

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 the [Laboratory Code of Behaviour](https://tinyurl.com/SITAS-elec-labs) (https://tinyurl.com/SITAS-elec-labs)
- Lab paper must be handed in to the instructor 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.

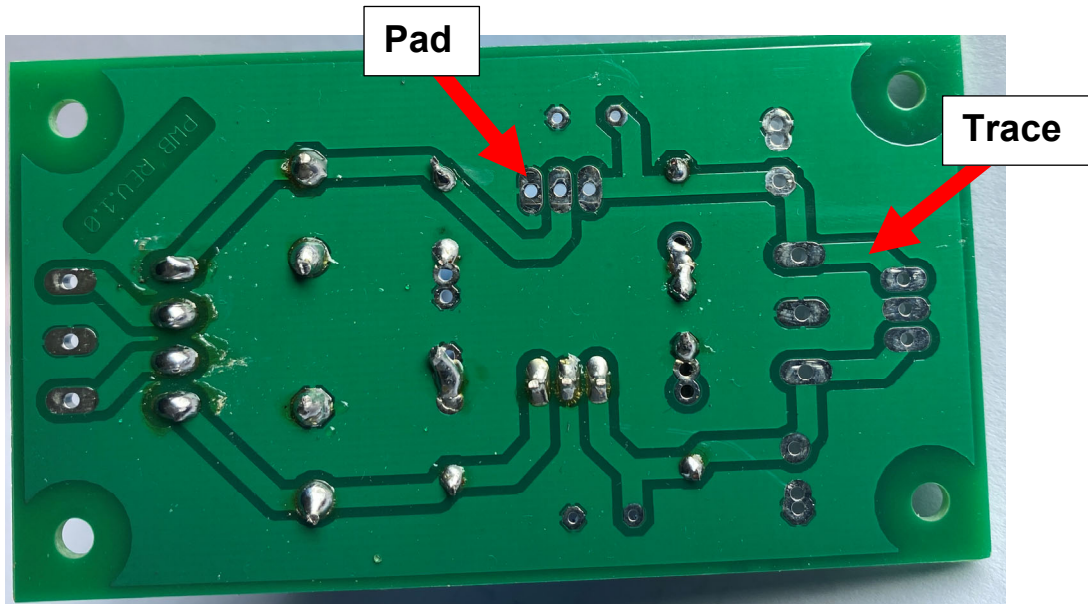


Figure 1 - Soldered side of a printed circuit board (PCB), noting pads and traces.

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!

If you are using this reference as a guide: Pay particular attention to the instructions in Step 7, which outlines 3 cases to make your design conform to the restrictions required by the PCB Lab.

A **SEMET.Ibr** file is now available to students that supports the first 3 semesters in electronics.

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 AD FUSION from MyApps. Login to your Autodesk account.
2. On the **Data Panel**, click **New Project**.

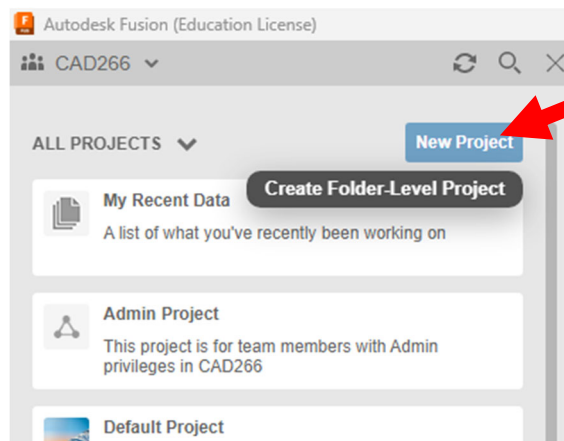


Figure 2 - Create a new project.

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**.

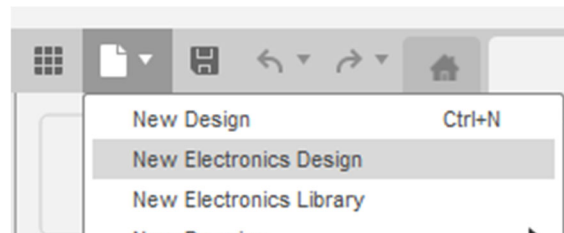


Figure 3 - Create a new electronics design.

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

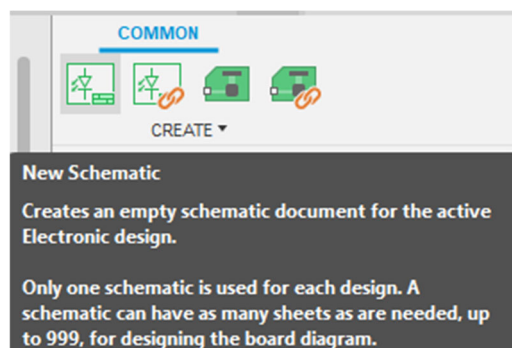


Figure 4 - Create a new schematic.

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.

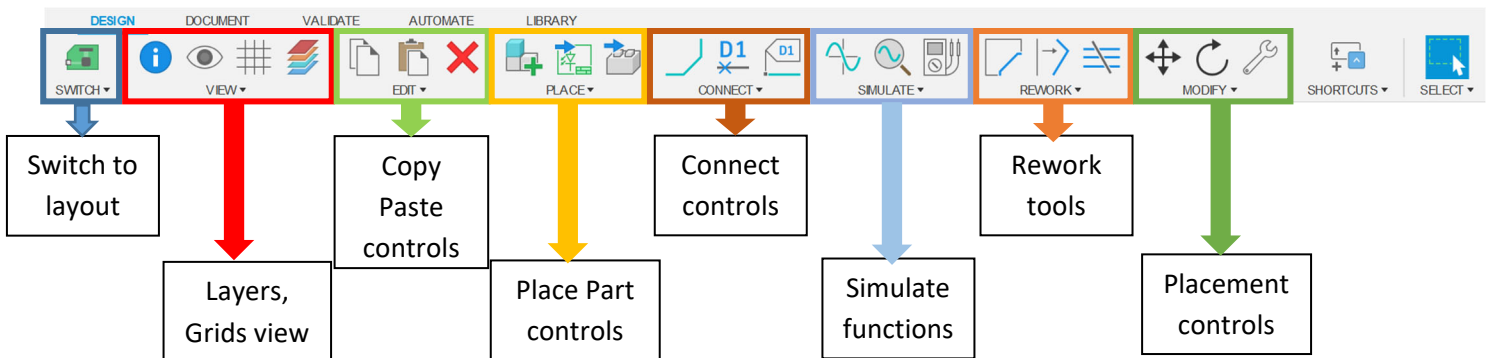


Figure 6 - Fusion schematic tool bar.

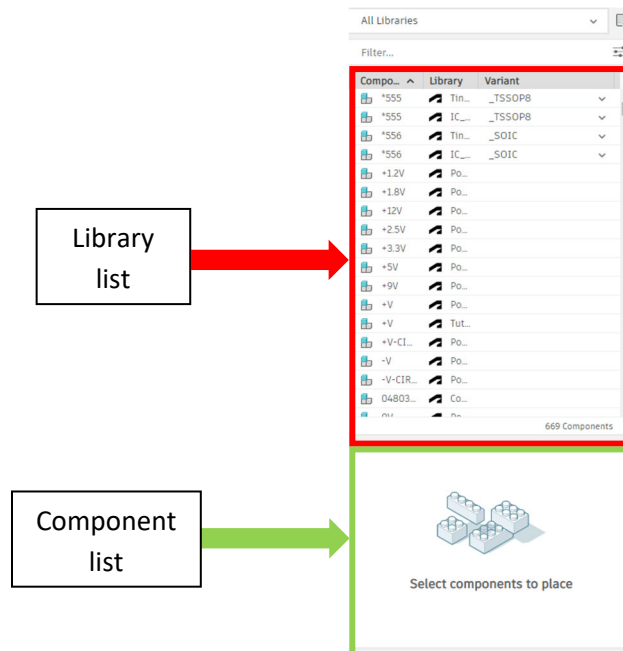


Figure 7 - Library manager.

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

- Add Parts** to the design by searching and selecting each one from the library,
- Connect** the parts using the Net tool, to indicate where the traces will go,
- Run the ERC** or Electrical Rules Check to check for errors and fix them.

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

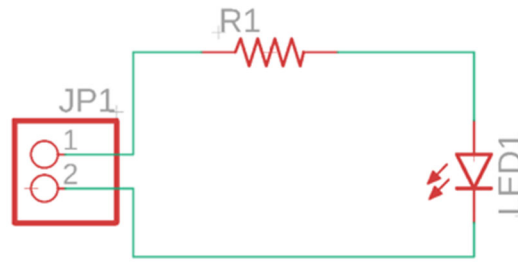


Figure 8 - LED circuit.

The schematic consists of:

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

8. Obtain the **SEMET.lbr** library from LIN155.CORE (SEMET site) and save to local computer.

9. Add SEMET.lbr library to FUSION.

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

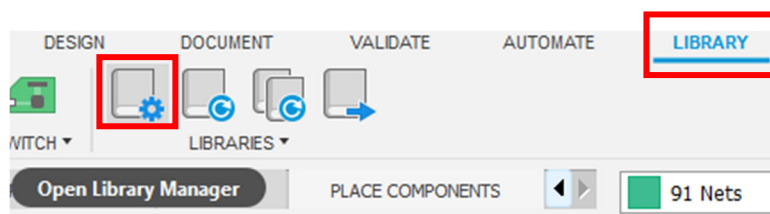


Figure 9 - Opening Library Manager.

- Once the **Library Manager** open, click on **Import Library**. Then select **Import from local disk**.

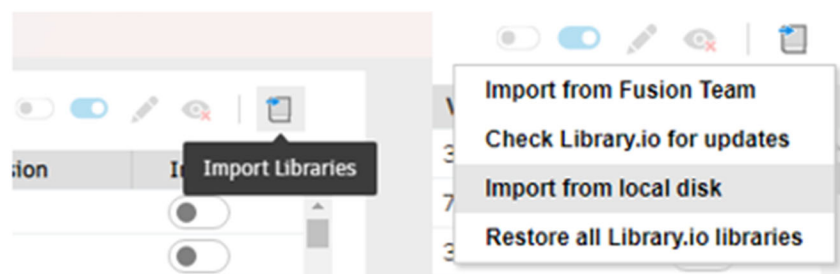


Figure 10 - Import libraries from local disk.

- Select **SEMET.lbr** then click **Open**.
- Once the library open, a list of components should be seen as the figure below.

Component ^	Library	Variant
555	SEMET	DIP
7400	SEMET	N
7402	SEMET	N
7404	SEMET	N
7408	SEMET	N
7432	SEMET	N
7486	SEMET	N
74LS47	SEMET	N
78XX	SEMET	
7SEG-CA	SEMET	
7SEG-CK	SEMET	
BJT_NPN	SEMET	2N3904
BJT_PNP	SEMET	2N3906
CAP-CER	SEMET	CAP-2...
CAP-POL	SEMET	E2,5-6E
DB9CON	SEMET	"
DIODE	SEMET	DO35-7
DISC...	SEMET	

Figure 11 - List of available components.

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

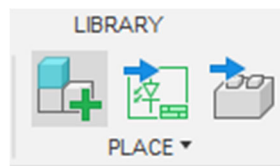


Figure 12 - Place component button.

11. To add a resistor:

- In the Filter... box, type **res**.

Component ^	Library	Variant
74LS47	SEMET	N
RESISTOR	SEMET	AXIAL-7...

Figure 13 – Searching for a resistor.

- Click on  symbol of the **RESISTOR** line and choose the **11.7mm** variant.



Figure 14 - Select 11.7 mm resistor variant.

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

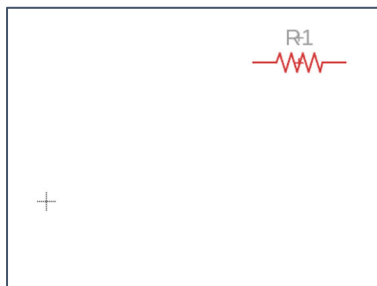


Figure 15 - Placing a resistor to the schematic diagram.

12. Next, search for **LED** and place a **LED_5MM**.

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

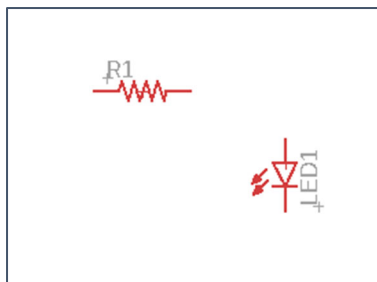


Figure 16 - Schematic showing a resistor and a LED, unconnected.

13. Now add a 2-position pin header to the schematic: search for **PIN** then add a **PINHD-1X2**



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

Notice that the pins of JP1 are facing LED1. Either **rotate** or **mirror** JP1.

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

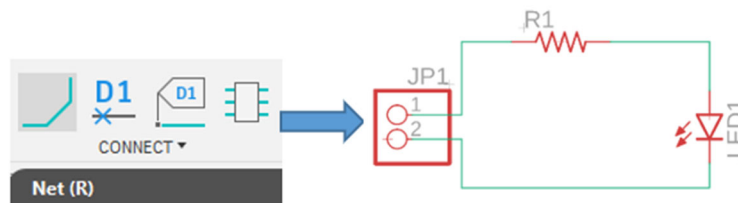


Figure 18 - Select Net tool to connect the parts.

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

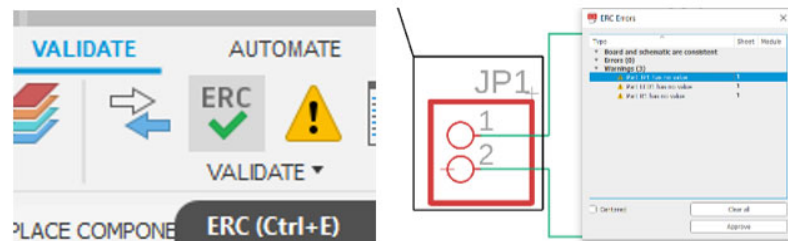


Figure 19 - Click "ERC" to run the Electrical Rules Check.

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

Review all errors and **warnings**. **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 with the board layout design.

Save your work.

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.

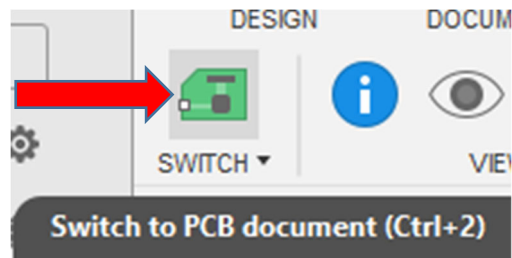
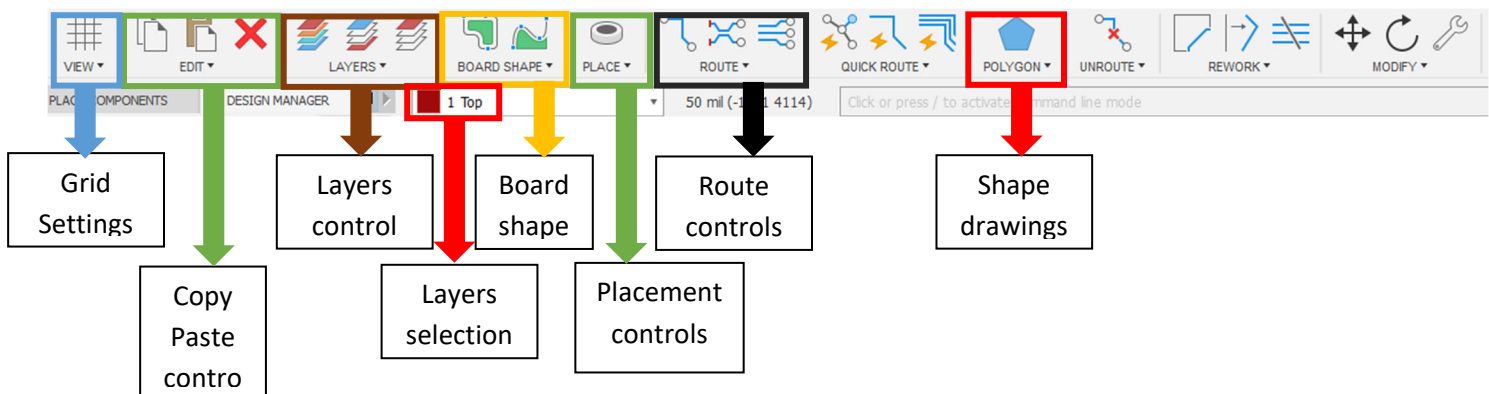


Figure 20 - Find the Switch to PCB document button at the top of the screen.

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.

Set Design Rules

3. Click on menu **RULES** and select **Design Rules**.

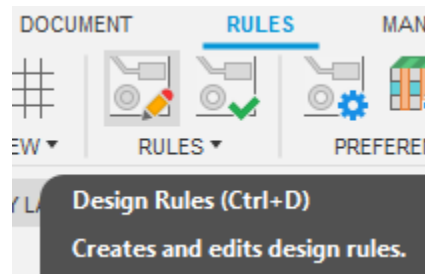


Figure 22 - Select Design Rules tool.

4. Set the signal clearance to 14 mil as **Figure 23** below.

Wire - Wire Clearance	14 mil
Wire - Pad Clearance	14 mil
Wire - Via Clearance	14 mil
Pad - Pad Clearance	14 mil
Pad - Via Clearance	14 mil
Via - Via Clearance	14 mil
SMD - Pad Clearance	14 mil
SMD - Via Clearance	14 mil
SMD - SMD Clearance	14 mil

Figure 23 - Clearance rules.

5. Set the **Minimum Width** and **Minimum Drill** as below. Then click **OK**.

Copper Width	18 mil
Drill Size	1mm

Figure 24 - Sizes rules.

Note: For Drill Size, enter “1 mm”. It will automatically convert to 39.37 mil.

6. Navigate to menu **RULES**, then select **Design Preferences**.

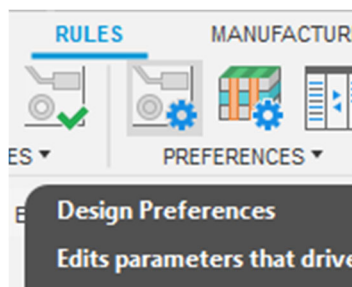


Figure 25 – Design Preferences.

7. Select tab **Annular Ring**. **IMPORTANT!** This step is essential in getting PCBs printed by the SITAS in-house PCB lab. Please follow the instructions carefully. If you are:

- a. **First semester** students and doing LIN155 Lab 8, you will follow these directions but won't need to find or add any additional parts from other libraries for today's lab.
- b. **Second or higher semester** students and using this document as a PCB guideline, read through this section fully and edit any footprints that do not meet the DRC.

NOTES FOR ALL STUDENTS:

- Sometimes you can find all your parts in one library, like the SEMET.lbr file. In future, you may need to add parts from additional libraries into the design. For example: Figure 26 below shows the 7400N part from SEMET.lbr as well as an unchanged footprint of RB-1215D inserted from another library.
- The drill holes and pad shapes and sizes for all the parts in the SEMET.lbr have been pre-verified. Parts from any other library will need you to edit them to pass the DRC.

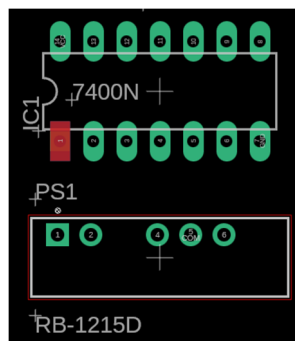


Figure 26 - Example of using SEMET.lbr with additional libraries.

- You cannot modify the footprint in a library that is not your own version.
- If you need to use parts from other libraries, create your own New Library first, save the footprint into that library, make the modifications to the footprint, and save changes.
- Good practice: Start your own library so it is set up for future classes that require PCBs.

i. Begin by verifying the Annular Ring values are set as shown in Table 1 below.

	Min	%	Max
Top	10 mil	25	20 mil
Inner	10 mil	25	20 mil
Bottom	10 mil	25	20 mil

Table 1 - Annular Ring default values.

- ii. **Verify** all the drill holes are at least 39.37 mils (or 1 mm). The ones that are too small will look like those highlighted in the example figure below.
- R2 drill holes are smaller than 39.37 mils => highlighted by yellow lines
 - IC2 drill holes are greater than or equal to 39.37 mils.

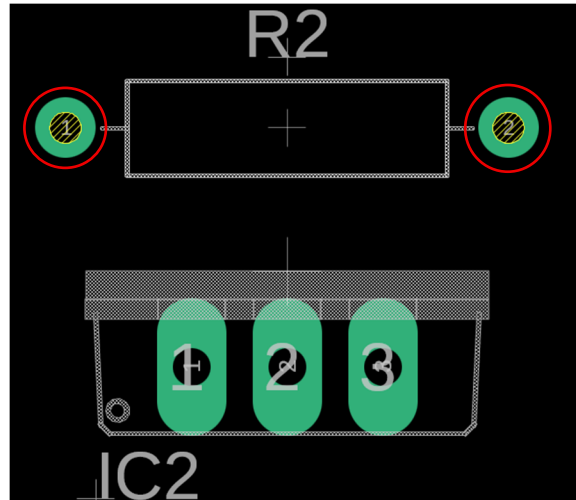


Figure 27 – Under-sized drill holes are highlighted.

- iii. When the footprint drill hole sizes require editing, follow these steps:
- a. Right click on the component with the highlighted drill holes then select **Edit Footprint**. Click **YES** when prompted.
 - b. Select all the pads then open the INSPECTOR window.
 - c. Change the Drill size and Diameter as below.

Diameter	70 mil
Slot Length	0 mil
Drill	39.37008 mil

Figure 28 - Set Diameter and Drill size for pads.

- d. Save the library. This step will save a copy of the original library to Fusion Hub.
- e. Navigate back to the PCB. Right click on the same component again then select Swap Library.
- f. Verify the Used Library is the Library.io version, and the Target Library is the Fusion Hub version. Then click SWAP.

- g. Verify the component has been swapped (yellow line highlight disappeared).

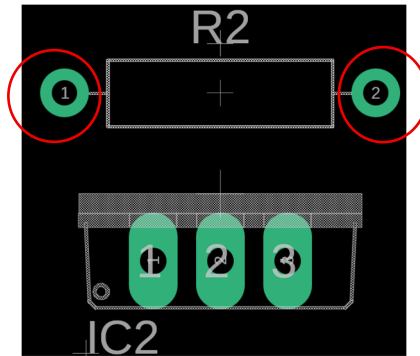


Figure 29 - Component swapped.

- h. Repeat from steps a) to g) for all other components, if needed.
- iv. **Verify** that all pads for all parts in the layout have an oblong shape. Looking at Figure 35, this means they look like the shape for IC2, not round like for R2.
- v. When the footprint pad shapes require editing, follow these steps:
 - a. Right click on the part with the incorrect pad shapes then select **Edit Footprint**. Click **YES** when prompted.

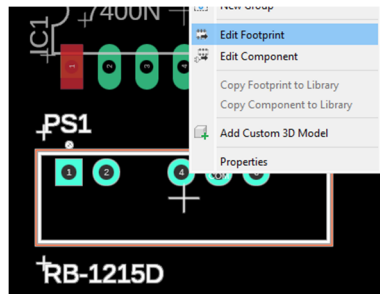


Figure 30 - Select Edit Footprint.

- b. Select all the pads then open the **INSPECTOR** window.
- c. Change the **Shape** to **long** and **Angle** to **90** degrees.

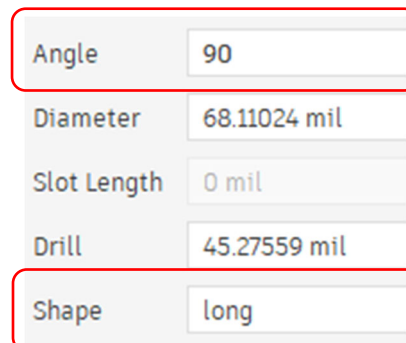


Figure 31 - Change the angle and shape of the pads.

- d. All the pads should change to the shape shown below.

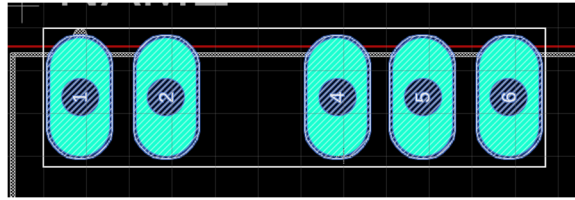


Figure 32 - Pads shape and angle changed.

- e. Save the library. This step will save a copy of this library to Fusion Hub.
 f. Right click again on the part you modified then select **Swap Library**.
 g. Verify the Used Library is the local disk version, and the Target Library is the Fusion Hub version. Then click SWAP.

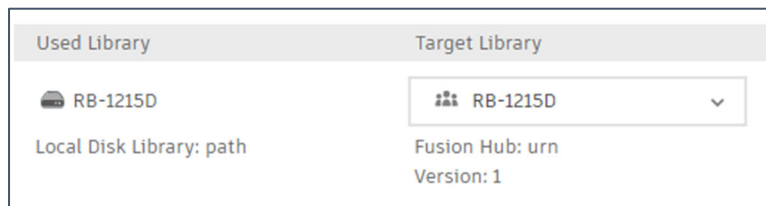


Figure 33 - Verify the used and target library before swap.

- h. Verify the part on the PCB has swapped to the new version. As shown in Figure 31 below, the drill hole sizes and pad shapes are now uniform.

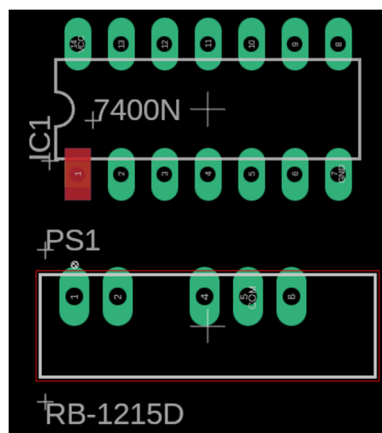


Figure 34 - Library swapped.

- i. Repeat from steps a) to h) for all other components, if needed.

8. Click on tab **Supply** and set the **Thermal isolation** to **20 mil**. Then click **OK**.

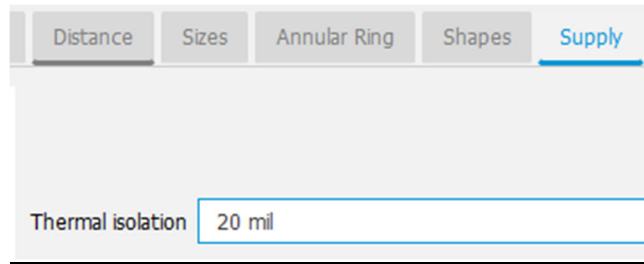


Figure 35 - Supply rules.

Board Outline

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

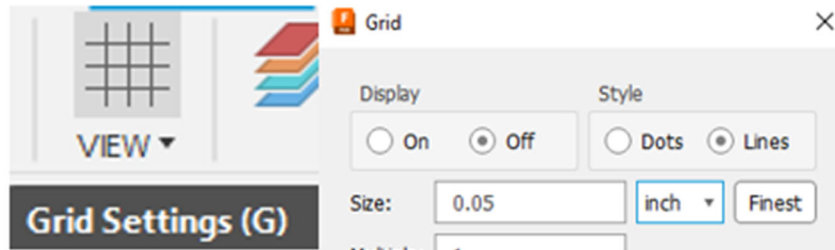


Figure 36 - Change grid unit to inch.

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

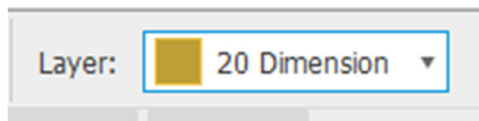


Figure 37 - Dimensions layer selected.

11. Select the **SELECT** tool.

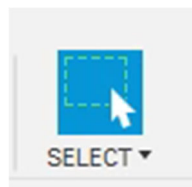


Figure 38 - SELECT tool.

12. Open the INSPECTOR window.

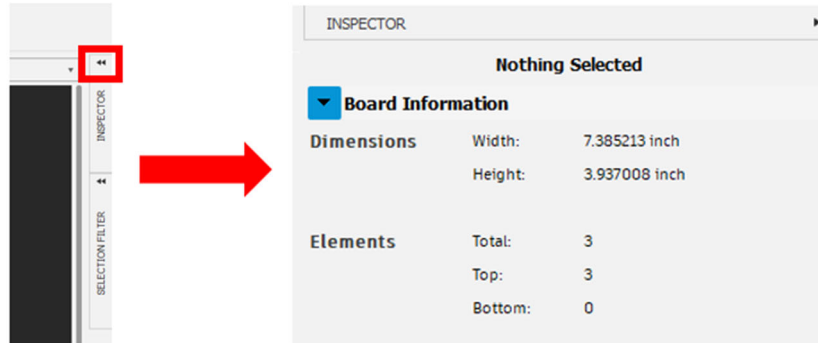


Figure 39 - Opening the inspector windows.

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

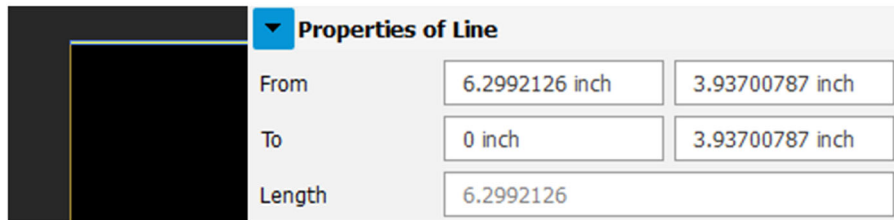


Figure 40 - Top edges coordinates.

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

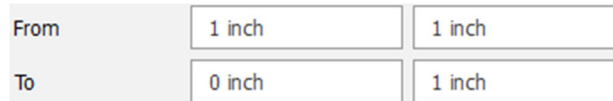


Figure 41 - Set new coordinates.

15. Select the bottom edge of the board outline.

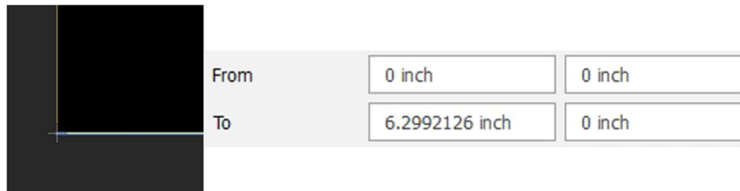


Figure 42 - Bottom edges coordinates

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



Figure 43 - Set new coordinates for the bottom edge.

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

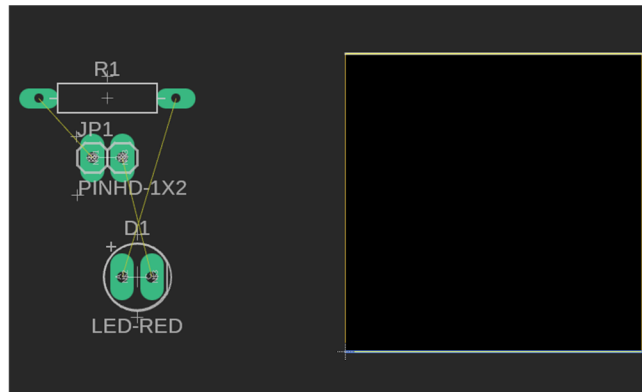


Figure 44 - Resize board outline to 1" x 1".

Component Placement

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.

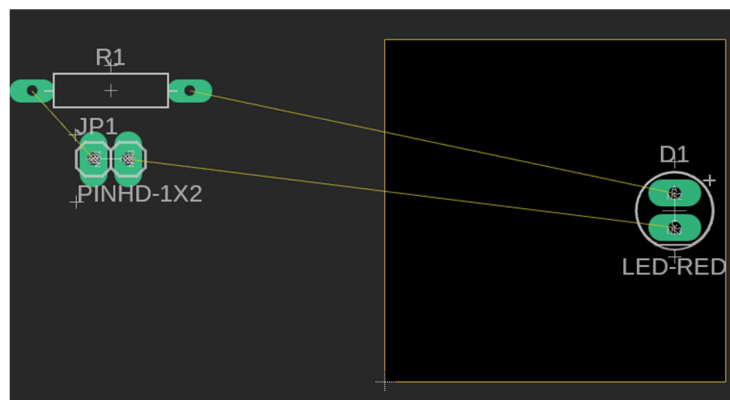


Figure 45 - Move the LED and rotate it into place.

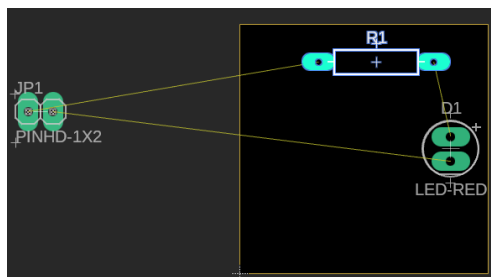


Figure 46 - Move the Resistor into place.

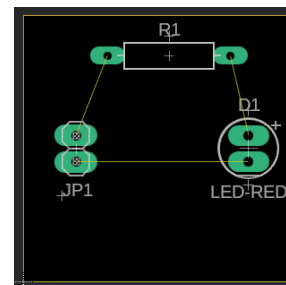


Figure 47 - Move the Connector into place.

Traces

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 48 - Select the Bottom layer.

20. In menu **DESIGN**, click on the **Route Manual** button from the **Route Controls** section.

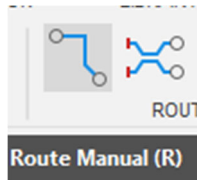


Figure 49 - Select the Route Manual button.

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

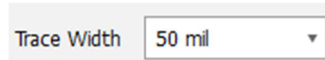


Figure 50 - Set trace width to 50 mils.

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

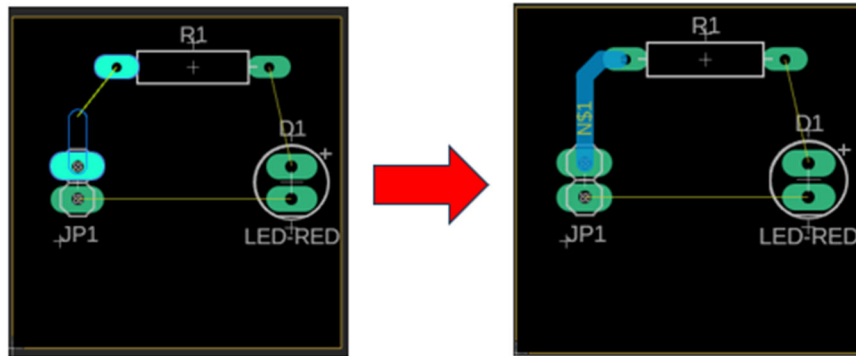


Figure 51 - Complete the first trace between JP1 and R1, labelled N\$1.

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

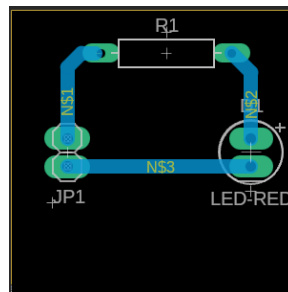


Figure 52 - All components are connected.

Silkscreen Text

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

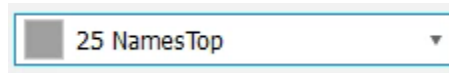


Figure 53 - Change the layer to NamesTop.

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



Figure 54 - Selecting Text tool.

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

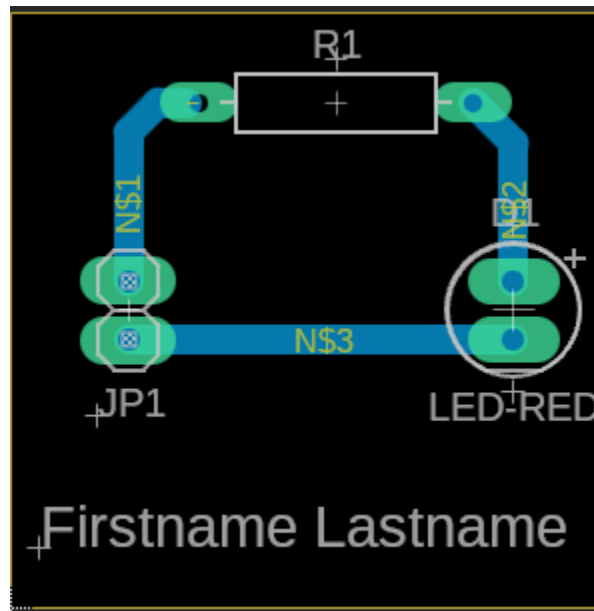


Figure 55 - Finished board with all parts and text in place.

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

27. Run the DRC process by go to menu **RULES**. Then click **Check Design Rules**.

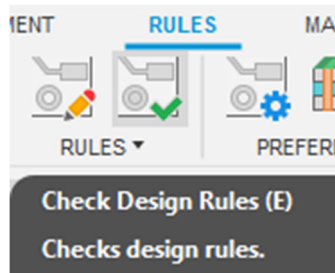


Figure 56 - DRC result of the current design.

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.

Save your work.

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**.

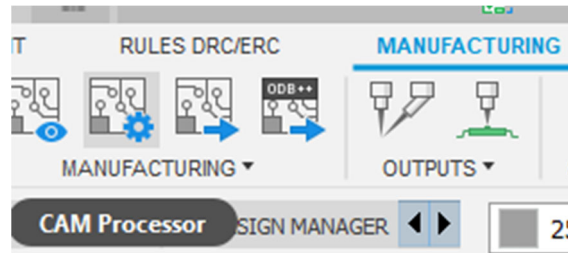


Figure 57 - Opening CAM Processor.

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

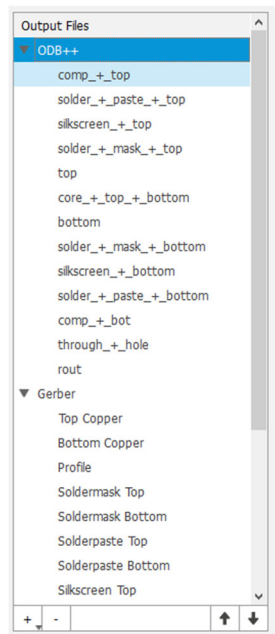


Figure 58 - Possible output CAM files.

3. A 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.

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

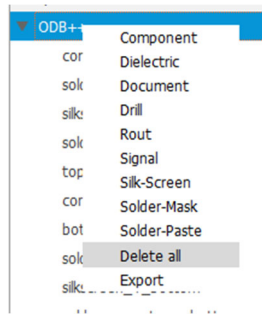


Figure 59 - Deleting ODB++ directory.

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

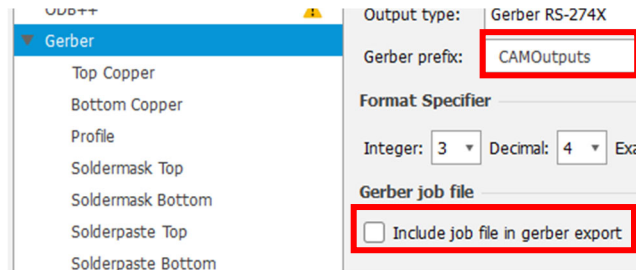


Figure 60 - Change the location for Gerber outputs.

- Under the **Gerber** directory, except for **Bottom Copper, Profile and Silkscreen Top**, delete all other files. (For future double-sided boards, you may need to keep **Top Copper** as well.)



Figure 61 - Necessary Gerber outputs.

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

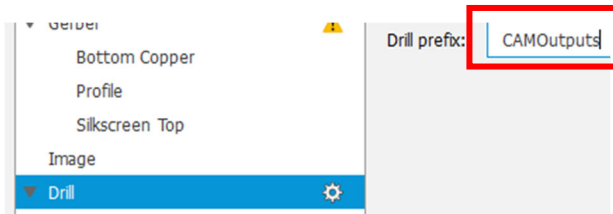


Figure 62 - Change location for Drill files.

- Click **Process Job**. Select a location and click save.

10. Export the board layout by selecting **Files > Export**.

The screenshot shows a dialog box titled "Export". It has three main sections: "Name:", "Type:", and "Location:". The "Name:" field contains the text "LIN155_Lab8_v1". The "Type:" dropdown menu is open, showing "EAGLE 9.X Compatible Board Files (*.brd)" selected. The "Location:" field is currently empty.

Figure 63 - Exporting the board layout.

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

The screenshot shows a dialog box titled "Export". It has three main sections: "Name:", "Type:", and "Location:". The "Name:" field contains the text "LIN155_Lab8_v2". The "Type:" dropdown menu is open, showing "EAGLE 9.X Compatible Schematic Files (*.sch)" selected. The "Location:" field is currently empty.

Figure 64 - Exporting the schematic.

Name: LIN155_Lab8

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

Locations: your storage drives.

12. 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.

13. 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 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.