THE UNIVERSITY OF CALGARY

Department of Electrical and Computer Engineering

ENCM 467

PSPICE Tutorial

This tutorial is loosely based on a Dr. Haslett's PSPICE tutorial

PSPICE is made up of a number of functional modules such as a schematics editor, a graphical output display screen called PROBE, a printed circuit board layout module, analog, digital and mixed-mode simulation modules, etc. We will only use the schematic editor, PROBE, and the analog/digital simulation portions of the program.

All simulations begin with the creation of a circuit schematic in the schematic editor. This is invoked by:

1. clicking on "capture" (**NOTE** use "schematics" in the student version of PSPICE which will take you directly to a window equivalent to Figure 4) in the program group. The opening screen should look something like this (it may take a while…):
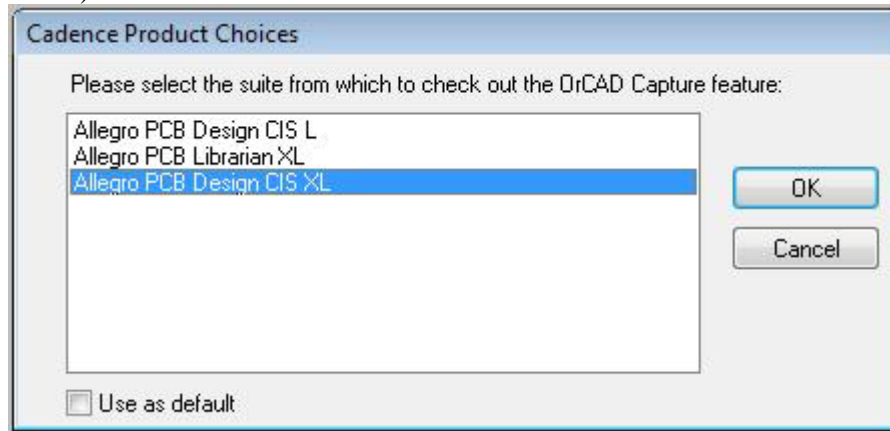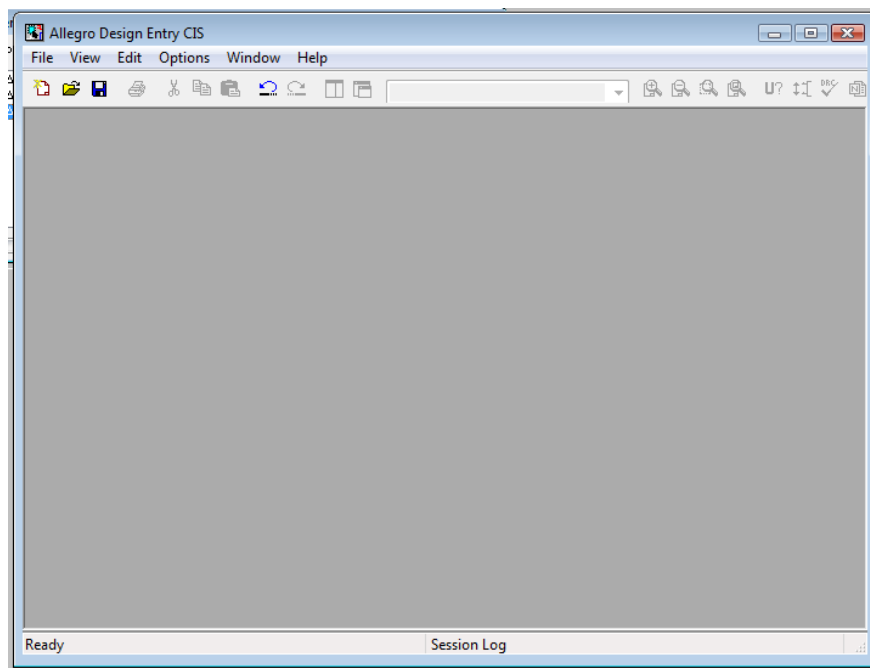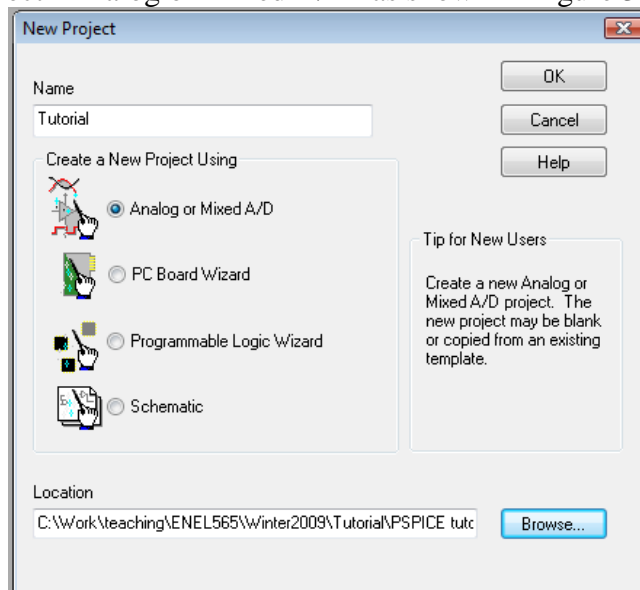

Figure 1

2. Select Allegro PCB Design CIS XL and click OK. This will result in the following

Figure 2

3. Under File menu, select New->Project, type the project name, for example Tutorial, select "Analog or Mixed A/D" as shown in Figure 3 and click OK

Figure 3

4. Another window will pop-up, select "Create a blank project" and click OK. This will create a schematic window in Figure 4 where the circuits are created.
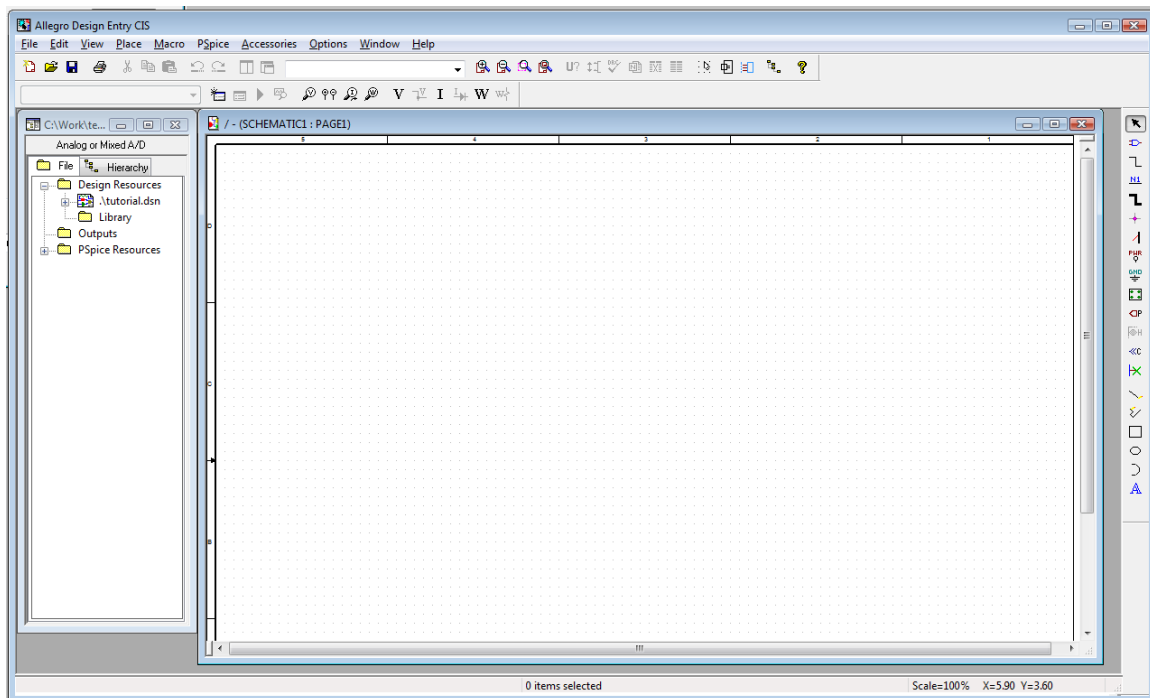
Figure 4

Once the schematic window is displayed you can start creating your circuit by
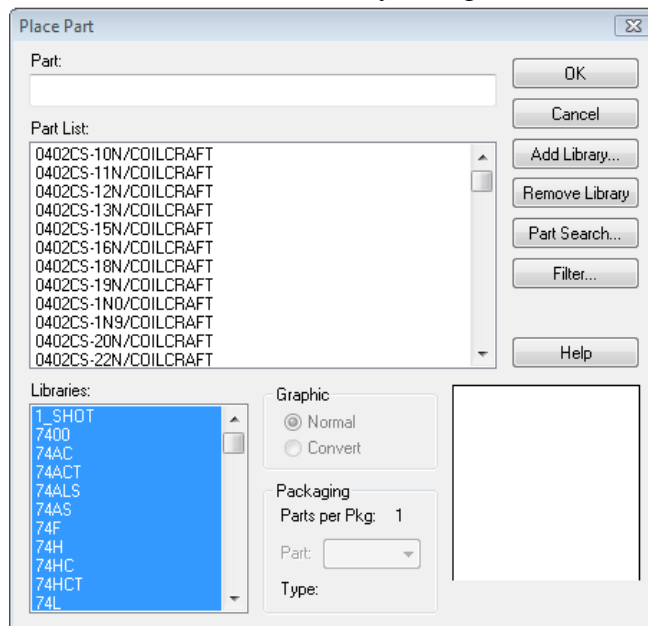1. On the menu bar click on Place->Part and you'll get the following screen

Figure 5

There are many libraries that come with PSPICE but the ones we'll use often are:
ANALOG - This contains passive components such as resistors and capacitors.
BREAKOUT-This library provides uncommitted parts that you can customize for your own models.
SOURCE - This is where we get dc and programmable signal sources.

When a part is selected it will follow your cursor on the schematic screen. You can then place the part by clicking on the schematic screen. Right click quits part placement.

As with other windows programs Ctrl+C and Ctrl+V will duplicate a selected part.

2. To connect parts with wires, use either the wire button on the toolbar, short cut "w", or menu Place->Wire. The wires are drawn by clicking and releasing to set a starting point. When finished, right click to return to the cursor. Note when connecting parts, be sure their leads do not overlap, otherwise a connection will not be made.
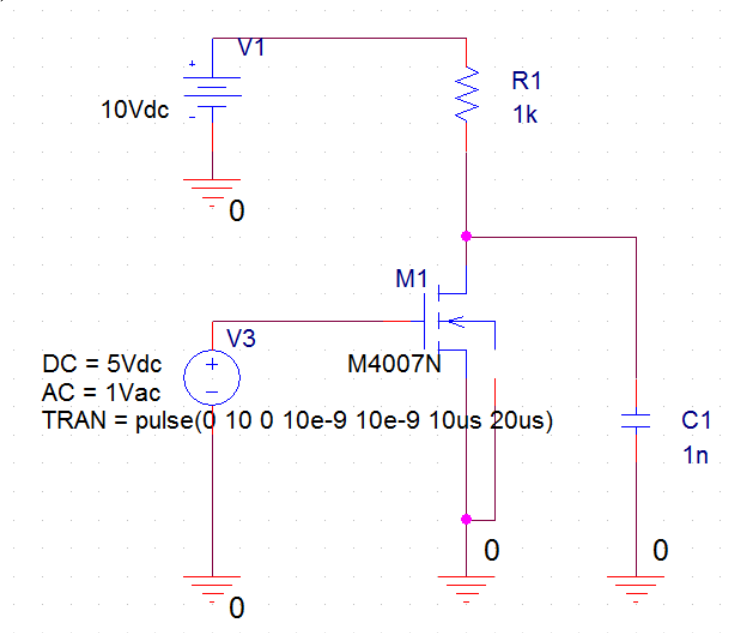
Following this, build a schematic shown below:

Figure 6

Where R1 and C1 are from Analog library, M1 is MbreakN transistor from Breakout library (note it will change its name once the model information below is added), VSRC and VDC are from Source library, the ground is found by clicking on Place->power menu, and the output voltage probe is found on the tool menu at the top of the schematic.

Since M1 is from the Breakout library, it does not have a model assigned to it. To assign a transistor model to M1
   1. Select M1 and the right click and select "Edit PSPICE model"
   2. In the window that appears type

```
.model M4007N nmos( Level=1 Tox=300n Uo=600 Kp=20.54u W=144u L=8u
+ Vto=1.3 Lambda=15m Cbd=4p Cbs=4p Cgdo=1.7n Cgso=1.7n Rs=1 Rd=1)
```
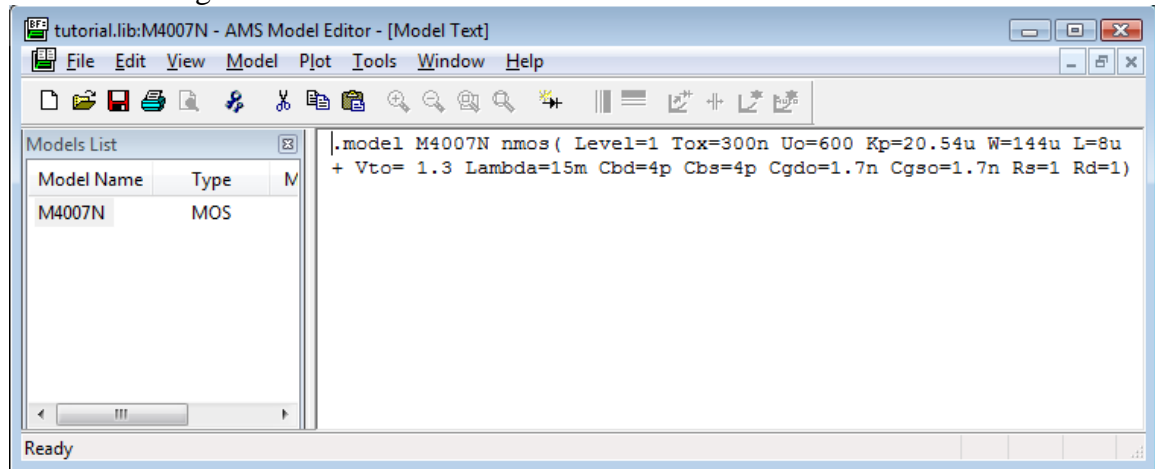
4

as shown in Figure 7:

Figure 7

3. Click on Save and then exit. This will convert an empty model to a model of M4007N NMOS transistor.
4. To change voltage V1, double click on 0Vdc, and change 0Vdc to 10Vdc
5. Do the same to change V3 to have 1Vac amplitude, 5Vdc offset, and a transient pulse waveform which steps from 0 to 10 volts with 0 time delay, 10ns rise and fall times, a pulse width of 10us and a period of 20us.

The resultant circuit is a simple amplifier set up with a VSRC source that can be used to do dc, ac and time domain analysis. To simulate this circuit, the simulation analyses need to be setup. To do this:

1. Click on PSPICE->New Simulation Profile, give the simulation profile a name, for example Tutorial, and click Create. This will bring up a window shown in Figure 8.
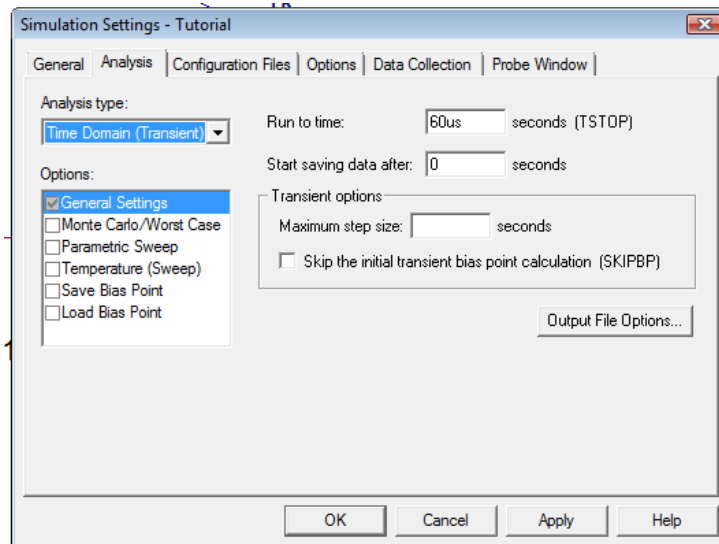

Figure 8

2. Modify Run to time to 60us and press apply.

3. This resultant setup window is used to set up a variety of analyses such as DC Sweep, AC Sweep, and Time domain (Transient) analysis as shown in Figure 8. These three are the ones you will use the most.

    *   Since in Figure 8 the Time domain (Transient) analysis is already setup, press OK and press run under PSPICE menu. Probe window will appear and run automatically. Once the simulation is finished the results can be displayed in the Probe graphical display screen by selecting Trace->Add Trace and then selecting V(M1,d) and V(M1,g), i.e. drain and gate voltages of M1. This will result in the following data display:
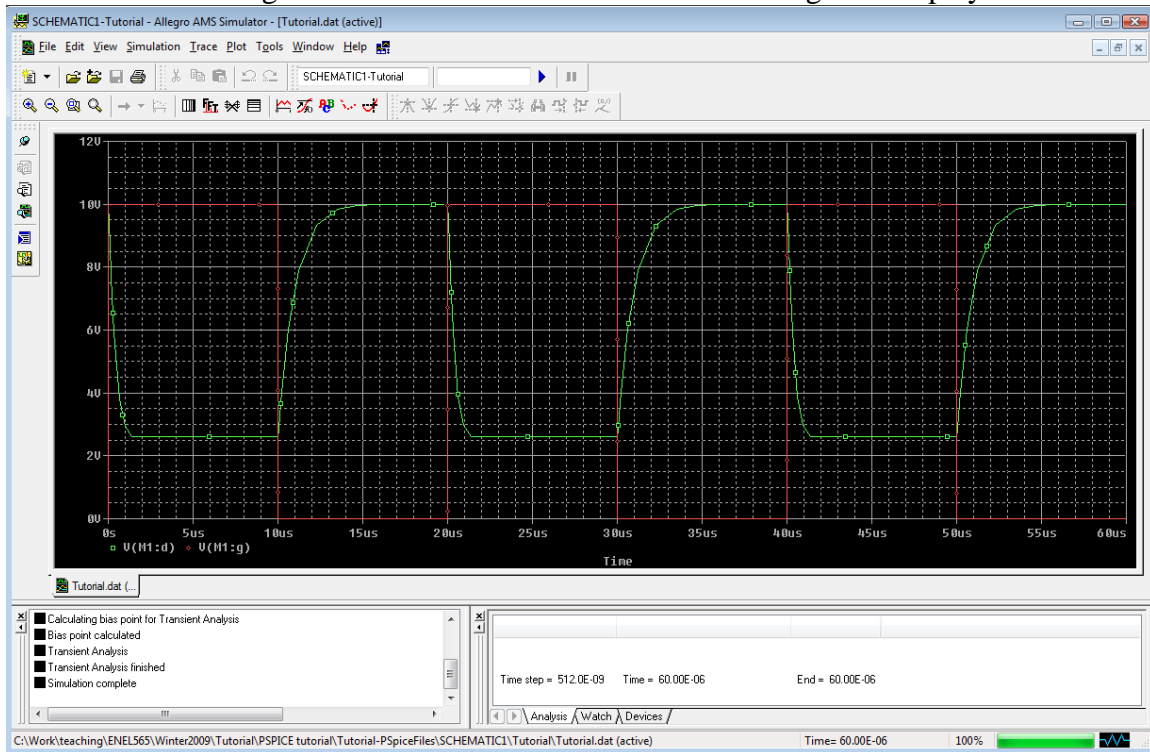

Figure 9

- For DC sweep, edit the simulation profile and select DC sweep. You will see the following setup screen:
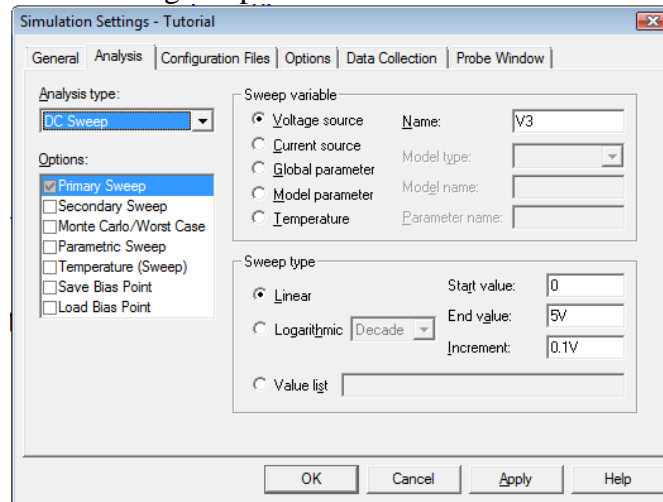

Figure 10

Where you enter sweep variable name and start, end and increment values as shown in Figure 10 and click on Apply. Run simulation the same way as for the Time domain analysis and display voltage on the drain of M1. You should get the following plot:
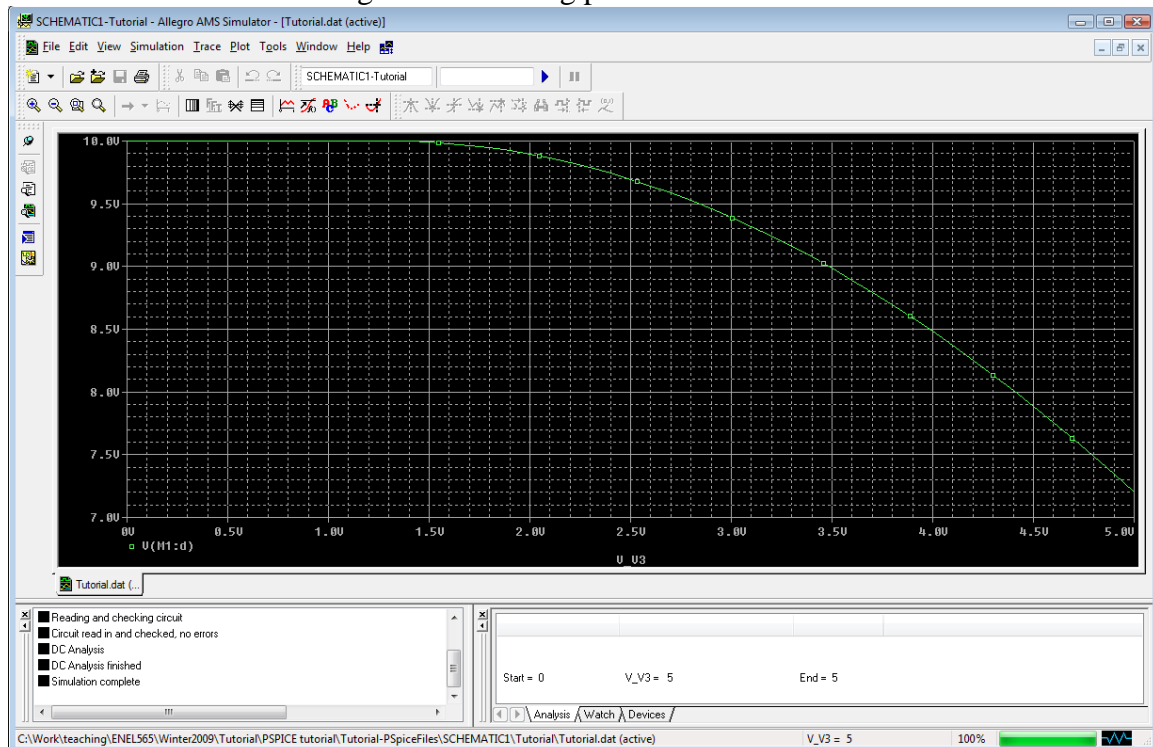

Figure 11

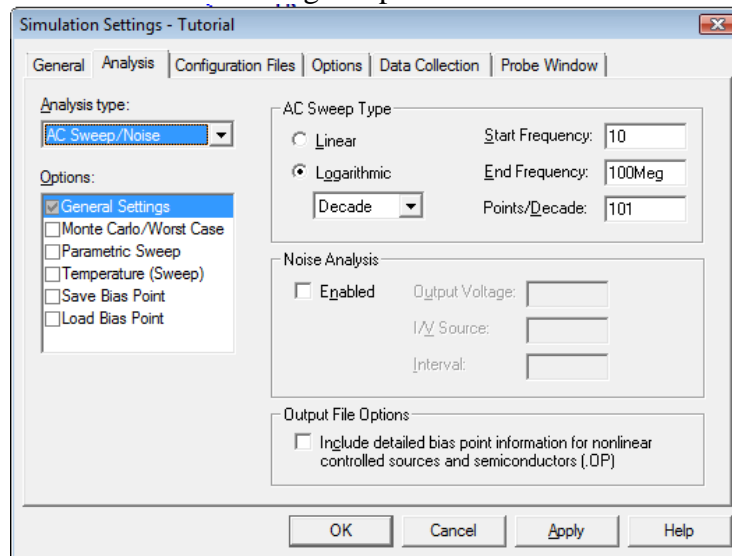- For AC sweep, edit the simulation setup and select AC sweep. You will see the following setup screen:

Figure 12

Enter values are shown in Figure 12 and click Apply. Run the simulation and verify that you obtain the following results:
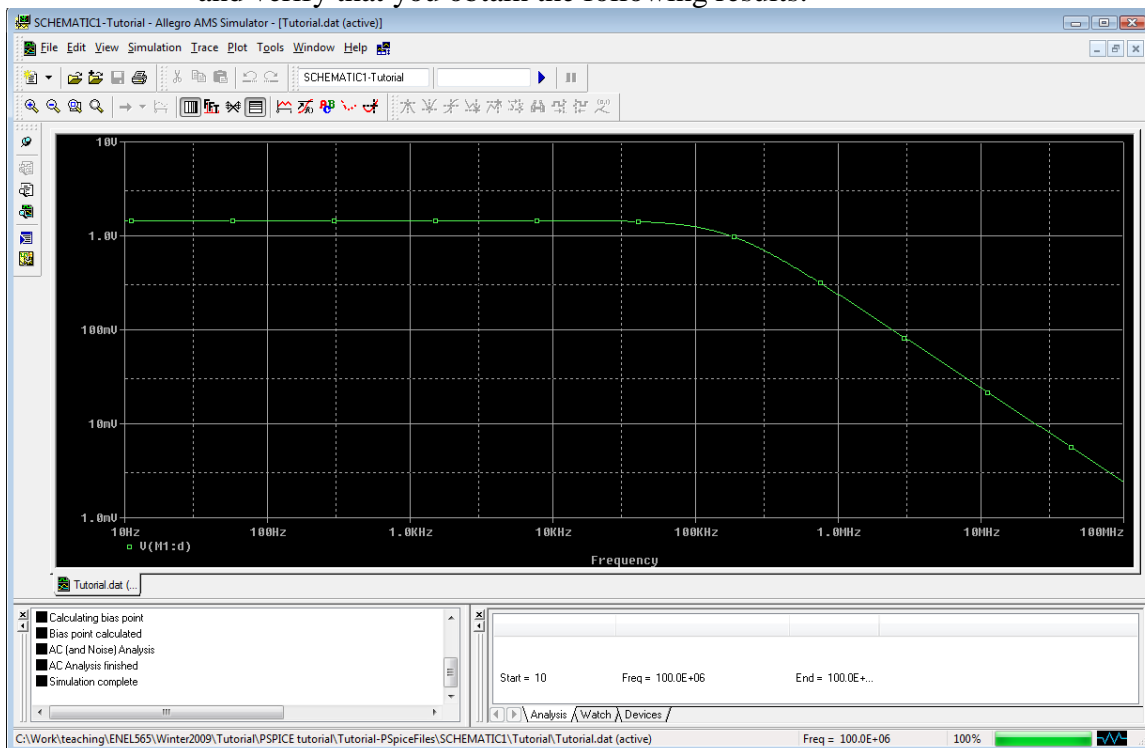

Figure 13

Note that the Y-axis is in the log scale.

These three screens in Figure 8, Figure 10, and Figure 12 are pretty much self-explanatory. In each case, intelligent choices must be made for the start and end points.

For example, in time domain analysis, if a final time is chosen that is much smaller than the period chosen in the transient source specification, then less than one cycle of the input will be seen in the transient output display.

Other notes
- The Probe screen allows the user to select cursors which can be moved along the traces to get exact values and differences. Explore many graphical functions of the Probe window.
- Voltages and currents can be also displayed on the main schematic by pressing "V" and "I" icons under the menu bar. This is quite helpful in some instances.
- The screen shots are for PSPICE 15.7, the student version 9.1 looks a bit different by the idea is the same.

This tutorial should be enough direction to get a novice user started. The program is very sophisticated, and we have only scratched the surface here. For a more detailed description of advanced capabilities, see the online documentation supplied with the program.