

# 8. Design and simulation of PCB circuit in EasyEDA

**Course: ECE1008 – Electronic Hardware Troubleshooting LAB**

-Dr Richards Joe Stanislaus

Assistant Professor - SENSE

Email: 51749@vitstudent.ac.in



**VIT<sup>®</sup>**

**Vellore Institute of Technology**

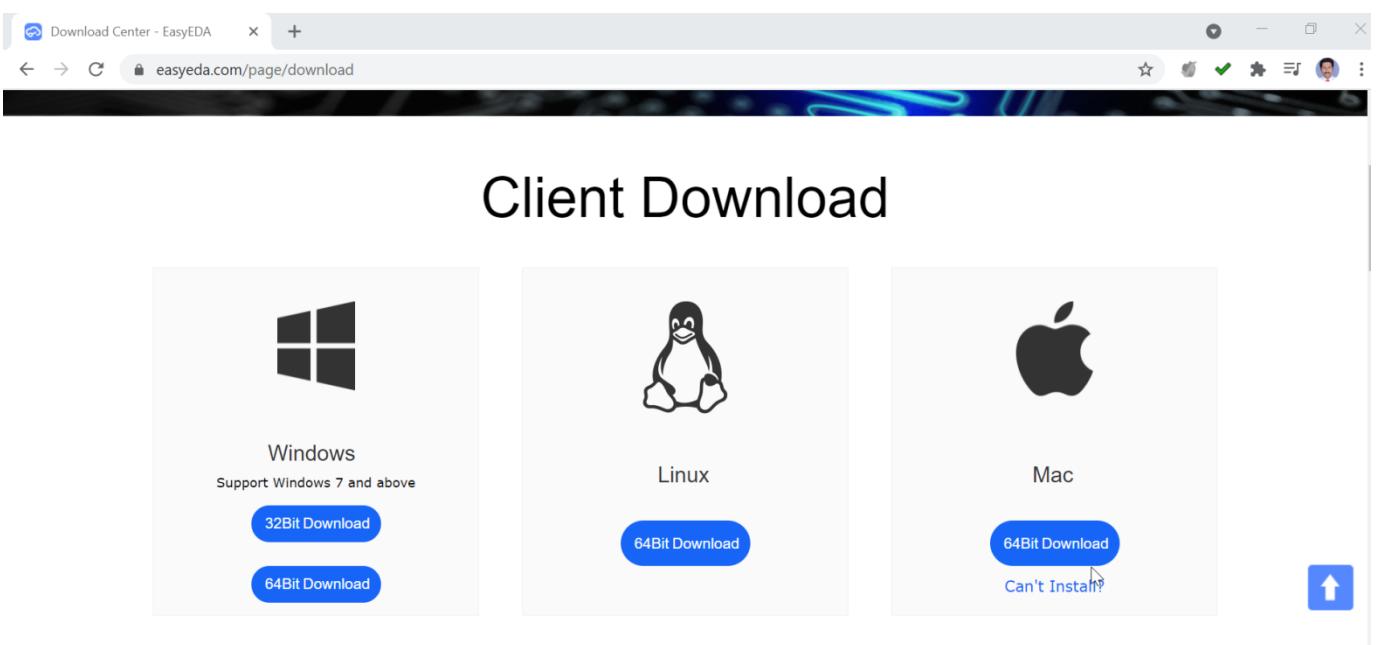
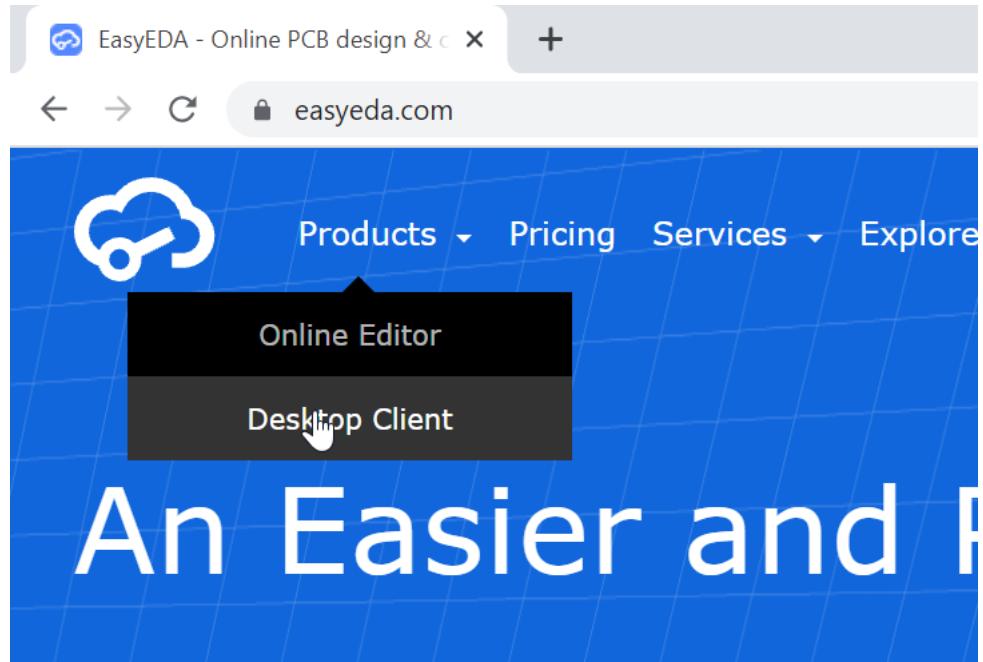
(Deemed to be University under section 3 of UGC Act, 1956)

CHENNAI

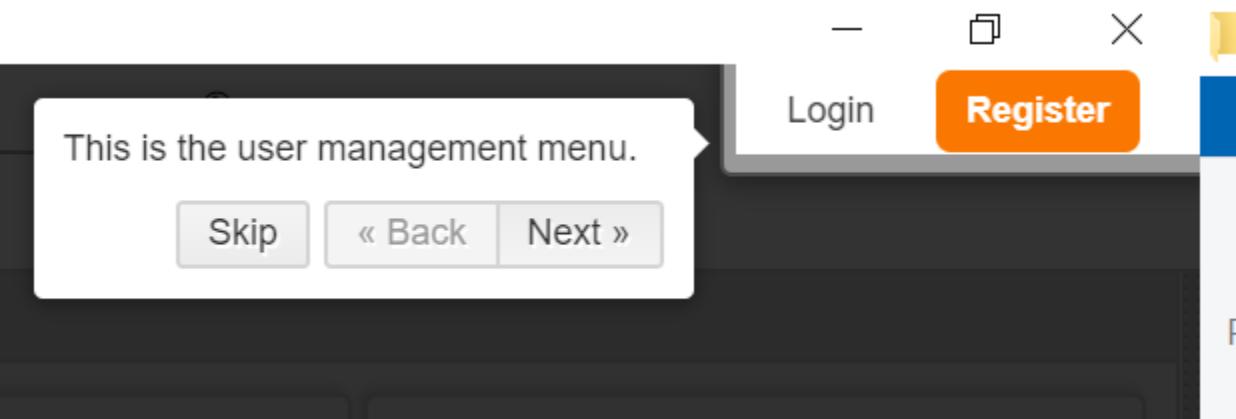
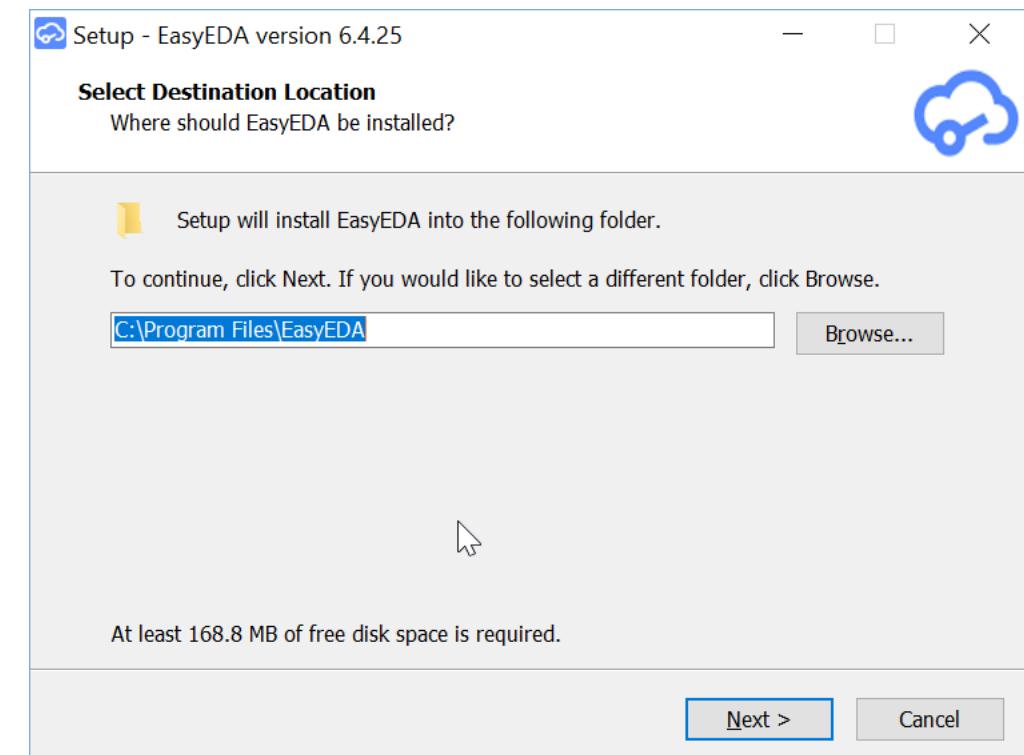


# 1. Download and install EasyEDA

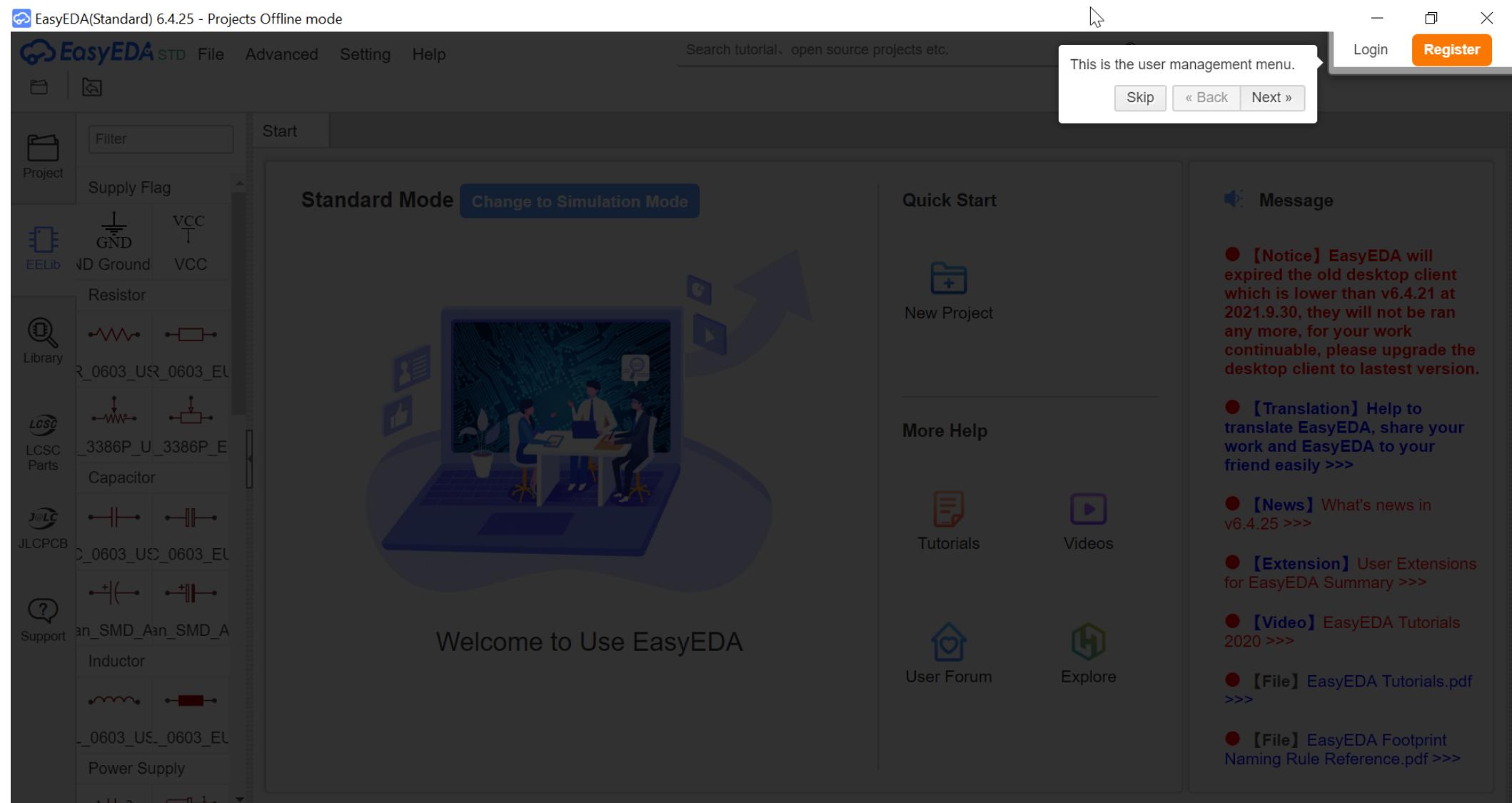
- [easyeda.com](http://easyeda.com)



# 1. Download and install EasyEDA (Windows)



# 1. Download and install EasyEDA (Windows)



# 1. Download and install EasyEDA (Windows)

The screenshot shows the main interface of the EasyEDA Standard 6.4.25 software. At the top, there is a toolbar with icons for file operations like New, Open, Save, and Print. Below the toolbar is a search bar labeled "Search tutorial, open source projects etc." and a user login/register section. A modal window titled "This is the toolbar." is displayed, containing buttons for "Skip", "<< Back", and "Next >>". The central area features a "Welcome to Use EasyEDA" message with a background illustration of three people working on a circuit board. To the left, there is a "Project" panel with a "Supply Flag" section showing GND and VCC options. Below it is a "Library" panel listing various component categories: Resistor, Capacitor, Inductor, Power Supply, and others. To the right, there is a "Quick Start" section with links for "New Project", "More Help", "Tutorials", "Videos", "User Forum", and "Explore". On the far right, a "Message" section contains several bullet points providing notices and links to various resources.

EasyEDA(Standard) 6.4.25 - Projects Offline mode

EasyEDA STD File Advanced Setting Help

Search tutorial, open source projects etc.

Login Register

This is the toolbar.

Skip << Back Next >>

Standard Mode Change to Simulation Mode

Welcome to Use EasyEDA

Project

Supply Flag

GND VCC

EElib ND Ground VCC

Resistor

LCSC Parts

JLCPCB

Support

New Project

More Help

Tutorials Videos

User Forum Explore

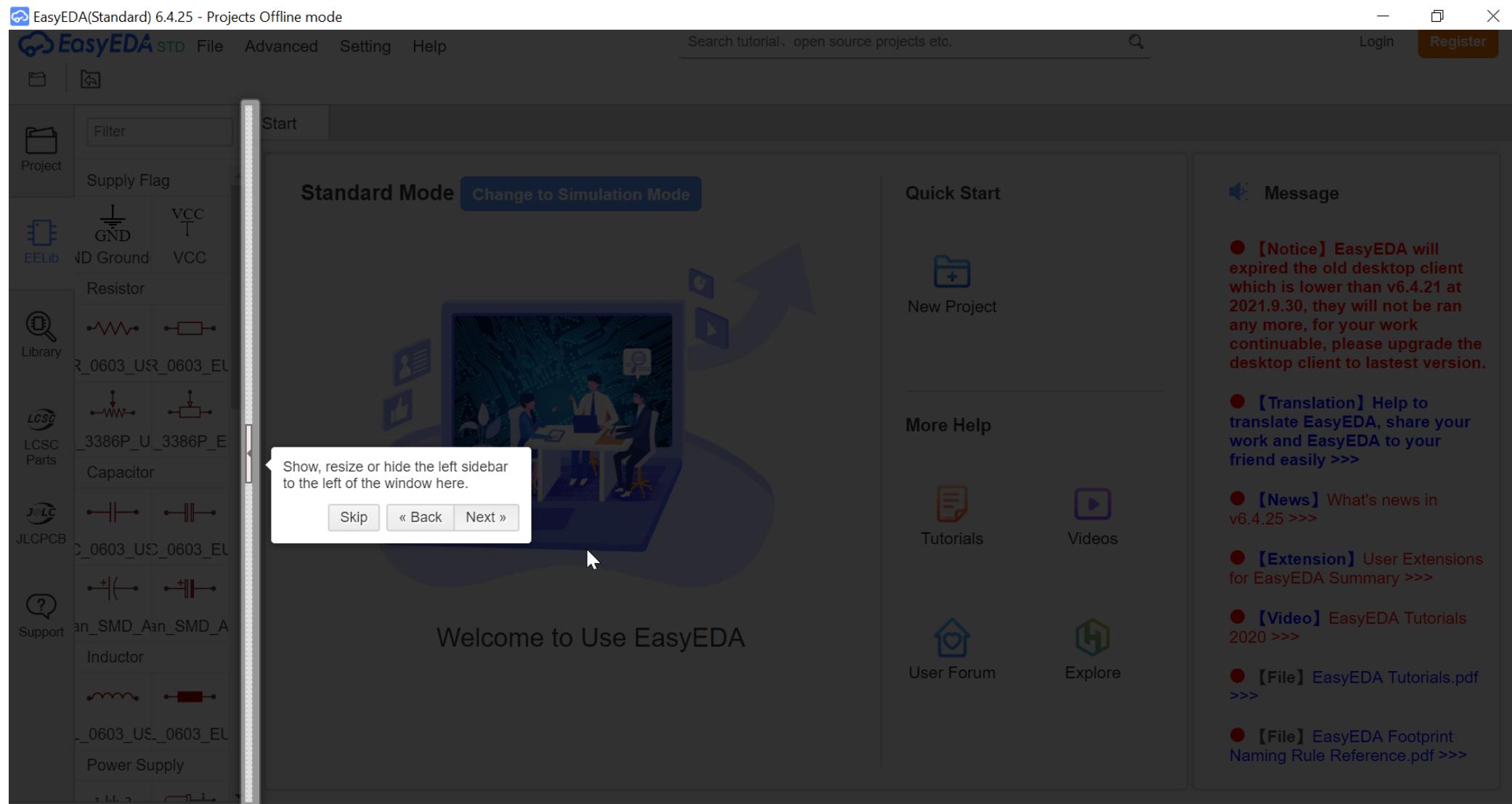
Message

- 【Notice】EasyEDA will expire the old desktop client which is lower than v6.4.21 at 2021.9.30, they will not be ran any more, for your work continuable, please upgrade the desktop client to lastest version.
- 【Translation】Help to translate EasyEDA, share your work and EasyEDA to your friend easily >>>
- 【News】What's news in v6.4.25 >>>
- 【Extension】User Extensions for EasyEDA Summary >>>
- 【Video】EasyEDA Tutorials 2020 >>>
- 【File】EasyEDA Tutorials.pdf >>>
- 【File】EasyEDA Footprint Naming Rule Reference.pdf >>>

# 1. Download and install EasyEDA (Windows)

The screenshot shows the main interface of the EasyEDA Standard 6.4.25 software. At the top, there's a navigation bar with icons for file operations, a search bar, and user authentication buttons for 'Login' and 'Register'. On the left, a vertical sidebar contains links for 'Project', 'EElib', 'Library', 'LCSC Parts', 'JLCPCB', and 'Support'. The main area features a large central window titled 'Standard Mode' with a 'Change to Simulation Mode' button. Below this is a 'Welcome to Use EasyEDA' message and a circular icon depicting two people working on a computer. To the right of the central window are three columns: 'Quick Start' with 'New Project' and 'More Help' sections; 'Message' with various notices and links; and 'Tutorials' and 'Videos' sections. A tooltip on the left side provides information about the sidebar navigation.

# 1. Download and install EasyEDA (Windows)



# 1. Download and install EasyEDA (Windows)

EasyEDA(Standard) 6.4.25 - Projects Offline mode

EasyEDA STD File Advanced Setting Help

Search tutorial、open source projects etc.

Login Register

Project Filter Start

Supply Flag

GND VCC

EElib ND Ground VCC

Resistor

R\_0603\_USR\_0603\_EL

3386P\_U\_3386P\_E

Capacitor

JLCPCB C\_0603\_USC\_0603\_EL

Support SMD\_A\_SMD\_A

Inductor

\_0603\_US\_0603\_EL

Power Supply

Standard Mode Change to Simulation Mode

Welcome to Use EasyEDA

Quick Start

New Project

Message

【Notice】 EasyEDA will expire the old desktop client which is lower than v6.4.21 at 2021.9.30, they will not be ran any more, for your work continuable, please upgrade the desktop client to lastest version.

【Translation】 Help to translate EasyEDA, share your work and EasyEDA to your friends.

Show or hide the right sidebar to the right of the window here.

Done Back Next

More Help

Tutorials Videos

User Forum Explore

【Extension】 User Extensions for EasyEDA Summary >>

【Video】 EasyEDA Tutorials 2020 >>

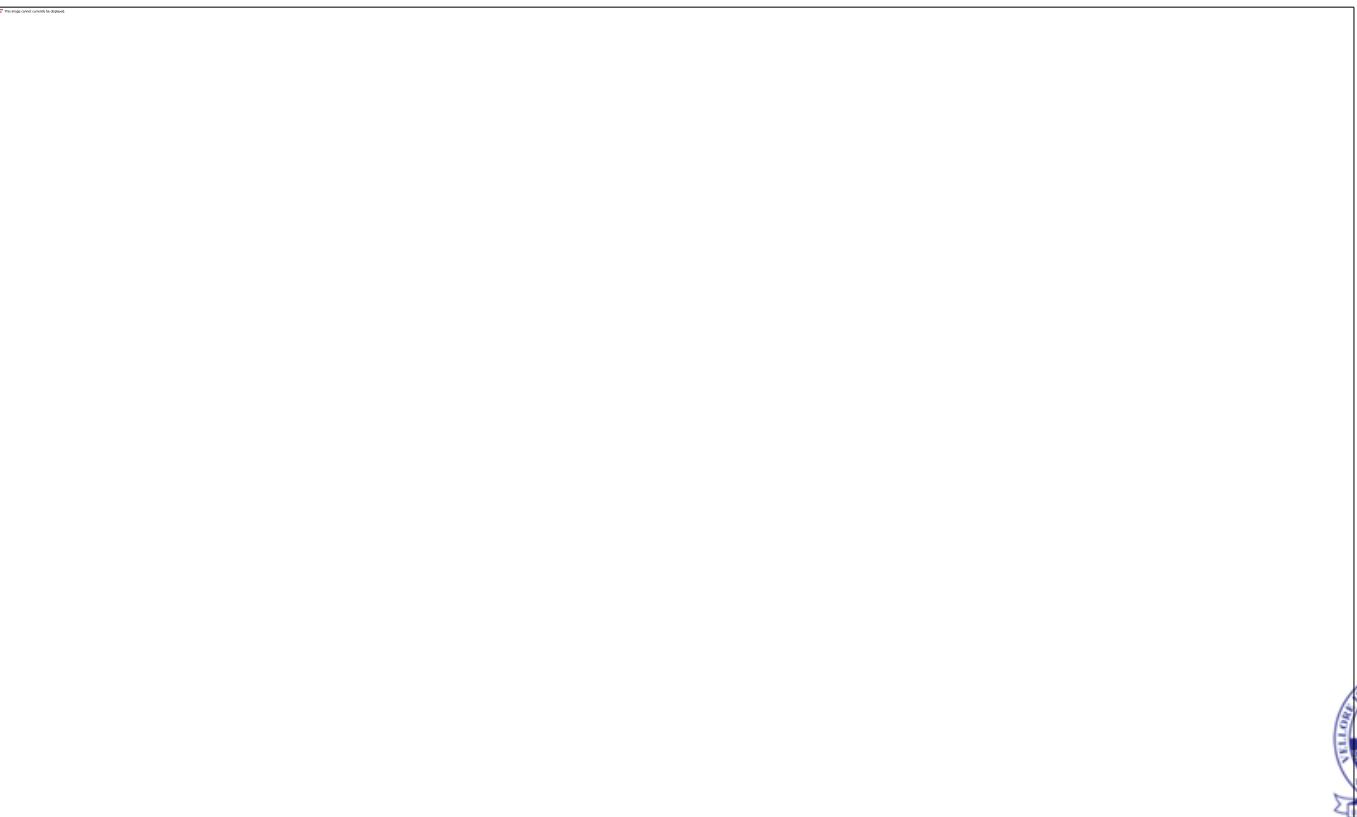
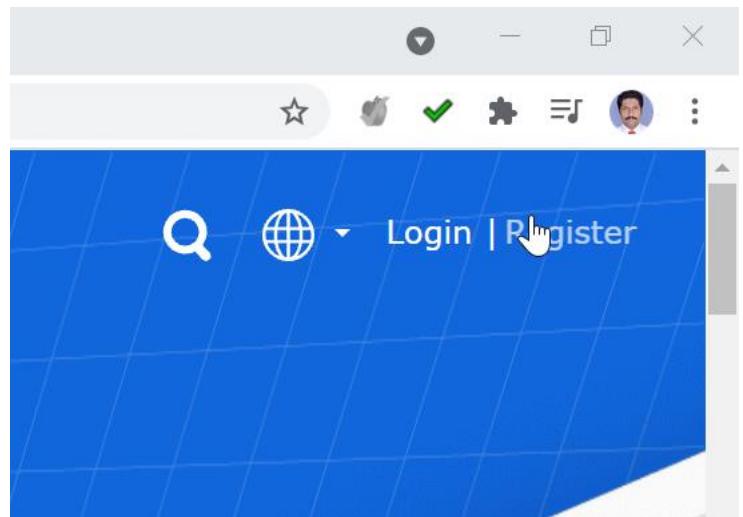
【File】 EasyEDA Tutorials.pdf >>

【File】 EasyEDA Footprint Naming Rule Reference.pdf >>

The screenshot shows the main interface of the EasyEDA Standard software. On the left, there's a library panel with categories like EElib, Library, LCSC Parts, JLCPCB, and Support, each listing various component models. The central area features a large 'Welcome to Use EasyEDA' message with a background image of people working on a circuit board. Below it, there's a 'Standard Mode' button and a 'Change to Simulation Mode' button. To the right, there are sections for 'Quick Start' (New Project), 'Message' (with a notice about an outdated desktop client), 'More Help' (Tutorials and Videos), and links to 'User Forum' and 'Explore'. A search bar at the top right allows users to search for tutorials and open source projects.

## 2. Register in easyeda.com

- Click register and select continue with google  
Login with vitstudent.ac.in email id



# 3. Build the circuit

The screenshot shows the EasyEDA software interface. On the left, there's a vertical toolbar with icons for EElib, Library, LCSC Parts, JLPCB, and Support. The main menu bar includes 'File', 'Advanced', 'Setting', and 'Help'. A search bar at the top right says 'Search tutorial, open source projects etc.' Below the menu is a user profile for 'Richards Joe Stanislaus'. The central area shows a 'Create New Project' dialog box with a red border. It has fields for 'Folder' (set to 'Offline Project'), 'Title' (set to 'VoltageRegulator'), and 'Description'. At the bottom of the dialog are 'Save' and 'Cancel' buttons. To the right of the dialog is a 'Message' sidebar with several bullet points in red text:

- [Notice] EasyEDA will expire the old desktop client which is lower than v6.4.21 at 2021.9.30, they will not be ran any more, for your work continuable, please upgrade the desktop client to lastest version.
- [Translation] Help to translate EasyEDA, share your work and EasyEDA to your friend easily >>>
- [News] What's news in v6.4.25 >>>
- [Extension] User Extensions for EasyEDA Summary >>>
- [Video] EasyEDA Tutorials 2020 >>>
- [File] EasyEDA Tutorials.pdf >>>
- [File] EasyEDA Footprint Naming Rule Reference.pdf >>>



### 3. Build the circuit: Enter your name, Reg No(in company) Experiment name (in Title)

EasyEDA(Standard) 6.4.25 - Projects Offline mode

EasyEDA STD File Edit Place Format View Design Tools Fabrication Advanced Setting Help

Richards Joe Stanislaus

All Projects(1) | Open Start \*VoltageRegulator

Project Filter

Offline Project VoltageRegulator

Design Manager

EELib

Library

LCSC Parts

JLCPCB

Support

Wiring Tools

Tip: Press **SPACE** or **R** to rotate the placing object or selected object.

Hotkeys Setting... Close

VCC +5V

Selected Objects 1

Text Attributes

Text Your Company

Color #0000FF

Font Family Verdana

Font Size 9pt

Font Weight (Auto)

Font Style (Auto)

Text Type common

ID gge72

Mouse-X 660

Mouse-Y 480

Mouse-DX -229.63

Mouse-DY 435.18

Sheet\_1

Sheet 1/1

EasyEDA Company: Your Company Date: 2021-10-02 Drawn By: Richards Joe Stanislaus

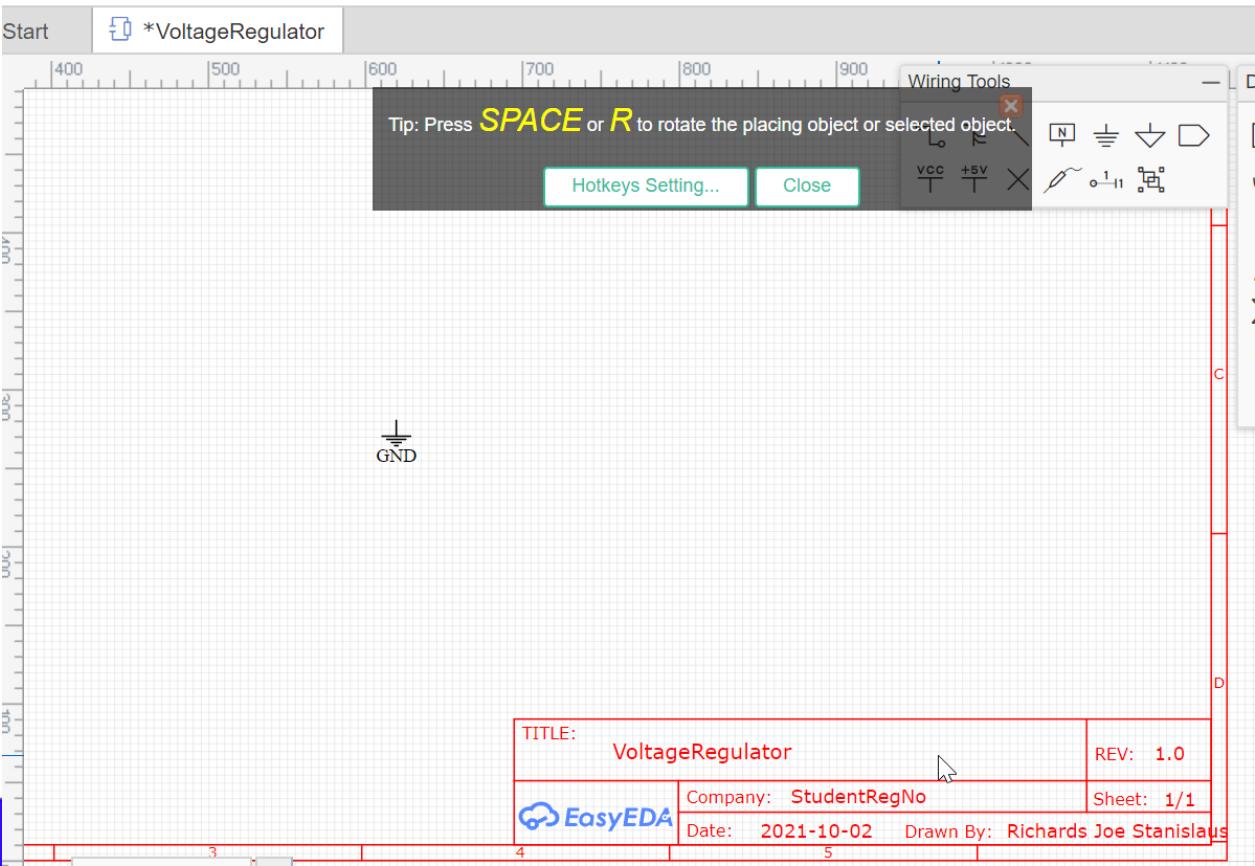
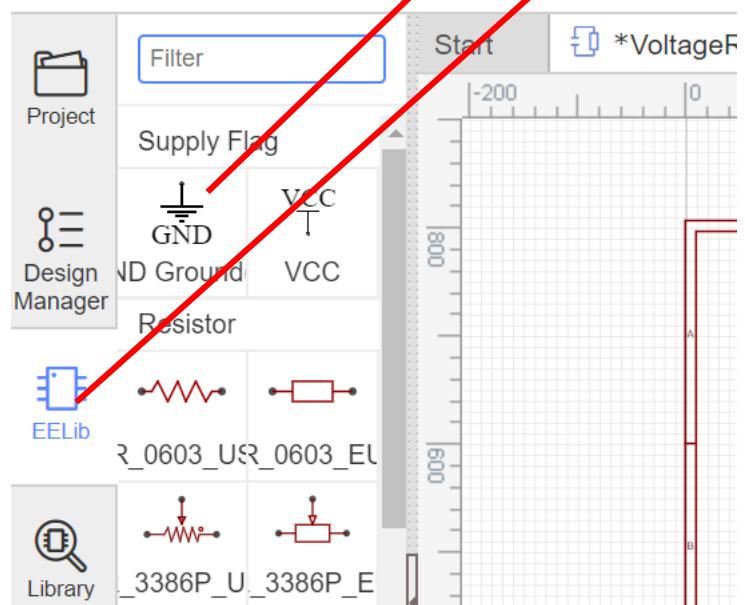


# 3. Build the circuit

- In EasyEdaLibrary (EEL) ->

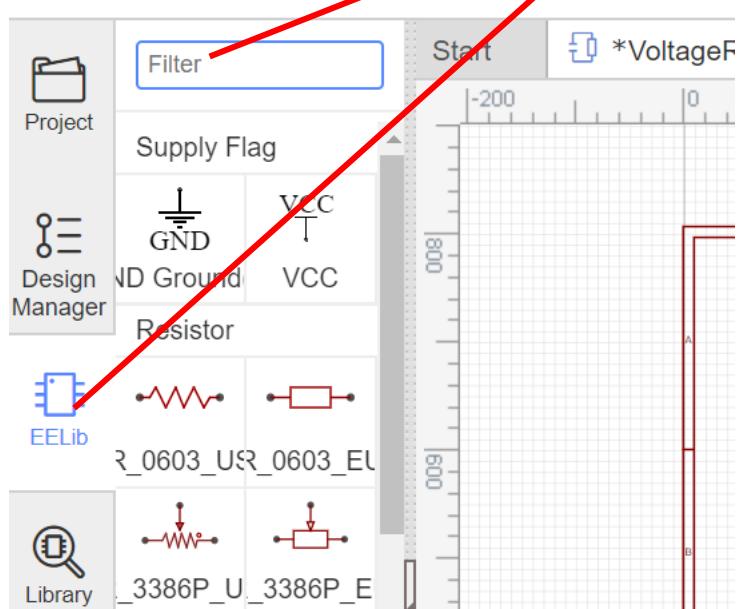
Select ground and place it in the schematic window

To zoom in / zoom out -> Mouse scroll in schematic

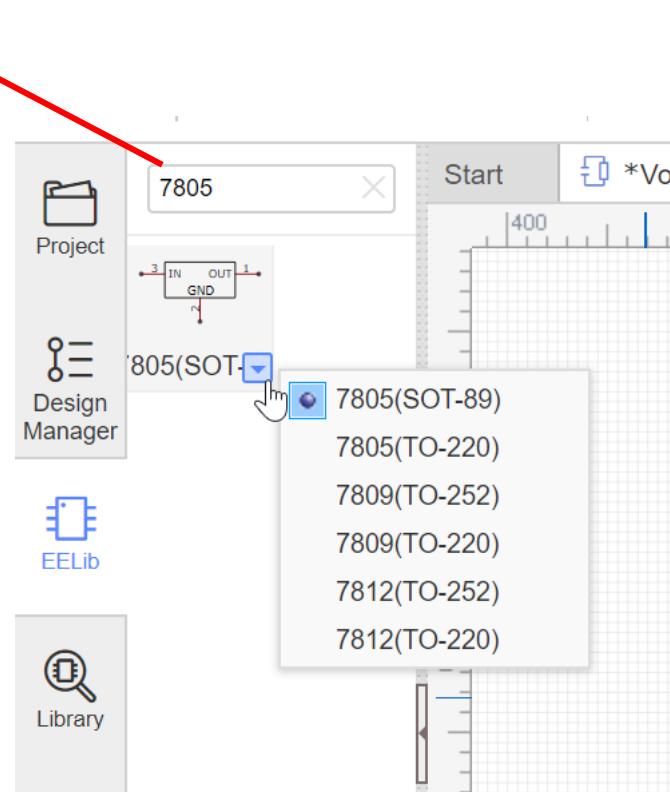


# 3. Build the circuit

- In EasyEdaLibrary (EEL) ->  
Find component in Filter box

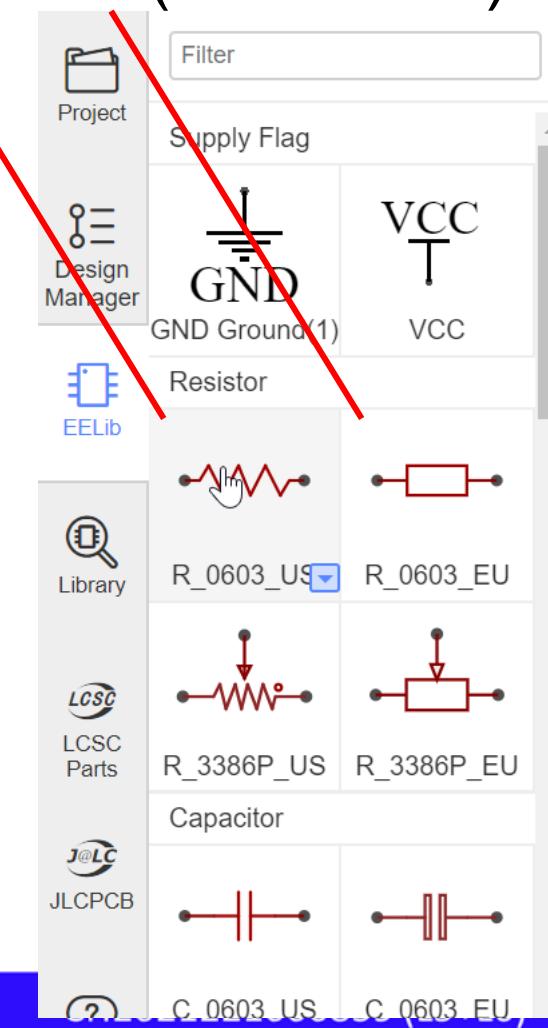
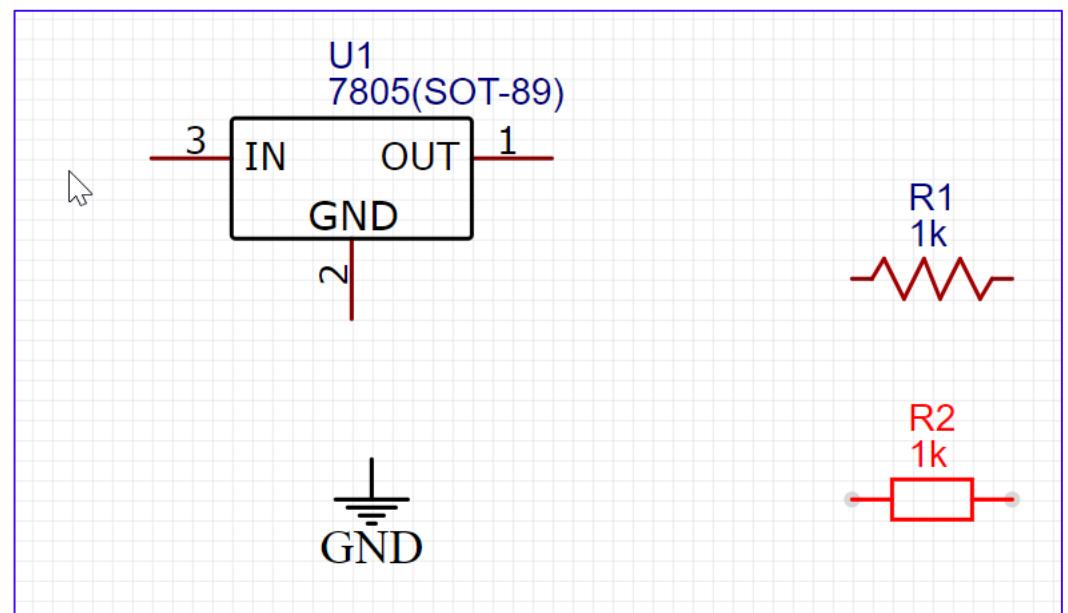


: 7805



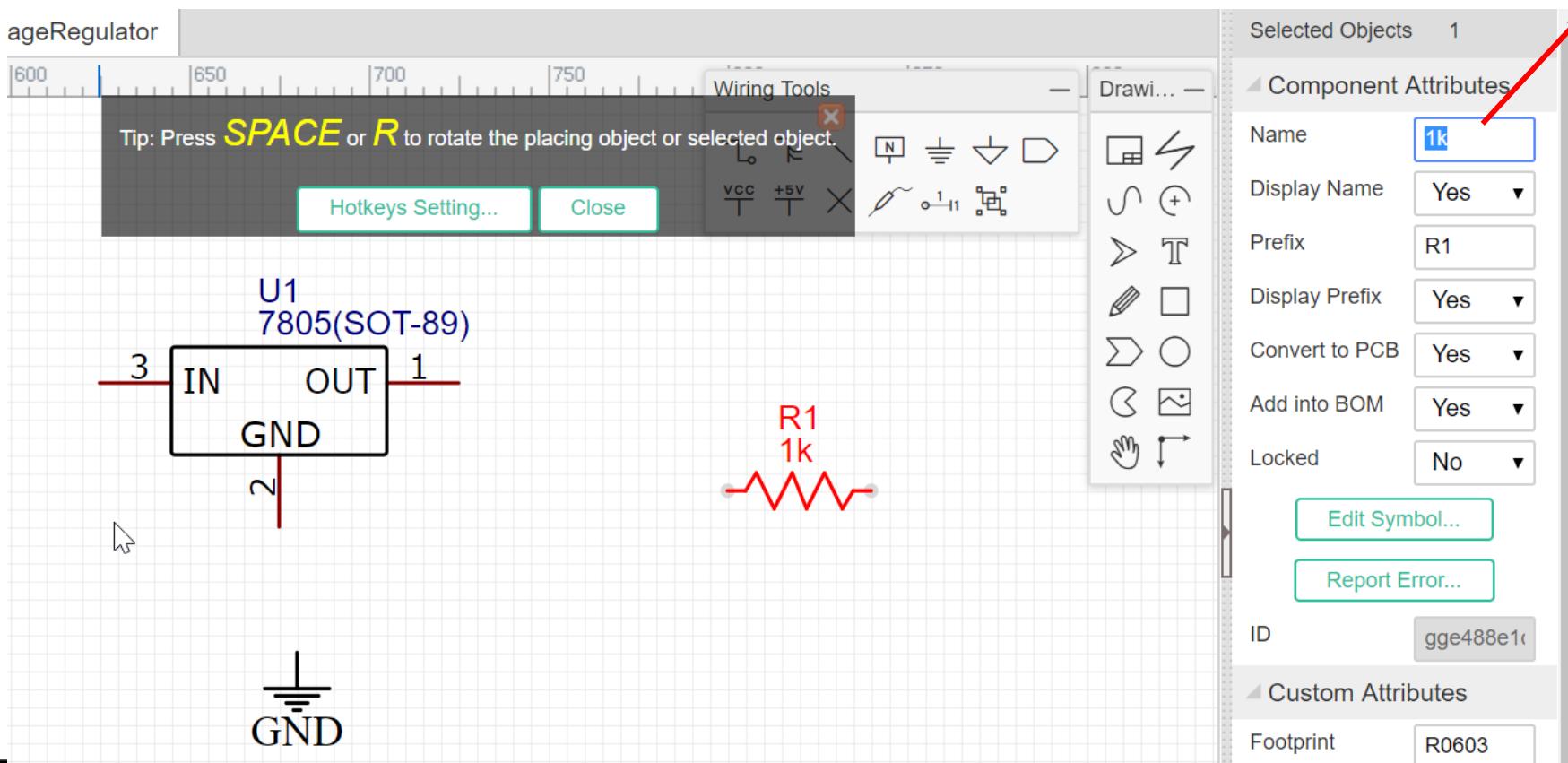
# 3. Build the circuit

- Place the 7805 Voltage regulator and resistor(US or UK) in the schematic window



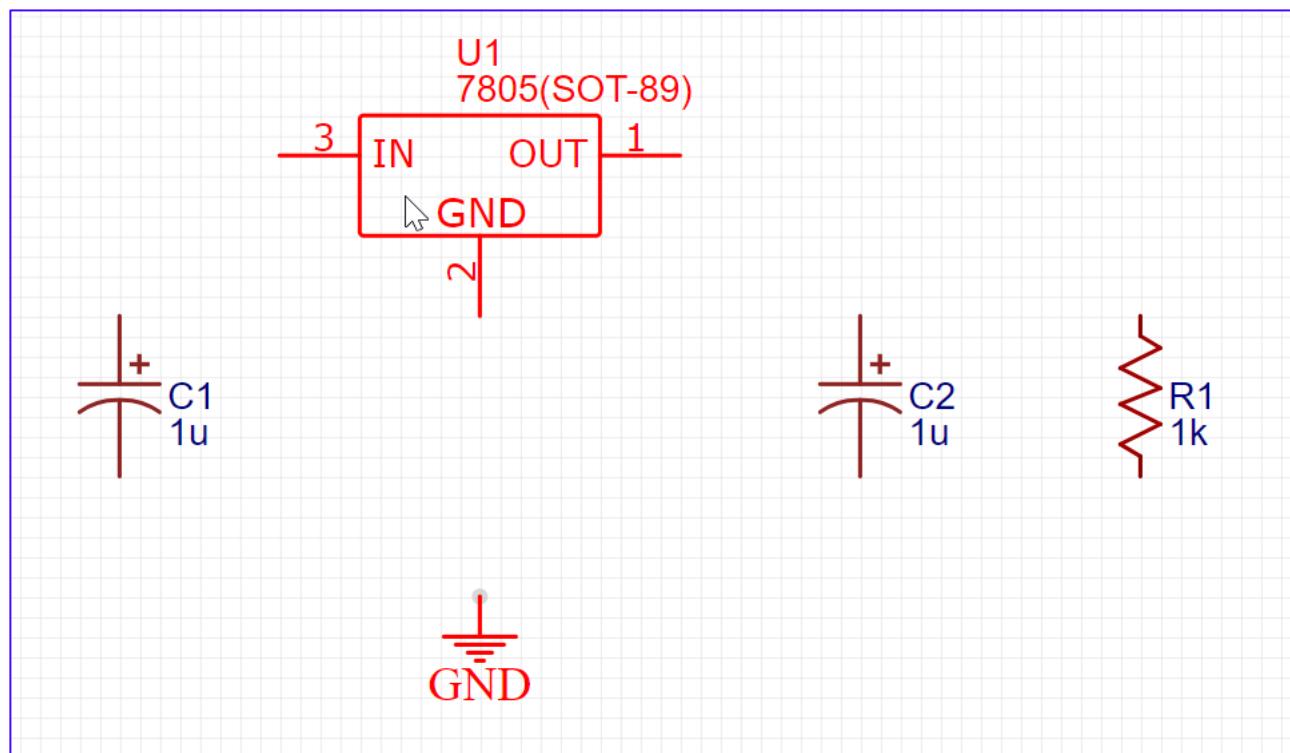
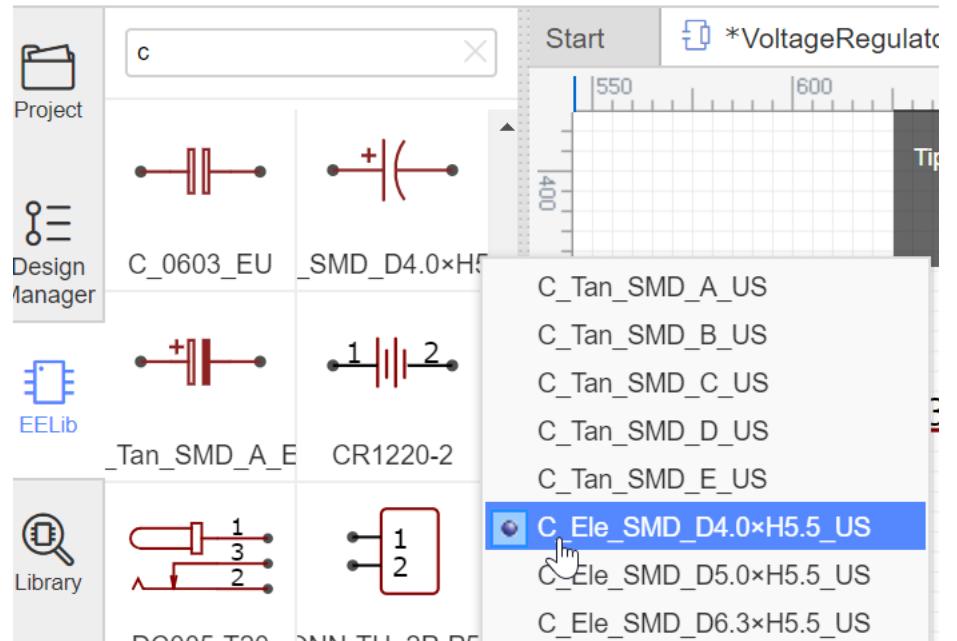
# 3. Build the circuit: Resistor

- I preferred US representation  
To modify the value of resistor, Select and edit component attributes



# 3. Build the circuit: Capacitor and Rotate

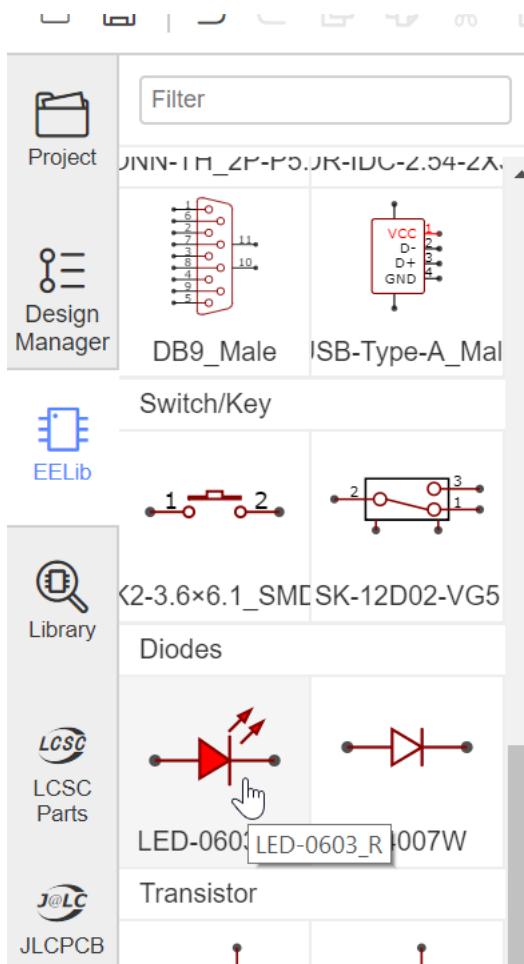
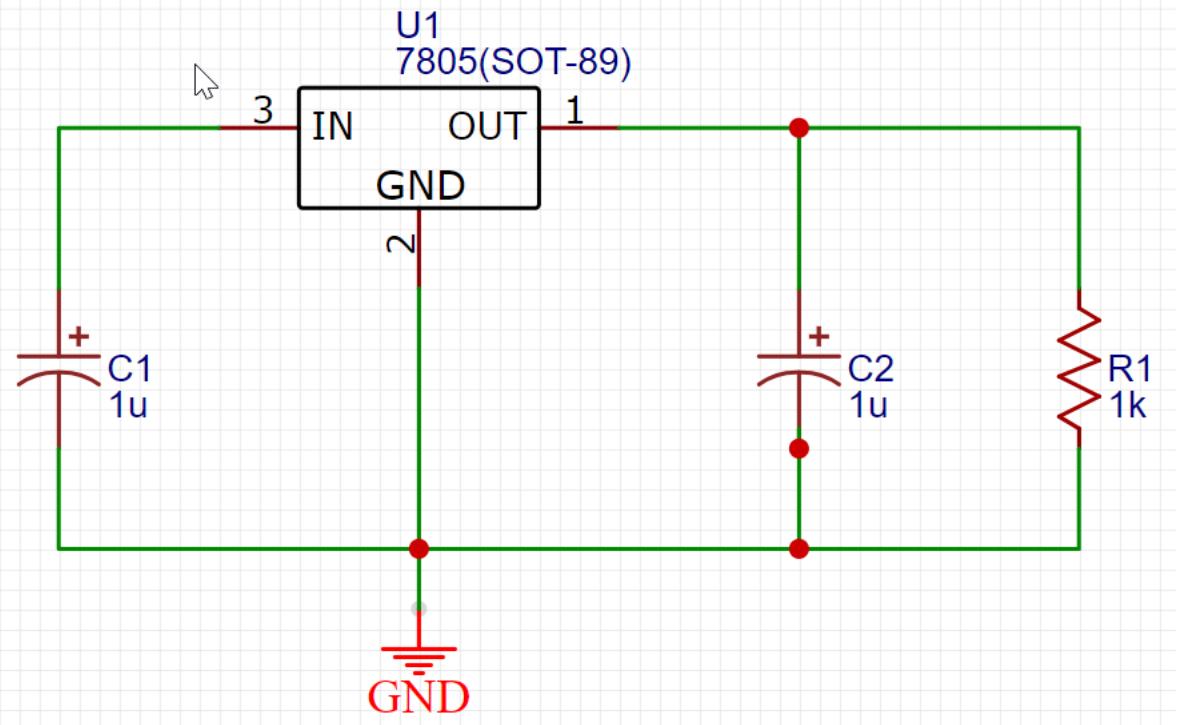
- Select C and select electrolytic capacitor



- Press R to rotate the selected component

# 3. Build the circuit: W for wire, select LED

- Connect all wires, select LED



# 3. Build the circuit: Select battery

Screenshot of the EasyEDA Library search interface:

Search Engine: EasyEDA (highlighted)

Keywords at least 3 characters:

Types: Symbol (highlighted), Footprint, Spice Symbol (highlighted with a red box), SCH Module, PCB Module, 3D Model

Classes: System (highlighted)

Filter sidebar:

- battery
- Analog Devices
- Digital Device
- Operational Amplifier
- Power Sources** (highlighted with a red box)
  - Controlled Source
  - Current Source
  - Spice Source
  - Three Phase Power
  - Voltage Source** (highlighted with a red box)
- Regulator
- Others

Title(PartNO)	Footprint	Description
Voltage_Source_VDC	HDR1X2	To use this symbol please see: <a href="http://ltwiki.org/LTspiceHelpXVII/">http://ltwiki.org/LTspiceHelpXVII/</a>
Behavioral_Voltage_Source_BV	HDR1X2	
Voltage_Source_VPULSE	HDR1X2	
Voltage_Source_VSFFM	HDR1X2	
Voltage_Source_VPWL	HDR1X2	
Voltage_Source_VEXP	HDR1X2	
Voltage_Source_VSin	HDR1X2	

Symbol preview: A circular voltage source symbol with a red circle around the positive terminal (+) and a yellow circle around the negative terminal (-).

Footprint preview: A black rectangle with two circular pads, one red and one yellow.

Navigation and actions:

EasyEDA > Spice Symbol > System > Voltage Source > Voltage\_Source\_VDC

Edit Place More Cancel

# 3. Build the circuit: Ammeter

The screenshot shows the EasyEDA Library search interface. The search term 'ammeter' is entered in the search bar, which is highlighted with a red box. The results table has columns for Title(PartNO), Footprint, and Owner. A blue row highlights the entry 'Ammeter copy' by saava, which is also highlighted with a red box. To the right of the table, there is a preview of an ammeter symbol, which is a circle with the letter 'A' inside.

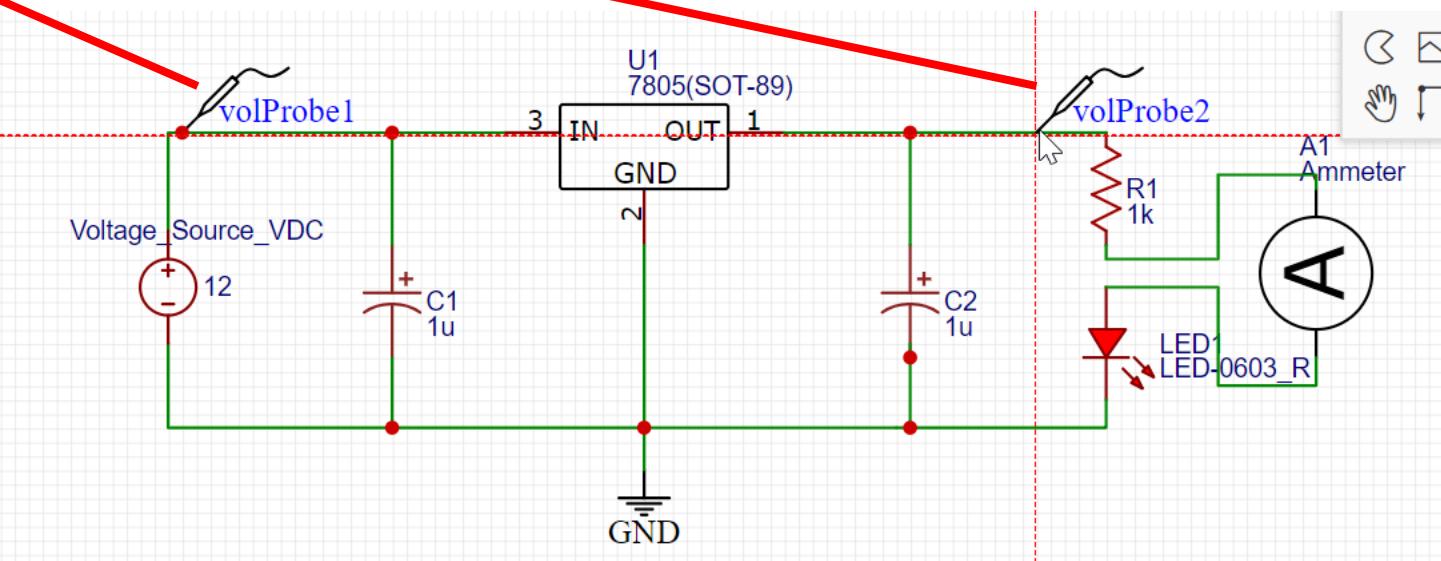
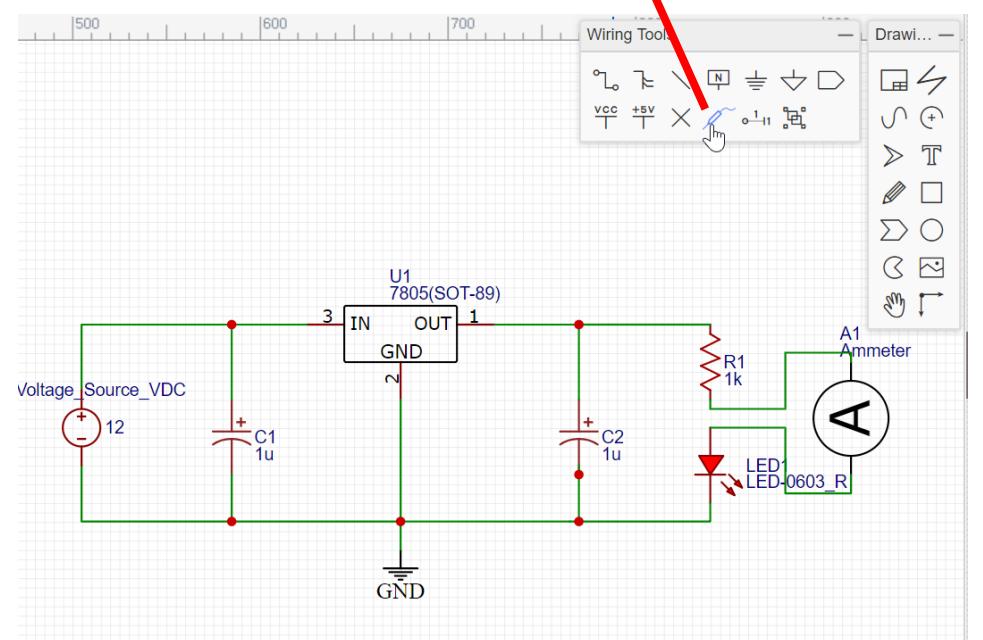
Title(PartNO)	Footprint	Owner
AMMETER	AMMETER	Summer 2020 - Design fo...
VOLT/AMMETER	VOLT/AMMETER	jukaloko
VOLT/AMMETER	VOLT/AMMETER	Junikos
AMMETER COPY	AMMETER	ทศพร อัญเชิญ
Voltmeter-Ammeter	VOLTMETER_AMMETER	Ed Paul Valencia
Voltmeter-Ammeter 200V 10A	VOLTMETER_AMMETER	kostecki
Voltmeter-Ammeter 200V 10A	VOLTMETER_AMMETER	Zrefer INC
Ammeter_wire_connector	AMMETER WIRE CONNECTOR	vitharana
DIGITAL VOLTMETER AMMETER	DIGITAL VOLTMETER AMMETER	hlitariandigital
<b>Ammeter copy</b>	<b>NONE</b>	<b>saava</b>
AC VOLT AND AMPEREMETER	NONE	ANDREW FISHER
Digital Voltmeter Ammeter5V	NONE	Scarlett Yau
Digital Voltmeter Ammeter12V	NONE	Scarlett Yau

EasyEDA > Symbol > User Contributed > Ammeter copy

Edit Place More Cancel

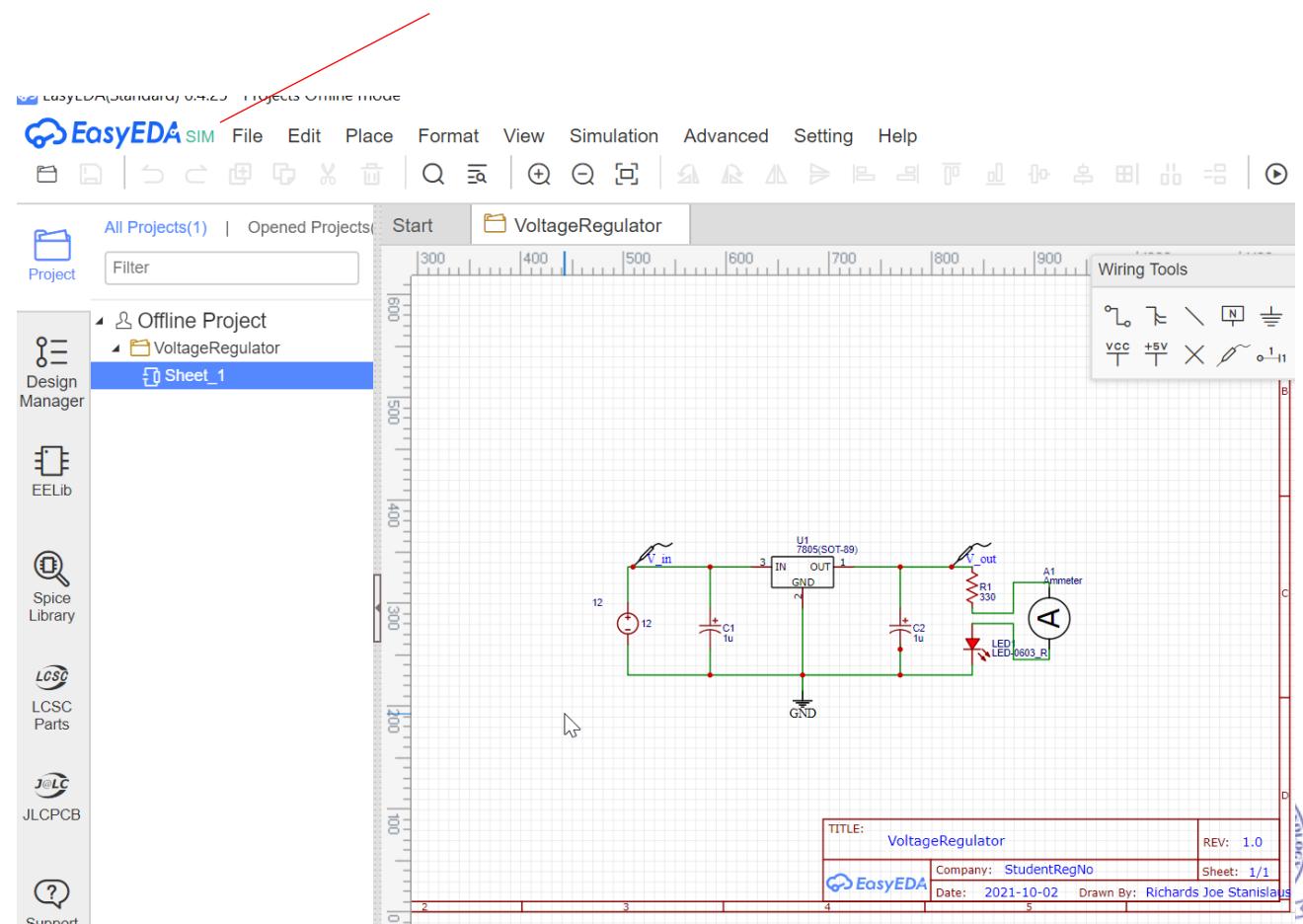
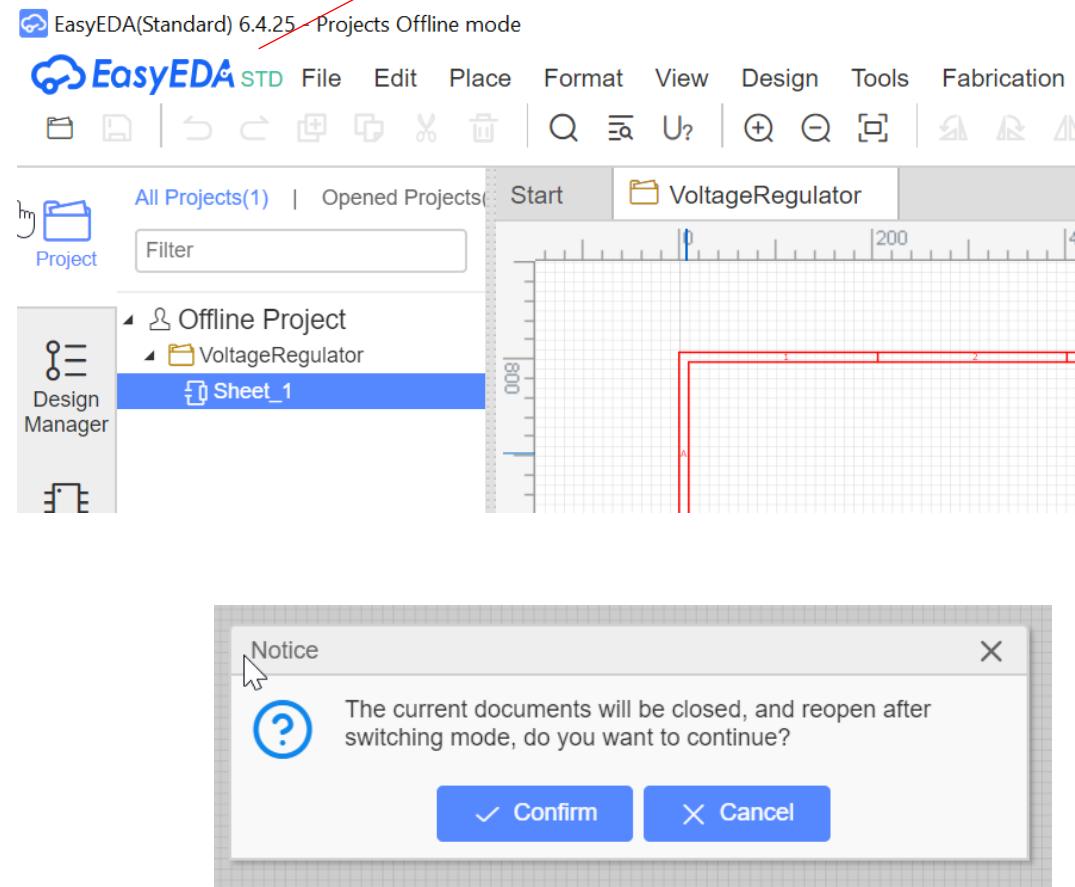
# 3. Build the circuit: Complete the circuit and place voltage probe

- Voltage probe



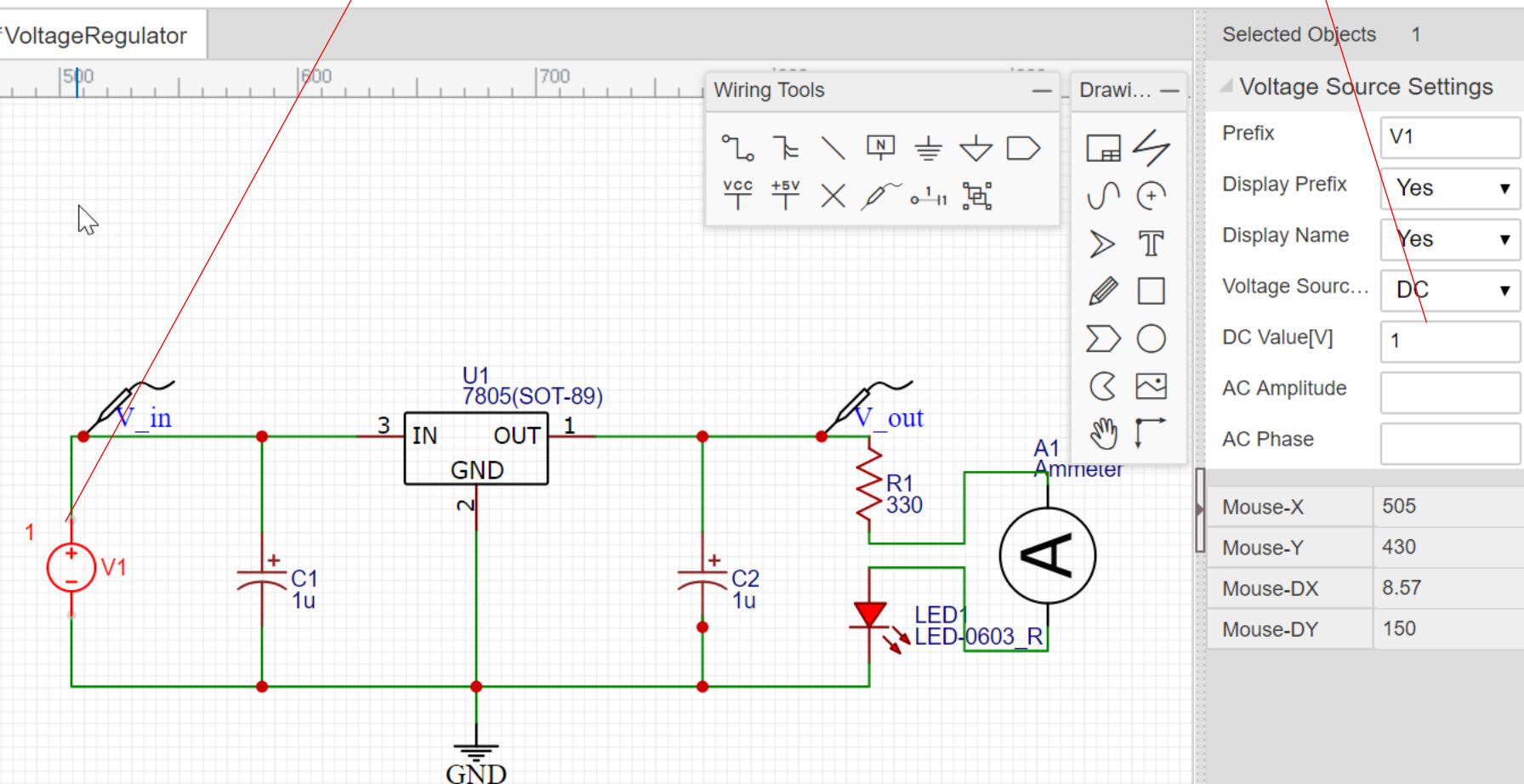
# 4. Switch from standard mode to Simulation mode

- Click at STD and confirm after saving -> Simulation mode



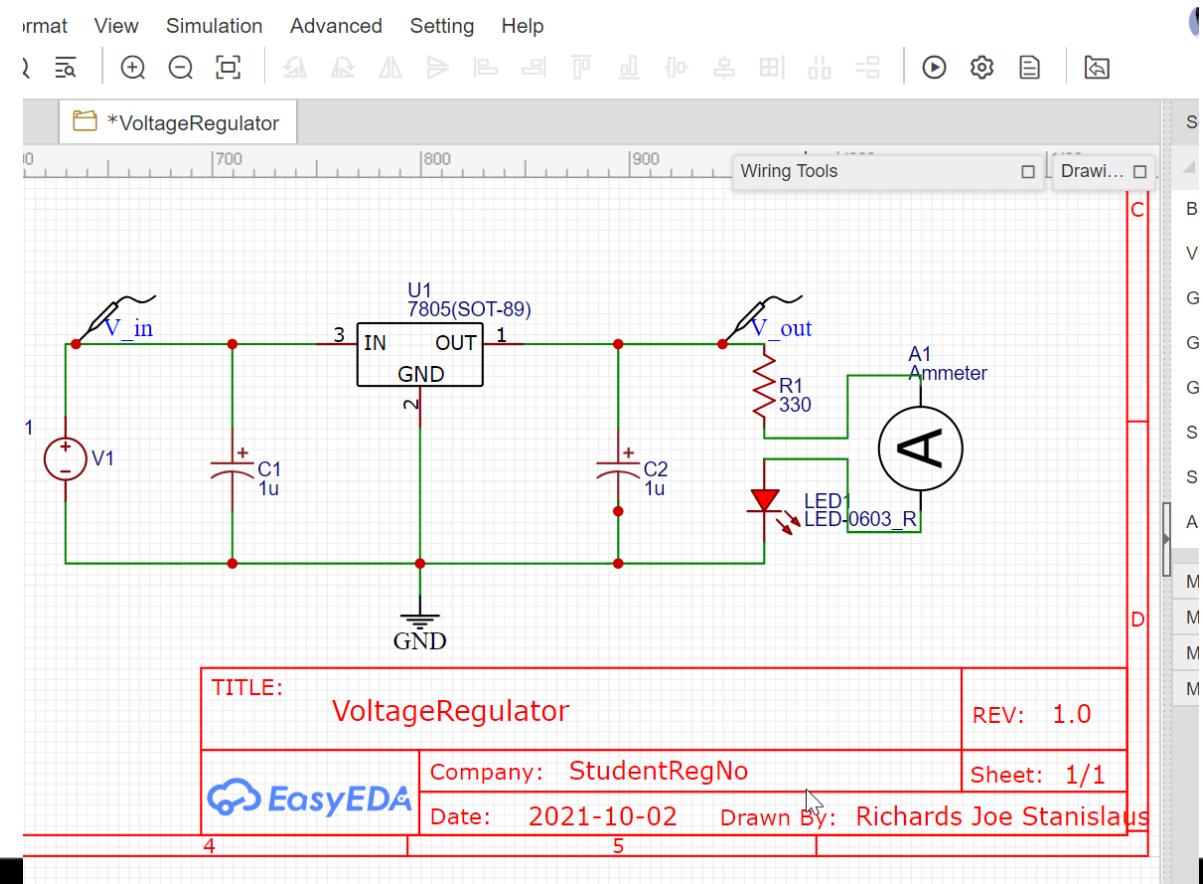
# 5. Voltage source value to 12V

- Select voltage source and change DC value = 12 in voltage source settings



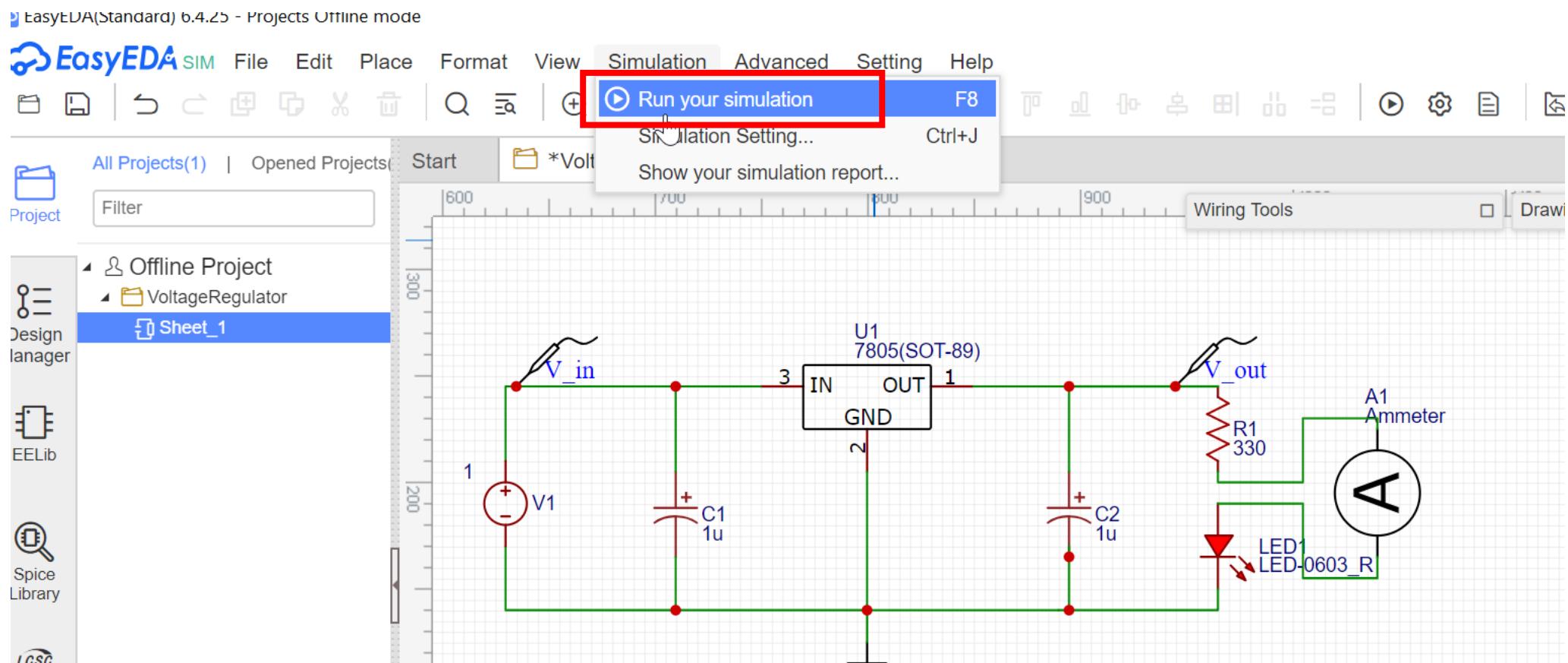
# 6. Move circuit near Student info

- Select all using mouse and move the circuit to the bottom right of schematic so that the name, registration number and title are visible.



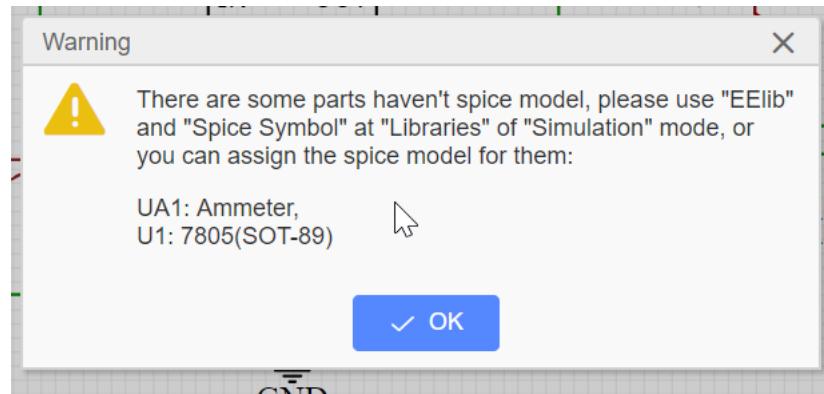
# 7. Run Simulation

- Note: If you do not find Simulation, then you have to perform step 4 (switching from standard to simulation mode)



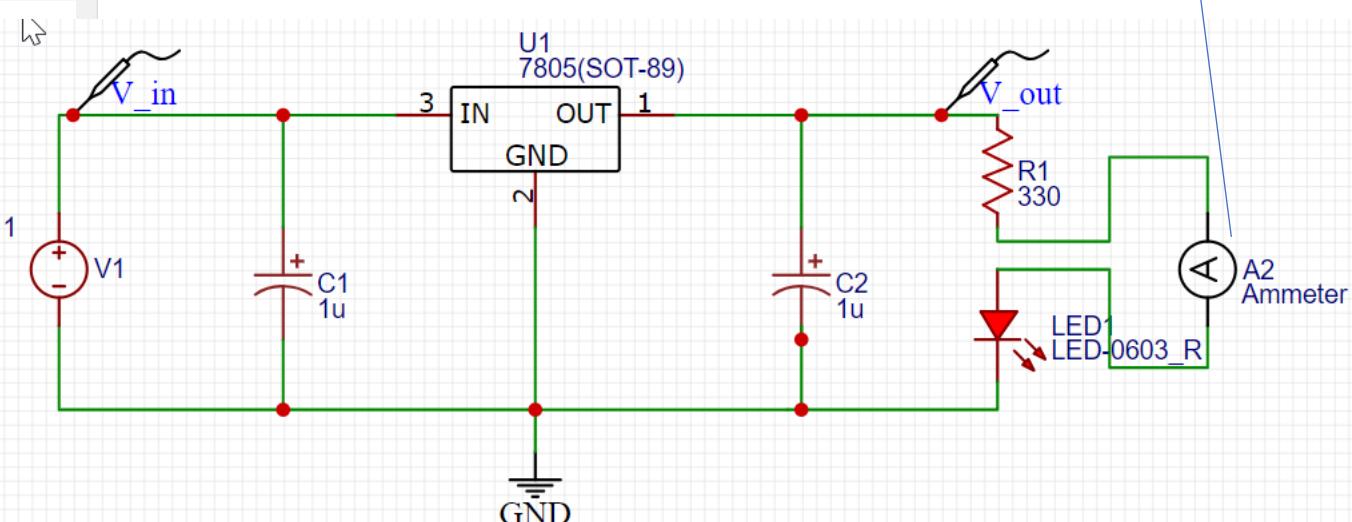
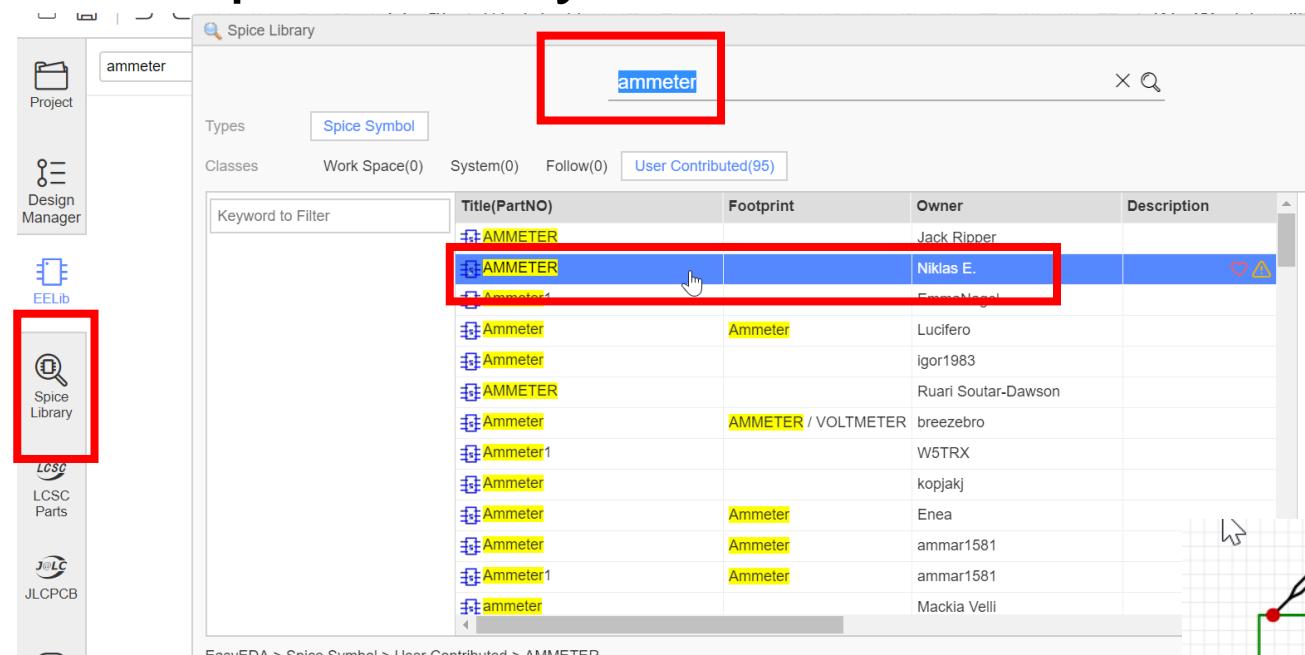
# 7. Run Simulation

- It will show error as we have to select the ammeter and voltage regulator in Spice symbol / Eelib inside the Simulation mode



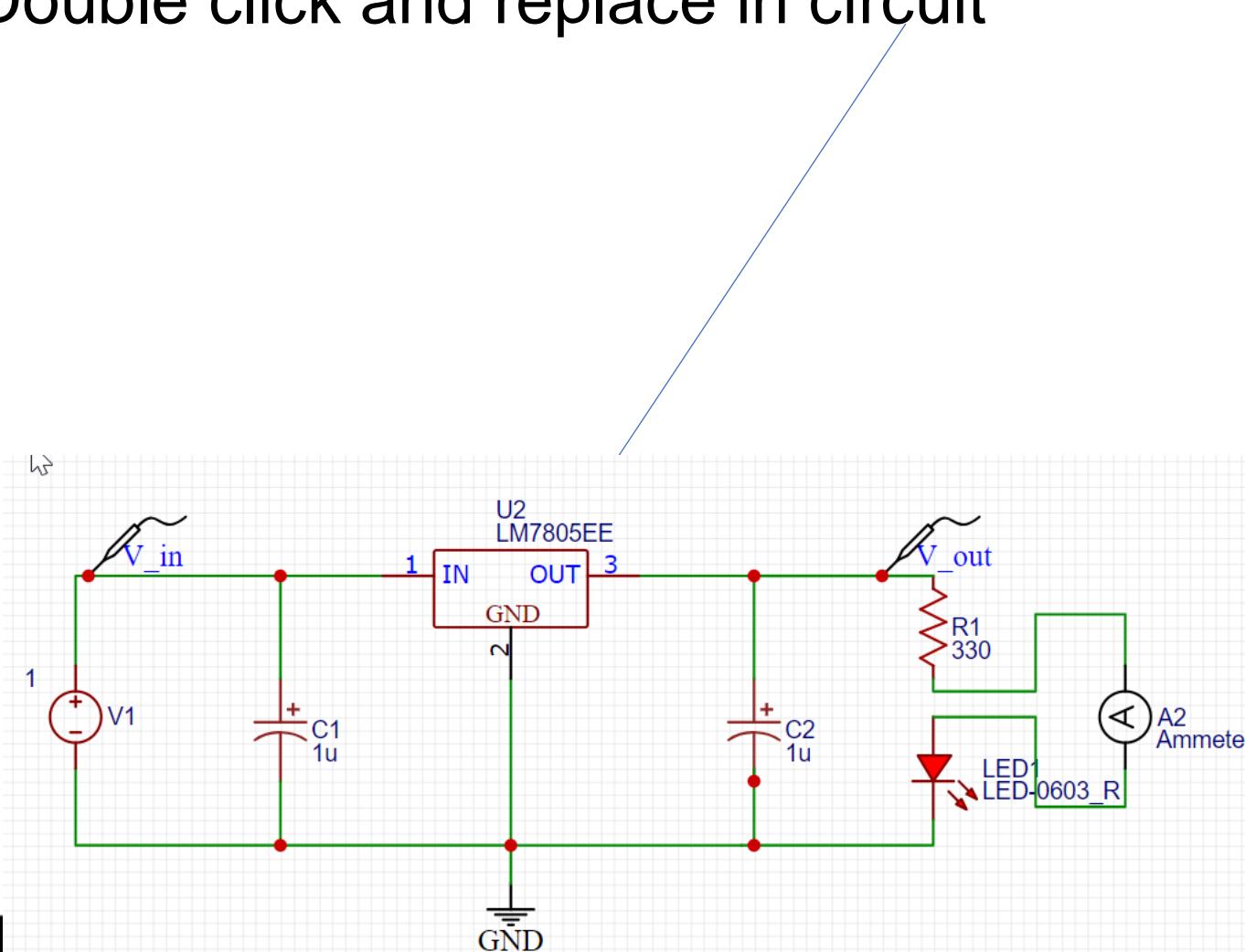
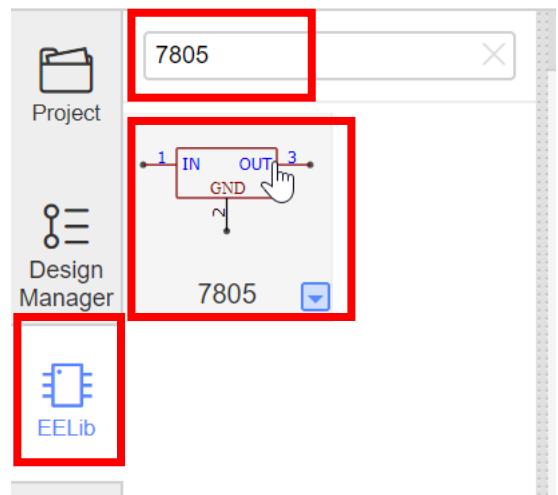
# 8. Update the circuit (Replace ammeter)

- Spice library -> Search ammeter -> Double click and replace in circuit



# 8. Update the circuit (Replace Voltage regulator)

- EELib → Search 7805 → Double click and replace in circuit



# 8. Update the circuit (Replace LED)

- Spice library -> DLEDR -> Double click and replace in circuit

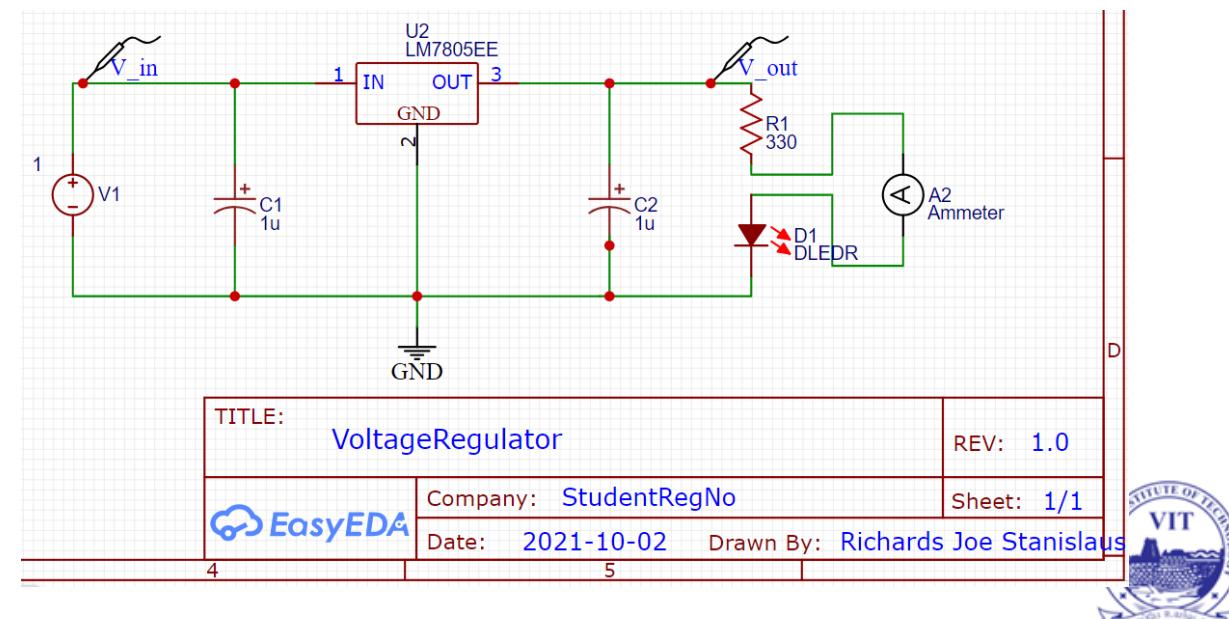
Spice Library

### DLEDR

Title(PartNO)	Footprint	Owner	Description
DLEDR	LED3MM	andyfierman	A basic red LED model
Diedr copy		nuriflyer	A basic red LED model
Diedr_custom		nuriflyer	A basic red LED model

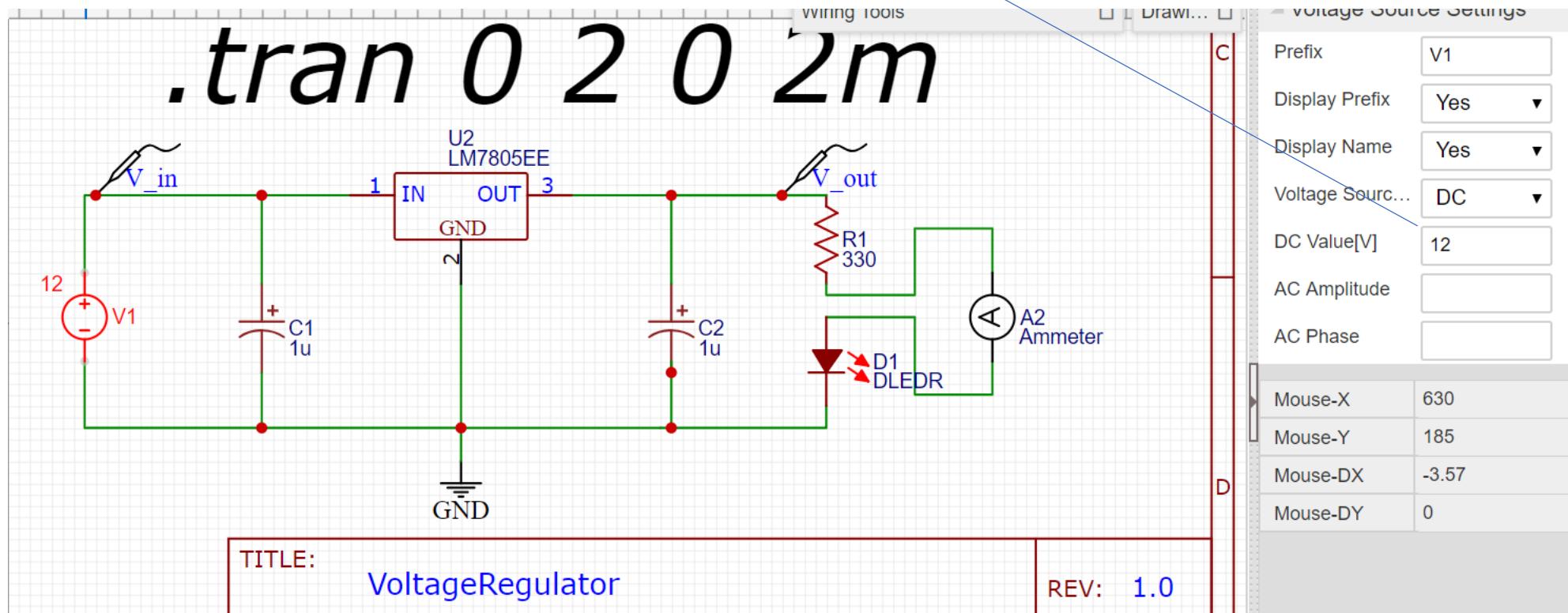
Spice Symbol

Information  
This library check before

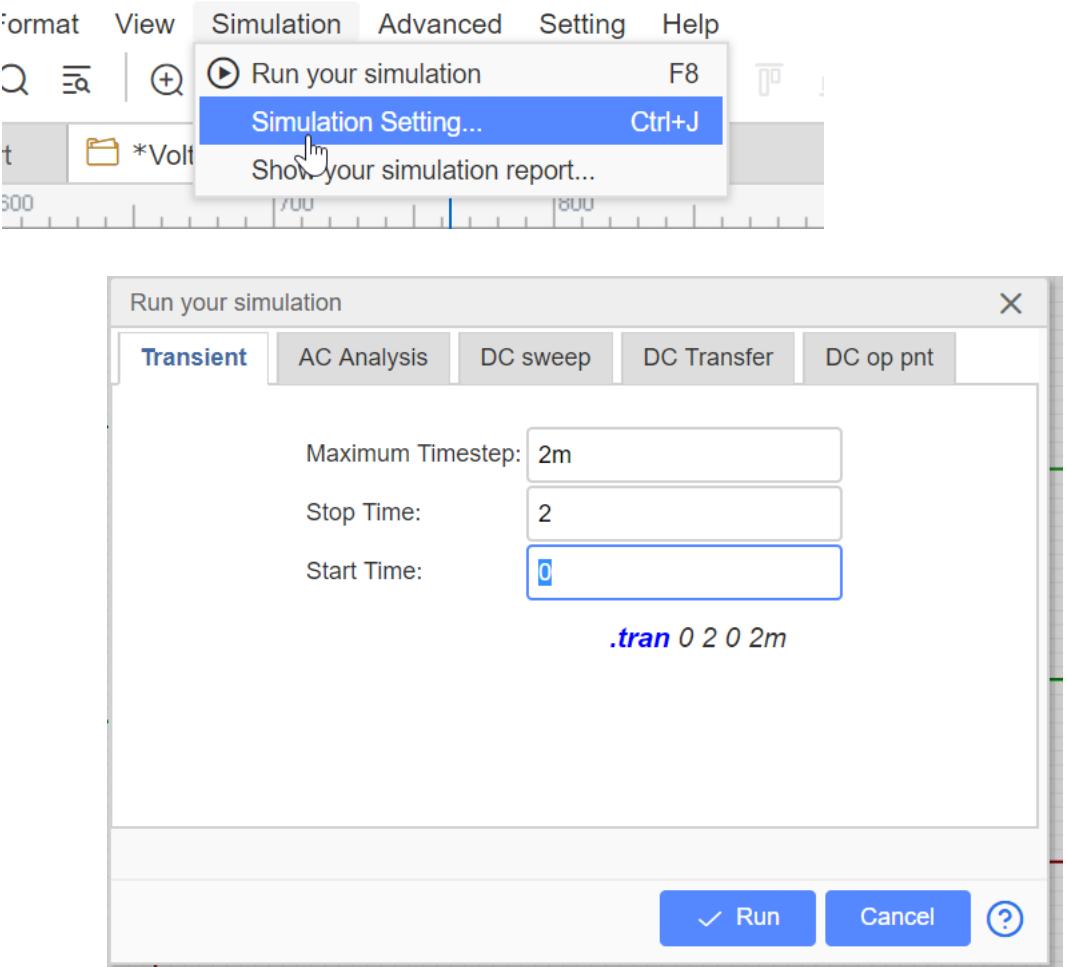


# 8. Update the circuit (Update Voltage source value if not updated)

- Select voltage source and update value to 12



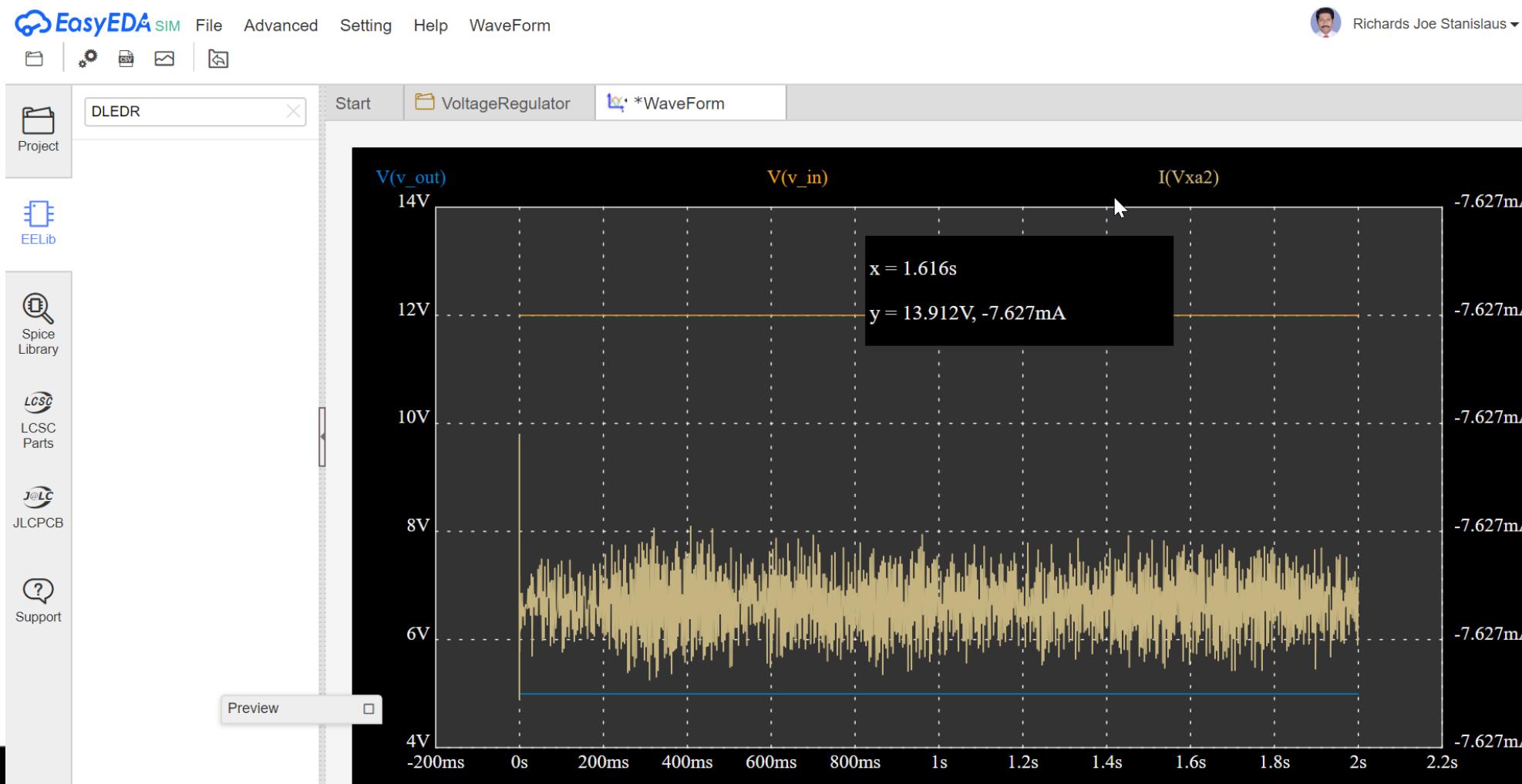
# 9. Run simulation



- Note, if waveform does not pop up, then you have to check if you have updated the components correctly.

# 9. Run simulation

- Note, if waveform does not pop up, then you have to check if you have updated the components correctly.



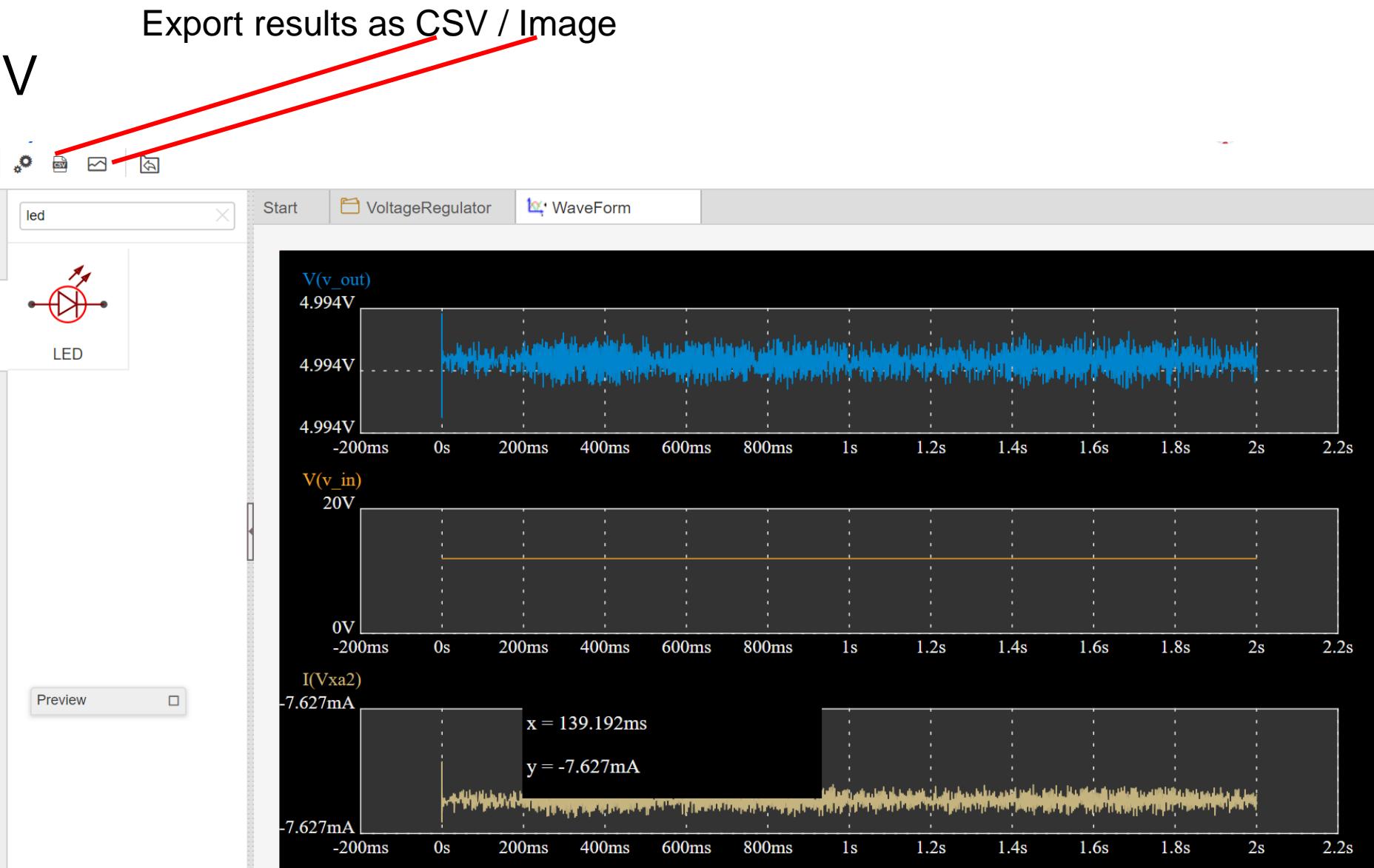
# 10. Waveform: Change to individual panes

- Select waveform config



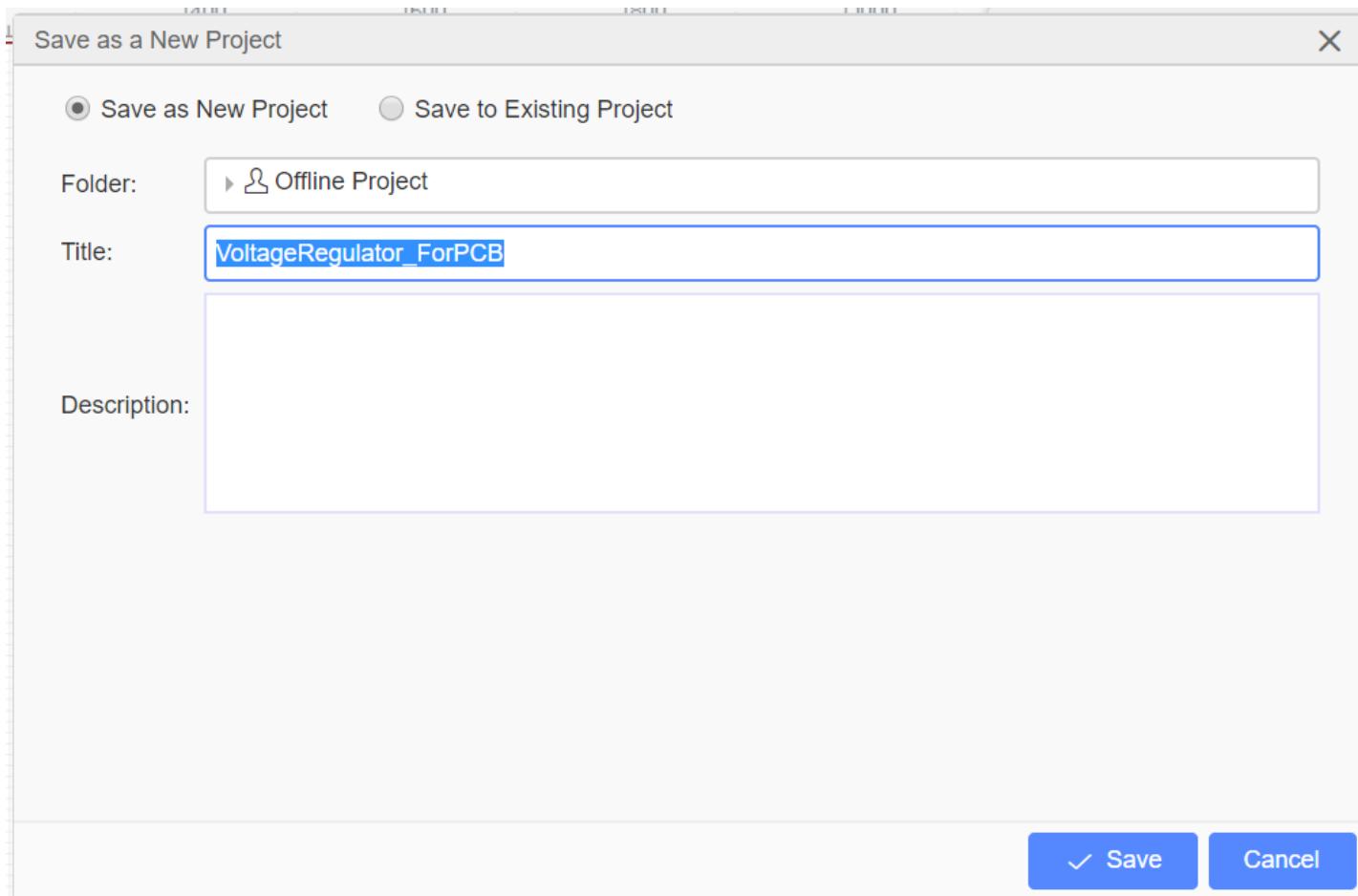
# 10. Waveform: Change to individual panes

- $V_{out} = 4.994V$
- $V_{in}=12V$
- $I=7.627mA$



# 11. Steps to convert Schematic to PCB

- File-> Save as ->VoltageRegulatorForPCB



# 11. Steps to convert Schematic to PCB

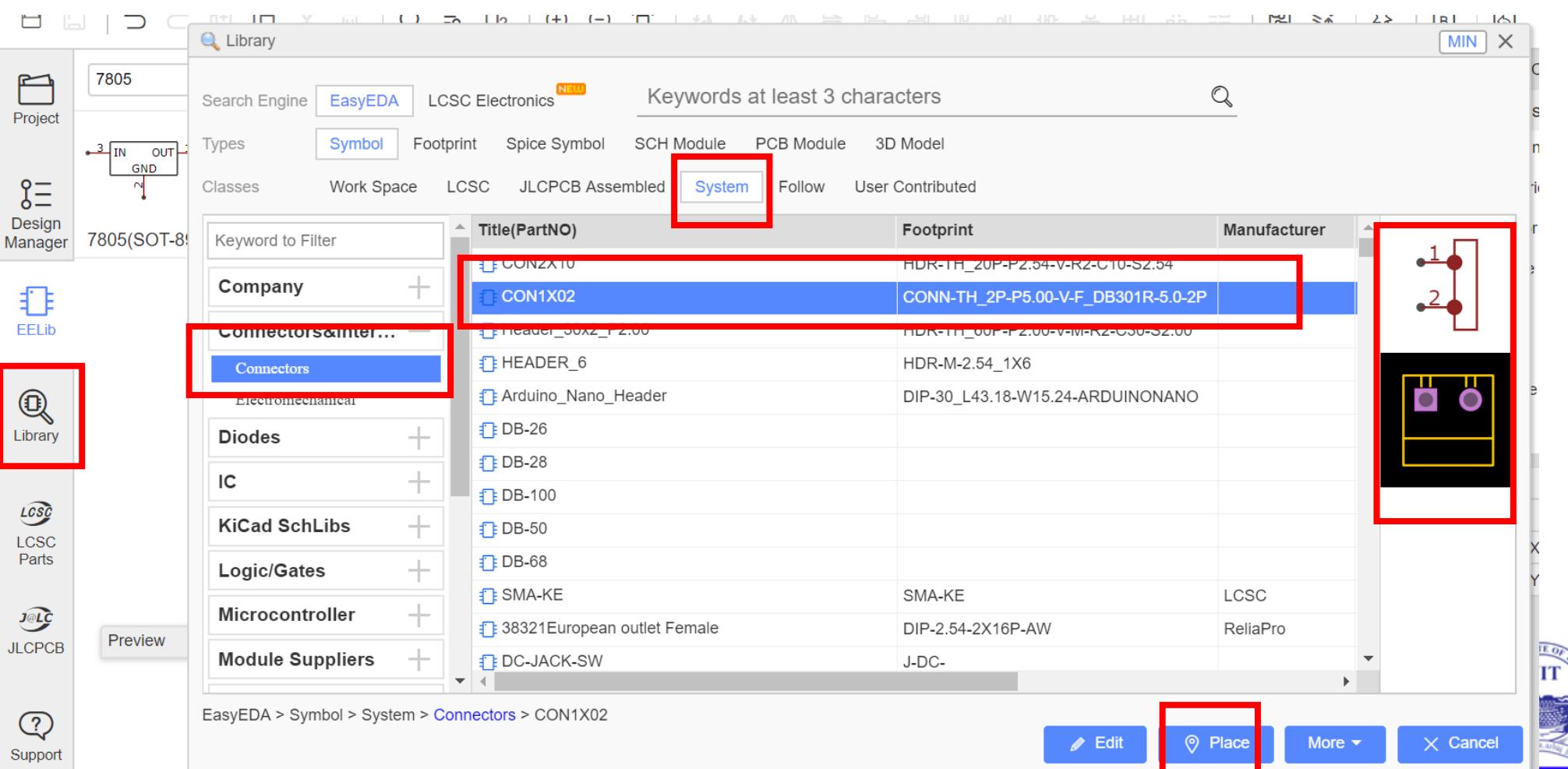
- Remove voltage sources, Voltage probe, Ammeter, transient command
- Change the settings from SIMulation settings to Standard settings



# 11. Steps to convert Schematic to PCB

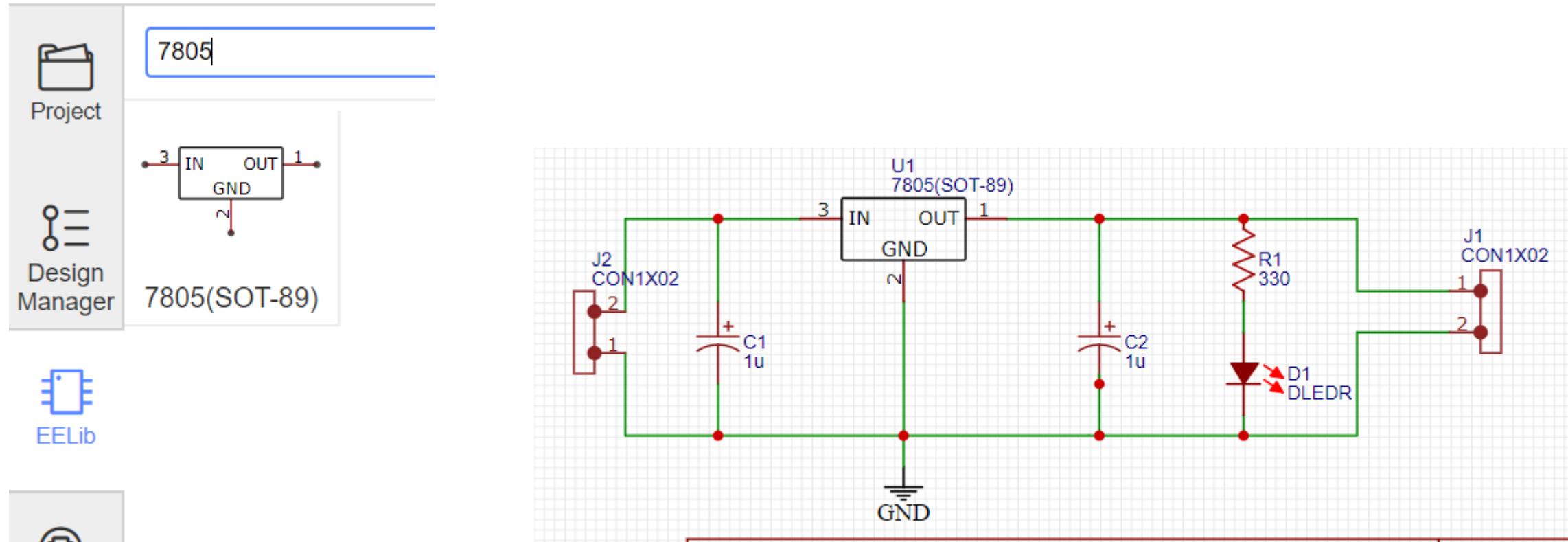
- Library-> System-> Connectors-> CON1x02

- Connect  
the  
connectors



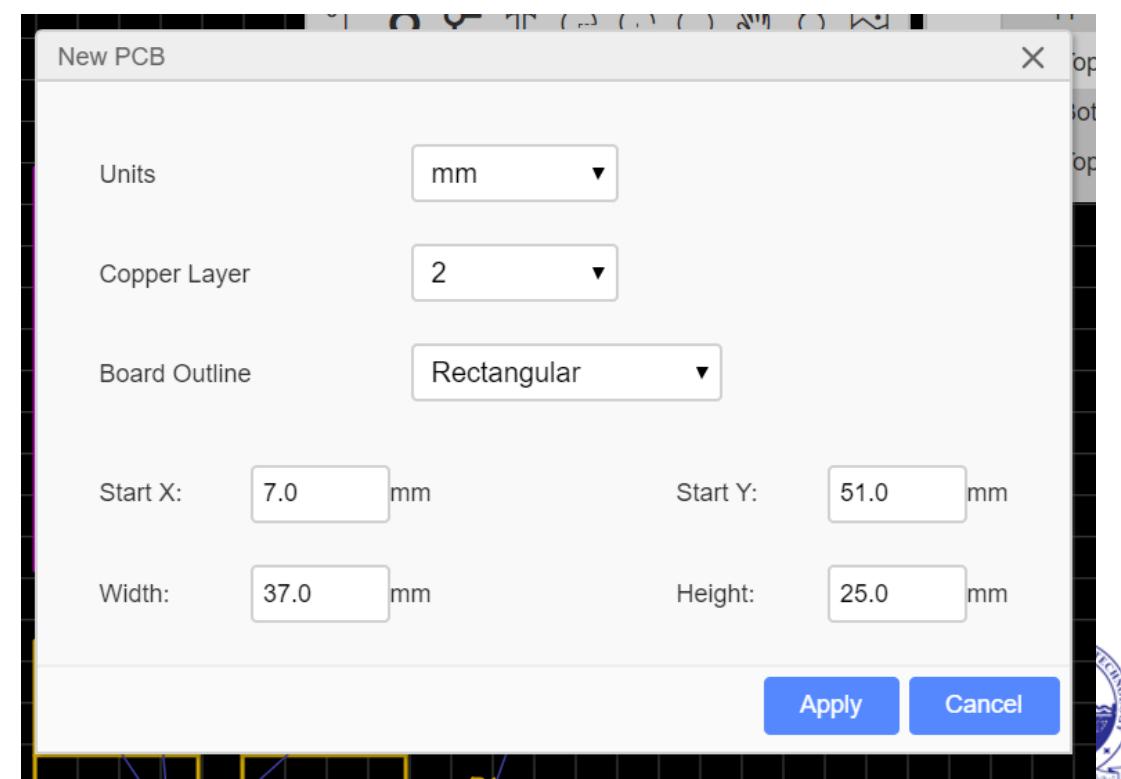
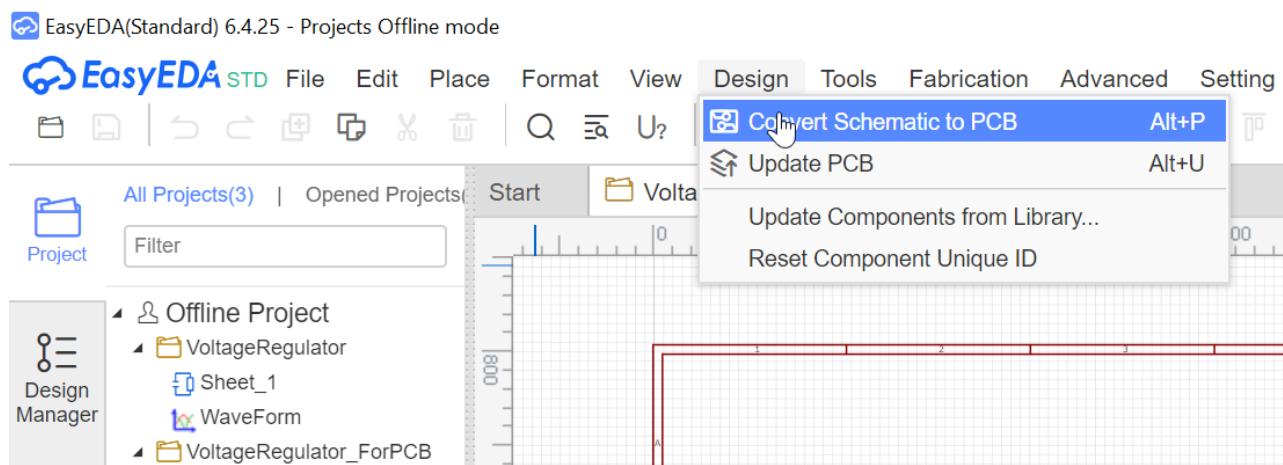
# 11. Steps to convert Schematic to PCB

- Also change the 7805 regulator as it should have schematic



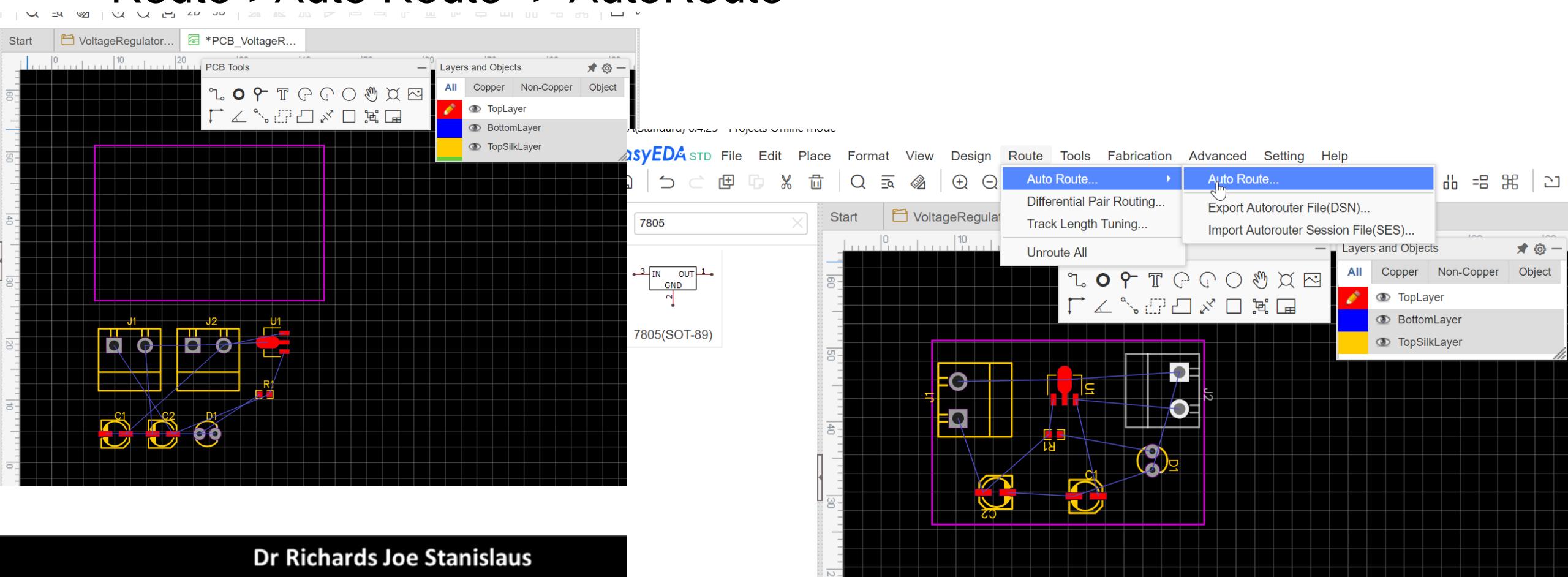
# 11. Steps to convert Schematic to PCB

- Save the project
- With Standard settings
- Design-> Convert schematic to PCB

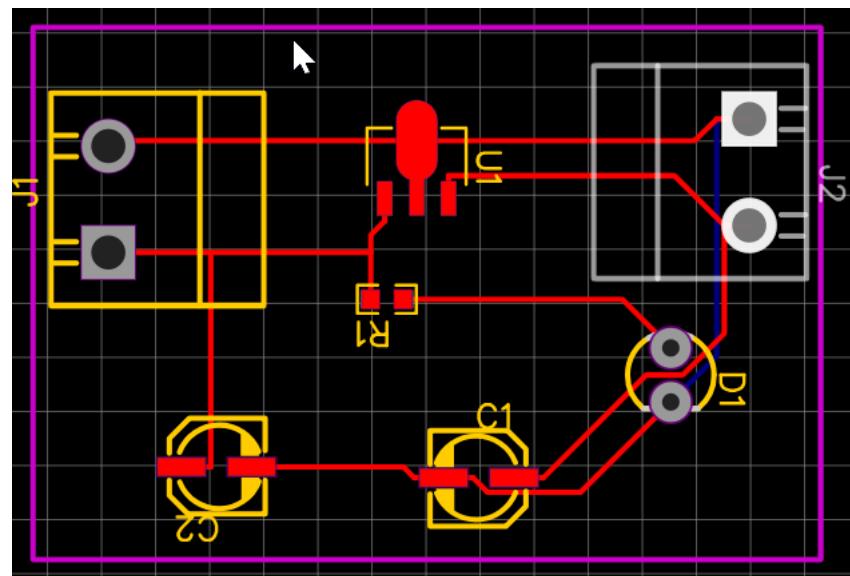
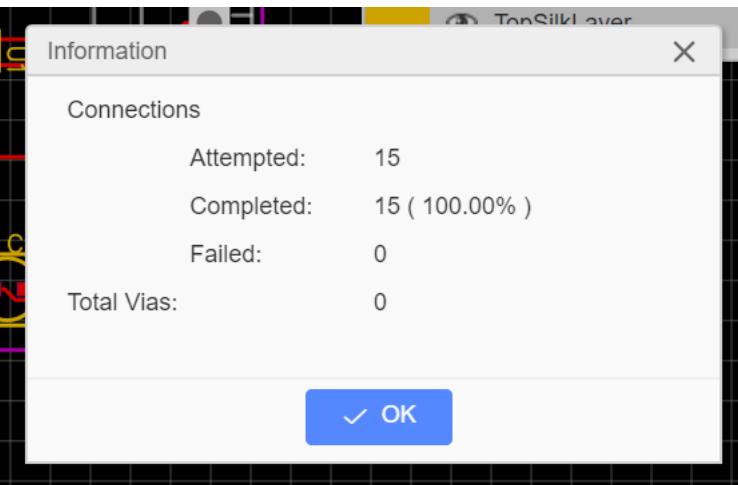
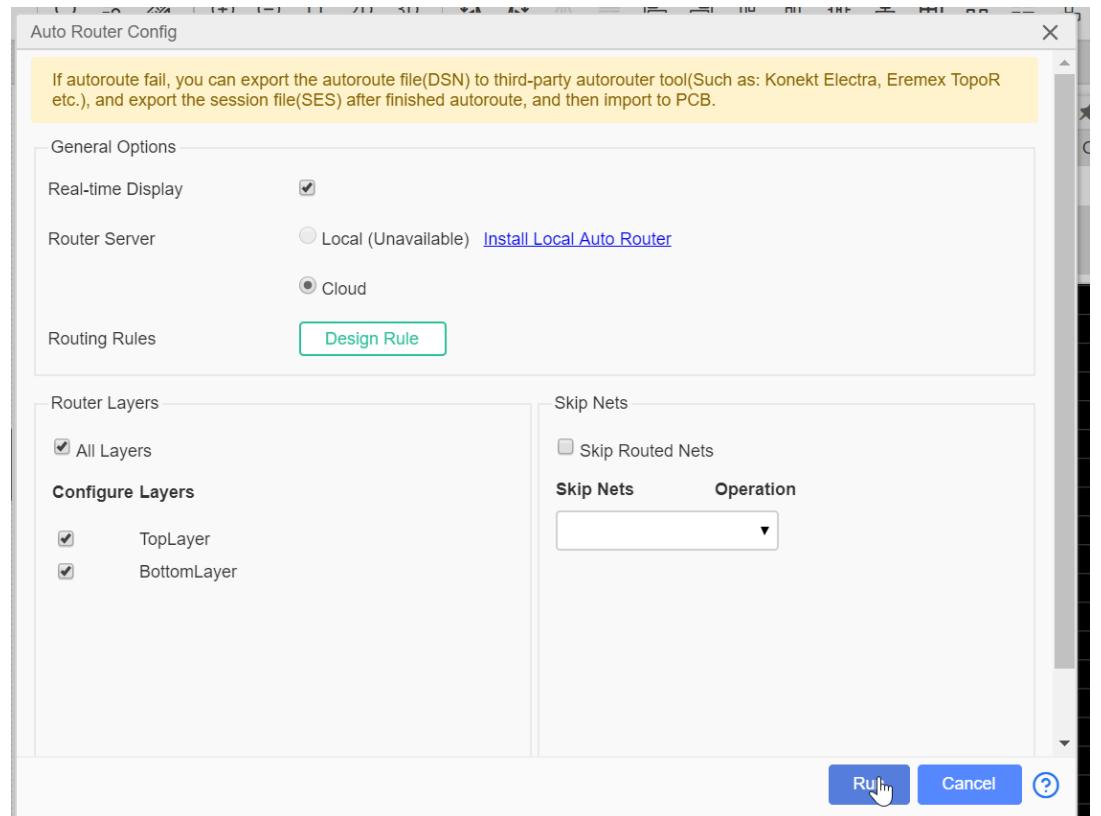


# 12. Steps to convert Schematic to PCB

- Drag all components inside board outline
- Route->Auto Route -> AutoRoute

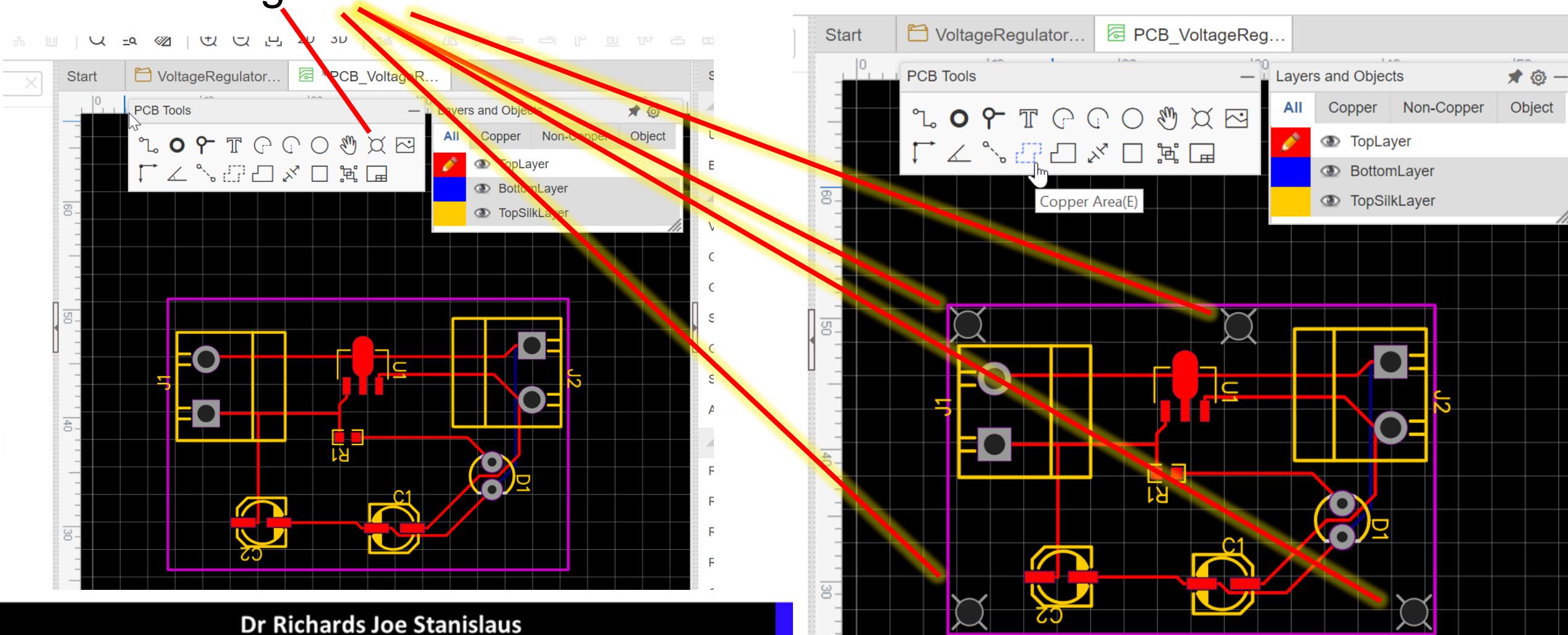


# 12. Steps to convert Schematic to PCB



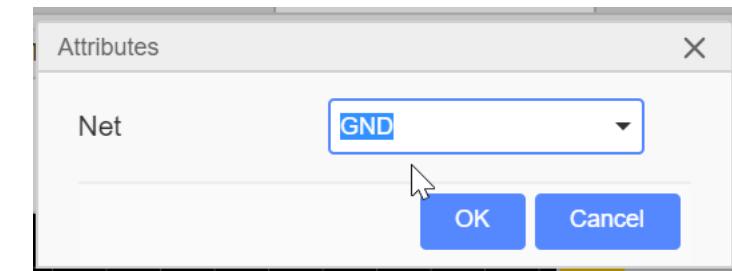
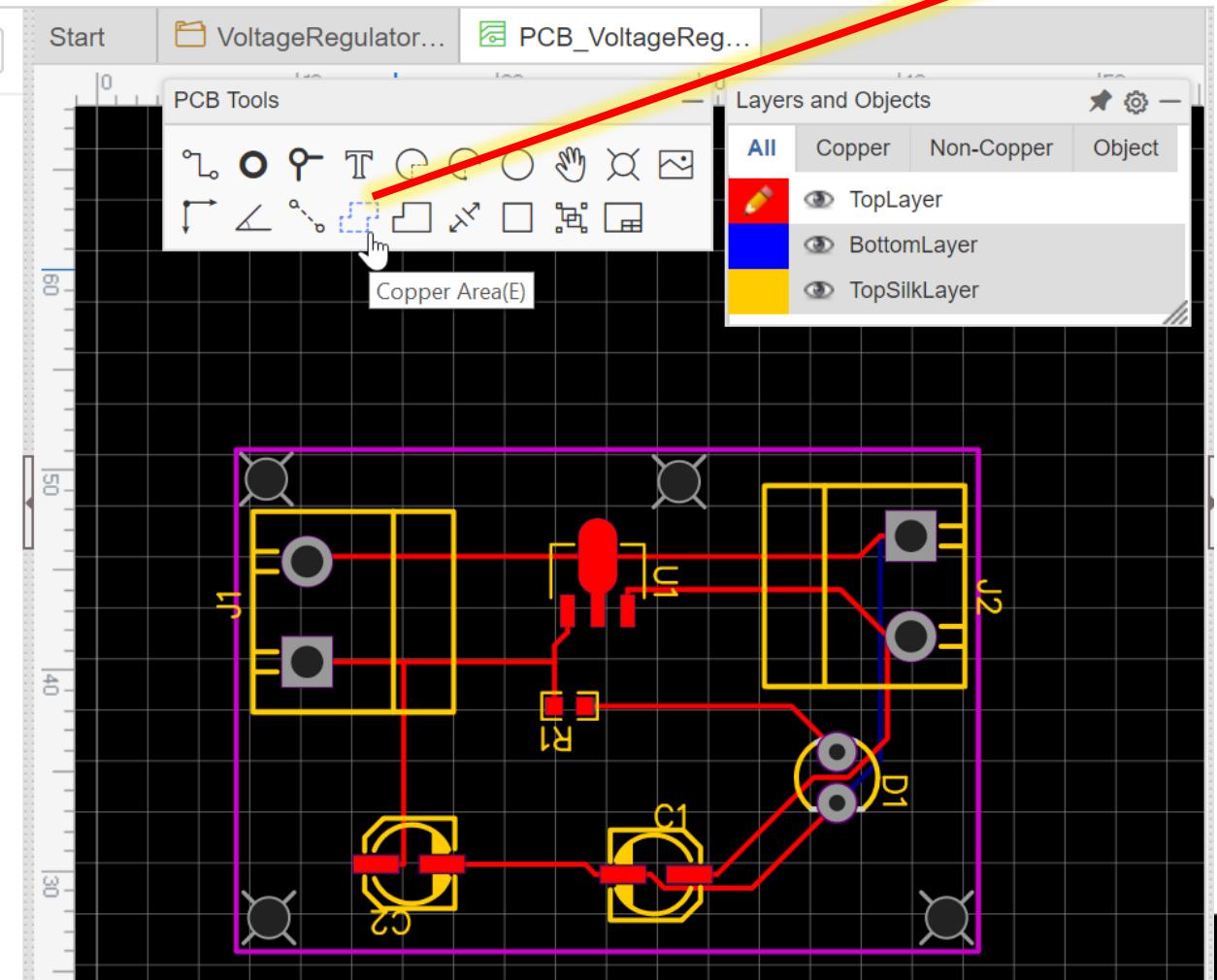
# 12. Steps to convert Schematic to PCB

- Mounting holes: To mount the PCB on external device



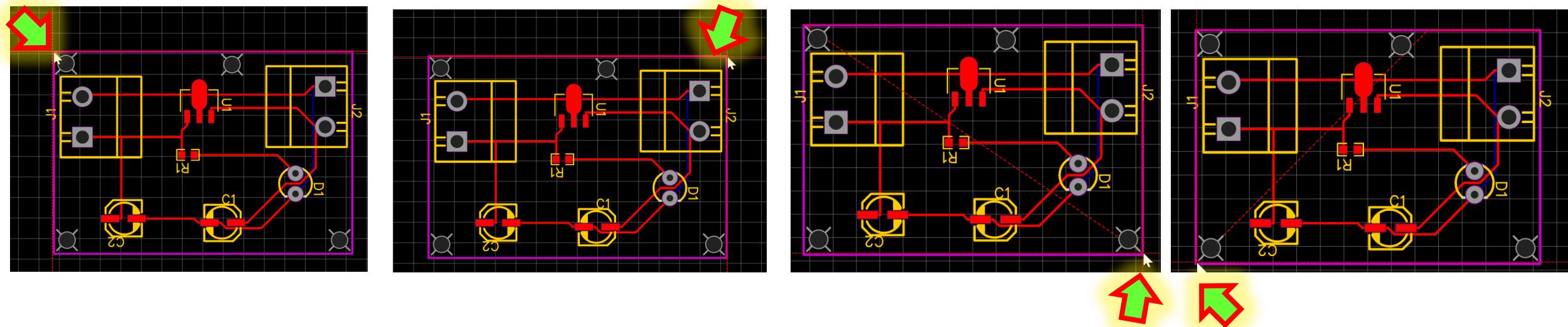
# 12. Steps to convert Schematic to PCB

- To create ground, select copper area

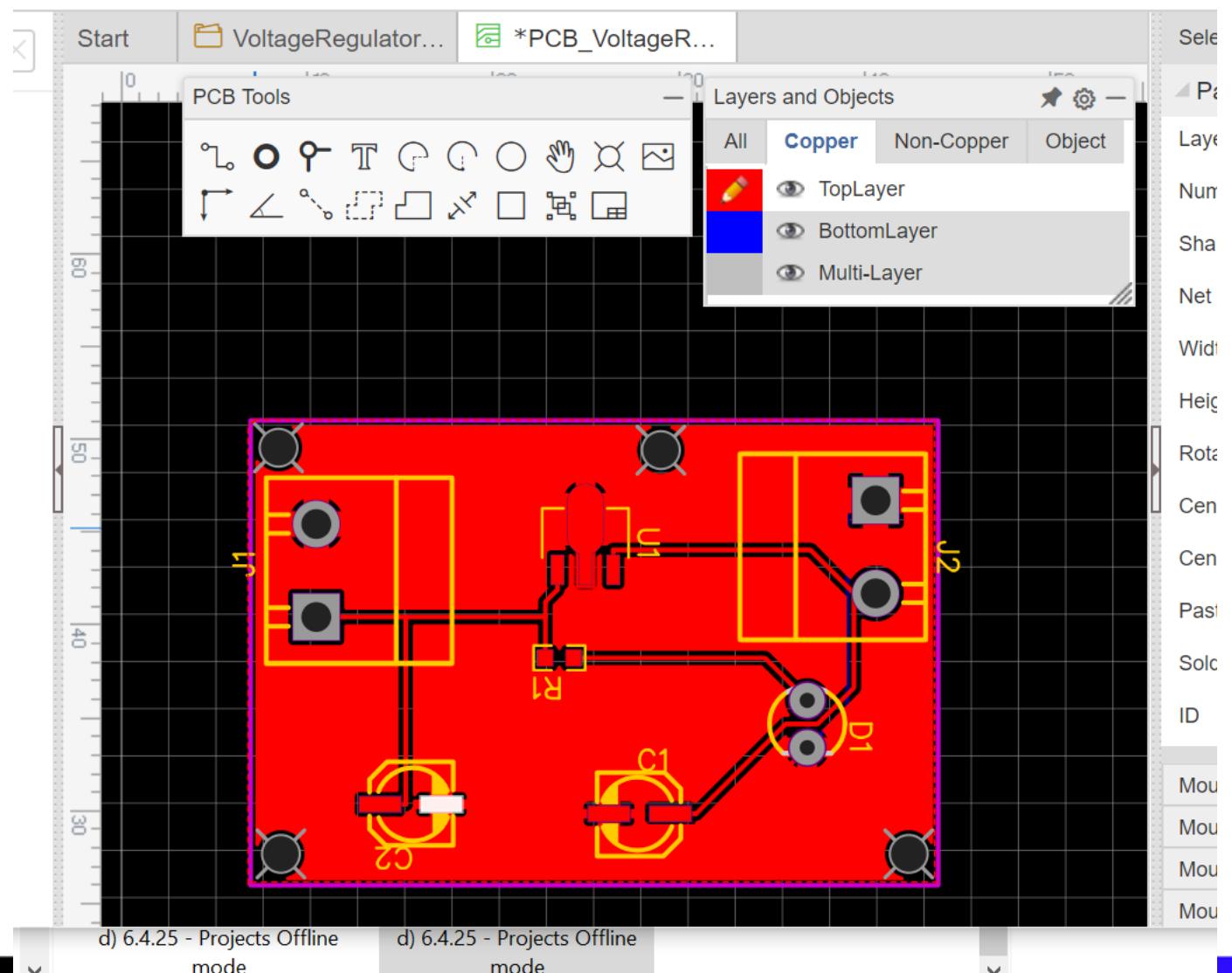


# 12. Steps to convert Schematic to PCB

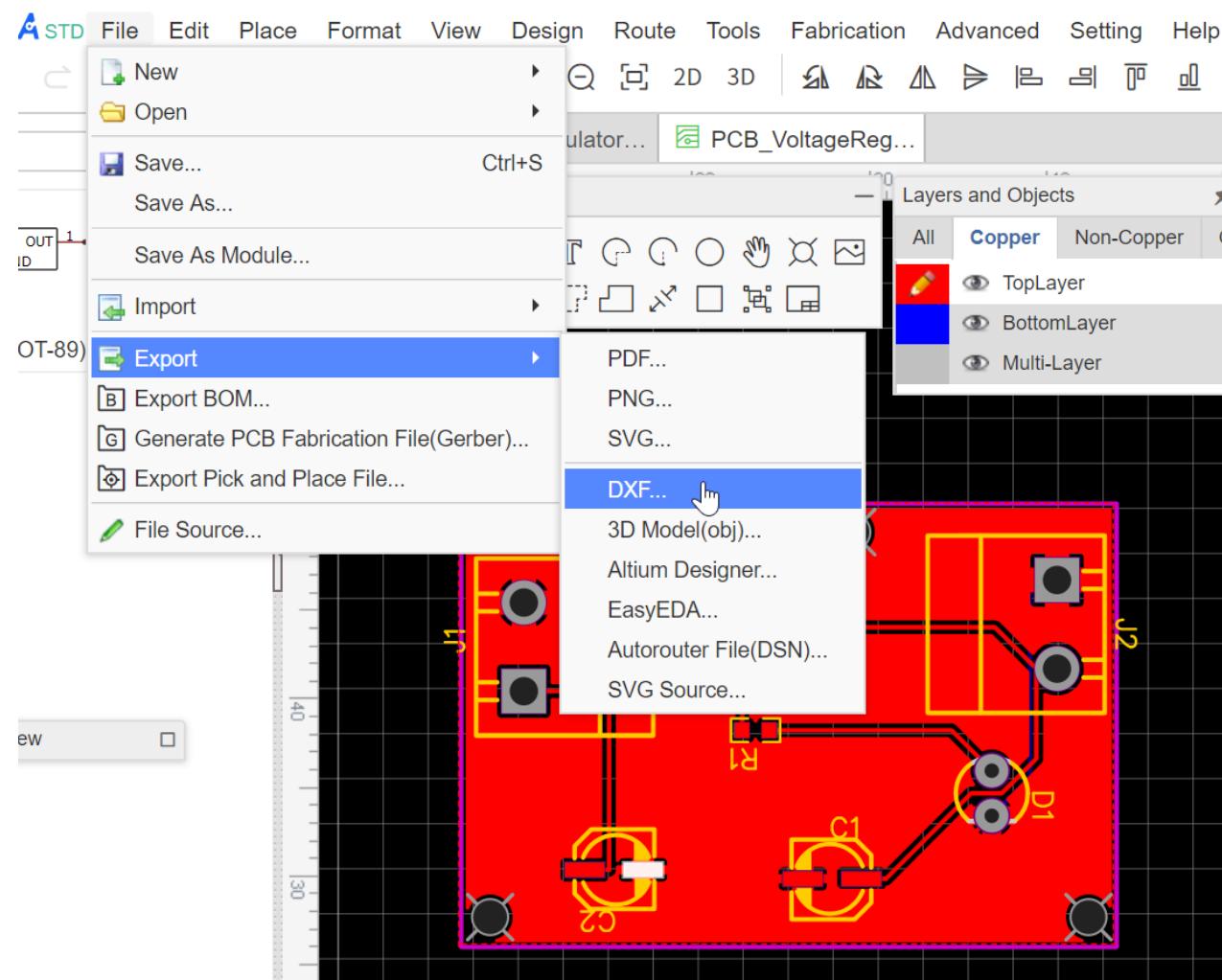
- Click at 4 corners and press escape



# 12. Steps to convert Schematic to PCB



# 13. To export and fabricate



# Important NOTE

- Enter your **registration number** and **Full Name** next to

**all your circuits, PCB and the output plots.**



# LAB record instructions:

For the lab experiment,

- Write the **Aim**.
- Complete the **Software/Hardware components used**.
- **Obtain the expression for the outputs.**
- Place the respective **circuits in LT Spice**.
- Connect the inputs and outputs. Name them and **write the same in the lab copy(inputs and outputs section)**.
- Use probe in LT spice to plot all possible combinations.
- Write a **concluding statement for each circuit**.
- **Submit** the document's soft copy **on time** in [lms.vit.ac.in](http://lms.vit.ac.in) when available.

