

Updated 5/28/2018 by Alan Sandoval

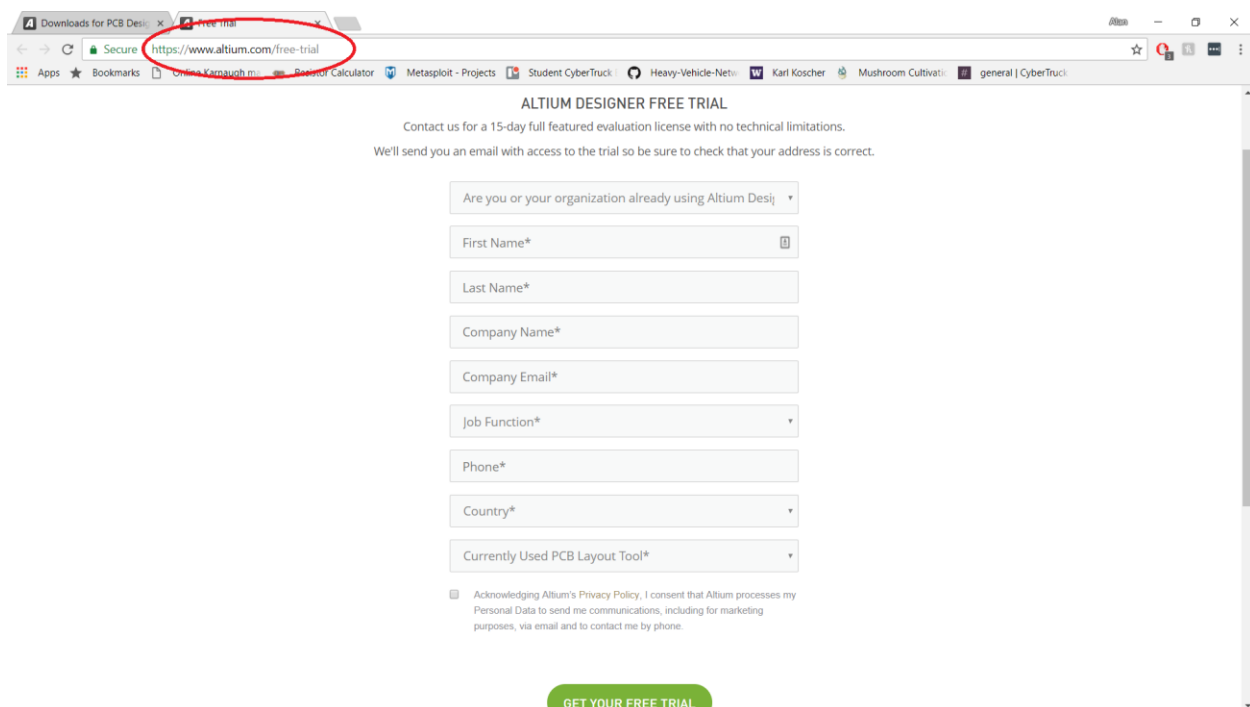
1 mil = 1/1000<sup>th</sup> of an inch

Press Q to to switch units (mils, mm, inches)

Contents of this guide include how to install and activate Altium, how to create and manage a project, how to create and use a schematic, and how to install new libraries from online. I plan to add how to use a PCB file, how to route a PCB, and how to create your own Schematic Library.

### STEP 1: Register for a “free trial” of Altium

Visit <https://www.altium.com/free-trial> and fill out the required fields.



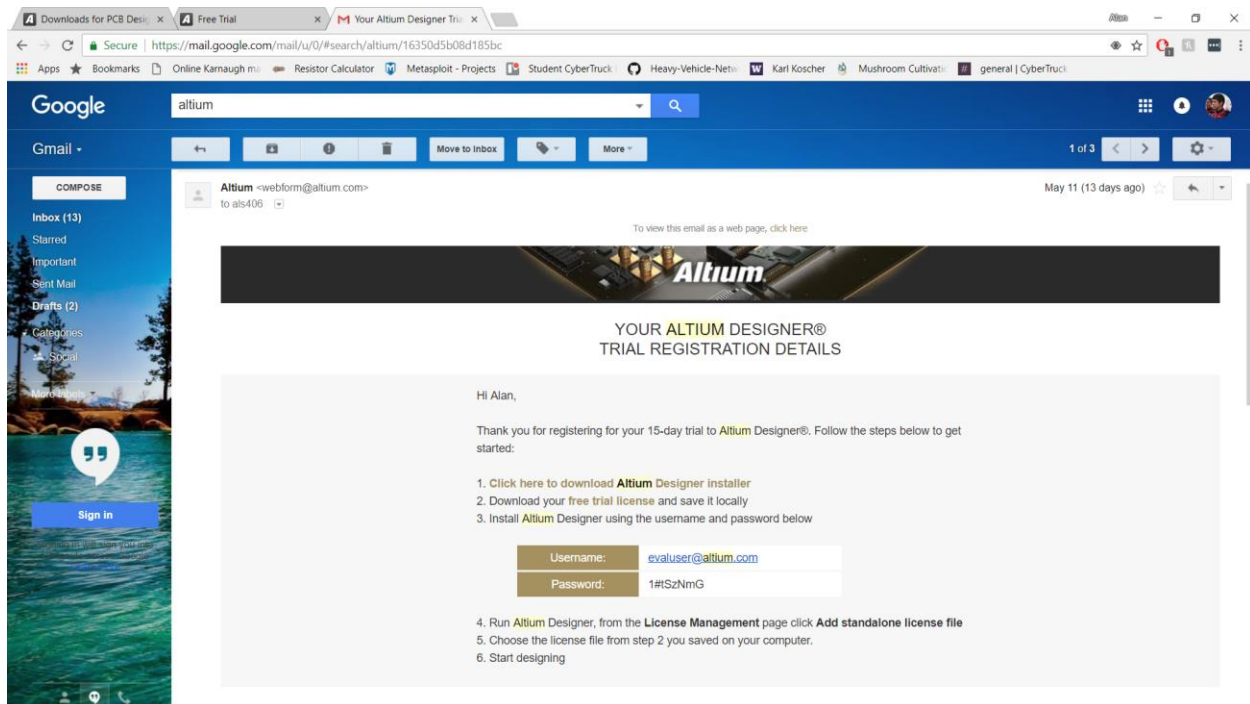
The screenshot shows a web browser window with the URL <https://www.altium.com/free-trial> in the address bar. The page title is "ALTUM DESIGNER FREE TRIAL". Below the title, it says "Contact us for a 15-day full featured evaluation license with no technical limitations. We'll send you an email with access to the trial so be sure to check that your address is correct." The form contains the following fields:

- Are you or your organization already using Altium Desig (dropdown menu)
- First Name\*
- Last Name\*
- Company Name\*
- Company Email\*
- Job Function\* (dropdown menu)
- Phone\*
- Country\* (dropdown menu)
- Currently Used PCB Layout Tool\* (dropdown menu)

Below the form, there is a checkbox labeled "Acknowledging Altium's Privacy Policy, I consent that Altium processes my Personal Data to send me communications, including for marketing purposes, via email and to contact me by phone." At the bottom of the page, there is a green button labeled "GET YOUR FREE TRIAL".

### STEP 2: Access download link from your email

The installer will prompt you for the login information you received in the email.

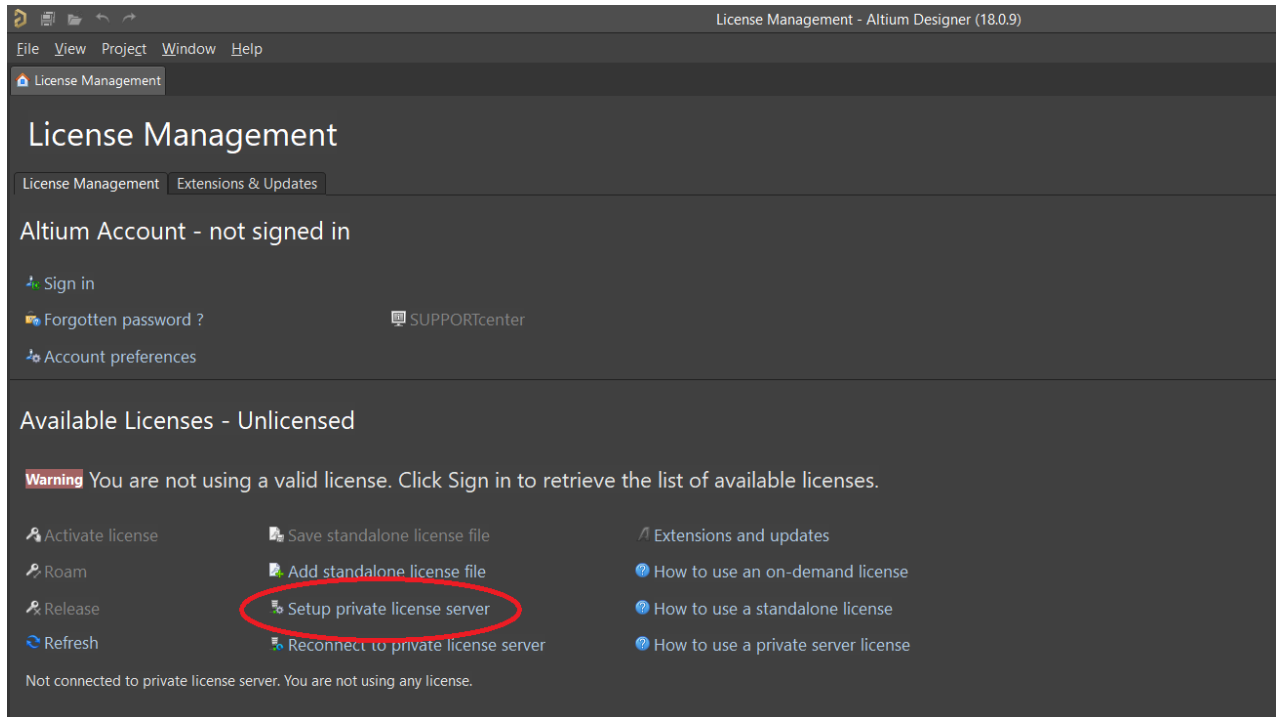


### STEP 3: Download

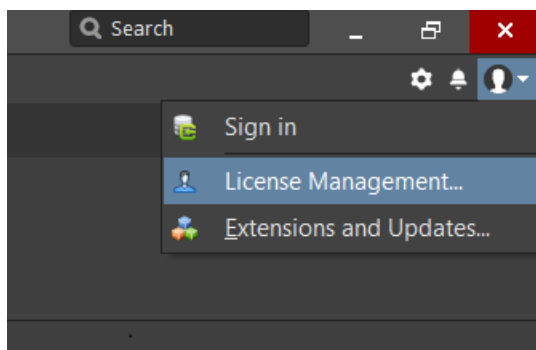


#### STEP 4: Log in to license server

When you first open Altium, this is the screen that you should see. Click on “Setup private license server”



If you are not automatically directed to the License Management screen, access to it is available in the top right corner of the screen.



These are the credentials to fill in after you click on “Setup Private License Server”



**dr.daily** 7:35 PM

uploaded this image: [Pasted image at 2016-09-01, 7:33 PM](#) ▾

**Private License Server Setup**

**Primary server**

Servers list: ▼

Server name:  ☒ Use name

Server address:  ☐ Use address

Server port:

**Secondary server**

Servers list: ▼

Server name:  ☒ Use name

Server address:  ☐ Use address

Server port:

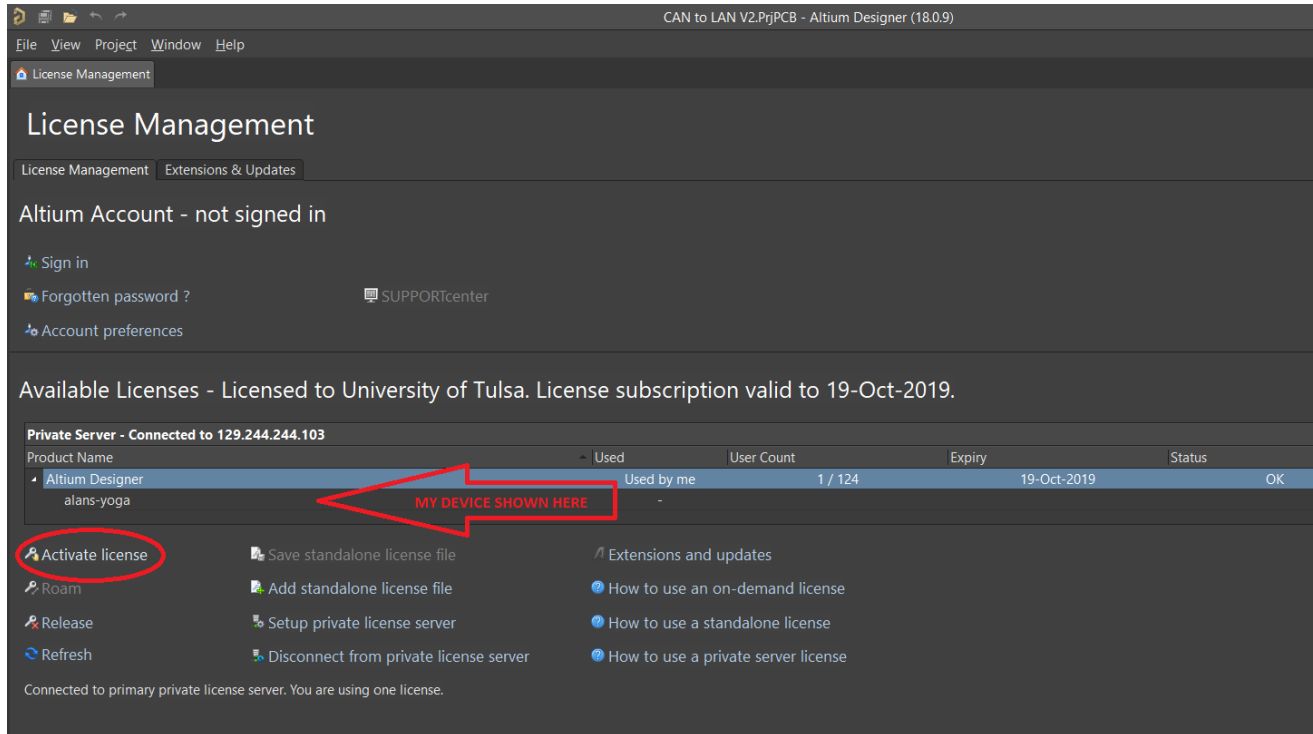


**dr.daily** 7:35 PM

[@haydenallen](#) Here is the license info

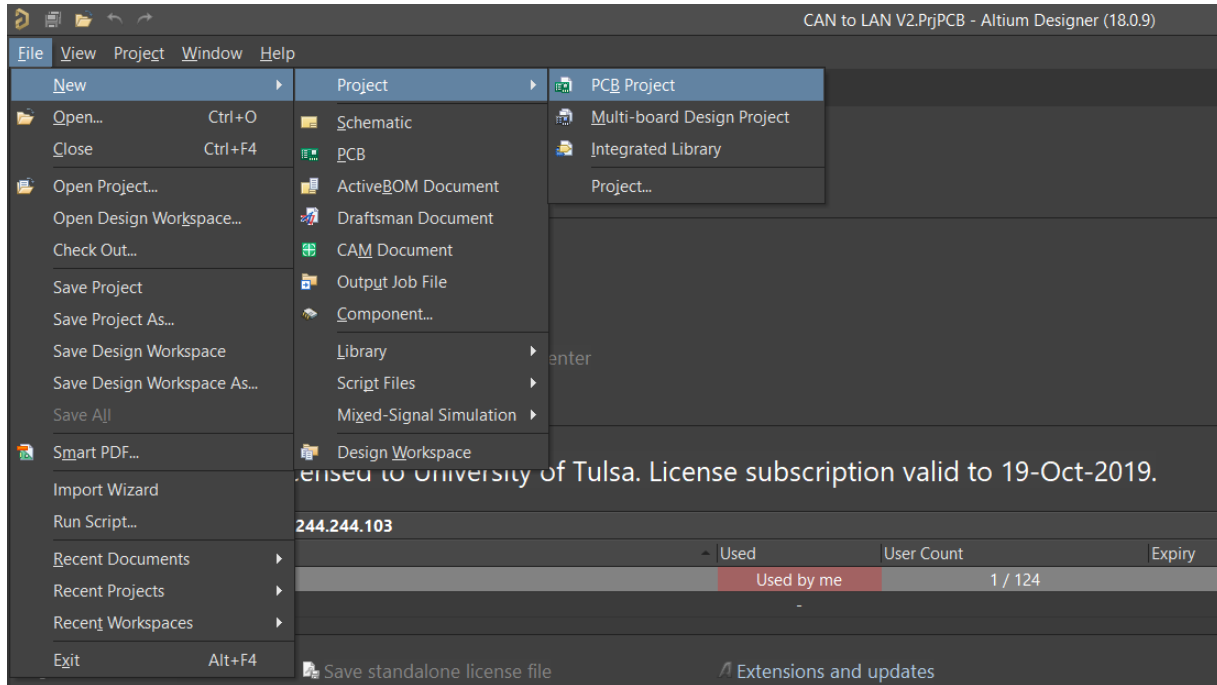
## STEP 5: Activate License

If your device doesn't appear on the list of activated licenses, click on "Activate License"



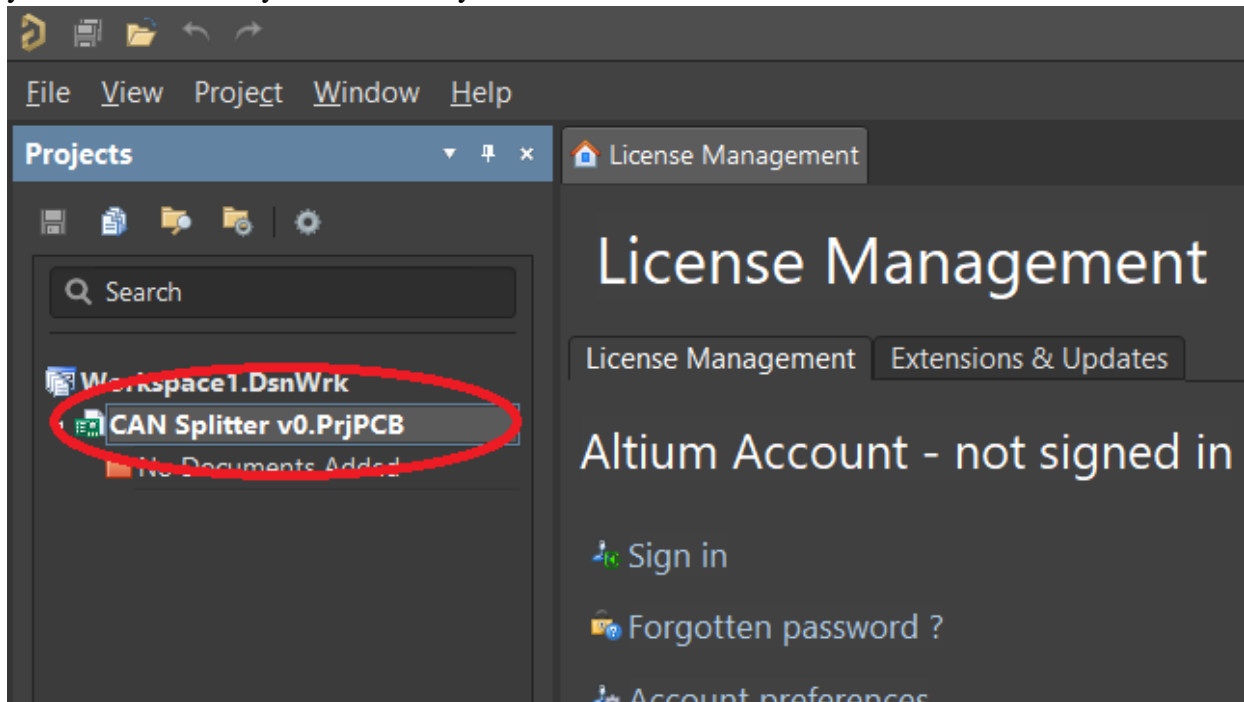
## STEP 6: CREATE A NEW PROJECT

File > New > Project > PCB Project



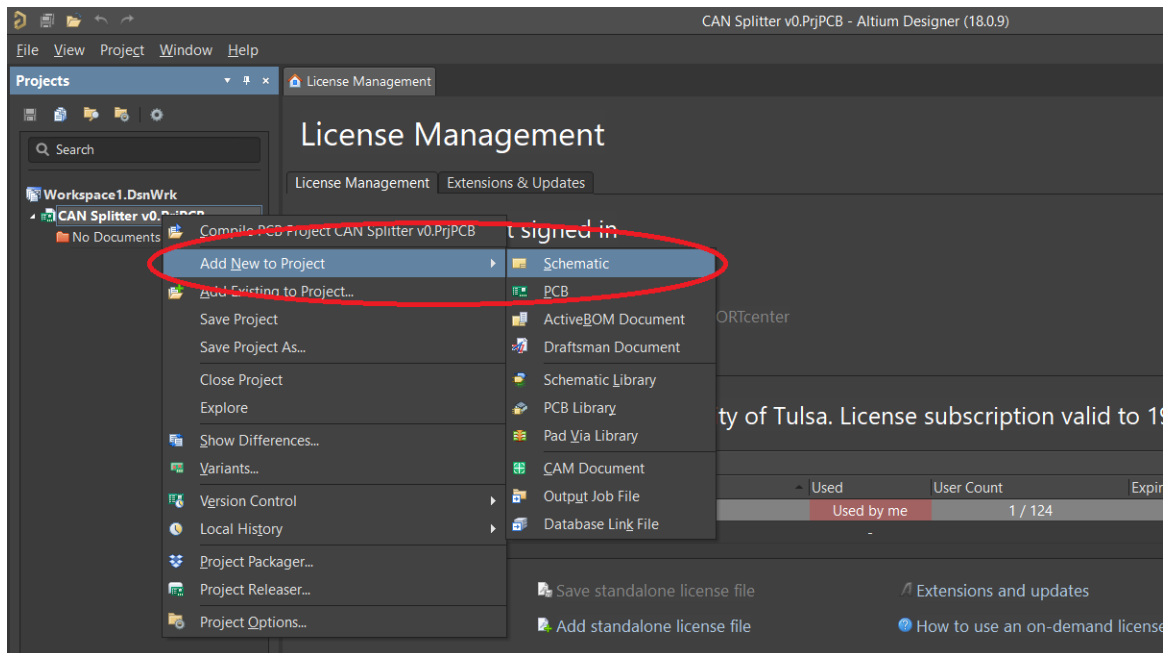
After you create a new project, it is a good idea to save your project. I save my projects on Dropbox to be able to access them from anywhere.

I also like to keep track of what edition of my project I'm working on. Saving old versions of your schematics may come in handy.

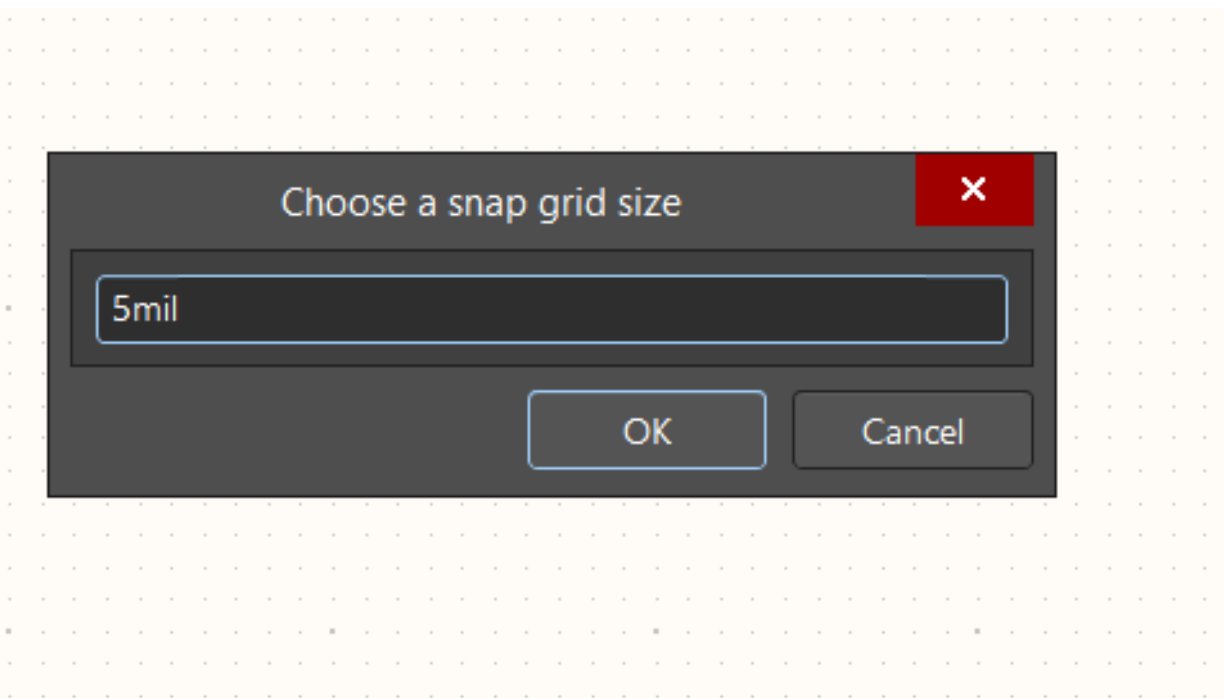
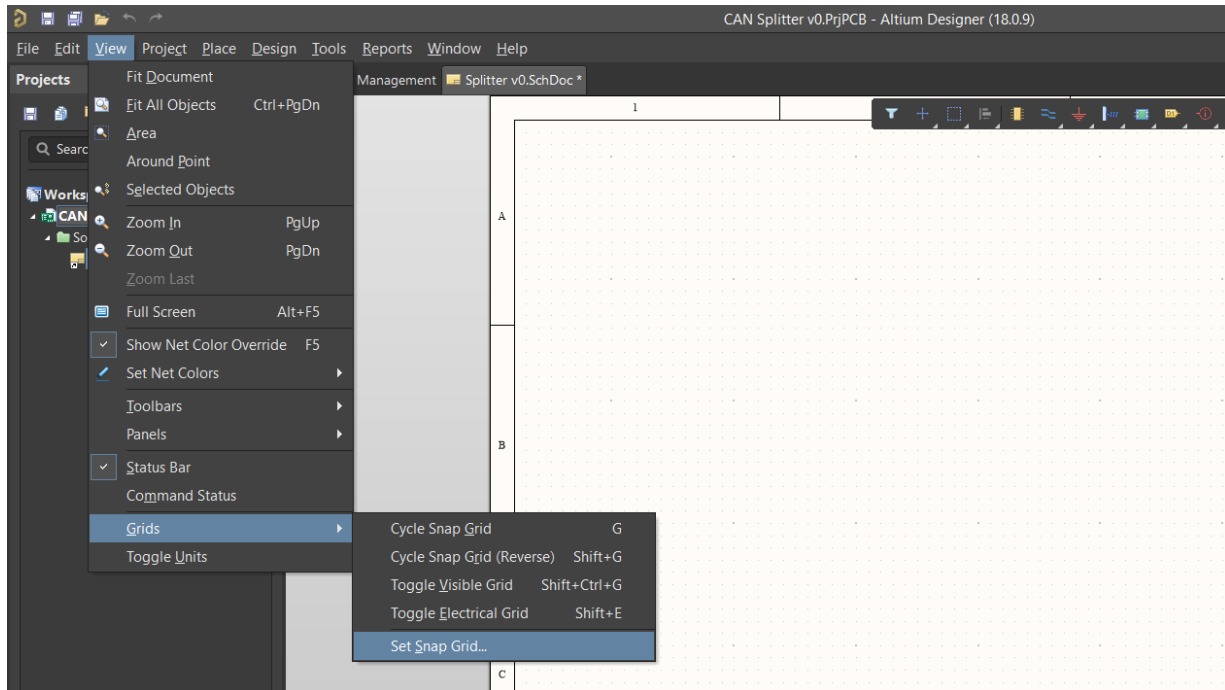


## STEP 7: WORKING WITH A SCHEMATIC FILE

The schematic is the place where you will drop all the components (chips, resistors, capacitors, LEDs, etc.) of your PCB and decide which pins to connect to each other. Create a schematic file by right clicking on your project and selecting Add New to Project > Schematic.

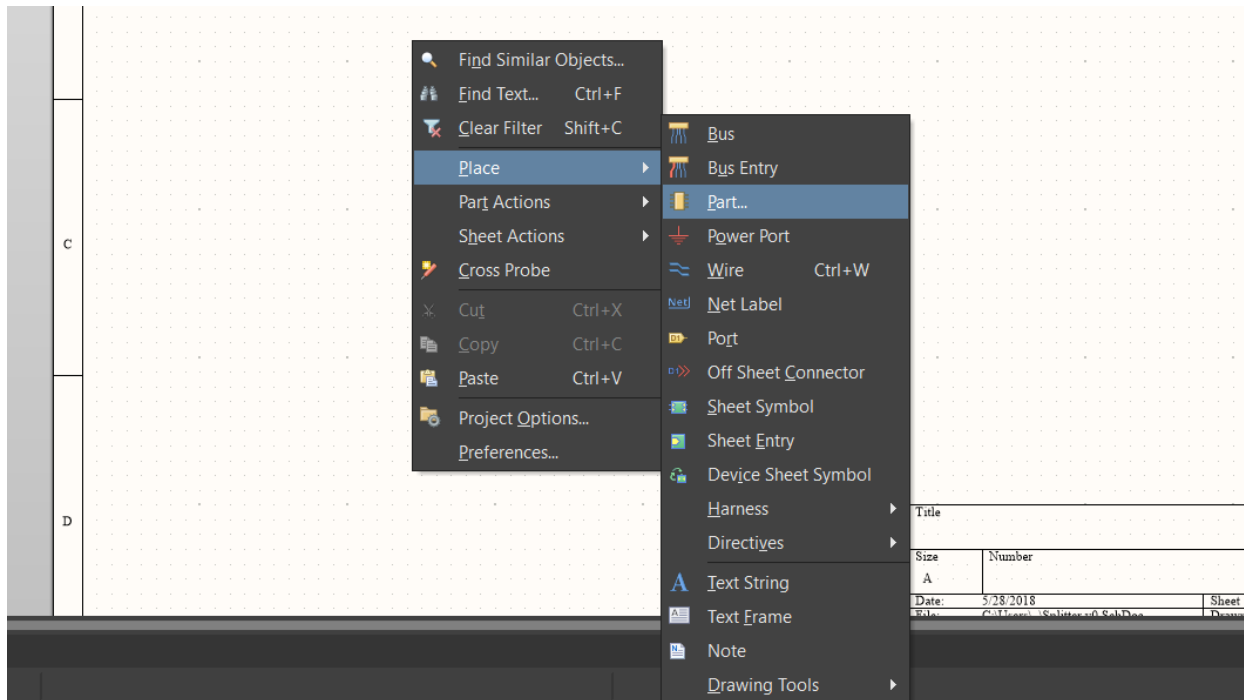


Before you begin adding components, set the “snap grid” to 5mil so that it is compatible with the libraries that we like to use.



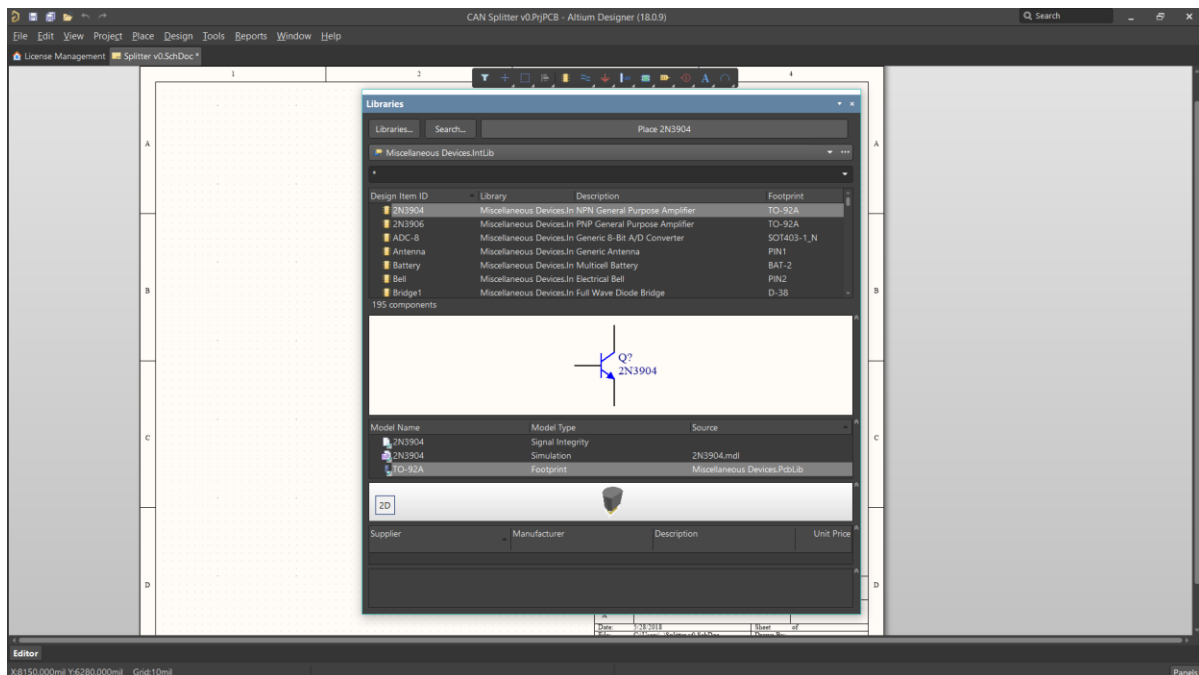


Let's add components to the schematic! You can do this several ways depending on your computer's user interface. You can right click on the schematic > Place > Part...

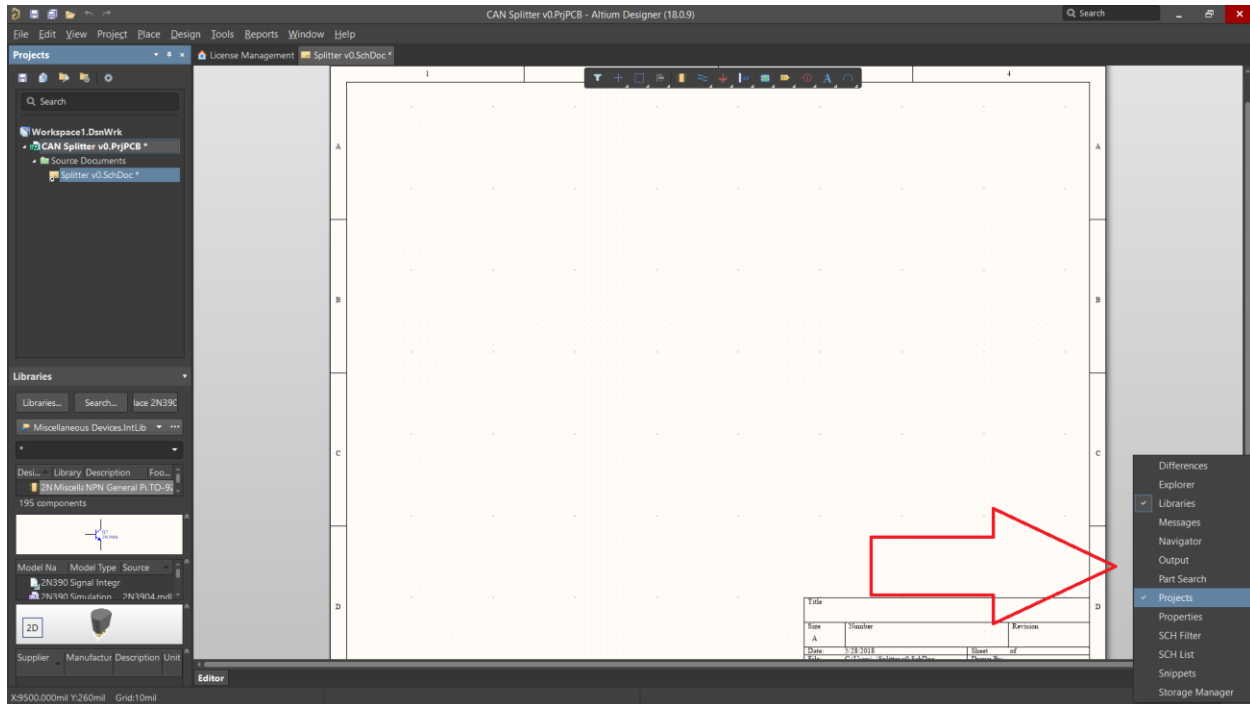


OR you can just click on the toolbar's Part symbol at the top of your screen.

This should open a window named Libraries. These libraries each hold a collection of components.



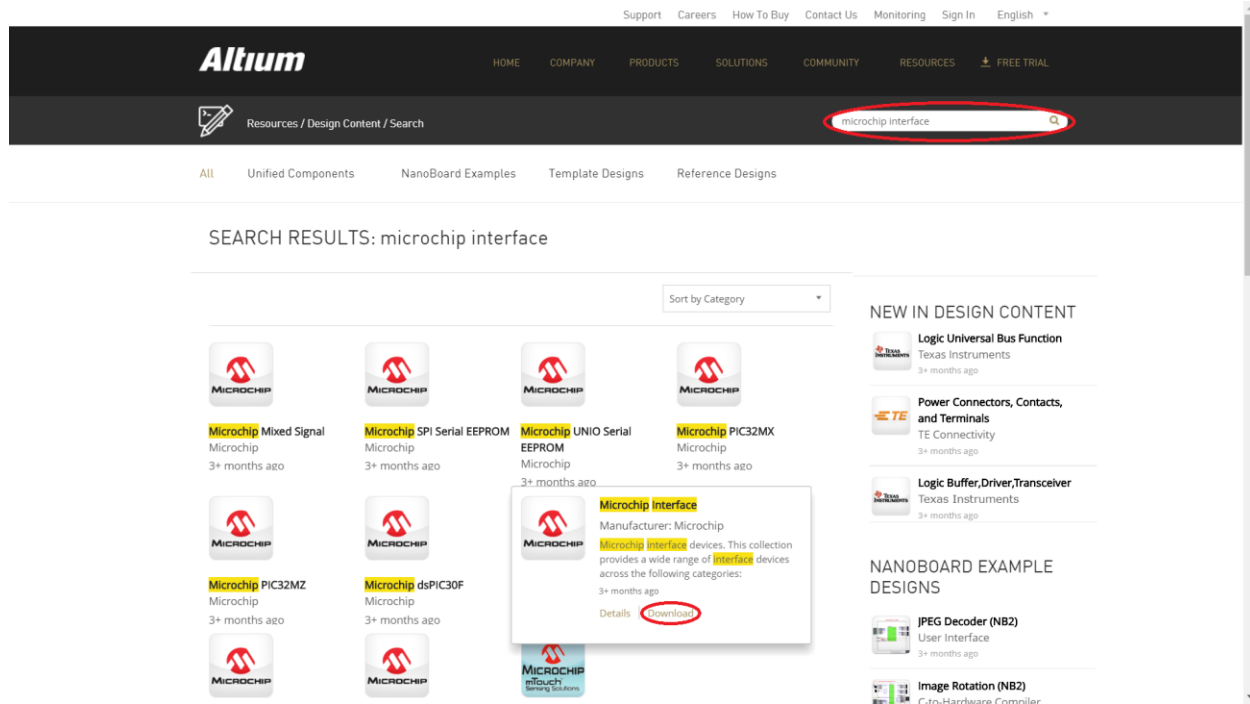
If your interface becomes too crowded with panels like it is below, you can manage panels with a button in the bottom right corner of your screen. When working on schematics, I like having the Libraries panel and sometimes the Properties panel open. The projects panel can be closed until later.



We can find new libraries full of components on Altium's website. Manufacturers publish schematics for their products in easy-to-install packages.

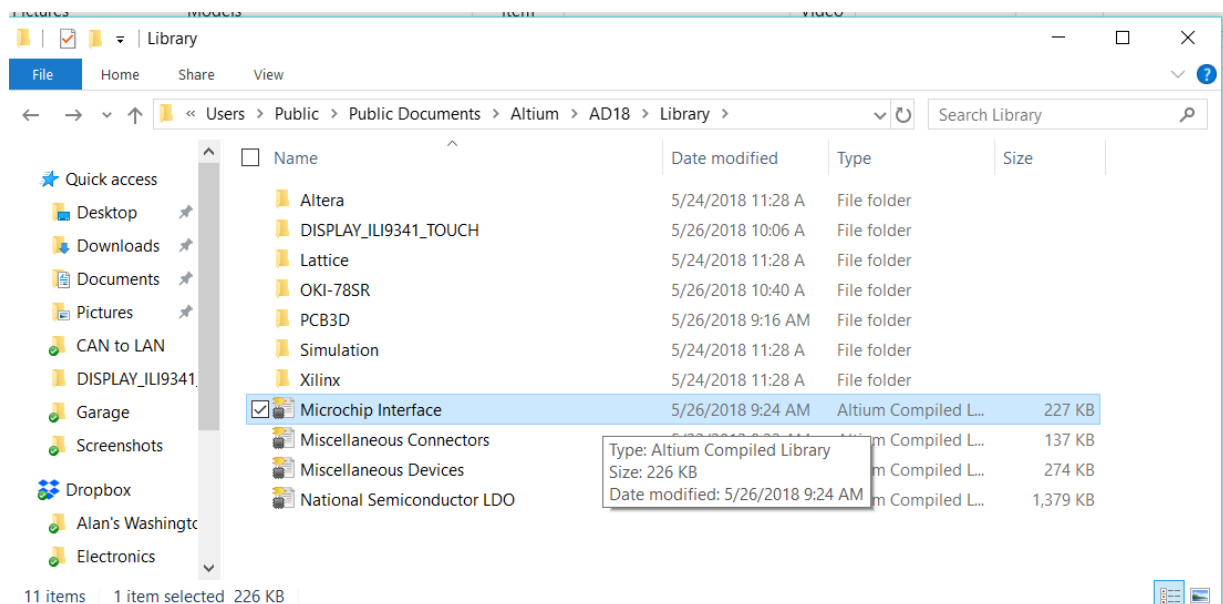
## HOW TO INSTALL A NEW LIBRARY

Visit <https://designcontent.live.altium.com/> and search for the name of your desired library. Here we will install a Library called “Microchip Interface”

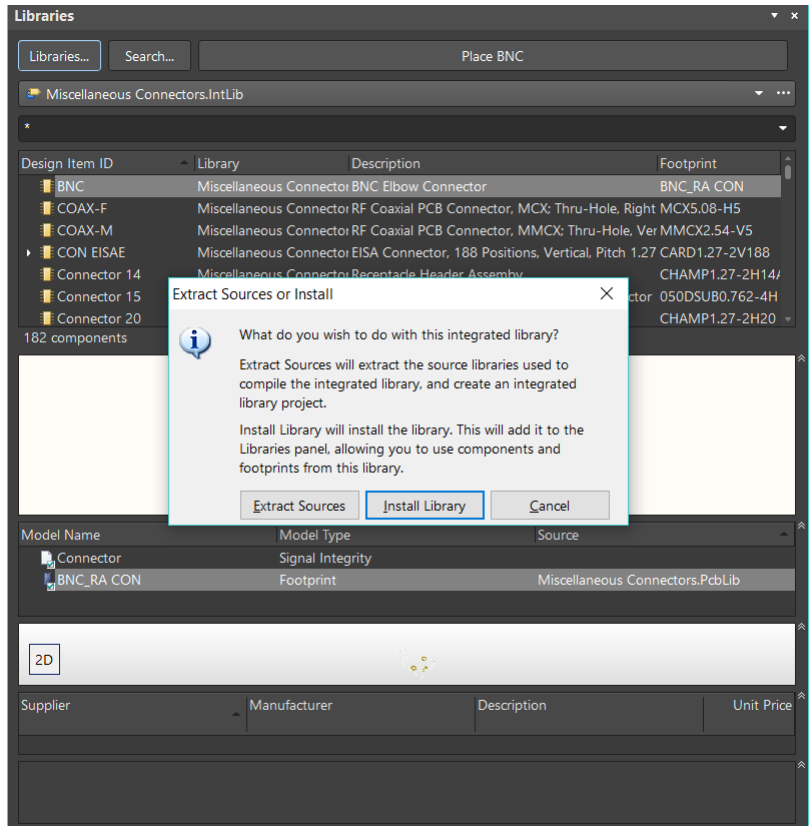


Download the file, unzip it, and move it to your computer's directory.

C:\Users\Public\Documents\Altium\AD18\Library



Above you can see three other Libraries I have downloaded before. I recommend you get these from the website, too. Double-click on the library, click on “Install Library” and it should upload to your Libraries in Altium.



Now let’s actually add some components to the schematic. Select the component that you want from your selected library and click on the schematic board.

The pins on the components will connect to each other if they are named the same thing. For example, all pins that require 5 volts should be named 5V. All of the pins labeled with “5V” will be connected to each other when we route the copper in the PCB sketch.