

## To simulate the circuit and generate PCB in eSim

1. Download eSim from [here](#).

### Creating a new project and adding Libraries:

1. Create a new project. Refer to [Spoken Tutorials](#) or [eSim manual](#) on how to create a project.
2. Make sure all the libraries are loaded. If the libraries are not loaded, add them using the 'Preferences' option. To add libraries:
  - In KiCad eeschema, click on the **Preferences** option from the top menu.
  - Select **Component Libraries** and click on the **ADD** button. It will be directed to the KiCad libraries.
  - Select All (**Ctrl+A**) and then press **OK**.

### Opening the KiCad schematic file

Open the KiCad schematic file or Append the schematic. To append a schematic:

- In the KiCad eeschema page, click on the **File** option from the top menu. Select **Append Schematic Page** option. Select the converted KiCad schematic file and press **Open**.

### PCB

Once the schematic is loaded, you can go for PCB generation. Refer to the [Spoken Tutorials](#) on how to create PCB.

### Simulation

For Simulation, there are two options:

- First option is to choose and replace the components from eSim libraries. Refer to the [Spoken Tutorials](#).
- Second option is, you can change the component reference according to the Ngspice format.

### Editing the references/designators

1. The Ngspice uses different designators/references based on whether the components are device model, subcircuit, linear or non linear dependent sources.
2. To edit a component:

- Place the cursor on the component. Right click and choose the *Edit Component* option .
  - Choose reference and change it from *U* to the required reference.
  - You can also use the shortcut key '*e*' to edit a component.
3. The reference for device models. You have to change the reference U to (It is case insensitive)
    - i. D: Diode
    - ii. Q: BJT
    - iii. M: FET
    - iv. J: MOS

Refer to Ngspice manual or eSim libraries

4. Most of the ICs used are categorized as subcircuit. The reference for Subcircuit is *X*. You have to change the reference *U to X*. (It is case insensitive). For digital devices such as logic gates, shift registers and flip flops change the reference from *U to D*. (It is case insensitive).
5. For any linear or non linear dependent sources, change reference U to :
  - i. E: for VCVS (Voltage controlled voltage sources)
  - ii. F : CCCS
  - iii. G: VCCS
  - iv. H:CCVS

### **Plotting and adding the Global labels**

1. To plot the output and input nodes, use components from eSim\_sources.
2. You can use Global labels and text to name the nodes. Follow the [Spoken Tutorials](#) on how to simulate a circuit in eSim.