



*“Blink your LED and mesmerize people”*

# Introduction:

This module will teach you how to use “Eagle CAD” Electronic layout software to design a schematic and corresponding circuit board suitable for manufacturing on the ECE Department/LPKF Milling machine or by any online circuit board manufacturing house.

Many of the devices that will be used for this class can be obtained by going to SERC (room 1217 Benedum Hall)

## Components and Tools:

The components and devices that will be used for this module are as follows:

1. Eagle CAD Circuit board design and layout software. Available by visiting [www.software.pitt.edu](http://www.software.pitt.edu) and searching for “Autodesk” Alternatively you can visit this link: <https://www.autodesk.com/education/free-software/featured> You will be required to create an account on their site. Use your “Pitt.edu” email to register.
2. Components needed for assembly. Notice one "surface mount" and the rest are "Through Hole"  
For an explanation of each please click here: [Surface Mount vs Through Hole](#)

# 1270 PCB

## Components

470 Ohm Res.



1K Ohm SMD (1210 Package)



LED



1 uF Cap.



.01 uF Cap.



Header



DC Power Jack  
2 Views



Fuse Holder  
2 Views



Fuse



1 Meg. Ohm Pot  
3 Views



555 Timer IC w/Socket







## 2. Pliers



## 3. Wire Cutters



## 4. Brain



## 5. Microscope

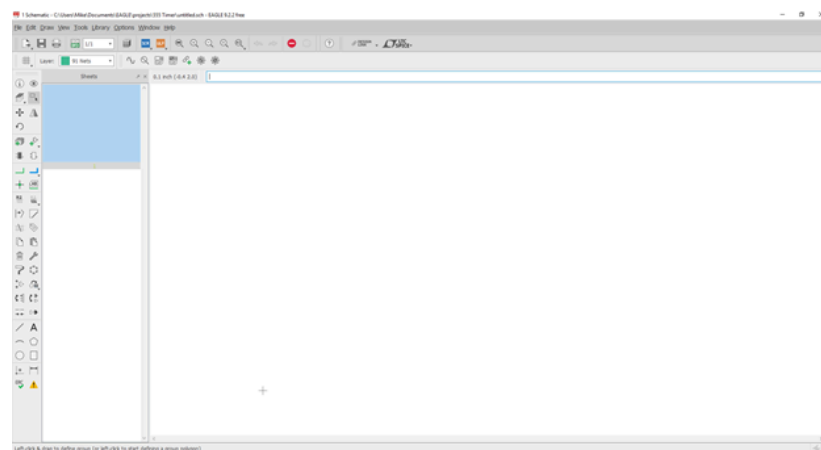
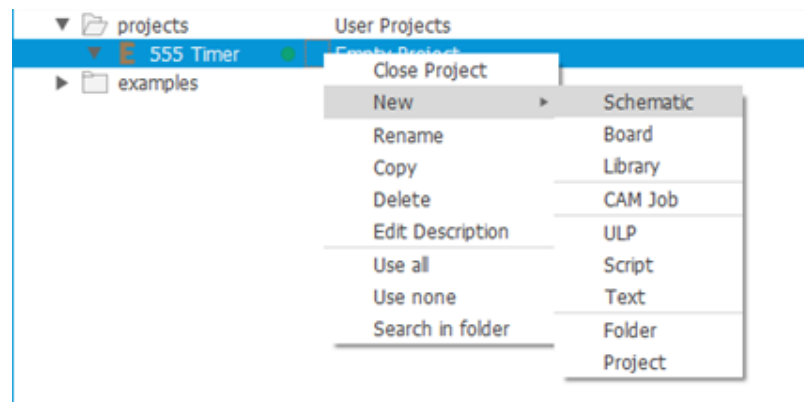


# Section 1: Schematic Layout

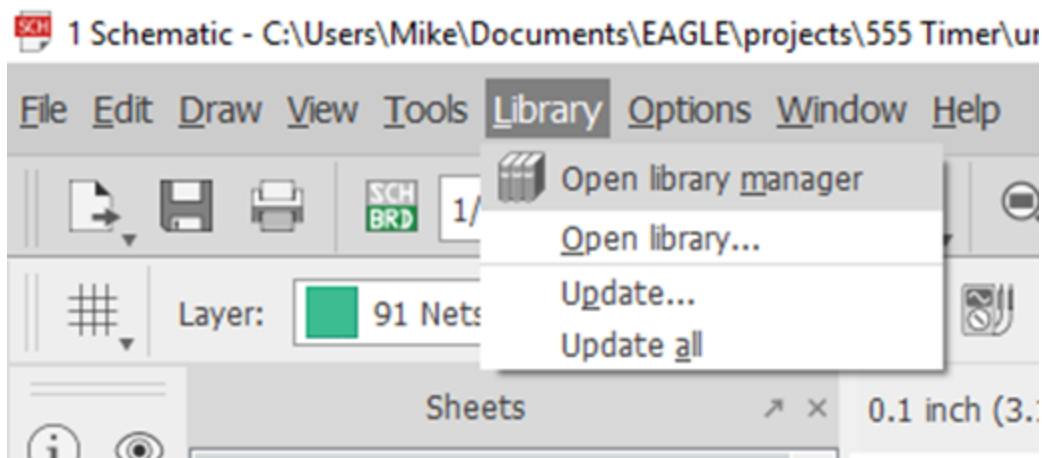
This section will teach you how to create a schematic. You need to think through this section carefully and methodically. All the electrical connections, parts and wiring need to be correct before you move to the board layout (section 2). Once you complete this section you will submit your schematic and BOM (bill of material) to courseweb for checkoff.

While laying out your circuit keep in mind that any mistakes you make will follow you to your board layout. This causes a problem because making electrical changes in the board layout will not update your schematic, thus compounding the problems and confusion. For the time being “Think Electrically” and make sure your parts and wiring are exactly as specified.

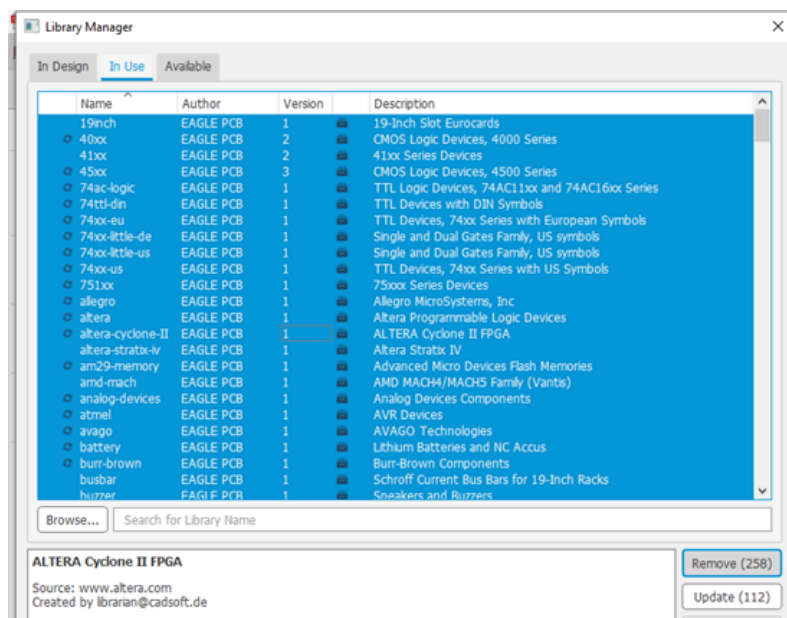
1. Download and install Eagle onto your computer. (PC/MAC) Eagle can also be found by logging into the computers in the 12th floor classrooms 1223ABC and 1211AB.
2. Open Eagle and create an autodesk account using your “pitt.edu” address or login with your autodesk account and the program will start up.
3. From the Eagle control panel, navigate to the projects folder and expand it.
4. Right click on projects and create a new project called “555 Timer”
5. Right click on your newly created project and open it.
6. Right click again on your project and choose “new” Then select “Schematic”. A new schematic window will appear. Maximise this window to full screen.



7. Go to the menu bar and choose “Library” Then open the Library manager.

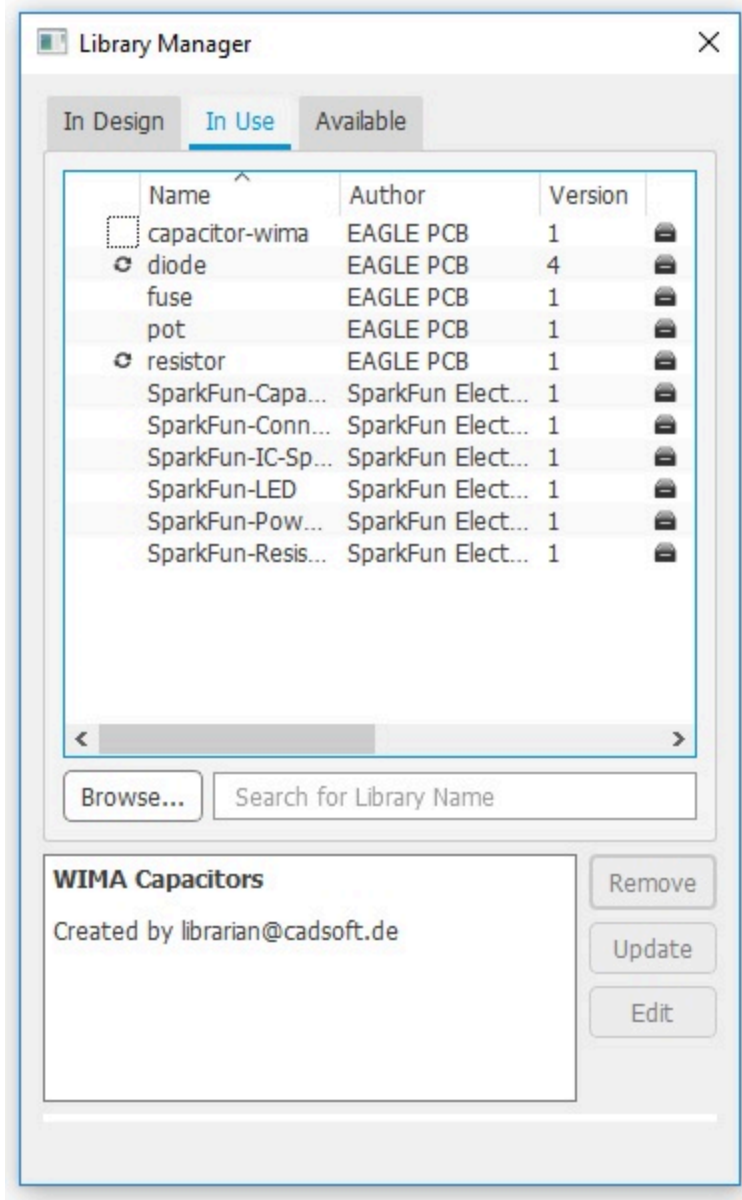


8. Choose the “In Use” tab and select everything by hitting ctrl + A. and choose “remove”.



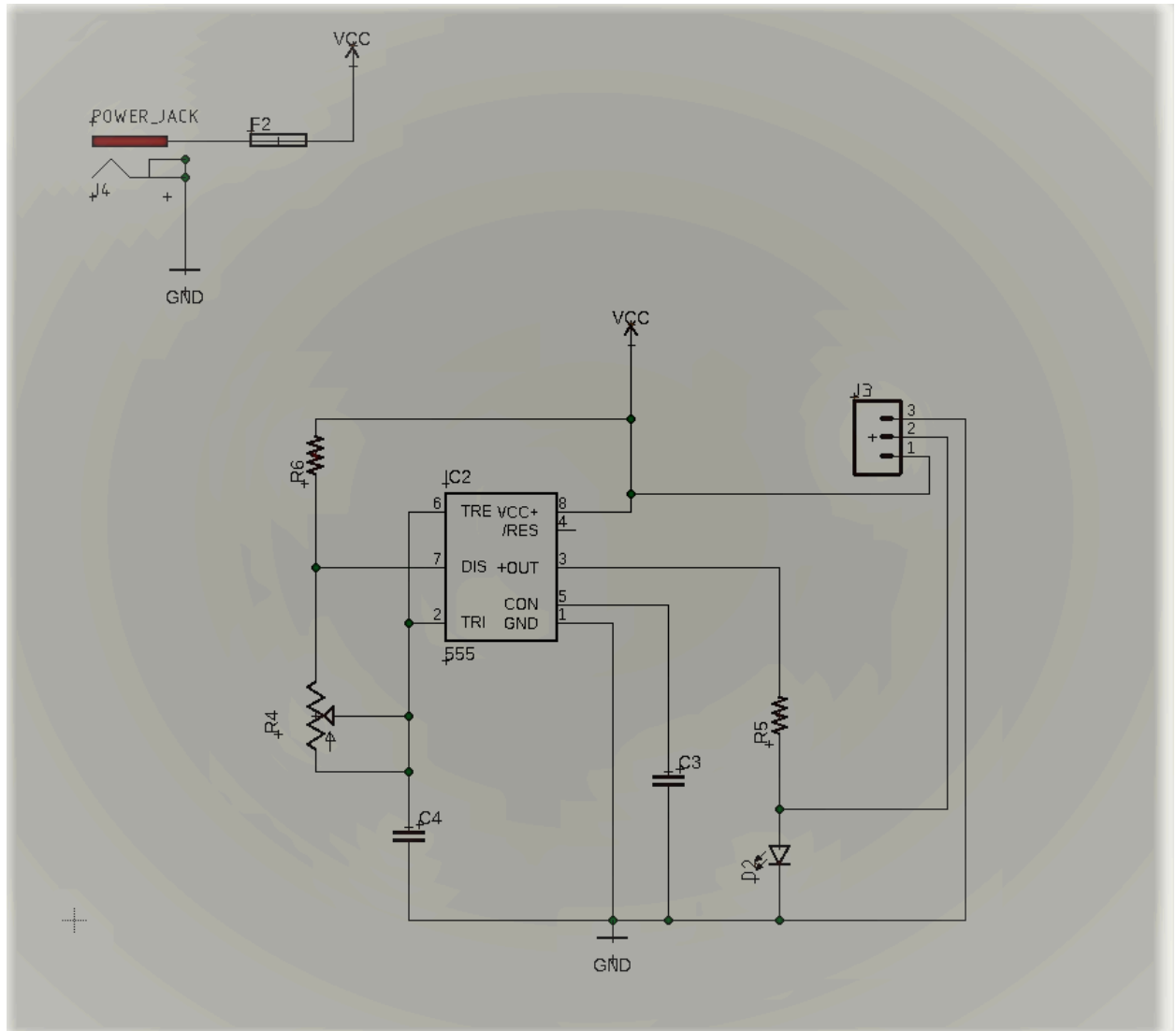
9. Now select the Available tab and browse for the following libraries and click “use” after locating each one. This will add them to your “In Use” Tab
- Capacitor - WIMA
  - Diode
  - Fuse
  - Pot
  - Resistor
  - Sparkfun
    - Capacitors
    - Connectors
    - IC Special Function
    - LED
    - Power Symbols

NOTE: Your “In Use” Tab should now look like this image.

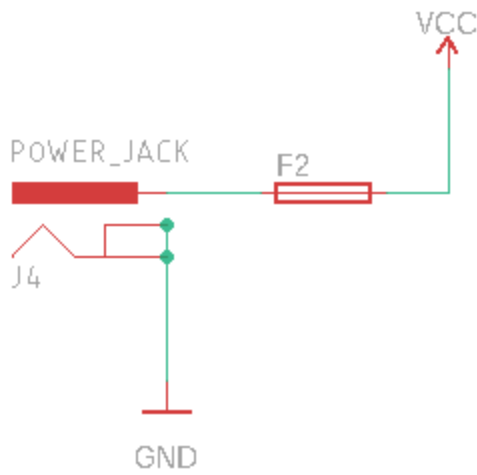


10. You can now close the library manager and return to the Schematic window.
11. You need to become familiar with the tool bar on the left side of your screen. This contains all the required functions to layout the “Schematic” portion of your circuit. The board layout will be discussed later in this tutorial.
12. Locate the “Add Part” icon on the toolbar. Opening it will bring up the library list we just created. Choose the parts listed below and place them into your design by clicking in the schematic layout window. When you hit esc after placing them Eagle returns you to the library. Chose the next component and continue until all are placed on your schematic.
  - a. 555 timer IC - Library: Sparkfun-IC-Special-Function/555 Part name: 555P
  - b. VCC - Library: Sparkfun-Power-Symbols Part name: VCC (qty:2)
  - c. GND - Library: Sparkfun-Power-Symbols Part name: GND (qty:2)
  - d. Potentiometer - Library: POT/Trim-US Part name: Trim\_US-CA6V
  - e. Surface mount resistor - Library: Resistor/R-US Part name: R-US\_R1210

- f. Through hole resistor - Library: Resistor/R-US Part name: R-US\_0207/12
- g. Capacitors - Library: Capacitor-WIMA/C Part name: C7.5/3 and C15/5
- h. LED - Library: Sparkfun-LED/LED Part name: LED 5mm
- i. Fuse holder - Library: Fuse/SH22 Part name: SH22
- j. Header 3 pin - Library: Sparkfun-Connectors/Conn\_03 Part name: Conn\_03
- k. DC Power jack - Library: Sparkfun-Connectors/Power Jack Part name: Power\_Jack (PTHLayout)
- l. Wire your schematic as the image below shows using the wire/line tool.

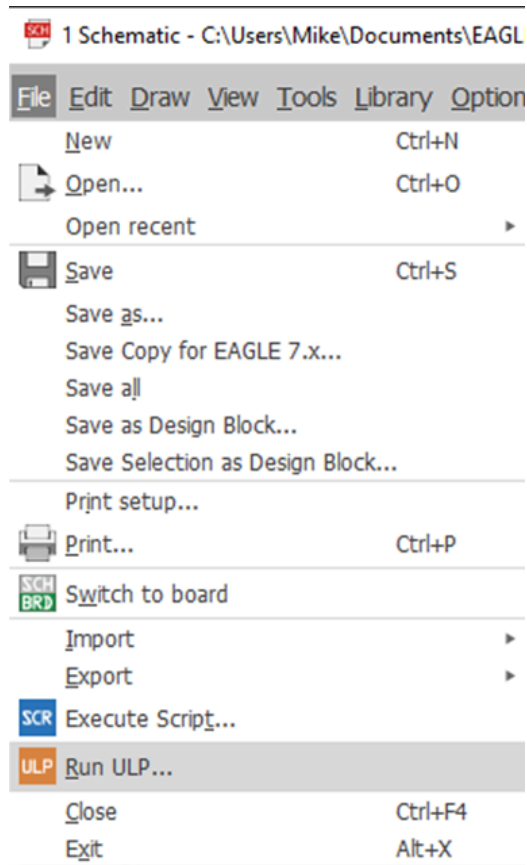


13. Notice how some of the wires have circles connecting them to certain wires of the circuit. These are called “junctions” they are there to make electrical connections. If you don’t put these in, the wires will simply cross over each other and not electrically connect.
14. You will also have to wire in the fuse holder and power jack as below to verify the power jack connections use a hand held meter.

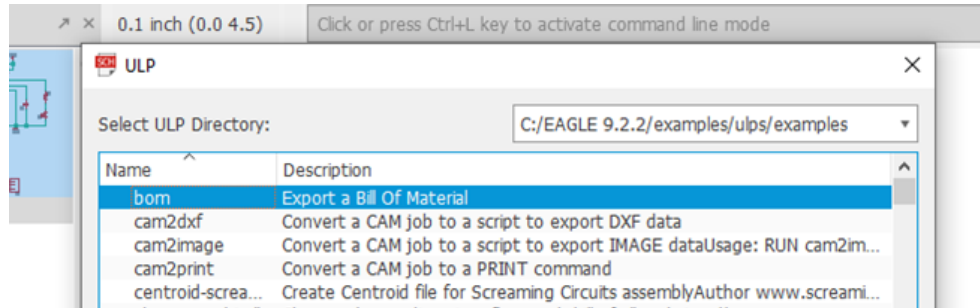


15. The final step is to produce your BOM (Bill of Materials)

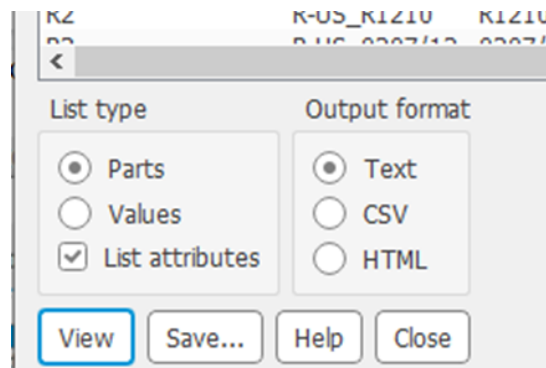
16. Click on the file menu and choose “Run ULP”



17. The complete list of jobs appears. Choose BOM to Export a Bill of Material.



- a. Please note: make sure you format the BOM by using one of the following export options. Doing so will format your output properly for viewing and printing via notepad.
  - i. List type: PARTS, Output format: Text
  - ii. List type: Values, Output format: Text

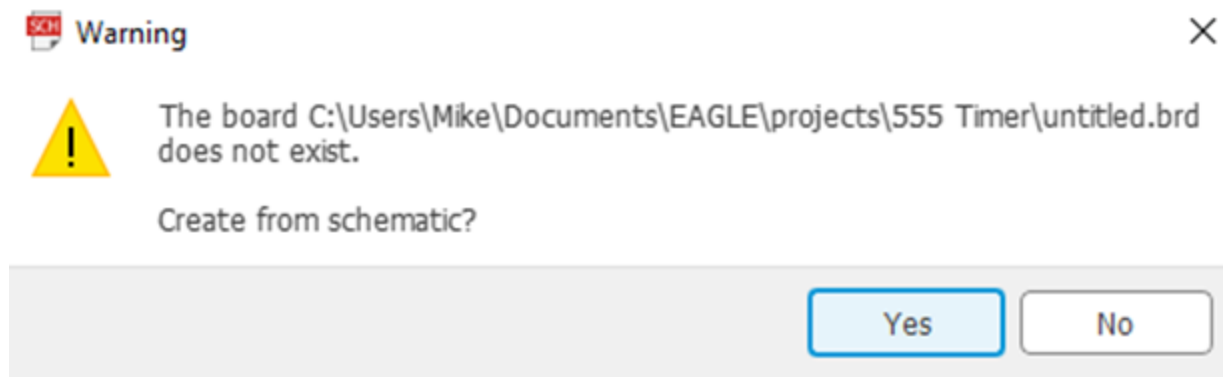
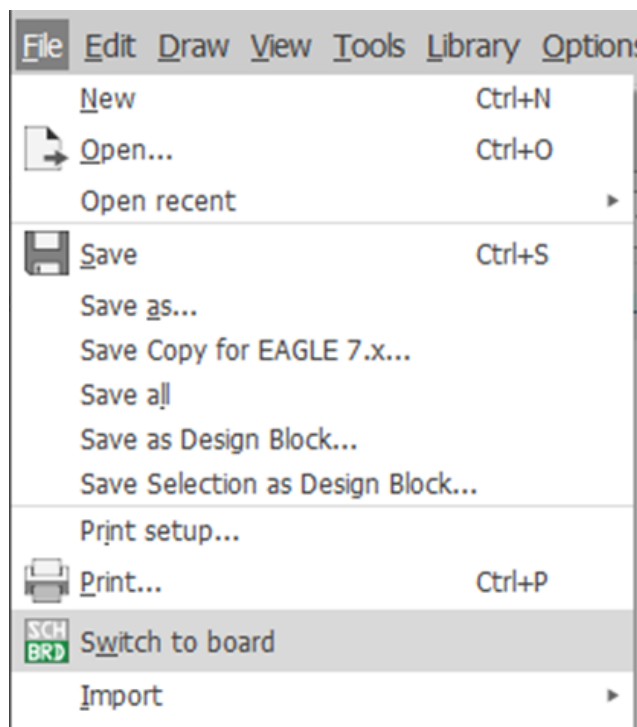


- b. Make sure you save your BOM to your project folder
18. You can now submit the schematic and BOM to courseweb for review. This concludes the Schematic portion of the class. Congratulations!

## Section 2: Board Layout

During this section of the class you will take your Schematic and create a board layout. This section requires you to stop thinking electrically and start thinking physically. You will be laying out the physical location of the components on a simulated circuit board. We suggest you imagine holding your circuit board in your hand and thinking how it will look to you once you have all the components soldered on. Let's get started.

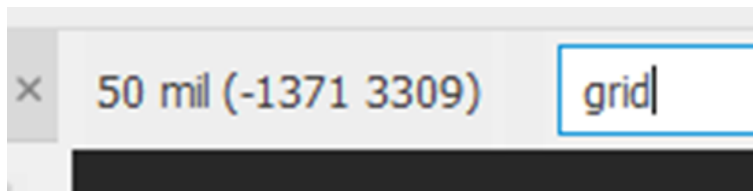
1. Open Eagle and your 555 timer schematic.
2. Double check to make sure your schematic is correct and has been approved. Any mistakes here will cause you more problems later.
3. Click file and then "Switch to Board". If you are prompted to create a board from the schematic, select "Yes". This will bring up the board layout screen. Maximise and notice that all your parts are in a big pile next to the board.



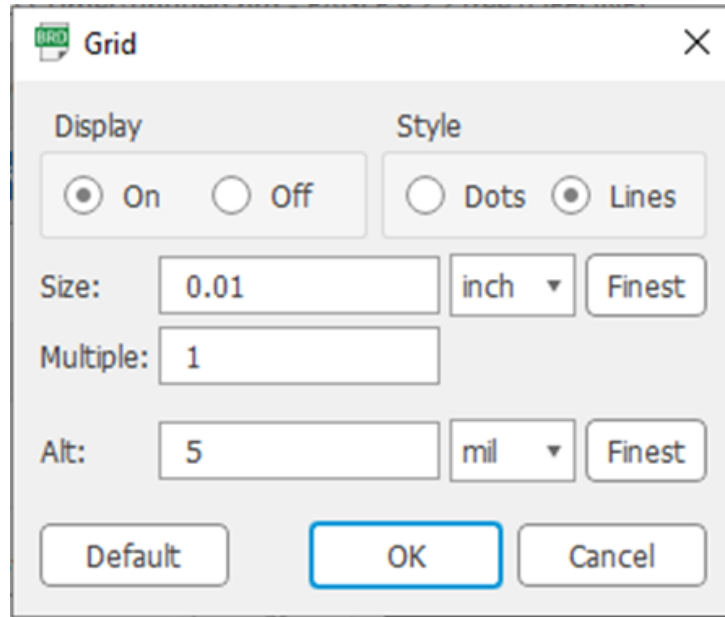
4. Also notice that the board outline is to the right of your pile of parts. This is what you need to resize to make it 3x3 inches as Bill has indicated approx the size of a post it note. If you're not exact its ok. But keeping it this size will be less expensive for you if the board is manufactured at a board fabrication house like: ALLPCB.com, Oshpark, Sunstone, etc...



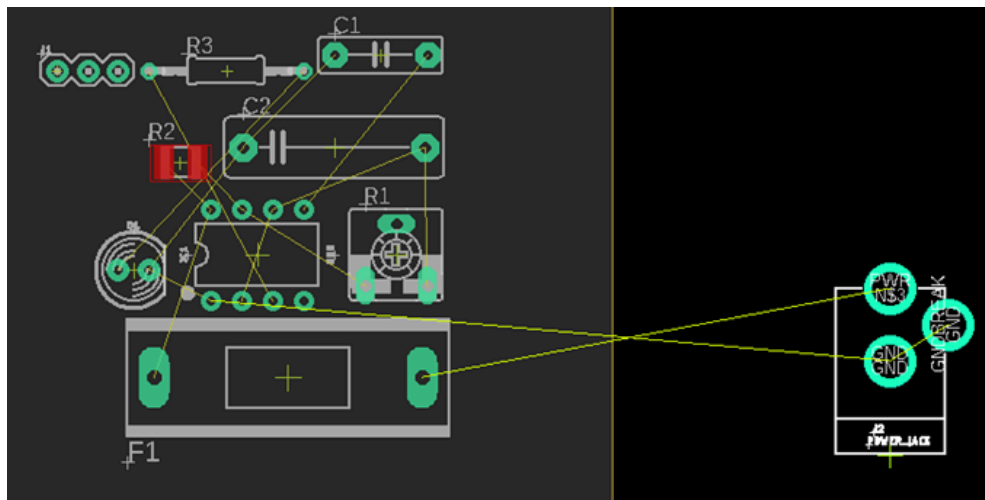
- Before you resize go to the command bar above your board and click it so you can enter a command
- Enter: grid



- You will see the grid window appear and can set the size to .01 and set the value to inches. Make sure you turn the grid **ON** else it won't be visible when you close the window. This will give you a grid and allow you to move about and place components with the same precision as a breadboard.



- d. Now grab the upper right corner of the board outline (yellow box) and move it while you watch the measurement indicator to the left of the command bar and set it to 3x3 as indicated above.
5. REMINDER: make sure you are thinking about physical placement now. This is how your board is going to look IRL.
6. Start by grabbing a component and dragging it over to the board. You will notice wires that are tied to it. These are called “Ghost Wires or Air Wires depending on your version of eagle”



- a. Ghost wires are just that. They are only electrical references to the connections you made in your schematic. Do not assume they are real wires connecting parts.
7. Continue to place all your components until you are pleased with the layout of your board.
8. Now you need to convert the “Ghost wires” to real traces. To do this you can right click a ghost wire and go to “Route”. You will see how the wire is transformed into a trace you

can drag around with your mouse. Also you will see a detailed menu of trace options above the command bar.

9. IT IS CRITICAL!!!! That you change the trace width. By default it is .006. This equates to the thickness of a human hair which is too small. **Change it to .032** this is a suitable width trace for signal lines.
10. SPECIAL NOTE: (optional) Power Traces should be slightly larger to allow for more current and reduced resistance. Remember the thinner the trace the more resistance it has.
11. Route all your ghost wires to their associated connection points.
12. Many students opt to use Autoroute to save time. However unless you know how to setup autoroute constraints and rule sets to follow I highly suggest you route your traces manually. Plus it's more fun to know where all your wires are going to be on your board. Especially for troubleshooting later. This circuit isn't complex so please try it manually first.
13. Once you have laid out your design choose a space somewhere there are no wires and add the following text to the TOP Copper layer. Be careful not to touch any of your traces with this text else you will produce a short.
  - a. Your name
  - b. Your project
  - c. Current Date
14. Adding in some holes for standoffs (mounts) will also help you later when you display your board or go to assemble it.

## Section 3: Gerber File export

Now that you have your virtual circuit board all laid out and looking great! The next step is to have it fabricated. In order to do this you need to export a series of files called "Gerbers" these files correspond to each of the individual layers of the circuit board and the machines (mills) that make your board require these files to accomplish their job. Below we will explain how to export your gerber files from your circuit board layout.

- 1) Start Eagle and open your board layout.
- 2) From the file menu choose "CAM Processor"

- 3) Next click on GERBER from the cam processor window (it has a little gear on it but don't click the gear, just highlight the line)
- 4) Click the "load job file" icon at the top of the window next to the save icon.
- 5) Choose "Open Cam file" navigate to C:\EAGLE 9.3.0\examples\cam\examples\Legacy
- 6) Choose the file "gerb274x.com" and click open.
- 7) Check the checkbox at the top that says: "Export to Project Directory" This will capture the files in a central location for you.
- 8) Notice you see a yellow exclamation point on the drill job above Gerber. Right click on it and select "Generate Excellon outputs based on PCB stackup"
- 9) Click "Process Job" You should get a successful popup or you will see a window about this being a "Mixed cam job" say YES to this and you will get "Job processed successfully"
- 10) Close the CAM Processor window and don't save changes.
- 11) Go to the Eagle Control panel and refresh the screen.
- 12) Under your project you will see a folder called CAMOutputs/DrillFiles and your drill output .XLN extension file. Right click and copy this file to a new folder somewhere on your computer or flash drive
- 13) Next copy the following files to the same location .brd/.cmp/.gpi/.plc/.pls/.sch/.sol/.stc/.sts
- 14) Zip up the folder containing all those files and submit them to courseweb for review.

Your boards will now be reviewed and created on the PCB Milling machine located in 1217 SERC (Student Electronics Resource Center) If you wish to see boards being made please contact your instructors for times when the machine will be running.

## Conclusion:

Congratulations! You have completed the 555 timer Circuit Layout/design and manufacturing class. You rock!

## References:

