Chapter 9

MOSFET CHARACTERISTICS

AIM

To design and implement a circuit for simulating the drain and transfer characteristics of a MOSFET.

DESIGN AND CIRCUIT DIAGRAM

Inorder to draw the MOSFET characteristics, we have to use a DC source of voltage which may be varied during simulation. The MOSFET in the circuit should be associated with a coresponding 'MOSFET model' during simulations. The resulting circuit diagram is shown in the Figure 9.1.

Drain charcteristics is a plot between the drain current and drain to source voltage keeping the gate voltage constant. Transfer charcteristics is a plot between the drain current and gate to source voltage keeping the drain voltage constant.

PROCEDURE

Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window

Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window.

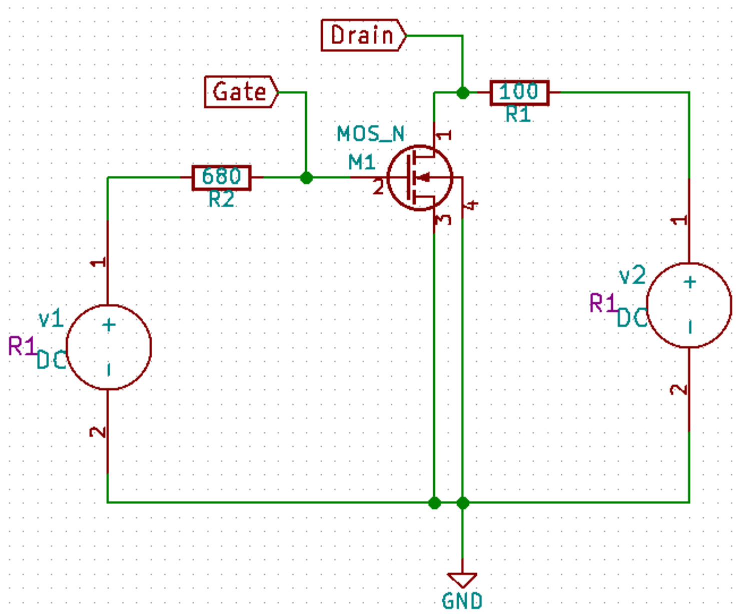

Figure 9.1: Schematic diagram for MOSFET characteristics

Create the Schematic

To create the schematic, click the very first icon of the left toolbar.This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components.

Clicking on the icon on the right toolbar opens the component library. After all the required components of the circuit are placed, wiring is done using the Place Wire option. Scroll up and down for zooming in and out.

Placing the Components: Normally all the components availbale in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose DC sources from eSim_Sources
- Choose resistors from eSim_Devices
- Choose MOS_N from eSim_Devices
- Choose GND from power

Wire the components to get the circuit. A global labels 'GATE' and 'DRAIN' hav been added to identify those nodes whose voltage will be later recorded and plotted.

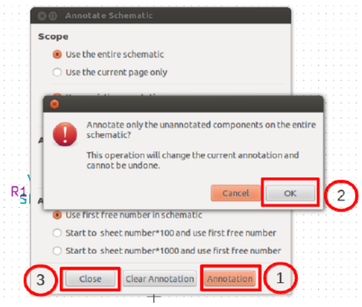

Figure 9.2: Annotation

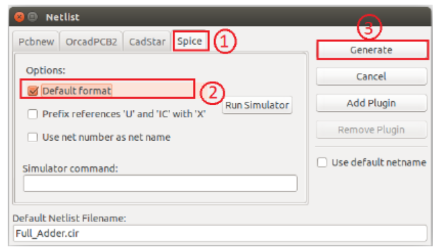

Figure 9.3: Netlist Generation

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the ‘question marks’ associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbarand in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 9.2.

Now we have the circuit diagram as shown in Figure 9.1.

Note: If some libraries are found missing, you can add them from the ‘Preferences‘ menu by following the procedure:

1. Choose ‘Component Libraries’ from Preferences menu.
2. Click on the Add button on the top right side of the window.
3. Choose the required libraries from ‘user/share/kicad/library’ and click OK button

Create Netlist

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving.See Figure 9.3. Now the netlist is ready to be simulated.

KiCad to Ngspice conversion

To convert KiCad netlist of MOSFET circuit to NgSpice compatible netlist click on KiCad to Ngspice icon as shown in Figure 9.4. 8Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

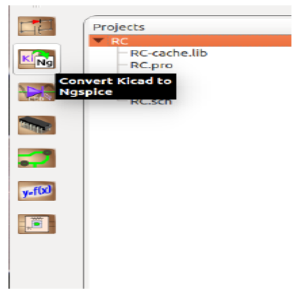

Figure 9.4: Choose Kicad to Ngspice tool

Analysis: Choose DC analysis type. On the same netlist you can simulate the drain characteristics as well as transfer charcteristics. Choose the values of two DC sources, V1 and V2 in the netlist properly as described below. Follow the procedures for drain characteristics first. After obtaining the required plots do the procedures for the transfer characteristics and obtain the required charcteristics curves.

- **Drain Charcteristics:** Give the values of DC variables as shown in Figure 9.5. Enter the name of your DC source **V2** and let its value be varied from 0V to 20V with a step of 0.1 V.
- **Transfer Charcteristics:** Give the values of DC variables as shown in Figure 9.6. Enter the name of your DC source **V1** and let its value be varied from 0V to 4V with a step of 0.1 V.

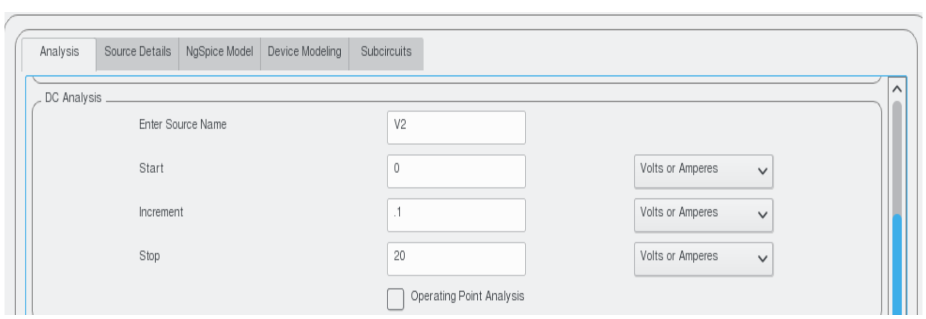

Figure 9.5: Choose DC analysis type and enter the values of V2

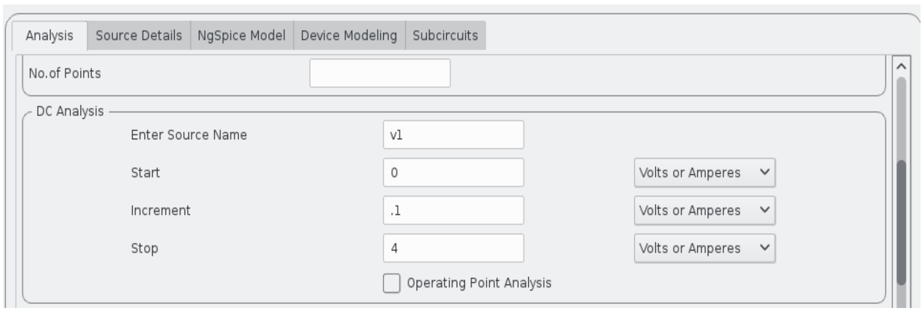

Figure 9.6: Choose DC analysis type and enter the values of V1

Source Details:

- **Drain Charcteristics:** Give the value of DC variables as shown in Figure 9.7. Leave the column of V2 blank. Give the value of V1 as 3V, which is the gate voltage. (You may repeat the experiment by varying the gate voltage as V1=4V, V1=6V etc.)

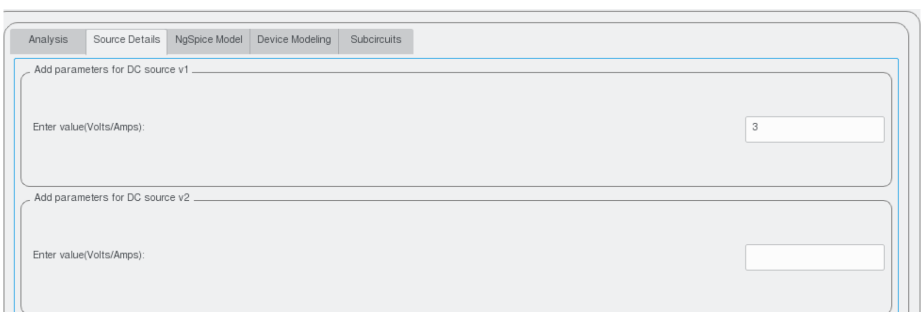

Figure 9.7: Enter the details of fixed source V1

- **Transfer Charcteristics:** Give the value of DC variables as shown in Figure 9.8. Leave the column of V1 blank. Give the value of V2 as 10V, which is the drain voltage.

Ngspice Model: No Ngspice model to be given.

Device Model: The MOSFET is a device whose model details must be given for simulation. Let us choose the generic N-chnnel MOSFET model availabe in the eSim model library. Browse it from `/opt/eSim/src/deviceModelLibrary/MOS/NMOS-5um.lib`. See Figure 9.9.

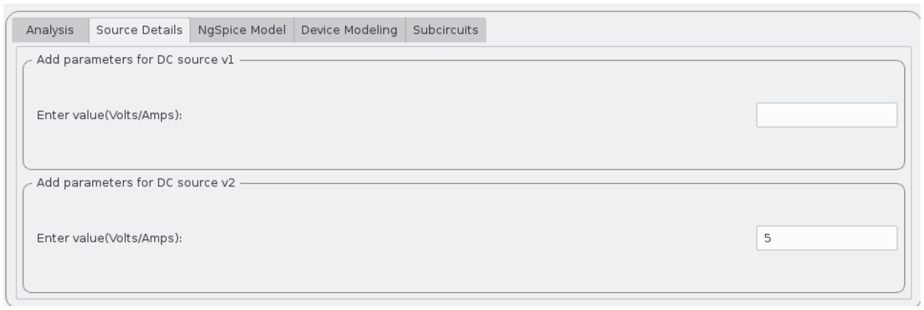

Figure 9.8: Enter the details of fixed source V2

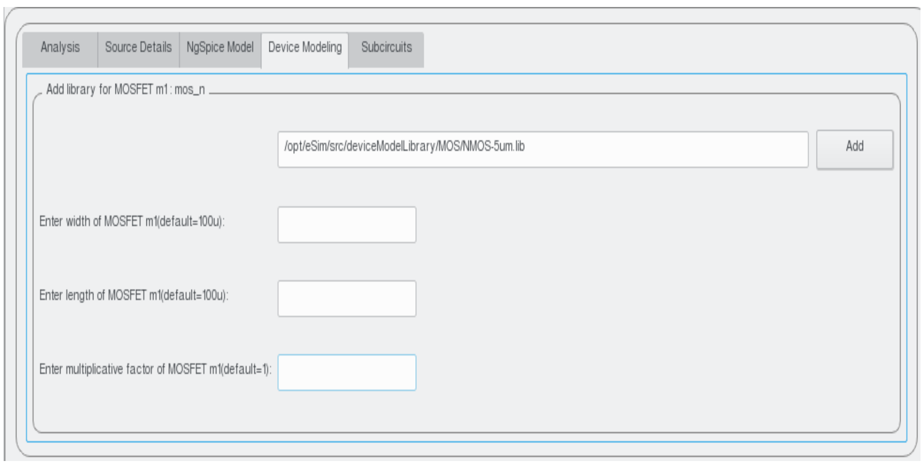

Figure 9.9: Choose the required MOSFET model

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the MOSFET characteristics let us use the commands in ngspice plotting window. We need to plot the drain charcteristics as well as transfer characteristics.

Drain Characteritics: In the ngspice plotting window, type the following command:

`plot -i(v2) vs v(drain)`

This would pop up the drain characteristics of the MOSFET as defined in the MOSFET model NMOS-5um.lib. For a differnt device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 9.10.

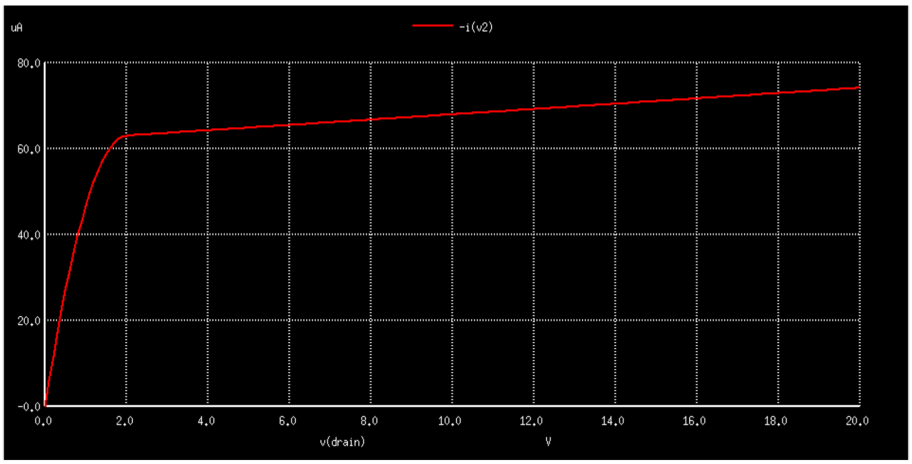

Figure 9.10: The drain characteristics of MOSFET with gate voltage =3V

Transfer Characteritics: In the ngspice plotting window, type the following command:

`plot -i(v2) vs v(gate)`

This would pop up the transfer characteristics of the MOSFET as defined in the MOSFET model NMOS-5um.lib. For a differnt device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 9.11.

RESULT

The circuit for plotting the charateristics of MOSFET was implemented and simulated.

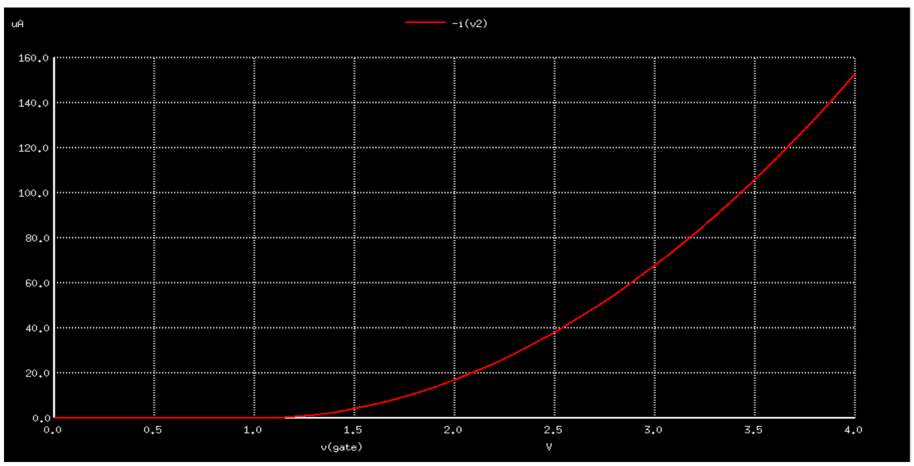

Figure 9.11: The transfer characteristics of MOSFET with drain voltage =10V