PoznaŒUniversity of Technology
Faculty of Computing and Telecommunications
Institute of Multimedia Telecommunications

COMPUTER AIDED DESIGN

LABORATORY

Instruction for the laboratory exercise

LTspice: Time/Transient Analysis and FFT and Fourier

dr inż. Michał Maćkowski (Ph.D.)
dr inż. Sławomir Michalak (Ph.D.)

1. The aim of the exercise

- learn about the possibilities of time analysis/transients *Transient*,
- determination of *FFT* and *Fourier* analyzes,
- creating new elements - transformer.

2. The course of the exercise

a) Cyclic charging of the capacitor

1. Create a circuit diagram as shown in Fig. 1. Use shortcut keys:
   - R – inserts a resistor,
   - C – inserts a capacitor,
   - G – inserts ground symbol,
   - F2 – open a list of elements,
   - F3 – turns on the connection drawing mode,
   - F4 – inserts a label (in/out).

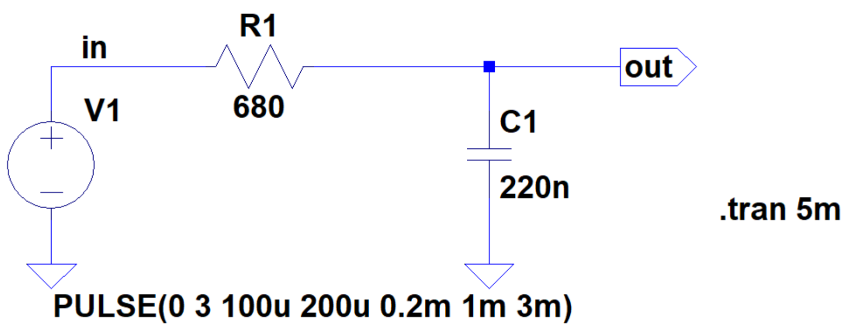

Fig. 1. Diagram of the tested RC circuit.

2. Set the values of elements according to the drawing. Set the voltage source value for the selected *PULSE* function, which is visible in the extended mode (*Advanced* button in the source settings).
3. Start the simulation command editor, in the *Simulate* menu select *Edit Simulation Command*, select the *Transient* tab, in the *Stop time* field, set the observation time to 5ms.
4. Run the analysis, analyze the output voltage V_{out} and input voltage V_{in}.
5. Check the effect of a capacitance change of ±25% on the output voltage.

b) RLC circuit

1. Create a capacitor charging diagram in the RLC circuit (Fig. 2)

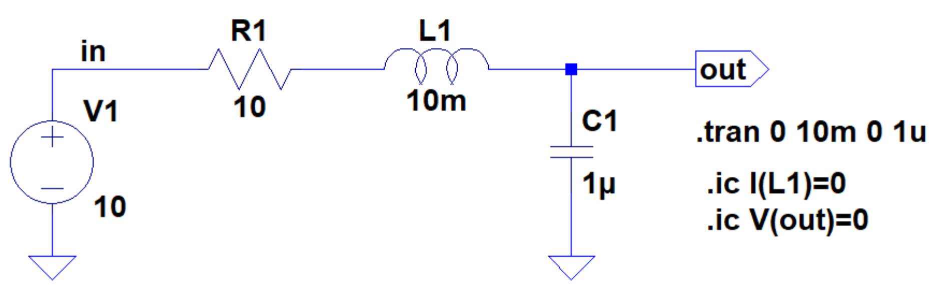

Fig. 2. Charging a capacitor in an RLC circuit.

2. Set the values of the elements as shown in Figure 2. Set the value of the voltage source 10 in the *DC value [V]* field.
3. Using the *.IC (Set Initial Conditions)* commands, set the initial values of the current through coil *I (L1)* and the voltage across the capacitor *V (out)* to zero.
4. Set the parameters of the *Transient* analysis, in the *Stop time* field set the observation time to 10ms, and in the field *Maximum Timestep* set the value 1μs.
5. Run the analysis, observe the waveform of the output voltage *V(out)* and the current flowing through the coil *I(L1).* Display the current and voltage on separate graphs (In the plot window, in the *Plot Settings* menu or after right-click mouse chose the *Add Plot Pane* option).
6. Check the influence of changes the resistance R1 value (take the values: 10, 50 and 100 Ω) on the shape of the waveforms.
7. Modify the circuit according to Figure 3 illustrating capacitor discharge in the RLC system.

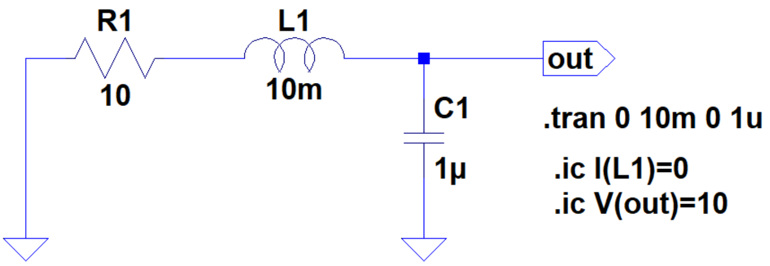

Fig. 3. Discharge of a capacitor in an RLC circuit.

8. Repeat analyzes in the same way as in the capacitor charging circuit (2-6).

c) Analog adder circuit

1. Create a voltage adder circuit according to Figure 4.

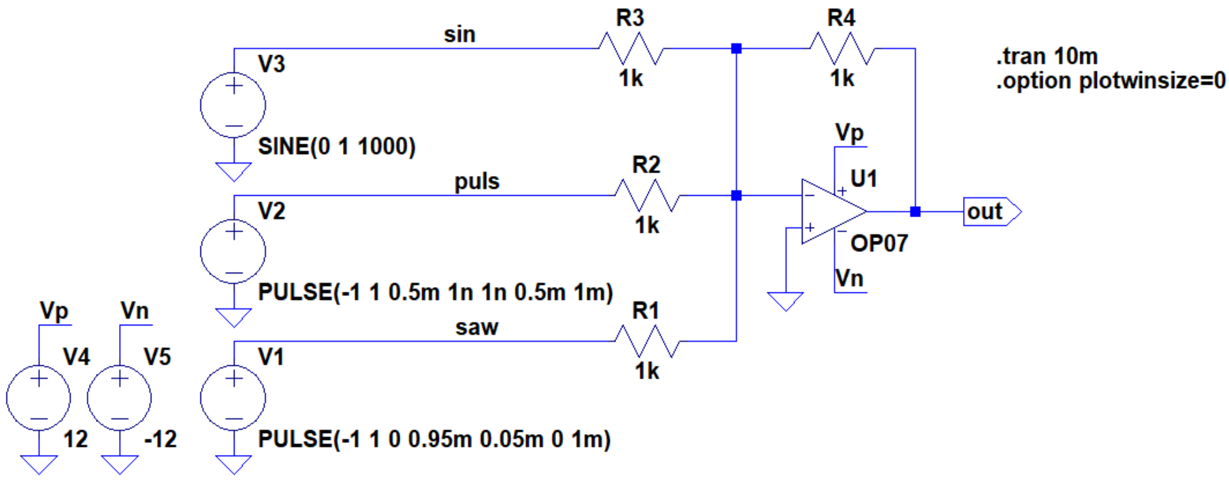

Fig. 4. Voltage adder.

2. Set the values of the elements according to Figure 4.
3. Use the command *.option plotwinsize=0* to disable output data compression.
4. Plot the voltage waveforms: input *sin*, *pulse* and *saw* and output *out* on separate graphs (Fig. 5).

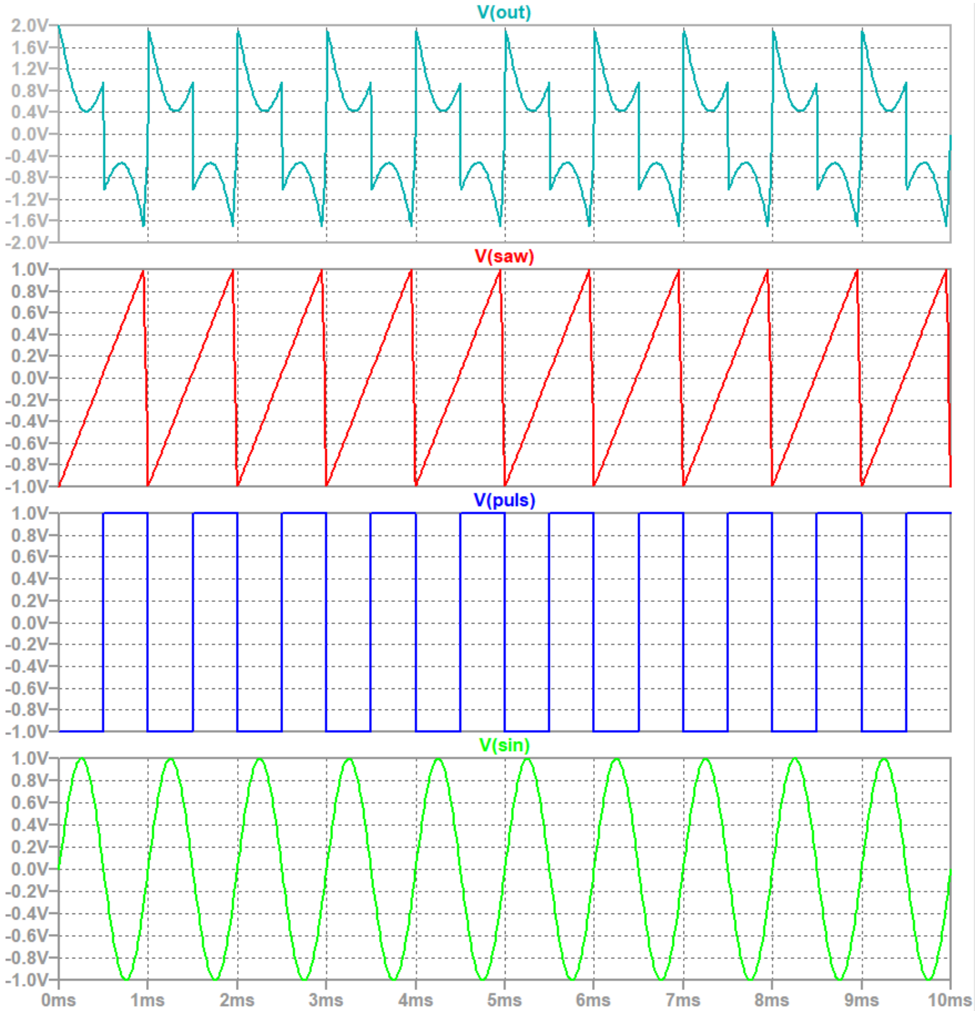

Fig. 5. Waveforms in the adder circuit.

5. View *FFT* analysis results for individual *sin*, *pulse*, *saw*, and *out* signals. To do this, select the *FFT* item in the *View* menu for the active plot window, or right-click on the plot window and select *View > FFT*. To make the FFT plots easier to read, choose a linear value representation (Y axis).
6. Compare the FFT results for the input signals with the output signal.
7. Use command *.four* to calculate the Fourier transform for input and output signals. Define the functions:

.four 1k 10 v(sin)
.four 1k 10 v(puls)
.four 1k 10 v(saw)
.four 1k 10 v(out)

8. The results of Fourier transform calculations are available in the **SPICE Error log** file, compare them with the FFT graphs of individual *sin*, *puls*, *saw* and *out* signals.

d) Analysis of the rectifier circuit with the Graetz bridge.

1. Create a new folder (give a name **CAD**) in your computer e.g.
   C: \ Users \...\ Documents \ LTspiceXVII directory.
2. In the LTspiceXVII program settings: *Tools/Control Panel/Sym. & Lib. Search Path (*)* add new paths for saving symbols and libraries:
   C: \ Users \ ... \ Documents \ LTspiceXVII \ CAD
3. Draw a diagram of the transformer as shown in Figure 6. *InV+* and *InV-* are labels set as inputs (*Port Type: Intput*), and *OutV+* and *OutV-* are labels set as outputs (*Port Type: Output*). Enter the coupling of coils **K1 L1 L2 1** as the Spice directive.

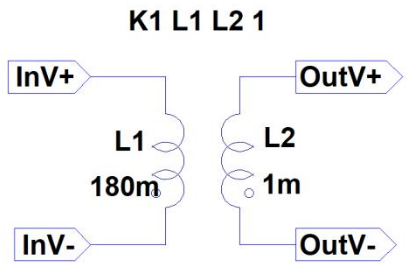

Fig. 6. Diagram of a transformer.

4. Set the series resistance of the coils L1 to 100mΩ and L2 to 10mΩ, respectively.
5. Save the transformer diagram in:
   C: \ Users \ student \ Documents \ LTspiceXVII \ CAD naming it *transformer.asc*.
6. From the *Hierarchy* menu, choose *Create a New Symbol*.
7. In the symbol editor, draw a rectangle (menu *Draw*) and insert the output pins *InV+* and *InV-* and output pins *OutV+* and *OutV-* (menu *Edit > Add Pin/Port* or the p key) (Fig. 7).

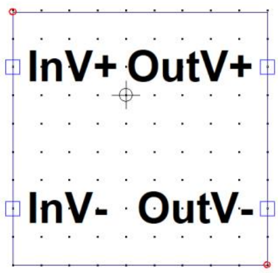

Fig. 7. Editing the transformer symbol.

8. Save the created symbol in:
   C: \ Users \ student \ Documents \ LTspiceXVII \ CAD
   directory and name it *transformer.asy*.
9. Prepare the rectifier diagram as shown in Figure 8 and save it in:
   C: \ Users \ student \ Documents \ LTspiceXVII \ CAD
   folder before starting the simulation.

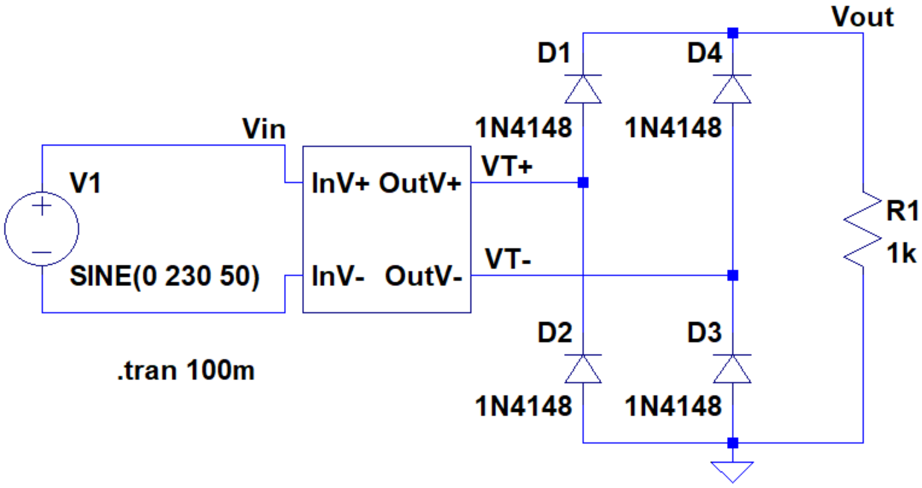

Fig. 8. Rectifier with Graetz bridge.

10. Run the simulation. Display voltages: input *Vin*, at transformer output (differential) *VT+* - *VT-* and output voltage at load *Vout*.
11. Connect a 1μF filter capacitor (parallel to R1). Run the simulation.
12. Investigate the effect of capacitance, take the values 1, 10 and 100 μF.
13. If you work on a computer in the university laboratory - after completing the exercises, clear the contents of the folder: *C: \ Users \ student \ Documents \ LTspiceXVII \ CAD*

e) Wien Bridge Oscillator

1. Create a Wien bridge diagram circuit as shown in Figure 9.

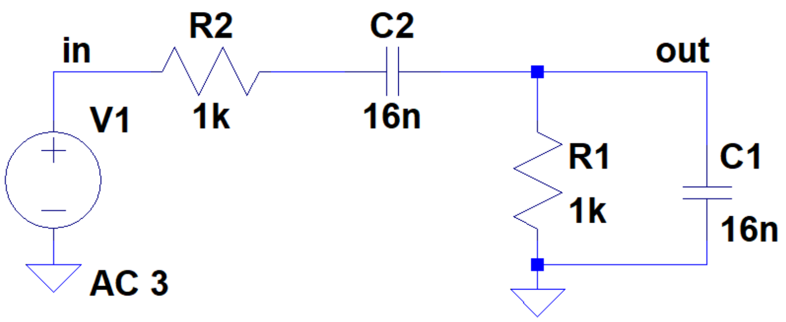

Fig. 9. Wien bridge circuit.

1. Using the *AC* frequency analysis, find the value of the frequency f_0 for which the phase shift between the output and the input is 0. Find the attenuation of the *Vout/Vin* bridge for this frequency.
2. Draw the Wien bridge generator circuit according to Figure 10.

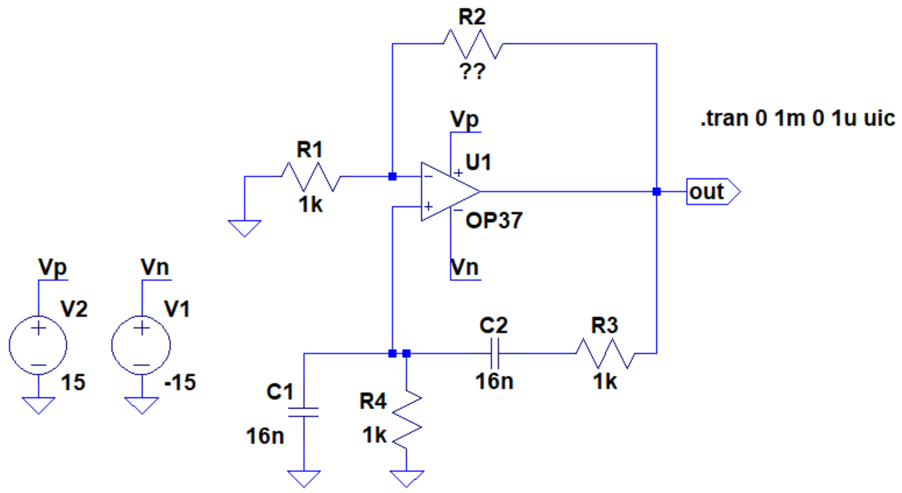

Fig. 10. Wien bridge oscillator.

3. Based on the results from point 2 and the oscillation conditions, determine the resistance value of the resistor R2.
4. In the *Transient* settings, select *Skip initial operating Point solution*. Run the simulation.
5. View the results of the FFT analysis.
6. Make simulation for changes in the resistance R2 value by ± 10%.
7. Modify the generator circuit as shown in Figure 11.

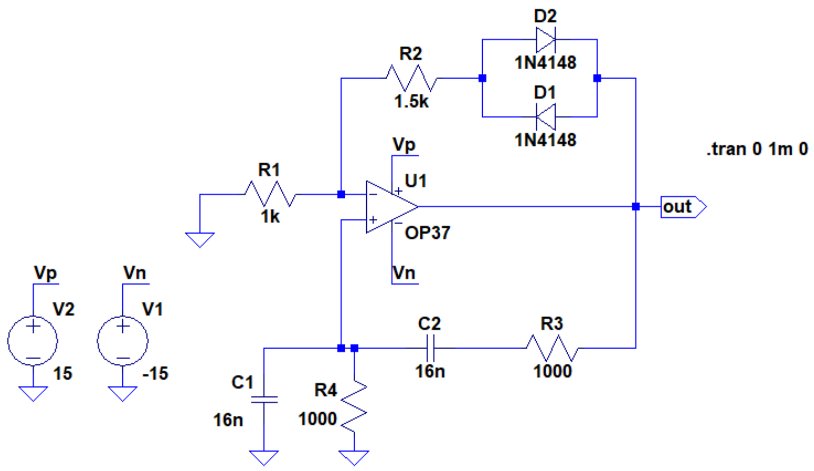

Fig. 11. Wien bridge oscillator (sine wave generator) with nonlinear elements (AGC).

8. View the results of the FFT analysis.
9. Perform Fourier analysis for the output signal.
10. Check the influence of changes in the resistance *R2* value on the shape of the output signal.

3. Tasks for students to do homework (obligatory)

- Create a new element (subscheme) - Wien's bridge (similar to the transformer) and use it in the previous generator circuit (Fig.11)

4. Additional tasks

- Simulate the operation of the Wien bridge oscillator (sine wave generator) with AGC (Fig. 11) in Micro-Cup 12 program (make perform the necessary AC and Transient simulation).

You can see also: https://www.youtube.com/watch?v=9Ur_JvUzcRU

5. Report

Should contain:

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