

Altium Student License Setup

1. First, you will need to be running a Windows operating system (Windows 7 or newer) in order to run Altium Designer. If you do not have a default Windows OS, you will need to set up a partition on your computer. These are the more detailed requirements to run this program

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

- a. To partition your MacOS with Windows, follow the instructions found here <https://support.apple.com/en-us/HT201468>
2. Go to <https://www.altium.com/solutions/academic-programs/student-licenses> and fill out your information, using your cornell.edu email to request a license
 3. You will get an email to verify your email within a few minutes, and after you confirm that, you will get another message with the following set-up steps

Step 1: Activate AltiumLive

AltiumLive is where you access the Altium online ecosystem. If you have not already done so you will need to activate your AltiumLive account before you can continue.

- [Activate AltiumLive](#)

Step 2: Download Altium Designer

Installers for Altium products are available from the downloads page. You will need to login with your AltiumLive account you activated in the previous step to download the installers.


- [Altium Downloads](#)



Step 3: Install and Activate Software

Double click the installer application, login with your previously created AltiumLive account, and follow the on screen instructions to install. Further details are available here:

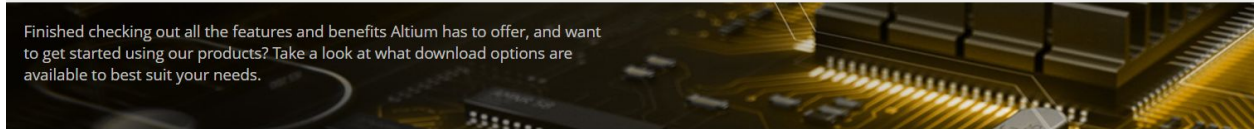
- [Installing Altium Designer](#)

4. Follow the instructions above to set up your Altium Live account, download the latest version of Altium (following the downloads link provided, the very first download on the page will be the most recent version).

Altium HOME PRODUCTS SOLUTIONS COMMUNITY RESOURCES COMPANY [FREE TRIALS](#) 

 **Altium Designer® 20: Available Now** [Free Trial](#) 

Finished checking out all the features and benefits Altium has to offer, and want to get started using our products? Take a look at what download options are available to best suit your needs.



ALTIVM DESIGNER

Leverage the most powerful, modern and easy-to-use PCB design tool on the market. Altium Designer brings together unified design and Native 3D™ PCB capabilities to help you create next-generation electronics.

Give Altium Designer a test run, download the free trial

Latest version: 20.0.13 

Release Date: 2020-02-25
Installation and management
Download (exe, 23.7 MB)
Need Offline Installer?

5. After you have downloaded the executable, follow the instructions listed here to install and run Altium for the first time
<https://www.altium.com/documentation/altium-designer/installing-altium-designer>