Lab 8 - PCB Design: FUSION Software - 15 Marks - 2% of Final Mark

Student Name: ____

Student ID Number: __________________ Date: __________________

Lab Rules:

- Safety glasses are mandatory in the lab when the power is ON.
- No food or drink is allowed in the lab.
- Follow all SEMET Lab Rules (http://inside.senecacollege.ca/semet/code-of-behaviour)
- Lab must be handed in by the end of the lab.

Lab Objectives:

1. Create a new printed circuit board (PCB) project using FUSION software.
2. Identify and select recommended parts in accordance with functional specifications.
3. Design a schematic that conforms to the Electrical Rules Check and correct any errors.
4. Design a PCB layout that conforms to the Design Rules Check and correct any errors.
5. Create and manage board dimensions, trace widths, silkscreen text, and overall design.
6. Create a set of manufacturing files (Gerbers) and upload them to MySeneca.

Lab Materials:

1. Lab computer for MySeneca and Autodesk FUSION software.

Part 1: Introduction and Background

All modern electronics rely upon Printed Circuit boards (PCBs) to provide a compact, efficient, and reliable connection between various components.

PCBs are usually comprised of one or more layers of fiberglass and copper laminated together, where the copper is designed in specific patterns. Circuit boards serve two main purposes:

- Provide a stable and rigid surface to mount components
- Create electrical connections between the components

To achieve these objectives, several features are etched into the copper of a PCB:

Pads - Pads are the locations in which components are inserted into the board and soldered.

Traces/Tracks - Traces are the metal paths used to connect various components together.

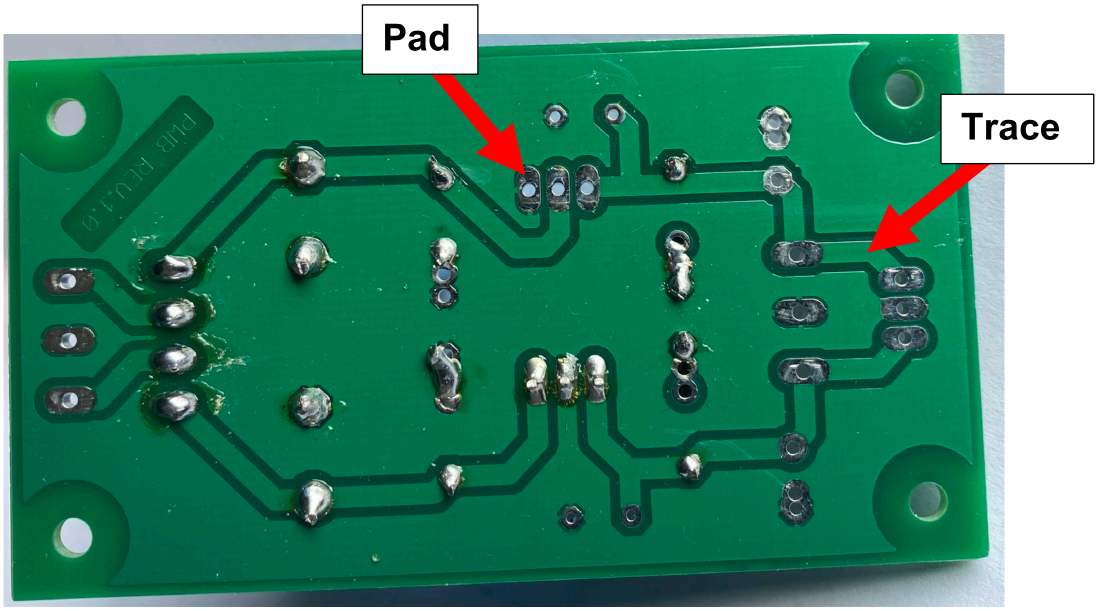

Figure 1 *- Soldered side of a printed circuit board (PCB), noting pads and traces.*

From **Figure 1** above, students can practice doing the following:

- Count the number of pads and traces present,
- Identify the location and number of mounting holes, and
- Locate the revision board number.

Note: Students can design and print PCBs for free using SEMET resources. This lab paper will cover all the steps to create manufacturing files which can be sent to the PCB lab for printing.

Keep this lab as a handy guide as you will be required to design PCBs for upcoming classes!

Part 2: Schematic Capture 6 Marks

Our first step in PCB design is to create a **Schematic** based upon the parts that will be used in the board layout. This task will cover the process of creating a project and Schematic Capture.

1. Launch Autodesk FUSION 360 from MyApps. Login to your Autodesk account.

2. On the **Data Panel**, click **New Project**.

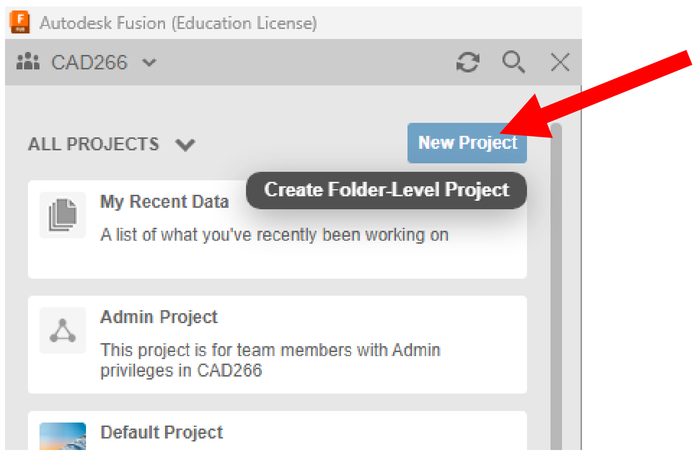

Figure 2 - *Create a new project.*

Name the project: ***LIN155NxxL_LAB8_Lastname_Firstname***

Put your first and last name and lab section as part of the project name.

3. Navigate to the **Menu bar**, select **File > New Electronics Design.**

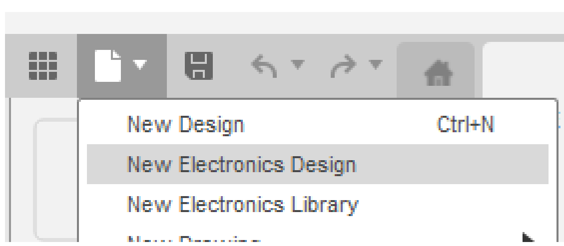

Figure 3 - *Create a new electronics design.*

4. Navigate to the **COMMON** tool bar and select **New Schematic**.

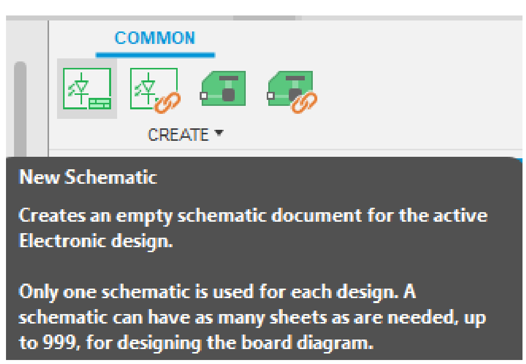

Figure 4 *- Create a new schematic.*

5. Click **Save** and name the schematic **LIN155_Lab8.**

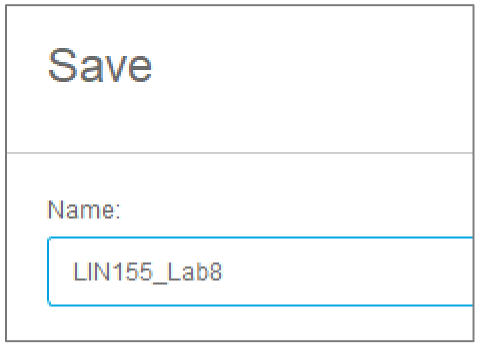

Figure 5 *- Save the project name.*

6. The schematic editor will open. Get familiar with locations of the main controls and their uses.

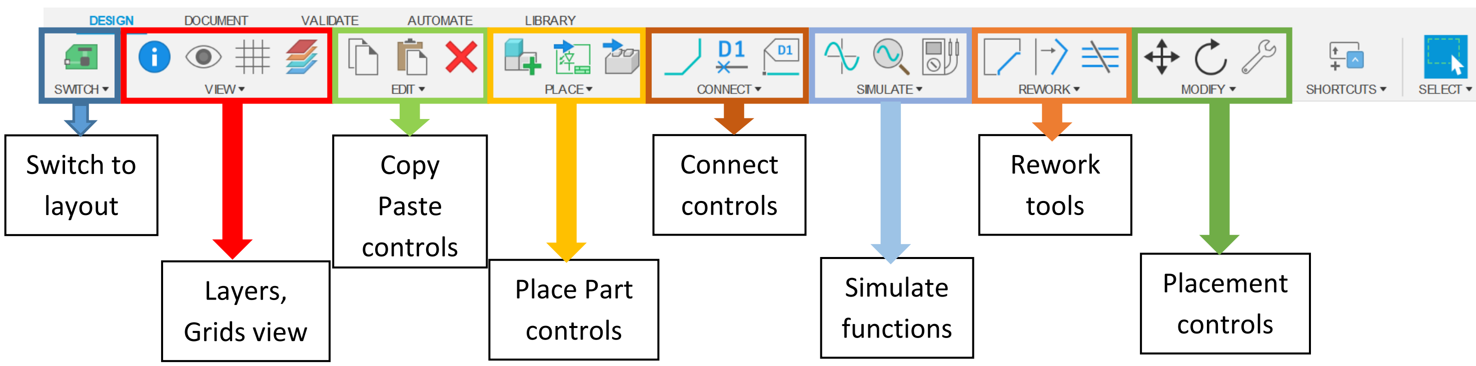

Figure 6 *- Fusion schematic tool bar.*

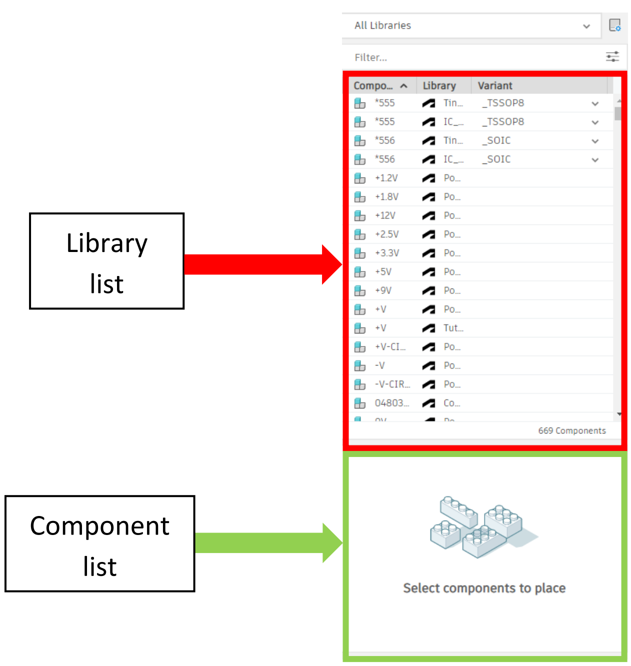

Figure 7 *- Library manager.*

We will follow each of these steps to complete the Schematic Capture design:

a) **Add Parts** to the design by searching and selecting each one from the library,

b) **Connect** the parts using the Net tool, to indicate where the traces will go,

c) **Run the ERC** or Electrical Rules Check to check for errors and fix them.

7. Let draw the schematic in **Figure 8** below.

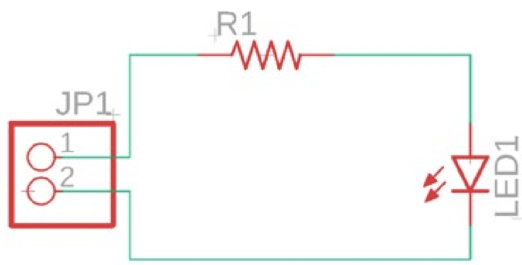

Figure 8 *- LED circuit.*

The schematic consists of:

- one **Resistor**: 1 kΩ, 1/4 W
- one **LED**: 5 mm
- one **Connector**: two-position, 2.54 mm/0.1” pitch (also called a header pin)

8. All components can be found under **rcl, pinhead** and **led** library under **EAGLE PCB** folder. We must add the libraries to our design first.

- Click on menu **LIBRARY** then select **Open Library Manager.**

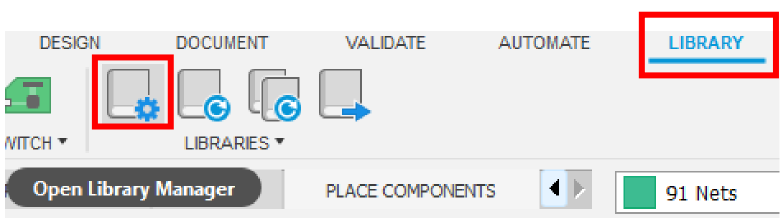

Figure 9 *- Opening Library Manager.*

- Search for **rcl** then enable the **rcl** library.

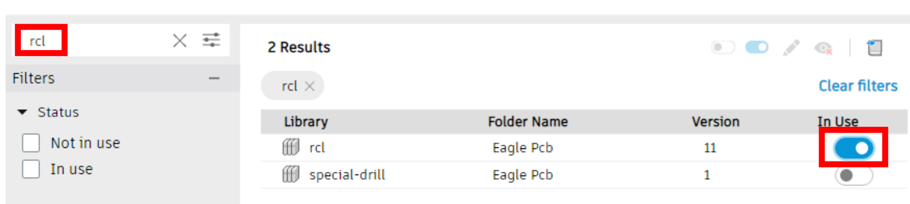

Figure 10 *- Enabling rcl library.*

- Search for **led** then enable the **led** library.

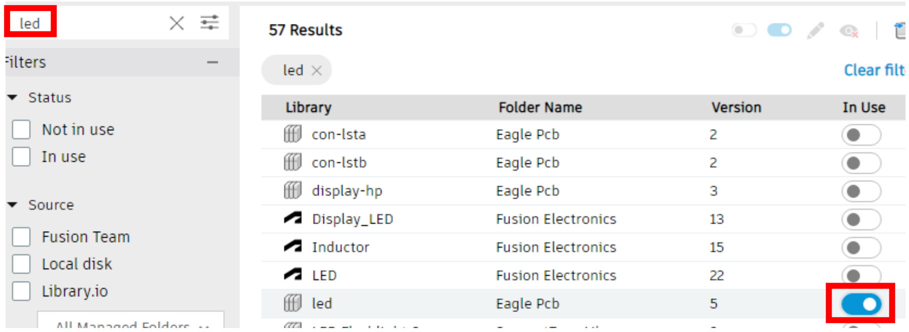

Figure 11 *- Enabling led library.*

- Search for **pinhead** then enable the **pinhead** library. Then close the **Library Manger**.

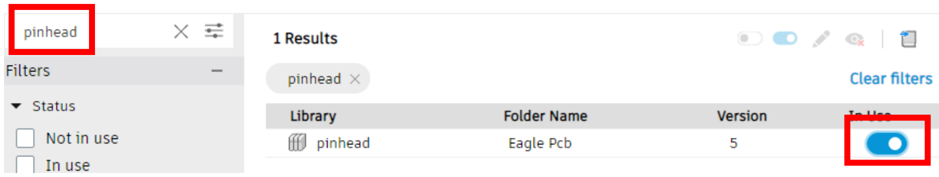

Figure 12 *- Enabling pinhead library.*

9. Begin adding parts using the **Place component** button, found under the **PLACE** section.

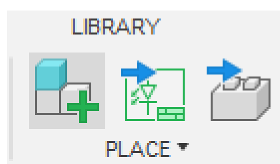

Figure 13 *- Place component button.*

10. On the Library list, click on **R-US_**. Click on the drop-down arrow and select the **0207/12** variant.

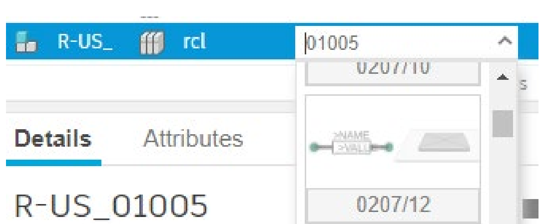

Figure 14 *- Adding a 0207/12 resistor to the schematic.*

11. Double-click to select then drag place the resistor on the workspace.

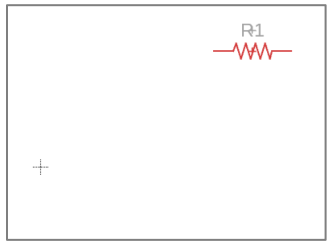

Figure 15 *- Placing a resistor to the schematic diagram.*

12. Next, place the LED into the circuit: **LED > LED5MM**.

    You can rotate the part by right-clicking before placement.

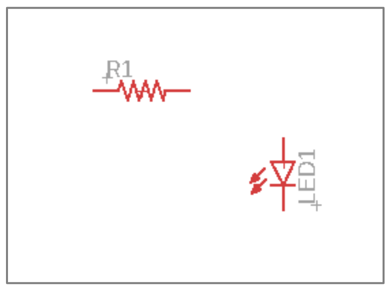

Figure 16 *- Schematic showing a resistor and a LED, unconnected.*

13. Now add a 2-position pin header to the schematic: **PINHD-1X2 > /90**.

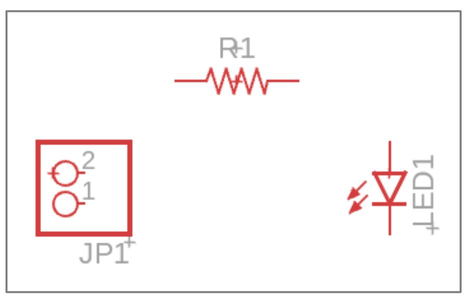

Figure 17 *- 2-position pin header is added to the schematic.*

Notice that the pins of JP1 are facing LED1.

14. Using the **Net** tool, connect the circuit parts together as shown in **Figure 18** below.

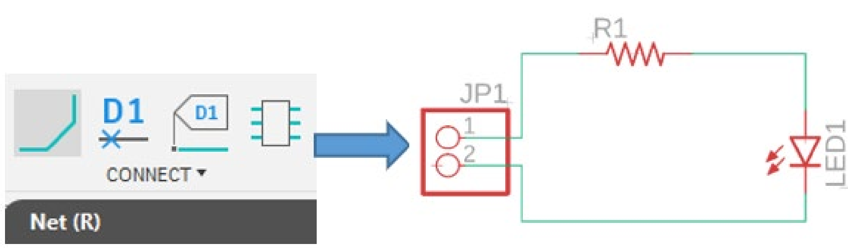

Figure 18 *- Select Net tool to connect the parts.*

15. Click on menu **VALIDATE** then click on the **ERC** button.

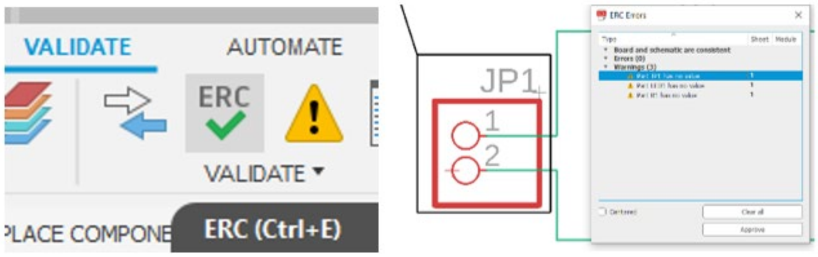

Figure 19 *- Click "ERC" to run the **E**lectrical **R**ules **C**heck.*

16. The ERC Errors window will pop up and highlight any errors or warnings, such as unconnected pins, accidental connections between power and ground, missing values, and others.

Review all errors and any warnings and fix them if necessary. For example, for any missing values, select the part and right-click it again to select its Value in the menu, then fill it in.

If no correction is needed on the error, click on "Approve" to remove it from the list.

When the schematic is complete and error-free, we can proceed to the board layout design.

Call your instructor for a signature: ____________________ **(1 2 3 4 5 6 Marks)**

Part 3: PCB Layout **6 Marks**

Our second step in PCB design is to convert the schematic information into a physical board layout. This task will cover the process of the PCB Layout.

1. Click the **Switch to PCB document** button to switch to the board editor.

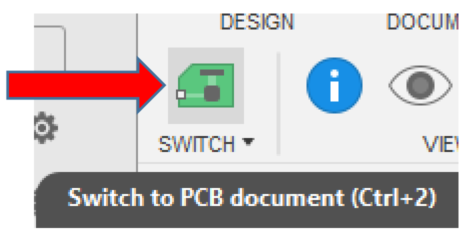

Figure 20 *- Find the **Switch to PCB document** button at the top of the screen.*

2. The board editor should open. Get familiar with locations of the main controls and their uses.

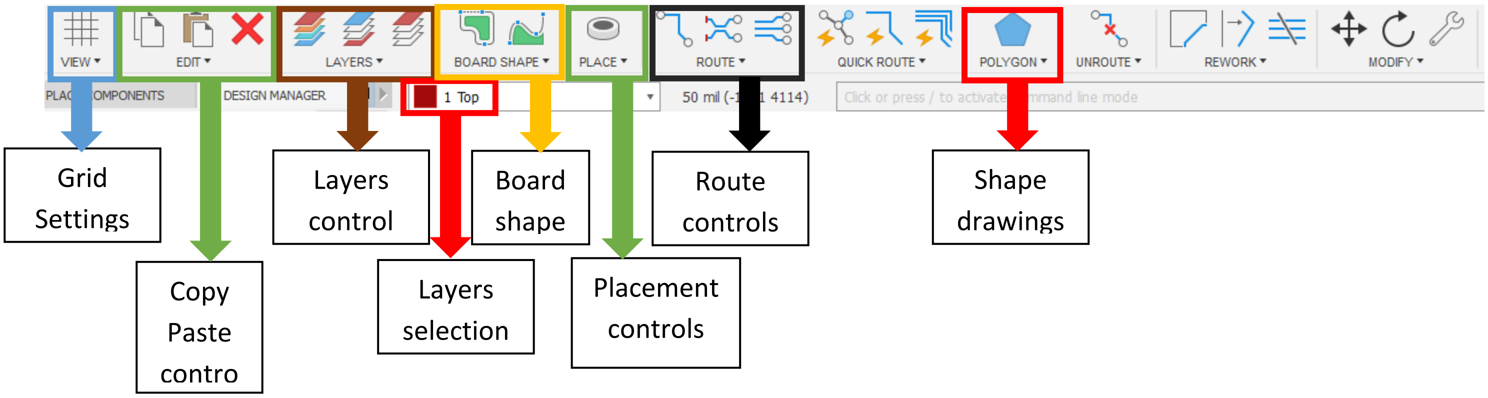

Figure 21 *- Board Editor window, side panel toolbar*.

We will follow each of these steps to complete the PCB Layout design:

a) Set **design rules**.

b) Set **board outline** dimensions.

c) **Place the parts** within the board dimensions in their desired locations,

d) Replace all schematic Nets with physical copper **Traces** on the board,

e) Add **text** to the silkscreen layer to indicate name, revision, etc.

<u>Set Design Rules</u>

3. Click on menu **RULES DRC/ERC** and select **DRC**.

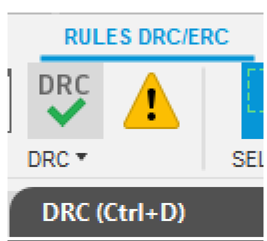

Figure 22 *- Select DRC tool.*

4. Click on tab **Clearance** and set the clearance as **Figure 23** below.

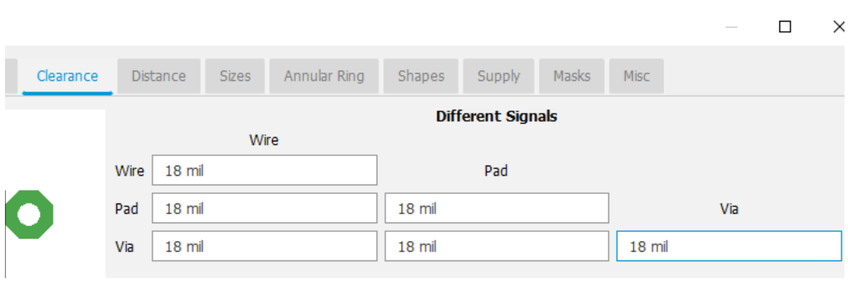

Figure 23 *- Clearance rules.*

5. Click on tab **Sizes** and set the **Minimum Width** and **Minimum Drill** as below.

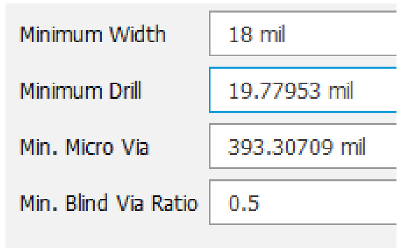

Figure 24 *- Sizes rules.*

6. Click on tab **Annular Ring** and set the **Min, Max** of **Pads** as below. Make sure the **Diameter** box is checked.

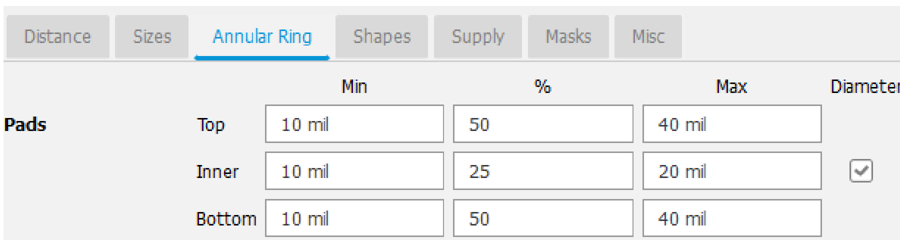

Figure 25 *- Annular ring rules.*

7. Click on tab **Supply** and set the **Thermal isolation** to **20** mil.

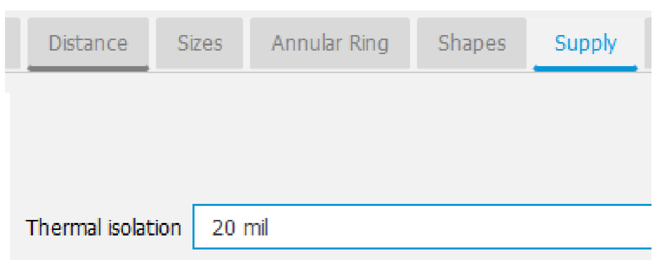

Figure 26 *- Supply rules.*

8. Click **Apply** then close the window.

<u>Board Outline</u>

9. Click on **Grid Settings** then change the unit to **inch**. Click **OK**.

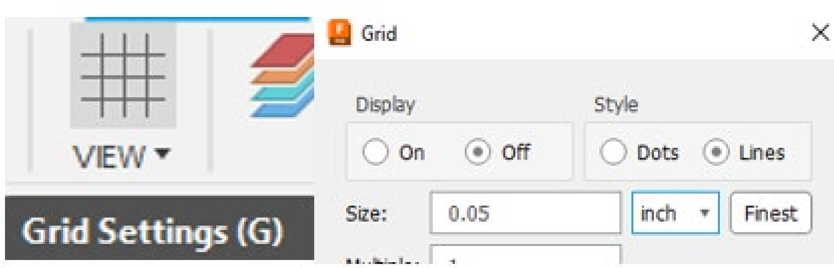

Figure 27 *- Change grid unit to inch.*

10. Set the active layer to the dimensions layer, as shown in **Figure 28**.

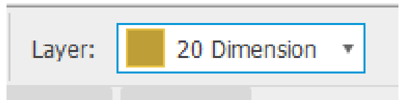

Figure 28 *- Dimensions layer selected.*

11. Select the **SELECT** tool.

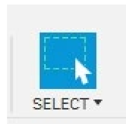

Figure 29 *- SELECT tool.*

12. Open the **INSPECTOR** windows.

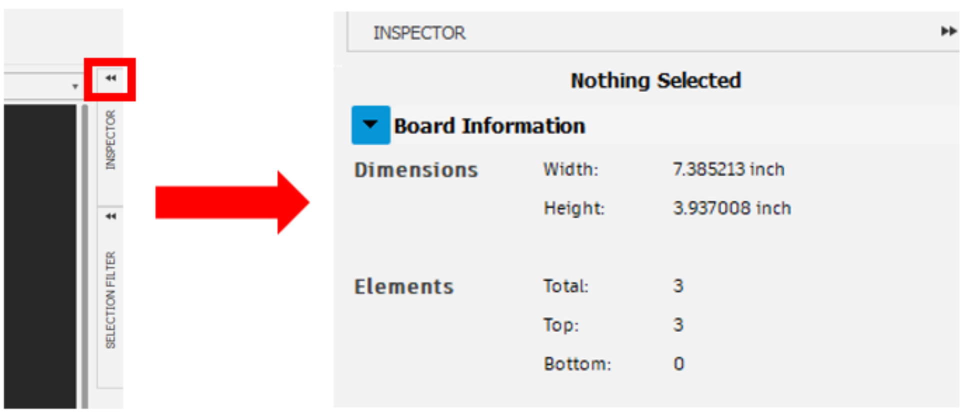

Figure 30 *- Opening the inspector windows.*

13. Click on the top edge of the board outline.

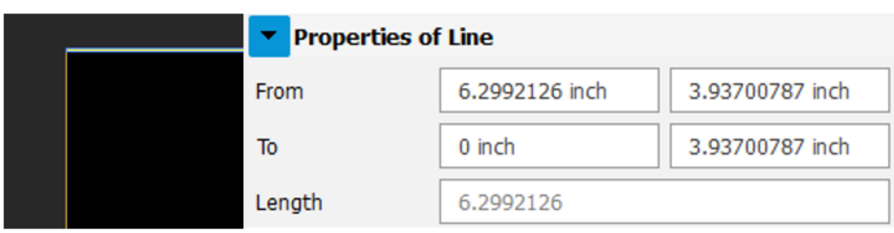

Figure 31 *- Top edges coordinates.*

14. Change the coordinate of this line as **Figure 32** below.

Figure 32 *- Set new coordinates.*

15. Select the bottom edge of the board outline.

Figure 33 *- Bottom edges coordinates*

16. Change the coordinate of this line as **Figure 34** below.

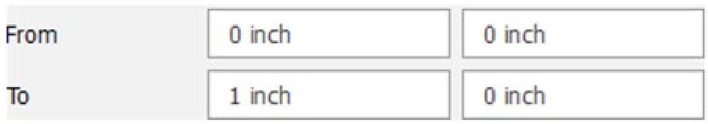

Figure 34 *- Set new coordinates for the bottom edge.*

17. The board outline size now set to 1” x 1”.

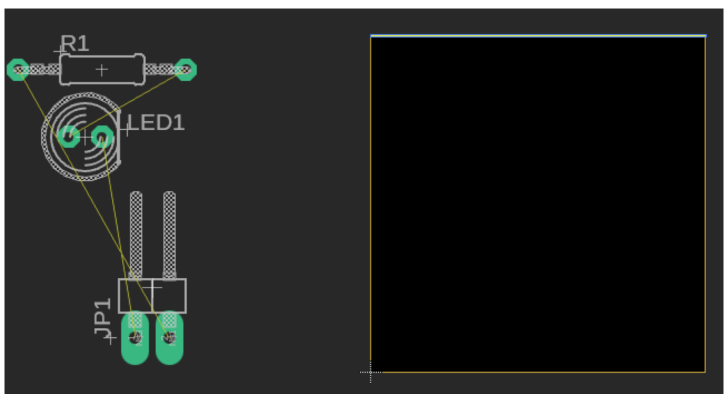

Figure 35 *- Resize board outline to 1" x 1".*

<u>Component Placement</u>

18. Ensure the **SELECT** tool is selected and move each part into the board outline area. When picking up parts, aim to click the yellow + in their centre. Rotate them while they are selected by right-clicking. Move the parts into place as shown in the following diagrams.

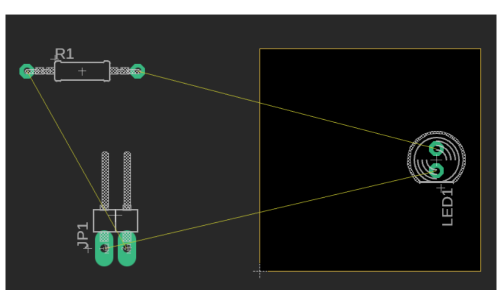

Figure 36 *- Move the LED and rotate it into place.*

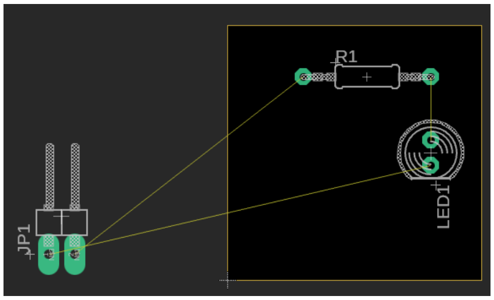

__Figure 37__ - Move the Resistor into place.

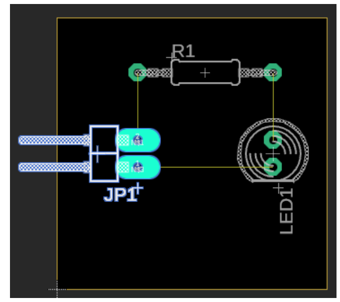

__Figure 38__ - Move the Connector into place.

<u>Traces</u>

Note: The “unrouted” yellow wires connecting the parts together indicate where the traces will go.

19. Select the Bottom layer as the active layer. Traces made on this layer will appear in blue.

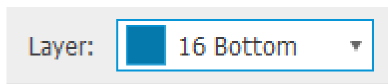

__Figure 39__ - Select the Bottom layer.

20. Click on the **Route Manual** button from the **Route Controls** section.

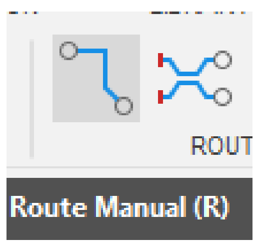

__Figure 40__ - Select the Route Manual button.

21. Set the trace width to **50** mils. This will control how wide the traces will be in the layout.

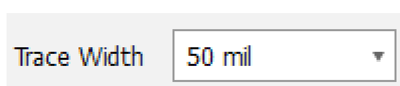

__Figure 41__ - Set trace width to 50 mils.

22. Click on the top pin of JP1 to start routing the trace. Complete the trace all the way to R1.

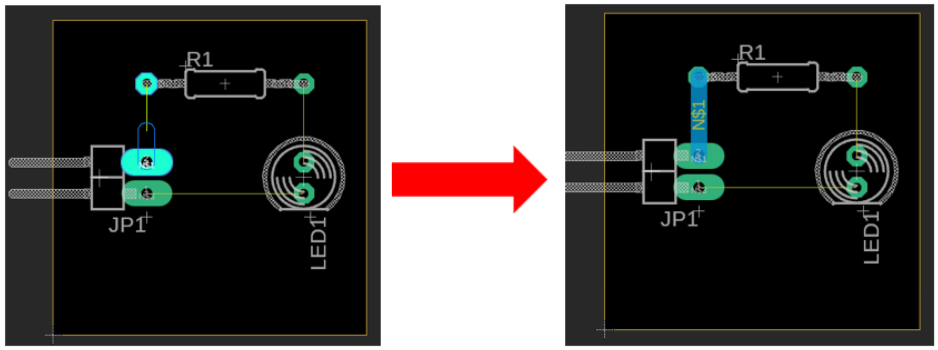

Figure 42 *- Complete the first trace between JP1 and R1, labelled N$1.*

23. Continue routing all traces so that there are zero remaining unrouted wires.

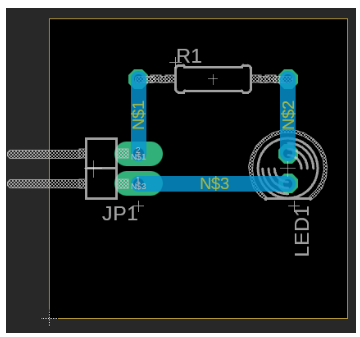

Figure 43 *- All components are connected.*

<u>Silkscreen Text</u>

24. To add text to the silkscreen layer, change the layer to **NamesTop**.

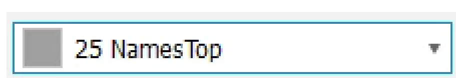

Figure 44 *- Change the layer to* ***NamesTop****.*

25. Click on menu **DOCUMENT** then the drop down at **DRAW** section. Select **Text** tool.

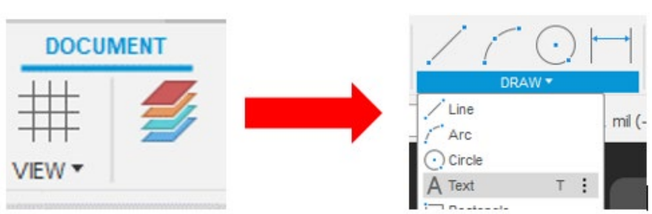

Figure 45 *- Selecting Text tool.*

26. Write your name in the text then place it in the PCB.

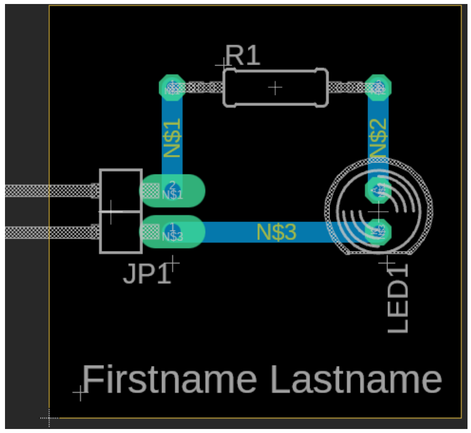

Figure 46 *- Finished board with all parts and text in place.*

Call your instructor for a signature: ___________________ (1 2 3 4 Marks)

27. Run the DRC process by go to menu **RULES DRC/ERC > DRC**. Then click **Check**.

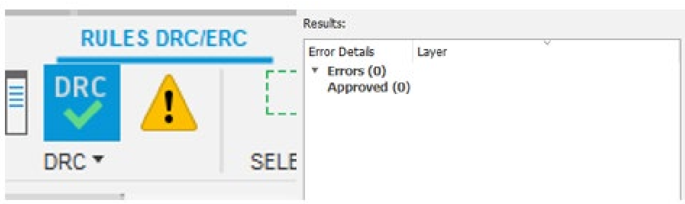

Figure 47 *- DRC result of the current design.*

The DRC Errors window will pop up and highlight any errors or warnings, such as unconnected pins, accidental connections between power and ground, missing values, and others.

Review all errors and any warnings and fix them if necessary.

If no correction is needed on the error, click on “Approve” to remove it from the list.

When the schematic is complete and error-free, we can proceed to the Gerber Generation.

Call your instructor for a signature: ___________________ (1 2 Marks)

Part 4: Gerber Generation & File Submission 3 Marks

The final stage of PCB design is to create a set of manufacturing files that the machine can use to physically create the board. Each layer of the PCB design will be exported into its own Gerber file. This task covers the steps to export all Gerber files.

1. On the menu bar, click **MANUFACTURING > CAM Processor**.

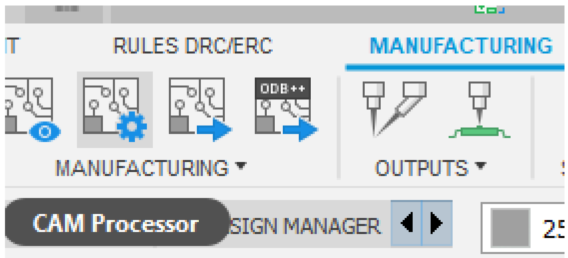

Figure 48 *- Opening CAM Processor.*

2. A list of possible output files shown as below.

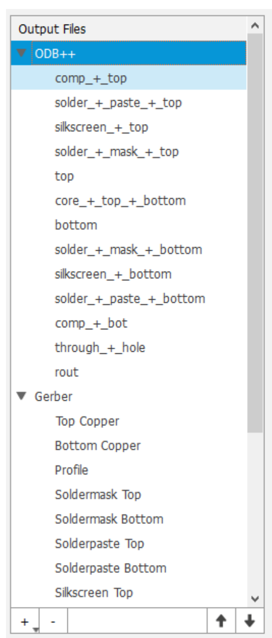

Figure 49 *- Possible output CAM files.*

3. At the very minimum, you must have the following Gerbers to manufacture a board:

- Board Outline Layer
- Drill Files (Plated and non-plated)
- Bottom-side Copper Layer
- Top-side Silkscreen Layer

eg. If you have a double-sided board, you will also need to include the Top-side Copper Layer.

4. Right click on **ODB++** directory and select **Delete all**.

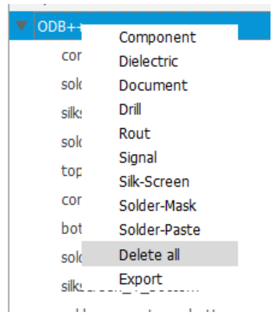

Figure 50 *- Deleting ODB++ directory.*

5. Delete the **Assembly** directory.

6. Click on the **Gerber** directory. Then change the **Gerber prefix** to **CAMOutputs**. Uncheck the **Include job file in the gerber export**.

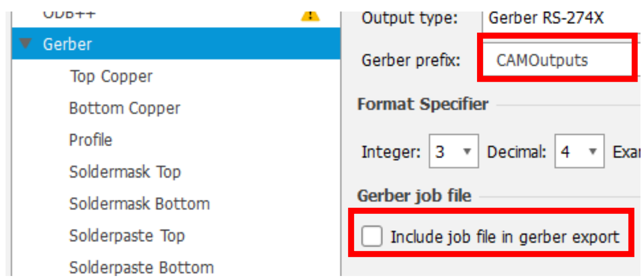

Figure 51 *- Change the location for Gerber outputs.*

7. Click **OK** and select the location to have all the Gerber files exported to.

8. Under the **Gerber** directory, except for **Bottom Copper, Profile and Silkscreen Top**, delete all other files.

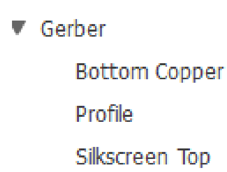

Figure 52 *- Necessary Gerber outputs.*

9. Click on the **Drill** directory and change the **Drill prefix** to **CAMOutputs**.

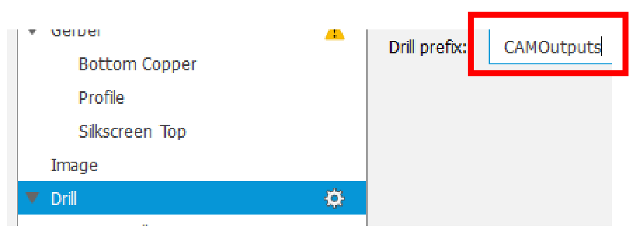

Figure 53 *- Change location for Drill files.*

10. Click Process Job. Select a location and click save.

11. Open the schematic then export it by going to menu **Files > Export**

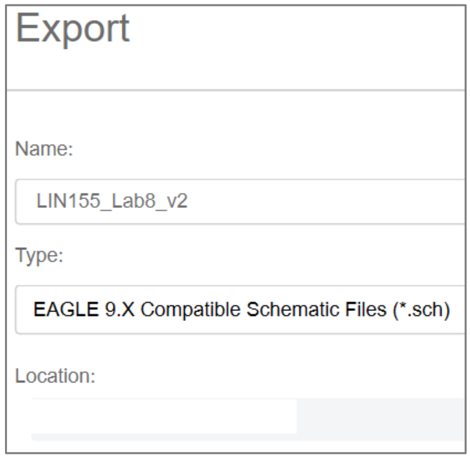

Figure 54 *- Exporting the schematic.*

Name: LIN155_Lab8

Type: EAGLE 9.x Compatible Schematic Files (*.sch)

Location: your storage drives.

12. Open the board layout then export it.

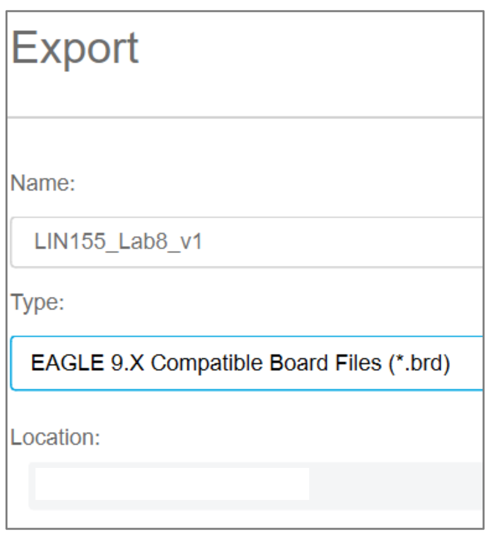

Figure 55 *- Exporting the board layout.*

13. Using a program like File Explorer, navigate to your Gerbers folder. Put all Gerber files, plus the original *.SCH and *.BRD files, into a **7zip archive**. Name the file in the following format:

LIN155NxxL_LAB8_LASTNAME_FIRSTNAME .7z

Your first and last name and lab section should be part of the 7zip archive name.

14. Upload the .7z file containing the **schematic** and **PCB** files, and **all Gerbers**, to Blackboard as per your professor's instructions.

Your instructor can check your .7zip archive on your screen to confirm you have included all the files, but no signature is required for this section.

No signature. You must upload your .7z file to get full marks for the lab. **(1 2 3 Marks)**

Total Marks: **/15** Comments: ______________________________

____

Lab 8 Checklist:

At the conclusion of this week's lab, you should be comfortable explaining the following items:

- Explain how the FUSION applications work together to produce Schematics & PCB designs.
- How do the schematic and board editors work, and how are they different from each other.
- Justify your selection of components from the parts library based on their size and function.
- Understand the software rules for inserting, rotating, and modifying components.
- Explain the importance, and differences between, the DRC and ERC.
- Explain the purpose of the board outline and how to create one.
- Understand how the clearances and design rules affect the outcome of the PCB design.
- Explain how to create the Gerber files for manufacturing, and what data they each contain.
- Practice sending files to a 7zip archive, which is the file format our PCB Lab accepts by email.