

# **PCB Artist Introduction**

## **1<sup>st</sup> Laboratory Report for ECE 383 Microcomputers**

**Submitted by  
Patrick Brooks  
11650957**

**The University of Alabama  
Tuscaloosa, AL 35487**

**1/22/21**

**Abstract:**

The objective of this lab was to familiarize students with a software they will be using throughout the semester, this software is PCB Artist. Students will learn how to design electronic circuits using this software. Their last objective of this lab is for the students to design and construct a PIC24HJ128GP502 component within PCB Artist. Task one ask students to draw and label a four layer PCB, define terms used in PCB design, and to capture these tasks. Task two instructs students to create a digital clock circuit schematic within PCB Artist using a LM555CJA integrated circuit. Students use two resistors and one capacitor along with the LM555CJA. Student screenshotted their schematic and included it in their lab report. Task three student take their schematic from task two in order to create a PCB. After reading through the tutorial on how to convert a schematic to a PCB, students will make sure all components and nets are automatically routed/placed by the software. Student will then screenshot their PCB and include it in their lab report. Task four asks students to create a PIC24HJ128GP502 using PCB Artist. They are instructed to create a library with the name “ece383”. Student will then use the symbol wizard tool in order to effectively create their PIC24. Once in the wizard student will follow the prompts to recreate the component given to them in the lab instructions. This includes creating the appropriate number of pins, labeling these pins, and saving to the ece383 library. In order to make this creation a useable component within a schematic, students must solidify this component by going to the “Components” tab, creating a new “.cml” library called “ece383”, and following the instructions while in the wizard. After students save this .cml file to the appropriate library, they will be prompted to assign terminal names to the pins. Lastly, after assigning terminal names to the correct pins, students will find this component in the “ece383” library and must simply drag the PIC24 into the schematic design window for a screenshot. After completion of the lab students learned how to create a schematic, convert a schematic to a PCB, creating your own library, and creating/saving a component.

## Introduction:

The objective of the lab is to familiarize the students with PCB Artist while having them create a schematic, a PCB, and a new library with a custom component stored inside. Task one focuses on introducing students to PCB design. They are asked to draw and label a 4 layered PCB as well as defining terms used in PCB design. In task two students are instructed to build a 555 timer schematic within PCB Artist. This timer uses two resistors, a capacitor and a LM555CJA integrated circuit. Task 3 is where students convert their schematic from task two into a PCB. They use the software to automatically route nets and components. Task four asks students to create a new library that will store a component in which they will design. This component is a PIC24HJ128GP502.

## Procedure/Results:

Task 1:

- a. Draw and label a four-layer PCB

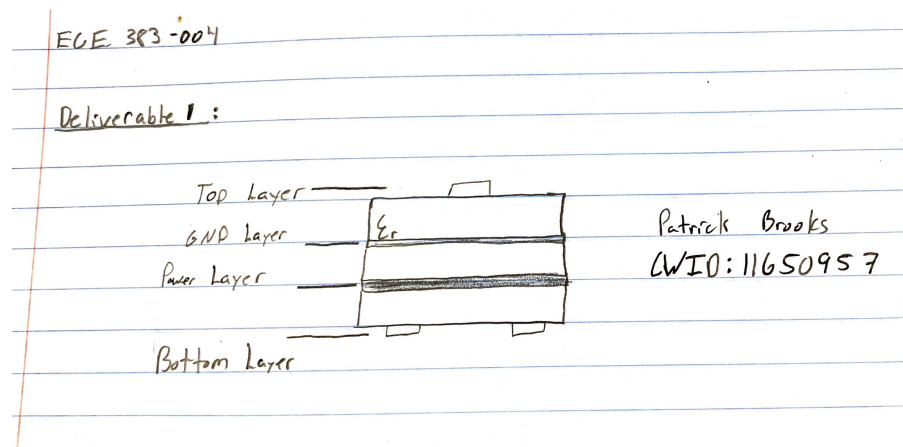


Figure 1. Drawing of four-layer PCB

Task1 (cont.):

b. Define given terms

Trace: The trace(s) of a PCB refers to the routes of wiring, fuses, copper, and insulation.

Pad: The pad(s) of a PCB refers to the exposed copper surfaces that allow the soldering of a component to the board.

Panel: The panel of a PCB refers to the nonconductive layers that make up the board.

Plated through hole: Plated through holes are holes that have been punched through a PCB and plated with copper. This allows for components to carry current through to the other side of the hole.

Silkscreen: Silkscreens are thin layers of ink printed on a PCB to identify components, warnings, test points, and parts of the PCB etc.

Solder mask: Solder mask is a thin layer of polymer that is used to protect against oxidation and soldering errors.

Surface mount: Surface mount technology is a method used to mount electrical components directly onto the PCB.

Via: A via consist of two pads that are electrically connected by a hole through the PCB. They are used to join traces and pads.

Plane: A plane is typically a large area of copper foil connected to the circuit's ground point.

Patrick Brooks  
ECE 383-002  
CWID: 11650957

Figure 2. Defining given terms in my own words

## Task 2:

- Find the  $R_A$  resistors, “C” capacitor, GND, VCC, LM555 and drag into window
- Set up components as seen in lab instructions
- Change values on  $R_1$  to 10K,  $R_2$  to 100K, and C to 1uF
- Route connections to components
- Screenshot schematic with name and CWID visible

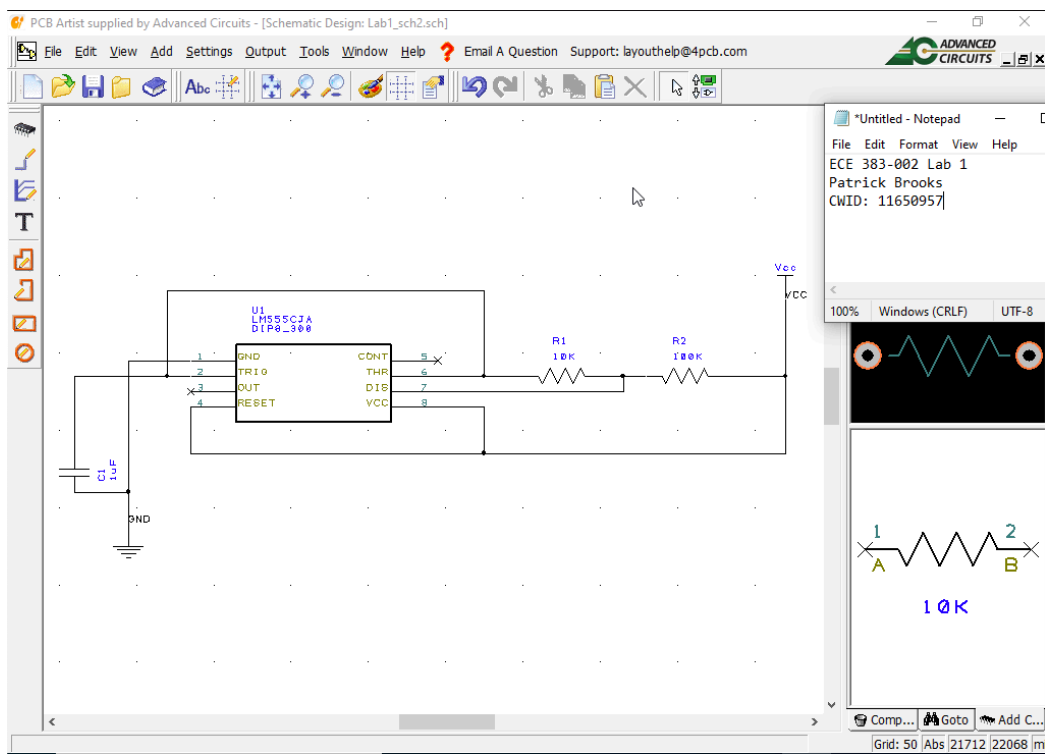


Figure 3. Schematic of LM555 timer

**Task 3:**

- a. Convert schematic from task two to a PCB by following the tutorial provided
- b. Once the PCB creation window appears, follow tutorial
- c. Upon exiting of PCB creation window, auto place components and nets
- d. Screenshot schematic with name and CWID visible

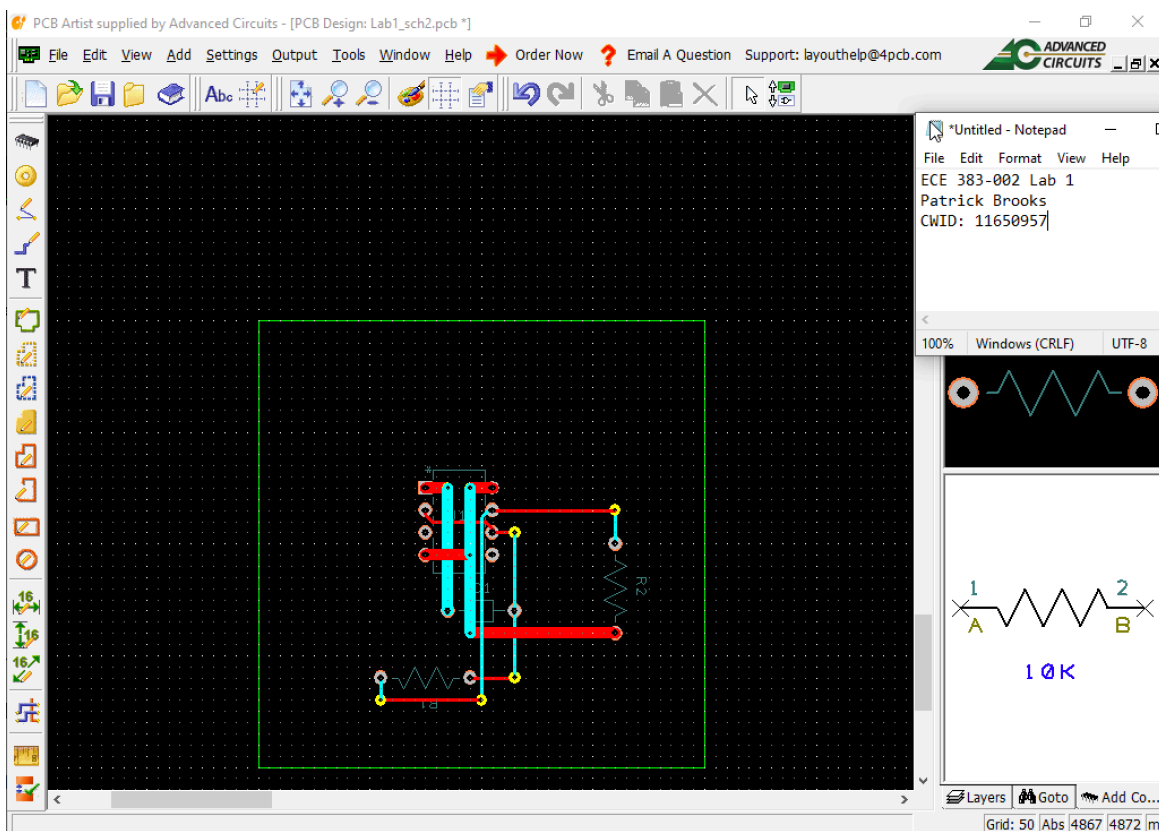


Figure 4. Converted schematic to PCB

## Task 4:

- a. Complete tutorial
- b. Within the Library Manager Create a “New Lib”
- c. Name this library “ece383.ssl”
- d. Use the symbol wizard tool to create PIC25HJ128GP502
- e. Refer to lab pdf once the “Technology” window appears
  - i. change units to mm
  - ii. change precision to 2
  - iii. select rectangle when asked “What kind of symbol”
  - iii. change origin to pin 1, change to 28 pins
- f. Save symbol as “PIC24HJ128GP502” and save to “ece383” library
- g. Click on “Components” tab and “New Lib”
- h. Create a new library named “ece383.cml”
- i. Click on “Component Wizard” and follow the lab pdf settings
- j. Upon saving your component, you will be prompted to name terminals
- k. Refer to lab pdf for correct terminal names along with matching pins, screenshot result

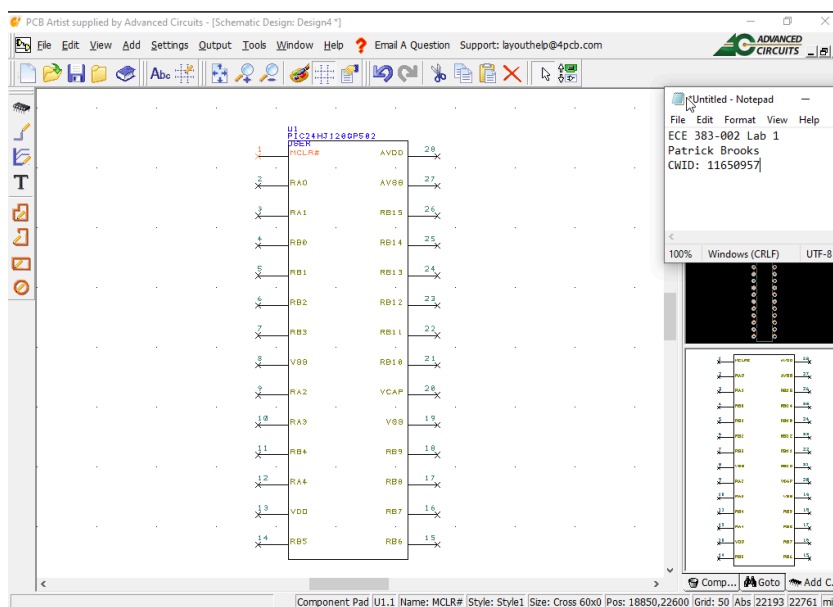


Figure 5. PIC24

Task 4(cont):

1. Provide names of two similar commercially available CAD programs

Two Similar CAD Softwares:

1. Solidworks PCB

2. Altium Designer

Patrick Brooks

CWID: 11650957

Figure 6. Similar commercial CAD softwares



**Conclusion:**

After completion of the lab students learned how to create a schematic, convert a schematic to a PCB, creating their own library, and creating/saving a component. Students now have a familiarity with PCB Artist that was nonexistent prior. Students also learned the lab report format, which will be used throughout their academic endeavors.