**Schematic and Board Layout Design Review Notes**

TEAM BEING REVIEWED: Team 7

REVIEWING TEAM: Team 14

DATE: 11/8/18

**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
  + MAJOR
    - Text overlap in title block, making it illegible if printed
  + MINOR
    - Text overlap of names and values of various components
* **LAYOUT**
  + CRITICAL
  + MAJOR
    - Minimum trace width on all lines? Assuming 6mil is the minimum trace width.
    - Sharp angles on traces, such as 90° or more, is not recommended as the etching could sever the trace at the corner. Use 45° angles if possible.
    - Air wires on bottom side of the board… (alt+2 to view bottom side, alt+3 to view both top and bottom, alt+1 to view just the top side)
      * Air wire intending on shorting out L2
      * Air wire to connect pins 3 and 6 of MCP1
  + MINOR
    - TP2 somewhat close to L2, maybe you want more distance?
* **Bill of Materials (BOM)**
  + Misspelled “Bourns” MFG on MPN: CR0805-FX-5623ELF
  + Aesthetics: ALL CAPS differences “Digikey” vs. “DIGIKEY”