**Schematic and Board Layout Design Review Notes**

TEAM BEING REVIEWED: Team 6

REVIEWING TEAM: Team 10

DATE: November 7th, 2018

**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
    - Coupling capacitors
    - Power supply
  + MAJOR
    - Not quite all All nets labelled
    - Put group # (and names) on schematic
  + MINOR
    - Perhaps regulate to 3.3v in order to prevent brownouts from USB voltage drops
* **LAYOUT**
  + CRITICAL
    - Trace widths, specifically speaker
    - Characterize current/power of speaker
* **Bill of Materials (BOM)**
  + Check specs and tolerances, specifically of speaker
  + Label general tolerances in lower right corner of schematic