

To open OSCAD,

Go to desktop and double click on oscad. Then click on “Run”. Maximize the OSCAD window
OR

Type “oscad” in terminal window command prompt. Maximize the OSCAD window

You will see the OSCAD window shown in Figure.



Figure 1. OSCAD Main Window

Opening an existing project.

1. Change directory to a project directory.

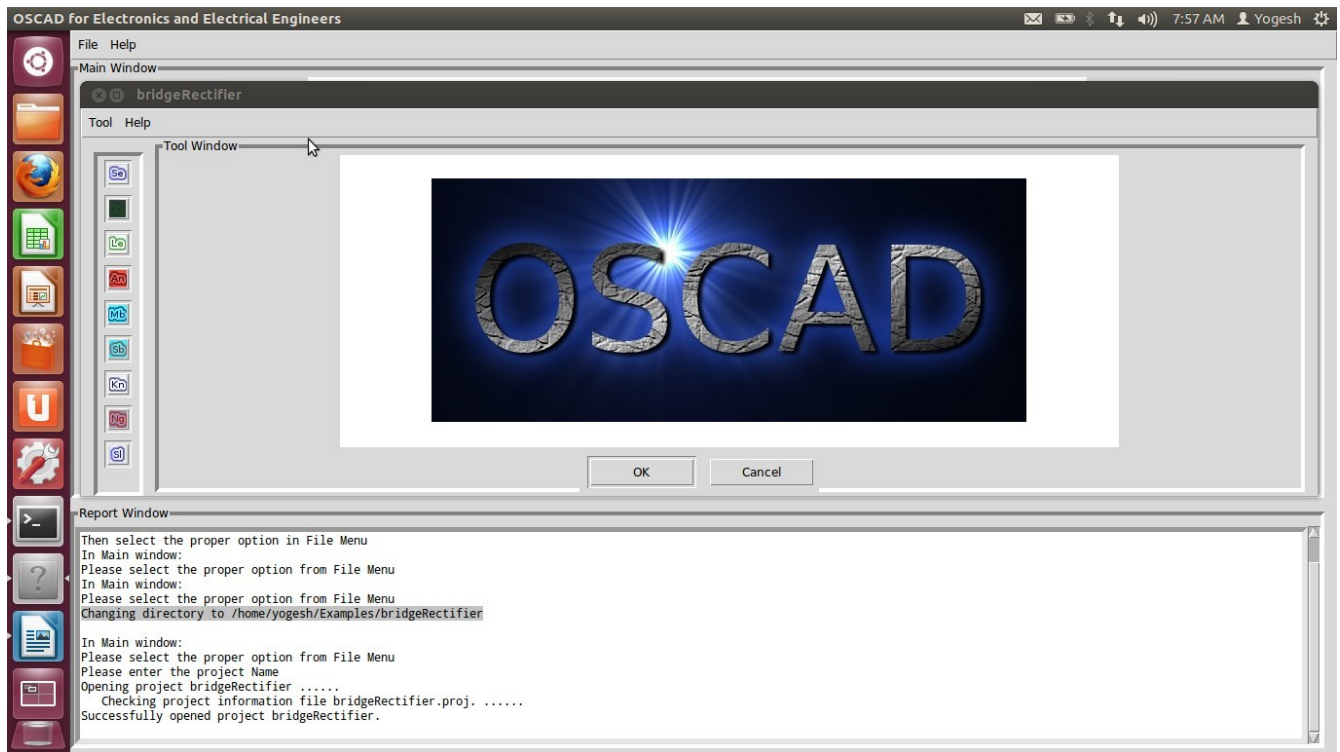
- Press F5, This will ask for choose directory
- Select /home/(login_name)/Downloads/Example/bridgeRectifier
- Press OK

In report window, you will see the message “Changing directory to /home/(login_name)/Downloads/Examples/bridgeRectifier”. Verify whether the chosen directory is correct or not.

2. Open the project

- Press F3. This will ask for the project name.
- Press OK.

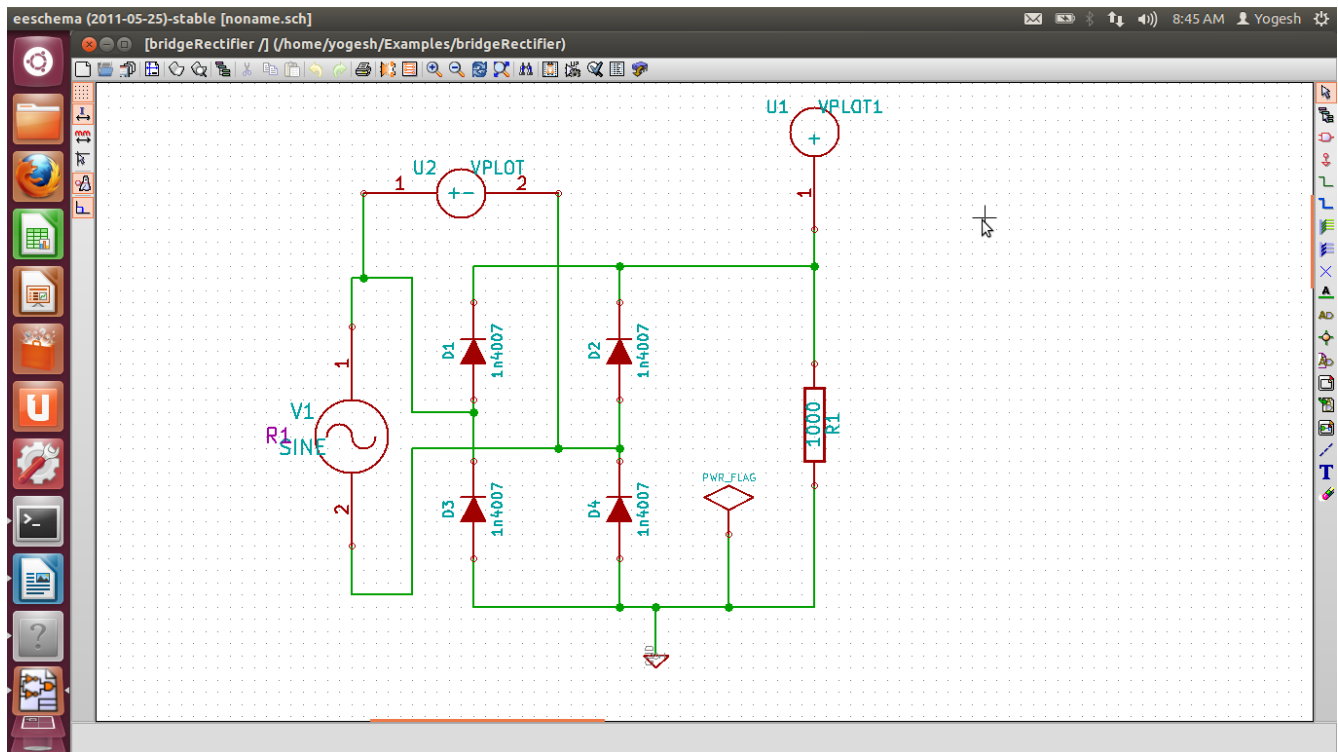
This will open OSCAD project window.



3. Open schematic editor (Visit

http://www.spoken-tutorial.org/show_video/%3Ftr%3D100%2526sval%3D1
for help)

- Press F2. This will open schematic diagram. There may be a warning “Some Library are missing”. In such case, set Library path using step 4
You will schematic window shown in Figure. Use zoom + button to adjust the schematic size.



4. Set Library path

- Choose library from preference (main menu of schematic editor)
- In user defined search path, click on add button.
- Choose (Home_directory)/OSCAD/library and \Press Open
- The message “Use a relative path” will be displayed. Press “No”
- Press OK. This will ask for “Save setting for project” Press “Save”

This will set OSCAD library path.

5. Annotate Schematic

- Click on “Annotate” button (4th button from right on top button window). Press Annotation.
- Press OK and close the window.

6. Perform electrical check rules

- Click on “electrical check rules” button (3rd button from right on top button window). Press
- Test ERC.
- If there is an error, message will be displayed.
- Correct the errors if any.
- Close the window.

7. Generate netlist for Spice

- Click on “Generate Netlist” button (5th button from right on top button window).

- Click on the Spice tab.
- Uncheck “Prefix references 'U' and IC with 'X' ” (This will appear in new version only)
- Press “NetList” and then click on “Save”
- This will generate spice netlist for simulation

8. Generate netlist for PCB

- Click on “Generate Netlist” button (5th from left in top button window).
- Click on the Pcbnew tab.
- Press “NetList” and then click on “Save

This will generate a netlist for PCB Layout

9. Close Schematic editor.

10. Open footprint editor (Visit

http://www.spoken-tutorial.org/show_video/%3Ftr%3D100%2526sval%3D1 for help)

- Press F3. This will open component and footprint.
- Assign footprint to the component (Refer to the above link).
- Close footprint editor.

11. Open Layout editor (Visit

http://www.spoken-tutorial.org/show_video/%3Ftr%3D101%2526sval%3D1 for help)

- Press F4. This will open PCB Layout editor
- Draw PCB layout.
- Close Layout editor.

12. Open Analysis inserter.

- Press F5. This will open Analysis inserter window.
- Click on Transient Tab.
- Change stop time to 40 ms.
- Change step time to 100 us.

- Press “add simulation data”.
- Press Save in File menu.
- Again Press Save. It will ask ‘do you want to replace existing file?’ Press Replace.
- Close Analysis inserter.

13. Open model editor.

- Press F6. It will open model Editor. On the screen you will all the components in the circuit which require models. In our example, diode 1n4007 requires proper model.
- Select 1n4007 Press OK.

- The warning message “model already exist Do you want to edit it” Press OK.
- Change Reverse saturation current “Is” to 880.5e-16.
- Press OK.
- Close model editor.
-

14. Open Kicad to Ngspice netlist converter

- Press F8. It will open terminal window.
- It will ask for parameter for sine wave.
- Add parameters for sine source v1
-
- Enter offset value (Volts/Amps): 0
- Enter amplitude (Volts/Amps): 5
- Enter frequency (Hz): 50
- Enter delay time (seconds): 0
- Enter damping factor (1/seconds): 0

15. Run ngspice

- Press F9.
- Observe the output.
- Close the terminal window.

16. Run SMCSim (Scilab based circuit simulator)

- Press F10.
- Select Symbolic option. Press Ok.
- Observe the output.
- Close scilab.