

Tool Instructions for Lab #1:

1. Copy the directory P:/CourseNotes/ELEC3500/Lab1_2006 to your work directory on your drive (ex. W:\ELEC3500\lab1)

2. Run Orcad 10.5 Demo – Capture CIS Demo.

3. File -> Open -> Project

Choose the appropriate project from your W: drive. Note that the project files correspond to the figures in the lab manual as follows:

NMOS1.opj => Figure 2.2

NMOS2.opj => Figure 2.4

PMOS1.opj => Figure 2.5

PMOS2.opj => Figure 2.6

As the project files were created using an older version, it will ask you if you would like to convert the project to the newer version when you open each of the above projects. DO THE CONVERSION.

4. *To open the schematic:*

In the project window, under the "Design Resources" folder, there is a design file with the same name as the project (for PMOS1.opj it is called pmos1.dsn). Click the plus sign beside this file to see its contents. You will see a "Schematic1" folder. Click on the plus sign to see its contents. Double click on "Page1" to open the schematic.

5. To ***change the values*** of the components such as resistors and capacitors, double click on the value and edit it. If the value is not displayed on the schematic, double click on the component and from the list of properties find the value you need to change and change it. If you want a property displayed on the schematic, select the box with its value and right click. From the Display menu item, choose the display format you want (Name and Value is usually most appropriate). If you get a warning saying "Cannot save changes to the property filters..." simply say yes and save the file in your project directory (W:\ELEC3500\lab1 for example). When changing wires be sure to use "place wire," not "place line."
6. Once the schematic is ready for ***simulation***, hit the "Play" Button in the toolbar, or from the "PSpice" menu, go to "Run".

7. After a few seconds of running the simulation, your **simulation output** will be displayed. You can go back to your schematic to see which color waveforms correspond to which nodes. You will see a probe symbol with the appropriate color on the nodes corresponding to the waveforms.
8. The simulation is set up as a **time domain simulation** initially. When you want to run a **DC simulation**, go to PSpice-> Edit Simulation Profile. From the Analysis type drop down menu, choose "DC Sweep." Make sure the name and type of the source you want to sweep is selected. Choose the range and increment for the sweep under sweep type. Hit OK, then run a simulation.
 - PSpice -> Edit Simulation Profile
 - Analysis Type = DC Sweep
 - Options: = Primary Sweep
 - Sweep Variable: Voltage Source "V1"
 - Sweep Type: Linear; fill in Start, End and Increment.
9. To add a **current or voltage probe**: Click on the icon (V) or (I) in the toolbar and click on the schematic. Select a wire for a voltage probe, or a component pin for a current probe. Or PSpice -> Markers -> [Voltage Level, Voltage Differential, Current into Pin]
10. To see **other voltage or current waveforms** in the Pspice output window, go to Trace ->Add Trace. Select a function from the list on the right hand side, and then select the appropriate voltage or current from the left hand side. Ensure the "Trace Expression" shown at the bottom is what you want, then hit OK, and it will be displayed.
 - It is a good idea to go to Plot -> Add Plot to Window before adding new traces. This way the new traces show up in their own plot, meaning the plot scales will be appropriate to the new trace.

You can run Simulations for different values of a component at the same Run. Here is how:

To create a parameter and sweep it (eg. make the value of R a parameter):

Click on its value and change it to {myparameter}

- Note that the { } brackets are IMPORTANT!
- Go to Place -> Part.
- Add Library.

- Choose Special.olb
- Find "PARAM
- click OK and drop anywhere on your schematic.
- Double click on the new PARAM block
- Choose "New Column"
- write your variable name (no curly brackets) and its default value.

To sweep this value in a simulation

- select your simulation type (DC, transient, etc...) and its values.
- Click On parametric sweep
- select Global Parameter
- enter your parameter name (no curly brackets)
- Select a minimum, maximum or step value of your parameter for your simulation
- Pspice -> Run , to perform the sweep