ECAD 1: Basic of Using Altium (GEEN 3010 Version)

This workshop is very similar to the ECAD Level 1 Workshop but relies on a different library and schematic reference to build the Stoplight PCB. This is important because students are then expected to assemble their own Stoplight PCB in the assembly workshop.

Find workshop material here: ECAD 1 - Basics of using Altium Designer (GEEN 3010)

Material Links

Dr. Zarske sends a link for students to download Altium on their personal computers. You can find that process here: Altium Student License Page

The schematic library can be found here: GEEN3010_Stoplight_SP2026. Once downloaded, students will drag and drop this into Altium and click install.

Find workshop material here: ECAD 1 - GEEN 3010 Version Only.pptx.

Key Points (IMPORTANT!!!)

Dr. Zarske is very intentional with this workshop and built a lab document with questions surrounding important learning objectives. When leading this workshop, it is important that you know the questions she wants answered and include these answers within your presentation and walkthrough.

The questions include:

1) Schematic Capture – The schematic is a roadmap for your entire design. It is the first step in the process of making the PCB (after building it on the breadboard). Make sure you also talk about the schematic etiquette and rules that are included on the ECAD slides.

2) Libraries – A library is a structured storage system for electronic components, containing essential models needed for design, including schematic symbols (.SchLib), PCB footprints (.PcbLib), and 3D models.

3) Node – Nodes or junctions are net lines or wires that branch to more than 2 pins. These are important as they allow connectivity between multiple aspects.

4) Errors and Warnings – Errors are the critical failures and high value checks that must be resolved. Warnings are design preferences or expected attributes that are missing (do not always need to be fixed). Warnings usually point to better practices or better etiquette.

5) PCB Layout – We want to make sure we emphasize that when routing you should not include 90 degree angles. That power signal should usually have a larger trace width if room permits. When showing the polygon pours, we want to emphasize that often we attach this to ground, and it is used to save copper and reduce noise. Use DRC (design rule check) to see if you have any errors or warnings.

6) Air Wires – Yellow thin wires on the PCB Layout between components are known as air wires. These are computer-generated lines that show the shortest distance between two elements that need to be connected. These represent the physical copper traces you need to lay down (in a sense), and it is important that you try and untangle these before you route.

7) Autoroute – Altium has a built-in auto routing feature that allows the software to generate traces that satisfy the schematic netlist. These tracs are the computers idea of “best way possible”. We do not use it because our brain allows us to think outside of the board and create better solutions based on circumstances.

8) Fabrication Outputs - CAM/Gerber files are the files needed to manufacture your board. These files contain the information and form factors that your manufacturer needs. Different manufacturers have different required files, and these can be found on the specific manufacturer's website.

Feedback and Recommendations

- This software can be quite intimidating at first, so making sure students feel comfortable navigating everything is crucial. Also offer students to use a mouse if they are not working on a lab station computer (we have them in ECAD kit).
- Because there is so much covered in this workshop and it is a valuable skill, we suggested that all students work on their own computer. The lab stations should work perfectly fine if you have access to these. The checkout laptops do have some finicky issues, ensure that you have looked over how to set these up (included in the slide deck) and are ready to troubleshoot as you go.
- Make sure all students are on the same steps. It is harder and less valuable for you to catch them up later than it is to stop really quickly and help. Make sure everyone is on the same page by asking if anyone needs help frequently. Then walk around and visually confirm this is true when giving them work time.
- Bring schematic diagrams for them to have in front of them as they work. Makes it a lot easier for them to reference, and you to point stuff out.

Revision #4
Created 2026-04-06 15:22:21 MDT by Tanner
Updated 2026-04-06 15:51:50 MDT by Tanner