



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.11b 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.

- Make users aware of the importance of circuit simulations.
- To recognise students and faculty who are good in this area.

Participate & earn attractive honorarium + certificate of internship from IIT Bombay.

Lab Migration

We help Colleges & Institutes shift their EDA labs based on proprietary tools to eSim.

The Lab Migration team helps in the following ways:

- Provide suggestions on the different ways eSim can be implemented in the lab.
- Coordinate lab migration.
- Provide solutions to the lab's problem statements.
- Provide support to the faculty and lab in charge.

Participate and earn attractive honorarium for your efforts.

Spoken Tutorials

The eSim team has created Spoken Tutorials on eSim. For self-learning, we recommend you to use the Spoken Tutorials available on our web site.

Forum

Forum is a place where one can post all their doubts and questions which users / developers get while using eSim. Please reach out to us with your queries on installation and use of eSim through our Forum page.

About FOSSEE

FOSSEE (Free and Open Source Software for Education) project is funded by the National Mission on Education through ICT, MHRD. The FOSSEE team works on 'Adaptation & development of Open Source simulation

packages equivalent to proprietary software', and is based at Indian Institute of Technology Bombay.

Other Projects under FOSSEE

Scilab, Python, DWSIM, Osdag, R, OpenFOAM, Xcos, QGIS, OpenModelica, Focal and Open hardware, etc.

Activities of FOSSEE

- Textbook Companion
- Lab Migration
- Niche Software Activities
- Forum
- Workshops and Conferences

Weblinks

eSim:

<https://esim.fossee.in>

Circuit Simulation Project:

<https://esim.fossee.in/circuit-simulation-project>

Lab Migration:

<https://esim.fossee.in/lab-migration-project>

Forum:

<https://esim.fossee.in/forum>

Spoken Tutorials:

<https://esim.fossee.in/downloads/tutorials>

Github repository:

<https://github.com/FOSSEE/eSim>

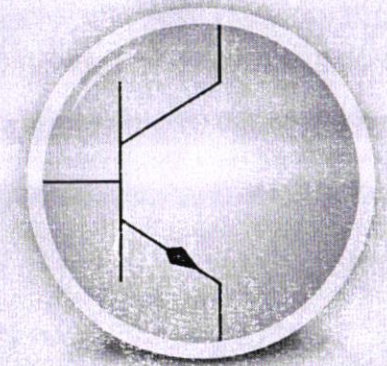
<https://github.com/FOSSEE/nghdl>

Contact us:

General help & Queries:
Email: contact-esim@fossee.in

Sponsored by:

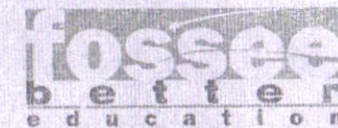
National Mission on Education through ICT, MHRD
<http://sakshat.ac.in>



eSim

A Free and Open
Source EDA Tool

<https://esim.fossee.in>



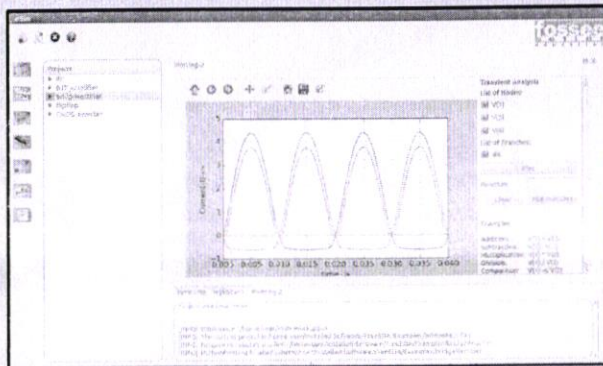
<https://fossee.in>



Introduction to eSim

eSim (previously known as OScad / FreeEDA) is a free/libre and open source EDA tool developed by the FOSSEE team at IIT Bombay. It can be used for circuit design, simulation, and PCB design. It also supports mixed-mode simulation.

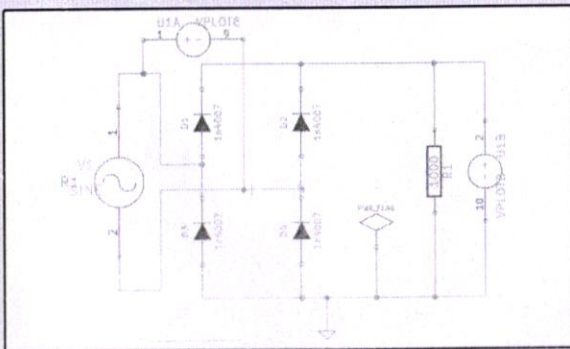
It is an integrated tool built using free/libre and open source software such as KiCad (<http://www.kicad-pcb.org>), Ngspice (<http://ngspice.sourceforge.net/>) and GHDL (<http://ghdl.free.fr/>). eSim is released under GNU GPL License and runs on Ubuntu Linux OS, Windows 7 and above versions of Windows OS.



Features

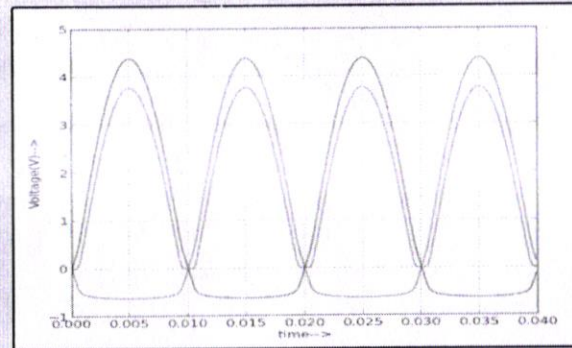
Create Circuit Schematic

- Generate netlists for simulation and PCB design.
- Perform Electric Rules Check (ERC).
- Create new components using Library Editor.



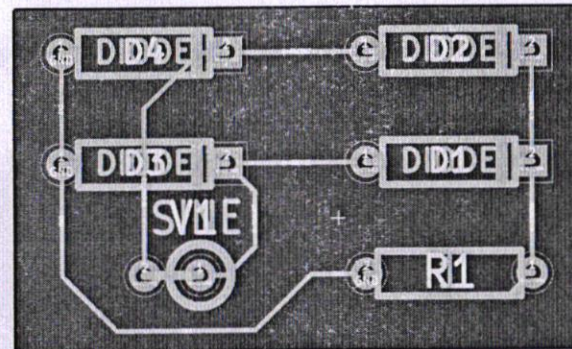
Perform Circuit Simulation

- Analog, digital and mixed signal circuit simulations.
- Perform AC, DC, DC operating point and Transient analyses.
- Interactive Python plotting.



Create PCB Layout

- Design multilayer PCB layouts.
- Create custom footprints or Modify the existing footprints per requirement.
- Export the design in formats such as Gerber, PDF, SVG and several other formats.



Advanced Features

Model Builder

- Create/upload spice model for semiconductor devices.
- Modify or edit existing spice models for semiconductor devices.

Subcircuit Builder

- Create a new subcircuit at schematic level.
- Edit existing subcircuits down to schematic level.

NGHDL

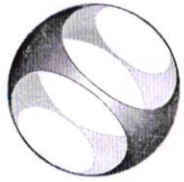
- Using NGHDL, user can create custom digital models using VHDL language. From simple multiplexers, counters to microcontrollers and ASICs, any custom component in the digital domain can be realized using the NGHDL tool.
- The created digital model can be used in either mixed-mode circuit or a standalone circuit operating in digital domain.
- NGHDL gives user the liberty to edit existing models supplied with eSim as per their needs, either for experimenting new ideas or to change the model as per their specific requirement.
- We are currently working towards including the support for simulations involving micro-controllers.

Circuit Simulation Project

FOSSEE, IIT Bombay, encourages students, faculty, and practitioners of electrical and electronics and allied fields to participate in the Circuit Simulation project using eSim. The Circuit Simulation project aims to port existing circuit designs and simulations using eSim.

The objectives of this project are to:

- Make available a large number of Circuit Simulation examples through crowdsourcing.
- Create a database of device models and subcircuits that can be distributed to other users.
- Form a community of users who can contribute and take advantage of the resources available.



Spoken Tutorial
Project at
IIT Bombay

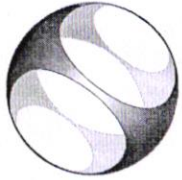
Certificate of Participation

M ADAN
OHAN
ALAVIYA
NATIONAL MISSION ON
TEACHERS AND TEACHING

This is to certify that **KAUSAR JAHAN** has participated in **Faculty Development Programme** from **2023-10-05** to **2023-10-18** on **eSim** organized by **Visakha Institute of Engineering & Technology (VSPT)** with course material provided by Spoken Tutorial Project, IIT Bombay.

Prof. Kannan M Moudgalya
IIT Bombay

Spoken Tutorial is a project at IIT Bombay, started with funding from the National Mission on Education through ICT, Ministry of Education (previously MHRD), Govt. of India



Spoken Tutorial
Project at
IIT Bombay

Certificate of Participation

M ADAN
OHAN
ALAVIYA
NATIONAL MISSION ON
TEACHERS AND TEACHING

This is to certify that **JEEVANARAO B** has participated in **Faculty Development Programme** from **2023-10-05** to **2023-10-18** on **eSim** organized by **Visakha Institute of Engineering & Technology (VSPT)** with course material provided by Spoken Tutorial Project, IIT Bombay.

Prof. Kannan M Moudgalya
IIT Bombay

Spoken Tutorial is a project at IIT Bombay, started with funding from the National Mission on Education through ICT, Ministry of Education (previously MHRD), Govt. of India



Spoken Tutorial

Toggle Navigation

- Software Training
 - **Software Training**
 - About the Training
 - Progress to Date
 - Software Offered
 - Contacts for Training
 - Change in Training Policy
- Procedures**
 - Organising Training
 - Instruction for Downloading Tutorials
 - Create Your Own Disc Image
 - Resource Centers
- Training**
 - Training & Payment Dashboard
 - Semester Training Planner Summary (STPS)
 - Student Dashboard
 - new Individual Learning
 - Individual Learning Workshop
 - Verify ILW test certificate
- Online Test**
 - Instruction for Invigilator
 - Instruction for Participants
 - Certificate Verification Link
 - Email Verification Link
 - new Job Recommendation
- Creation
 - - Creation Process
 - Outline and Script
 - Creation Dashboard
 - Suggest a Topic
 - Suggest an Example
 - Collaborate with Us
- News
 - Testimonials
 -
 - Articles on Spoken Tutorial project
 -
 - Events & Happenings
 -
 - Research / Popular Articles
- Academics
 - MOOCs acceptance

-
- [Official Letters and Links](#)
-
- [Articles on University tie-ups/workshops](#)
- [About](#)
 - [About Us](#)
 -
 - [Team](#)
 -
 - [Mission](#)
 -
 - [Brochures](#)
 -
 - [FOSSEE Semester-long Internship 2022](#)
 -
 - [Health and Nutrition page](#)
 -
 - [Archived FOSS](#)
 -
 - [Project Documents](#)
- [Forums](#)
- [Statistics](#)
 - [Training](#)
 -
 - [FDP Training](#)
 -
 - [Creation Content](#)
 -
 - [Online-Test](#)
 -
 - [Academic Center](#)
 -
 - [India Map](#)
 -
 - [Motion Charts](#)
 -
 - [ILW Training](#)
- [jeevanarao.1006](#)
 - [Change Password](#)
 -
 - [Edit Profile](#)
 -
 - [View Profile](#)
 -
 - [Log Out](#)

Participant List

[Go to Master List](#)

Invalid Users or Verification Pending Users List

Edit Department Year First Name Last Name Email Gender Status

Verified Users List

#	<u>Edit</u>	<u>Department</u>	<u>Year</u>	<u>First Name</u>	<u>Last Name</u>	<u>Email</u>	<u>Gender</u>	<u>Status</u>
1	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	JEEVANARAO	B	ecehod@vietvsp.com	Male	Verified Delete
2	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	KAUSAR	JAHAN	kausar.465@gmail.com	Female	Verified Delete
3	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	T	KALYANI	kalyanithoram@gmail.com	Female	Verified Delete
4	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	PUSHPA	K	kmangapushpa@gmail.com	Female	Verified Delete
5	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	HEMANTH KUMAR	M	hemanth@vietvsp.com	Male	Verified Delete
6	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	BHASKAR NAIDU	B	bhaskarmadhireddi@gmail.com	Male	Verified Delete
7	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	KRISHNA CHAITANYA	CH	krishnachaitanyach4@gmail.com	Male	Verified Delete
8	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	SANDHYA	K	sandhya.ece@vietvsp.com	Female	Verified Delete
9	<u>Edit</u>	Faculty Development Programs (FDPs) (PMMMNMTT)	2023	KEERTHI	LANKA	patrudu88@gmail.com	Female	Verified Delete
10	<u>Edit</u>	Faculty Development	2023	T	MALASRI	malasripattem222@gmail.com	Female	Verified Delete

Programs
(FDPs)
(PMMMNMTT)

- [FOSSEE Project](#)
 - [Scilab](#)
 - [eSim \(Oscad\)](#)
 - [Python](#)
 - [OpenFoam \(CFD\)](#)

 - [Aakash Labs](#)
 - [Co-learn](#)
 - [Education Mission](#)

 - [Software Training](#)
 - [Contacts for Training](#)
 - [Training Dashboard](#)
 - [Creation Dashboard](#)

 - [About Us](#)
 - [News](#)
 -
- 145,098,731 **145,098,732** page loads

[Click here to view stats](#)

Developed at IIT Bombay.



Spoken Tutorial by IIT Bombay is licensed under a [Creative Commons Attribution-ShareAlike 4.0 International License](#), except where stated otherwise
Based on a work at <https://spoken-tutorial.org>. Permissions beyond the scope of this license may be available at <https://spoken-tutorial.org>.



1 The procedure to install eSim in Ubuntu Linux 16.04 or 18.04 or 20.04 LTS (64-bit machines)

1.1 Getting ready for eSim installation

1. To follow the installation procedure, you need to be connected to the Internet
2. Following system requirements should be met:
 - i. Linux: 64-bit x86 Ubuntu 16.04/18.04/20.04 LTS
 - ii. At least 4GB disk space should be available
 - iii. Minimum of 2GB RAM
 - iv. 2 GHz dual-core processor or above
3. Please note: A text editor (for example: Gedit/Vim) is required to view and edit certain files.

1.2 Installing eSim on your machine

1. Go to <https://esim.fossee.in/downloads>
2. Now from the section eSim 2.1 go to Installers. To download and save the file, click on the zip file eSim-2.1.zip under Ubuntu 16.04 and above LTS versions section
3. Locate the downloaded file on your computer. It will be in the Downloads folder ideally
4. Open a terminal by pressing Ctrl+Alt+T keys
5. Change the directory using cd command to the location where the downloaded eSim-2.1.zip file is kept. For example, cd Downloads
6. Unzip the file by typing unzip eSim-2.1.zip and press Enter key
7. Change the directory to eSim-2.1 using the cd command by typing cd eSim-2.1
8. Type the following commands and press Enter after typing each command:
 - a. `chmod +x install-eSim.sh`
 - b. `./install-eSim.sh --install`Note: On Ubuntu 16.04 LTS, in case you are facing difficulties, please use the command `sudo ./install-eSim.sh --install`
9. If prompted, type the admin password and press Enter
10. You may get the following prompt: Is your internet behind proxy? (y/n)
If you are behind a network proxy, you should type y and enter the details to continue.
If not, then type n to continue
11. eSim will be installed on your computer and a shortcut icon will appear on your Desktop

2 The procedure to install eSim in Windows 10 (64-bit)

2.1 Getting ready for eSim installation

1. To follow the installation procedure, you need to be connected to the Internet
2. Following system requirements should be met:
 - i. Windows 10: 64-bit
 - ii. At least 4GB free space should be available in your C:/ Drive
 - iii. Minimum of 2GB RAM
 - iv. 2 GHz dual-core processor or above

3. Disable any anti-virus running on your system (e.g., Norton, Kaspersky, AVG, Avast etc.)
4. Please note: Notepad++ is required to view and edit certain files

2.2 Installing eSim on your machine

1. Go to <https://esim.fossee.in/downloads>
2. Under the section **eSim 2.1** go to the subsection **Installers** . To start the download, click on the zip file **eSim-2.1_install.exe** form the **Windows 7 and about(64 bit PC)** section
3. Locate the file on your computer. It will be in the **Downloads** folder ideally
4. Open the location where this file is downloaded and double click on **eSim-2.1_install.exe**
5. If a pop-up window appears asking, "Do you want to allow the following program from an unknown publisher to make changes to this computer?", click **Yes** to continue
6. In the **License Agreement** window, select the **I Agree** option
7. In the **Choose Install Location** window, click on **Next**
8. In the **Choose Start Menu Folder** window, click on **Install**
9. A progress bar will appear; once it reaches 100%, **Installation Complete** message will be displayed at the top of the eSim setup window. Click on **Close**.
10. eSim shortcut icon will be visible on your Desktop

3 Procedure to check eSim installation

1. Double click on the **eSim icon** on your Desktop and wait till an interface appears. Please do not close the terminal while using eSim because it will close eSim as well
2. Click on the **Open Project** button from the top-left corner of eSim interface.
Windows OS users: go to the location **C:/FOSSEE/eSim/Examples/**and select **Halfwave_Rectifier**
Ubuntu Linux OS users: go to **~/Downloads/eSim-2.1/Examples/** and choose **Halfwave_Rectifier**
3. Double click on the project's name from the project explorer window and click the **Open Schematic** button from the left-most toolbar. A schematic window will appear
4. On the eSim interface, for the same **Halfwave_Rectifier** project click on **Simulation** button from the left-most toolbar. The output plots will be displayed. This indicates that the installation is successful.