A Simple PSpice Example

<u>Topic</u>: PSpice Familiarization

<u>Date</u>: 01 September 2016

<u>Preliminary</u>:

n/a

<u>Introduction:</u>

The purpose of this lab is:

1. Load the PSpice software onto your PC.
2. Enter the sample circuit.
3. Print out the circuit, voltage waveform and current waveform (with formatting)

If you desire to use another version of PSpice for your lab assignments, that is fine.

<u>Pre-lab:</u>

n/a.

<u>In the Lab</u>

Bring your PC to the lab and we will get the software running. You can download the PSpice program from the F drive (F:/dept/ece/ee221). We will have Pspice on DVDs available in lab if you have trouble getting on the network.

<u>Analysis and Conclusions</u>:

n/a

<u>Lab Report Checklist:</u>

Hand in (no formal lab report required):

1. A printout of the circuit,.voltage waveform and current waveform (with formatting). Put your name on the printouts.

<u>!</u>

A Simple PSpice Example

This installation was verified:

- On 24 August 2016 on one of the lab PCs which was running the Windows XP operating system.
- On 02 September 2016 on a Windows 7 system

1) Copy the “91pspstu.exe” file to a known directory, perhaps “c:\” or c:\pspice91\” or create a directory for it of your choosing. If you have the file on a DVD, you can run it from the DVD, but you will still have to specify where the unzipped files will go.
2) Run the “91pspstu.exe” file to unzip the PSpice program. You might be able to do this by double clicking on the file. Make a note as to where the un-zipped files are located.
3) Run the “setup.exe” file. You will need administrative privileges on your pc to do this.
   i) “OK” to the ”turn off your virus detection…” message.
   ii) “Yes” to the “admin privileges…” message.
   iii) Coose the “Capture” selection as the schematic editor.
   iv) Specify the destination folder that you want the program to be installed in. On my pc, the destination default was C:\Program Files\OrCad_Demo” and click on “Next”.
   v) The default program folders selection will be “PSpice Student”. Click “Next”.
   vi) If the current settings are ok, click “Next”.
   vii) The installation should take about 30 seconds or so. Click “Finish” if all went well.
   viii) You may get a few pages of release notes displayed after the installation completes.

4) Start / All Programs / PSpice Student / Capture Student to start the schematic capture program.
5) Start / All Programs / PSpice Student / PSpice AD Student to start the pspice simulator program. Leave both of these programs running in their windows.
6) Go back to the “OrCAD Capture” window and maximize the window. You will enter your schematic in this window.

7) File New Project
   a) Name = Simple Example
   b) Select “Analog or Mixed A/D”
   c) Location Browse
      i) Click on a directory that will hold the new project directory you will create in the next step.
      ii) Click “Create Dir”
      iii) In the Name box: simple example OK
      iv) Click on the “simple example” directory you just created OK OK
   d) In “Create PSpice Project” window, “Create A Blank Project” selected OK
   e) You should now see the schematic entry screen
   f) Minimize, then maximize to see the buttons on the far right side

8) Click on the “Place part” button to add dc voltage source
   a) Add Library = source.olb Open
   b) Click on “Source” library
   c) Select “VDC” as dc voltage source
   d) OK
   e) Place voltage source symbol on the schematic by clicking on the schematic page
   f) ESC
   g) Click on the “0Vdc” text and change it to “10” (ten) OK

h) ESC

9) Click on the “Place a part” icon (button on the right side of screen) to add resistors
   a) Add Library = analog.olb Open
   b) Click on the Analog library
   c) Select “R” to place resistors OK
   d) Place 3 resistor symbols on the schematic. Each click on the schematic page will insert one resistor. See example (attached) for circuit component placement suggestions.
   e) ESC
   f) Click on each of the horizontally placed resistors and hit “R” three times to rotate the resistors to a vertical position. This places terminal 1 of each resistor toward the top of the schematic.
   g) ESC

10) Click on “Place wire” icon (button on the right side of the screen), and connect the resistors and the voltage source in a series circuit. ESC ESC

11) Click on “Place a ground symbol” and place the ground node.
   a) Select the “GND/CAPSYM” symbol
   b) Super Important: CHANGE THE NAME OF THE SYMBOL FROM “GND” TO “0” (zero).
   c) OK
   d) Place the ground symbol on the bottom wire of the circuit.
   e) ESC ESC

12) Add voltage probes to the top node of each resistor. You can also enable digital value displays of circuit parameters on the schematic with the “Enable bias voltage / current” buttons. ESC.
13) Add a current probe to the upper node of the top resistor ESC.

14) Set each resistor value by clicking on it and changing the default 1k values to:
   a) R1 30 OK
   b) R2 40 OK
   c) R3 60 OK
   d) ESC

15) Update the documentation block in the lower right hand corner:
   a) Click the “<Title>” entry in the Title block and replace the text with “Getting Familiar with PSpice” OK
   b) Click the “<Doc>” entry in the Document Number block and replace the text with “project = simple example.opj” OK
   c) Click the “<Rev Code>” entry in the Rev block and replace the text with a “-” (dash symbol) OK
   d) ESC

16) FILE SAVE

17) PSpice New Simulation Profile Name = “simple example” Inherit From: None Create
   a) Analysis tab: Analysis type = Time Domain (Transient)
   b) Check the General tab to verify that the name was set to “simple example”.

c) Go back to the Analysis tab: Run to time: 100m seconds (see chart below for PSpice scale factors)
d) Start Saving After: 0 (zero) seconds
e) Maximum Step Size: .1m seconds
f) Check the box “Skip the initial transientbias point calculation”
g) Apply
h) OK

18) File Save

19) PSpice Run

20) Select the OrCAD PSpice A/D Demo screen. You should see the voltage and current waveforms in different colors on the graph.

21) Thereafter, you should be able to make changes on the schematic, do a “PSpice Run”, and the changes should be reflected as new data traces on the PSpice A/D Lite screen.

22) We can format the voltage and current plot for a pleasing printout.
a) File / Page Setup / Header
b) Remove all text in the “Left Side” text area.
c) Remove all text in the “Right Side” text area.
d) Remove all text in the “Center” text area.
e) OK
f) Footer
g) Remove all text in the “Left Side” text area.
h) Remove all text in the “Right Side” text area.
i) Leave the “Page &N” text in the “Center” text area.
j) OK OK
k) Window / Title / type in “Simple Example” OK
l) File / Print Preview. A nice, printable plot should appear. CLOSE
m) File / Print / select a printer / OK

23) We can format the schematic for a pleasing printout.
a) Go back to the schematic window.
b) File / Print Preview. A nice, printable plot should appear. CLOSE
c) File / Print / select a printer / OK

24) Close and Save all windows when done.

A Simple PSpice Example

PSpice Scale Factor Chart (from “Introduction to PSpice Manual for Electric Circuits”, Nilsson & Riedel, Prentice Hall

Table 1.1: PSpice Scale Factors

SYMBOL	EXPONENTIAL FORM	VALUE
F (or f)	1E−15	10^{-15}
P (or p)	1E−12	10^{-12}
N (or n)	1E−9	10^{-9}
U (or u)	1E−6	10^{-6}
M (or m)	1E−3	10^{-3}
K (or k)	1E3	10^{3}
MEG (or meg)	1E6	10^{6}
G (or g)	1E9	10^{9}
T (or t)	1E12	10^{12}

Notes:

1. While in the PSpice A/D Lite window, you can plot other circuit variables that are calculated during the simulation for example:
   a. TRACE Add Traces:
   b. V(R1:1) V(R2:1) V(R3:1)