



PCB Design

www.greytechnolabs.com

www.greytechnolabs.com



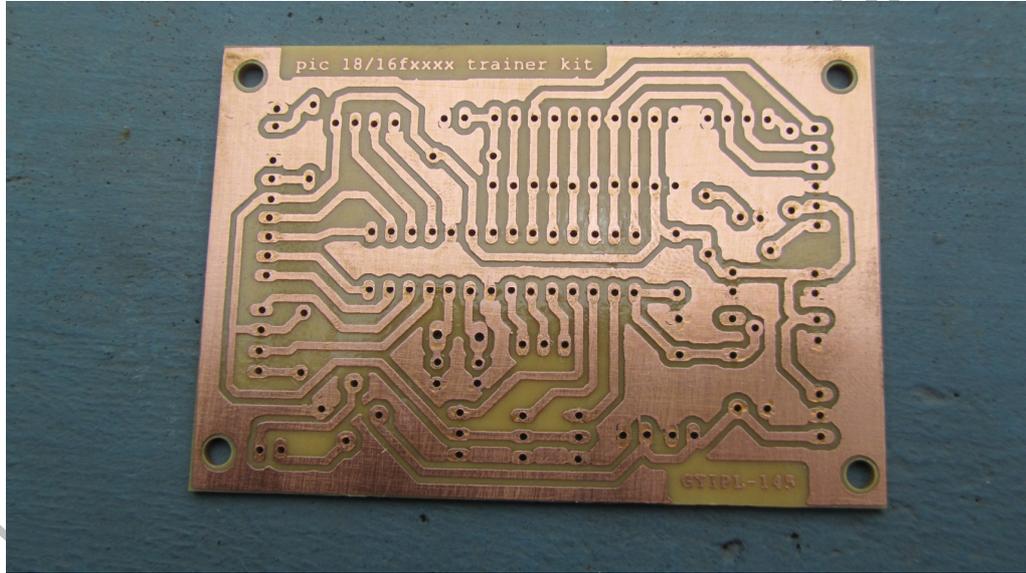
Introduction

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



www.!



- 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

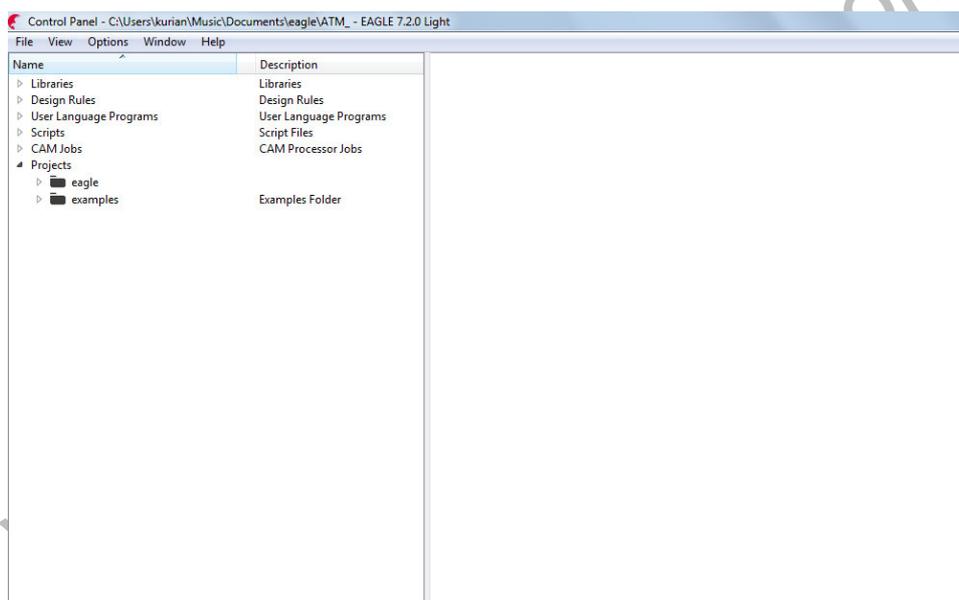
- **Control Panel:** is the main window, it launches everything else and when you close it, all subordinate windows get closed.
- **Schematic:** where you draw the schematic for your project.
- **Board:** where you lay out the pieces of your project and physically connect the correct pins as defined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



www.greynotech.com



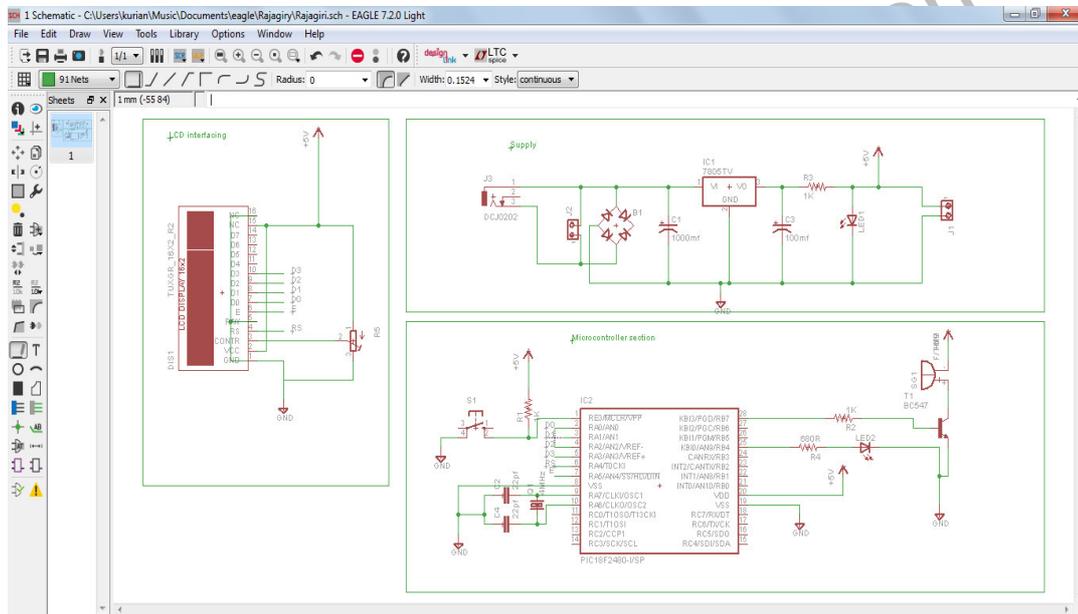
Control Panel



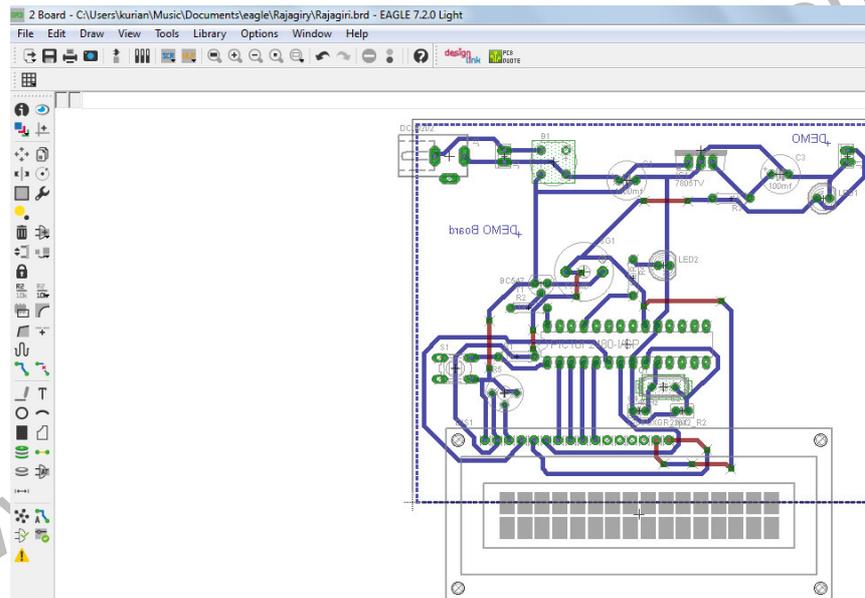
www.greynotech.com



Schematic Window



Board Window





Familiarization of Eagle Lite



www.greytechnolabs.com



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



www.greytechnolabs.com



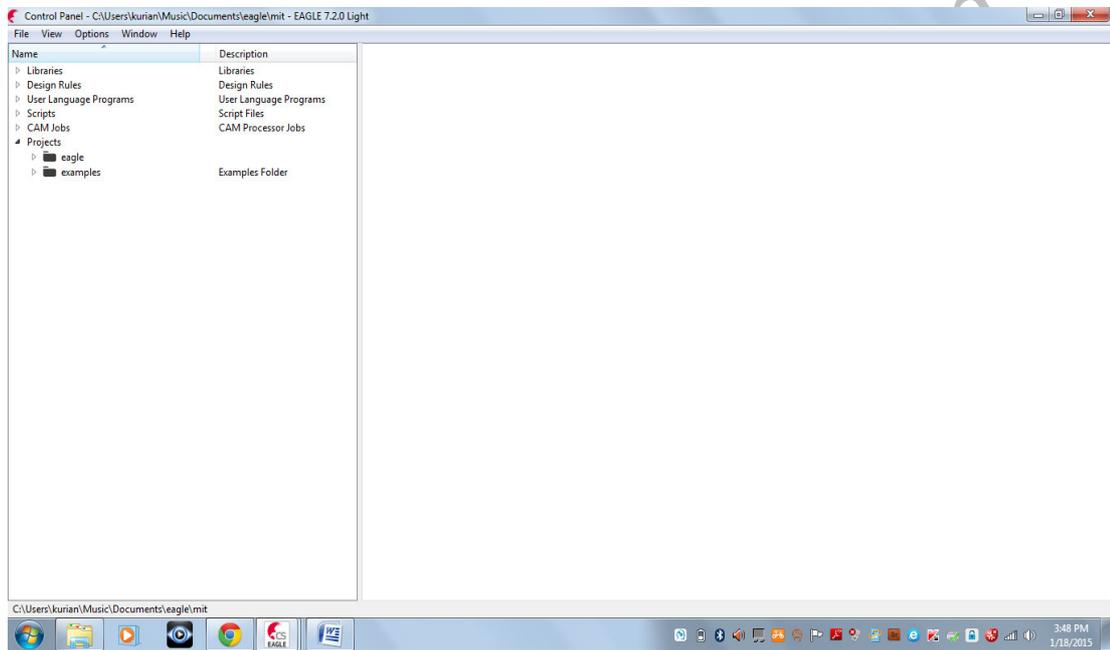
Part-2

- Board Layout
- The Design Rule Check
- DRC Results

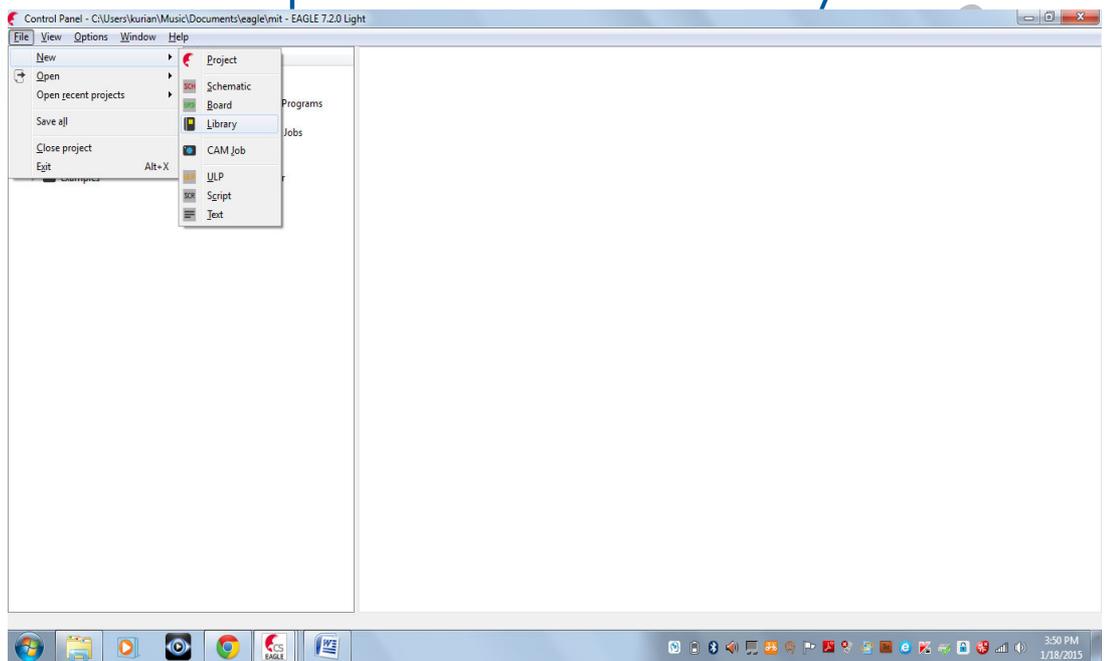


CREATE NEW LIBRARY IN EAGLE

Step 1: Go to control Panel

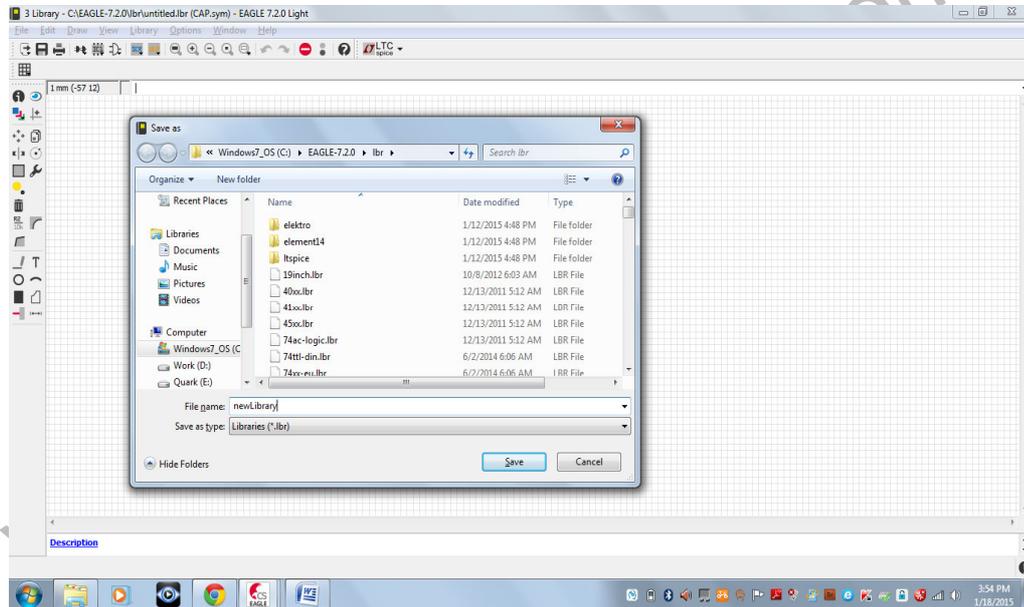


Step 2: Create new Library file

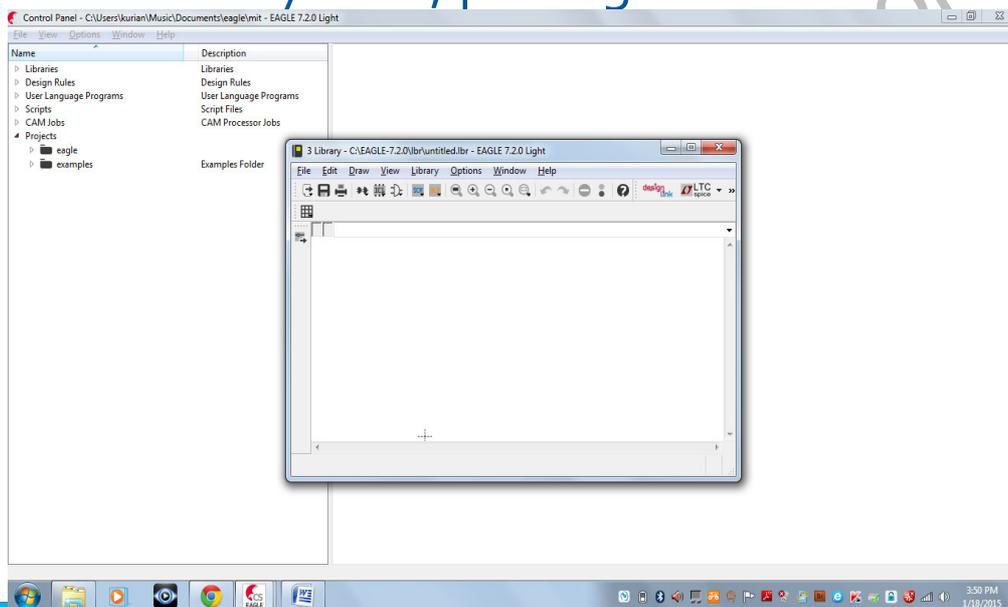




Step 3: Give a **name** to your Library, (here file name set as **newLibrary**)

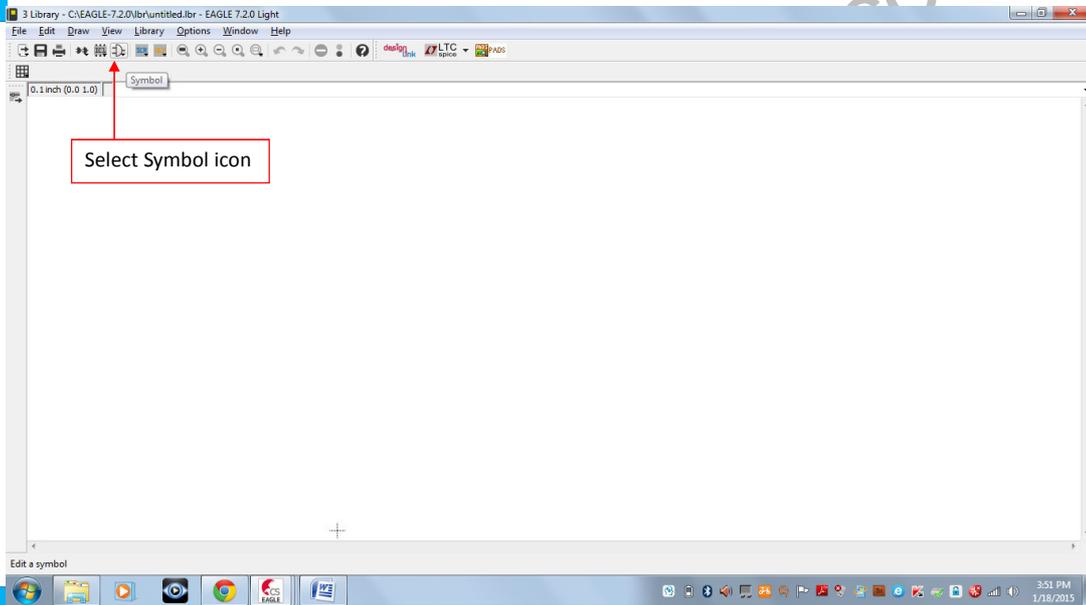


Step 4: Here you need to create new symbol, package and Device

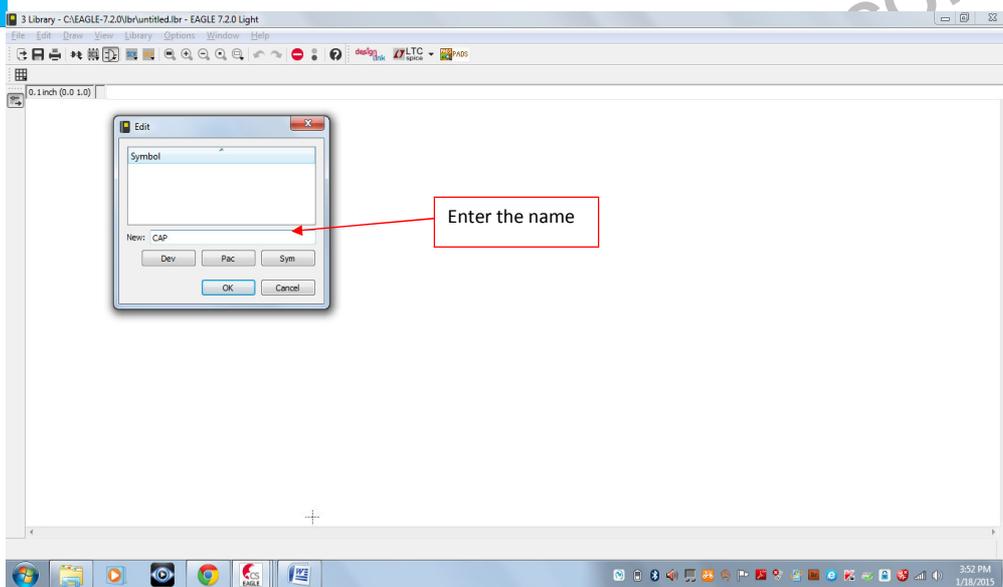




Edit Symbol: Step 5: Select symbol icon

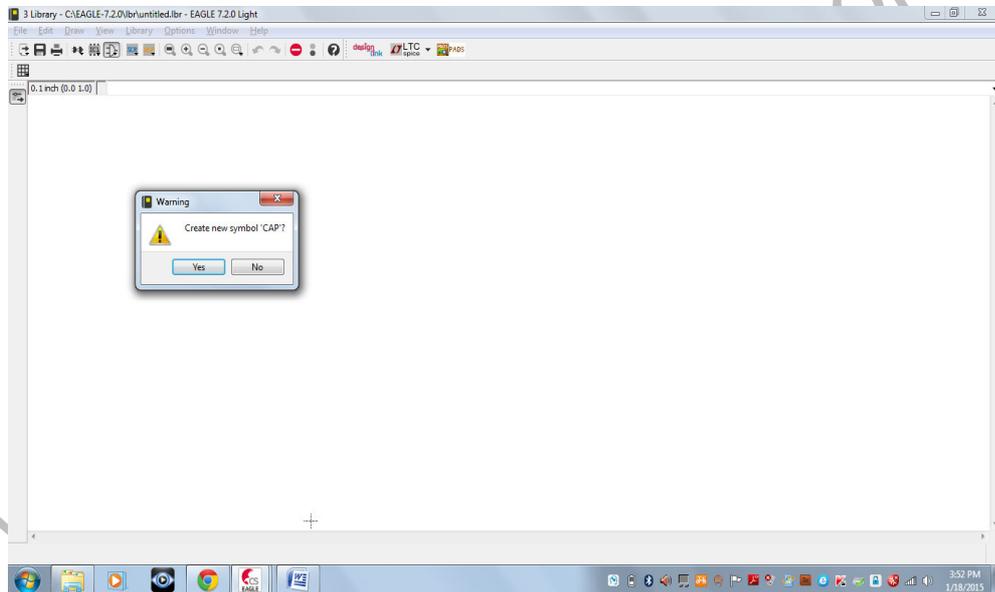


Step 6: Enter the name of symbol

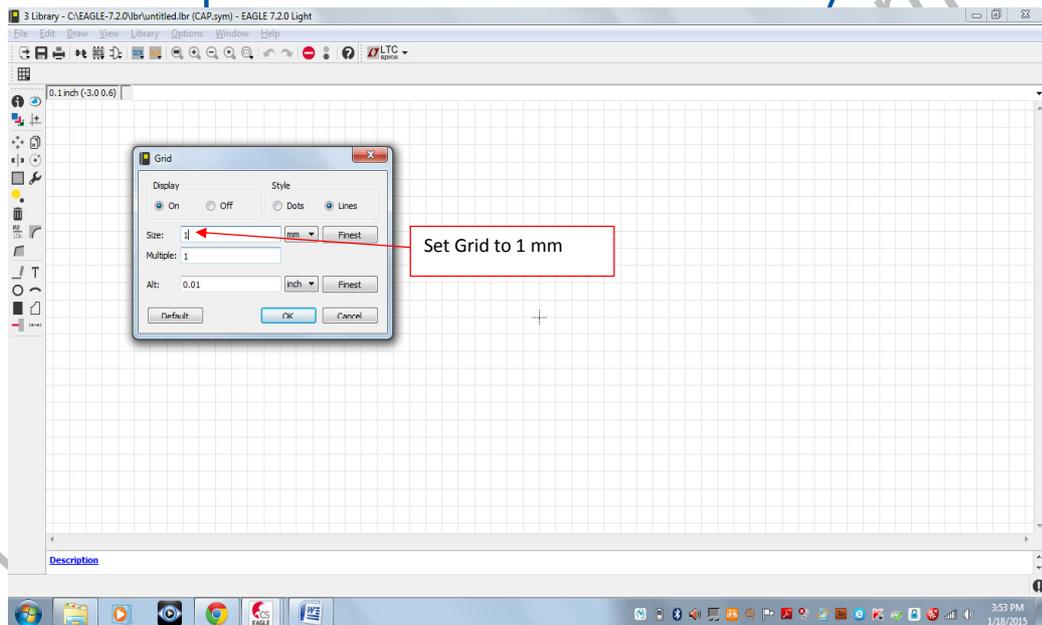




Step 7: Select 'yes' to close the warning window.

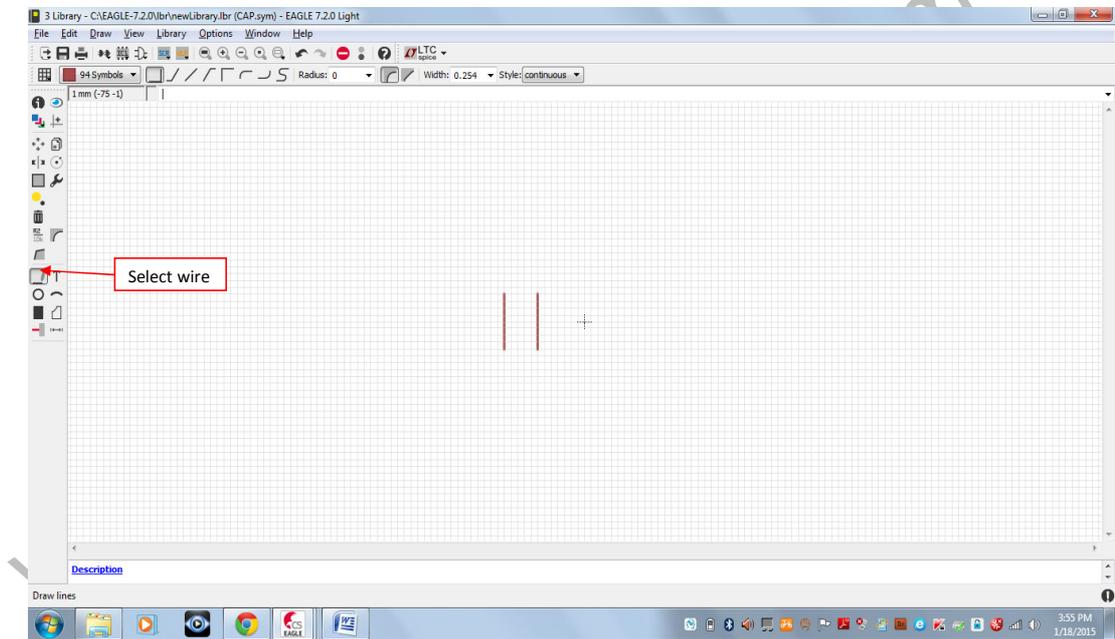


Step 8: Set **Grid** size for the Symbol.

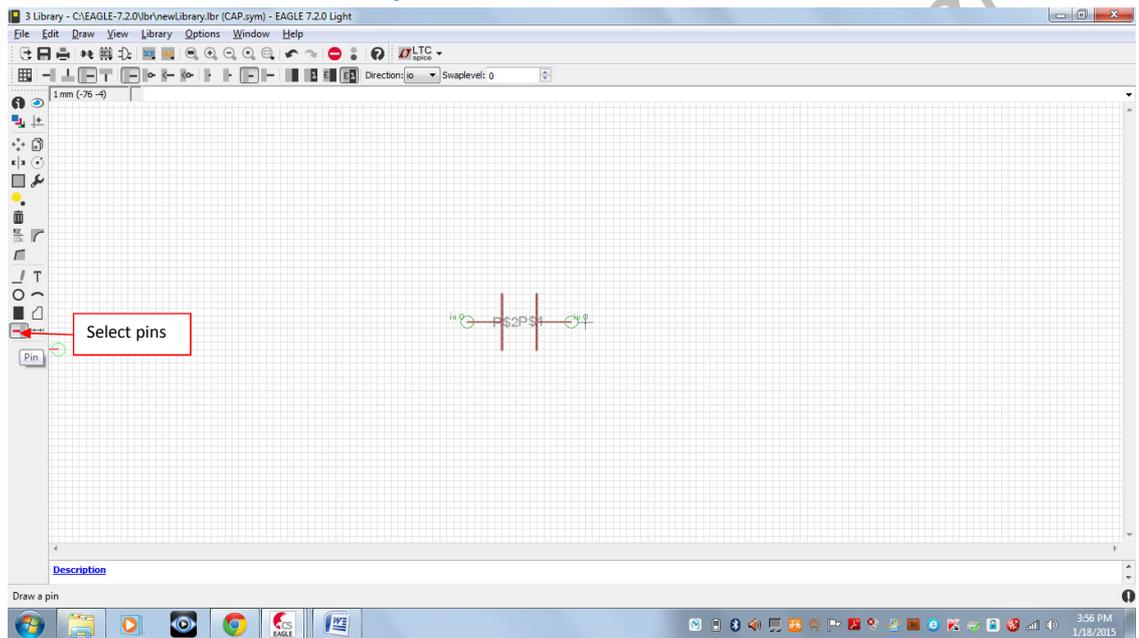




Step 9: Draw the schematic symbol

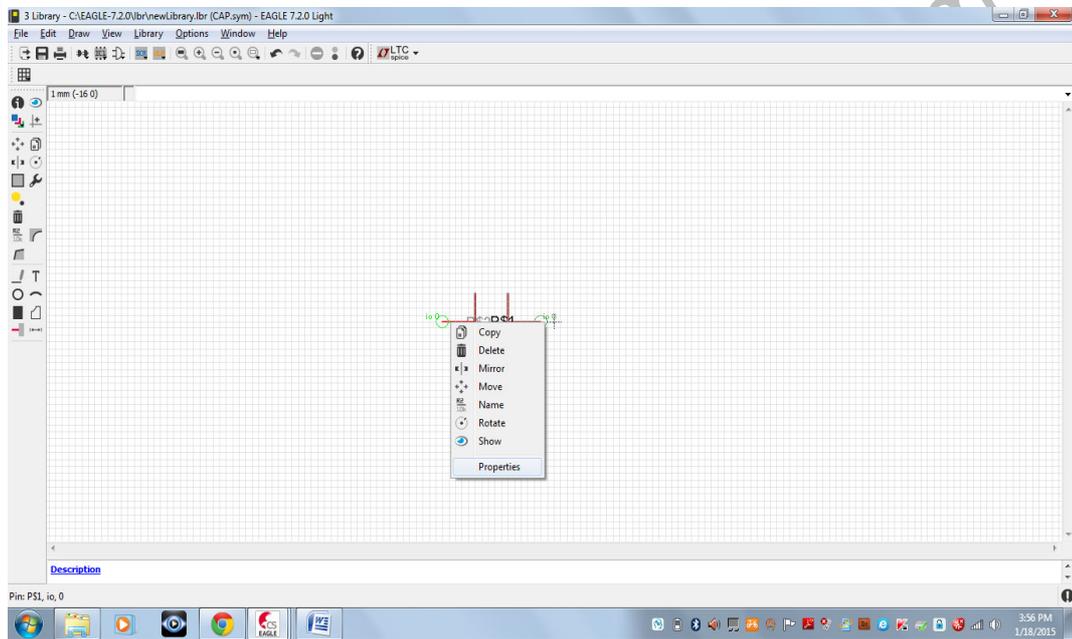


Step 10: connect Pins

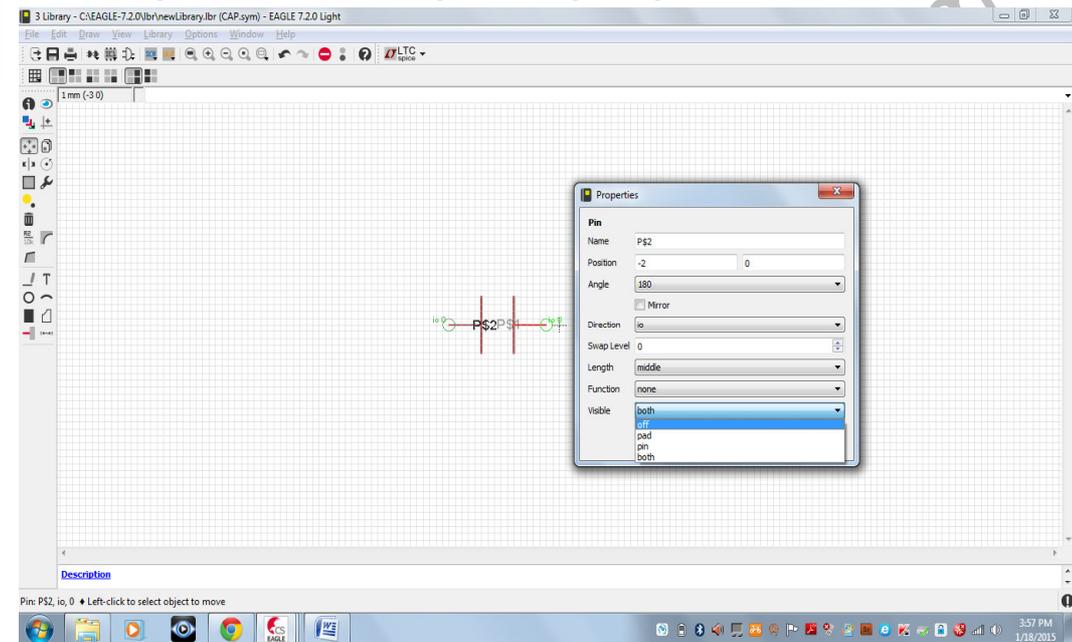




Step 11: select pin1>properties > visible>off

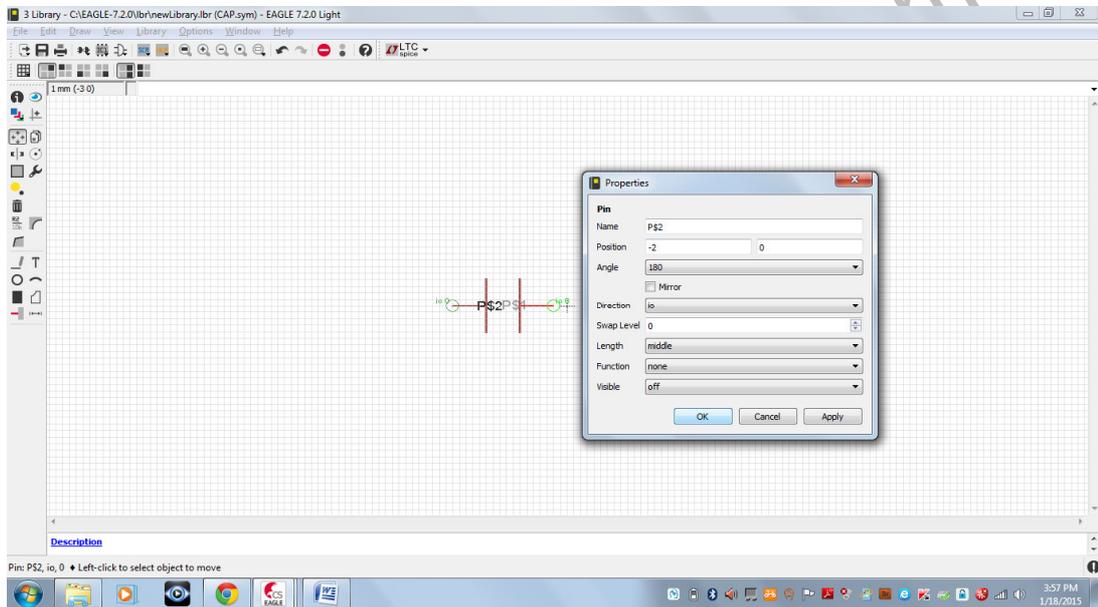


Step 12: select pin2>properties > visible>off



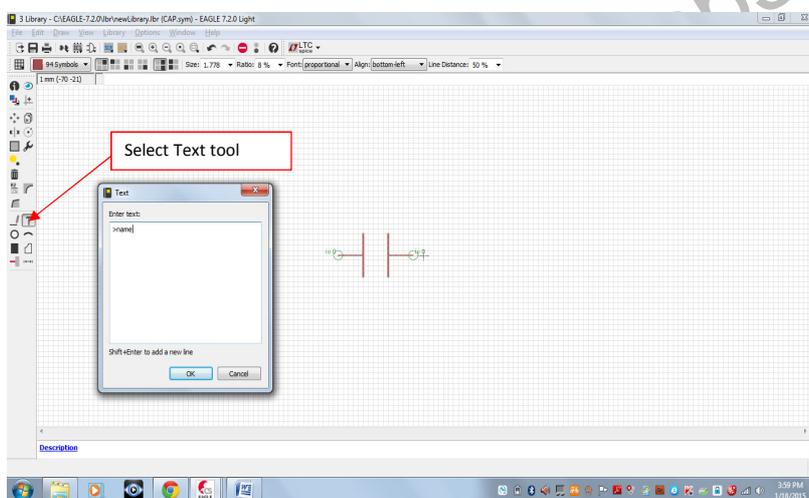


Step 12.a:



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

- Step 13.a: select Text Tool



Step 13.b:

3 Library - C:\EAGLE-7.2.0\lib\newLibrary.lib (CAP.sym) - EAGLE 7.2.0 Light

File Edit Draw View Library Options Window Help

94 Symbols Size: 1.778 Ratio: 8 % Font: proportional Align: bottom-left Line Distance: 50 %

1 mm (-77.5)

Text

Enter text:

>value|

Shift+Enter to add a new line

OK Cancel

>name

Description

3:59 PM 1/18/2015

Step 13.c:

3 Library - C:\EAGLE-7.2.0\lib\newLibrary.lib (CAP.sym) - EAGLE 7.2.0 Light

File Edit Draw View Library Options Window Help

94 Symbols Size: 1.778 Ratio: 8 % Font: proportional Align: bottom-left Line Distance: 50 %

1 mm (-29.32)

>name

>value

Description

Stop command

4:00 PM 1/18/2015

Step 14: Give description

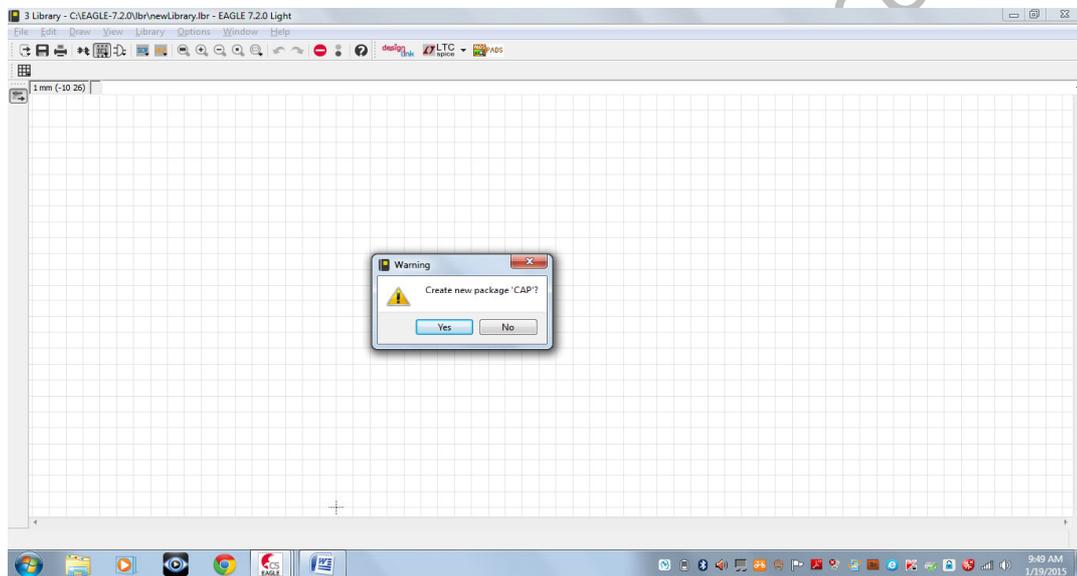
The screenshot shows the Eagle 7.2.0 Light interface. A schematic diagram of a capacitor is visible in the workspace, with labels >name and >value . A dialog box titled "Description of CAP" is open, containing two text input fields for description. A red box labeled "Click Description" points to the "Description" button in the bottom-left corner of the workspace.

Edit package: Step 15: Select package icon

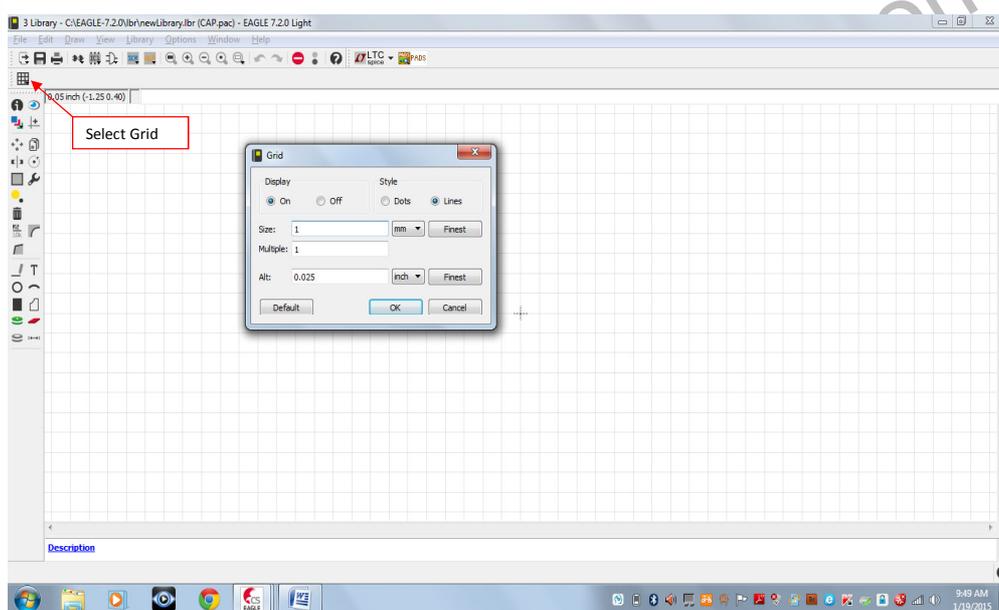
The screenshot shows the Eagle 7.2.0 Light interface. A red box labeled "Select Package icon" points to the package icon button in the workspace. The "Edit" dialog box is open, showing the "Package" field. A red box labeled "Enter Package name" points to the "New:" text field in the dialog, which contains "CAP".



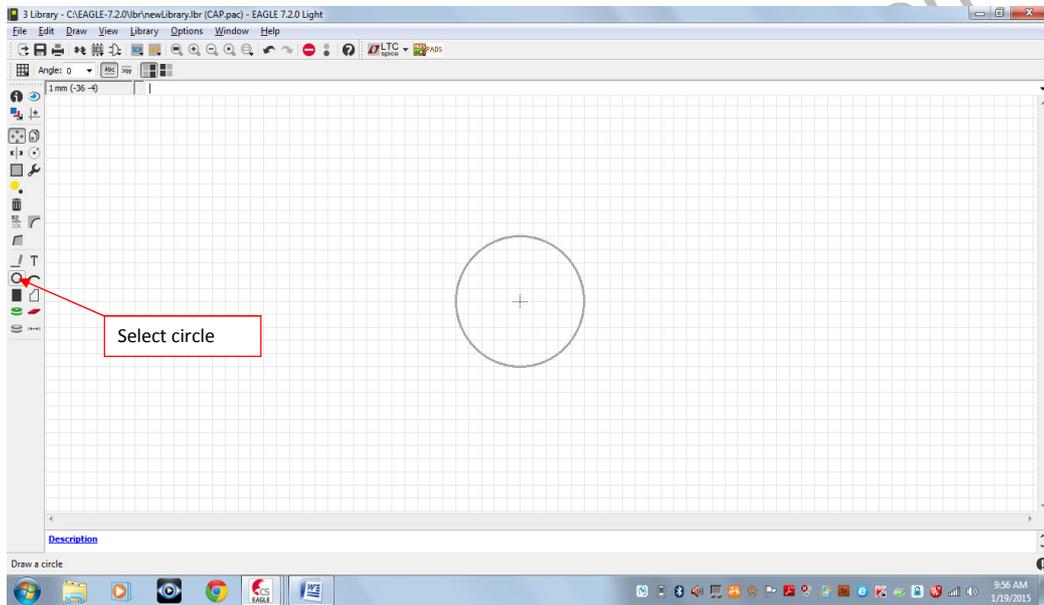
Step 16: Select 'Yes', to terminate the warning window.



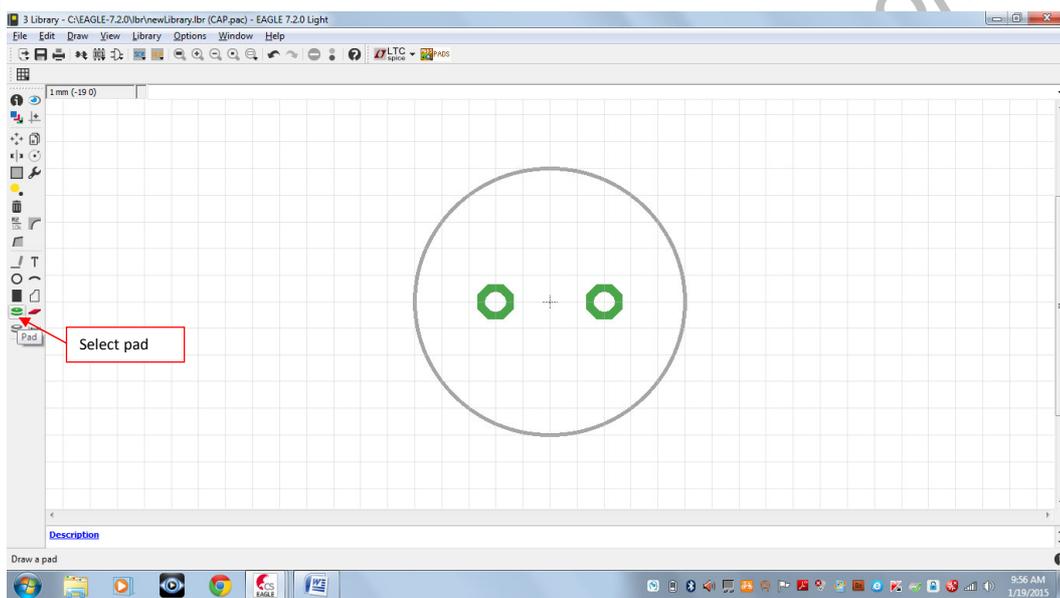
Step 17: Set Grid size to 1 mm.



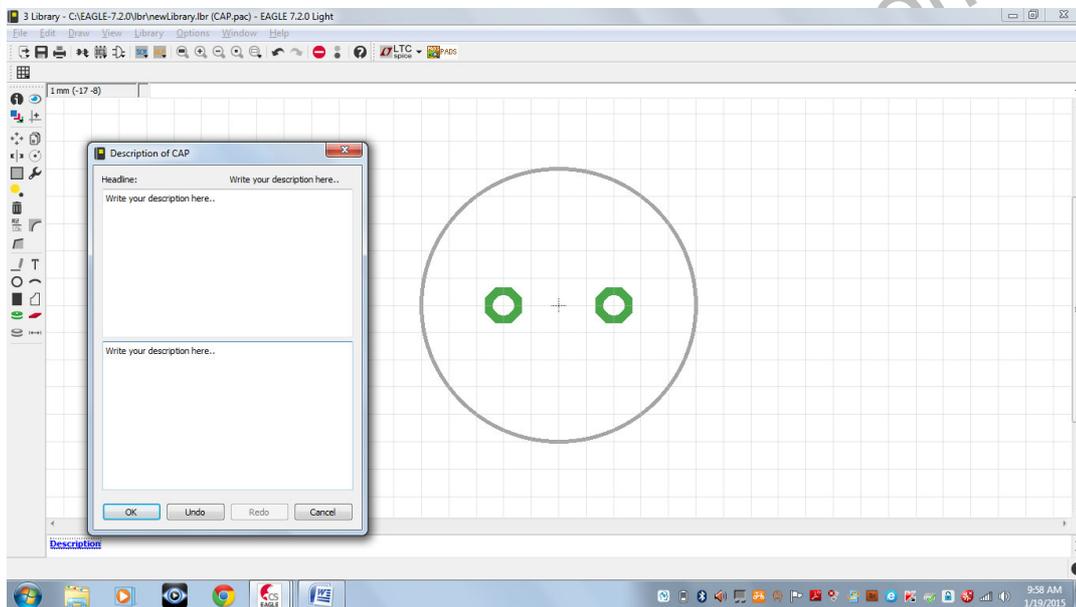
Step 18: Draw package outline.



Step 19: Select Pad

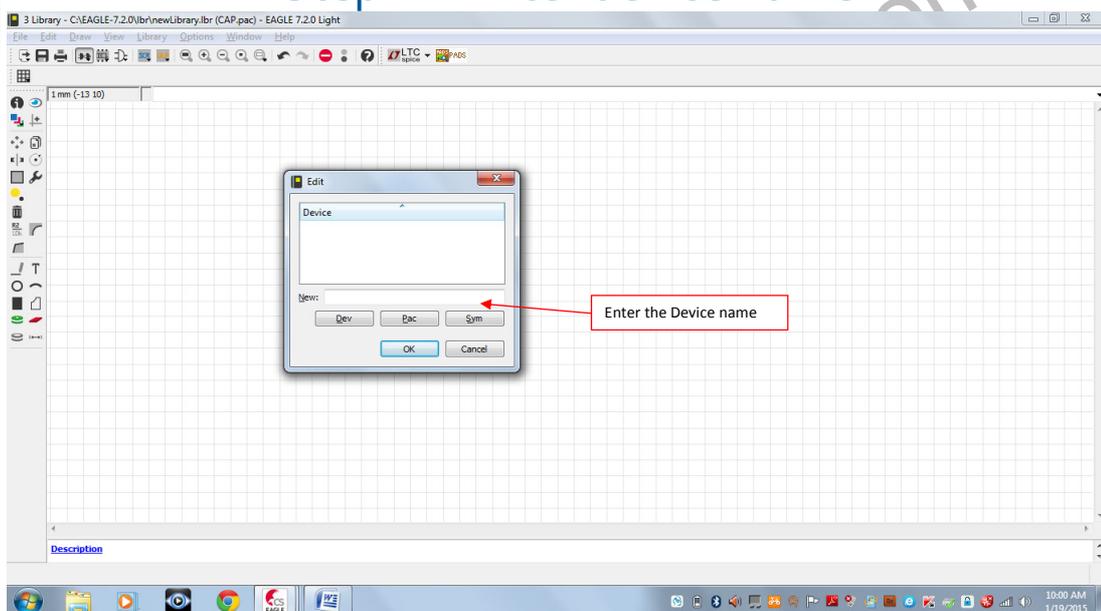


Step 20: Give description for the package.



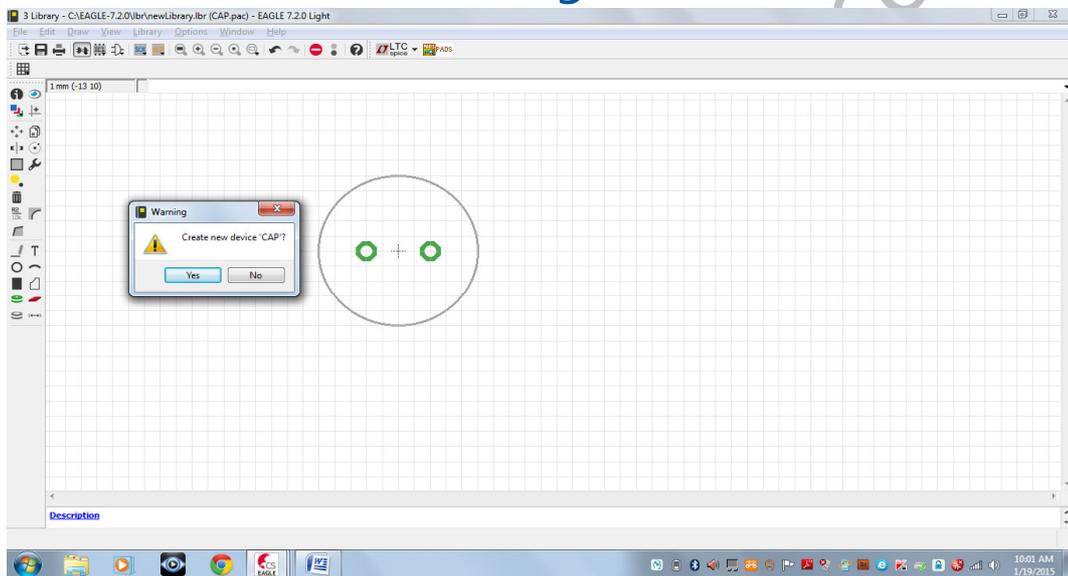
Edit Device:

Step 21: Enter device name.

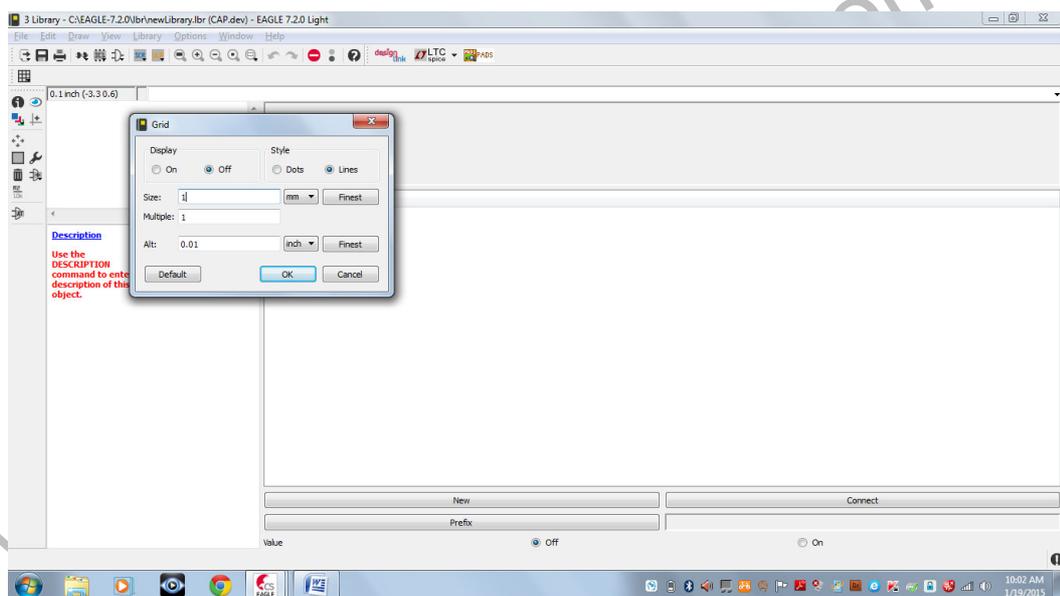




Step 22: Select 'Yes', and terminate the warning window.



Step 23: Set the Grid size for the Device.



Step 24: Add the symbol

3 Library - C:\EAGLE-7.2.0\lib\newLibrary.lbr (CAP.dev) - EAGLE 7.2.0 Light

File Edit Draw View Library Options Window Help

Address: next Swapslevel: 0

1mm (4-4-35)

Add

Symbol
CAP

OK Cancel

Select add icon

Use the DESCRIPTION command to enter a description of this object.

Select OK button

Description Technologies Attributes
CAP

New Connect

Prefix Value Off On

10:03 AM 1/19/2015

Step 25: Selected symbol is shown in slide.

3 Library - C:\EAGLE-7.2.0\lib\newLibrary.lbr (CAP.dev) - EAGLE 7.2.0 Light

File Edit Draw View Library Options Window Help

Address: next Swapslevel: 0

1mm (13-15)

Add=next Swaps=0

G\$1

>value

Description Technologies Attributes
CAP

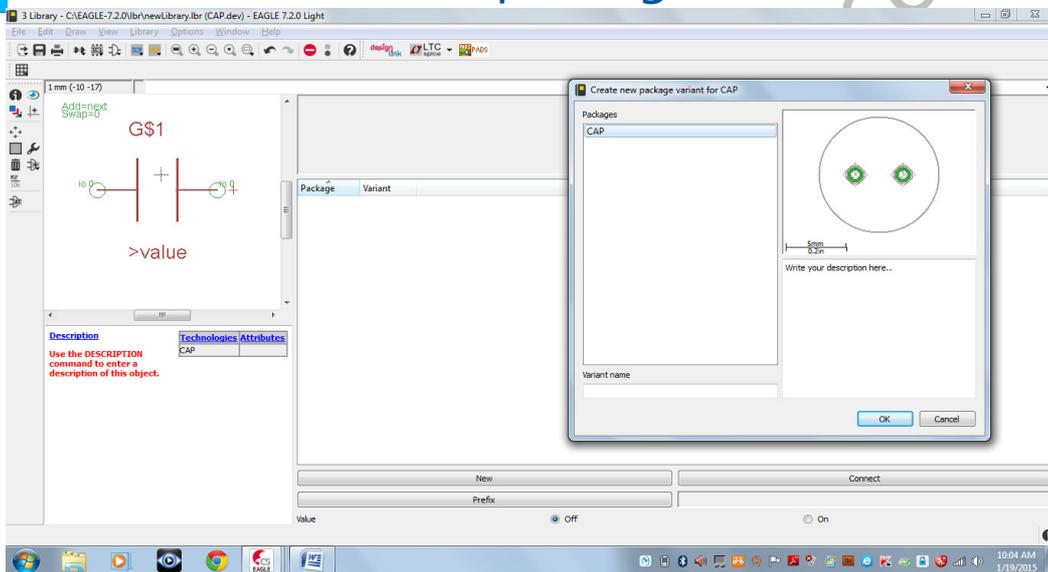
New Connect

Prefix Value Off On

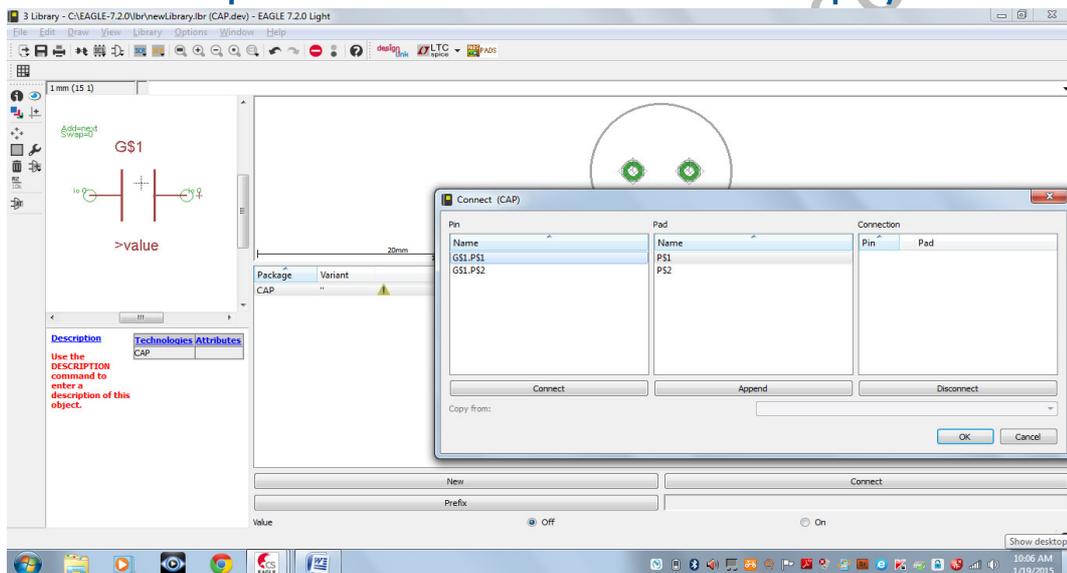
10:03 AM 1/19/2015



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

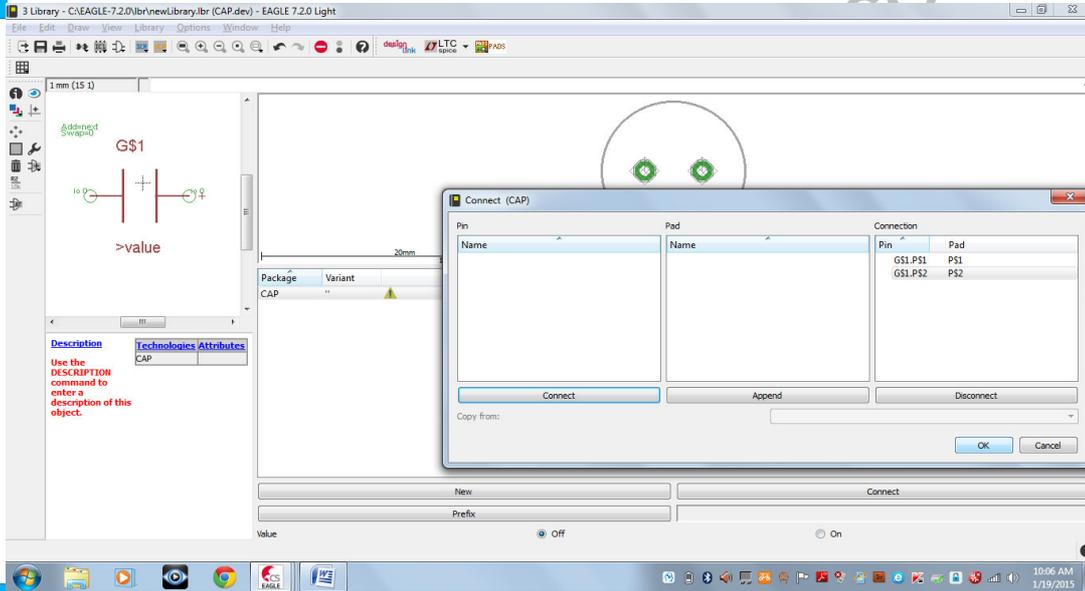


Step 27: click 'connect' button till the Pin & pad columns become empty.

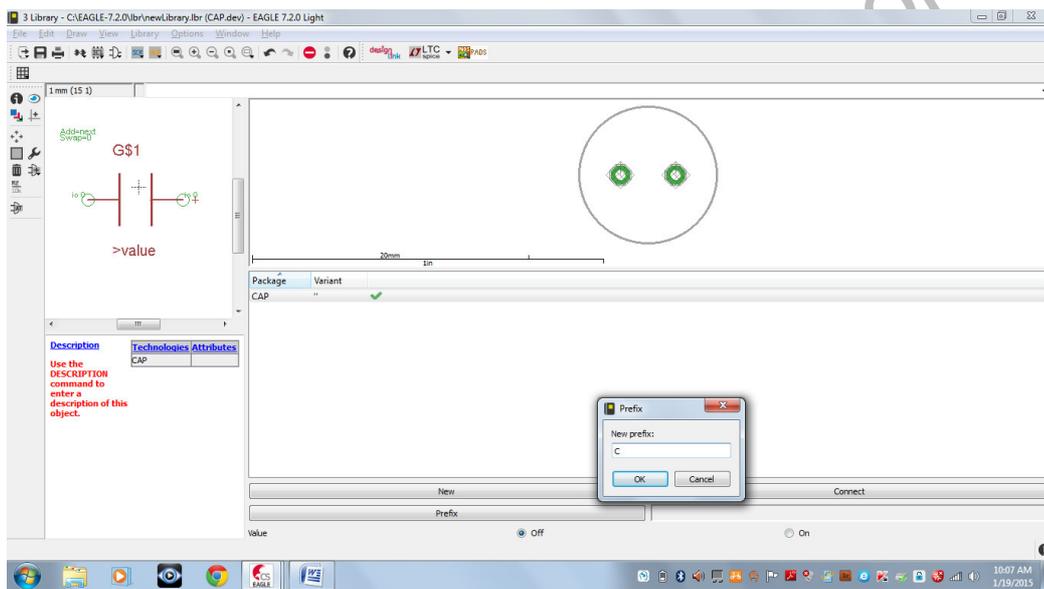




Step 28: Select 'OK' button to finish the connections.

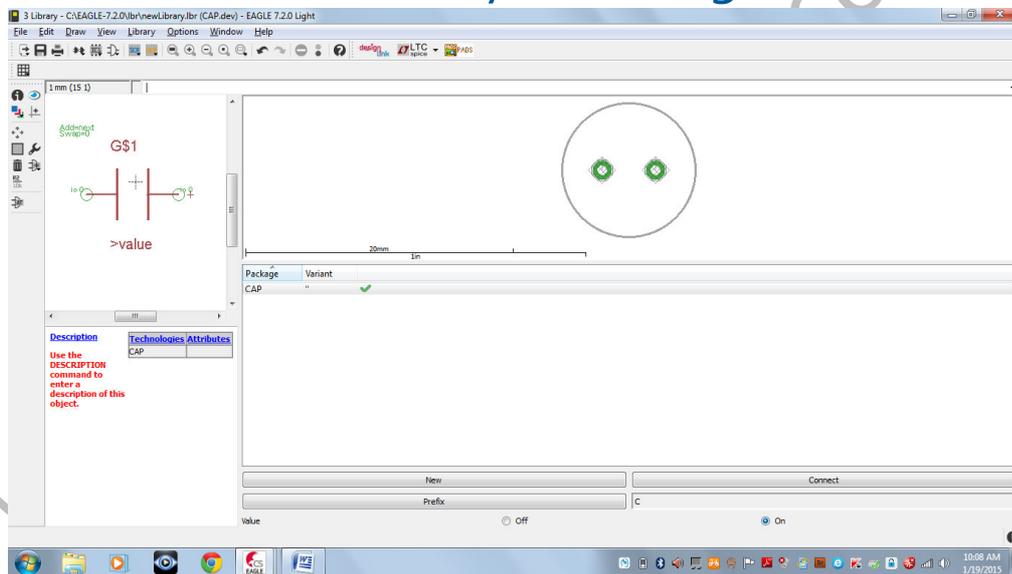


Step 29: Enter 'Prefix'.

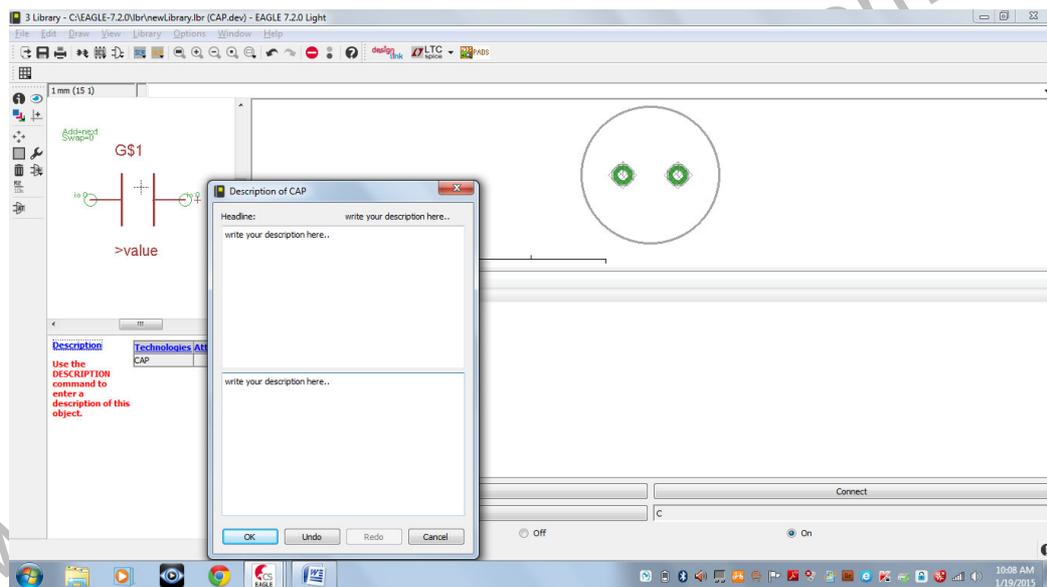




Step 30: Enable the Value by selecting **on**,
disable by selecting **off**.

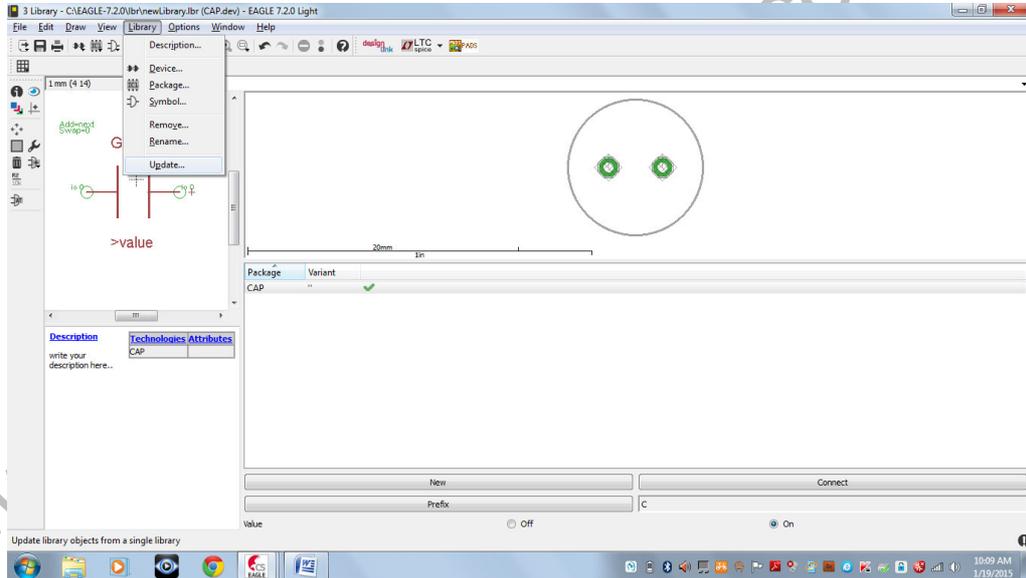


Step 31: Enter the Device description.

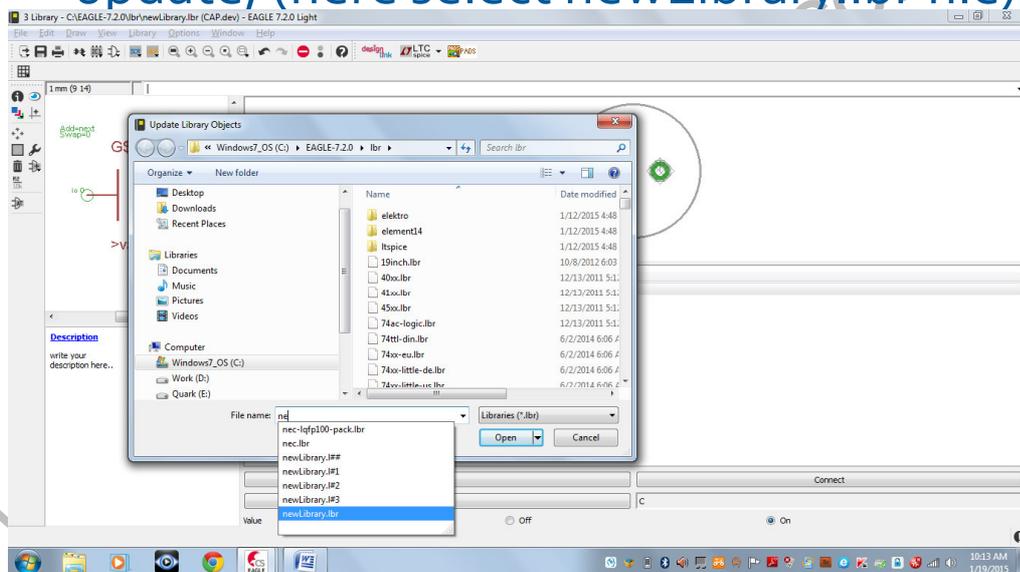




Update new Library: Step 32: Select Library > Update

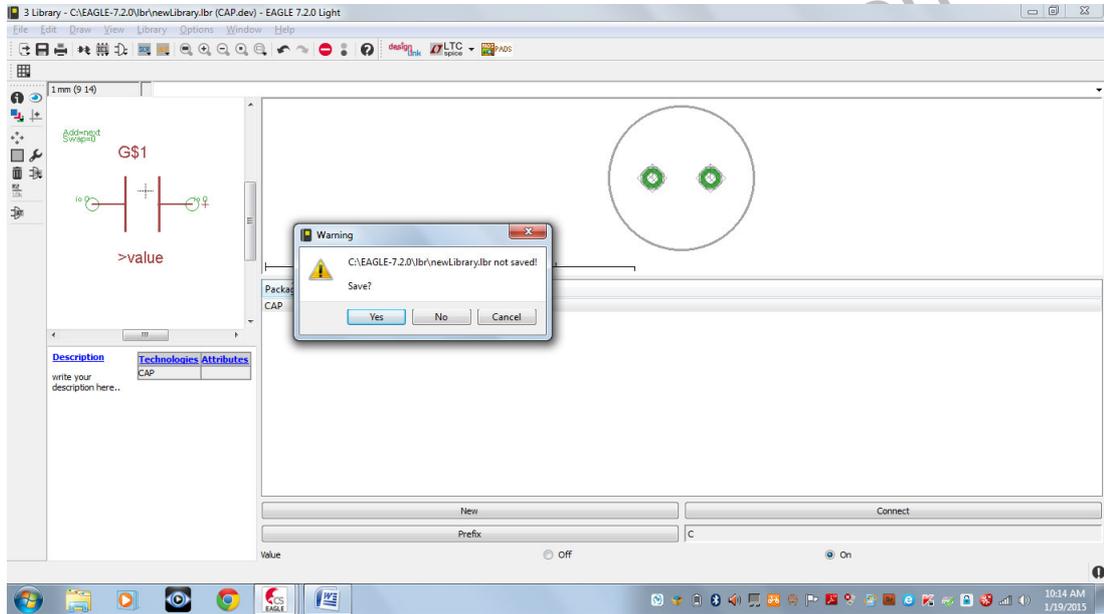


Step 33: Select the Library name and update, (here select newLibrary.lbr file).

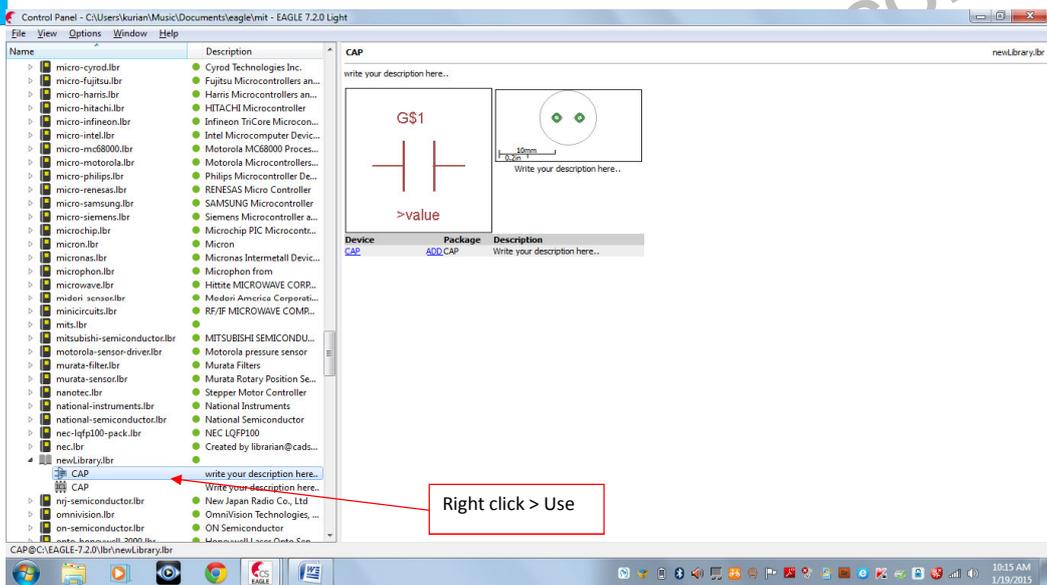




Step 34: Save the changes in Library.



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





Gerber File in Eagle



www.greytechnolabs.com

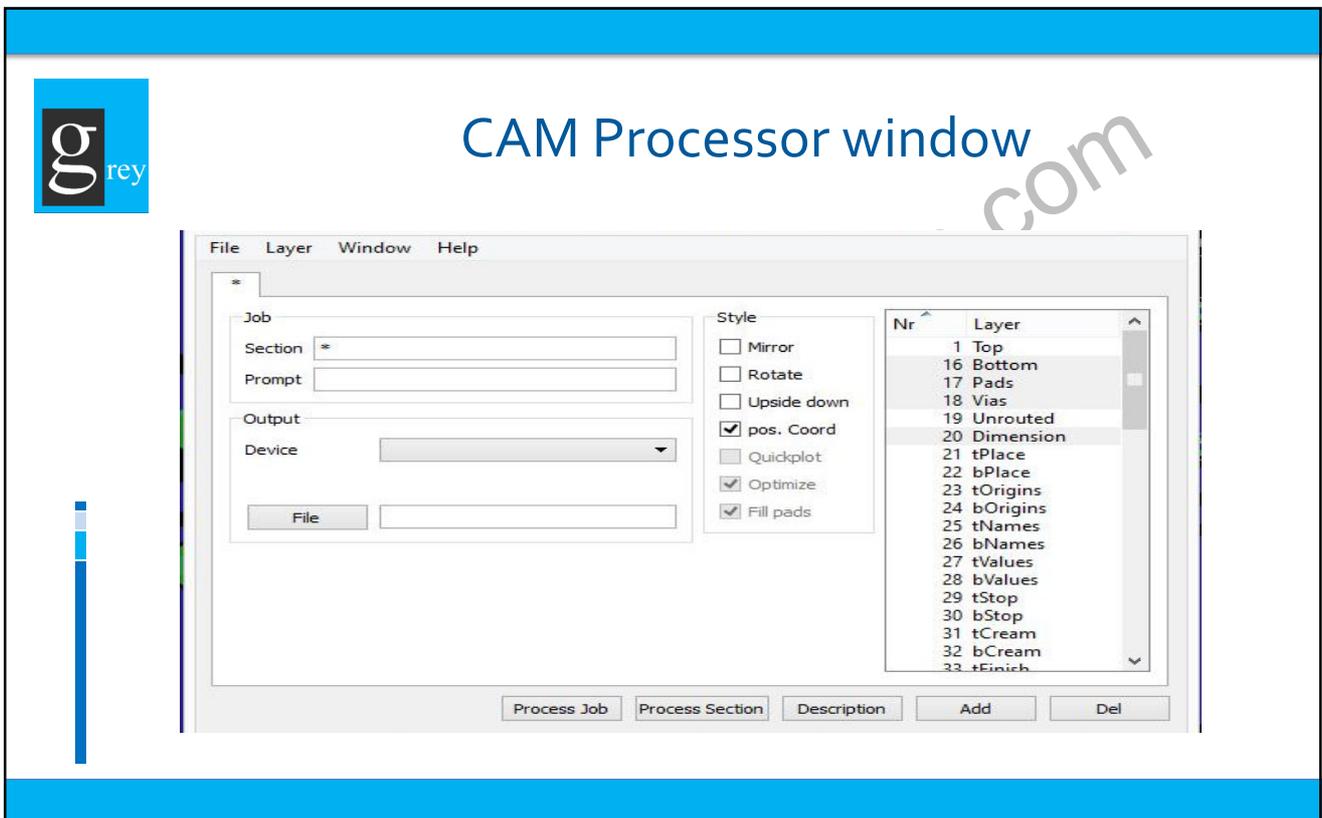
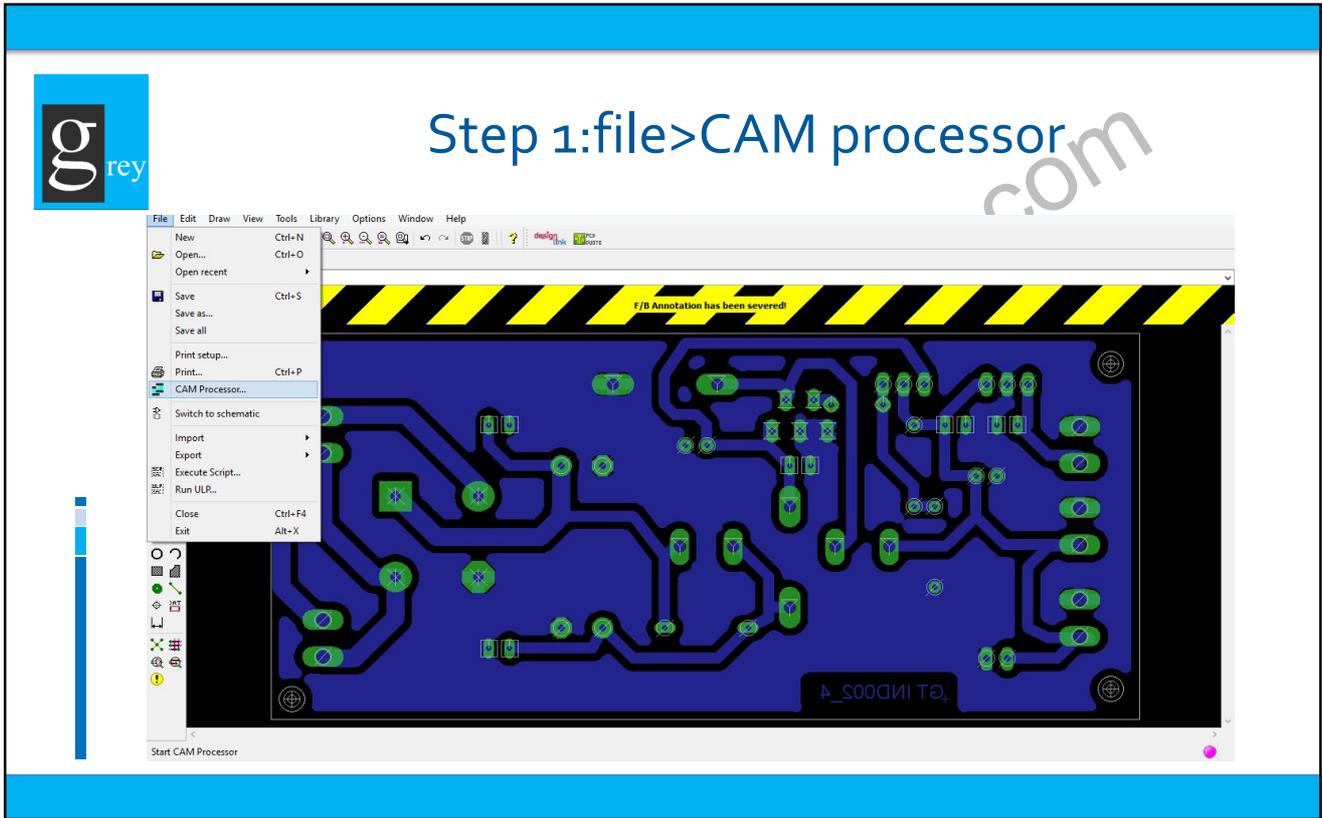


What is a Gerber file?

- 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



www.greytechnolabs.com





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

File Layer Window Help

Component

Job

Section: Component

Prompt:

Output

Device: GERBER_RS274X

File: ::/Users/Avi/Desktop/Gerber/ps.cmp

Offset

X: 0inch

Y: 0inch

Style

Mirror

Rotate

Upside down

pos. Coord

Quickplot

Optimize

Fill pads

Nr	Layer
1	Top
16	Bottom
17	Pads
18	Vias
19	Unrouted
20	Dimension
21	tPlace
22	bPlace
23	tOrigins
24	bOrigins
25	tNames
26	bNames
27	tValues
28	bValues
29	tStop
30	bStop
31	tCream
32	bCream
33	tFinich

Process Job Process Section Description Add Del



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

File Layer Window Help

Solder Component

Job

Section Solder

Prompt

Output

Device GERBER_RS274X

File C:/Users/Avi/Desktop/Gerber/ps.sol

Offset

X 0inch

Y 0inch

Style

Mirror

Rotate

Upside down

pos. Coord

Quickplot

Optimize

Fill pads

Nr	Layer
1	Top
16	Bottom
17	Pads
18	Vias
19	Unrouted
20	Dimension
21	tPlace
22	bPlace
23	tOrigins
24	bOrigins
25	tNames
26	bNames
27	tValues
28	bValues
29	tStop
30	bStop
31	tCream
32	bCream
33	tFinish

Process Job Process Section Description Add Del

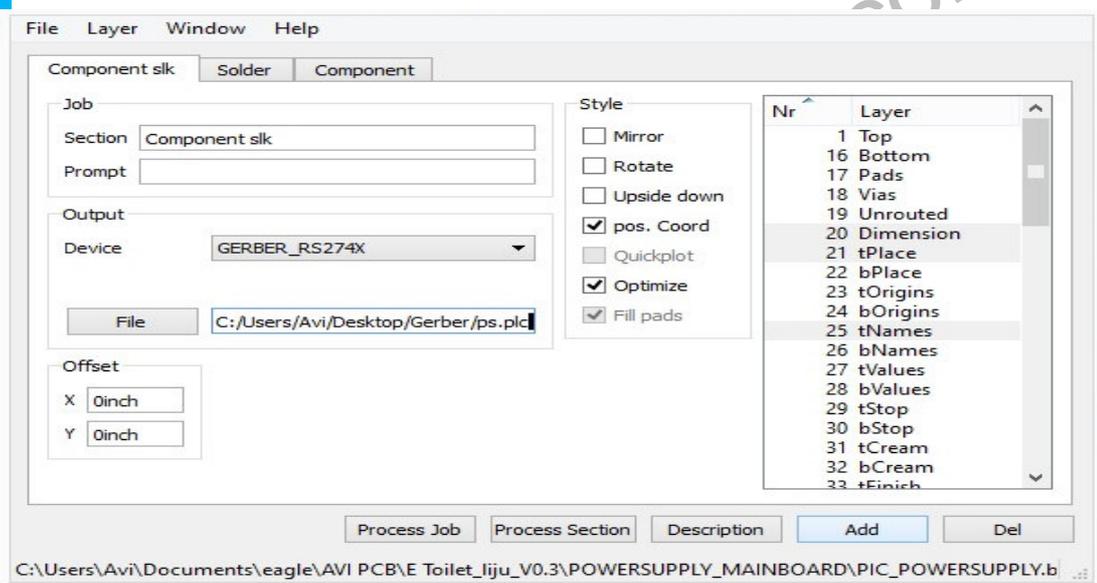


Step 4: Component Slk Layer Creation

- 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



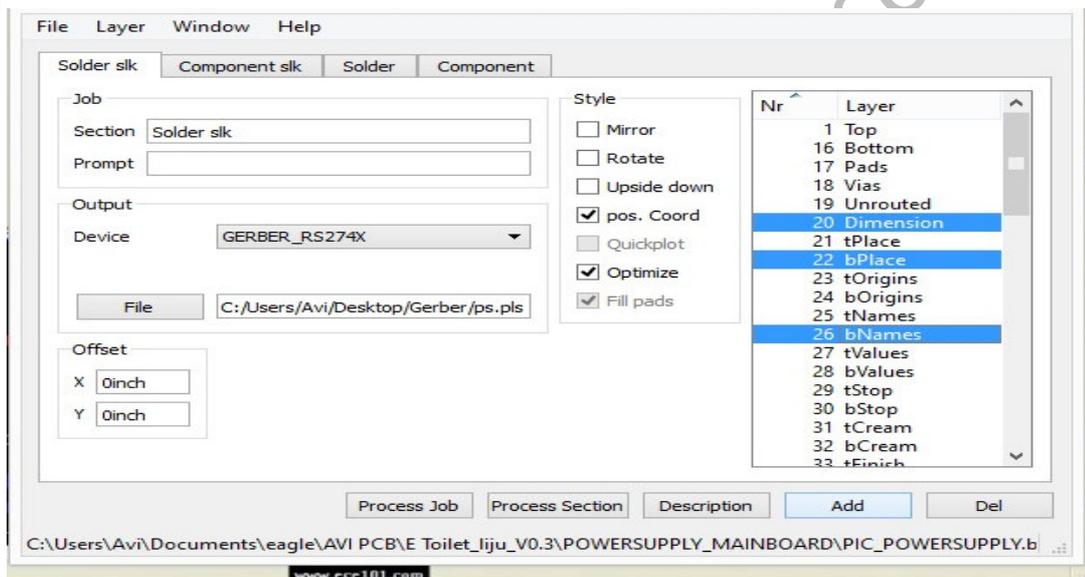


Step 5: Solder slk Layer creation

- 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



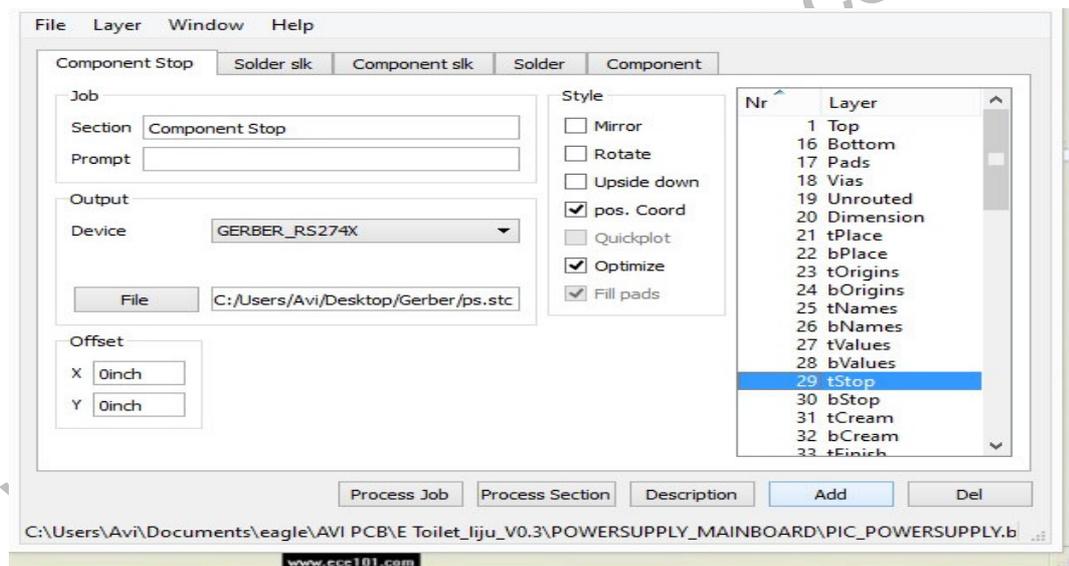


Step 6: Component stop Layer creation

- 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



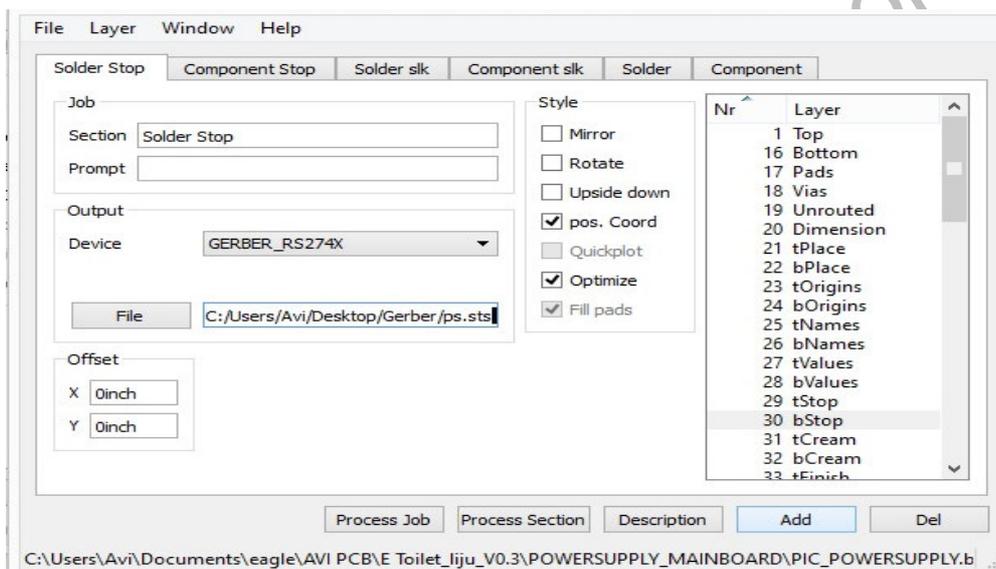


Step 7: Solder stop Layer creation

- 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



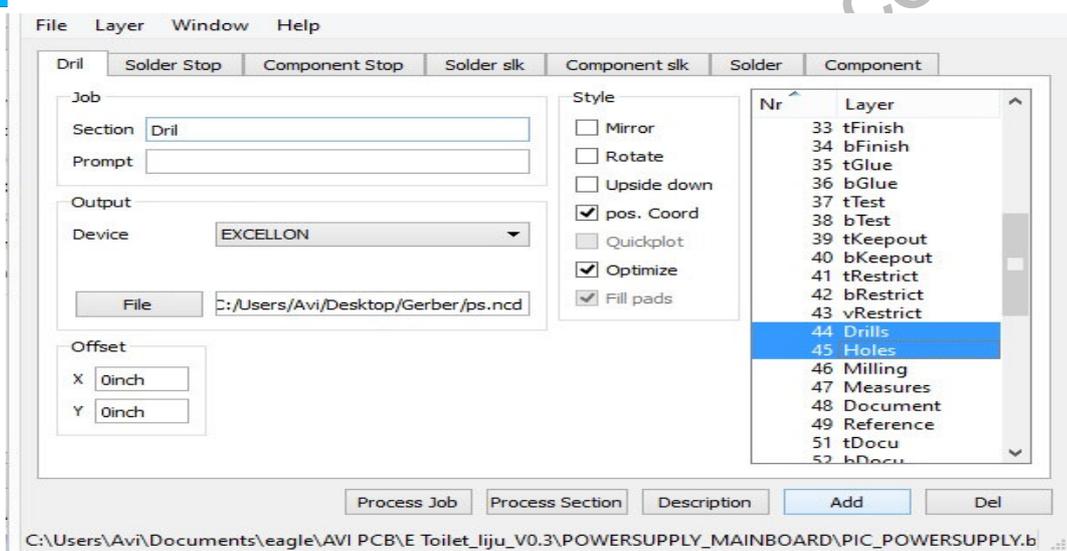


Step 8: Excellon Layer creation

- 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.

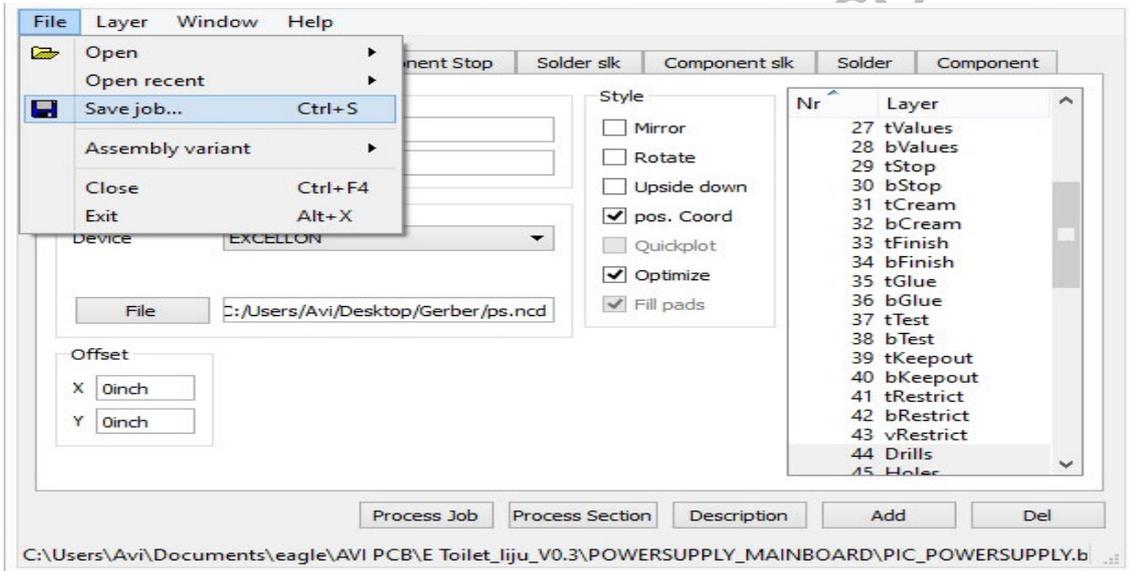


Excellon window

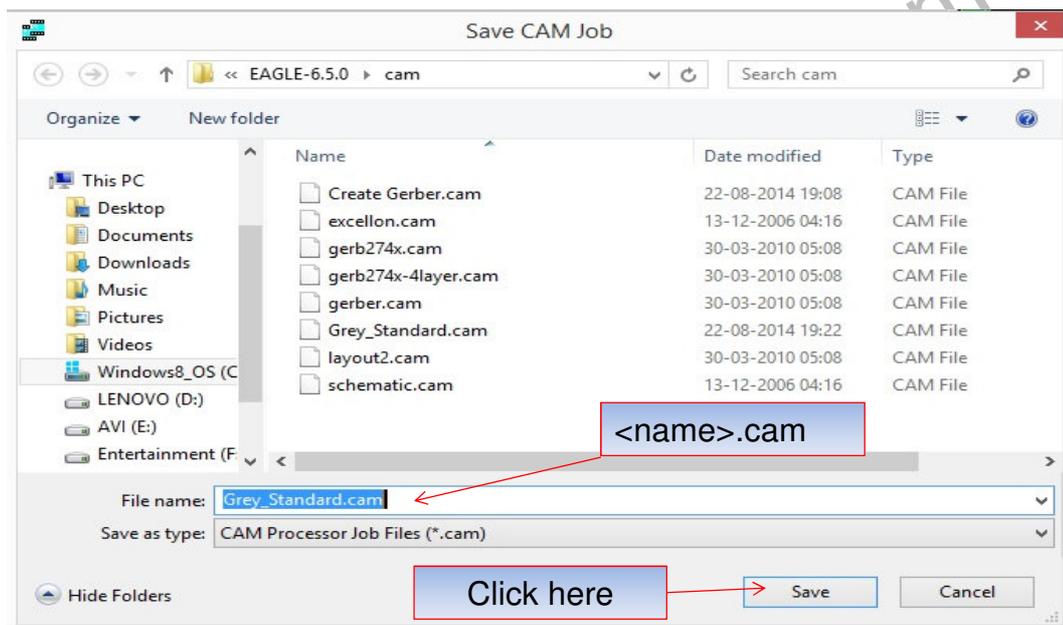




Step 9: File > Save Job

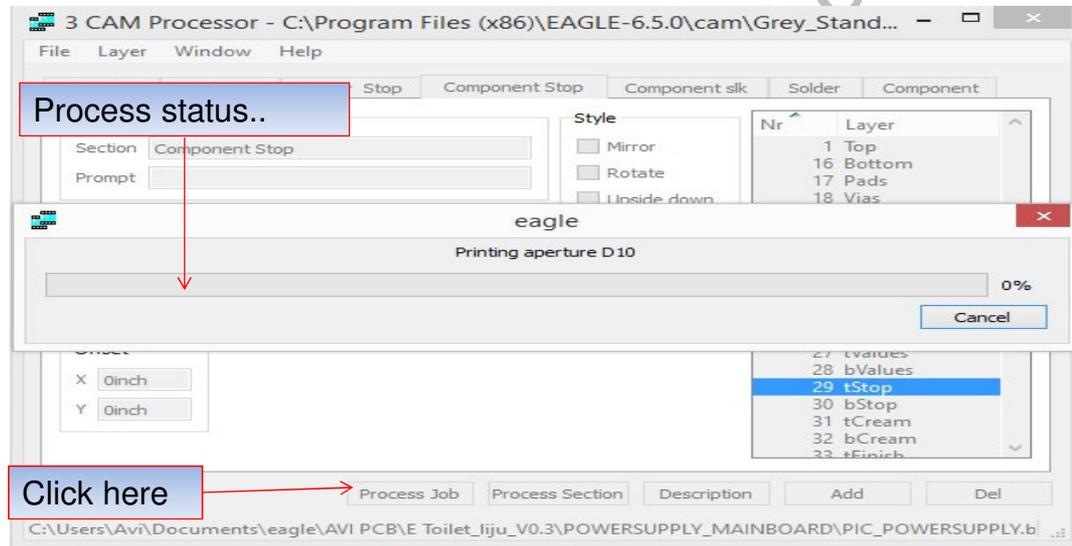


Save CAM Job





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



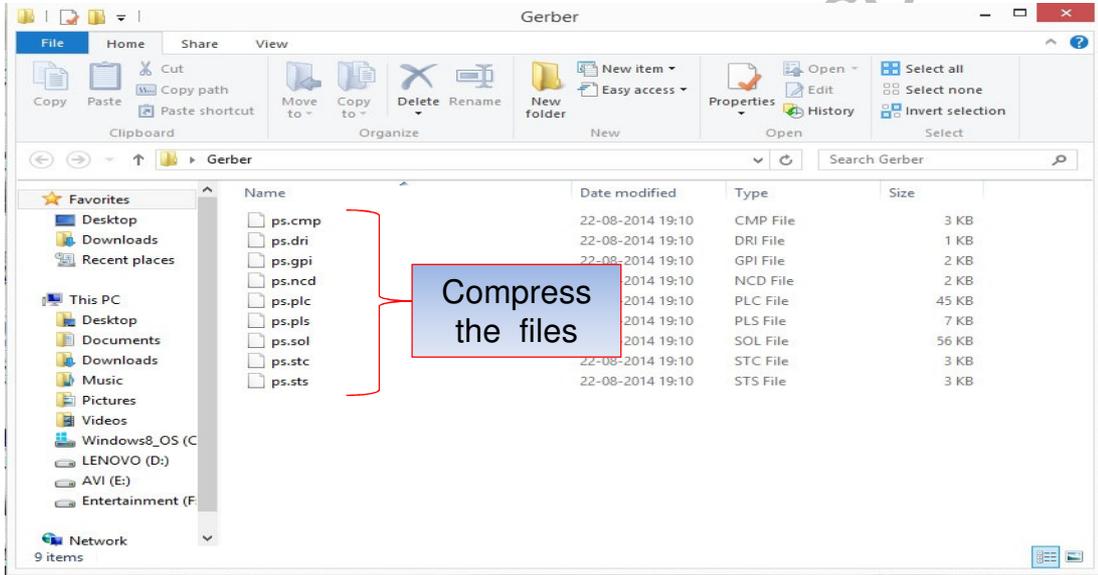
That's it, **You are done.**

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

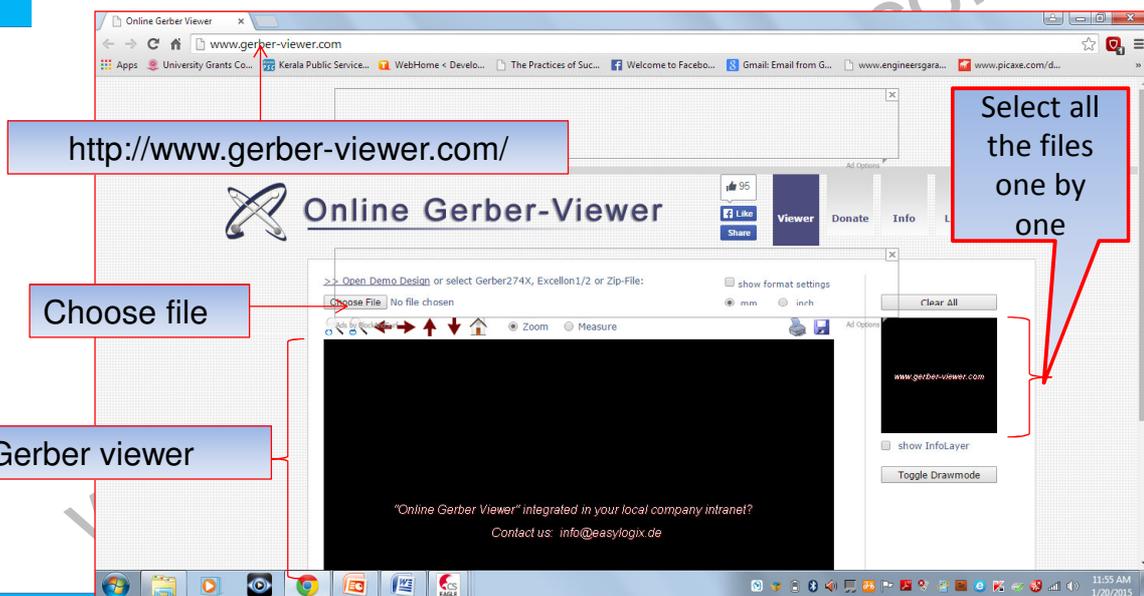
You can see files **with extension .cmp, .sts etc.**



Step 11: select all > compress the files.zip



Step 12: Upload your zip



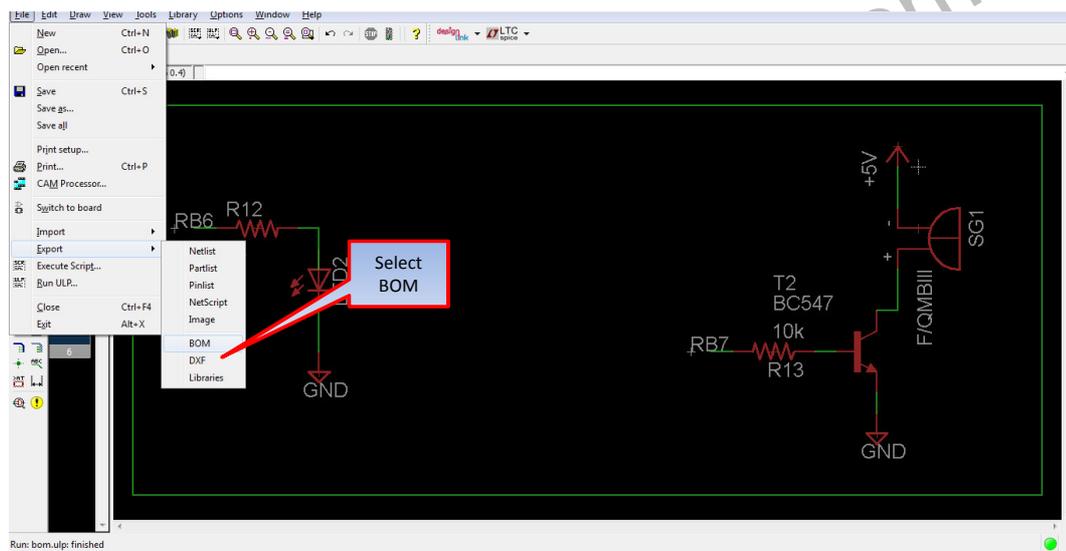


Bill Of Material - BOM

www.greytechnolabs.com



Step 1: File>Export>BOM





Step 2: Save the file

The screenshot shows the Eagle Bill of Material (BOM) dialog box. The table below represents the data shown in the dialog:

Qty	Value	Device	Package	Parts	Description	MF	MPN	OC_FARN
1	220R	R-US_0207/7	0207/7	R11	RESISTOR, American symbol			
1	7805TV	7805TV	TO220V	IC4	Positive VOLTAGE REGULATOR			
1	F/QMBIII	F/QMBIII	F/QMBIII	SG1	BUZZER Source: Buerklin			
1	GSV-2	GSV-2	GSV-2	K1	OMRON PCB Relay		GSV-2 5DC	unknown
1	L293D	L293D	DIL16	IC2	PUSH-PULL 4 CHANNEL DRIVER		L293D	9949488
1	MAX232	MAX232	DIL16	IC3	RS232 TRANSCEIVER	MAXIM	MAX232ACPE+	9723773
1	PIC18F4550_40	PIC18F4550_40	DIL40	IC1	USB Microcontrollers with nanoWatt Technology High-Performance, Enhanced Flash, 28/40/44-Pin, MICROCHIP	MAXIM	PIC18F4550-I/P	9321357
1	RACON12/	10-XX	B3F-10XX	RESET	OMRON SWITCH		B3F-1000	176432
1	TL46PO	TL46PO	TL4XPO	S1	TINY SWITCH ON - ON			
2	C-US025-024X044	C025-024X044	C1, C2		CAPACITOR, American symbol			
2	LED5MM	LED5MM	LED1, LED2		LED			
2	MTA02-100	10X02MTA	GSN, I1		AMP connector			
2	0.1MF	C-EU025-025X050	C025-025X050	C3, C10	CAPACITOR, European symbol			
2	BC547	BC547	TO92	T1, T2	NPN TRANSISTOR			
3	MTA02-100	10X02MTA	MOTOR_L, MOTOR_R, POWER		AMP connector			
4	680ohm	R-US_0207/10	0207/10	R2, R3, R5, R6	RESISTOR, American symbol			
				C6, C7, C8, C9	POLARIZED CAPACITOR, American symbol			

Annotations in the screenshot include:

- Two red boxes labeled "List type by" pointing to the "Parts" and "Value" radio buttons.
- A red box labeled "Save" pointing to the "Save" button.
- A red box labeled "List type by" pointing to the "List" dropdown menu.



Example

The screenshot shows a Notepad window titled "BOM - Notepad" containing the following text:

```

Partlist exported From c:/Users/kurian/Music/Documents/eagle/ATM_/ATM_V0.sch at 1/20/2015 12:52:14 PM
Qty Value Device Package Parts Description MF
2 C-US025-024X044 C025-024X044 C1, C2 CAPACITOR, American symbol
1 CRYSTALHC495 HC495 Q3 CRYSTAL
1 DC30303 DC30303 9V/12V_DC DC POWER JACK
2 LED5MM LED5MM LED1, LED2 LED
3 M05H M05H X2 SUB-D
1 MTA02-100 10X02MTA MOTOR_L, MOTOR_R, POWER AMP connector
1 MTA03-100 10X03MTA GSN, J1 AMP connector
1 MTA04-100 10X04MTA TOUCH_PAD1 AMP connector
1 MTA06-100 10X06MTA PIC_KIT AMP connector
1 P1NH0-1X20 1X20 JF1 PIN HEADER
1 R-TRIMM3339P RTRIM3339P R7 Trimm resistor
1 R-US_0207/10 0207/10 R1 RESISTOR, American symbol
1 R-US_0207/7 0207/7 R4 RESISTOR, American symbol
2 C-EU025-025X050 C025-025X050 C3, C10 CAPACITOR, European symbol
1 1000MF CPOL-USE3,5-10 E3,5-10 C4 POLARIZED CAPACITOR, American symbol
1 100MH L-US0207/7 0207/7 L2 INDUCTOR, American symbol
1 10K R-US_0207/7 0207/7 R13 RESISTOR, American symbol
5 10UF CPOL-USE2,5-5 E2,5-5 C3, C6, C7, C8, C9 POLARIZED CAPACITOR, American symbol
1 1N4004 1N4004 D041-10 D2 DIODE
1 1N4007 1N4004 D041-10 D1 DIODE
1 1k R-US_0207/7 0207/7 R12 RESISTOR, American symbol
1 220R R-US_0207/7 0207/7 R11 RESISTOR, American symbol
4 680ohm R-US_0207/10 0207/10 R2, R3, R5, R6 RESISTOR, American symbol
1 7805TV TO220V IC4 Positive VOLTAGE REGULATOR
2 BC547 BC547 TO92 T1, T2 NPN TRANSISTOR
1 F/QMBIII F/QMBIII SG1 BUZZER Source: Buerklin
1 GSV-2 GSV-2 K1 OMRON PCB Relay
1 L293D L293D IC2 PUSH-PULL 4 CHANNEL DRIVER
1 MAX232 MAX232 IC3 RS232 TRANSCEIVER
1 PIC18F4550_40 PIC18F4550_40 DIL40 IC1 USB Microcontrollers with nanowatt Technology High-Performance, Enhanced Flash, 28/40/44-Pin, MIC
1 RACON12/ 10-XX B3F-10XX RESET OMRON SWITCH
1 TL46PO TL46PO TL4XPO S1 TINY SWITCH ON - ON
  
```

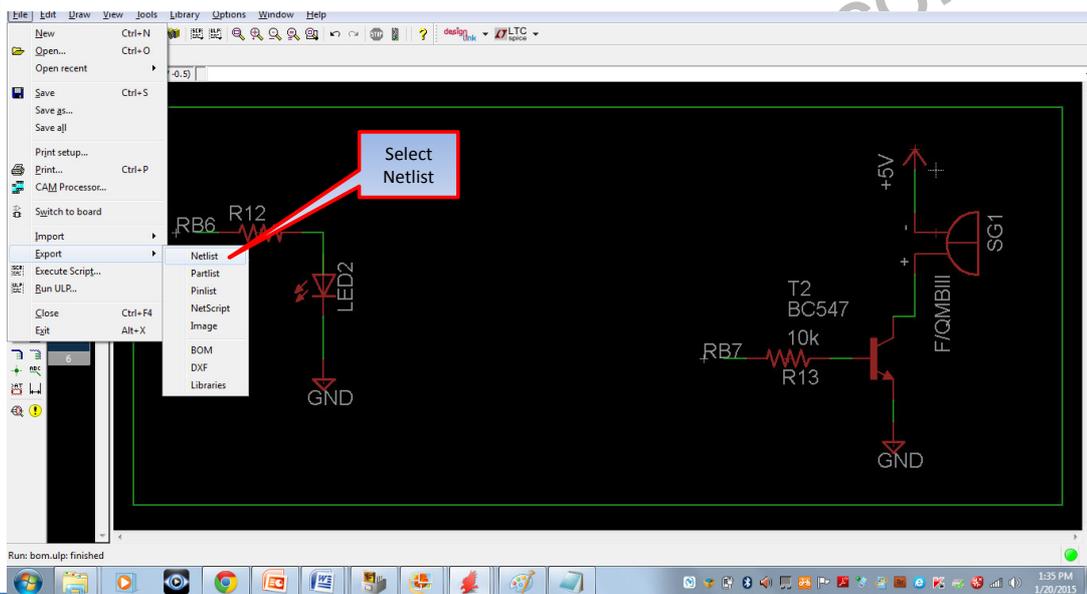


Netlist

www.greytechnolabs.com

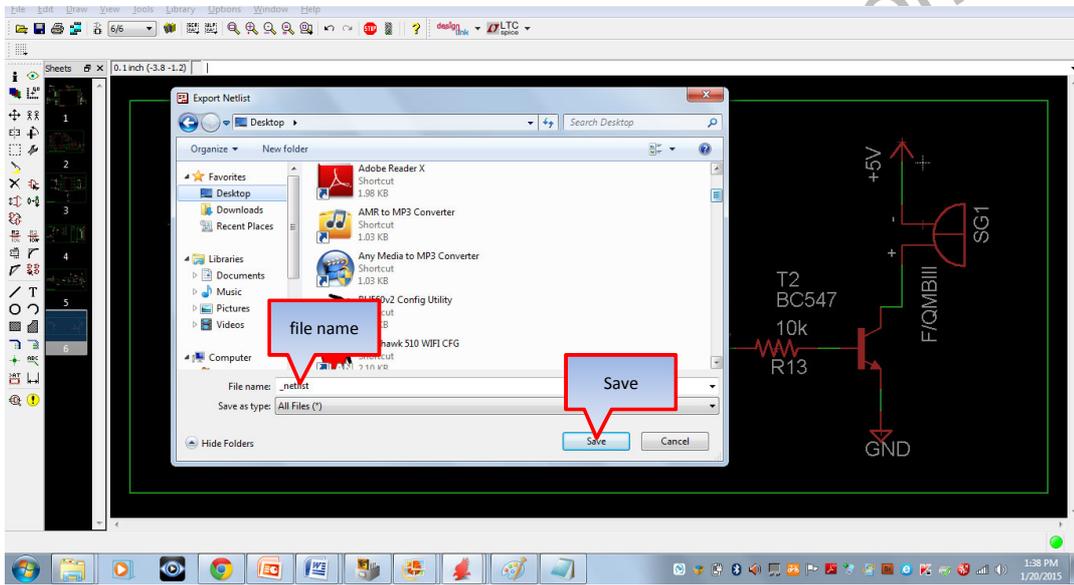


Step 1: file > Export > Netlist

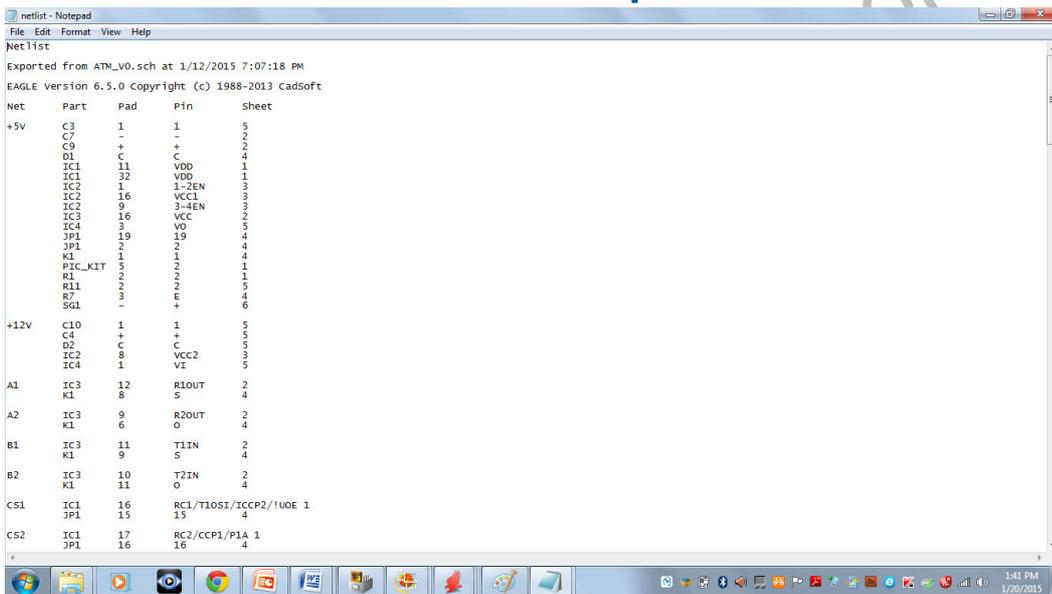




Step 2: Save the file



Example





THANK YOU



Grey Technolab (India) Pvt. Ltd
Kochi | Thiruvananthapuram
Kerala, India
www.greynetechlabs.com

www.greynetechlabs.com