



1 The procedure to follow

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 commands shown in the video.
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.

2 Oscad

1. Click on "Select FOSS Category" drop-down list and choose "Oscad".
2. Click on "Select Language" drop-down list and choose the language (English, Hindi, Marathi ...) in which you wish to learn.
3. Click on "Locate Tutorial" button.
4. You will see a list of tutorials based on your selection.
5. Start with the first tutorial in the displayed list.

3 First tutorial: Introduction to Oscad

1. Locate the topic "Introduction to Oscad".
2. To view the tutorial, click on the video player icon on the right of the selected topic.
3. The **Outline** of the tutorial and the **Prerequisite** will be visible to the right of the player.
4. The links for **Code Files** and **Assignment** will be available below the player.
5. Click on the player and view the tutorial.

3.1 Instructions to practise on Linux OS

- (a) The tutorials are explained for **Linux OS**.
- (b) It will be easy for Linux users to follow the steps shown in the tutorial.
- (c) At 8:00 min, the tutorial shows that you will get a small window called "Please select the model", on clicking Model Builder. But, you will not get this window. You will get a blank window titled "Ngspice Model Editor", instead.
- (d) Play-pause-practise the whole tutorial.

3.2 Instructions to practise on Windows OS

- (a) "Introduction to Oscad" teaches how to install Oscad and test run Oscad using an example in Linux OS.
- (b) For installing Oscad in Windows, kindly follow the instructions given in the Oscad Installation sheet.
- (c) After installing Oscad, launch Oscad by doing the following:
 - Click on **Start** and type **Oscad** in the search box (In Windows 8/8.1 machines, the search button is available on the left panel).
 - Click on the Oscad launcher (it will appear with Oscad icon) that appears in the search results.
 - This opens up the Oscad project window.

NOTE: If you are unable to launch Oscad even after performing the above steps, please try again after disabling the anti-virus on your computer.
- (d) Skip this step if you could launch Oscad using the steps given in (c). If you are still unable to launch Oscad, then do the following:
 - i. Click on **Start** and type **cmd** in the search box.
 - ii. Click on **cmd.exe** to open the **Command Prompt**.

- iii. Type `cd C:\` at the Command Prompt and press **Enter**.
 - iv. Now type `cd OSCAD\OSCAD\forntEnd` and press **Enter**.
 - v. Next, type `python oscad.py` and press **Enter**.
 - vi. This will open up the Oscad project window.
- (e) After launching Oscad, listen to the Oscad tutorial **Introduction to Oscad** from 6:37 min onwards.
 - (f) Reproduce the steps shown in the tutorial to validate the installation of Oscad.
 - (g) At 8:00 min, the tutorial shows that you will get a small window called "Please select the model", on clicking Model Builder. But, you will not get this window. You will get a blank window titled "Ngspice Model Editor", instead.
 - (h) Note: If, on clicking any of the tools from the Oscad toolbar (e.g., Schematic Editor), you get a message saying "Windows Firewall has blocked some features of this program", then click on **Allow Access** and proceed.

3.3 Common instructions for Assignments

- (a) Create a folder on the Desktop with your Name-RollNo-Component (Eg. `rakhi-04-Oscad`).
 - (b) Give a unique name to the files you save, so as to recognize it next time. (Eg. `Practice-1-Oscad`).
 - (c) Remember to save all your work in your folder.
 - (d) This will ensure that your files don't get over-written by someone else.
 - (e) Remember to save your work from time to time instead of saving it at the end of the task.
 - (f) Attempt all the given **Assignments** and save them in your folder.
6. Once the tutorial is complete, click on the back button on the browser (top-left corner left-arrow button).

4 Second tutorial: Schematic Creation and Simulation

1. Locate the topic "Schematic Creation and Simulation" from the list.
2. Follow this tutorial and reproduce all the activities as shown.
3. At 09:54 min, the video says that you have to watch KiCad tutorial - **Designing Circuit schematic in KiCad**.
4. Please refer to the KiCad series of tutorials for further instructions.
5. First go back to the tutorial list page by clicking on the back button on the browser (top-left corner left-arrow button).
6. As instructed earlier, select KiCad from the FOSS Category drop-down list and select **English** from the Language drop-down list.
7. From the tutorial list, select the first topic.
8. Reproduce the **Astable multivibrator circuit schematic** shown in the video, using **Oscad**.
9. Once the tutorial is complete, click on the back button on the browser (top-left corner left-arrow button).
10. Once again, select **Oscad** from the FOSS Category drop-down list and select **English** from the Language drop-down list.
11. From the tutorial list, select the first topic.
12. Resume watching the video from 09:54 min.
13. Once the tutorial is complete, click on the back button on the browser (top-left corner left-arrow button).

5 Third tutorial: Designing Printed Circuit Board

1. Locate the topic "Designing Printed Circuit Board" from the list.
2. For this tutorial, you will need to use the practice files created in the previous tutorial.
3. Follow this tutorial and reproduce all the activities as shown.
4. At 10:34 min, the video says that you have to watch KiCad tutorials -

- (a) Electric rule checking and netlist generation.
 - (b) Mapping components in KiCad.
 - (c) Designing printed circuit board in KiCad.
5. Locate these tutorials as mentioned earlier and watch them, before proceeding further.
 6. Reproduce the layout of Astable multivibrator shown in them, using Orcad.
 7. Once the tutorial is complete, click on the back button on the browser (top-left corner left-arrow button).