ECE 424 – INTRODUCTION TO VLSI DESIGN + LAB

LABORATORY WORK 1

In this laboratory work, we will learn how to use pspice and how we can simulate the characteristics of NMOS Transistor.

First of all, you should download **pspice student edition** to your computer and install it. You can google it to find the setup or you can directly find the setup of pspice by clicking on this link: http://www.electronics-lab.com/downloads/cnt/fclick.php?fid=513

This link is giving you a zipped file. Unzip that file and install the program on your computer.

Figure 1

While you are installing the program, don't forget to check schematics as I have shown it in the figure 1.

When you install it correctly and if it is ready to use, you can find the program in start program as you can see on figure 2. **<u>Please, don't forget to run the program as administrator by right-clicking. Otherwise you will have problems while you are making simulations.</u>**

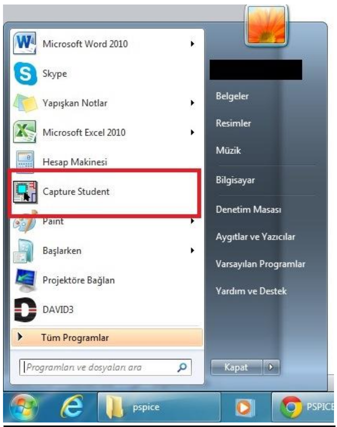

Figure 2

Starting a New Project in PSpice Student Edition:

1. When you run the program, PSpice Capture will launch and you see the following interface. No projects will load automatically.

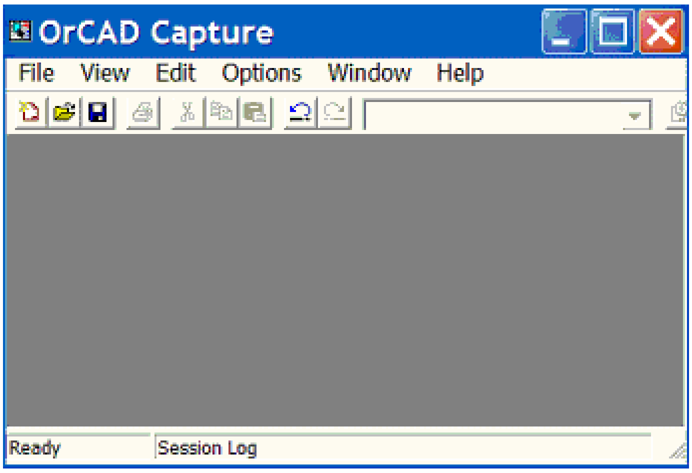

Figure 3

2. From the **File** menu in Capture choose **New** → **Project**. A dialog box will pop up.

3. In the **New Project** dialog box, create a name for your project. Also select the type of project as "Analog or Mixed A/D", choose a location to store your Pspice files. (You can create a folder in your computer hard drive, and click "Browse" to point to the directory you created. All your Spice files will be saved in this directory.)

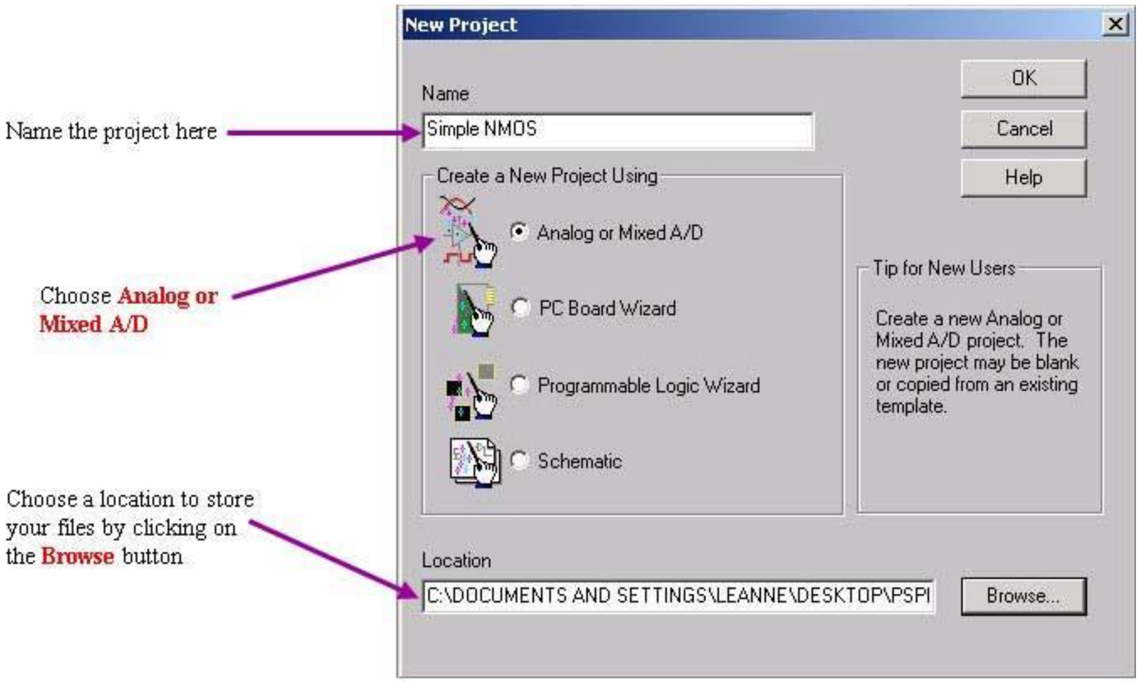

Figure 4

4. After clicking **OK**, the **Create PSPICE Project** dialog box will pop up. It will ask you to choose which type of project you want to create.

Figure 5

5. Once you have clicked **OK** in the **Create PSPICE Project** dialog box, the schematic window will open. If you are using PSPICE for the first time on your computer or you are using a lab computer, the parts libraries will need to be added. Go to the place menu and choose part as shown in figure 6.

Figure 6

6. Library files have the extension **.olb**. If this type of file does not appear in the **Browse File** dialog box, in the Look in: drop down menu you will need to go to: **C -> Program Files -> OrCAD_Demo -> Capture -> Library -> Pspice,** after you have reached this location, the **Browse File** dialog box should look like the one below. Please select all the available libraries.

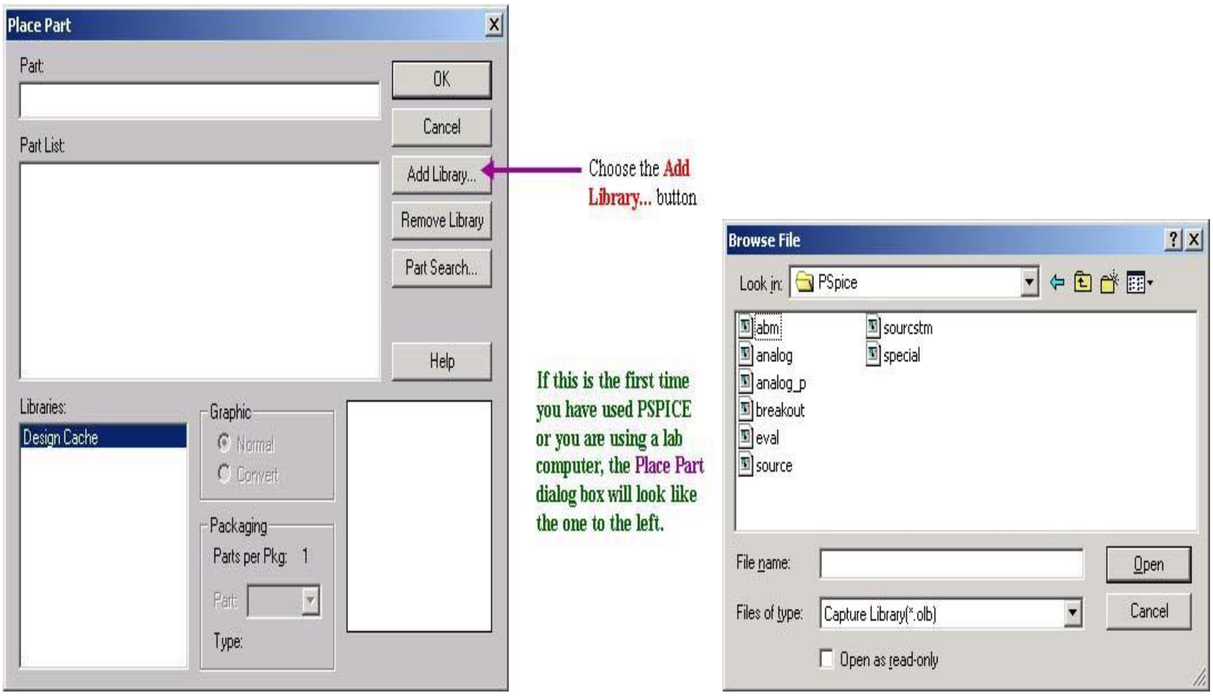

Figure 7 – 8

7. You will use the toolbar on the right side as shown in figure 9 frequently. If the icon bar does not appear in the right side, just left click anywhere in the schematic window and the icon bar will appear.

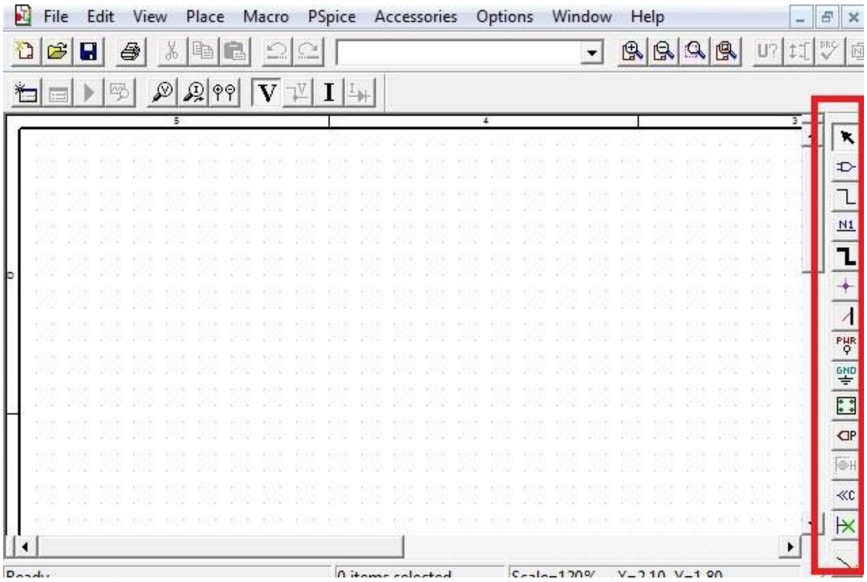

Figure 9

8. You will also need to add a library to use grounds in your circuit. Pspice must have this ground in order for proper simulation. Otherwise it will give error result. Select the **Ground** button from the icon bar on the right. The **Place Ground** dialog box will look like the one below if you are using PSPICE forthe first time on your computer or if you are using a lab computer. Once you choose the **Add Library...** button, go to the same location as we did above to add part libraries and add the **source** library.

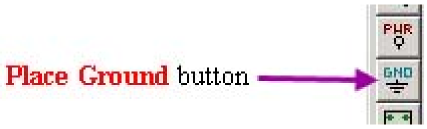

Figure 10

Please note that you only need to add library for the first time. In the future, you will not need to add the library anymore.

Designing a Circuit in PSpice:

1. If we want to design a circuit which is similar with the one in figure 11, we should add these components; DC power supply, NMOS transistor and a resistor. Get to the **Place Part** dialog box by using **Place->Part** from the menu or using the **Place Part** button on the icon bar.

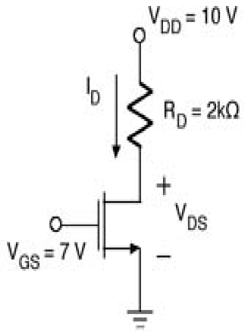

Figure 11

2. Write “VDC” to the part space as shown in figure 12 below. When you click ok button, you can add a voltage source to your project. Add two voltage source and then click place part button again to add another component.

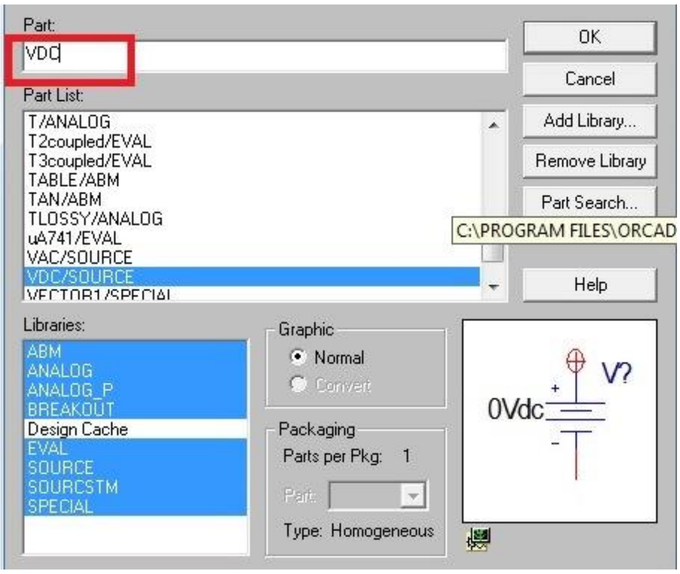

Figure 12

Do this process again to add a resistor and a transistor. To add a resistor, write “R”. To add a NMOS transistor, write “MbreakN3”.

3. Select the Place Ground button from the icon toolbar. Select “GND” from the list that is shown in figure13. You can give a name to it like “0”.

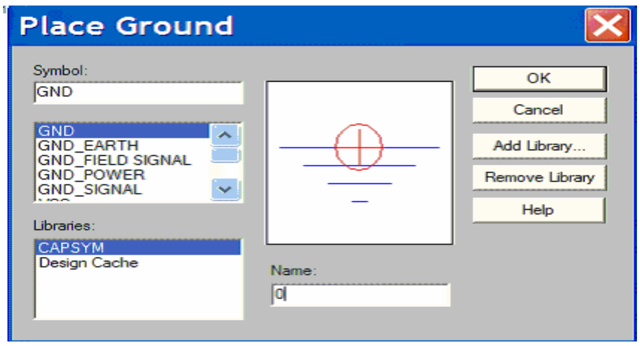

Figure 13

4. After you add all components that you need to the project, you should wire them to each other as shown in the figure 14.

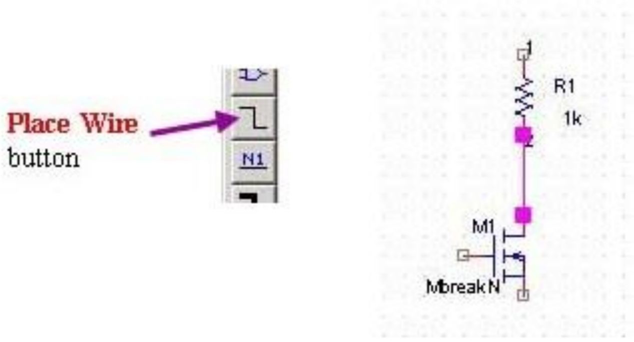

Figure 14

5. When you want to change the parameters of components, you can do it by double-clicking the value of it. You can see an example of it on figure 15. The value of power supply is changed in that figure.

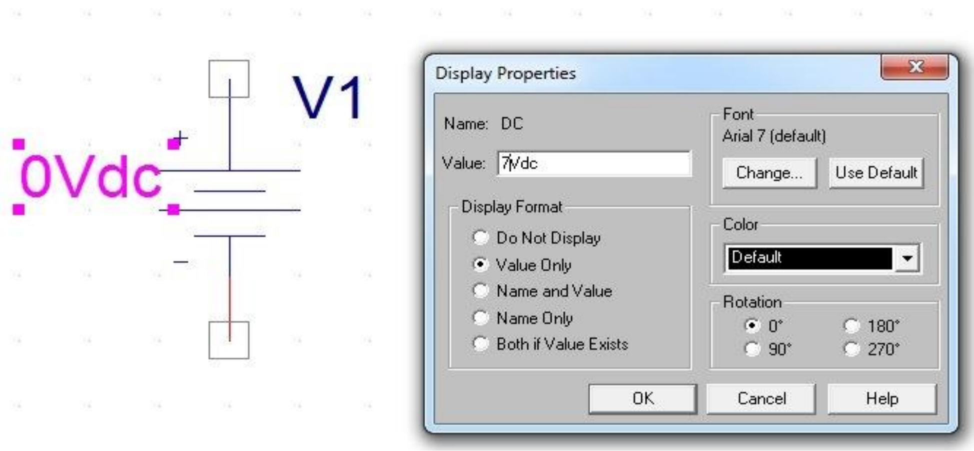

Figure 15

6. When you change the parameters and set the circuit, it will lok like the figure 16. Now you are ready to start a simulation.

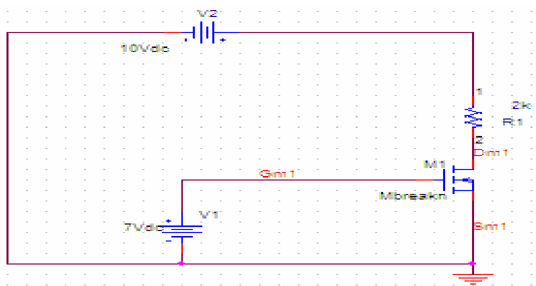

Figure 16

Creating a Simulation for Analysis:

1. Under the PSpice menu choose New Simulation Profile. The New Simulation dialog box will open up. After giving a name to it, click on “create” button.

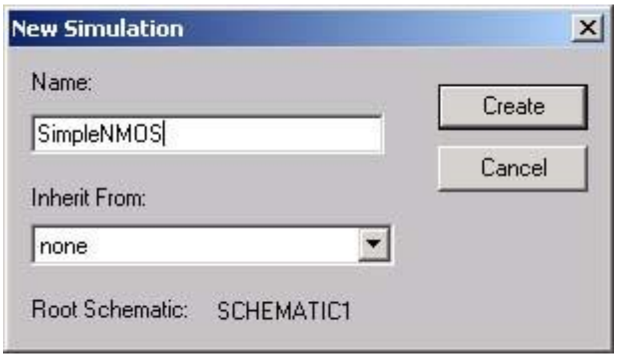

Figure 17

2. For our analysis, we want to do a **Time Domain (Transient)**. All of the settings in the dialog box will be correct as long as the type of analysis is correct. In this simulation, we would like to set the total simulation time (TSTOP) to be 10ms, and the maximum step size as 0.01ms. We can change the settings and the **Analysis** tab of the **Simulation Settings** dialog box should look like the one below.

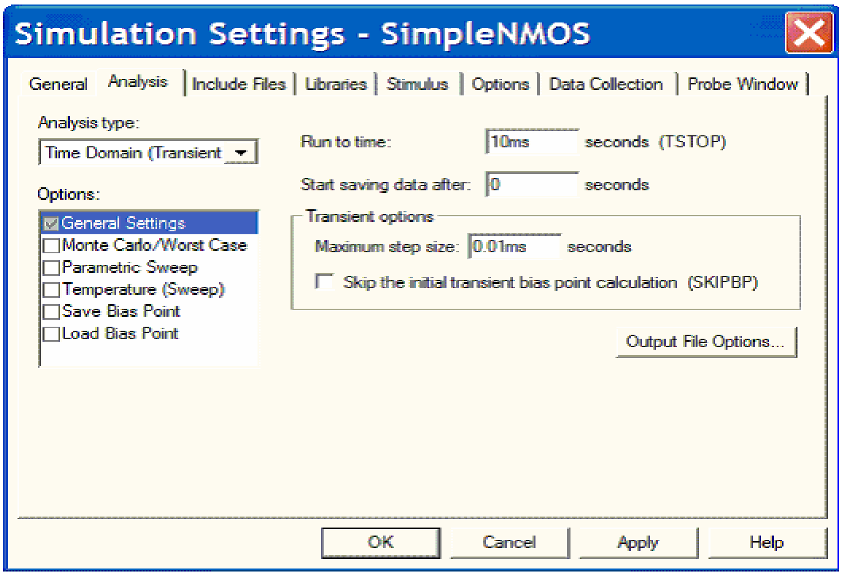

Figure 18

3. Click **OK** and you will return to the schematic. The next step is to run the simulation. This can be done by choosing **Run** from the **PSpice** menu, or by hitting **F11**, or by choosing the **Play** button from the icon bar at the top of the screen.

Figure 19

Now, you can see the simulation of voltage or current by choosing the wire which one you want. But, this simulation will show us the change of voltage or current in time. You can use the buttons that is shown in figure 20 to choose the wire that you want to analyse.

Figure 10

HomeWork:

- **We have learnt to use pspice and creating a simulation in time domain. By using this information, try to simulate the I-V characteristics of NMOS Transistor. The graph that you will find should be like figure 21.**

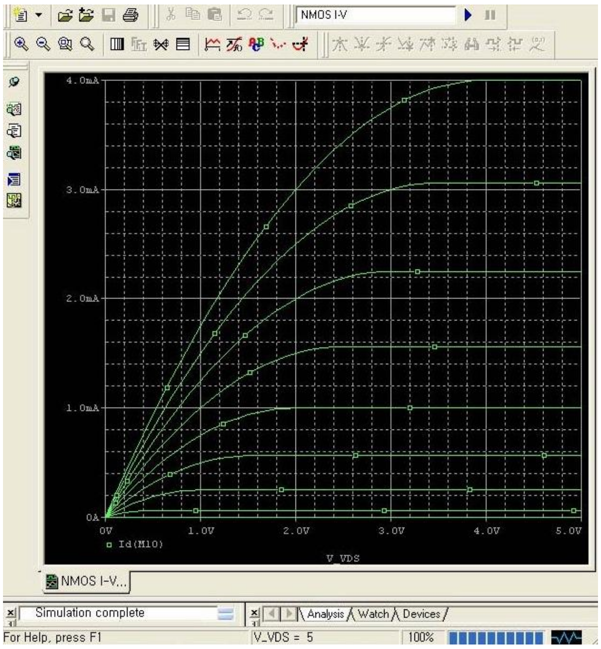

Figure 11

Prepared By: **İbrahim BOZKURT**
e-mail: ibrahimbozkurt@cankaya.edu.tr