

Microelectronic Circuits

8th Edition

A. Sedra, K.C. Smith
T. Chan Carusone, V. Gaudet

PSpice Simulation Guide

Prepared by: Nijwm Wary

2019

PSpice is a commercial SPICE simulator available for purchase. Many colleges and universities make PSpice available to students. Moreover, old student versions of PSpice are readily available and perfectly suitable for the examples in this book. The PSpice examples for Microelectronic Circuits were prepared with PSpice student version 9.1.

A wide array of tutorials and guides for PSpice are also easily found. For example, here are some websites with information to help you get started, current at the time of publication:

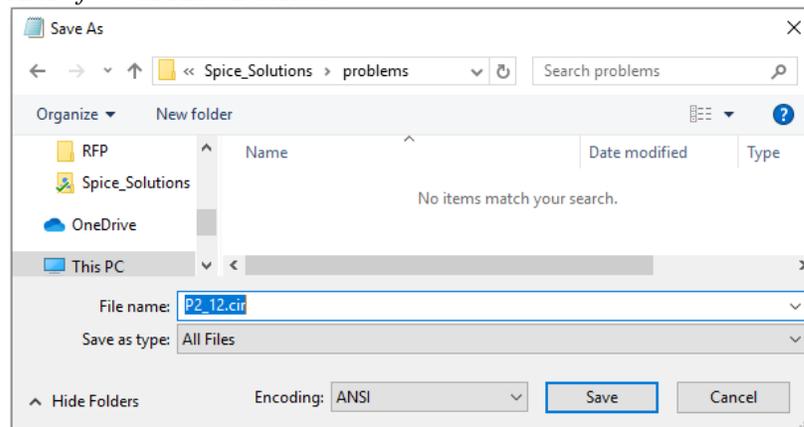
http://www.eng.auburn.edu/~troppel/pspice_links.html

<https://www.electronics-lab.com/downloads/circutedesignsimulation/?page=5>

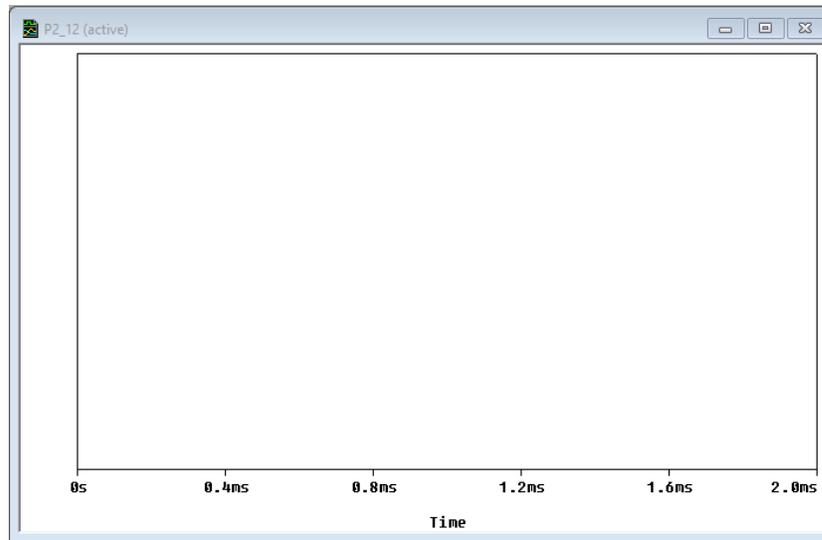
<http://www.uta.edu/ee/hw/pspice/>

Using the PSpice simulation examples for Microelectronic Circuits

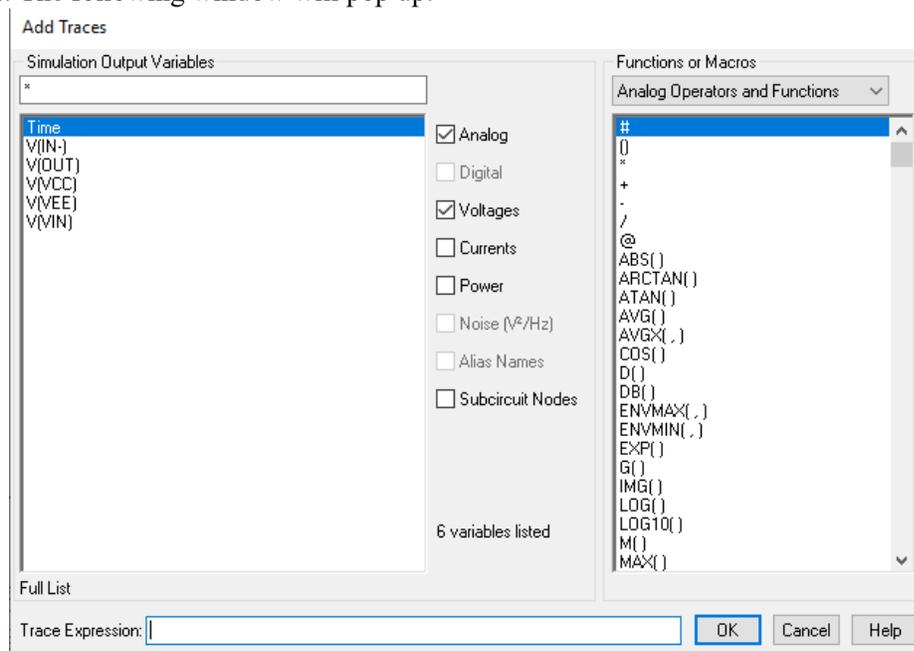
1. Copy any of the provided netlists into a text file and save it with the *.cir extension. In Windows, this can be done using the *Save As* option. Give it a filename <filenam>.cir and in *Save as type* chose the option *All files* as shown below.



2. Start *PSpice AD Lite* from the *Program Menu*.
3. Select *File>Open* and go to the folder where you have saved the *.cir file. Note that to see the file, the *File of type* should be set to *Circuit Files (*.cir)*.
4. To run the simulation (Transient, AC Sweep, DC Sweep, Operating Point) select *Simulation>Run*.
5. To view the details of the operating point, select *View>Output file*.
6. In most of the problems, upon running the simulation, you will see a blank simulation results window as shown below



7. To plot a waveform (voltage or current), select *Trace>Add Trace*. Or use the key *Insert* from the keyboard. The following window will pop up.



8. Select the waveform you want to plot, or type an expression in the text box *Trace Expression* to perform various mathematical functions on the simulation results.
9. You can use cursors to read any specific point on the output waveforms by selecting *Trace>Cursor>Display*.
10. If you are plotting multiple waveforms, you can find the information for any particular trace by right-clicking the trace and selecting *Trace Information* as shown below.