ENCE461 PCB Review

Bring this checklist, annotated with ticks or question marks, a printout of your new schematics, and the schematics annotated at the schematic rewiew.

Schematic

1. Battery is fused before any other electrical connection
2. Battery has reverse polarity protection (hat board only)
3. Radio has power supply filtering

Silk screen layer

1. Has group number
2. Has student names
3. Has clearly labelled test points
4. Has clearly labelled battery polarity for hat board
5. Has clearly labelled connectors

Mechanical

1. Drill sizes meet requirements (min 0.3, steps of 0.1)
2. USB connector hangs over edge correctly
3. Radio antenna hangs over edge or there are cut-outs in the PCB planes

Plane layers

1. Has a ground plane
2. The ground plane is connected to the GND net
3. Has a power plane (this can be partitioned)
4. The power planes are connected to the appropriate power nets
5. Has cut outs for the radio antenna if the antenna does not hang over edge

Signal layer

1. All vias go from top to bottom layers
2. There are no long power/ground traces to planes (use more vias)
3. Large vias and wide traces are used for power/ground and high-current traces
4. IC pins are not connected between the pads
5. SPI, I2C, PWM, and UART signals do not run over gaps in a plane
6. SPI, I2C, PWM, and UART signals do not change layers more than twice
7. Ground test points are positioned for ease of use

Design rule check

1. There are no clearance violations when performing a design rule check
2. There are no trace width violations when performing a design rule check
3. There are no unrouted nets

Note, Altium gets confused with the pad to plane check when adding decoupling caps, so ignore warnings with distances of 2.5 m.

Further checks

See `http://ecewiki.elec.canterbury.ac.nz/mediawiki/index.php/PCB_guidelines` and `http://ecewiki.elec.canterbury.ac.nz/mediawiki/index.php/PCB_advanced_guidelines`.