EE 230 – Analog Circuits Lab - 2021-22

Homework 1: Familiarization with NGSPICE Circuit Simulator and Lab Equipment
(Jan 09, 2022)

Part A – NGSPICE Familiarization

A.1 Installation

Introduction: Ngspice is a mixed-level/mixed-signal circuit simulator based on three open source software packages: Spice3f5, Cider1b1 and Xspice. Spice3 is the most famous and used circuit simulator developed University of California at Berkeley (UCB).

We will be using the latest version, viz. NGSPICE, Ver 34 (Jan 2021). In case you have an earlier version of NGSPICE, it should be ok, since we will be using only the basic commands.

Download and Installation details (Windows)
NGSPICE-34 can be downloaded from http://ngspice.sourceforge.net/download.html

In the above webpage, see the section "**Ngspice installation (quick intro)**" for the link for MS Windows (64 bit, Windows 10) version. Expand the contents of the zip file to an arbitrary location on your computer, e.g. to D:\.

Download and Installation details (Linux version)

For installing ngspice on Linux, macOS and other OSs please see:
http://ngspice.sourceforge.net/packages.html

A.2 WEL Lab Resources on NGSPICE

WEL Lab links (for access within IITB campus):

Download the following NGSPICE resources (Tutorial Slide and SPICE files)

http://wel.ee.iitb.ac.in/teaching_labs/WEL%20Site/ee236_temp/ngspice.html

Download NGSPICE Video Tutorial for NGSPICE Linux version (see under Expt 1, Download links)

http://wel.ee.iitb.ac.in/teaching_labs/WEL%20Site/ee236_temp/labsheets_2019.html

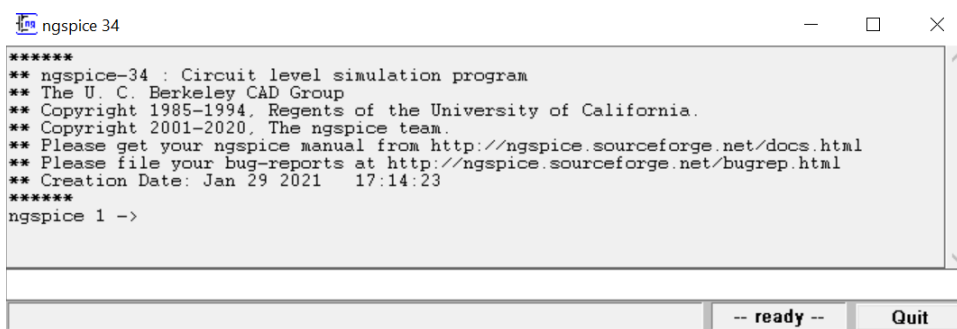**Google drive link (for access from outside IITB campus):**

WEL Lab resources on NGSPICE are copied in the following Google drive

https://drive.google.com/drive/folders/19tCwquDaB4wkwRbvcO-GF_t6GAC4S0vB?usp=sharing

A.3 Basic NGSPICE Operations

1. Windows 10 version:
   Run NGSPICE (from Spice64/bin folder) by double-clicking 'ngspice.exe'. You will see the following.

   ```
   ngspice 34                                                    —  □  ✕
   ******
   ** ngspice-34 : Circuit level simulation program
   ** The U. C. Berkeley CAD Group
   ** Copyright 1985-1994, Regents of the University of California.
   ** Copyright 2001-2020, The ngspice team.
   ** Please get your ngspice manual from http://ngspice.sourceforge.net/docs.html
   ** Please file your bug-reports at http://ngspice.sourceforge.net/bugrep.html
   ** Creation Date: Jan 29 2021   17:14:23
   ******
   ngspice 1 ->

                                                    -- ready --    Quit
   ```

2. Please see the WEL Lab resources:

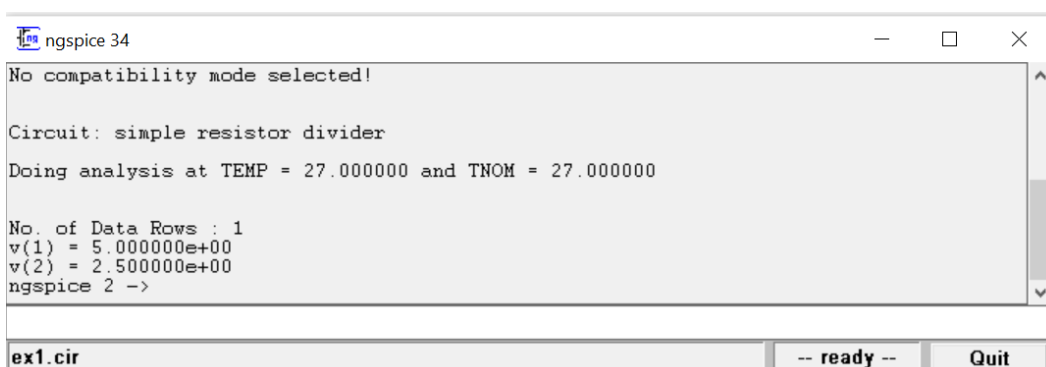   **NGSPICE Tutorial Video (Ngspice_tut.mp4) and PPT (Ngspice-PPT-2017.pdf)**

   The above two resources discuss a variety of NGSPICE simulation examples. The above resources were made for the Linux version. However, the commands and the results are identical with the Windows version.

   A Sample NGSPICE Example:

   Create a sample netlist file ('.CIR'file) using the Notepad. A simple example of resistor divider is shown below.

   > Simple resistor divider
   > * <element-name> <nodes> <value/model>
   > r1 1 2 1k
   > r2 2 0 1k
   > v1 1 0 5
   > *analysis command
   > .op
   > .control
   > run
   > *display command
   > print v(1)
   > print v(2)
   > .endc
   > .end

   Note that the first line is assumed by NGSPICE as a Title (or Comment line). All subsequent comment lines should start with '*'. Run the file by typing 'Ex1.cir' in the command window. The resulting display is shown below.

   ```
   ngspice 34                                                    —  □  ✕
   No compatibility mode selected!

   Circuit: simple resistor divider

   Doing analysis at TEMP = 27.000000 and TNOM = 27.000000

   No. of Data Rows : 1
   v(1) = 5.000000e+00
   v(2) = 2.500000e+00
   ngspice 2 ->

   ex1.cir                                          -- ready --    Quit
   ```

3. Familiarize yourself with the basic functions of NGSPICE, by creating and running all the examples given in the Tutorial video. You need to be familiar with the commonly used analysis commands (.op, .dc, .tran, .ac). You need not do the last example in the Tutorial video (Opamp integrator).
4. Experiment 1 requires the use of sine and pulse waveforms. Familiarize yourself with the correct format for these waveforms.
5. Experiment 1 comprises the following RC and RLC circuits: RC integrator, RC differentiator, RC Low-pass filter, RC high-pass filter, RC bandpass filter and RLC bandpass filter.

Part B : NGSPICE Simulation of RC and RLC Circuits

B.1 RC Integrator
Simulate and plot time response of the RC integrator circuit shown below for the cases indicated. Choose R = 10 kΩ and C = 0.1 µF. Your plots should show both V_{in} and V_{out}. Ensure that V_{out} waveform has at least two or three cycles of the steady state waveform.

Hint: Choose square pulse waveform with amplitudes as shown and use .tran analysis. Choose pulse width and square waveform period as per the requirement. (T = input pulse width, and τ = RC time constant).
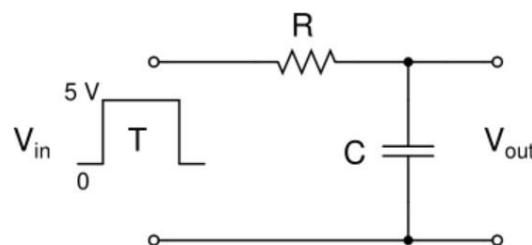
Format for the pulse waveform:
        pulse(Low-value Pulsed-value TD TR TF PW period)
        TD = time delay, TR = rise time, TF = fall time, PW = pulse width, period = pulse waveform period

Format for .tran analysis:
.tran TSTEP TSTOP

        Cases: i) T = 10τ ; ii) T = 5τ ; iii) T = τ ; iv) T = 0.1 τ ; vi) T = 0.05 τ

.

B.2 RC Differentiator
Simulate and plot time response of the RC differentiator circuit shown below for the indicated cases. Choose R = 10 kΩ and C = 0.1 µF. Your plots should show both V_{in} and V_{out}. Ensure that V_{out} waveform has at least two or three cycles of the steady state waveform.
.
        Cases: i) T = 10 τ ; ii) T = 5 τ ; iii) T = 1 τ ; iv) T = 0.5 τ ; v) T = 0.1 τ ; vi) T = 0.05 τ
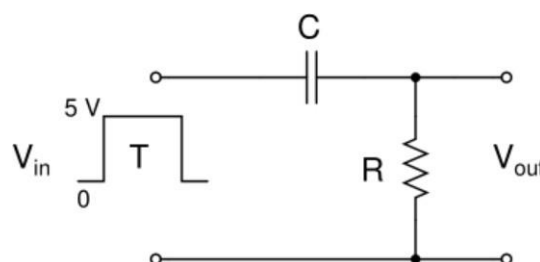
B.3 RC Lowpass Filter

Simulate and plot amplitude frequency response (amplitude Bode plot) of the RC lowpass filter circuit shown below. Choose R = 10 kΩ and C = 0.1 μF.

Hint: Choose sine waveform with amplitude 1 V. Choose .ac analysis with decade frequency variation.
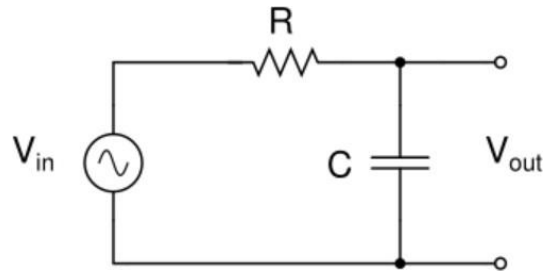
Format for the .ac command:

.ac DEC ND FSTART FSTOP

DEC stands for decade variation, and ND is the number of points per decade.

FSTART is the starting frequency, and FSTOP is the final frequency.

Specify an AC source with zero dc and ac amplitude 1V(Example: vin between nodes 1,0): vin 1 0 dc 0 ac 1
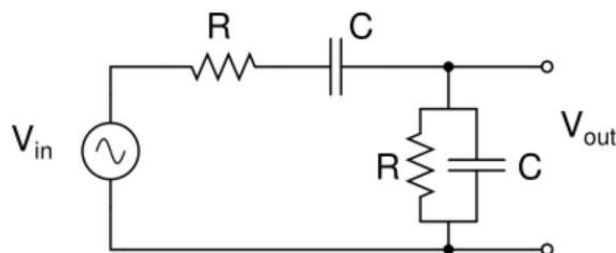

B.4 RC Highpass Filter

Simulate and plot amplitude frequency response (amplitude Bode plot) of the RC highpass filter circuit shown below. Choose R = 10 kΩ and C = 0.1 μF.
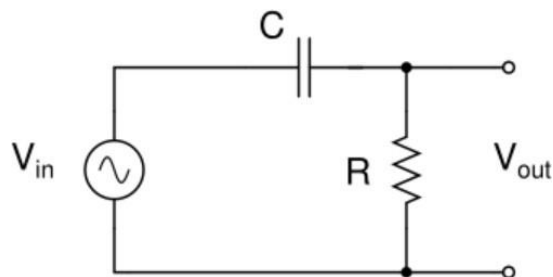
B.5 RC Bandpass Filter

Simulate and plot amplitude frequency response (amplitude Bode plot) of the RC bandpass filter circuit shown below. Choose R = 10 kΩ and C = 0.1 μF.

Note its peak amplitude, centre frequency(f_0), and the lower (f_L) and the upper(f_H) -3 dB points. Compare your simulation results with theory.

B.5 RLC Bandpass Filter

Simulate and plot amplitude frequency response (amplitude Bode plot) of the RLC bandpass filter circuit shown below. Choose R = 1 kΩ, L = 10 mH and C = 0.1 µF.

Note its peak amplitude, centre frequency(f_0), and the lower (f_L) and the upper(f_H) -3 dB points. Compare your simulation results with theory.
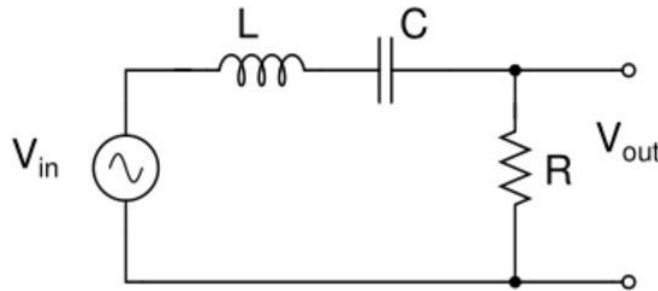
Part C – Familiarization with Lab Equipment

C.1 Digital Multimeter (DMM)

You will be using a Laboratory Table-top Digital Multimeter (DMM). This is an electronic meter used for measuring voltages, currents and resistances. The meter display shows voltages, currents and resistances on the LED display panel.

Measurements: First of all you need to decide the parameter to be measured. We shall use the DMM primarily for measuring voltages and resistances. Choose the Voltage or Resistance function and the appropriate range (based on the maximum expected magnitude) for the parameter. Choose the ranges carefully.

Note: Resistance mode of the DMM assumes that there is no current flowing in the resistor. Take extra care when using the resistance mode.

You may think of DMM essentially as a voltage measuring equipment. Resistances and currents are converted into voltages by the DMM circuitry. AC voltages need to be < 400 Hz.
Input resistance of the DMM is 10 Mohms.

C.2 Digital Storage Oscilloscope (DSO) - Tektronix TDS 1002B

Oscilloscopes are versatile electronic instruments used for displaying and measuring parameters of time varying voltage signals. They are very useful in measuring the amplitude and frequency/time period of a signal. The instrument most commonly used is what is called a Cathode Ray Oscilloscope (CRO). CROs are now slowly getting replaced by Digital Storage Oscilloscopes (DSO), which makes signal measurements easier compared to a CRO.

CROs/DSOs are useful in measuring signals up to their rated bandwidth. TDS 1002B has bandwidth of 60 MHz. However, DSOs cannot measure DC/AC voltages as accurately as a DMM. ADC Resolution for the DSO vertical signal is typically 8 bits (as compared to 12 bits or more in a DMM).

C.3 Arbitrary Function Generator

Arbitrary Function Generator (AFG) is a special function generator which can choose a variety of waveforms (sine, square, ramp, arbitrary, etc) whose parameters, such as amplitude, frequency, offset can be adjusted. (Refer to the AFG.pdf for more details on usage of this instrument.
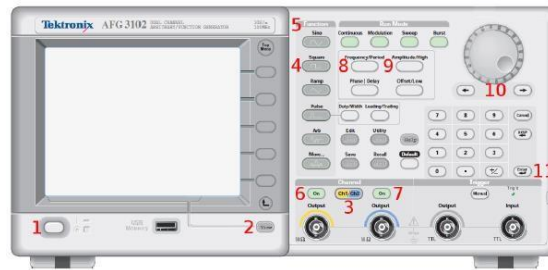

Fig. 2.1 Arbitrary Function Generator Panel

C.4 CRO Probes

CRO probes are special cables with a built-in RC/LC network to compensate for the input capacitance of the CRO. Most CRO probes have 1X and 10X modes. The 1X mode would display the signal as it is, whereas the 10X mode would attenuate the input signal by 10. DSO channel input parameters can be adjusted for the probe setting chosen.

For high frequency signals, and also for measuring fast rising waveforms, the 10X mode should be used. It is best not to use 1X except when the input signal is < 20 mV.

CRO probe (10X mode) and the DSO input equivalent circuit
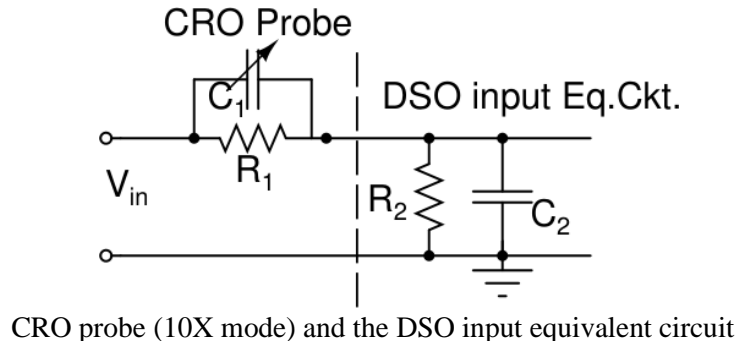
See the CRO schematic diagram shown. The input equivalent circuit of a CRO/DSO inputs is 1 MΩ (say R_2) in parallel with a capacitor (say C_2). CRO probe is essentially a 'compensated attenuator' used for compensating the CRO/DSO input capacitance. In the 10X mode the probe introduces a 9 MΩ resistance (say R_1) in series with the signal. The probe has an adjustable Capacitor (say C_1) connected parallel to R_1. If the input resistance of the DSO is $R_2 (= 1$ MΩ) and the input capacitance is C_2, then the signal at the DSO input (i.e. appearing across R_2) would be $(V_{in} R_2/(R_1+R_2))$, when $R_1 C_1 = R_2 C_2$. In order to achieve this condition, the capacitor C_1 in the CRO probe is made adjustable. DSO front panels are provided with a 1 kHz Probe CAL Pulse signal.

> Note: Students often mistake the CRO probe to a useful BNC cable, and are often tempted to use it in the 1X mode, and treat it like a BNC cable. This is a wrong practice as in the 1X mode the probe introduces an LC circuit in series with the signal (to partially compensate for C_2) which distorts the signal.