University of Pittsburgh
Department of Electrical and Computer Engineering
ECE 1895: Junior Design Fundamentals
Dr. Samuel Dickerson

Assignment 2: 555 Timer PCB Design

Prior to attempting this assignment, **you should have completed the at least one of the Altium tutorials listed on CANVAS**

For this assignment, you will

- Implement your 555 Timer Design as a PCB
- Verify its manufacturability
- Generate the PCB Output files

To get started:

- Refresh your memory on your 555-timer design. Please refer back to the datasheet if necessary.
- Create a new Altium project
- Import the 555 timer chip into your library

Sometimes, the schematic symbol and PCB footprint for a circuit component will not be available within Altium Designer's default libraries and you will need to search for them. There are several places where this information can be found, however we will focus on obtaining schematic symbols and PCB footprints from DigiKey, a well-known and widely used distributor of electronic components..

Visit DigiKey's website and do the following:

1. Search for the specific 555 timer component we are using on DigiKey's website using its part number
    - We are using the ***296-NE555P-ND***, a 555 timer from Texas Instruments.

2. In the ***"Media and Downloads"*** section of the part's webpage, select ***"Download from SnapEDA"***.

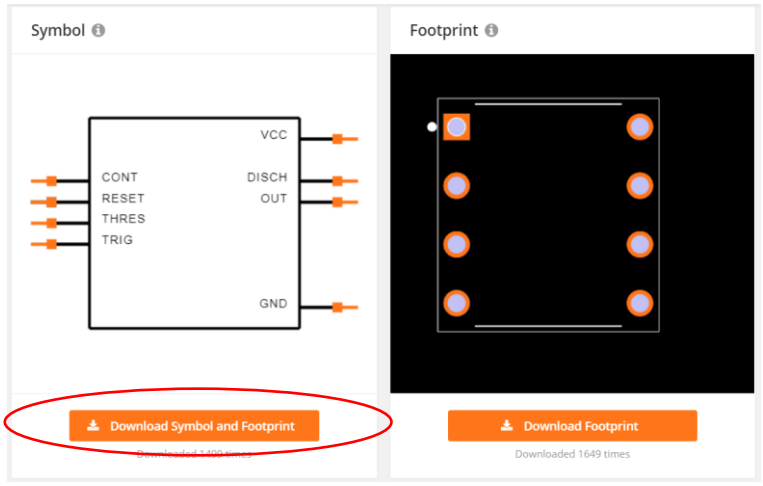
Media & Downloads

RESOURCE TYPE	LINK
Datasheets	NE555DE4
Featured Product	xx555 Single Precision Timers
PCN Design/Specification	Material Set 30/Mar/2017
PCN Assembly/Origin	Qualification Revision C 23/Dec/2014
EDA / CAD Models	NE555P by SnapEDA NE555P by Ultra Librarian

NOTE: If you search for a part on Digikey and it does not have an SnapEDA model listed, then the schematic symbol and/or PCB footprint will need to either be downloaded from the manufacturer's website (if available) or you will have to create it yourself in Altium Designer.

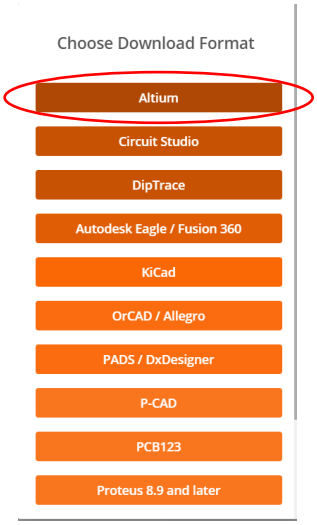
SnapEDA is a website that maintains a vast library of ready to use schematic symbols and PCB footprints.

3. After you have created an account, click ***"Download Symbol And Footprint"*** for the component you are searching for. This will make it easy to use the part in both the schematic and PCB layouts.

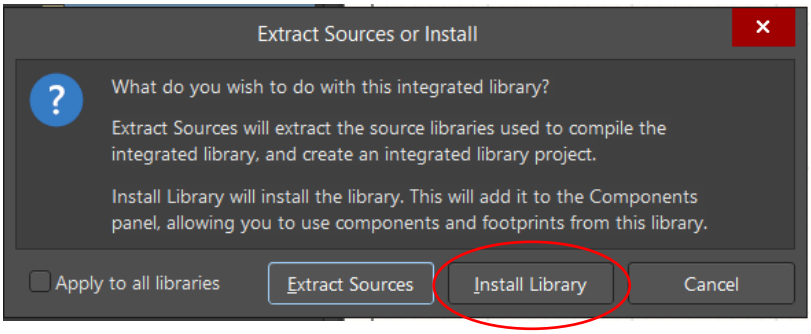


4. When the menu appears to ***"Choose Download Format,"*** choose ***"Altium."***

5. ***Create an account if it prompts you to***, then open the file downloaded. Make sure you have Altium Designer opened.

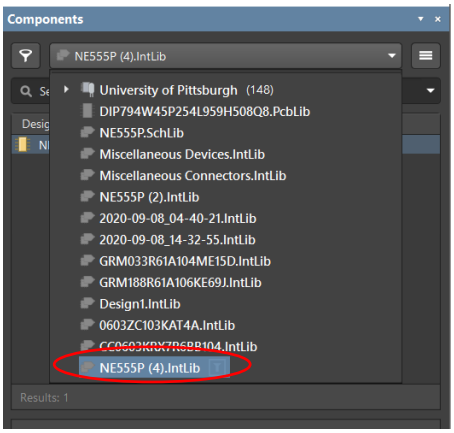


6. When the file is opened, the following menu will appear.

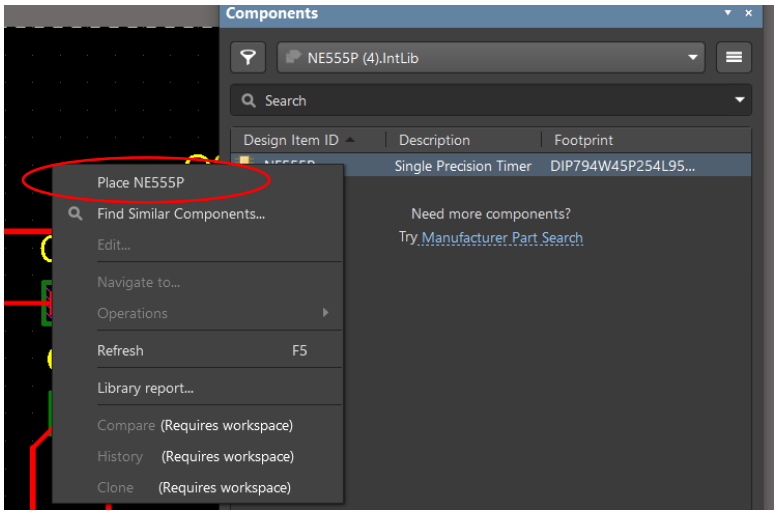


Click **"Install Library"** so that the component is easily accessible from Altium's **"Components Panel."**

7. In Altium, open the **"Components"** Panel. In the drop-down menu, you should be able to see the component you have just installed.



8. Once you click on the component, you can place the component by right clicking it and then selecting **"Place NE555P."**



You are now ready to create your design

- **Begin by creating a new Altium Project.**
- **First, create your 555-timer schematic. your schematic**
    - Don't forget, you must include a way (i.e. a connector) for you to deliver VDD and GND to your circuit.
    - You are free to use any components that you wish to complete your design.
- **Next, create a corresponding PCB layout.**
    - You may use through-hole or surface mount components. Just make sure that you are cognizant of what you are using.
- **Verify the connectivity of your board and make sure that the extracted layout netlist is equivalent to the schematic netlist (LVS)**
- **Make sure that your design passes all Design Rule Checks (DRC)**
- **Use Altium to generate the Bill-of-Materials (BOM)**

Submission

Submit the following to CANVAS before midnight

- A brief, 1-2 Page description of your PCB design. In your document (saved as a PDF file), include screenshots and explanatory text for the following items from your design
    - Circuit Schematic
    - PCB Layout. Make sure to report the approximate X-Y dimensions of your board
    - Report logs showing that all connectivity and manufacturing design rule checks successfully passed
    - Altium Generated Bill-of-Materials
- A zip file containing all your Altium Designer files
- A second, separate zip file containing all the manufacturing files (GERBER) and the exported Bill-of-Materials (BOM)