



## 1 Online / Offline content

1. The online content of Spoken Tutorials can be accessed from :  
<https://spoken-tutorial.org/tutorial-search/>
2. You can also download the Spoken Tutorials for offline learning from :  
<https://spoken-tutorial.org/cdcontent/>
3. From this link download the FOSS categories in the language you wish to learn.
4. The Spoken Tutorial content will be downloaded as a zip file on your machine.
5. Extract the contents of the zip file & access them.

## 2 The procedure to practise

1. You have been given a set of spoken tutorials and files.
2. You will typically do one tutorial at a time.
3. You may listen to a spoken tutorial and reproduce all the steps shown in the video **Side-by-Side Method**.
4. If you find it difficult to do the above, you may consider listening to the *whole* tutorial once and then practise during the second hearing.

## 3 eSim

1. Click on "Select FOSS" or "All FOSS Categories" drop-down and choose "eSim".
2. Click on "Select Language" drop-down or "All Languages" drop-down and choose the language (English).
3. Click on "Submit" or "Search" button.
4. You will see a list of tutorials based on your selection.
5. Start with the first tutorial in the displayed list

## 4 General Instructions for eSim:

1. Always select components from the libraries starting with eSim for precise simulation results.
2. Whenever the Missing libraries pop-up window appears, click on Close.
3. Whenever the Project Rescue Helper dialog box appears, click on OK.
4. At the end of each tutorial, close the schematic by pressing Ctrl and Q keys together on the eSim schematic editor window.
5. At the end of each tutorial, remove the project selected from the *Projects* window. To do so right click on the Project and select the *Remove Project* option.
6. To see all the files listed under a particular project, right click on that project's name from the *Projects* window and select *refresh* option.

## 5 Common instructions

1. To view the tutorial, click on the Play icon which is located in the player.
2. The **Pre-requisite** and **Slides** will be visible below the player (only for Online contents).
3. **Outline**, **Assignments**, **Code Files** and **Additional Reading Material** are available below the player.
4. Attempt the **Assignments** as instructed in the tutorial, wherever applicable.
5. Adjust the size of the browser in such a way that you are able to practise in parallel.

### 5.1 Common instructions to use Code files

- (a) Click on the link "Code files" located below the player and save it in your folder.
  - (b) Extract the downloaded zip file.
  - (c) You will see all the code/source files used in the particular tutorial.
  - (d) Use these files as per the instructions given in the particular tutorial.
6. Play-pause-practise the whole tutorial.
  7. Once the tutorial is complete, choose the next tutorial from the playlist which is located on the right side or below the player.
  8. Reproduce all the actions as shown in the video.
  9. Follow all the above instructions, till you complete all the tutorials in the series.

## 6 First tutorial: Schematic Creation and Simulation

1. Locate the topic **Schematic Creation and Simulation** and click on it.
2. To view the tutorial, click on the Play icon which is located in the player.
3. Select components only from the libraries starting with eSim.
4. When assigning a new value to any component or changing the existing value, follow the exact method demonstrated at time 7:11. Please take care that you do NOT change the field reference.
5. If you have changed values of components in the schematic, follow the procedure mentioned from time 9:17 to 11:32 to reflect those changes in the simulation compatible netlist.
6. Make sure the analysis parameters and source details are same as mentioned from time 9:52 to 11:20.
7. If you are using Windows OS, at time 14:32 in the tutorial, please go to C:/FOSSEE/eSim/src/deviceModelLibrary/Diode/ and select D.lib from there.

## 7 Second Tutorial: Simulating an Astable Multivibrator

1. When assigning a new value to any component or changing the existing value, follow the exact method demonstrated at time 6:52. Please take care that you do NOT change the field reference.
2. If you have changed values of components in the schematic, follow the procedure mentioned from time 8:16 to 10:00 to reflect those changes in the simulation compatible netlist.
3. Make sure the analysis parameters, source details and device modelling tab values are exactly as mentioned from time 8:37 to 9:47.

If you are using Windows OS:

1. At time 9:14 in the tutorial, please go to C:/FOSSEE/eSim/src/deviceModelLibrary/Diode/ and select LED.lib from there.
2. At time 9:36 in the tutorial, please go to C:/FOSSEE/eSim/src/deviceModelLibrary/Transistor/ and select BC547B.lib from there.

## 8 Third Tutorial: Mapping components with Footprints

1. Examples folder of eSim is present under C:/FOSSEE/eSim/

## 9 Sixth Tutorial: Setting Parameters for Designing PCB

1. Download the Code files provided for this tutorial and extract the downloaded folder.
2. Place the extracted folder 7805VoltageRegulator on your Desktop.

## 10 Fifth Tutorial: Laying Tracks on PCB

1. Try this tutorial *only after* you have performed **Setting Parameters for Designing PCB** tutorial.

2. Use the same project that was used for the **Setting Parameters for Designing PCB** tutorial, i.e. 7805VoltageRegulator, which is saved on your Desktop.
3. At 3:10 in the Copper Zone Properties dialog box, in the setting tab, Clearance(mm) value should be set to 0.8mm.

## 11 Sixth Tutorial: PCB Layout for Astable Multivibrator

1. Download the Code files provided for this tutorial and extract the downloaded folder.
2. Please select total of 6 layers for which the Gerber files are to be generated, as mentioned at time 8:21 onward. These layers are: B.Cu, F.Silks, Edge.Cuts, F.Mask, B.Mask and F.Cu.

## 12 Seventh Tutorial: Creating a Device Model

1. If you are using Windows OS, download and install the application Notepad++ to view and edit files with .lib extension.

## 13 Ninth Tutorial: Subcircuit Builder

1. At 13:57, in the Python plotting section, the narrator mentions the node's name as **cy**. Please note that the nodes name is actually **Cout**.

## 14 Eleventh Tutorial: Uploading a Spice Subcircuit file

1. Some of the plots when simulated will appear on top of each other so you might see more plots in your computer than shown in the tutorial.
2. At time 5:20 onward, upon opening the ujt.sub file, the line `.include HA.sub ** This is an example line**` appears in the ujt.sub file. Please note it was added for reference purpose and is not part of the actual subcircuit, as the comment suggests. This line will **not** be present in **your** file.