF electrolytic bypass capacitor for power supply
    - Enable internal pull-up resistors on digital pins
  + MAJOR
    - Label crystal oscillator’s frequency (16MHz)
    - Label regulators with voltage and max current output levels
    - Check USB port: Will it read/write if the power output pin is floating?
      * If power pin must be connected to something, limit its current draw to 100mA, which is the computer port standard max output current.
  + MINOR
    - Label pins of servo headers
    - Rotate labels so they’re level
    - Reduce clutter
    - Possibly move button pad headers to the right side, since that’s the input to the system
    - A little hard to read around the switch
    - No other visible potential issues, nice layout
* **LAYOUT**
  + CRITICAL
    - Add ground plane
  + MAJOR
    - Change micro usb port to thru-hole??
    - Place team number on silkscreen (tplace) layer!
  + MINOR
    - Watch out for sharp corners: try to limit angles to 45° at most. Don’t want the etching process to sever the trace on the corner.
    - Don’t use minimum trace width
    - Use large trace widths for high power traces
    - Minimize length of PWM (or signal) traces, maximize width.
    - Orient components vertically on the pcb to minimize temperature gradient side effects
    - A little hard to read around the switch
    - No other visible potential issues, nice layout
* **Bill of Materials (BOM)**
  + Left justify ‘22’