



# Altium Setup Guide

January 2025



# Table of Contents

Introduction.....	3
Creating an Altium Account .....	4
System Requirements .....	6
Installing Altium on your Windows Computer .....	6
Next Steps .....	7

It is **required** that you create an **Altium account** and receive an **Altium license prior** to the workshop.

It is also **required** that you use your **personal Windows laptop** for this workshop, as ECF has unfortunately removed Altium from all their machines.

Without a license, it may not be possible for you to follow along with the workshop on your computer.

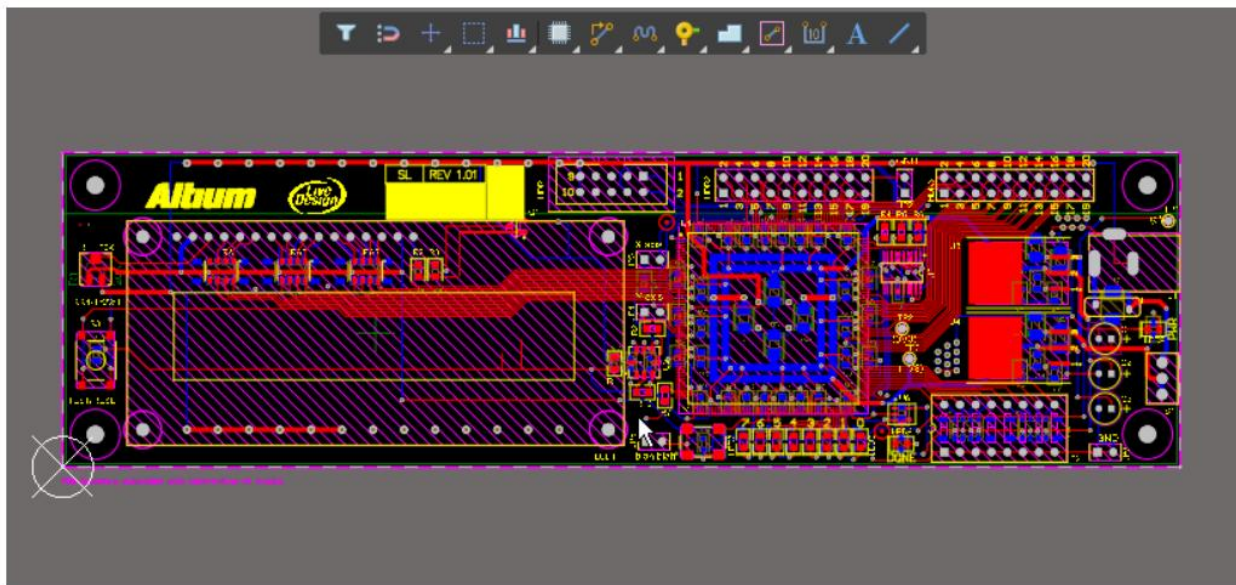
Written by Geoffrey (Tech Director) and Athavan (Tech Associate),

IEEE Tech Team

# Introduction

Altium Designer is a Printed Circuit Board (PCB) and electronic design automation (EDA) software package to create custom PCBs. Altium has two main functionalities:

- Schematic Capture – where you design the electrical circuit of your design
- 3D PCB layout design – where you layout the physical components

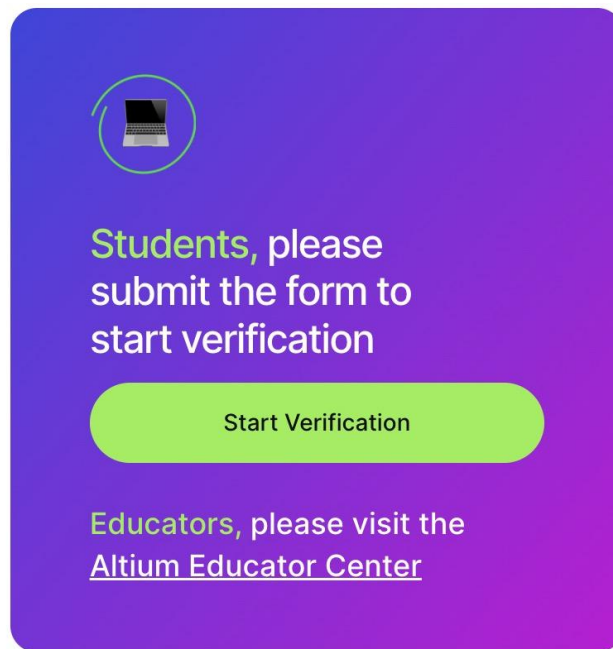


Altium Designer is free for students, who will receive a one-year renewable license when signing up. This guide will walk you through how to directly install Altium on your computer.

# Creating an Altium Account

This section will walk you through how to create an Altium account to receive your license.

1. Go to [this website](#), and scroll down until you see a green button to the right side that asks you to submit a form to start verification. Fill it out using your **UofT email address**. It should look like this:



2. The website with navigate through a series of forms and verifications. **Keep using your UofT email address.**

Once you have filled out all the forms, you should receive a few emails to your UofT address in the next 10 minutes.

If you do not receive any emails from Altium, contact us on Instagram at [@ieee\\_uoft](#).

3. One email will ask to verify your email – be sure to do so.

4. Another email will contain your license information for Altium Designer<sup>1</sup> along with instructions to activate an **AltiumLive** account – be sure to click on the link and follow the steps. The email should contain something like this:

#### **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**

You now have an Altium license! The next step is to see if you can download Altium on your computer.

Note that renewing your license is the same process as above, if you have had an Altium license previously. If you do not receive an email with your license after 24 hours of completing the forms, please contact us on Instagram at [@ieee\\_uoft](#), and we will help you troubleshoot.

---

<sup>1</sup> You will also get an email containing license information for Power Analyzer by Keysight. Keep, but ignore that email for now.

# System Requirements

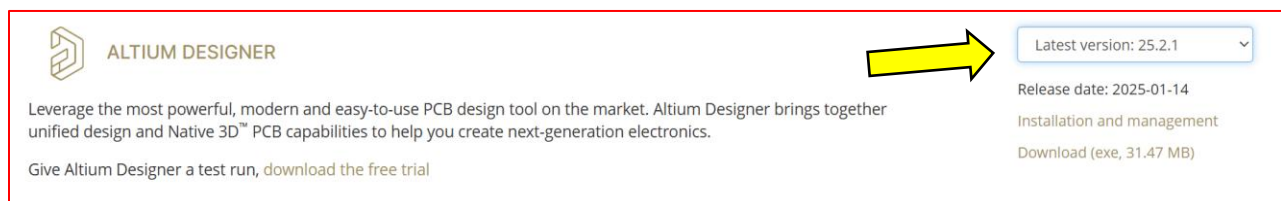
Altium is **only available on Windows machines**. You will need to bring your own Windows machine for this workshop, and it should meet the following minimum requirements:

- Intel® Core™ i5 processor or equivalent
- 4GB RAM
- 10GB hard disk space (Install + User Files)
- Graphics card preferred, but not necessary
- Monitor with at least 1680x1050 (widescreen) or 1600x1200 (4:3) screen resolution

The *recommended* requirements can be found on the [Altium website](#). Note that dual monitors and the 3D mouse are not *necessary* for the workshop, but **please bring a mouse to use with your laptop**.

## Installing Altium on your Windows Computer

1. Go to [this website](#) to view the installation options for Altium. Select version “25.2.1”, which is the latest version at the time of writing.

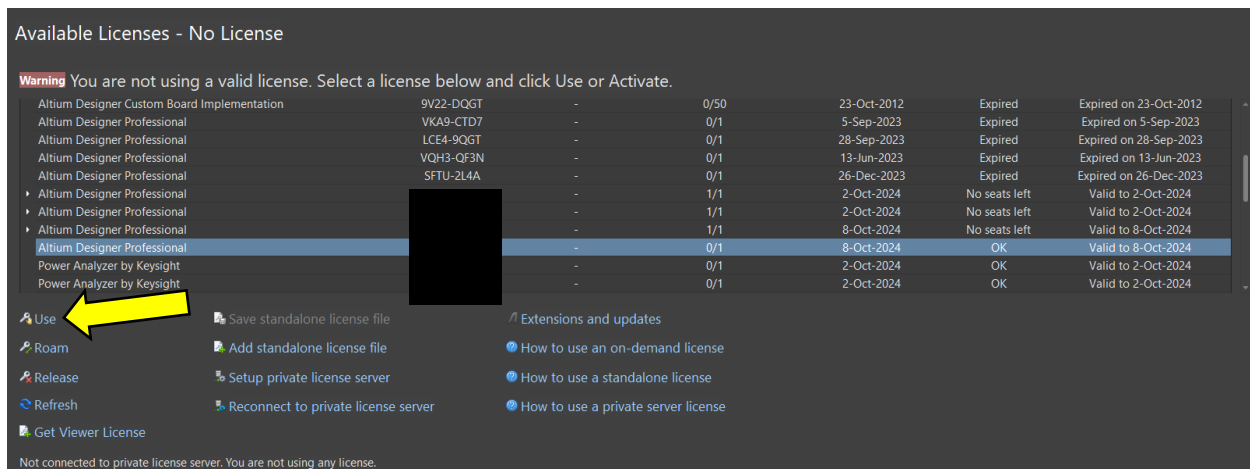


2. Click on “Online Installer” – an executable file will be downloaded.
3. Open the installer and follow the instructions to download Altium. You can leave the default installation settings as is.
4. Once installation is complete, run Altium if it doesn’t do so automatically.

5. Another pop-up will appear asking to update the extensions. Click on “No”.
6. Altium should now display a page called “License Management”. Find the license that matches with the license you received in your email when you first signed up for the student Altium account. Click on the license, then click “Use”.

If you do not see your license but see other licenses displayed<sup>2</sup>, find one that is for “Altium Designer Professional” and under the status column, is labelled “OK”. Click on the license, then click “Use”.

If you do not see your license or any other licenses, please contact us on Instagram at [@ieee\\_uoft](#), and we will help you troubleshoot.



*Valid license codes have been covered up for security.*

You are all set to begin using Altium!

## Next Steps

At this point, you are now able to open and begin using Altium! Feel free to check out the free resources provided to you by Altium as part of your student account, like the [tutorials here](#).

<sup>2</sup> As a new Altium user, you should only see 2 licenses upon startup – one for Altium Designer and one for Power Analyzer by Keysight. You may see more licenses if you are/were enrolled in ECE295.

Otherwise, stay tuned for our upcoming workshop and future events, like MakeUofT! As always, feel free to reach out to us on Instagram at [@ieee\\_uoft](#) with any questions or concerns.

Looking forward to seeing you at our Altium Workshop!



*This page has been intentionally left blank.*



UNIVERSITY OF  
**TORONTO**  
IEEE STUDENT BRANCH