

University of Utah
Electrical & Computer Engineering Department
ECE 1250 Lab 4b
Pulse Width Modulation Simulation

A. Stolp, 2/14/12
rev

Note: This is an individual project lab. It may be done on a computer in our normal lab room, in MEB 2265 or in MEB 2555. Bring last week's lab handout.

Objective

Learn a little about the Spice circuit modeling software as implemented by OrCAD Cadence.


Experiment

Sign on to one of the computers like you did in the first lab to run Matlab. Make sure there is an OrCAD Capture icon on the desktop and that the program will open for you. (Be patient, it's slow, OrCAD checks its license several times during the startup and project loading.) Select the "OrCAD PCB Designer Professional w/PSpice" option. If no, move to another computer. If yes, minimize OrCAD, open a browser and download:



<http://www.ece.utah.edu/~ece1250/PWM.zip>

(Available on http://www.ece.utah.edu/~ece1250/ECE1250labs_S12.htm)

- Unzip the files into a folder called PWM (either in `..\..\Downloads\PWM` or on `X:\.`. Not the desktop)
- Restore OrCAD capture and open `PWM.opj` (this is a OrCAD Pspice project file). Wait.
- If the schematic doesn't open automatically, open "Design Resources" under the File tab of the window that does open. Look for `PWM.dsn\PWM\PWM` and double-click on it.
- Examine the schematic. You should immediately recognize it as the PWM circuit that you built in lab last week.
- Click Pspice - Run to run the simulation. (F11 or the  (play) button will also work.)
- Click the new icon on the taskbar to view the results. (Be patient.)
- These waveforms should look familiar.
- Move windows or look back and forth between the results and the schematic to see how the colored probe markers correspond to the colored traces on the results page.
- On the schematic, double-click on the `SET = 0.7` near the pot. Change the value to some other value between 0.2 and 0.8. This sets the pot wiper position.
- Run the simulation again and see how the output changed.
- Repeat for several pot positions SET values).



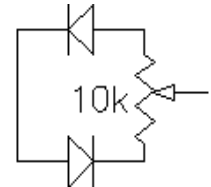
Installation on your own computer

Checkout a PSpice 16.2 DVD from the lab and make yourself a copy. On your own computer, find `Cadence\Cadence_OrCAD_Demo_162.zip` on the DVD. Unzip it into a folder on your hard drive and run `setup.exe`. Install `OrCAD_Capture_CIS` and `Pspice`. The rest isn't needed and they say that `OrCAD_Signal_Explorer` won't even work in demo mode. This will take about 800 MB of hard drive space and adds about 2.7 MB to your registry— just your modern mega-program. Consider trying Altium Design instead.

Your pro gram may work, mine didn't. Don't rely on it as your only way to finish this lab.

Now you will edit the schematic by taking out R3 and replacing it with the two diodes and the pot like you did in the final circuit of the last lab.

- Hit the top button down the right-hand column of buttons-- the one with the arrow symbol. This is the selection tool and can be used to select parts which can then be copied or moved or deleted, etc.. Delete the third op amp and all the surrounding wires and connections EXCEPT the pot. Delete the Vcc and Vee connections.
- Move the pot into a clear section of the drawing.
- Hit the "P" key on the keyboard or the "Place part" button (1st button down the 2nd right-hand column of buttons). This should bring up a whole "Place Part" window on the right side of the screen.
- Find the DIODE library and click it. If you have to, first add it by typing alt-A or clicking the tiny squarish symbol "Libraries" area of the "Place parts" area. Add it from the C:\Cadence\SPB_16.5\tools\capture\library\psice subdirectory.
- Find the D1N4148 or the D1N914 part and double-click it.
- Place two diodes near the pot so that you can connect them as needed to pot. Hit esc or the "Select" button to quit placing diodes.
- Select the top diode, right-click and hit "Mirror Horizontally" to turn it around.
- Hit the "W" key on the keyboard or the "Place wire" button (third button down the right-hand column of buttons). Use this to connect the little □ boxes with wires.
- Delete the center resistor (R3).
- Select and stretch the circuit as necessary to make room for the diodes and pot.
- Move the diodes and pot into position.
- Connect the necessary wires.
- Move the probes as needed so they are connected to parts, not just wires, or they won't work.
- Run to run the simulation and view the simulation results.
- Change the SET value of the pot.
- Run the simulation again and see how the output changed.
- Repeat for several pot positions (SET values).



Finish and submit your results.

- If you are in lab with your lab instructor, ask your him or her to come to your computer. Show off your schematic and your results. Your lab instructor will record you as finished and you're done-- that simple. But.. You may wish to save your project... just in case.

If you're on your own:

- Manipulate your windows until you get some nice results and the schematic showing on your display at the same time.
- Click on the Windows Start button and your username should show up near the top, right.
- Hit "Print Screen" on the keyboard to perform a screen capture.
- Open Paint, paste the screen capture and save it.
- Attach it to an email and send it to your lab instructor using "ECE 1250 lab 4b" as the subject line. Make sure that your name is in the email text. Email address are given below and on the Lab web page. That's it, no notebook, no nuttin'else.

"Arhatha Bramhanand" <arhatha.b@gmail.com>

"Suhass Kallappa Teli" <suhass.teli@utah.edu>

"Peter Hillyard" <peterhillyard@gmail.com>