Poznan University of Technology
Faculty of Computing and Telecommunications
Institute of Multimedia Telecommunications

COMPUTER AIDED DESIGN

LABORATORY

Instruction for the laboratory exercise

LTspice: Design and apply analog filters

drinż. Michał Maćkowski (Ph.D.)
dr inż.SławomirMichalak (Ph.D.)

1. The aim of the exercise

- Learning how to design and apply analog filters,
- Attach to LTspice signals from external files.

2. Working with WAV files in LTspice

a) Saving signal as a WAV file

1. To save a waveform from a simulation as a WAV file, add a spice directive to the simulation by menu *Edit > Spice directive...,* then in the edit box enter:
   .wave "filename" nbits, sample_rate V(node_name)
   Where nibts is usually 16 bits and the sample rate is 8000, 16000, or 44100.
   An example might be: **.wave "my_file.wav" 16 16000 V(OUT)**

2. Create sin+noise signal according to Fig.2. Save this sample noise signal as wav file.

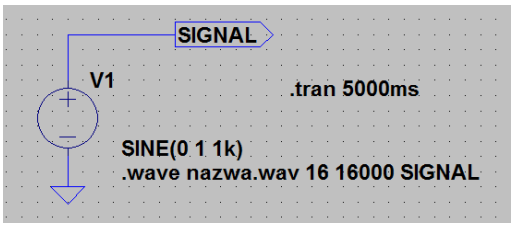

Fig. 1. An example of saving wav file.

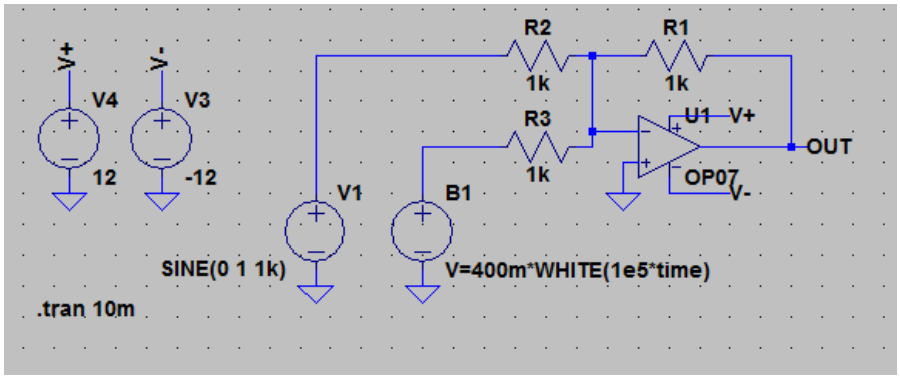

Fig. 2. Adding noise to signal (use BV voltage).

b) Adding WAV file to the simulation

1. Create the circuit diagram as shown in Figure 3.
2. To add a WAV file input to a LTspice simulation, right click a voltage source, then in the DC value field enter:
   wavefile="file_name.wav" chan=0

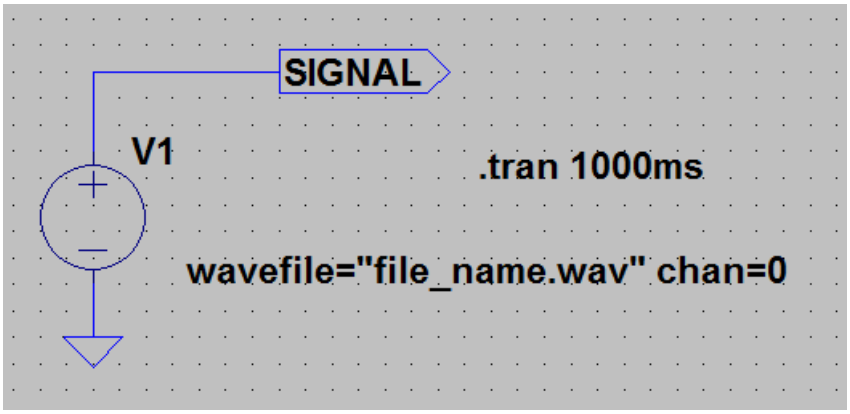

Fig. 3. Diagram of the tested circuit.

3. Run simulation and observe the signal.

3. Design and use filter to remove noise

Open the online filter design page: **https://tools.analog.com/en/filterwizard/**

a) Low-pass filter

1. Design an active filter with the following parameters:
   - low-pass,
   - gain: 1 V/V,
   - cut-off frequency: 2500 Hz,
   - filter order: 4.
   - filter response: Bessel
   - select multiple-feedback (MFB) circuit implementation
     Design the filter, assuming the values of E96 series resistors (1%) and capacitors from the E24 series (5%)
2. Perform frequency analysis of the tested filter (AC Sweep analysis).
3. Compare the determined characteristics with the design assumptions.

b) High-pass filter

1. Design an active filter with the following parameters:
   - high-pass,
   - gain: 1 V/V,
   - cut-off frequency: 500 Hz,
   - filter order: 4.
   - filter response: Bessel
   - select multiple-feedback (MFB) circuit implementation
     Design the filter, assuming the values of E96 series resistors (1%) and capacitors from the E24 series (5%)
2. Perform frequency analysis of the tested filter (AC Sweep analysis).
3. Compare the determined characteristics with the design assumptions.

c) Band-pass filter

1. Connect low-pass and high-pass band filter in series and crate band-pass filter.
2. Make frequency analysis of the tested filter (AC Sweep analysis).
3. Compare the determined characteristics with the design assumptions.

d) Filtering noise

1. Use sample WAV file as input source (bird voice).
2. Save the result of the simulated filter as a wav file and listen to it.
3. Compare original wav file and filtered wav file.

4. Tasks for students to do homework (obligatory)

1. Open the online filter design page: https://tools.analog.com/en/filterwizard/
2. Design filter as one band pass filter (passband 1k, stopband 5k, center frequency 1kHz, 6th order Butterworth).
3. Design the filter, assuming the values of E96 series resistors (5%) and capacitors from the E24 series (10%)
4. Repeat noise removing from sample wave file as in point 3.d.
5. Create BOM of your filter.

5. Additional tasks

- Design two independent filters: low-pass (1kHz) and high-pass (5kHz)
- Record sample of your voice as wav file or you can use sample of your favourite song.
- Filter your sample file - first with the low-pass filter, and then with the high-pass filter (save both output samples). Listen and compare both audio files.

6. Report

Should contain:

- all schemes of simulated systems,
- simulation results,
- answers to the questions contained in the manual,
- conclusions.