

Build Schematics with EAGLE

Qi Luo

11/13/2015

When our team is working on the dryer sensor project, we need build circuits and schematics for our transmitter unit, receive unite and voltage regulator. I have finally came up with build them in EAGLE. EAGLE, the **E**asy **A**pplicable **G**raphical **L**ayout **E**ditor is a powerful PCB design software tailored to meet the needs of professional engineers.

Easy to use. Custom circuits parts. Many useful feature.

1. Starting EAGLE

Since EAGLE is launched in DESC desktop, it opens several built-in directories and allows users to define a project folder within default directories. However, I want to save in my personal folder,I will want to create a project directory within your personal or class network space.

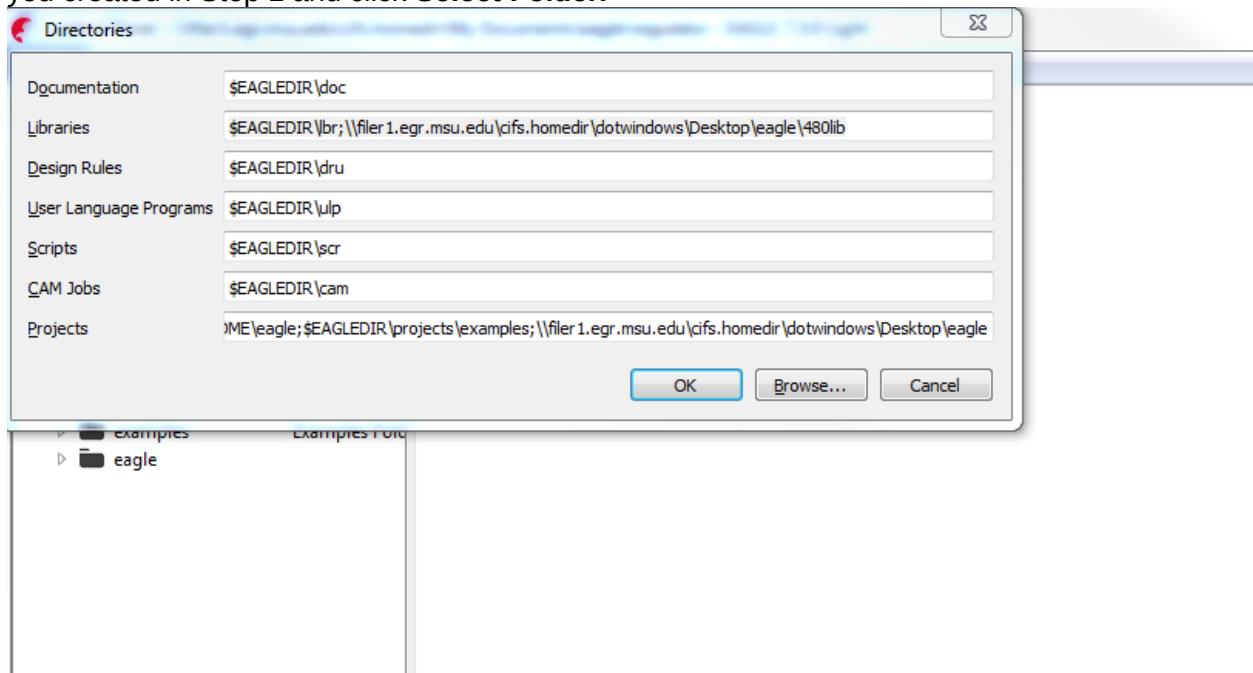
1. Launch the Eagle software in Windows 7, by search in start menu:

Start >> search EAGLE >>EAGLE 7.3.0

When launched, EAGLE will automatically create a project directory for PCB schematics and layouts.

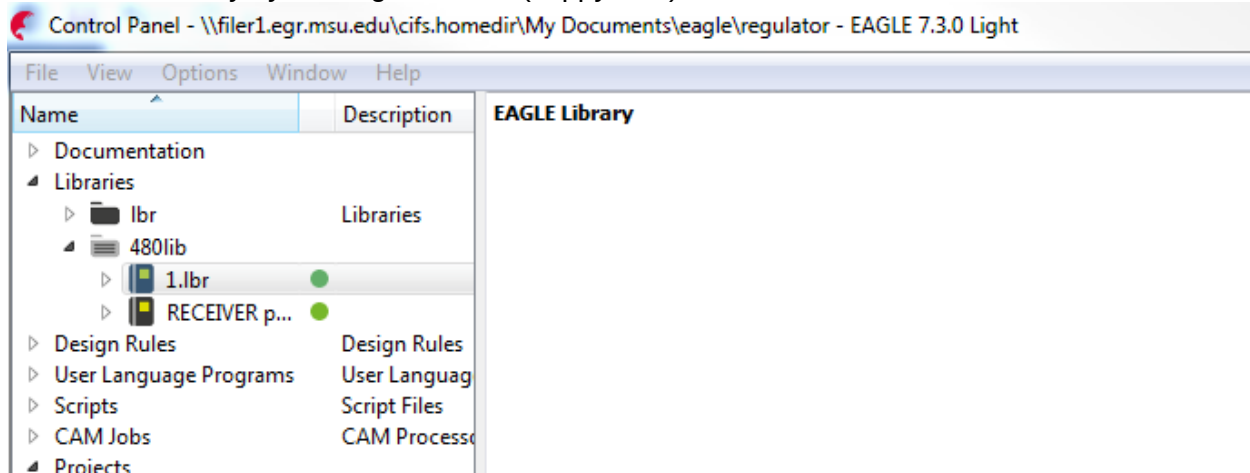
2. In the EAGLE **Control Panel**, go to **Options >> Directories...** to set your project directory.

5. Click on the dialog box next to **Projects** and select **Browse...**. Navigate to the “PCB” folder that you created in Step 1 and click **Select Folder**.



2. Creating a New Project and Project Library

1. In the EAGLE **Control Panel**, double-click on **Projects**
2. Right-click the “Projects >> PCB” folder and select **New Project**. Name the new project folder “Regulator”.
3. Select **New >> Library**. This will open an empty Schematic window where you can create a symbol for a new library component.
A **library** contains the schematic *Symbols* and *Package-Footprints* for components that can be utilized to design a PCB. You may want to create custom component for your project.
4. Save the library by clicking the **Save** (floppy disk) icon on the toolbar



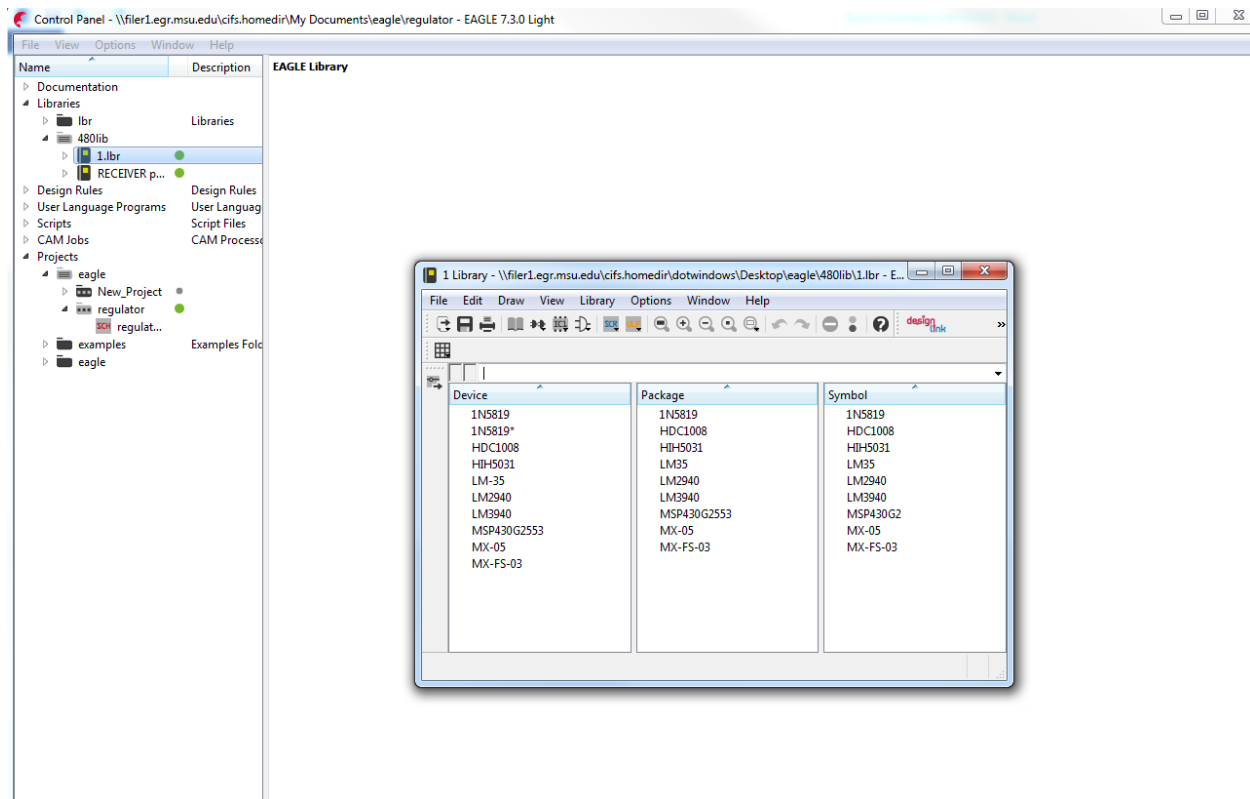
3. Designing a New Component

To demonstrate how to construct a new device for any component that is not found in EAGLE's build-in libraries, a project library containing a simple Red-Green LED will be created

A. Component Symbol

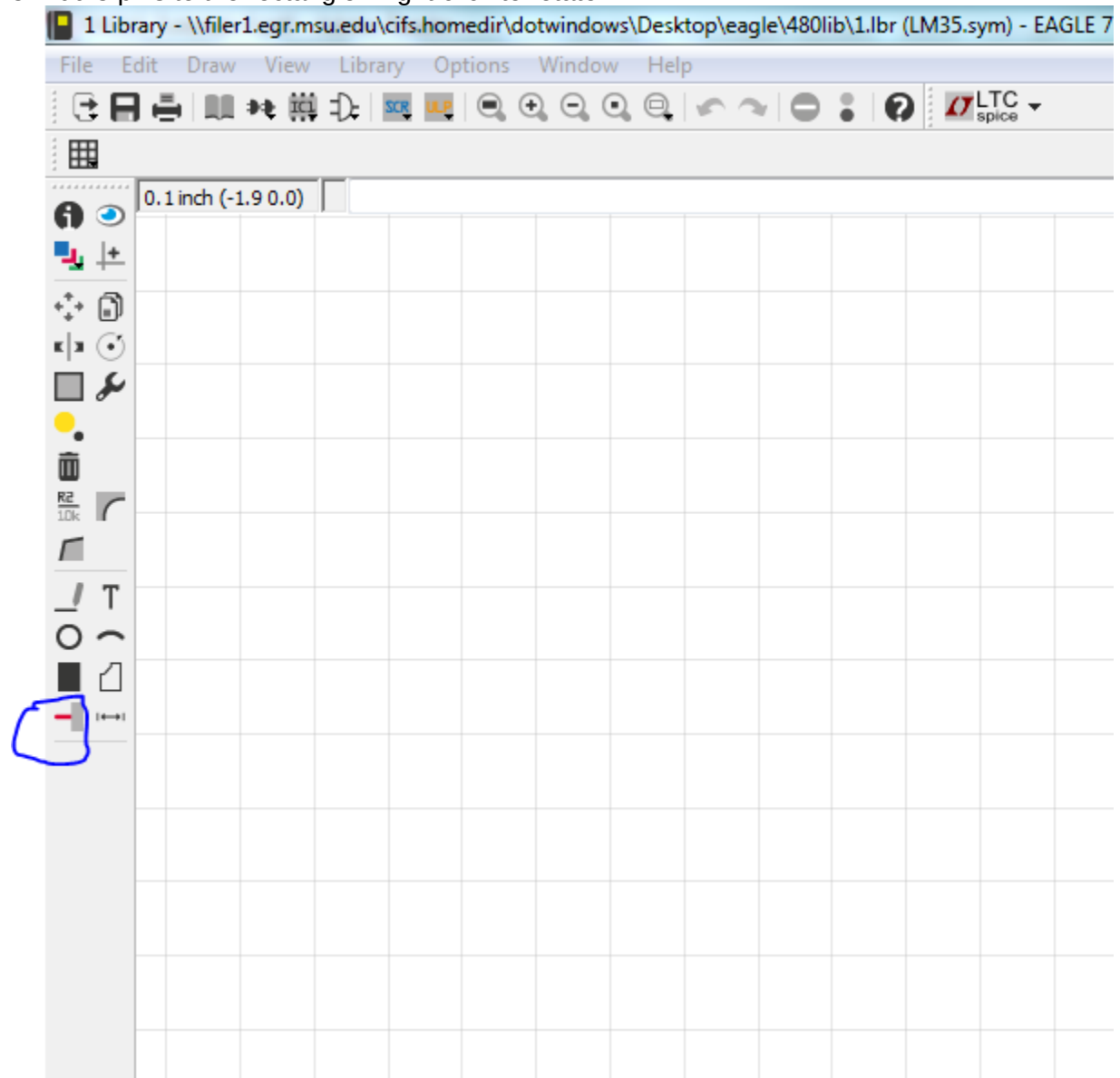
The first step to designing a PCB is to create a schematic of the circuit. To make a schematic, all components of the design must have schematic symbols, elements that represent the device within a schematic and define its connections points (pins). Thus, the first step to creating a new device is to create its schematic symbol.

1. Open the 1 library by double-clicking and a new **Library** window will open.

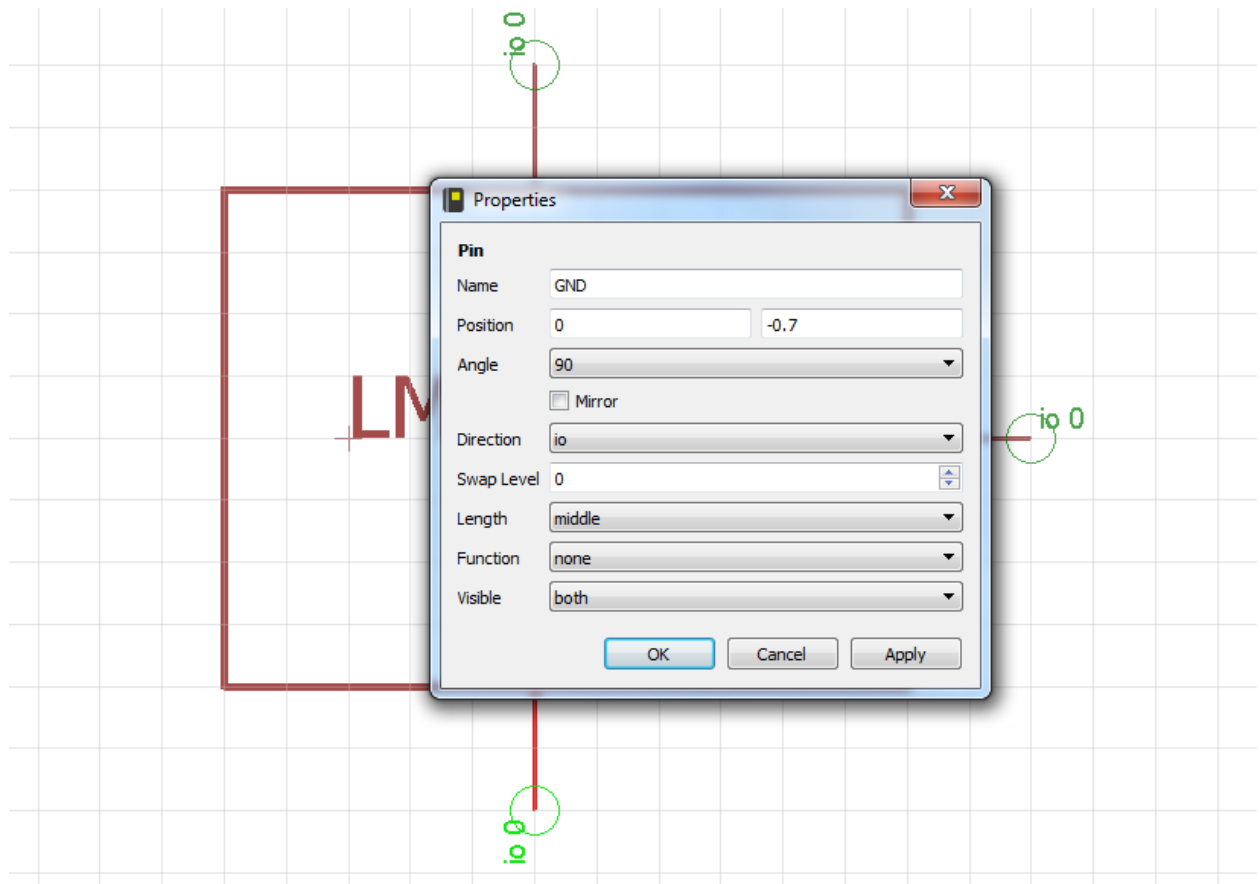


2. In the **Library** window, click the **Symbol** icon
3. In the **Edit** window, next to New: type "LM35". Then click the sym and ok.
4. Draw a rectangle as the basic symbol for LM35 sensor.

5. Add 3 pins to the rectangle. Right click to rotate.

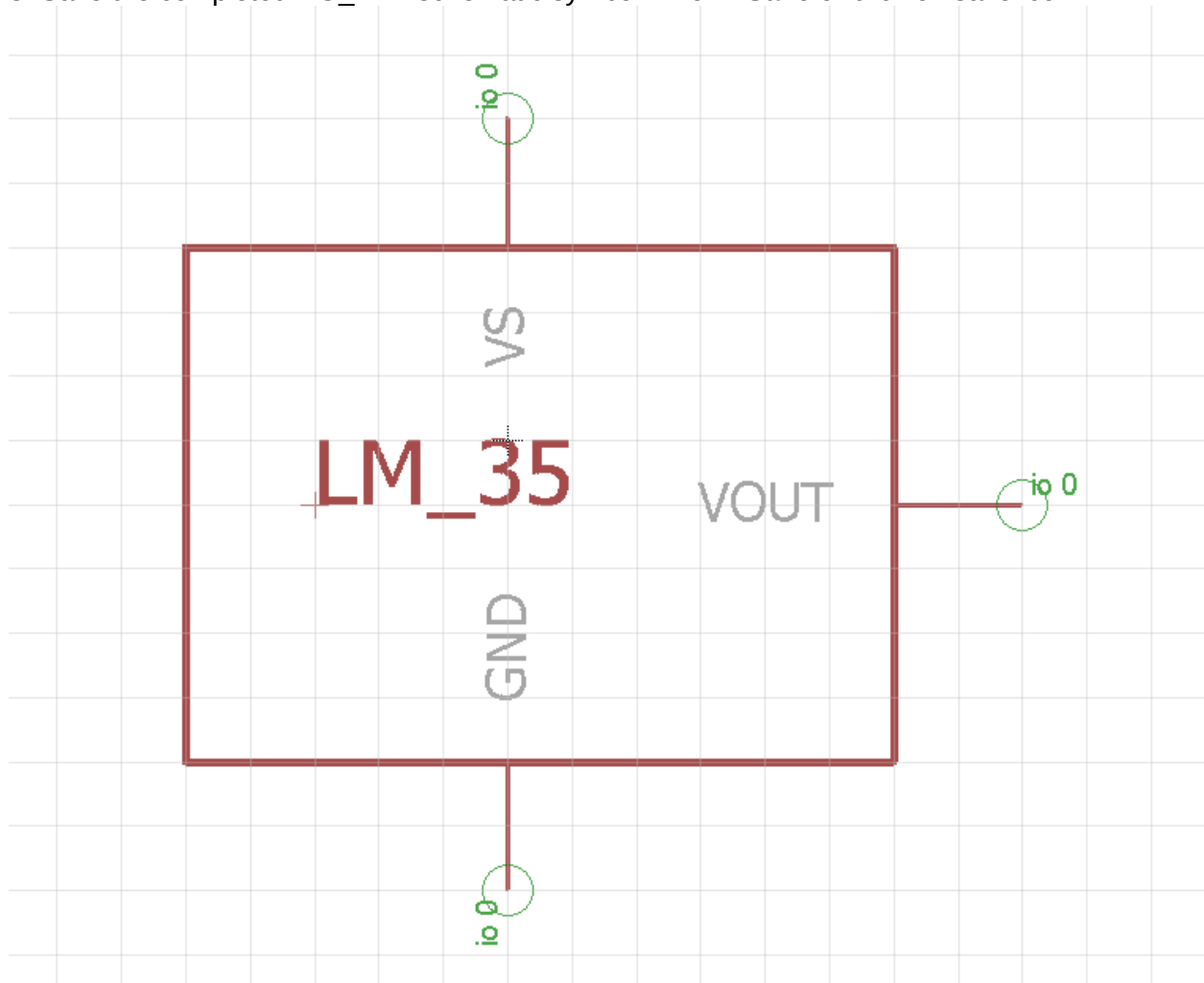


6. Right click on the pins to set new property.



7. use txt tool to add LM_35 on the symbol

8. Save the completed RG_LED schematic symbol: **File>> Save** or click on save icon.



B. Component Package-Footprint

While the symbol view defines the functionality of a component, EAGLE uses a **package** view to define the physical size and pin position of a component on the actual printed circuit board. This means that the package has to accurately represent the physical dimensions of the component being designed. Most components have a technical data sheet with their dimension provided in units of "mils". A mil is one thousandth of an inch.

1. In the **Library** window, click the **Package** icon (cmd: *EDIT*) and type "LM_35". Then click the **Pac** button at the bottom. Ensure the bar at the top says "Package" and press **OK**.

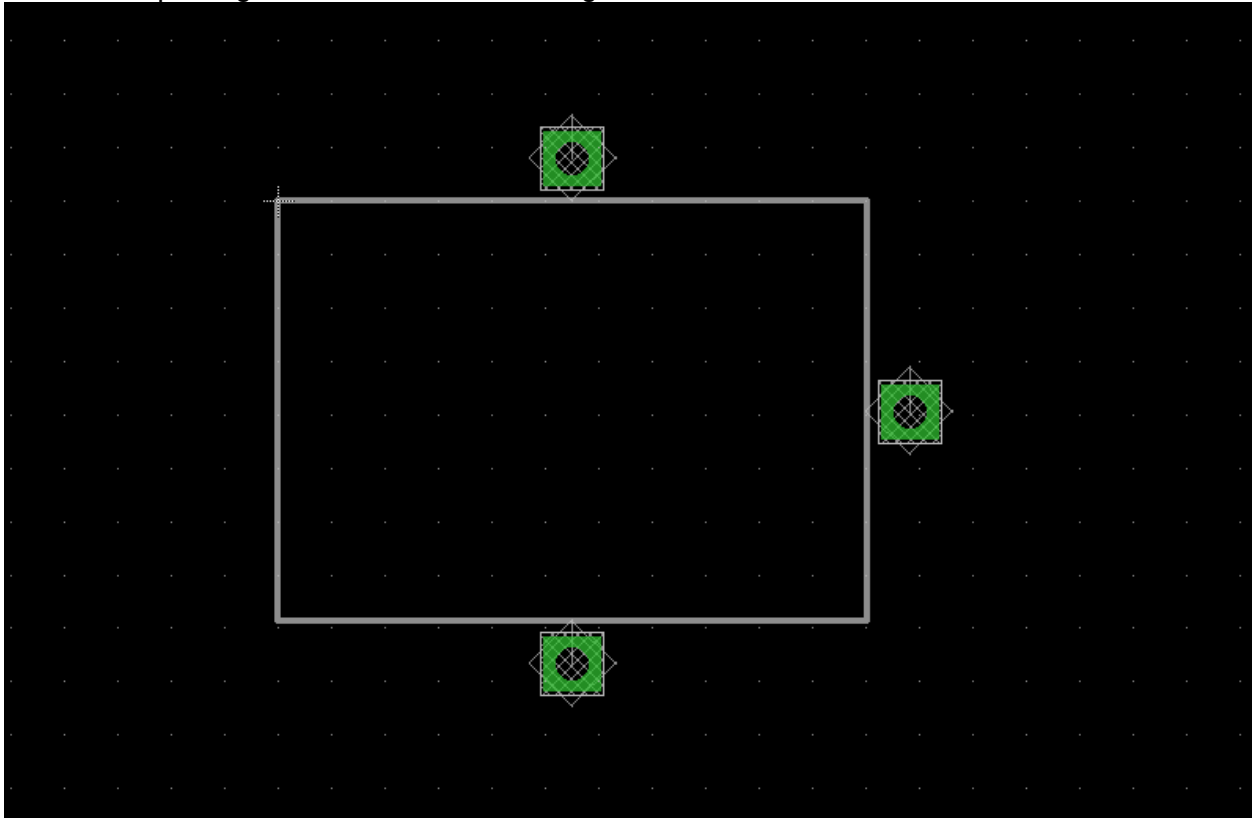
2. Begin by clicking the **Grid** icon (cmd: *GRID*) and setting the Display **On**, Style to **Dots**, and a standard size of 50mil (select *mil* from drop down menu next to Size). Then Press **OK**.

3. Use the **Wire** command to draw the rectangle based on the data from data sheet to match the physical dimensions

4. To complete the Package-Footprint, **pads** that identify input/output pins need to be placed at precise locations within the package outline. First we need to measure the proper place for the pads.

5. Assign names to the three pads using the **name** tool. Name the top pad "VS", middle pad "VOUT" and "GND" for the bottom one.

6. The final package should look like the image shown here:



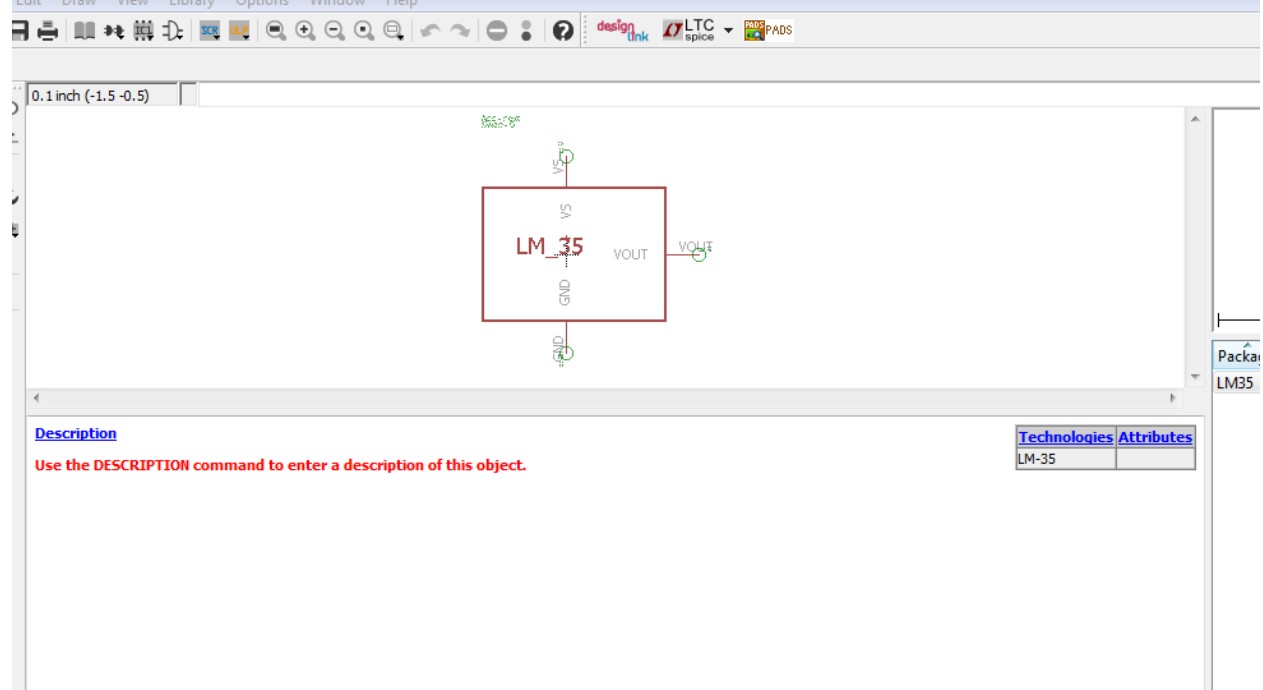
7. Save the PCB package: File>> Save.

C. Creating a New Device

Once the (schematic) Symbol and (board layout) Package-Footprint have been created, the two parts of a component must be linked together in a process called creating a **new device**. New devices can be created from new or existing symbols and package-footprints or any combination of both. The primary goal of creating a device is to identify the connections between pins on the schematic symbol and pads on the layout package-footprint. This allows connections made on a board layout to be verified against connections made on a circuit schematic.

1. In the **Library** window, select the **Device** button (cmd: *EDIT*) from the top of the EAGLE toolbar and type “**LM_35**” in the entry box. Then click the **Dev** button at the bottom (to identify this as a Device element for the component. Ensure the bar at the top says “Device” and press **OK**.

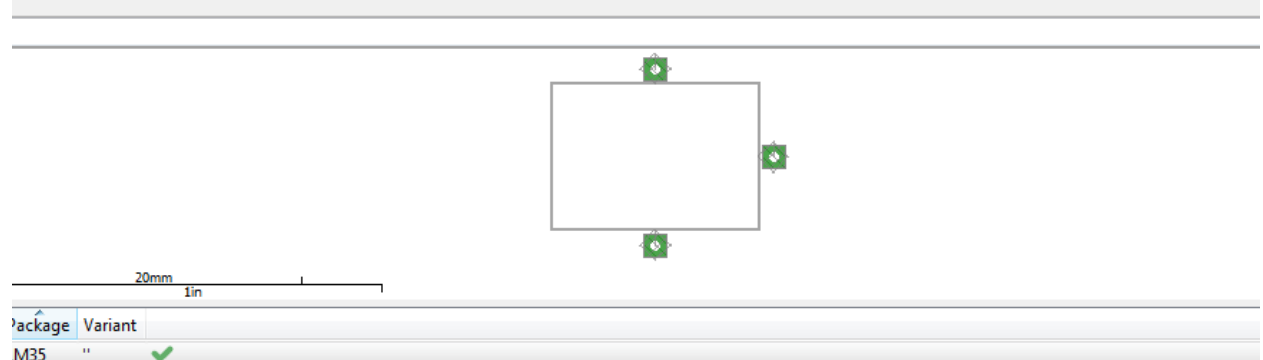
2. To link your LM_35 symbol to this new device, select the **Add** button from the side toolbar. Select the “LM_35” symbol and press **OK**.



3. Place the LM_35 symbol at the origin of the schematic window. Then hit **Esc** to exit the Add tool.

4. On the bottom right of the **Library** window, click **New**.

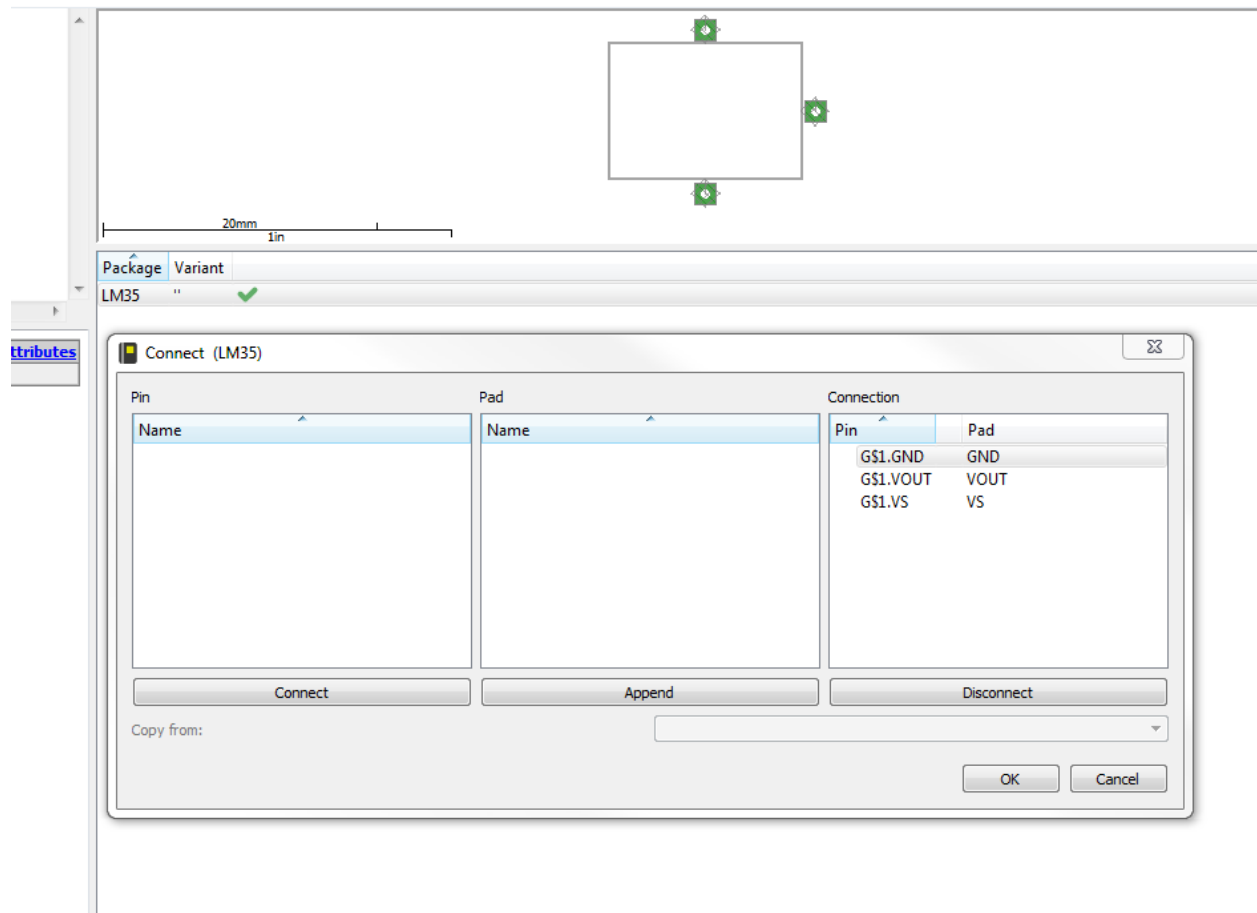
The LM_35 package should be selected by default. Press **OK** to place the package symbol.



5. Click **Prefix** (below New) and specify “LM-35” as the naming prefix for this device.

6. The final step of creating a device is connecting the pins of the schematic to the pads on the package-footprint. Click on the **Connect** button (next to the “New” button) to open the connection panel shown below.

7. Use the graphical interface of the symbol to determine which (symbol) Pin Name connects to each (layout) Pad Name. Then click **Connect** in the **Connect panel** to link the symbol and package. Once all connections are complete click **OK** and the Pad numbers will appear on the symbol, as shown on the image to the right.



If you make any mistakes, go back to the Connect panel and disconnect/reconnect pin-pad connections.

Make sure that the number assigned to the pad closest to the notched (flat) side matches with the ground pin on the LM35.

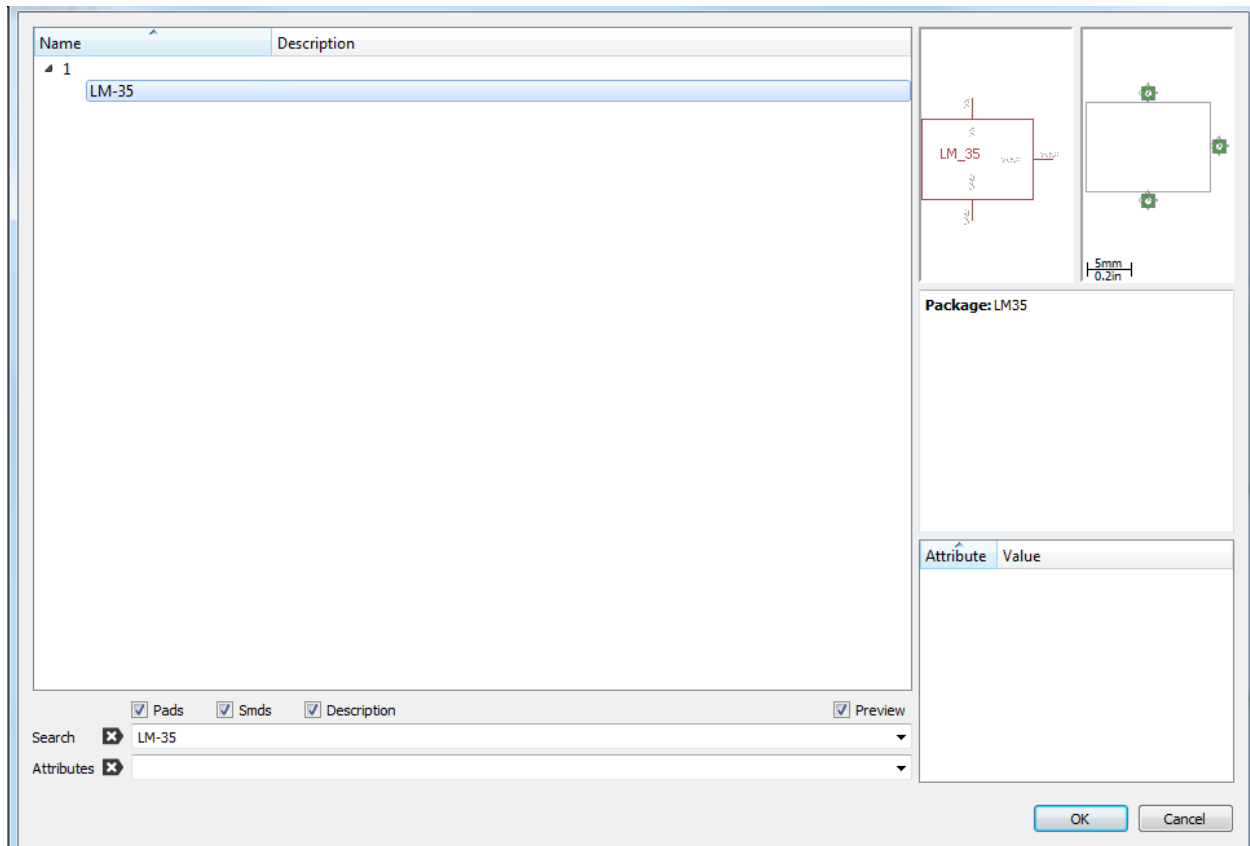
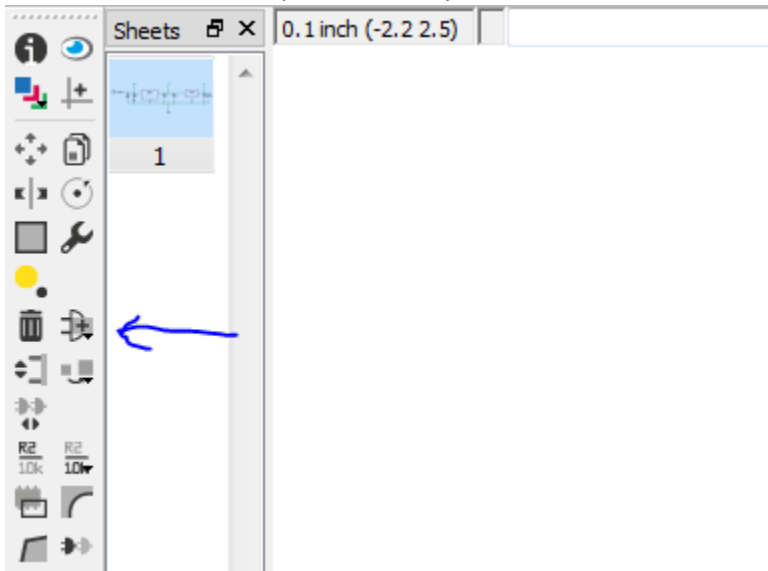
8. Check that your final device looks like the image shown here.

9. **Save** the Device.

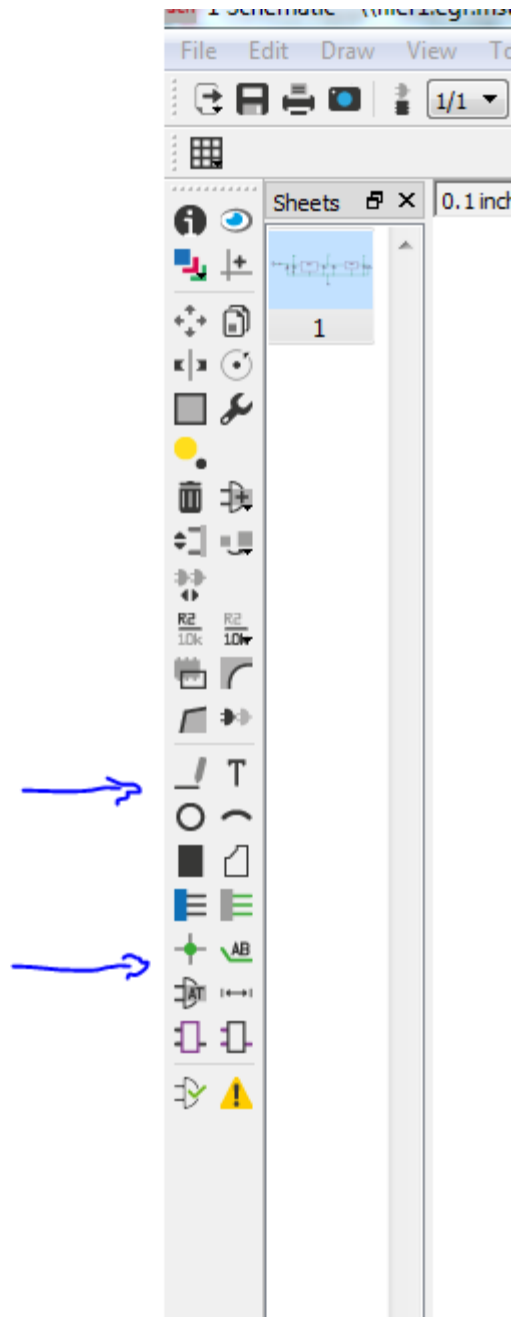
10. Exit the **Library** window to return to the EAGLE **Control Panel**.

4 build the PCB Schematic

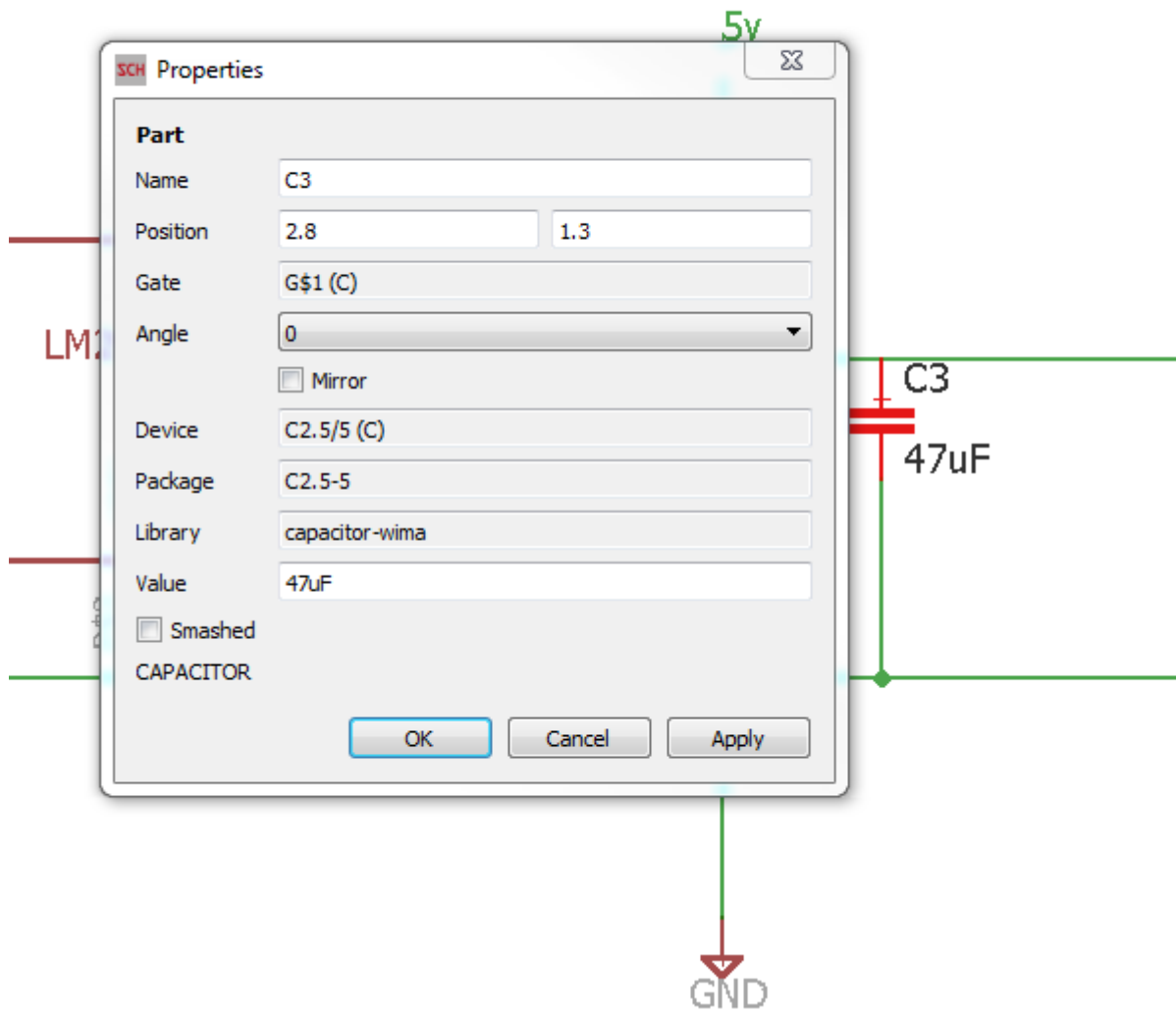
1. Click the ADD icon to Open the library and use the search feature to find the component.



2. Click ok and put it on the Schematic area.
3. After put all the component on the Schematic area use wire tools to wire the component. Remember to add junction on the wire junction.



4 . Right click on the Component which need assign a attribute (Resistor, capacitor) and left click on properties to assign the value.



5 .Then add the ground from library to finish the Schematic. You may also want to use text tool to add comment.

6. This the final schematic:

