

## Tutorial on Oscad Subcircuit Builder

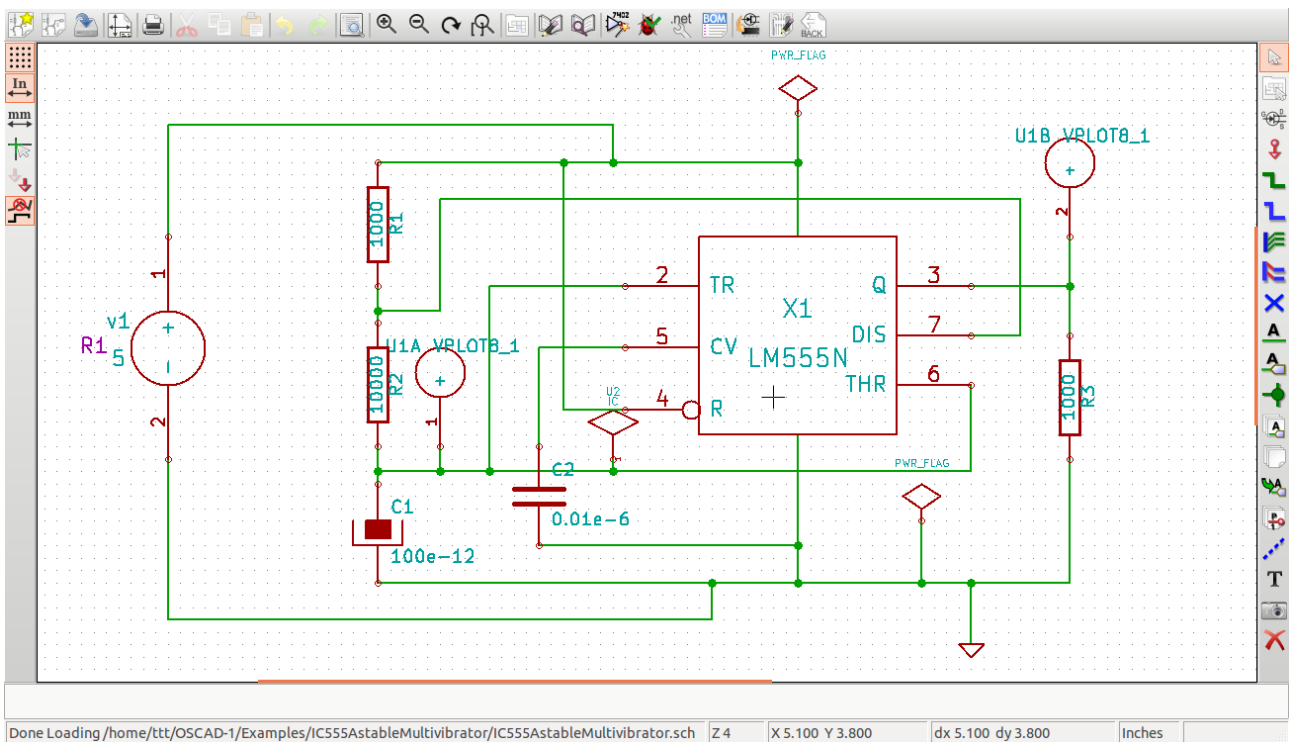
Subcircuit is a way to implement hierarchical modeling. Once a subcircuit for a component is created, it can be used in multiple projects. Oscad provides an easy way to create a subcircuit.

In this tutorial, let us see how to create a new subcircuit, export it and import it for other projects.

### A. Creating new subcircuit

Let us take an example of building a subcircuit for the component **IC 555 timer (LM555N)** which is a part of **Astable Multivibrator circuit** shown in figure below.

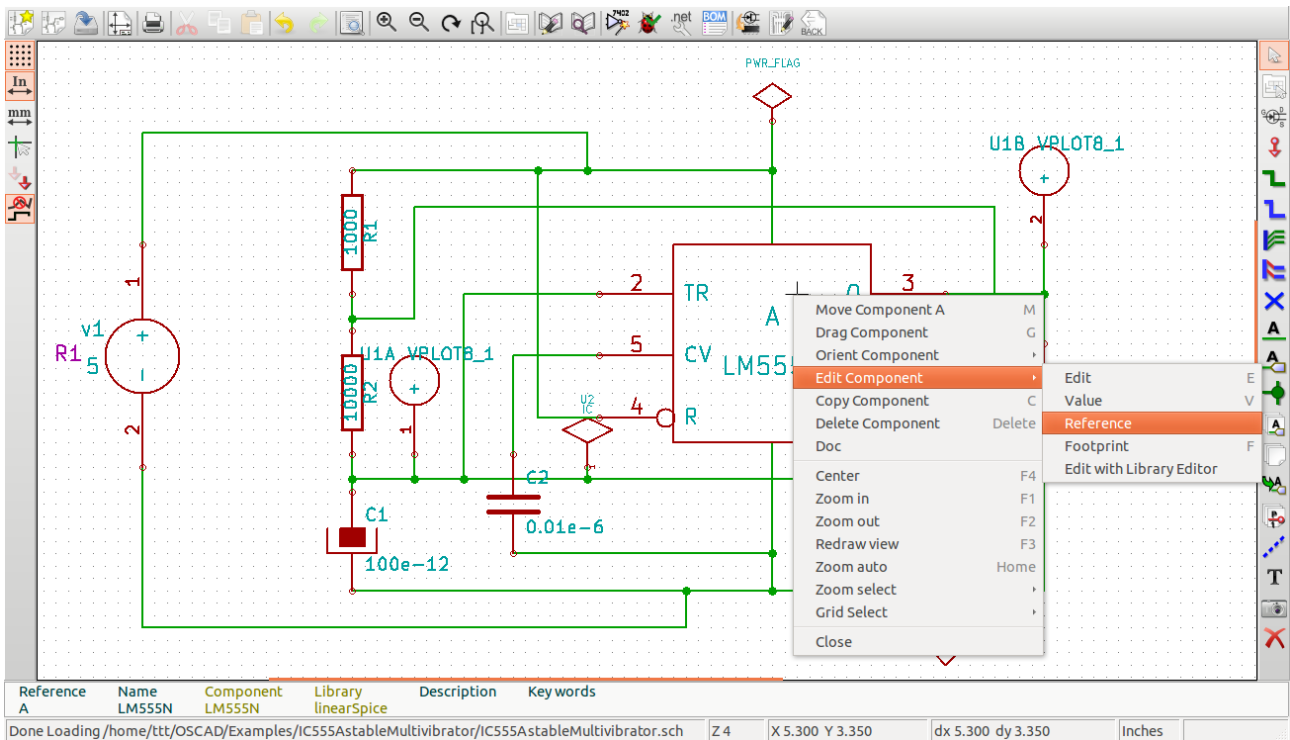
1. For this first create the main schematic of the Astable Multivibrator using **LM555N** as shown in figure below.



**Fig: Astable Multivibrator circuit with LM555N as subcircuit**

2. Before creating the subcircuit for any components, make sure that the **Reference Field** of these components should always be **X**.

Now, let us see how to change the **Reference Field** of a component. Right click on the **LM555N** component, select **Edit component** and then select **Reference**. In the pop up window that shows up change the reference field of LM555N from **A** to **X**. This is shown in Figure below.



**Fig: Astable Multivibrator circuit with LM555N as subcircuit**

Now annotate the schematic, do ERC check and generate the netlist as explained in the spoken tutorial “Schematic Creation and Simulation using Oscad”.

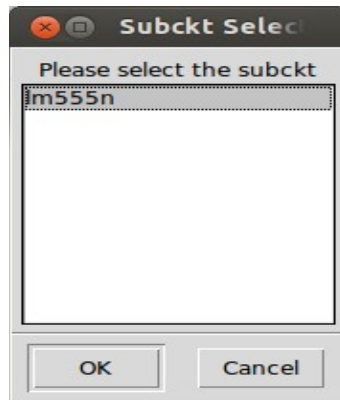
3. After this, to create subcircuit for the LM555N component, click on the **Subcircuit Builder** tool from the **Oscad toolbar** as shown in figure below.



**Fig : Oscad tool bar**

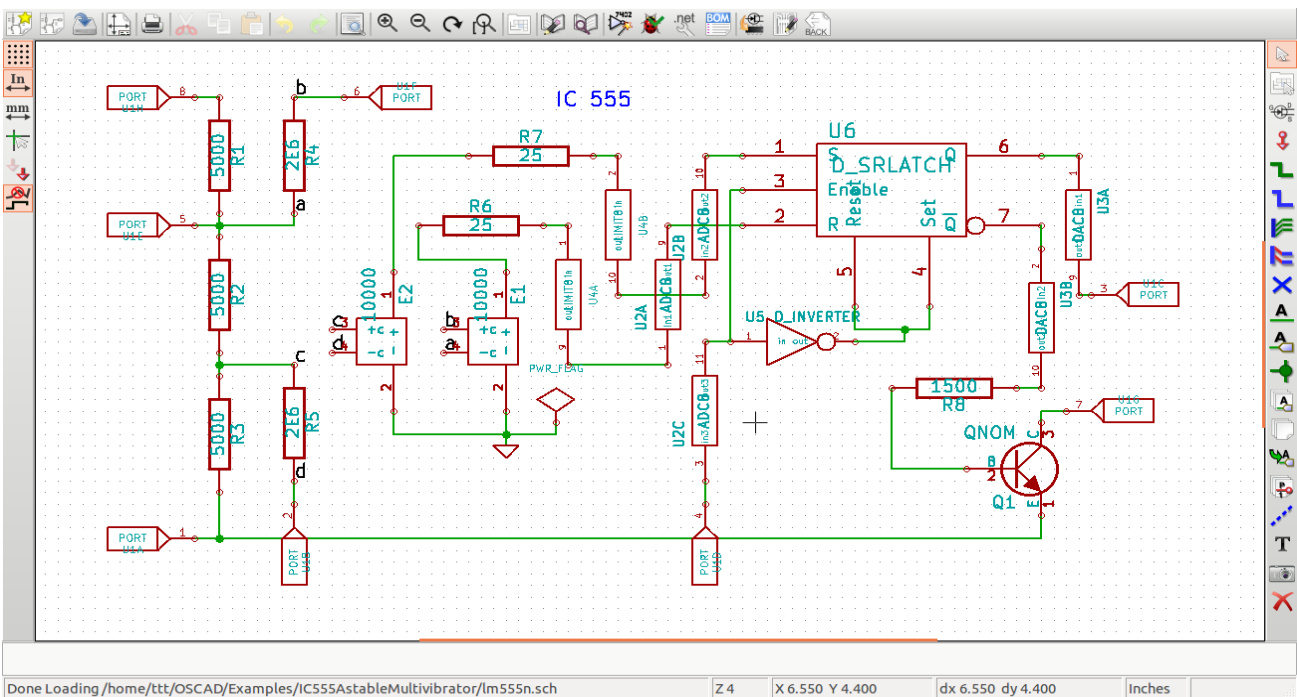
4. A window named **subcircuit select**, displaying the list of components which are referred as  $X$  in

the **Astable Multivibrator circuit** schematic, will open up. Select the subcircuit which you wanted to create. In our case we will select **lm555n** and click on **ok** as shown in below figure.



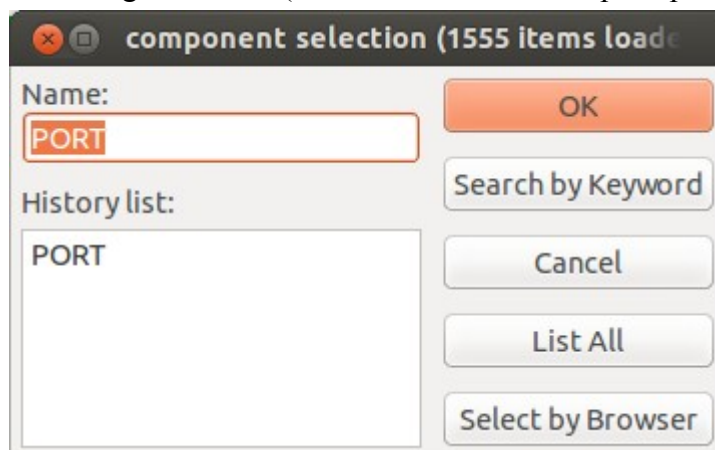
5. It opens up a schematic editor window where you can create the schematic of the LM555N subcircuit as shown in the figure below.

Before creating the schematic, load the Oscad libraries to the schematic as explained in the tutorial “Schematic Creation and Simulation”

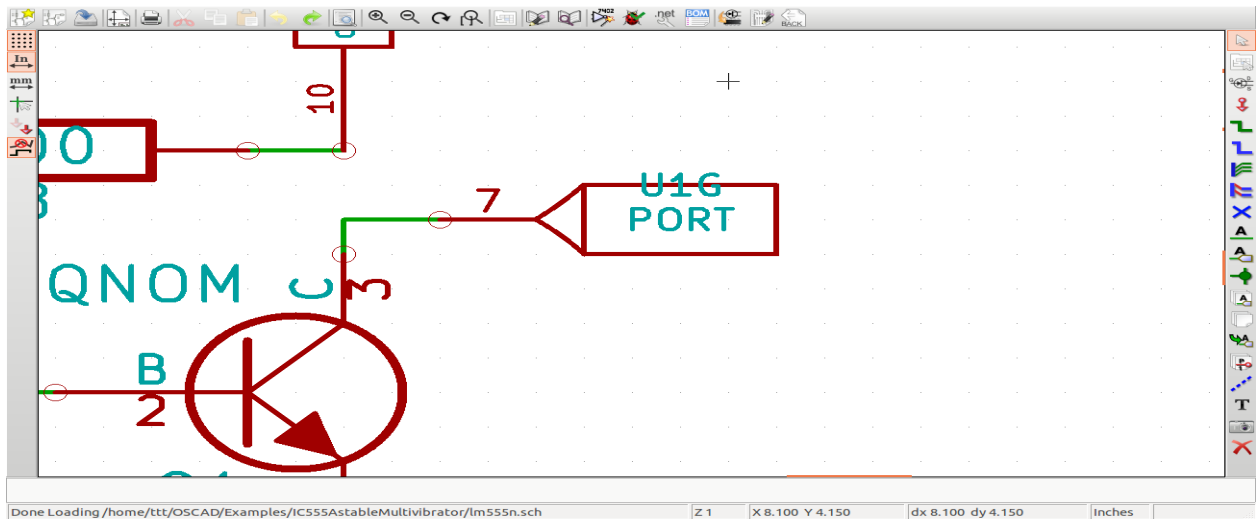


**Fig: LM555N subcircuit**

6. Once the subcircuit schematic is created, you will have to connect a **port** to the open pins of the subcircuit (input and output terminals). **Port** can be searched from the component library of the schematic editor as shown in figure below. (Port is available in the portSpice library).

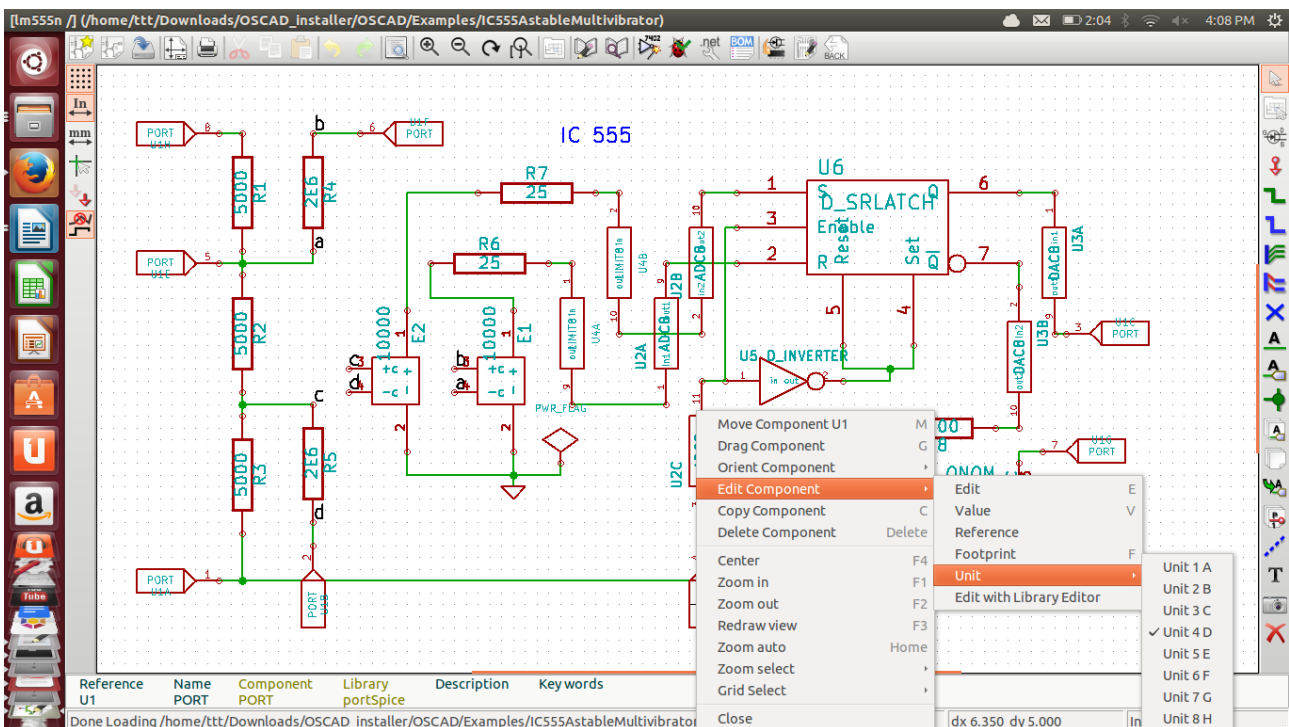


Connecting the port is shown in the figure below.



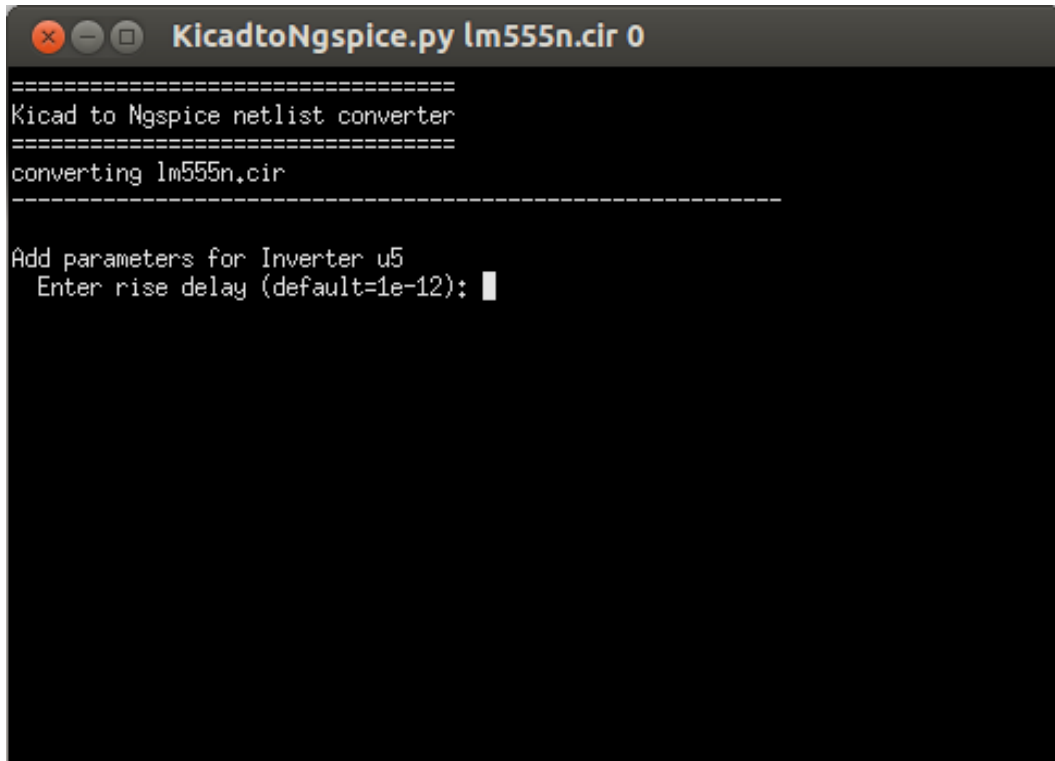
Then, also give the proper **number to the ports** according to the pin numbers of the subcircuit component that you have used in **astable multivibrator circuit** schematic. For example, for LM555N, you should connect a port numbered 7 to that input/output which represents pin number 7 of your IC.

To change the number of the port, right click on port and choose Edit Component and choose Unit and then select the number of choice. This is shown in the figure below.



Once the creation of subcircuit is complete, save the schematic as LM555N.sch in the the project folder of the astable multivibrator circuit and close the schematic editor window.

7. After you close the schematic editor window, a terminal window will pop-up and asks for the values of different parameters of subcircuit as shown in the Figure below.



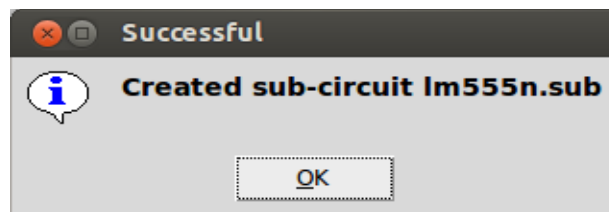
```
=====  
Kicad to Ngspice netlist converter  
=====  
converting lm555n.cir  
-----  
Add parameters for Inverter u5  
Enter rise delay (default=1e-12): █
```

After you enter all the parameters, a message will be displayed on the terminal as shown in figure below.

```
KicadtoNgspice.py lm555n.cir 0
-----
Add parameters for Limiter u4
Enter out lower limit (default=0,0):
Enter out upper limit (default=5,0):
Enter offset for input (default=0,0):
Enter gain (default=1,0):
-----
-----
Add parameters for digital to analog converter u3
Enter output low level voltage (default=0,2):
Enter output high level voltage (default=5,0):
Enter output for undefined voltage level (default=2,2):
-----
-----
Add parameters for analog to digital converter u2
Enter input low level voltage (default=0,8):
Enter input high level voltage (default=2,0):
-----
-----
The ngspice netlist has been written in lm555n.cir.out
The scilab netlist has been written in lm555n.cir.ckt
Press Enter to quit
```

This message tells you that, netlist for the subcircuit is created. Press Enter to quit

8. Finally it opens up a window that says Created subcircuit lm555n.sub. Click on **OK** as shown in the figure below.

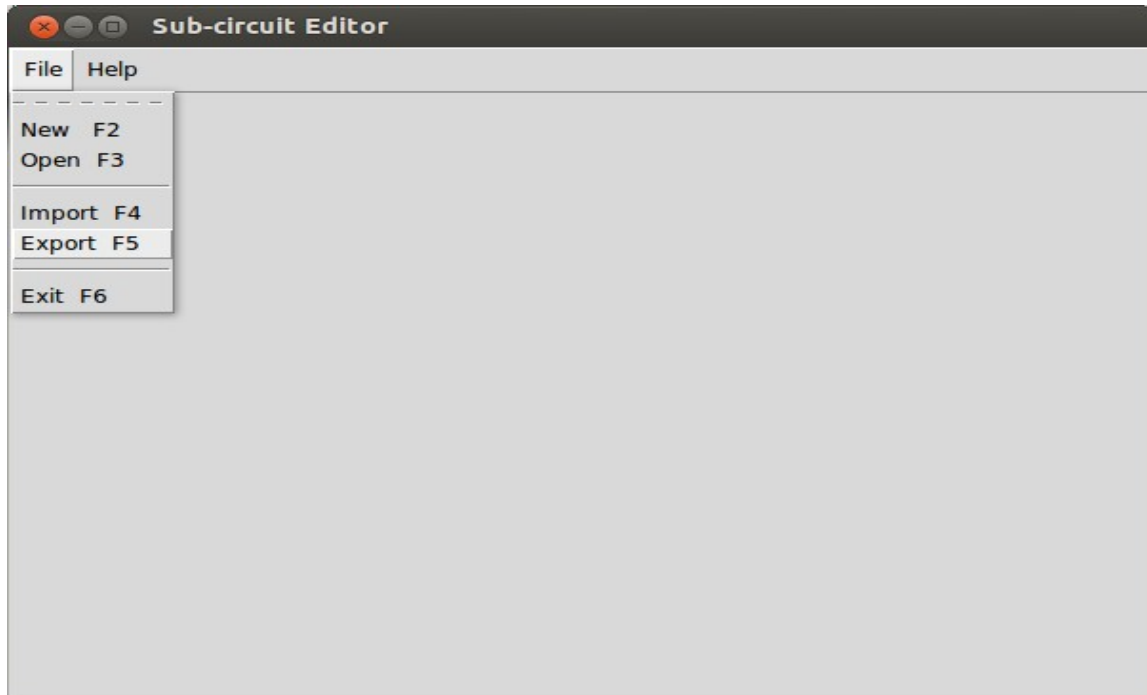


This completes the creation of new subcircuit.

Following this, you should do kicad to Ngspice conversion as explained in the spoken tutorial **Schematic Creation and Simulation using Oscad.**

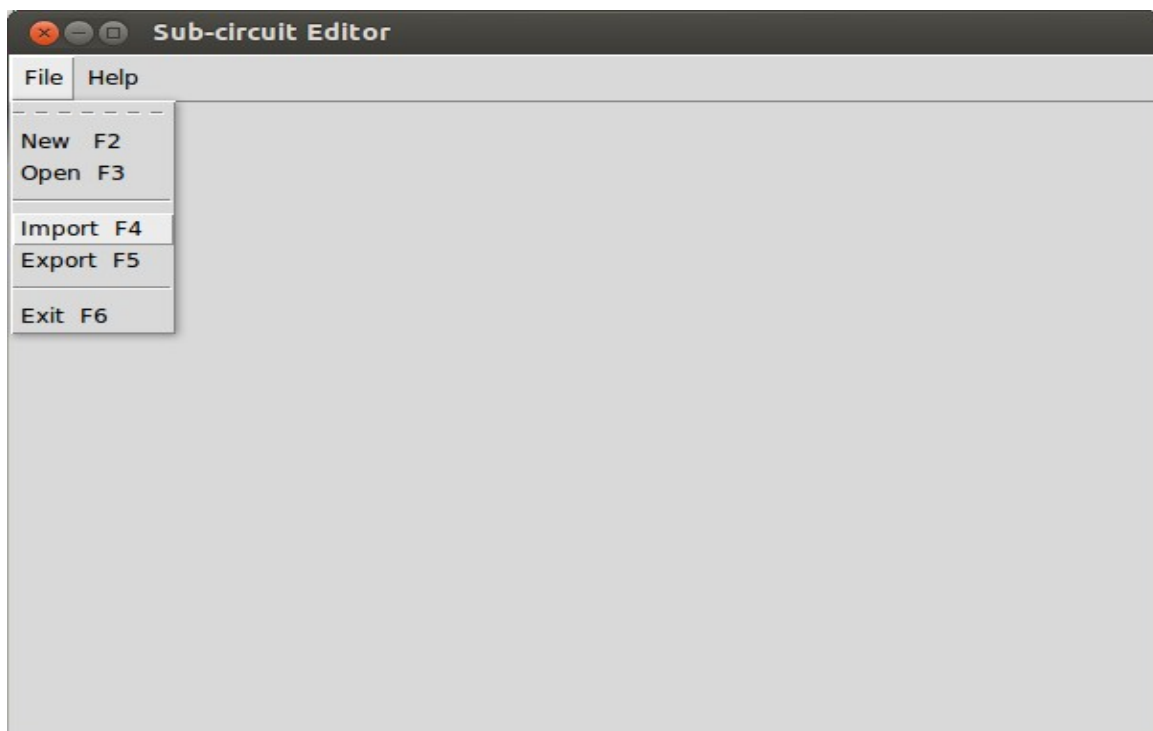
## 2. Exporting the subcircuit to subcircuit library

This newly created subcircuit can be exported to the Oscad subcircuit library which could be used for other projects in future. For this use the **export** option and select the subcircuit which you want to export. This is shown in the figure below.



### 3. Importing the subcircuit

You can import the already created the subcircuit like LM555N in above example to other projects. The steps to be followed are:  
Create your main schematic and include LM555N in your subcircuit. Change its reference to X.  
Now click on subcircuit builder and click on the *import* option and select the subcircuit wanted to use. This is shown in the figure below.



Now do kicad to ngspice netlist conversion and do the simulation.