Additional Reading Material
eSim - Open Source EDA Tool
Spoken Tutorial Team
IIT Bombay

1 FAQs

1.1 Schematic Related

1. After placing a component, I am unable to find the Component or I have zoomed out or zoomed in a lot, how to get the default view?
   Ans : Click on `View from top toolbar of Schematic editor and select Fit On Screen option Alternately, you can also press the home key on your keyboard.`
2. How do I assign a value to a capacitor or resistor?
   Ans : right click on the capacitor/resistor, and scroll to the Edit Component option, select the Value option from the list.
3. How do I assign a value to a Diode/LED/BJT/MOSFET?
   Ans : For semiconductor devices such as Diode/LED/BJT/MOSFET, you will have to assign spice files to them.
   For this purpose, make use of eSim's KiCadToNgspiceConvertor tool. Details given in the tutorial.
4. How do I assign a value to the DC and/or Sine and/or Pulse and/or pwl source?
   Ans : This is again covered in the tutorial Schematic Creation and Simulation, where we have demonstrated an example of adding source details in
   eSim's KiCadToNgspiceConvertor tool.
5. What is the difference between netlists ending in .cir and .net?
   Ans : .cir netlists are Spice compatible netlists. These are used for Simulations. .net netlist however when a PCB layout is to be drawn.

2 Common ERC errors, warnings, and what do they mean?

2.1 ErrType(1): Duplicate sheet names within a given sheet

Ans Review sheet names in hierarchy and remove duplicates.

2.2 ErrType(2) Pin not connected and no `No Connect Symbol`

Ans Check the pin and wire overlaps or place No connect symbol if pin should left unconnected.

2.3 ErrType(3): Pin connected to some others pins but no pin to drive it

This means the problem is typically easily solved by adding a PWR_FLAG symbol to the schematic.

2.4 ErrType(4): Conflict problem between pins. Severity: warning

Two pins connected but their function needs complementary signals (for ex. input -> output).

2.5 ErrType(5): Conflict Problem between pins, Severity: Error

This is because you can only have one output or power output on the same net.

2.6 ErrType(6): Mismatch between hierarchical labels and pins sheets.

There is mismatch between hierarchical label and existing pin sheet, try to re-import hierarchical pins to replace wrong pin sheet.

2.7 ErrType(7): A no connect symbol is connected to more than 1 pin.

"No connect" symbol should be put at the end of the pin,and this pin should be left unconnected at all costs.

2.8 ErrType(8): Global label not connected to any other global label.

There is a global label which has no pair in other sheet(s).

2.9 ErrType(9): Two labels are equal for case insensitive comparisons

Review schematic labels to find possible duplicates, watch for similarity of large and small letters.

2.10 ErrType(10): Two global labels are equal for case insensitive comparisons.

Review global labels in hierarchy to find possible duplicates, watch for similarity of large and small letters.

3 Ngspice Errors

3.1 Unknown parameter error

If any component value is not set, unknown parameter error will be displayed in Ngspice terminal.

3.2 Unknown device error

If the component is not selected from eSim_*libraries, as these are Ngspice compatible.

3.3 Missing pad error or singular matrix error

Missing pad error or singular matrix error will be generated if there is no junction.

3.4 Singular matrix (gmin) error

Singular matrix (gmin) error will be generated if the analysis parameters are not proper.

4 Python Plotting

4.1 Error while opening python plotting Editor.

This is generated mainly due to 2 reasons
1. If the Simulation calculations takes more than 2 seconds to get final output, OR
2. If the components are not assigned value or if any junction is missing or if any connection performs modulo by 0.

4.2 Use of the *Functions:* feature of Python Plotting

Use the *Function:* feature to analyze the plots for the standard functions box (You can use python plot to get addition, division etc. of any two signals)

5 Source Declaration

5.1 DC/AC source

1. You can declare the values of DC, Sine and other sources in the Source Details tab of KiCadToNgspice conversion tool.
2. The value can be declared as 9V in case of a 9V DC voltage source.
   or 9A in case of a 9V DC current source.
   Writing simply the numeric value 9 also works.

6 The shortcuts that can be used in the eSim Schematic Editor

6.1 What are the shortcut keys, how to access them and what do they do?

1. Open the schematic editor and press Shift and ? keys simultaneously.
2. This will display the total shortcut keys, also called as Hotkeys.
3. Please note that, if the shortcut key is related to a component, for example, changing its value or its orientation etc, then your cursor must be located on that component.

6.2 V

This edits the Component's value

6.3 R

This rotates a component.

6.4 A

This calls the place component tool through which you can add components in your schematic.

6.5 M

This moves a component. After pressing M key, the component you chose will be tied to the cursor and you can place it anywhere in the schematic by clicking once on the schematic editor

6.6 C

This copies a component. After pressing C key, the component you chose will be tied to the cursor and you can place it anywhere in the schematic by clicking once on the schematic editor

6.7 Ctrl + H

This is to create a Global label. After pressing Ctrl + H keys simultaneously, the global label will be tied to the cursor and you can place it anywhere in the schematic by clicking once on the schematic editor.

6.8 W

This calls the place wire tool through which you can add components in your schematic.