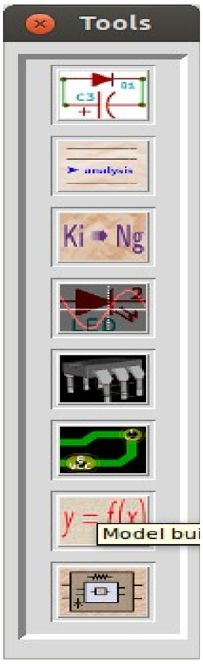
Tutorial on Oscad Model Builder

Model builder allows the accurate modeling of semiconductor devices. Using model builder, you can build the model for the devices such as diodes, Transistors(MOSFET, BJT, JFET, IGBT) and Magnetic core etc.

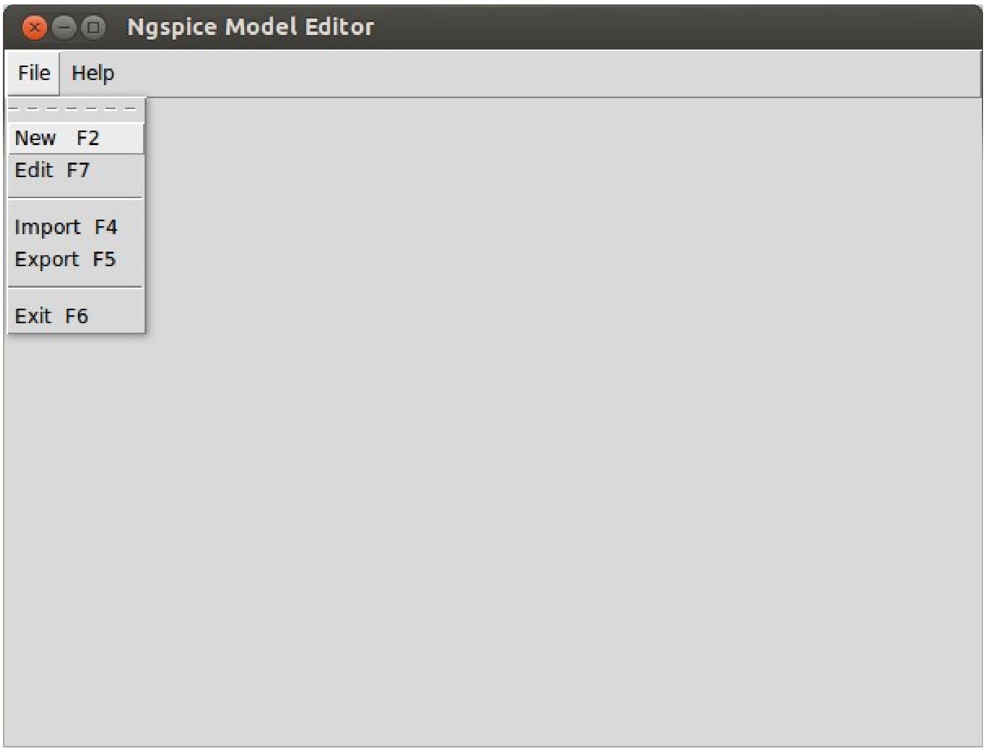
In this tutorial, Let us take an example of **MOSFET** and see how to build a Model, export the model and import the model.

A. How to build a new model

1. Once you complete the circuit schematic creation and netlist generation, click on ***model builder*** button on the Oscad tool bar. *(shown in fig below)*.



2. In the ***Ngspice Model Editor*** window that opens up click on ***file*** and then click on ***new*** as shown in figure below.

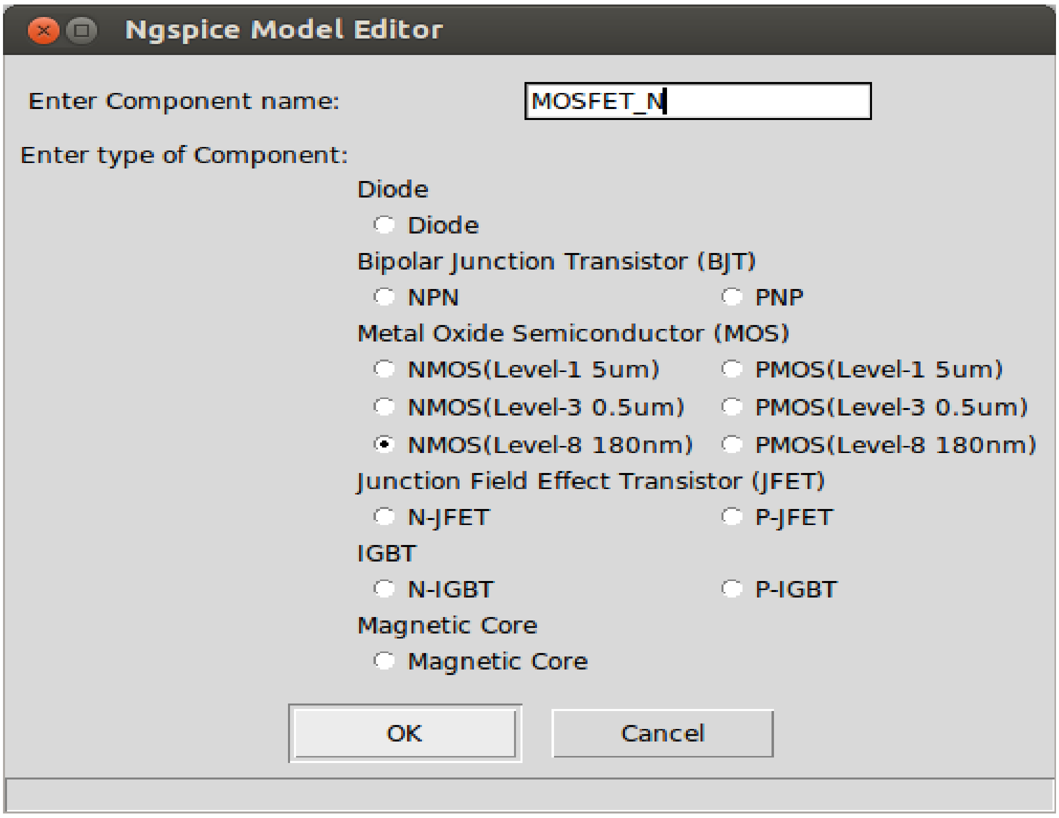


3. A new window, shown below, opens up.

For ***Enter Component name,*** type the model name (same as the field value of the component in the schematic) and select the type of the component.

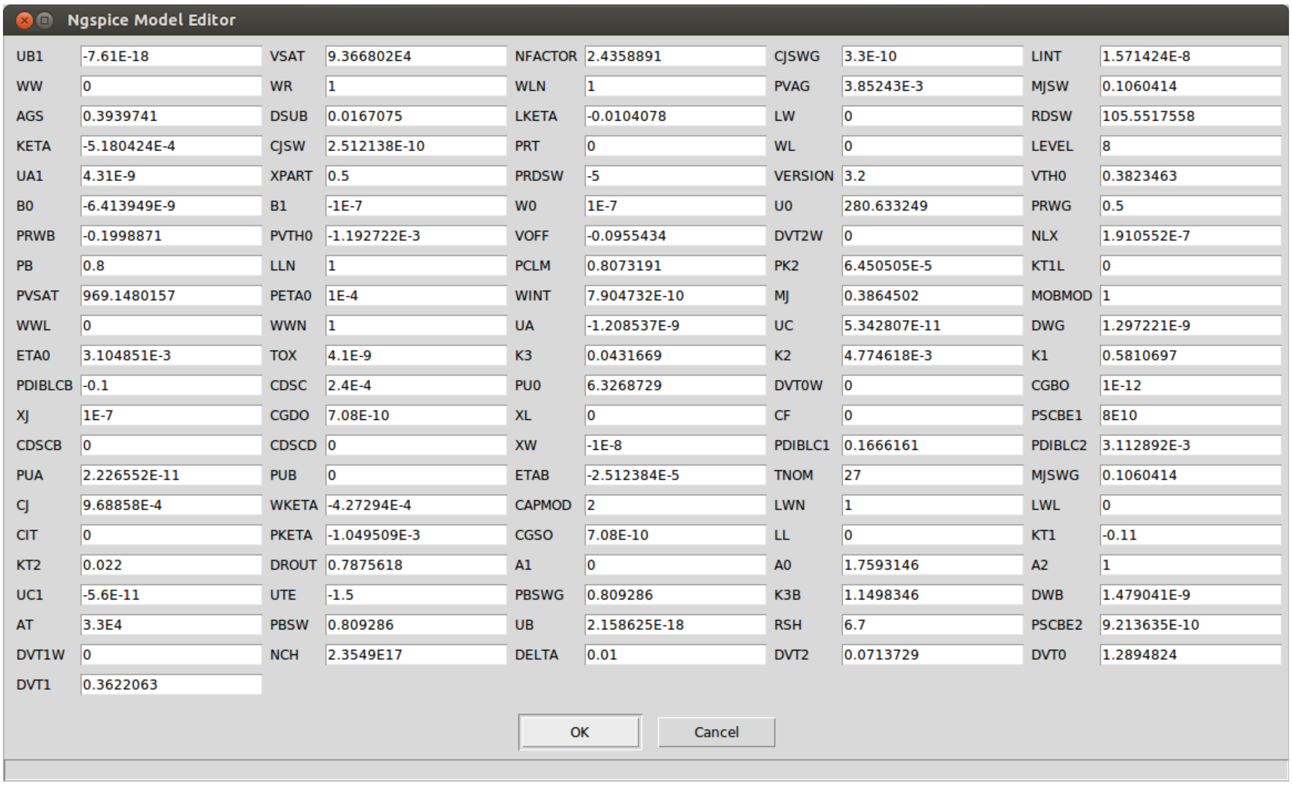
For the MOSFET in our case,

For ***Enter Component name*** I will type ***MOSFET_N*** for N channel MOSFET
And then for ***Enter type of Component*** I will select ***NMOS(level-8 180nm),*** as shown below



For P channel MOSFETs select the options accordingly. Finally click on *OK.*

4. When you click on OK, a window opens up where you can key in the model parameters of the device. You can change these parameters as per the requirement of your device model and then click on OK button to save the model. *(shown in fig below).*



This will generate a new model of the N channel MOSFET. Click on ***OK***. *(shown in fig below)*.

Finally close the winow "*Ngspice model editor*".

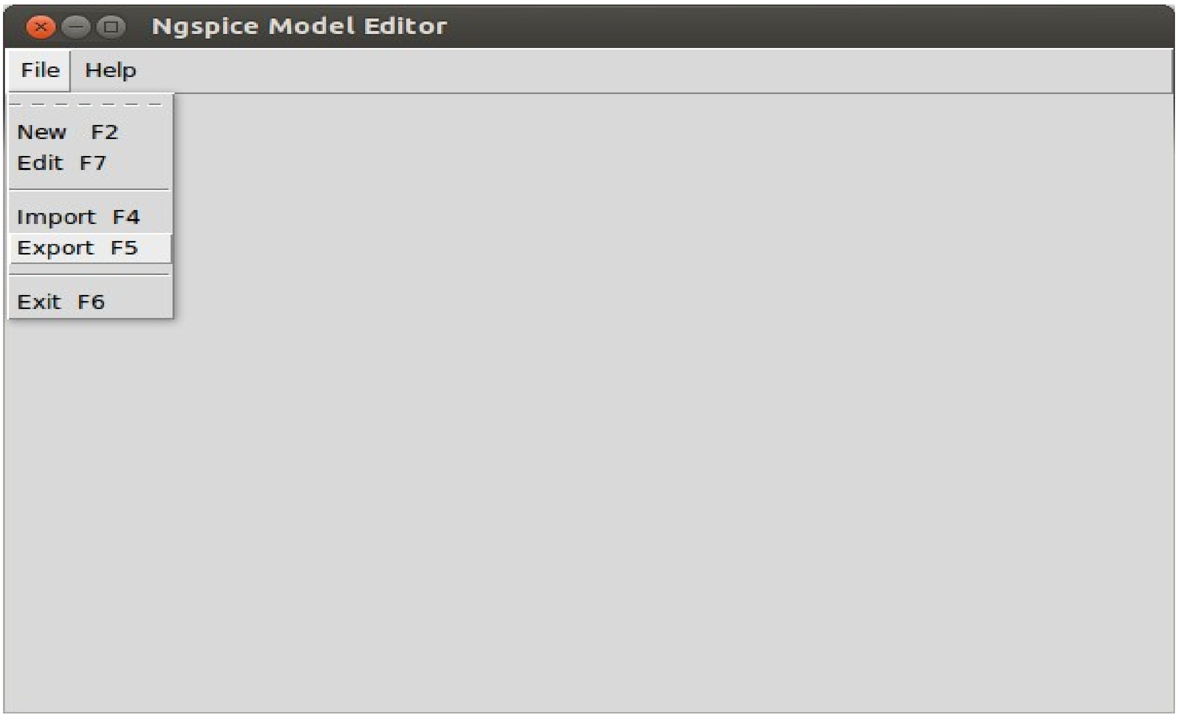
You can follow the same procedure for the other devices like, P channel MOSFET, Diode, BJT, Transformer etc.

B. Export the Model

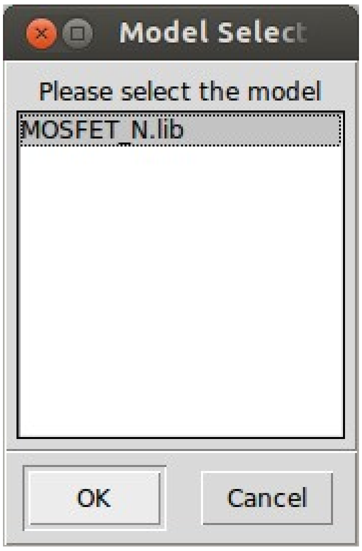
You can also use the device models that you created to other projects in future. For this use the ***export*** option present on the ***Ngspice Model Editor window***.

Now let us do it for the **N channel MOSFET** model that we have created in **part A.**

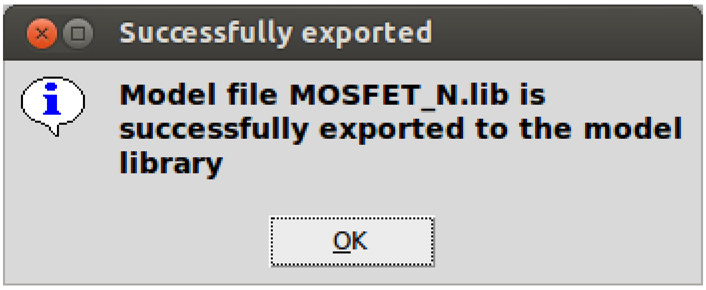
1. Click on export option **p**resent on the ***Ngspice Model Editor window*** as shown in figure below.



2. Then select the ***MOSFET_N.lib*** option and click on ***OK.***



3. Then it shows the message saying MOSFET_N.lib exported successfully to the model builder library.



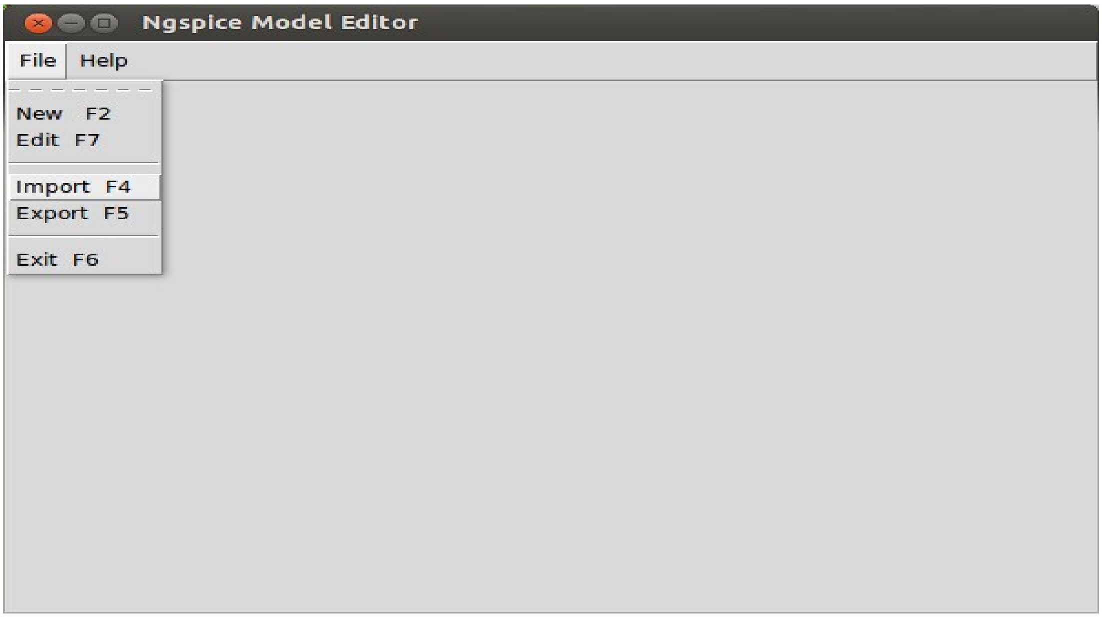
Once this is done it will export the MOSFET_N device model to Oscad device library which you can import for other projects later.

C. Import Model

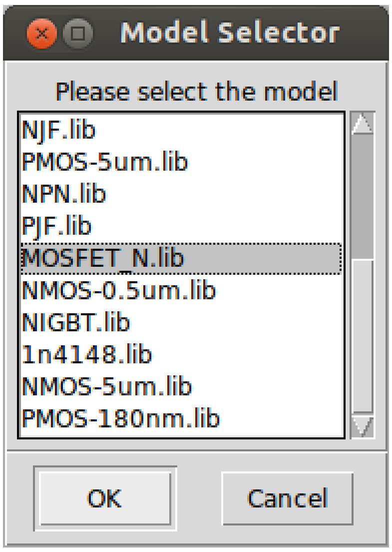
You can use the existing device models from the Oscad device library instead of creating new model. For this use the ***import*** option present in the ***Ngspice Model Editor window.***

Now let us see how to import the MOSFET_N device model that we have created earlier to a new project.

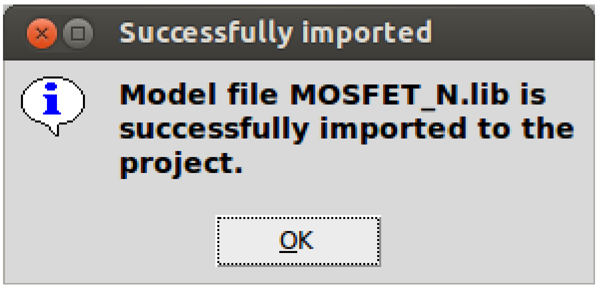
1. Here I assume that, you have created a new Oscad project where you wanted to import an existing N channel MOSFET model from the Oscad device model library. Launch the Model Builder tool from the Oscad toolbar. Now click on ***import*** option present in the ***Ngspice Model Editor window*** as shown in fig below**.**



2. In the window that opens up select the ***MOSFET_N.lib*** and click on ***OK***



3. Finally click on ***OK*** as shown in fig below.



This will import the N Channel MOSFET to your new project.