PCB Design

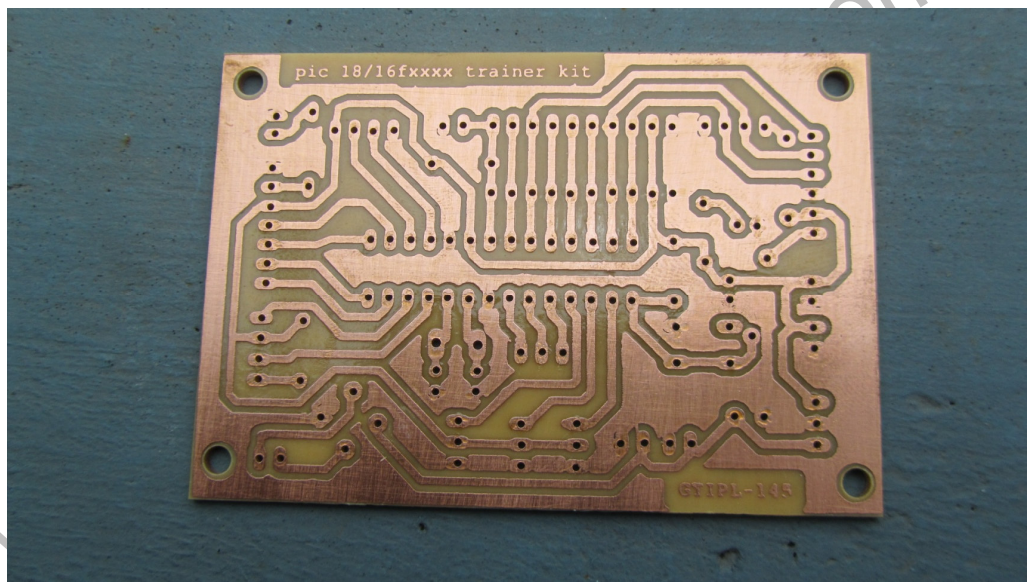
www.greytechnolabs.com

Introduction

About PCB

- Mechanically supports and electrically connects electronic components.
- Technically called as Printed wiring board (PWB).
- TWO SIDES:-
  1. PRINTED SIDE
  2. COMPONENT SIDE

Printed Side



Component Side

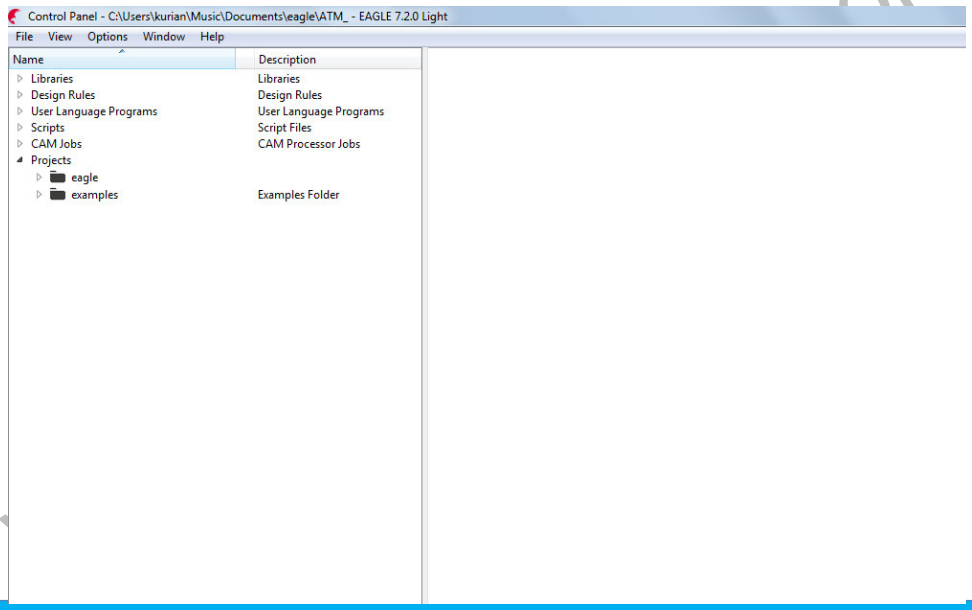
- Starting with schematics
- Preparing the board layout
- Create new Library
- Gerber file creation
- BOM
- Netlist

A Quick Note On How Eagle Works

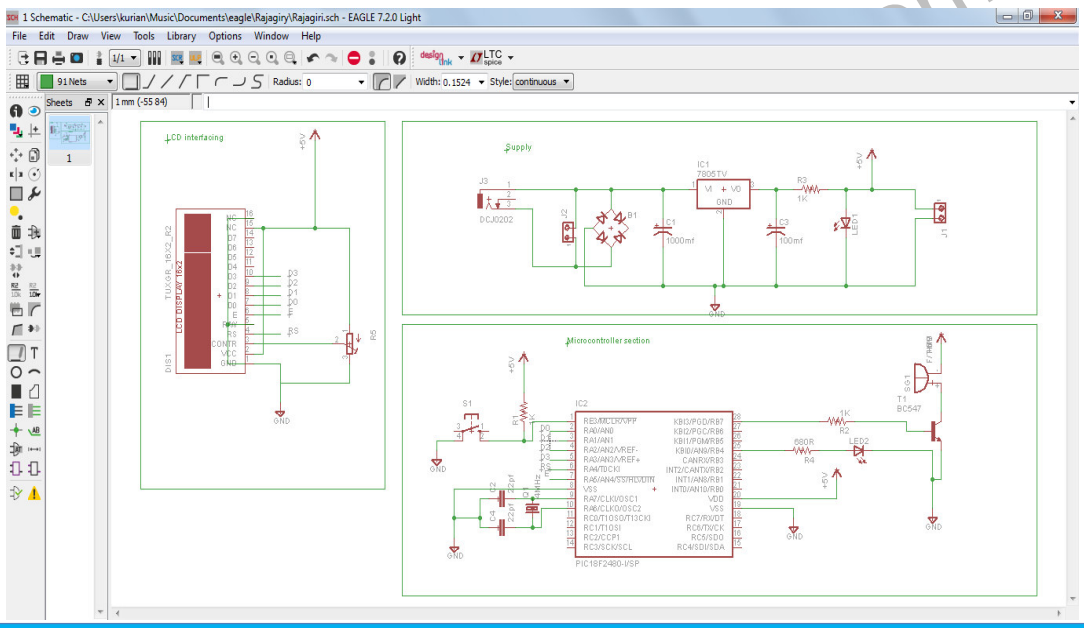
Eagle has three basic views

- Control Panel: is the main window, it launches everything else and when you close it, all subordinate windows get closed.
- Schematic: where you draw the schematic for your project.
- Board: where you lay out the pieces of your project and physically connect the correct pins as defined in the Schematic.

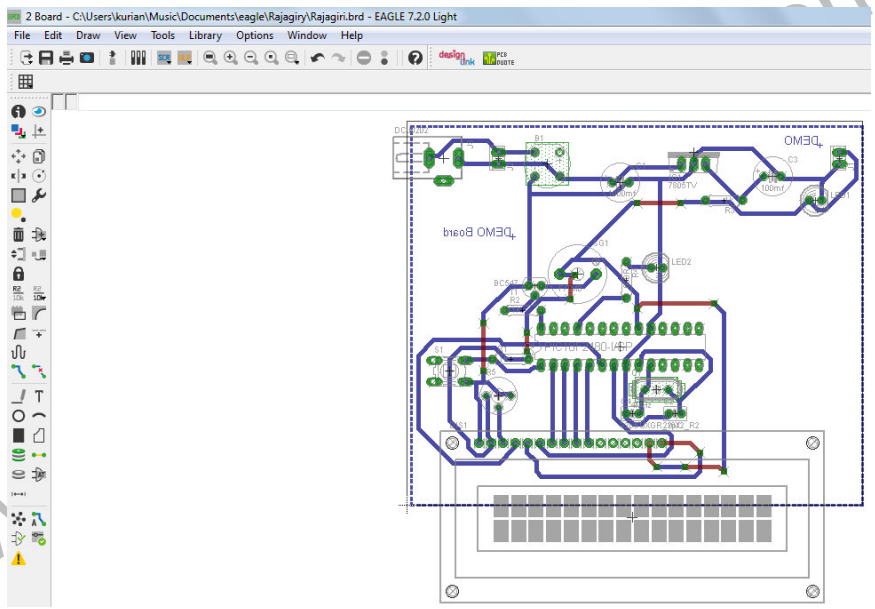
Control Panel



Schematic Window



Board Window



Familiarization of Eagle Lite

Highlights:
Part-1

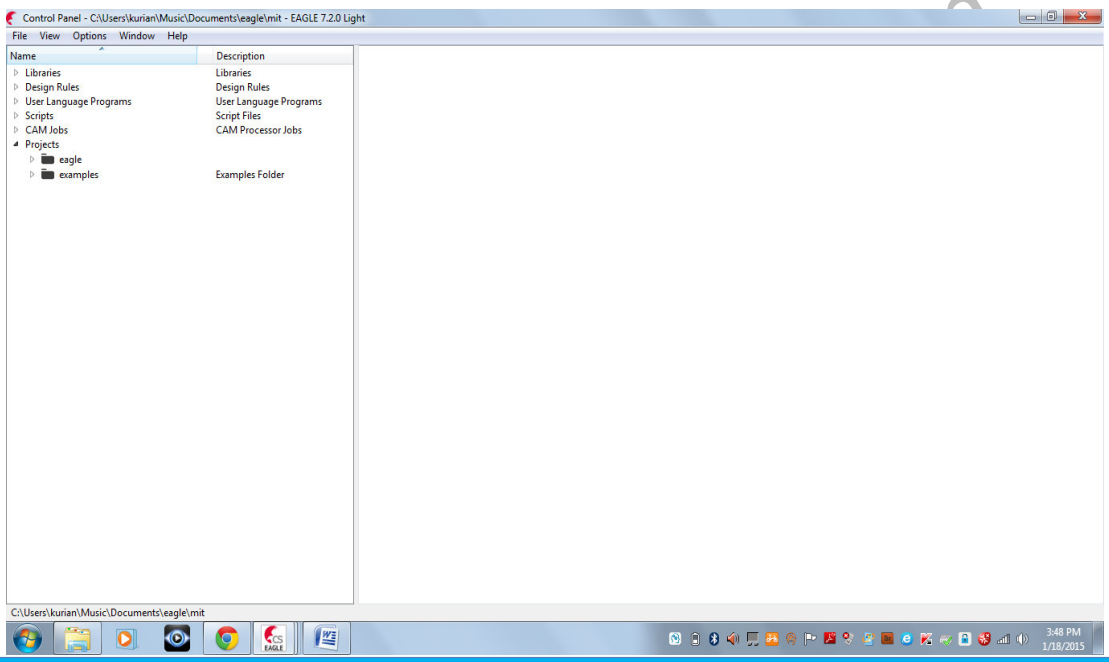
- Create a New Project and Schematic
- Add the Parts to the Schematic
- Connect the Parts
- Label and Name all of the Nets
- Give the Parts Some Values
- Electrical Rule Check

Part-2

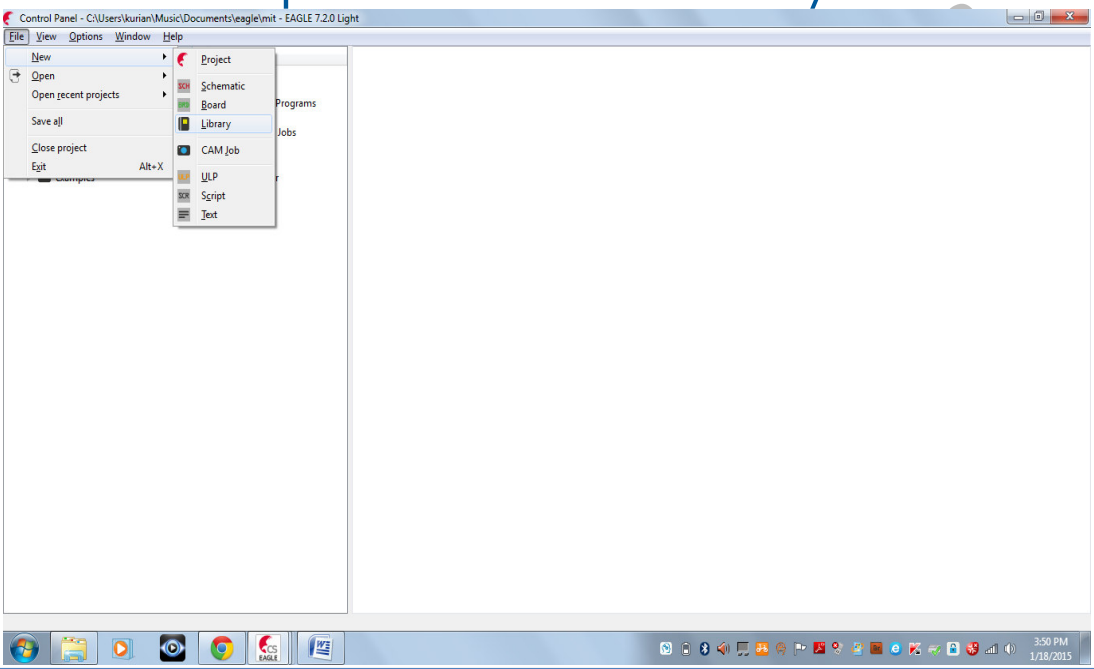
- Board Layout
- The Design Rule Check
- DRC Results

CREATE NEW LIBRARY IN EAGLE

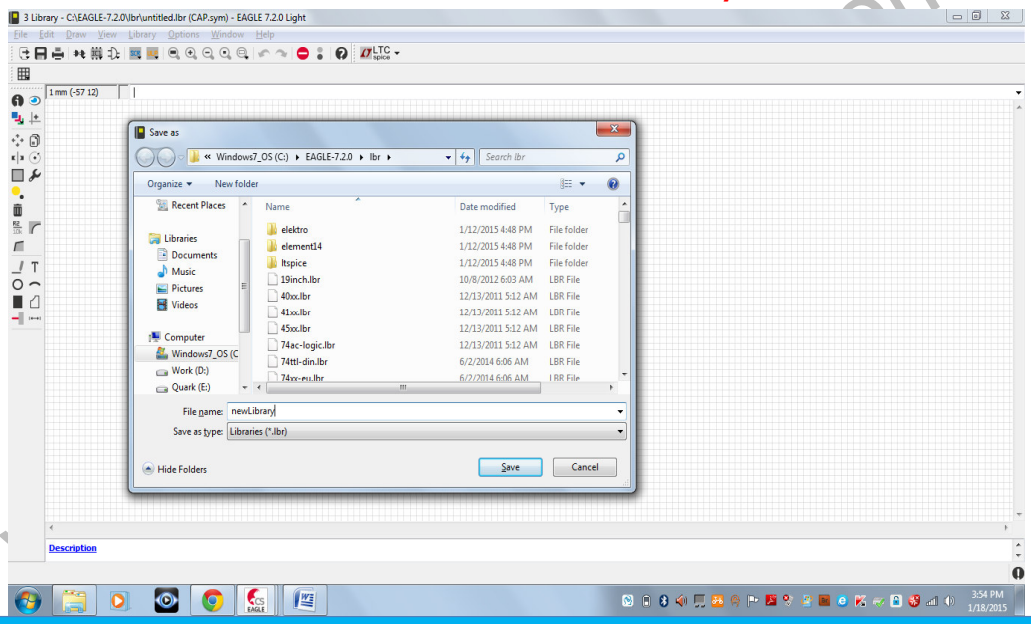
Step 1: Go to control Panel



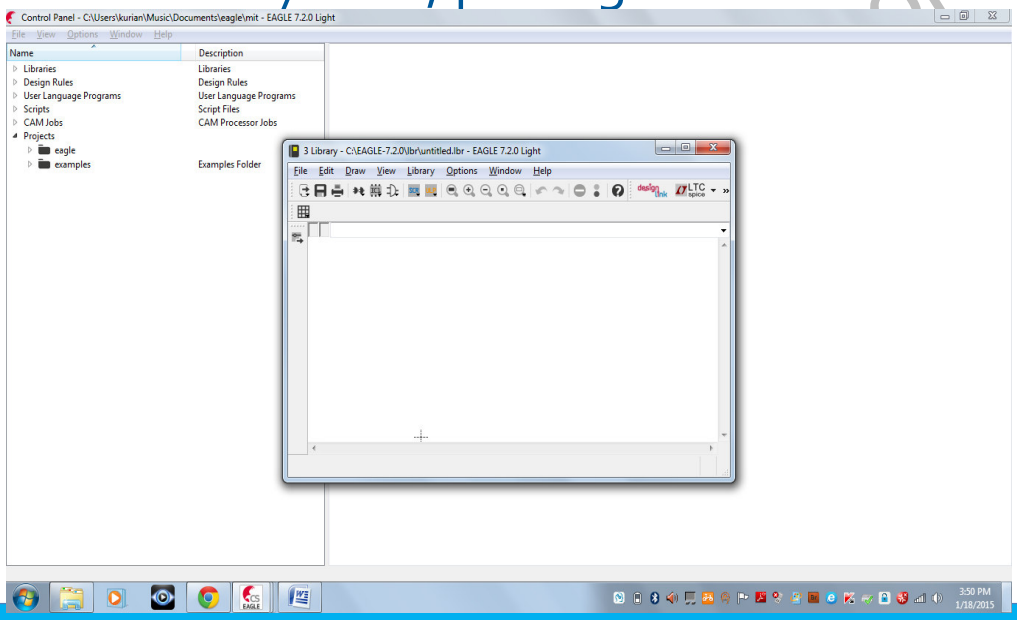
Step 2: Create new Library file



Step 3: Give a **name** to your Library, (here file name set as **newLibrary**)

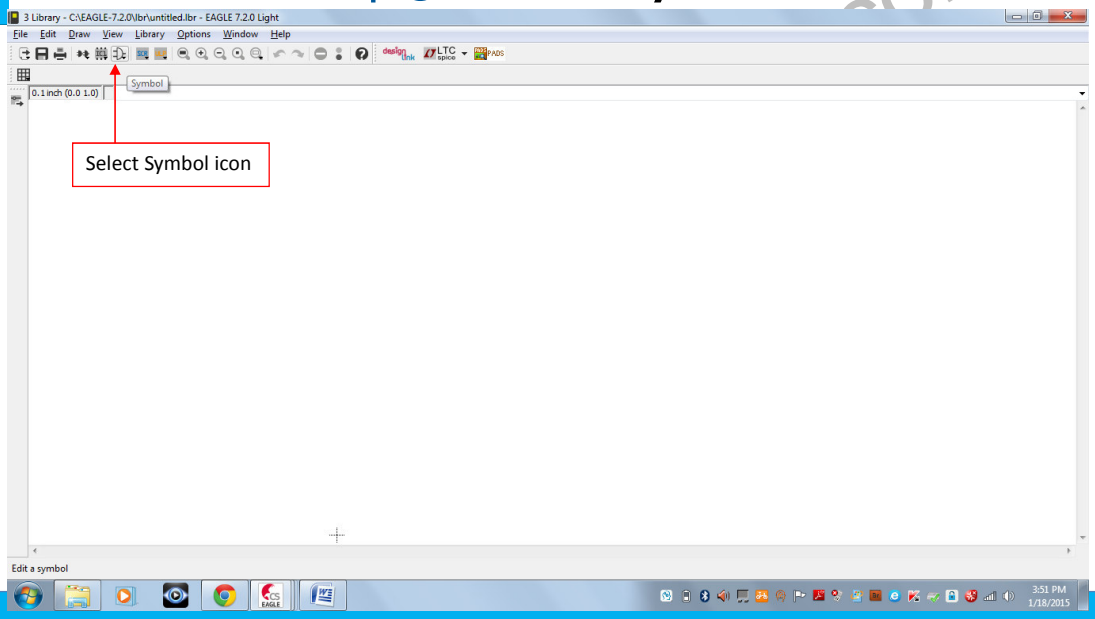


Step 4: Here you need to create new symbol, package and Device

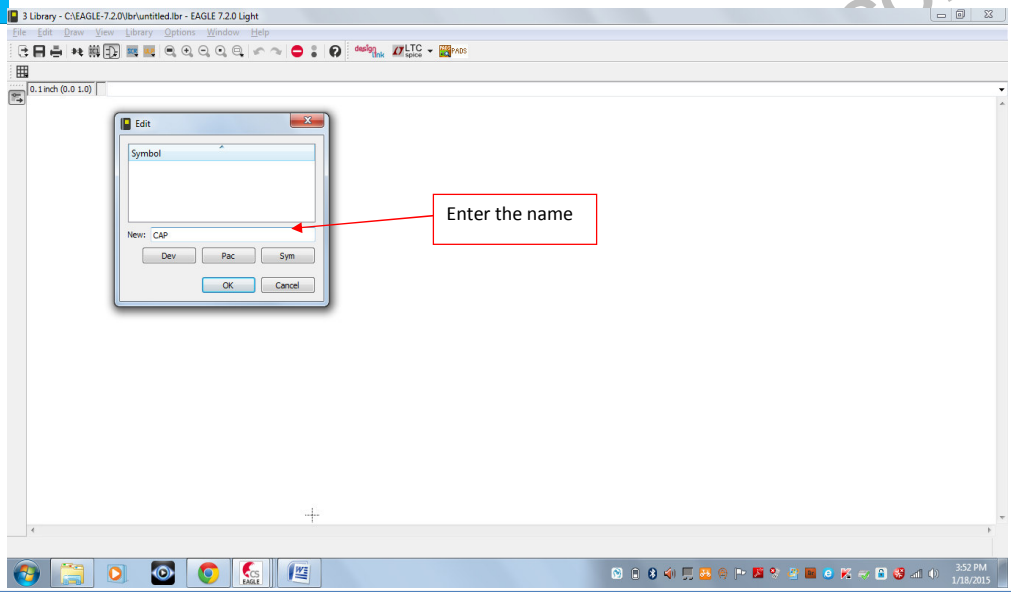


Edit Symbol:

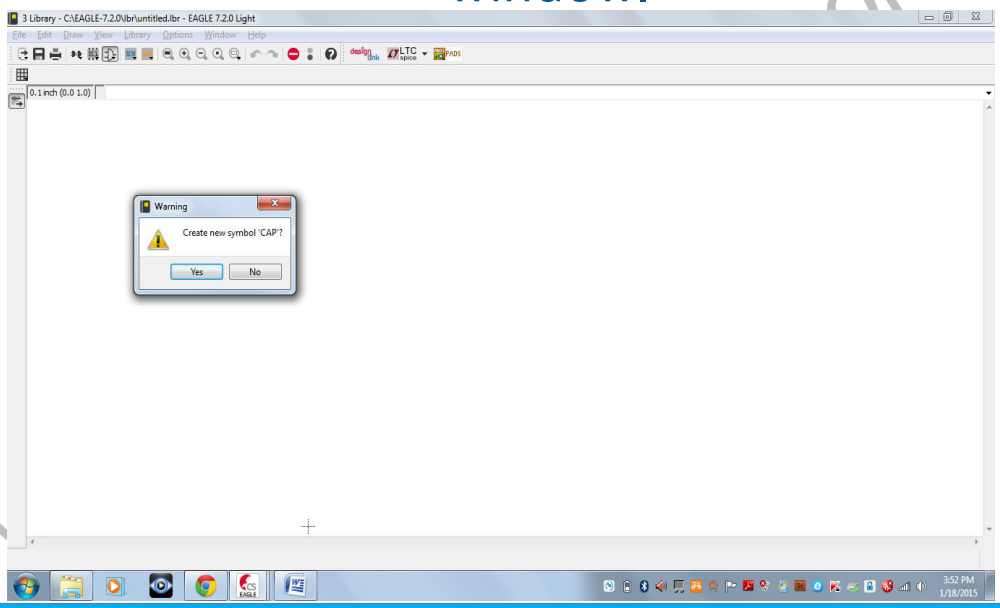
Step 5: Select **symbol** icon



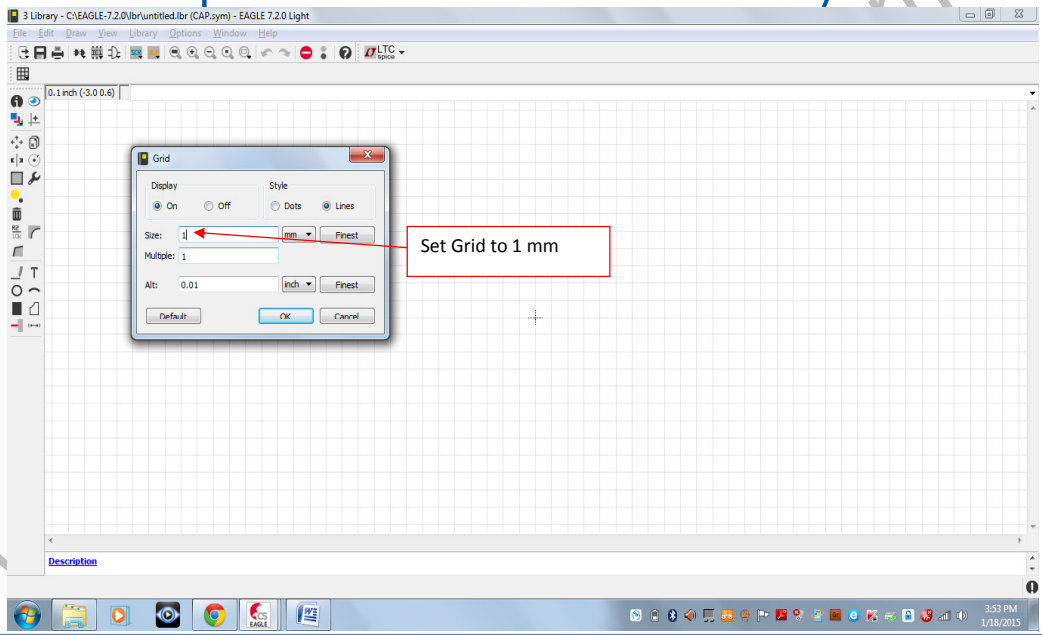
Step 6: Enter the **name** of symbol



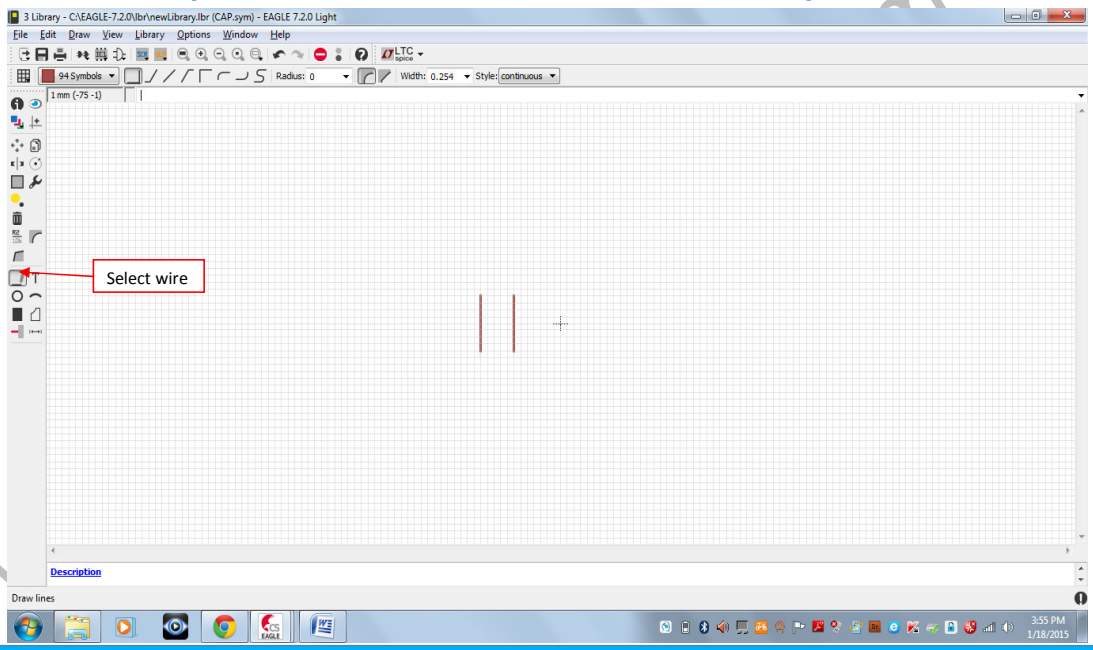
Step 7: Select '**yes**' to close the warning window.



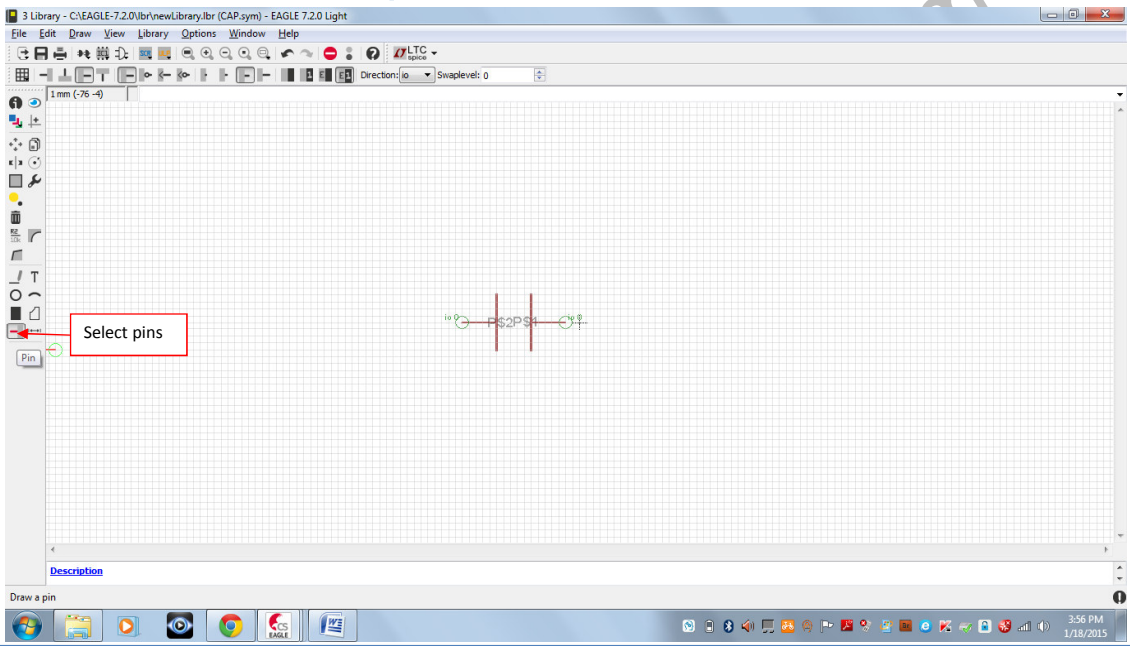
Step 8: Set **Grid** size for the Symbol.



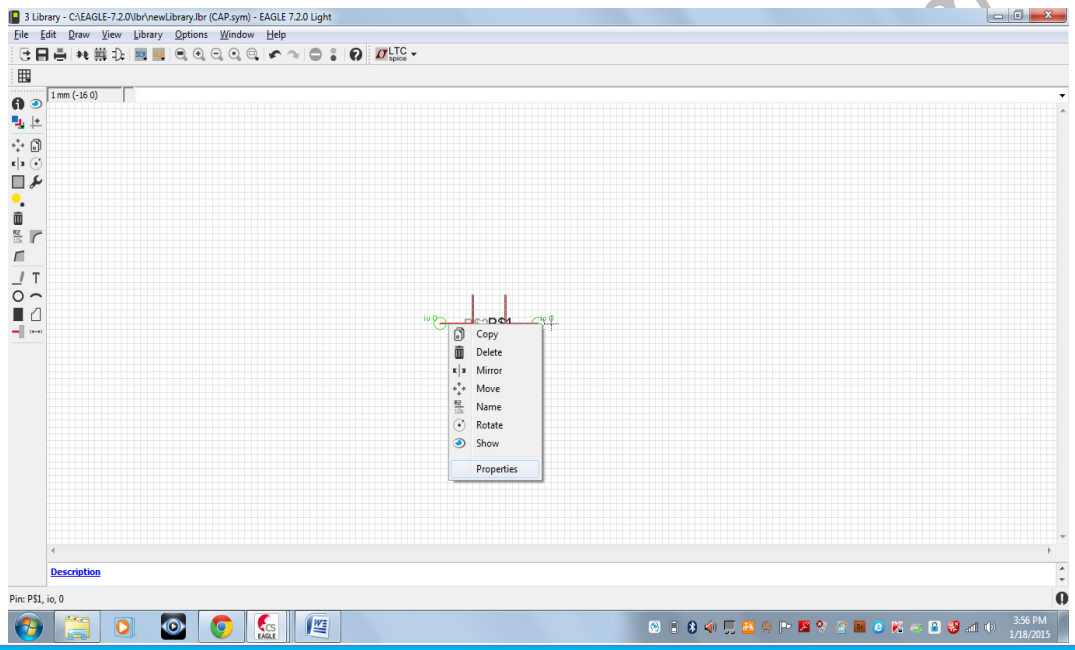
Step 9: Draw the schematic symbol



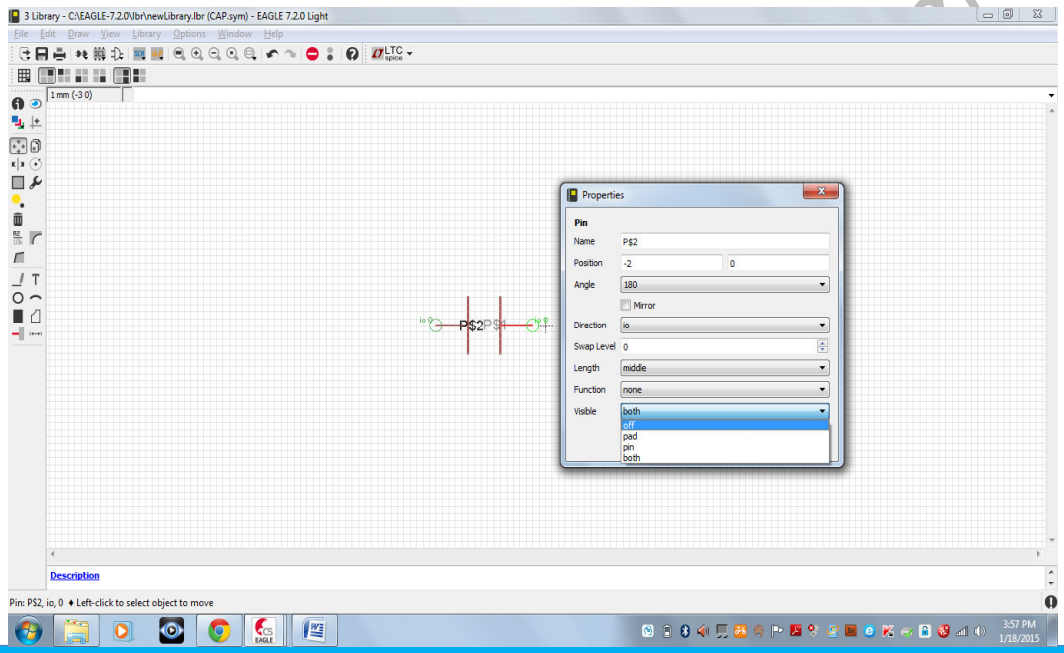
Step 10: connect Pins



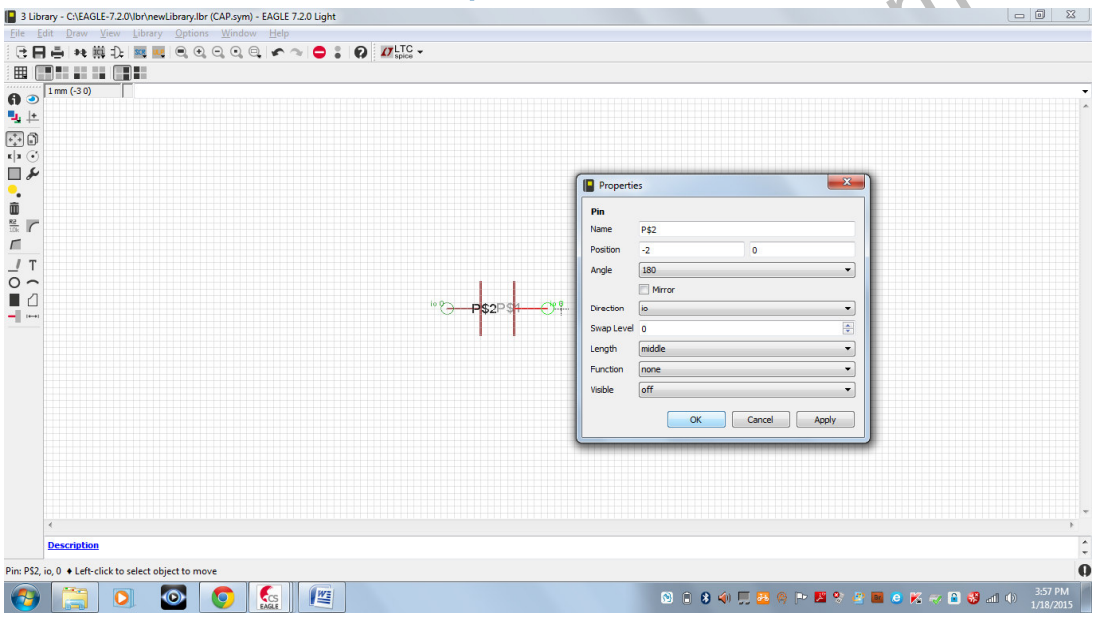
Step 11: select pin1>properties > visible>off



Step 12: select pin2>properties > visible>off

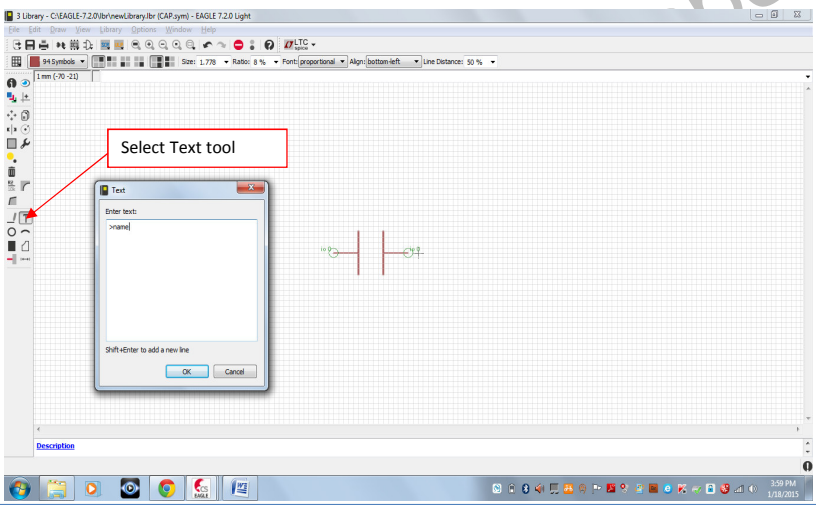


Step 12.a:

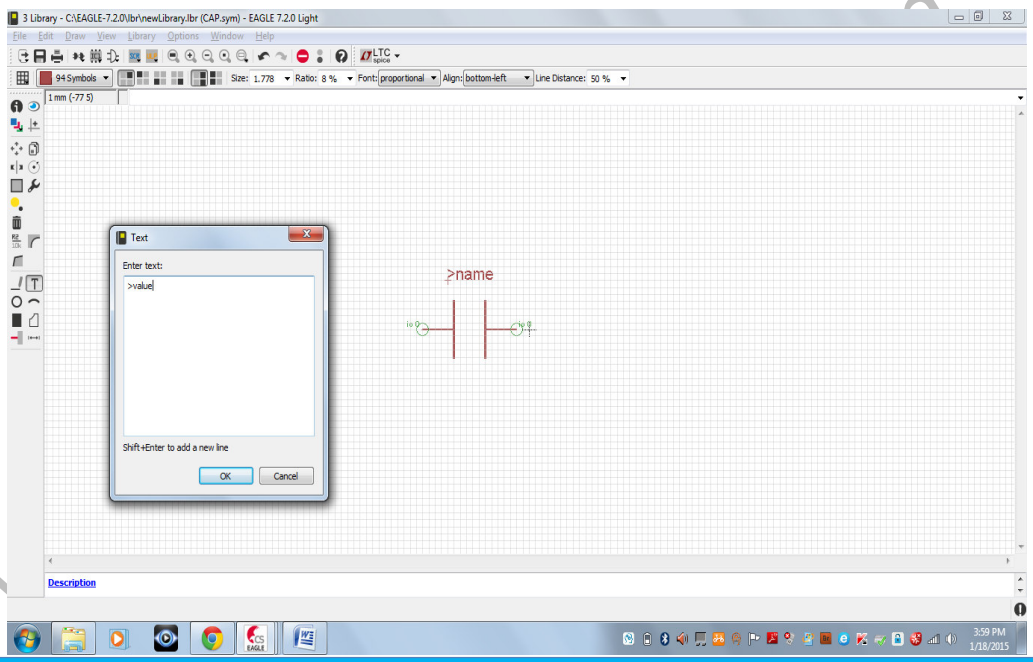


Give option for **Name** and **Value** for the new symbol

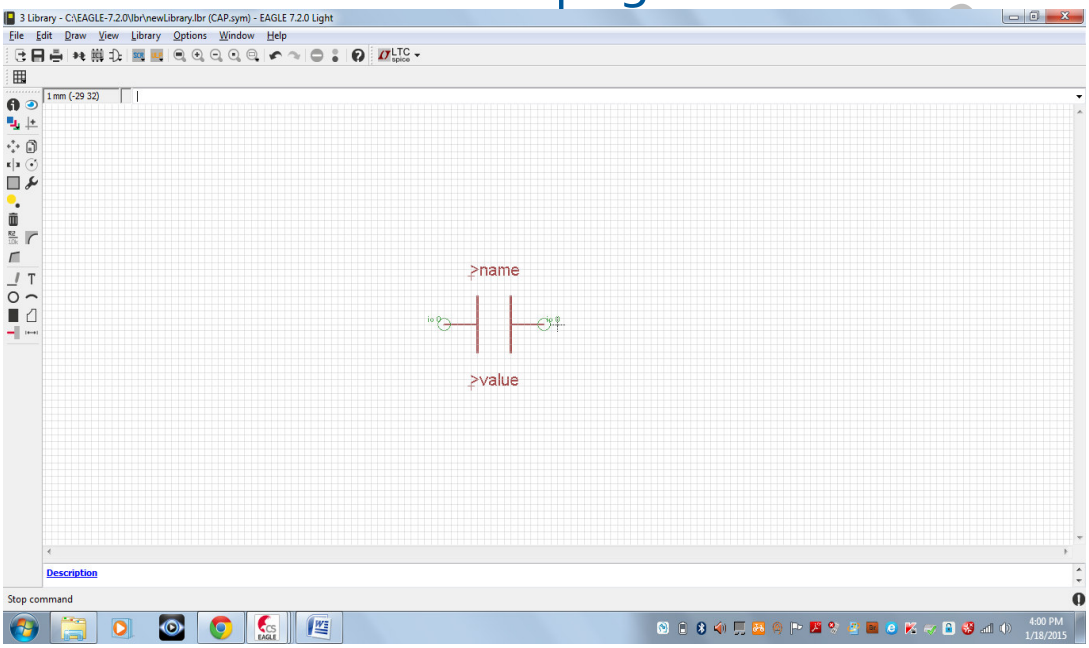
- Step 13.a: select Text Tool



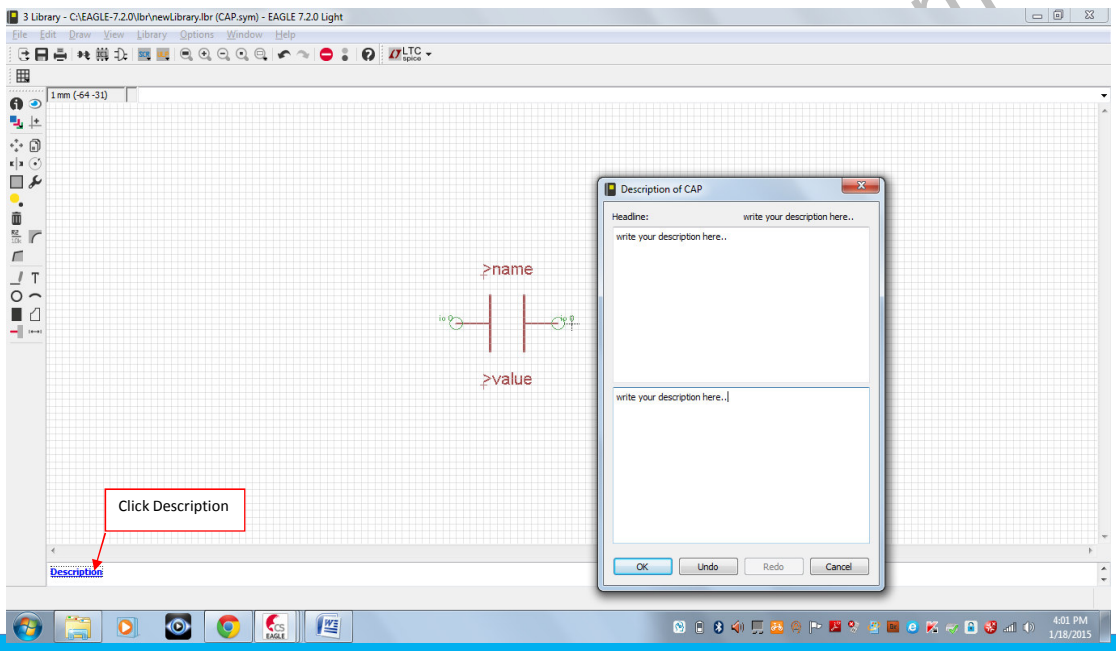
Step 13.b:



Step 13.c:

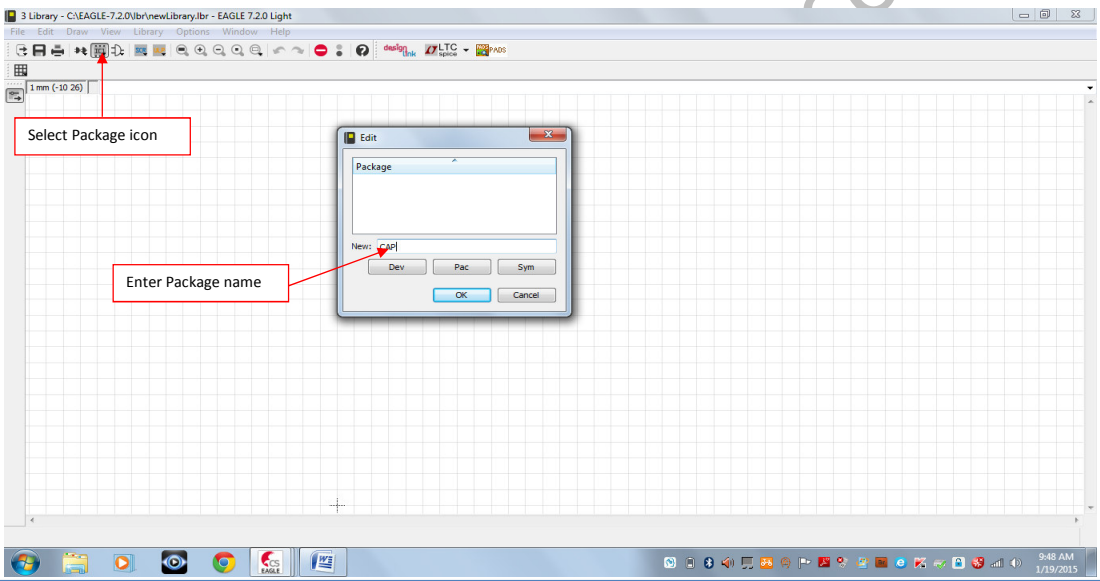


Step 14: Give description

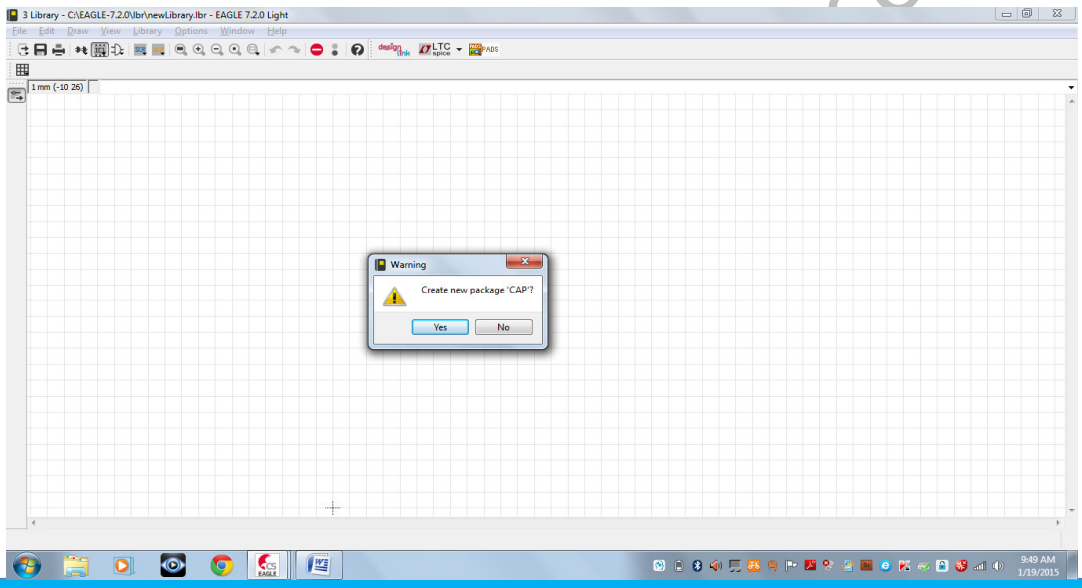


Edit package:

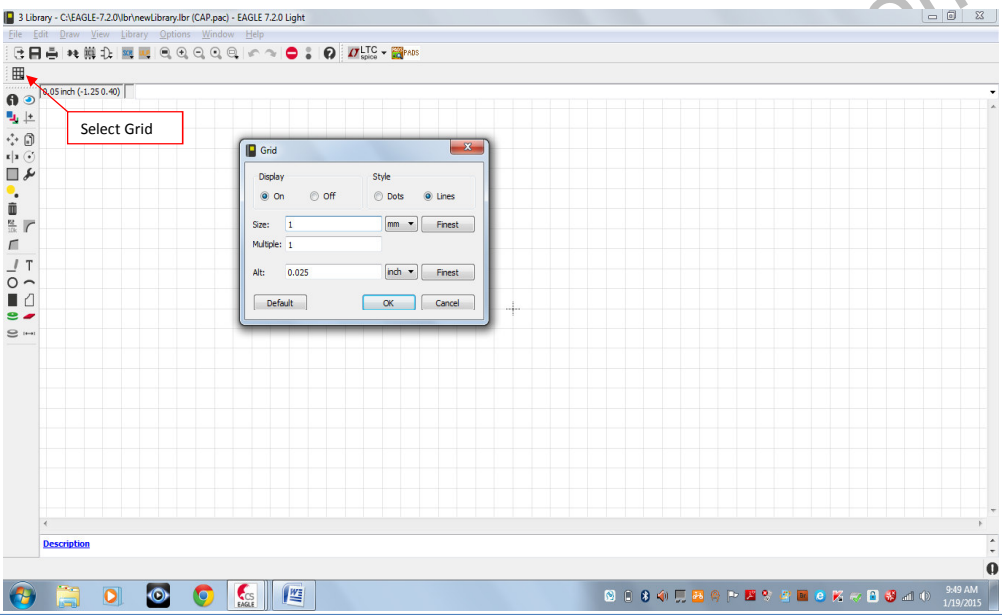
Step 15: Select package icon



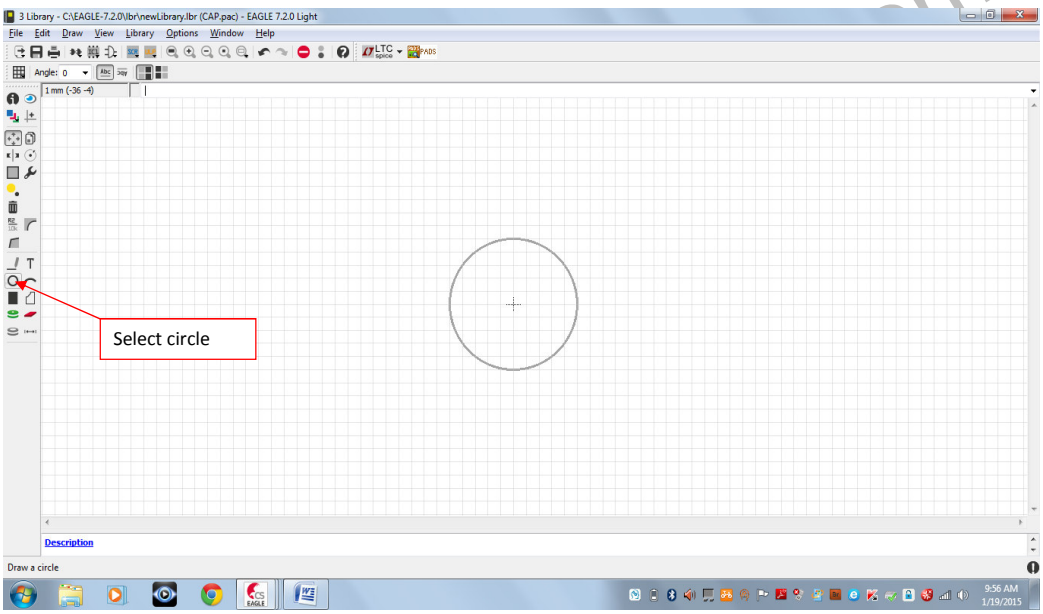
Step 16: Select '**Yes**', to terminate the warning window.



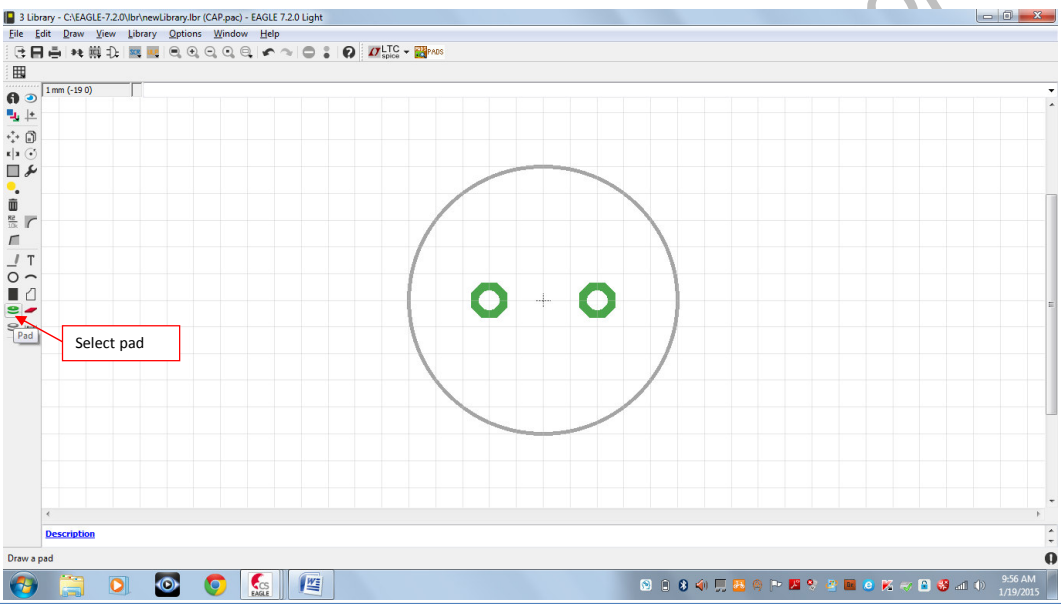
Step 17: Set Grid size to 1 mm.



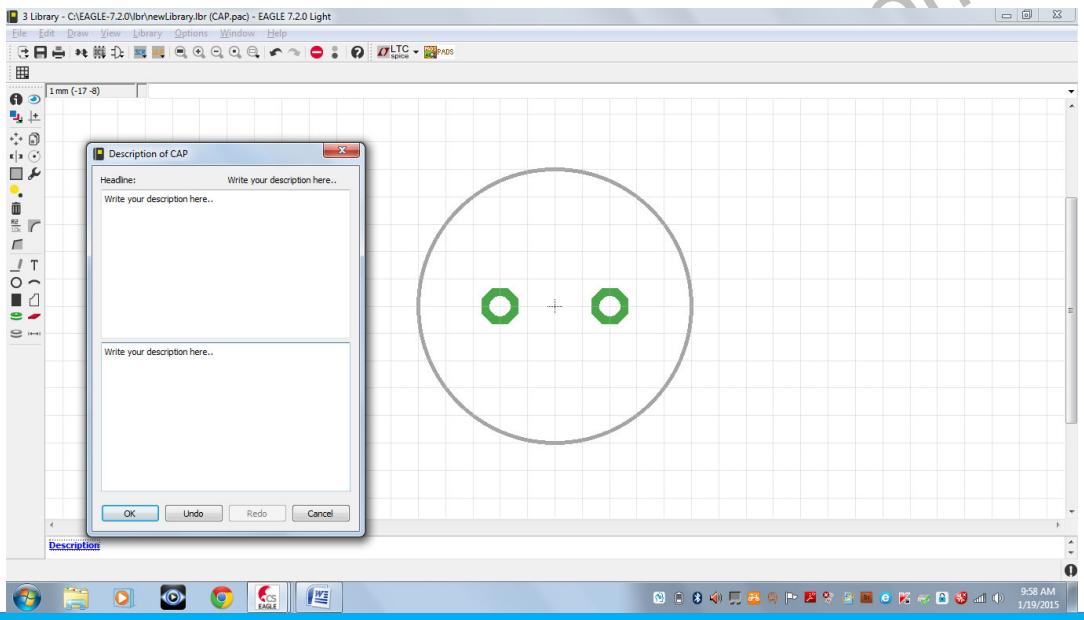
Step 18: Draw package outline.



Step 19: Select Pad

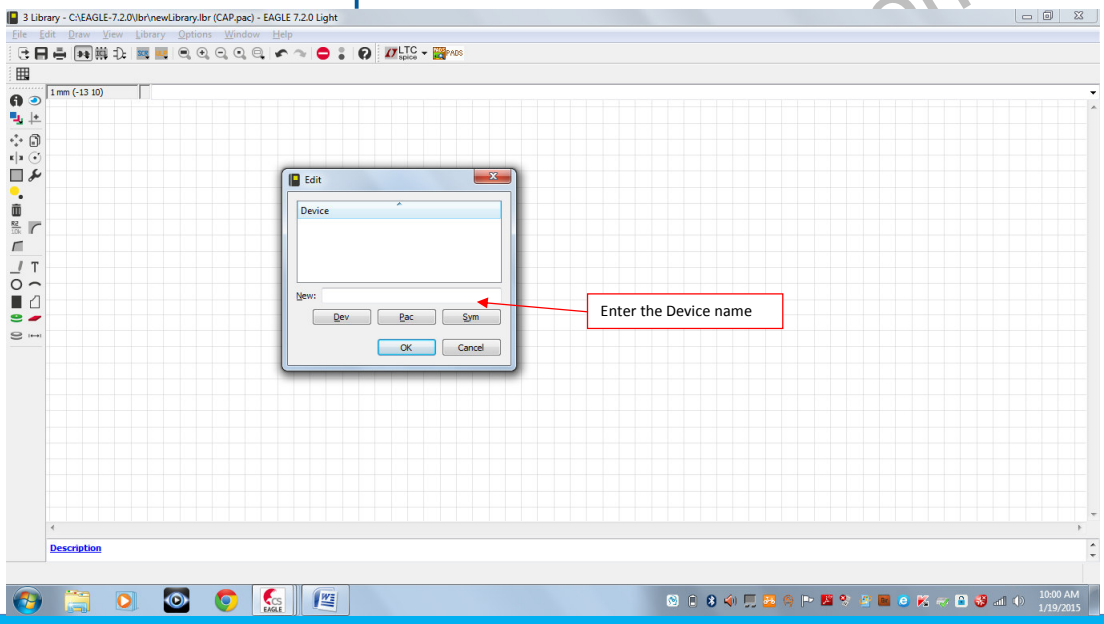


Step 20: Give description for the package.

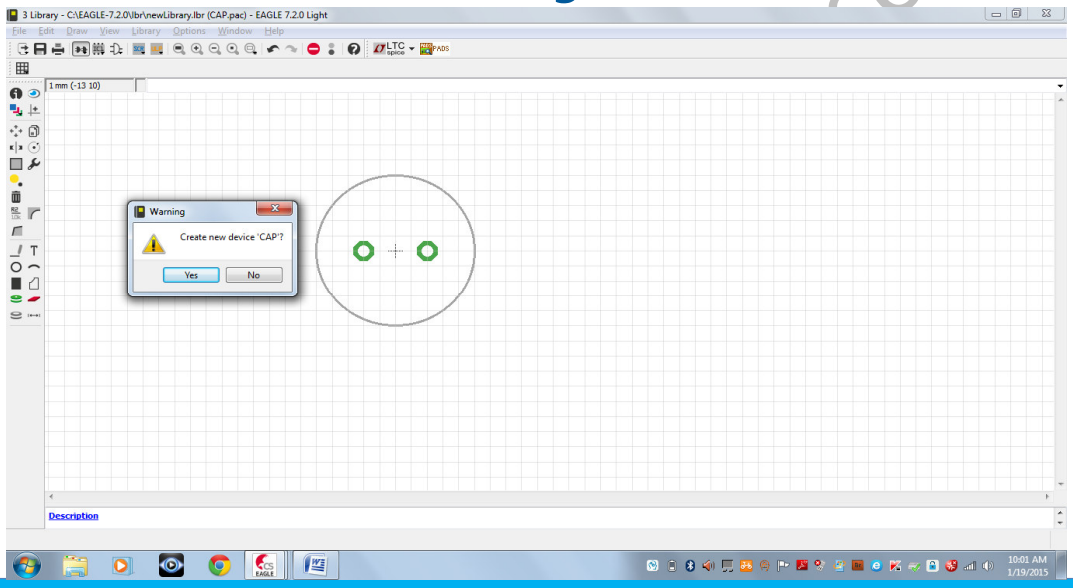


Edit Device:

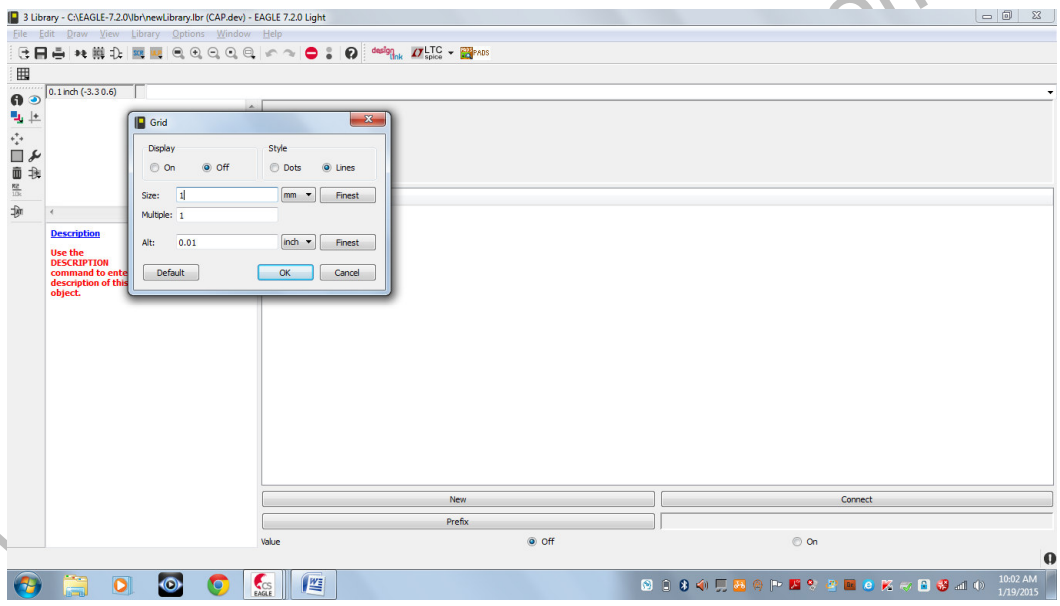
Step 21: Enter device name.



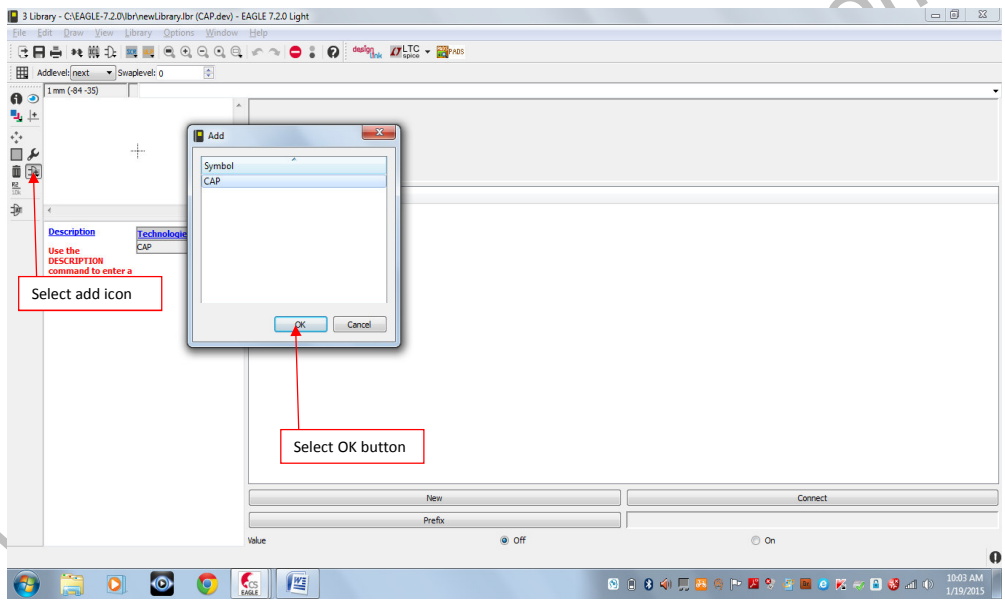
Step 22: Select '**Yes**', and terminate the warning window.



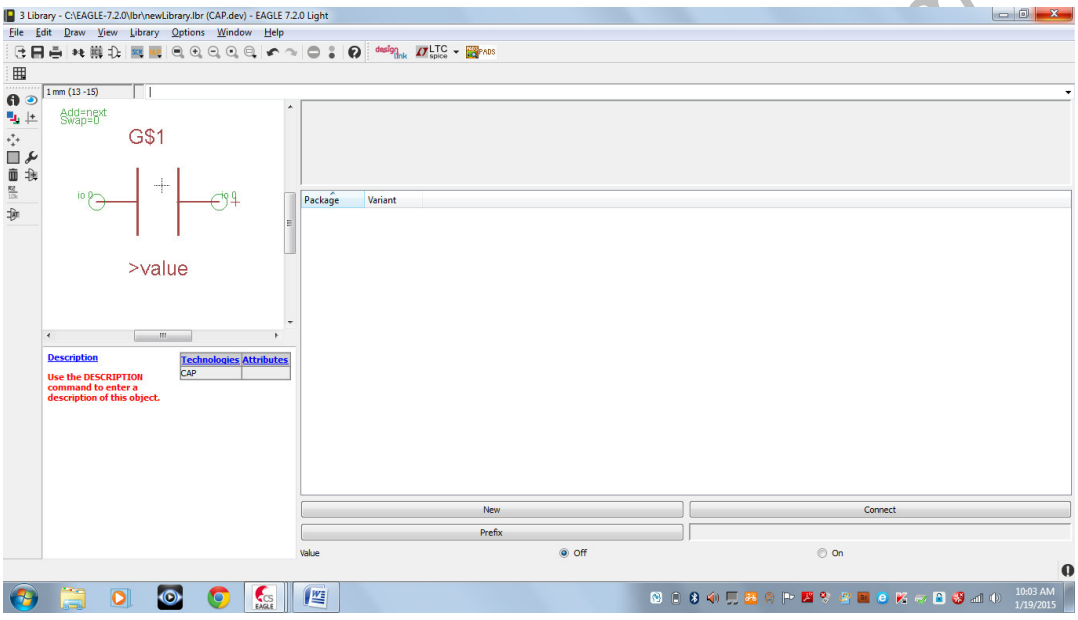
Step 23: Set the Gird size for the Device.



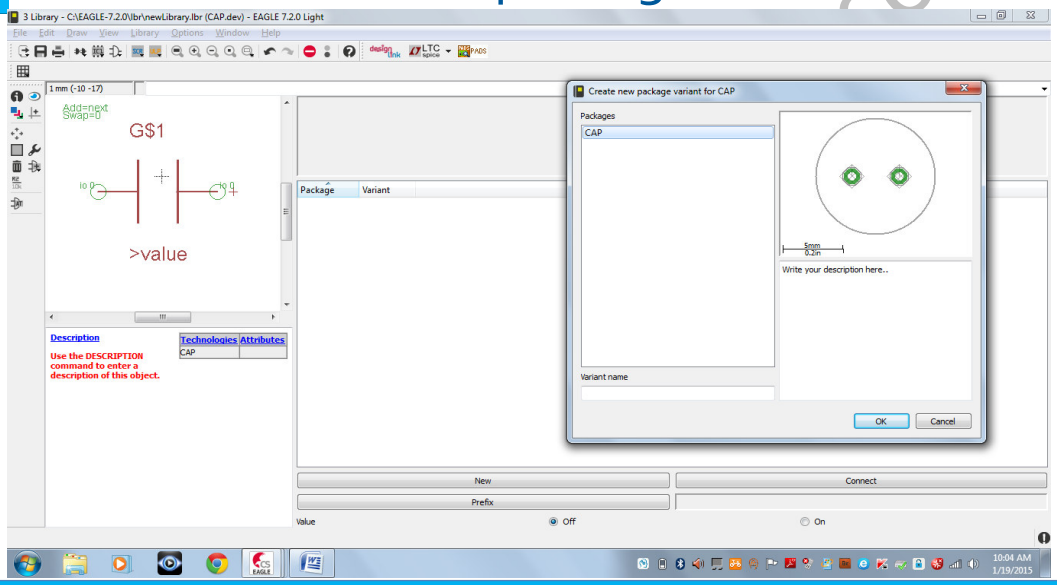
Step 24: Add the symbol



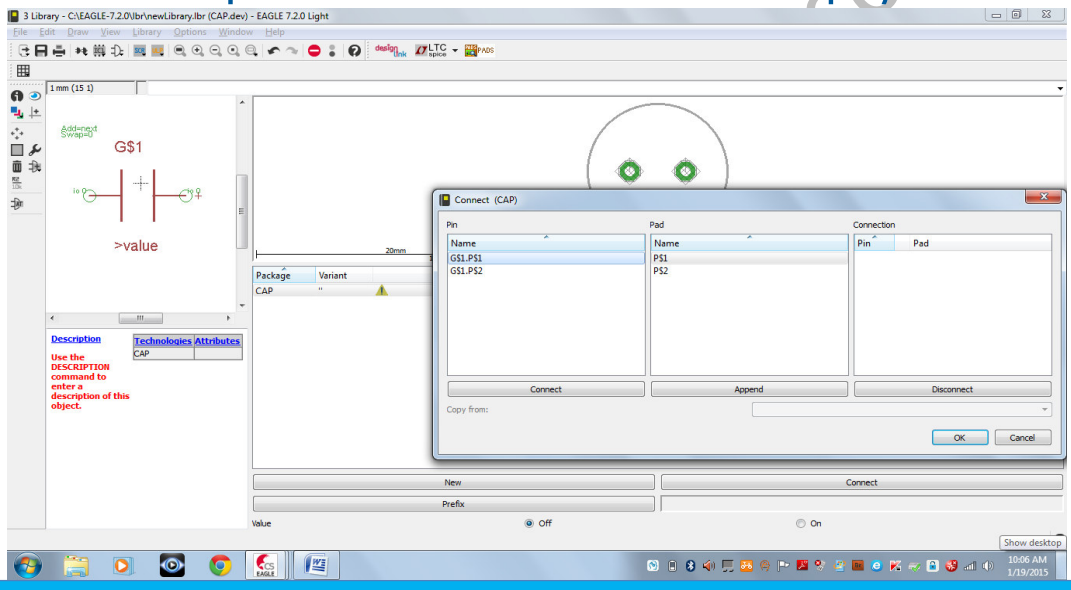
Step 25: Selected symbol is shown in slide.



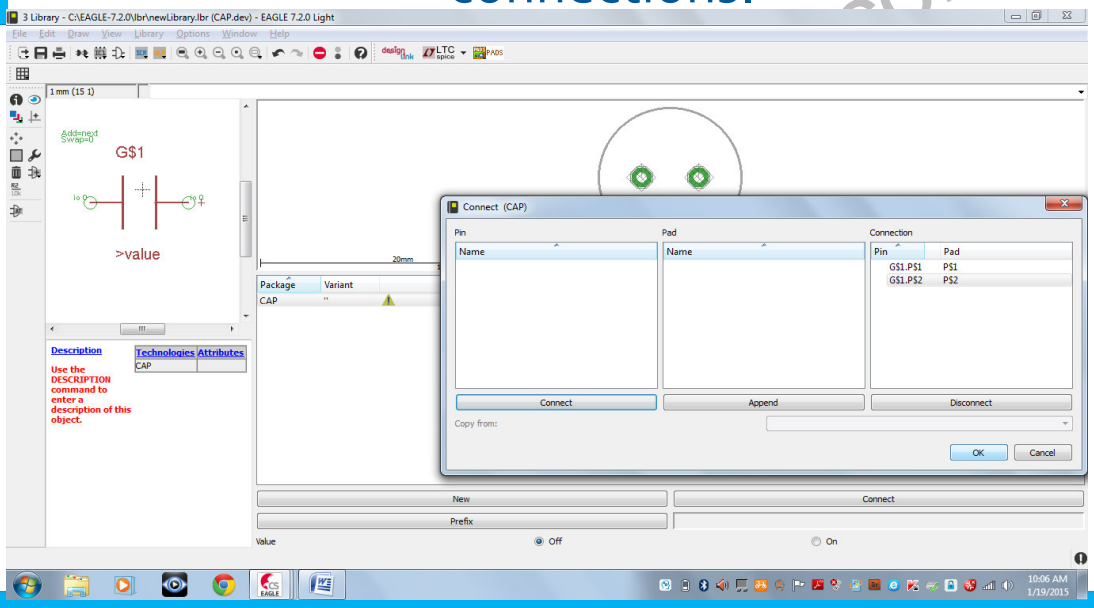
Step 26: Click '**new**' button and select package.



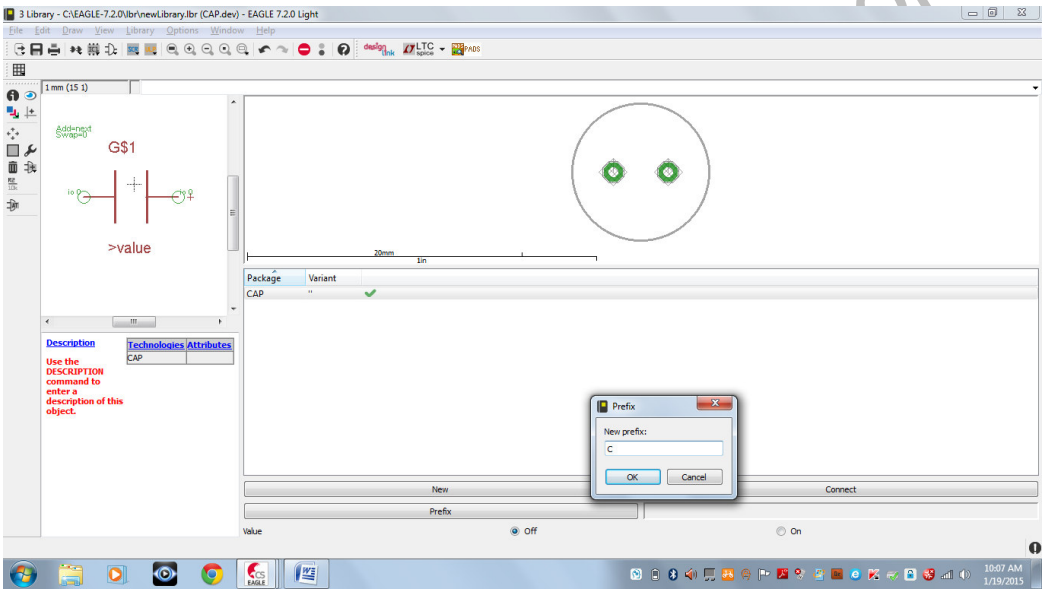
Step 27: click '**connect**' button till the Pin & pad columns become empty.



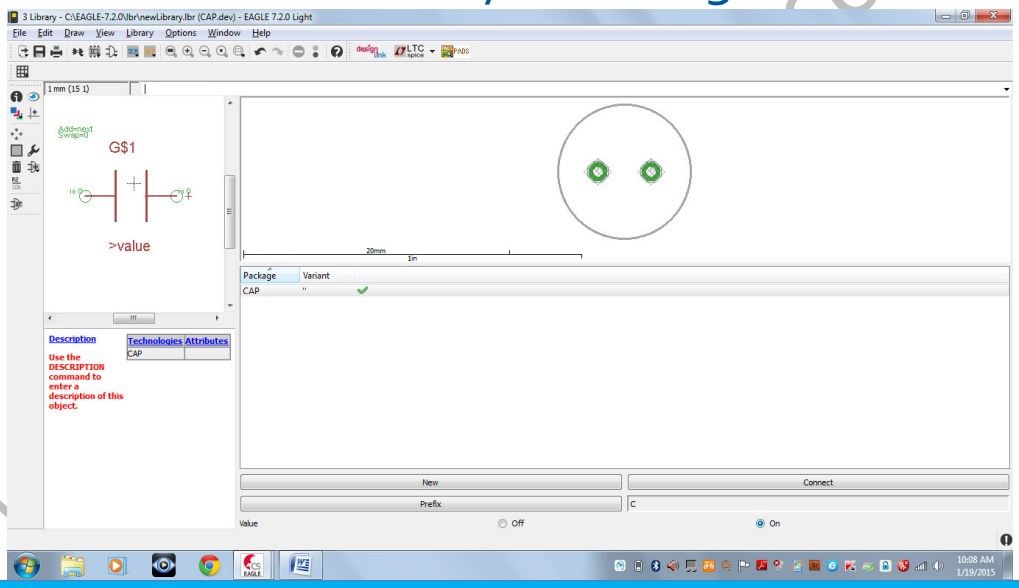
Step 28: Select '**OK**' button to finish the connections.



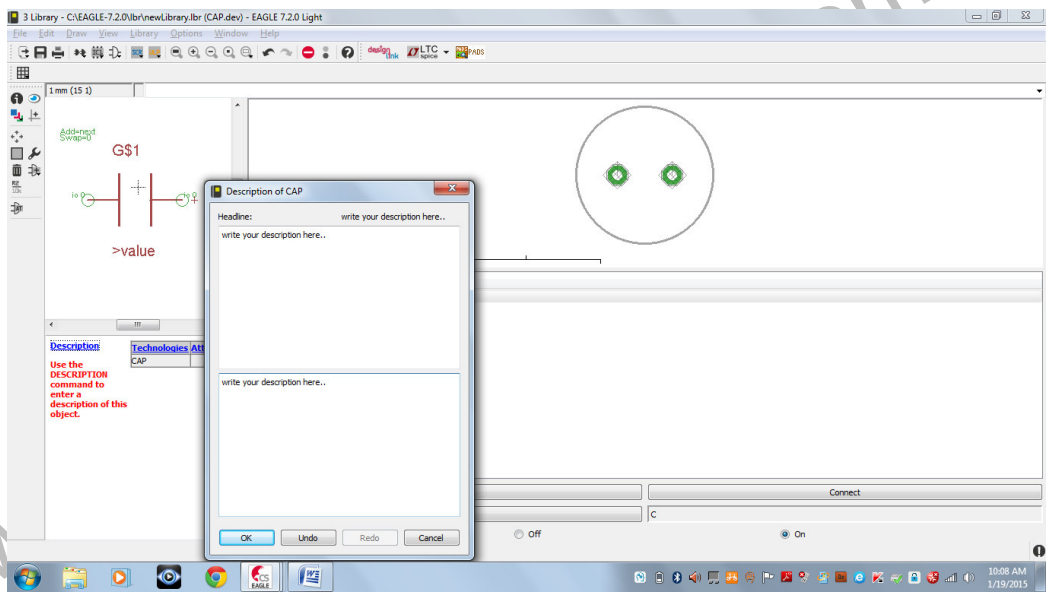
Step 29: Enter '**Prefix**'.



Step 30: Enable the Value by selecting **on**, disable by selecting **off**.

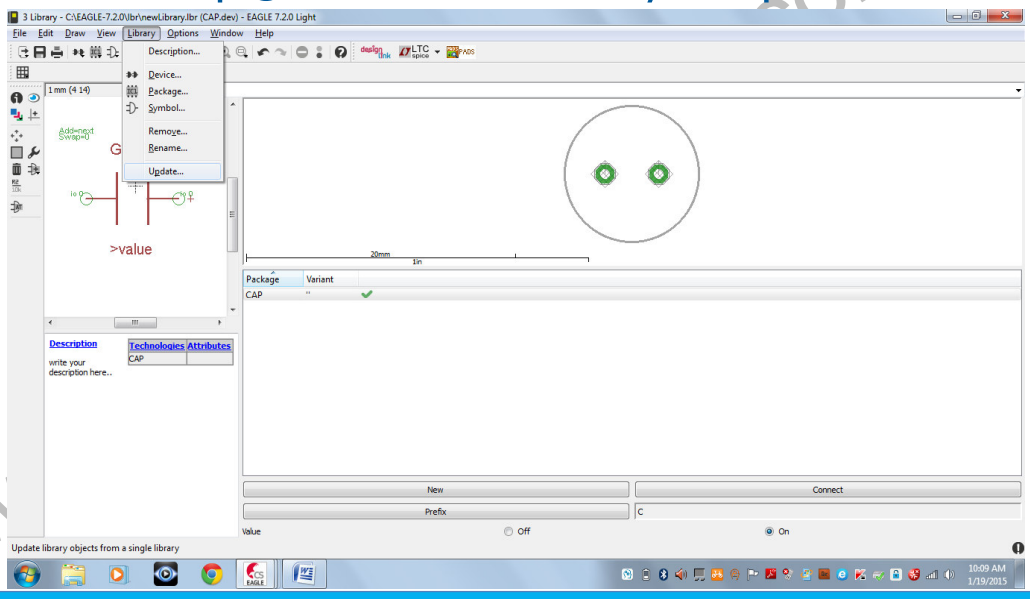


Step 31: Enter the Device description.

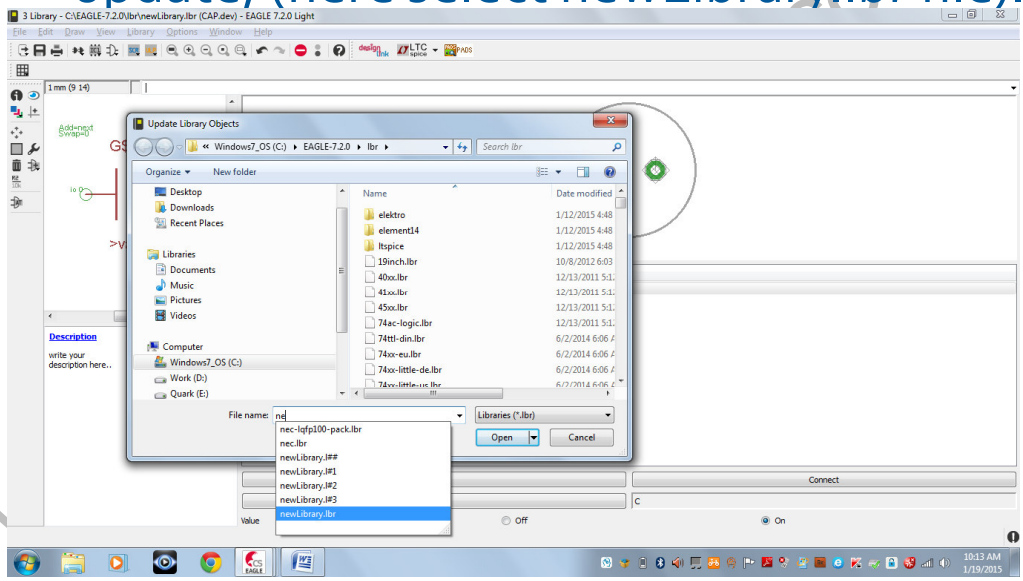


Update new Library:

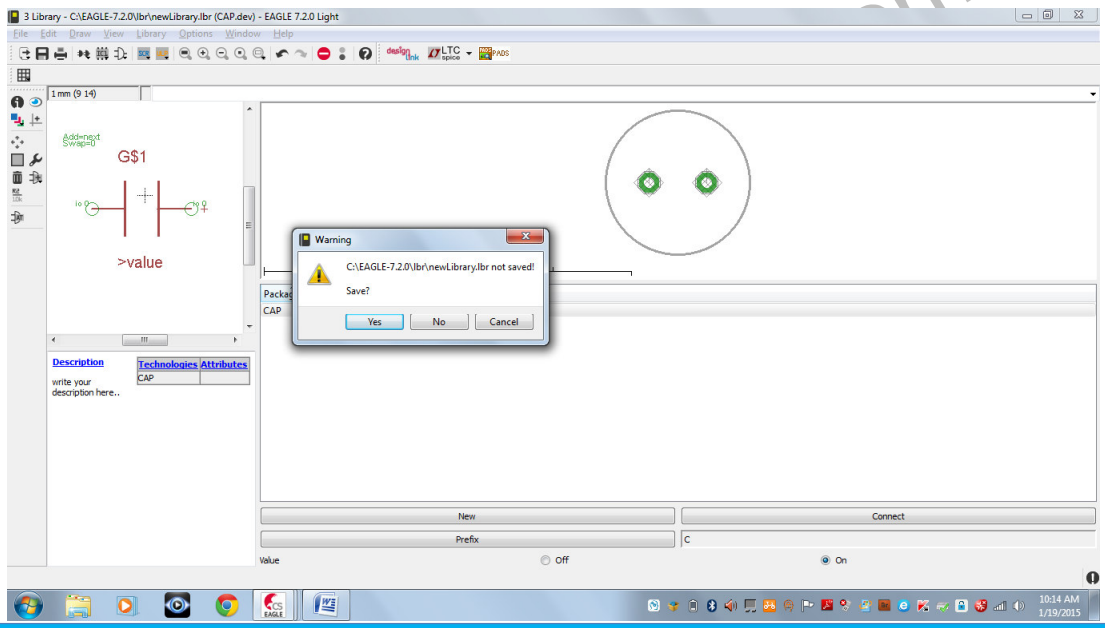
Step 32: Select Library > Update



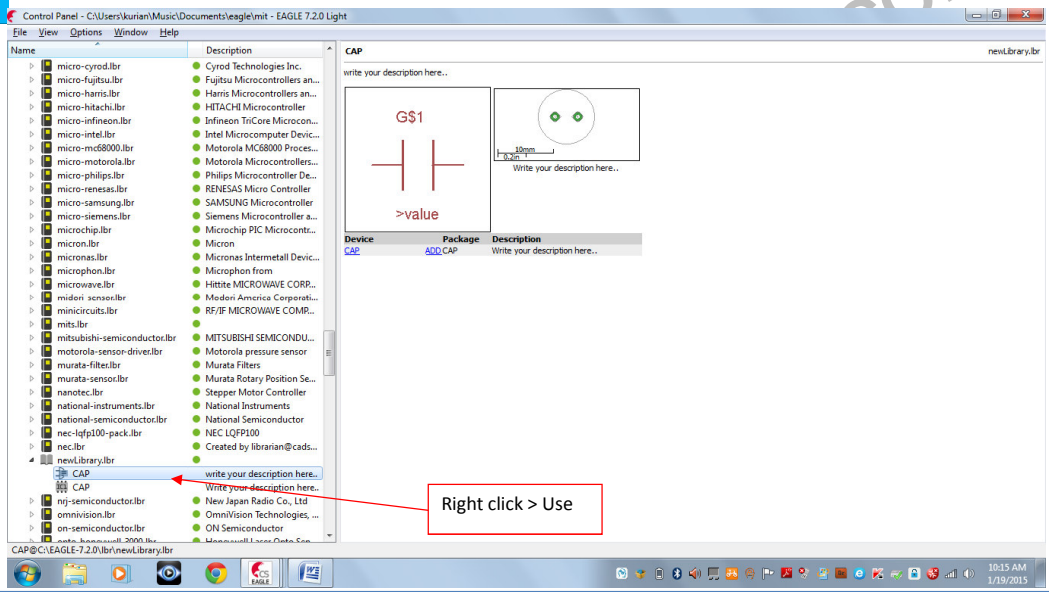
Step 33: Select the Library name and update, (here select newLibrary.lbr file).



Step 34: **Save** the changes in Library.



Step 35: Search the new library and enable it to **use**.

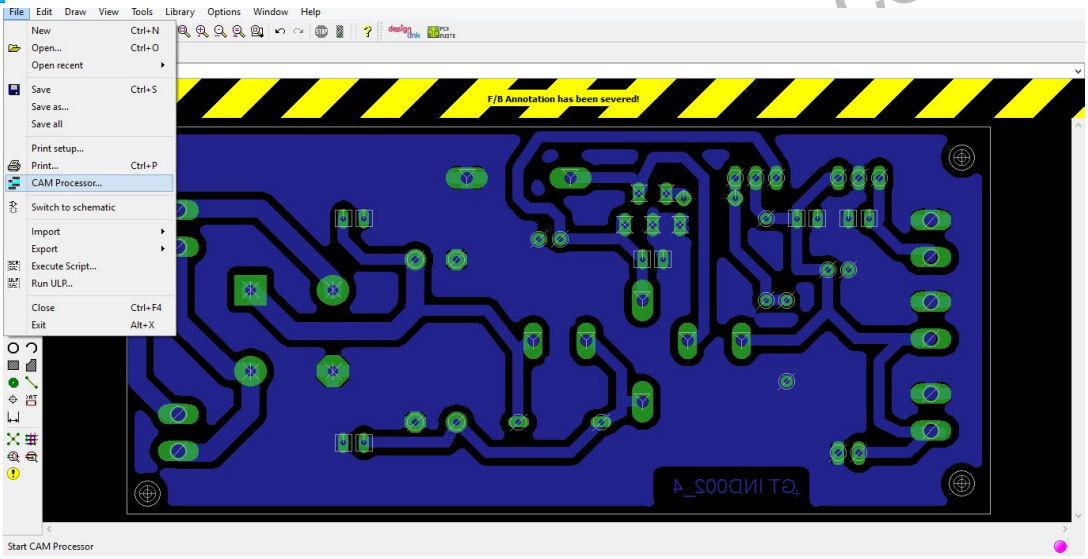


Gerber File in Eagle

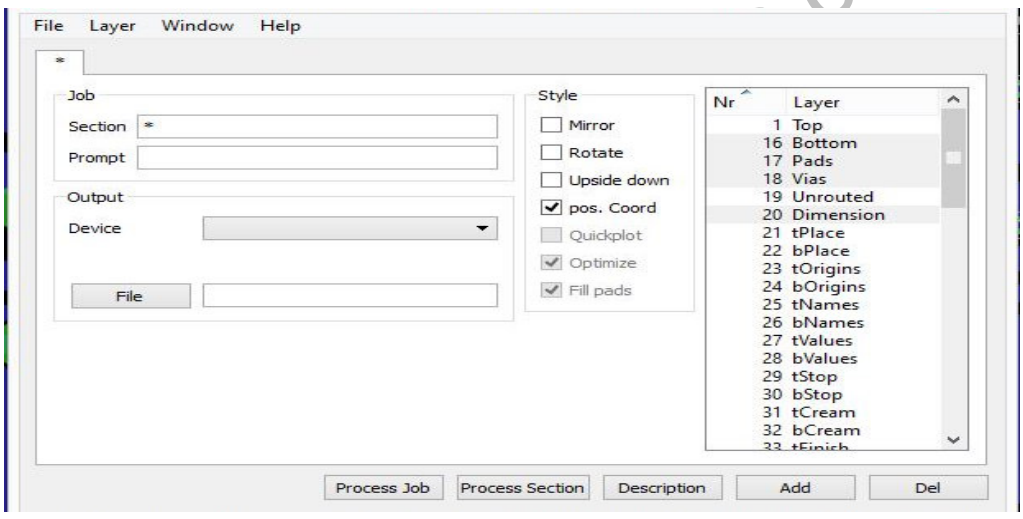
What is a Gerber file?

- Open 2D bi-level, vector image file format
- To describe the printed circuit board images: copper layers, solder mask, legend, etc
- two major generations of Gerber format
  - Gerber, or RS-274X
  - Gerber, or RS-274D

Step 1:file>CAM processor



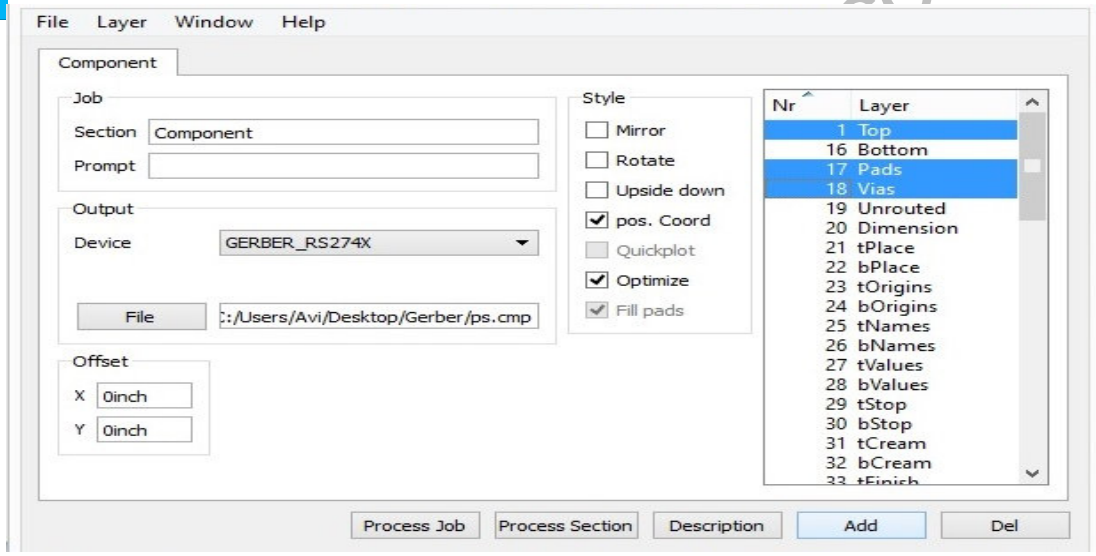
CAM Processor window



Step 2:Component Layer Creation

- In Device select GERBER_RS274X
- Go to Layer select Deselect all option.
- In section type Component
- In file navigate to the location where you want to store the Gerber file, name as projectName.cmp.
- In right side Layer window select Top, Pads and Vias.
- Click Add in the bottom.

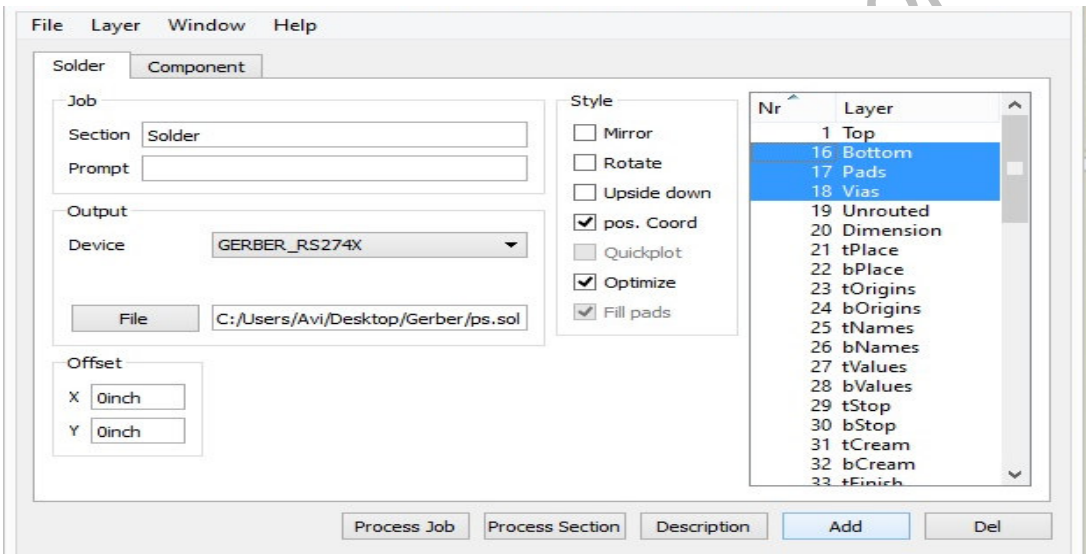
Component Window



Step 3:Solder Layer Creation

- In section type Solder
- Go to Layer select Deselect all option.
- In file navigate to the location where you want to store the Gerber file, name it projectName.sol.
- In right side Layer window select Bottom, Pads and Vias.
- Click Add in the bottom.

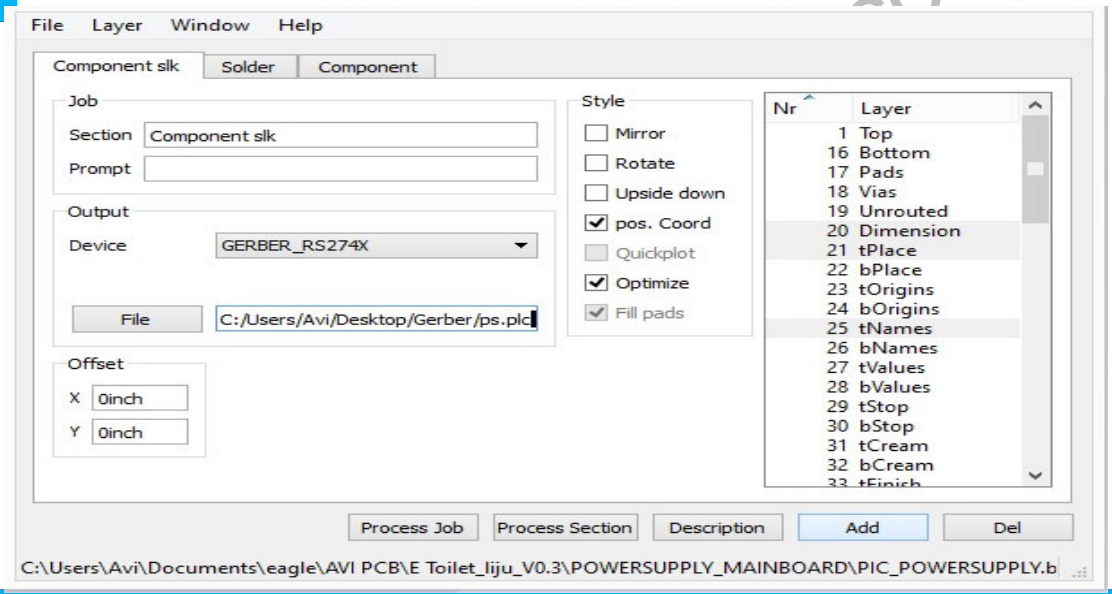
Solder window



Step 4: Component Slk Layer Creation

- In section type Component Slk
- Go to Layer select Deselect all option.
- In file navigate to the location where you want to store the Gerber file, name it projectName.plc.
- In right side Layer window select Dimension, tPlace and tNames.
- Click Add in the bottom.

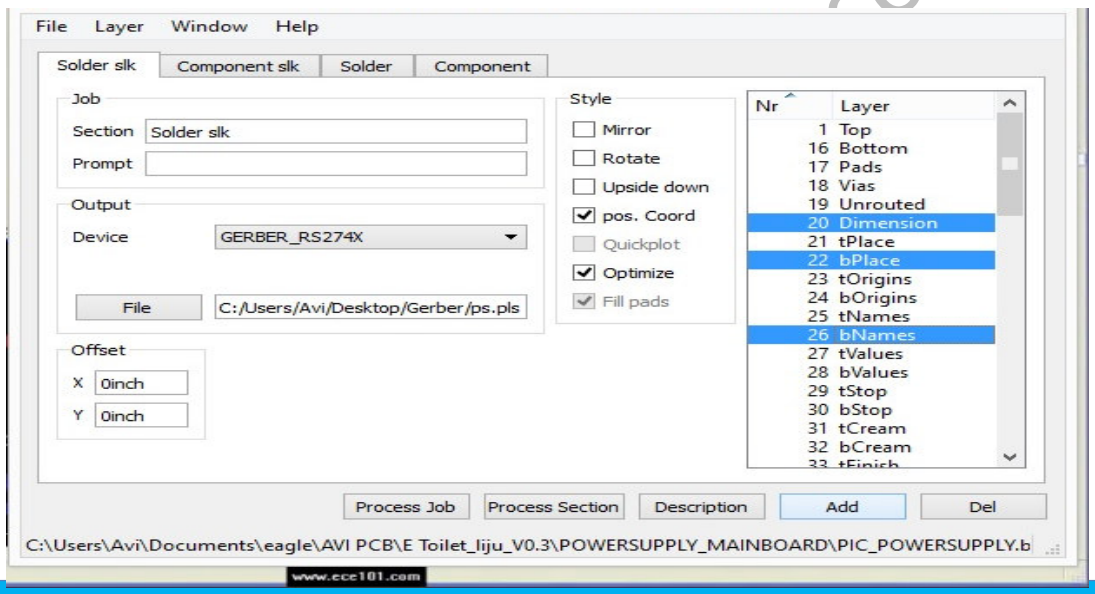
Component slk window



Step 5: Solder slk Layer creation

- In section type Solder Slk
- Go to Layer select Deselect all option.
- In file navigate to the location where you want to store the Gerber file, name it projectName.pls.
- In right side Layer window select Dimension, bPlace and bNames.
- Click Add in the bottom.

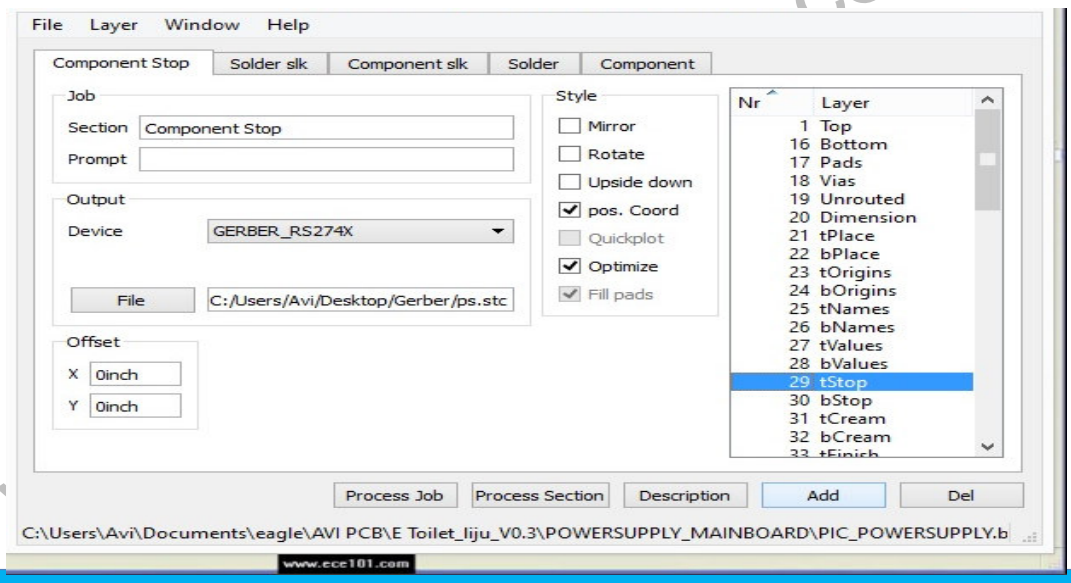
Solder slk window



Step 6:Component stop Layer creation

- In section type Component Stop
- Go to Layer select Deselect all option.
- In file navigate to the location where you want to store the Gerber file, name it projectName.stc.
- In right side Layer window select tStop.
- Click Add in the bottom.

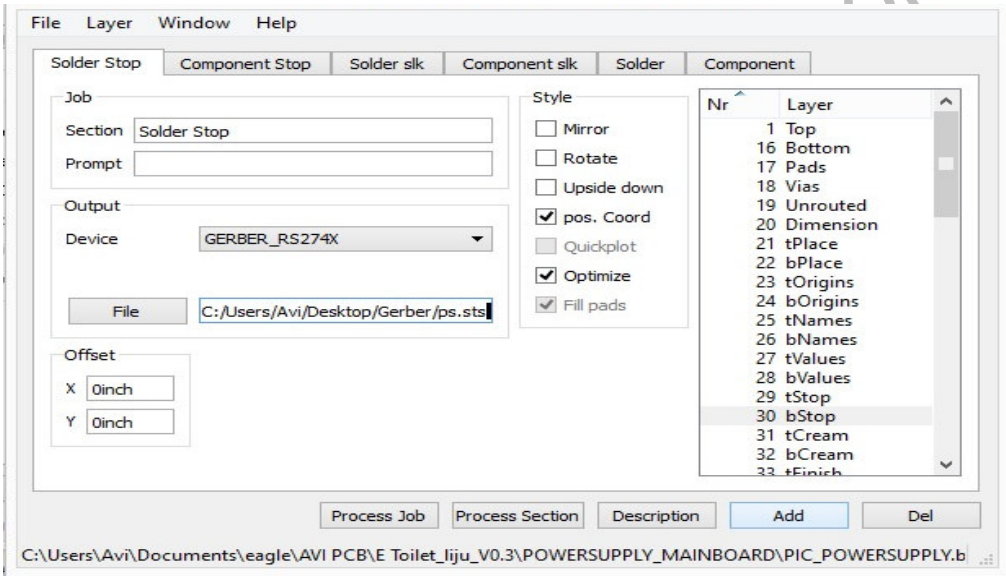
Component stop window



Step 7: Solder stop Layer creation

- In section type Solder Stop
- Go to Layer select Deselect all option.
- In file navigate to the location where you want to store the Gerber file, name it projectName.sts.
- In right side Layer window select bStop.
- Click Add in the bottom.

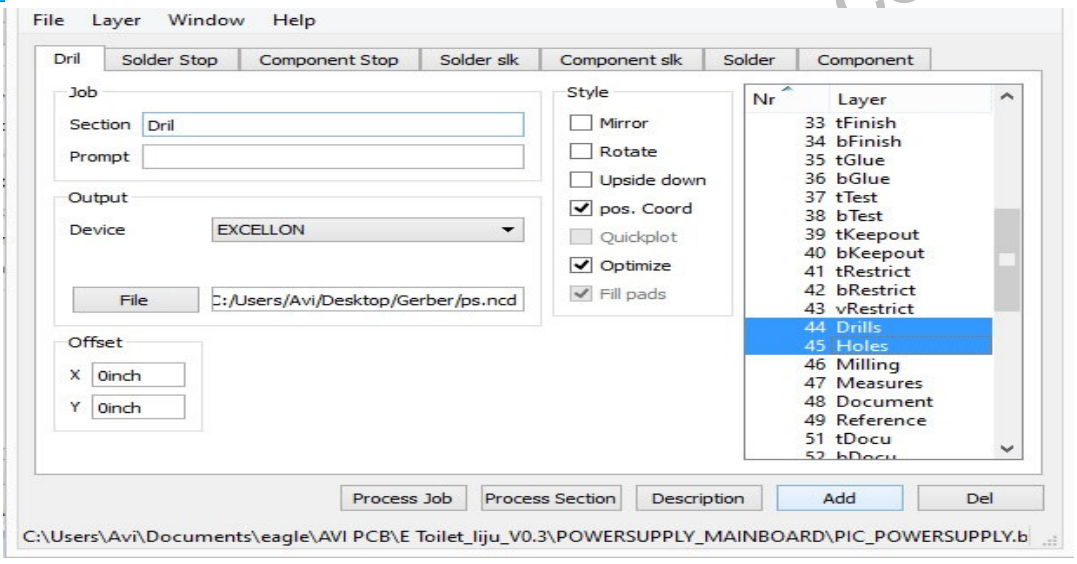
Solder stop window



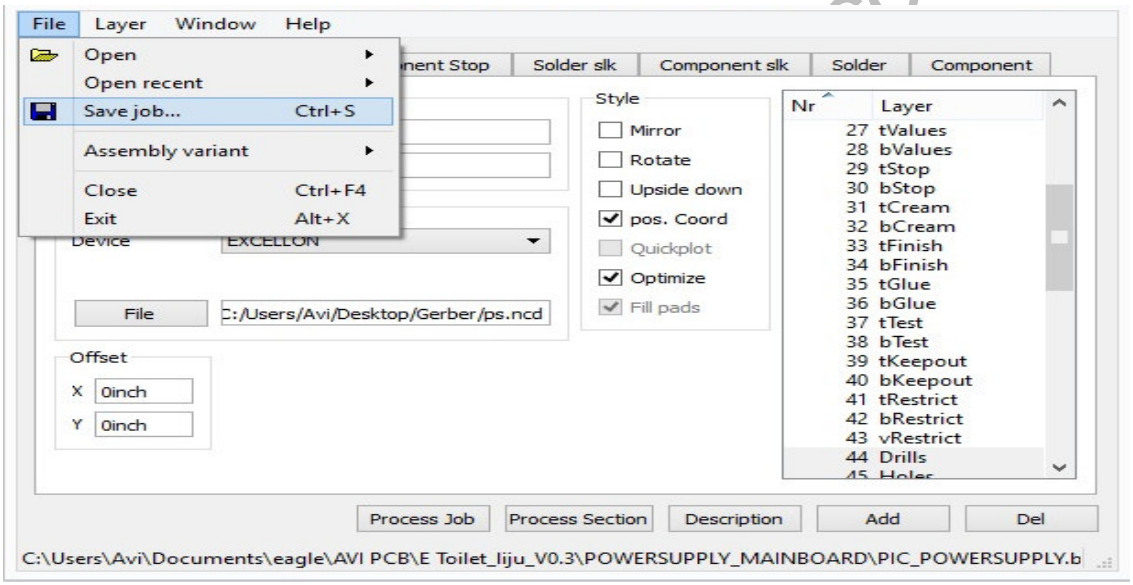
Step 8: Excellon Layer creation

- Change Device to EXCELLON
- Go to Layer select Deselect all option.
- In section type Drill
- In file navigate to the location where you want to store the Gerber file, name it projectName.ncd.
- In right side Layer window select Drills and Holes.
- Click Add in the bottom.

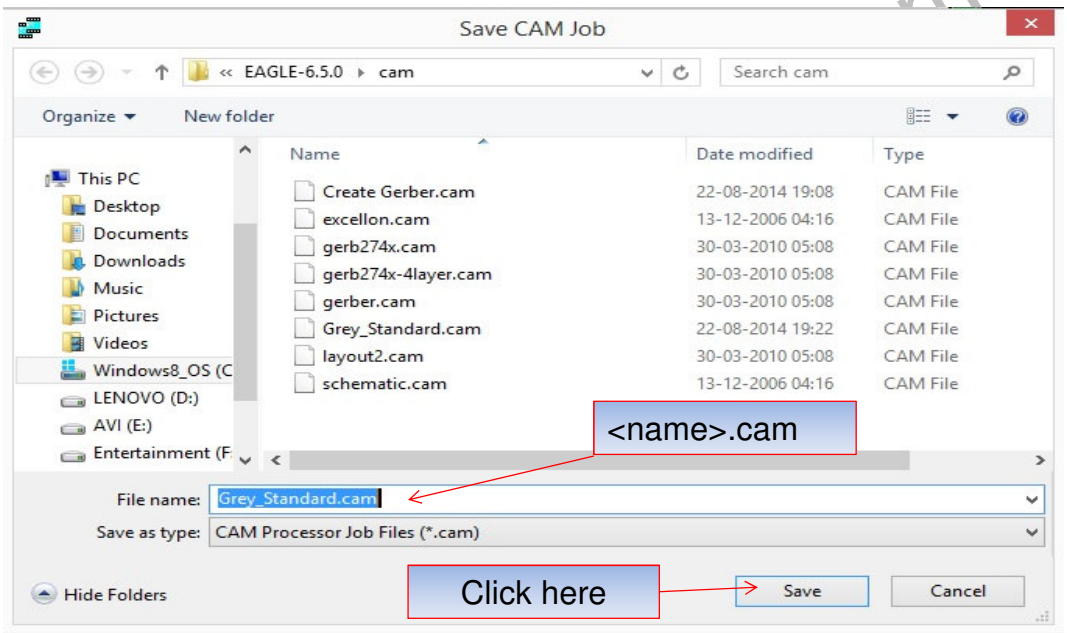
Excellon window



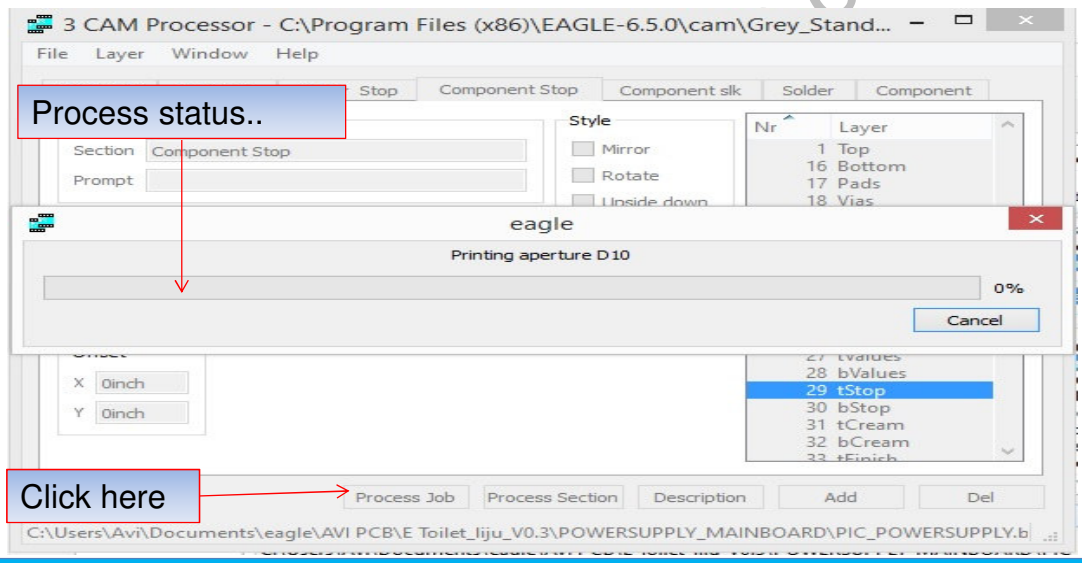
Step 9: File >Save Job



Save CAM Job



Step 10: Click the **Process Job** button in the bottom

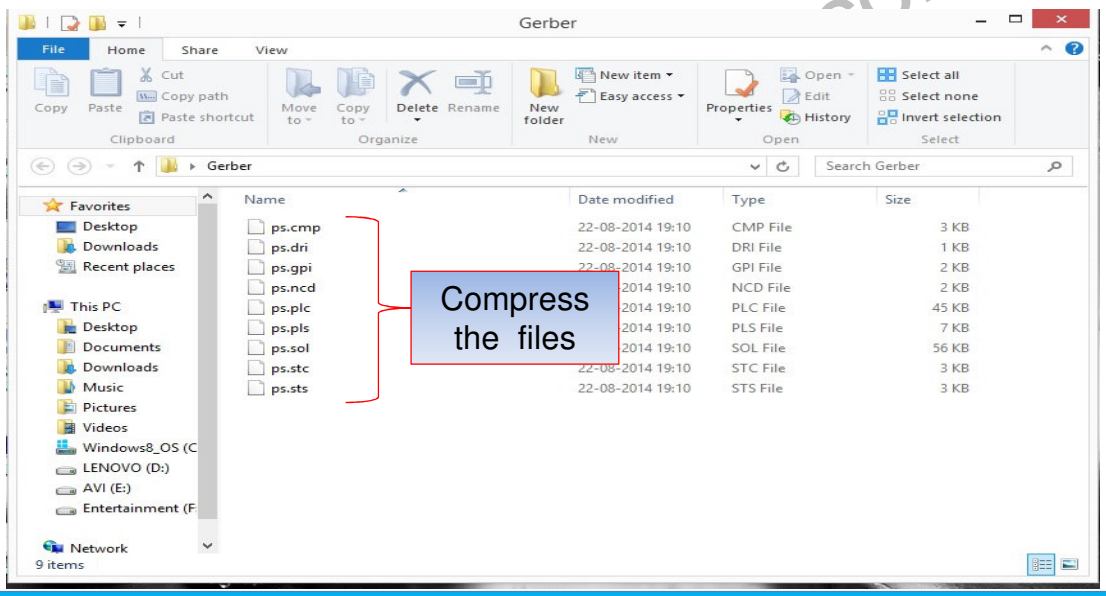


That's it, You are done.

Now open and see the Folder where you stored the files.

You can see files with extension .cmp, .sts etc.

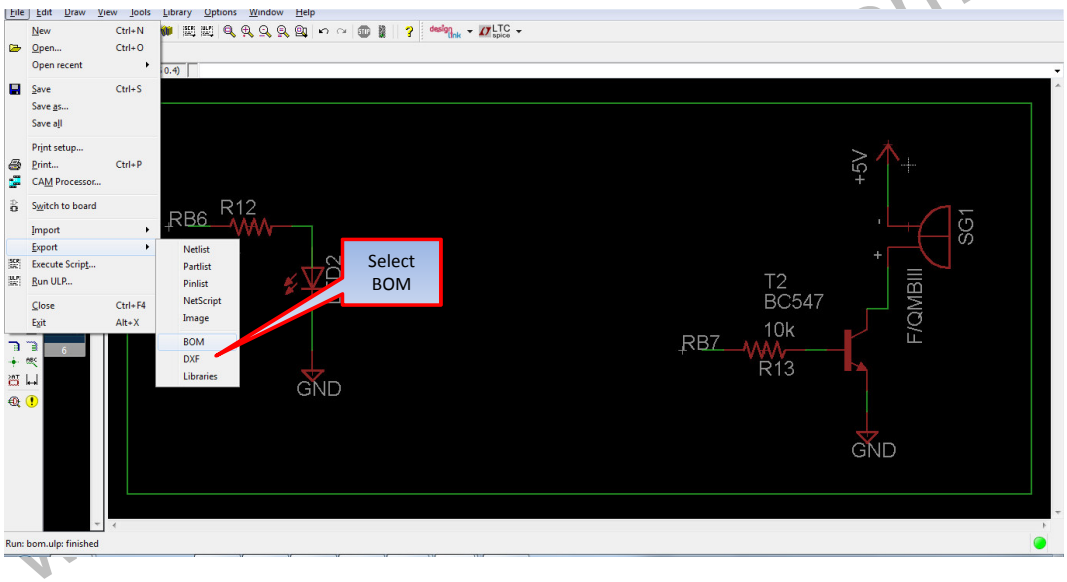
Step 11:select all>compress the files.zip



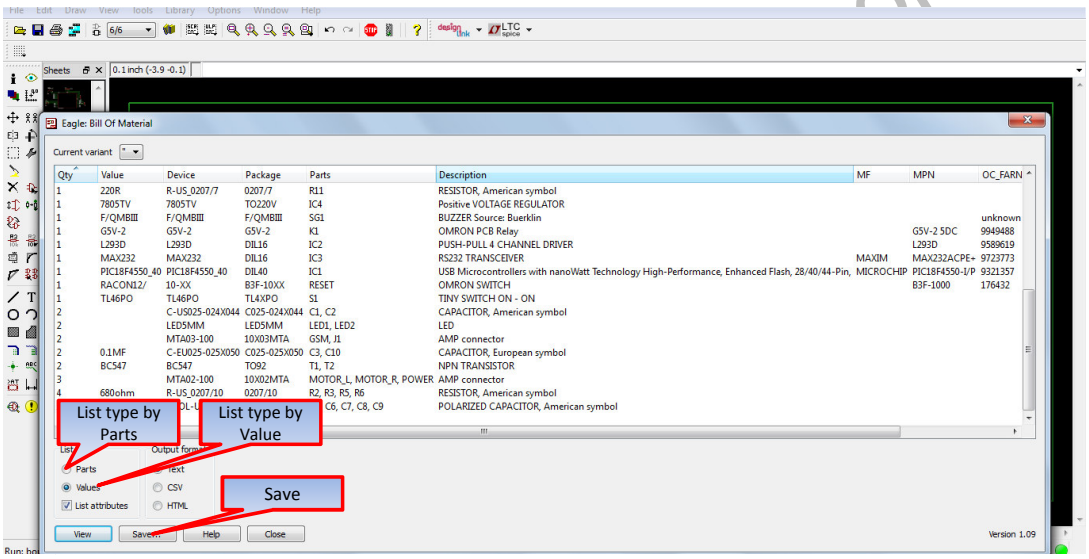
Step 12:Upload your zip

Bill Of Material -BOM

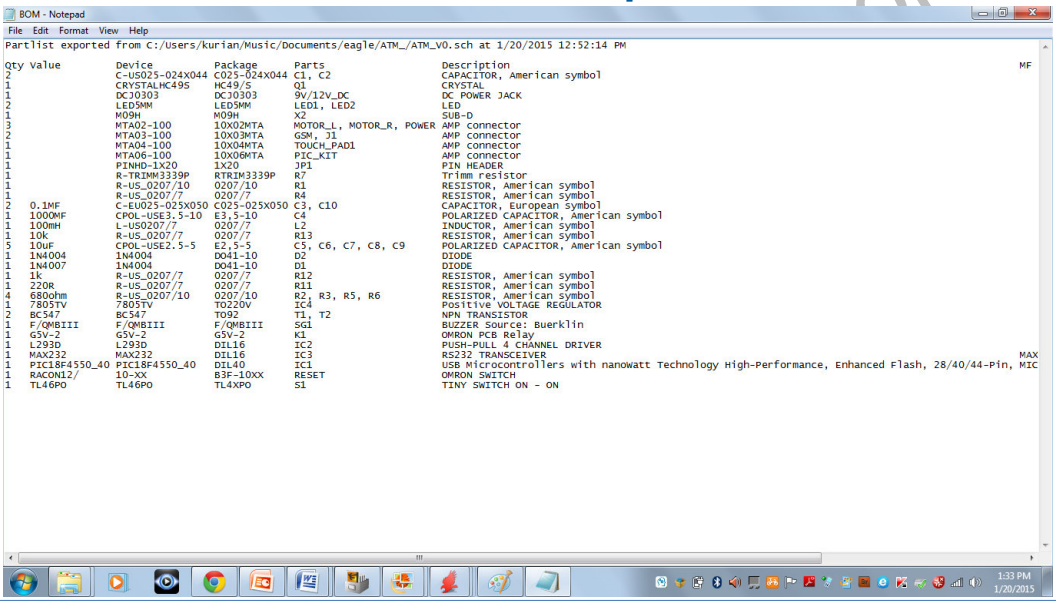
Step 1: File>Export>BOM



Step 2:Save the file

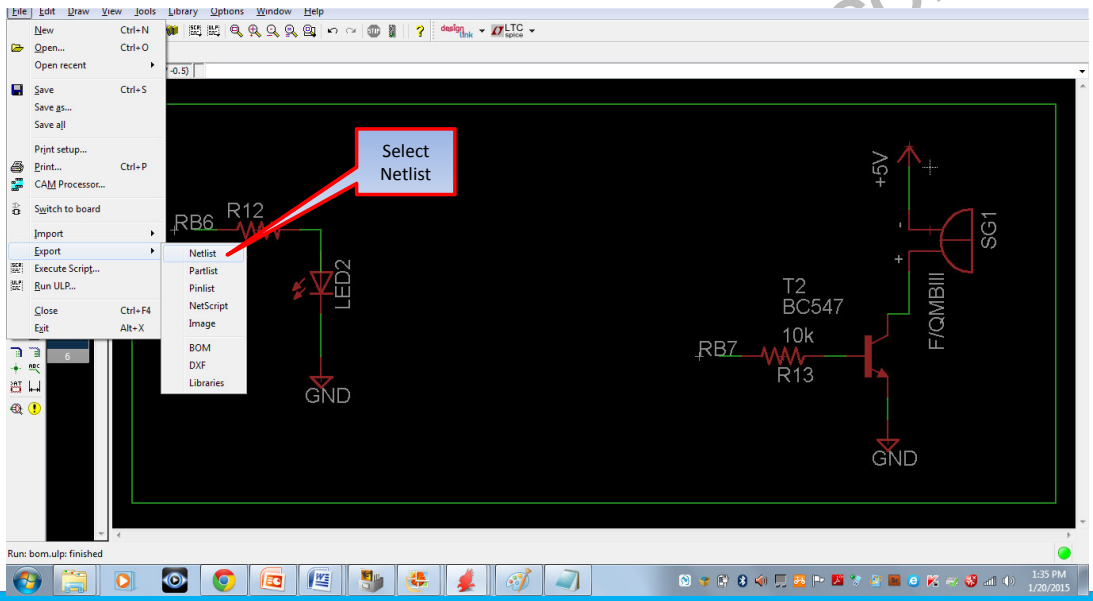


Example

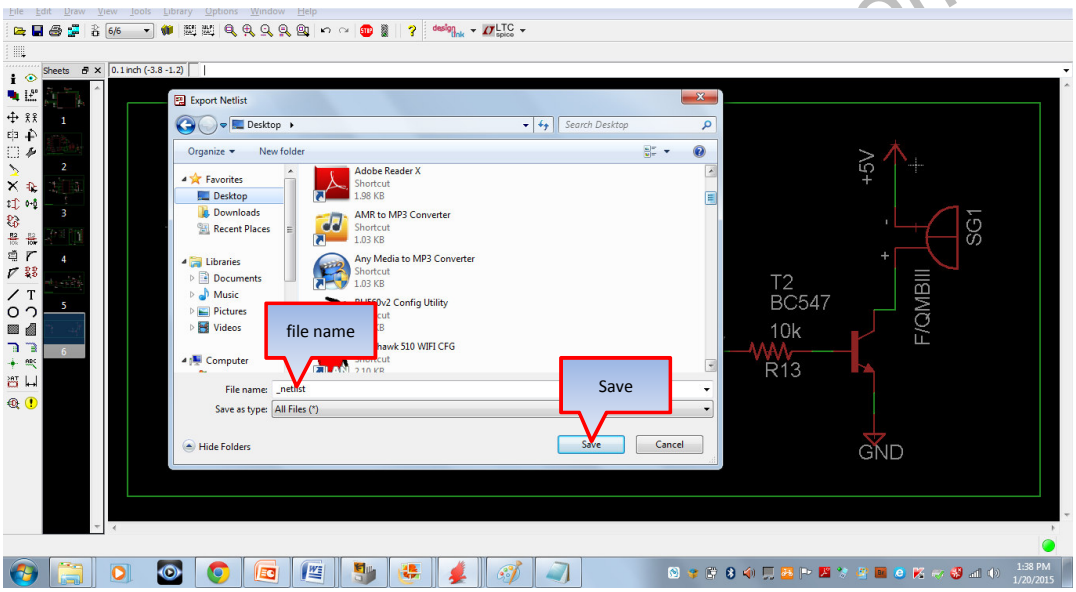


Netlist

Step 1:file>Export>Netlist



Step 2: Save the file



Example