A03G Schematics II

A **group** submission to be submitted on Github Classroom & Gradescope.

**Github Classroom assignment:** <https://classroom.github.com/a/Ba1hqn3X>

Remember to read the [ESE5160 S24 Assignment README](https://docs.google.com/document/d/1pPXQByy8eTxTJ--3vO8KpTjMk5yBHF8wQXoLJ55w5a8/edit) before starting!

If you need to use a late day, you must submit using [this form](https://docs.google.com/forms/d/e/1FAIpQLSd2hfFc7tIAqP-B1GouC5sP6Zbl59p7JXJa_yGTR60CJHRU3A/viewform).

# Table of Contents

[**Table of Contents 1**](#_fchudxiwvnp6)

[**Learning Outcomes 1**](#_wenhizp776g3)

[**1. PCBA Bill of Materials (Altium) 1**](#_rddhy93yduep)

[Steps 1](#_2jalxtibmv4)

[**2. Non PCBA Bill of Materials (Excel) 2**](#_lw2pwqehrkzy)

[**3. Updated Schematic Capture 3**](#_t7k0fjm1dmxh)

[**Rubric 4**](#_psy3hy66jdwd)

# Learning Outcomes

* Learn how to review and implement feedback from the teaching team and your peers to improve your schematic design.
* Understand the detail required in finalizing your Bill of Materials for a successful PCBA build and delivery on time.
* Determine the difference between components mounted on the PCBA and thos e that will be soldered by you.

# 1. PCBA Bill of Materials (Altium)

Use Altium to create a full BOM that includes all of your components on the PCBA. You must add all components

## Steps

1. Familiarize yourself with the ActiveBom, a Bill of Materials file you can add to your project.
   1. Read: [Managing Your Bill of Materials (BOM) with ActiveBOM in Altium Designer](https://www.altium.com/documentation/altium-designer/activebom-bom-management)
   2. This will explain what the document is and how to add a solution to a component (how to add a Manufacturer Part Number to a device). Read it in its entirety and return to it if you have questions.
2. Open the ActiveBom file in your starter project. Once loaded, the ActiveBom documents will add a row for each of your components on the schematics. **It is your task to add a solution to each component.** The restrictions are:
   1. Components must be selected from **DigiKey** or **Mouser**.
   2. Components must be **in stock** and are the **correct footprint**!
      1. It can be easier to look for the part number on DigiKey or Mouser, then use that distributor part number to search for the correct BOM component in Altium.
         1. Ex: When looking for BQ24075, Mouser’s distributor number would be **595-BQ24075TRGTT**. I would add a solution in the ActiveBOM specifically looking for **595-BQ24075TRGTT**.
   3. **Resistors:** Use only Stackpole Electronics Family RMCF ([Link](https://www.digikey.com/short/9rv5fjn7))
      1. If you have special resistor requirements (tighter tolerance, size larger than 0805, larger power rating, etc.), you may expand your search on DigiKey or Mouser to find suitable MPNs.
   4. **Capacitors:** Use only Yageo ([Link](https://www.digikey.com/short/f7cr1phv))
      1. If you have special capacitor requirements (tighter tolerance, size larger than 0805, large capacitance value, etc.), you may expand your search on DigiKey or Mouser to find suitable MPNs.
   5. If you are not mounting components, set them as **DNP (Do Not Populate)**. Use variants as seen in class to do this: [Altium Documentation](https://www.altium.com/documentation/altium-designer/design-variants)
3. **Double Check:** With your team mate, go through each item of the BOM and make sure the solution you added is the correct value, footprint, voltage, tolerance, etc. Sometimes, Altium can select a component with a different footprint if you let it select for itself.   
   **Things to look for:**
   1. Is the footprint correct?
   2. Is it in stock at DigiKey or Mouser?
   3. Is the Minimum Order Quantity 1?
      1. Don’t put something where the MOQ is 10,000!

**Submission:** In the Github README.md, add the share link to your Altium 365 project.

# 2. Non PCBA Bill of Materials (Excel)

Starting from the template spreadsheet in the Github starter repository, list the information for the components that **will not be populated on your PCBA**. Our PCBA manufacturer will not procure these components; the Detkin lab staff will purchase these for you separately from our PCBA build.

Items included on the Non-PCBA BOM include:

1. All through hole components
   1. Components considered surface mount: USB connector, SD card holder, and programming header. (they will be populated on your PCBA by the manufacturer!)
2. Motors, fans, or **development boards** that will not be soldered to the PCBA because it will be mounted away from the PCBA and connected with a cable.
3. Power supplies, if needed for high current applications (like an AC/DC wall wart)

If you have any concerns, please ask on ED!

**Submission:** Commit your updated Excel sheet to your Github repository.

# 3. Updated Schematic Capture

Update your schematics with feedback from the teaching staff and peer review (see below). Continue working in the **same Altium project file** - remember, this is version controlled, so it’s easy to move forwards and backwards in your commit history, if needed. The feedback will be in the form of [comments](https://my.altium.com/altium-designer/getting-started/commenting-your-design).

Add the Penn Template to your schematics and update the title block (See L05 slides/video for more info). Finally, ensure that you can import your schematics into your PCB without any errors. This will identify and part linking or footprint errors in your design.

**Submission:** Submit a link to your project in the UPenn ESE projects folder. Make sure your updated project is committed to A365 before the deadline - we can view the commit time.

# 

# Rubric

While the rubric attempts to capture all assignments details, points assigned may vary based on submission quality and teaching team review. Please ensure you read the assignment carefully so as not to miss details and lose points. Poor readability / formatting can lose you points on any assignment.

**For all questions, 0 points will be awarded if the submission is non-existent, very poorly done, or doesn’t compile (for firmware assignments).**

| **Points** | **Question** | **How to achieve full credit** |
| --- | --- | --- |
| 10 | 1. PCBA Bill of Materials (Altium) | PCBA BOM is complete – all components are there! There are no errors and component footprints on BOM match the schematic (example: You choose a resistor that is 0603 for an 0603 resistor on the schematic). |
| 5 | 2. Non-PCBA Bill of Materials (Excel) | Non-PCBA BOM is complete - all components not soldered to the PCBA are listed, including purchase links. |
| 20 | 3. Updated Schematic Capture | Comments from A02G were implemented, Penn template is used on all schematic sheets, title blocks are complete with information (authors, title, etc) for all schematics.  **If we can’t find your project, we can’t give you any points!** |
| 35 |  | Total Achievable Points |