

Printed Circuit Board Design

Overview

In this activity, you will learn to use Altium Designer to enter a schematic design and generate a printed circuit board (PCB) definition. This will give students experience in designing PCBs and an understanding of how hardware components are created via schematics and how components are integrated within them.

Each student will receive a schematic; every schematic is different. Students will work on their own schematics and printed circuit design. Some values may be hard to read on the schematics; in such cases, don't worry about the values but ensure the correct part type (resistor, capacitor etc.).

Instructions

This activity is broken into four steps: installing the software, creating the schematics and design files, generating a report of the schematic development, and completing a demonstration.

Setup

First, we need to install the supporting software, including Altium Designer and libraries. Note that this will require approximately **5 gigabytes** of storage just to download the files, before installing.

Download

To set up Altium Designer, you will need to download the following files:

[Altium Designer Installer](#) 21.6.4 or newer

[Altium Component Library](#) (2.1GB)

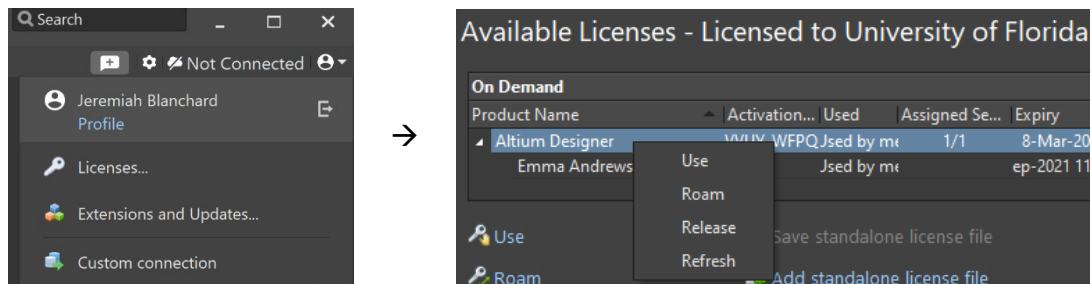
[ECE Component Library](#) (167K)

For libraries, note that the URL is http, **not https**. UF-hosted files require access from the VPN.

Installation

Fill out the form on the [student licenses website](#), making sure to use your UFL email address. Once the form is submitted, you will receive an email (may take several minutes) asking you to verify your email address. Click on the provided link to verify your email address. You will then receive another email (may take several minutes) containing your license information. Follow the activation in the email to create your Altium account. After your account is created you will be provided with a link to download the installer. Download the installer and then run it. During the installation process, you will be prompted to sign into your Altium account. Enter your UFL email address as the username and the password you just created. Select the PCB Design, Extensions, and Importers/Exporters features in the installer.

Once installed, run Altium Designer. Open the License Manager by clicking on the profile icon in the top right, then selecting “Licenses”. If you signed into your account during installation, your license will be auto-populated. Otherwise, sign into your Altium account, and your license will appear. Right click and select "Use" or click on “Use” under the Available Licenses pane to activate the license. If not signed in, sign into your Altium account from this page.



Schematic Assignment

The course instructor will assign a unique schematic to each student; you will recreate this schematic in Altium Designer.

Submission

Your submission will be composed of the following elements:

- Synchronous demonstration of PCB Design (in-person or Zoom, depending on availability)
- Altium project files within a **ZIP** archive on Canvas
- Report in **PDF** format on Canvas

Demonstration

Your demonstration will be completed either in person during office hours or via Zoom, depending on availability. It must include the following, at a minimum:

- Displaying PCB/circuit schematics
- Showing board outline on “keep-out” layer
- Showing polygon pours on both layers
- Demonstration to verify files are correct

Report

We recommend that you complete your demonstration in advance of the report so that you can receive feedback that will provide you the opportunity to correct mistakes. In addition to all other template elements, your report should include a page for each of...

- Sample schematic provided from lab (original image of circuit you’re creating)
- Altium schematic (.SchDoc file)
- PCB layout (.PcbDoc file)
- Drill files (.Cam files)