



03/31/2022

Accessing and Using Altium Designer Software

The University has a site license for Altium Designer, a PCB design, PCB component, and a data management software which is available to those in ECE and ETEC.

Anyone who needs to create a printed circuit board (PCB), Altium is an excellent piece of software to use. Knowledge of Altium is an awesome addition to any resume.

The following will get you started:

1. Accessing the Software.

- a. **RECOMMENDED** –Enroll in the “*Free College and University Level Curriculum and Altium Designer Student License*”
 - i. This provides you with a full copy of Altium Designer that you download to your own computer. Link is ...
<https://education.altium.com/>
 - ii. This course not only provides you access to the software and also provides you a well laid out training course on how to create a printed circuit board and how to use the software.
 - iii. If you complete the training course, save your certificate for potential future credit at the university as well as in industry (can add the certificate to your resume)
- b. Access through vLabs (**CURRENTLY NOT WORKING**):
 - i. Altium is available through vLabs. When you log into vLabs, select the Electrical Engineering “pool,” which is a new login menu option.
 - ii. To start Altium, either click on the desk top icon or select Altium through the menu.
- c. Downloaded Software (Alternative – **HIGHLY** recommend using method 1.a. above):



03/31/2022

- i. You **MUST** associate the software to your university email address.
- ii. <https://www.altium.com/solutions/academic-programs/student-licenses>

NOTE: You may need to turn off pop-up blockers to view the forms.

- d. **IF and only IF** none of the above meet your needs, contact Dohn Bowden mailto:dohn_bowden@uml.edu and explain why none of the above work for your situation. If you meet the requirements, a license will be provided.

i. Step 1: Activate AltiumLive

1. Your AltiumLive account enables you to access the Altium online ecosystem. If you have not already done so, please activate your AltiumLive account now before continuing.

<https://live.altium.com/>

ii. Step 2: Download Altium Designer

1. Installers for Altium products are available on the downloads page. First, you need to login using the AltiumLive account then you can access and download the installers.
2. Download the software from <https://www.altium.com/products/downloads> . If you do not have an Altium account, you will need to create one. You will need to login to download the software
3. Installers for Altium products are available on the downloads page. First, you need to login using the AltiumLive account Then you can access and download the installers.

iii. Step 3: Install and Activate Software

1. Double click the installer application and login with your previously created AltiumLive account. Then follow the on-



03/31/2022

screen instructions to start the installation.

iv. **Step 4: License**

1. Obtain the license from Dohn Bowden
dohn_bowden@uml.edu

2. Tutorials and other Training Aids:

- a. Use the following document to get you started:

<https://www.dropbox.com/s/une25wyd1q8xdvg/Getting-started-with-pcb-design.pdf?dl=1>

The above document is also available as a web page that is updated regularly:

<https://www.altium.com/documentation/altium-designer/from-idea-to-manufacture-driving-a-pcb-design-through-altium-designer>

- b. There is also an Atrium Designer “YouTube Channel,” which contains tutorials and other videos. The YouTube channel is located at:

https://www.youtube.com/channel/UCWLoHp3WJG_ats8waVCu7Mw

- c. If you are new to Altium Designer, then the following link will be useful:

<https://www.altium.com/documentation/altium-designer/exploring>

3. Additional Resources:

- a. How to Create a PCB Layout from a Schematic in Altium Designer:

<https://resources.altium.com/pcb-design-blog/how-to-create-a-pcb-layout-from-a-schematic-in-altium-designer>



03/31/2022

If you are aware of any additional resources that will aid others in learning Altium, please email dohn_bowden@uml.edu and the links will be added to the resources in this document.