Microelectronic Circuits
8th Edition

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

LTSpice Simulation Guide

Prepared by: Nijwm Wary

2019

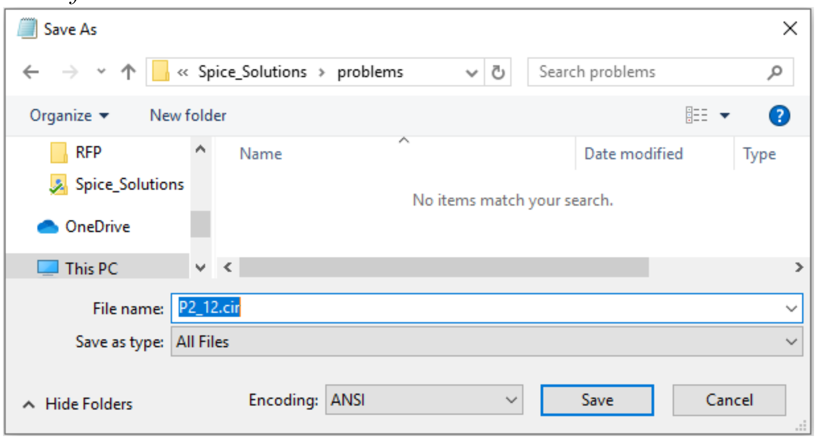
LTSpice is a free SPICE simulator available from Analog Devices. It is currently available here, along with extensive documentation and examples to help you get started:

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

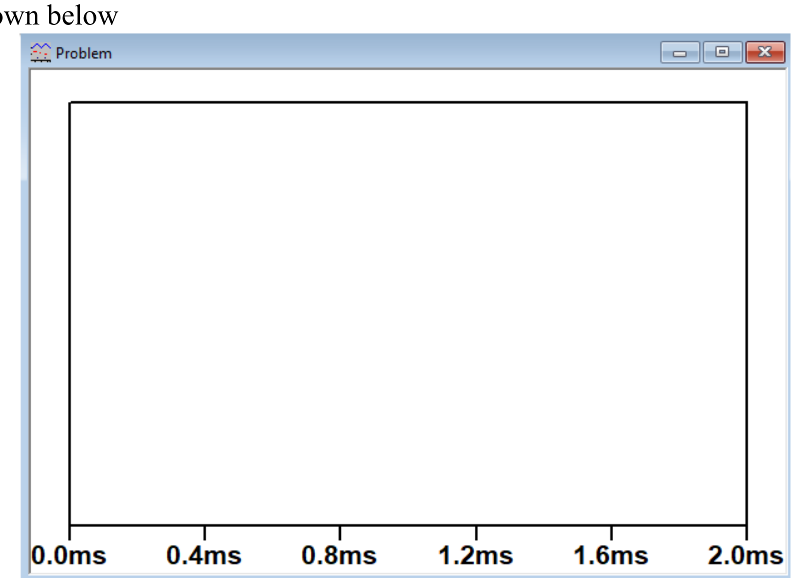
Furthermore, a wide array of tutorials and guides are readily available online.

Using the LTSpice simulation examples for Microelectronic Circuits

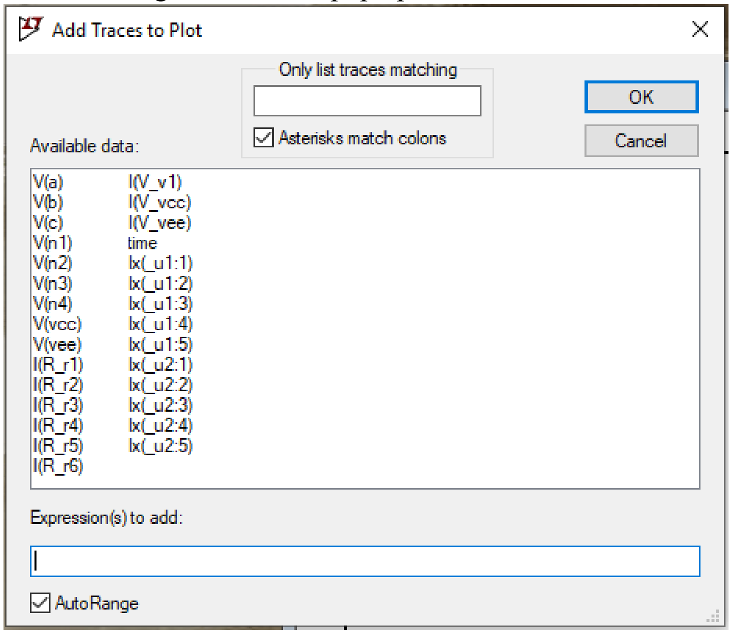
1. Copy any of the provided netlists into a text file and save it with the *.cir extension. In Windows, this can be done using the *Save As* option. Give it a filename <filenam>.cir and in *Save as type* chose the option *All files* as shown below.



2. Start *LTSpice* 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 *Simulate>Run*.
5. In most of the problems, upon running the simulation, you will see a blank simulation results window as shown below



6. To plot a waveform (voltage or current), right click in the blank area of the above window and select *Add Trace*. The following window will pop up.



7. Select the waveform you want to plot. Or type the expression in the text box *Trace Expression*. You can also perform various mathematical functions on the simulation results.