



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

COMPUTER AIDED DESIGN
LABORATORY

Instruction for the laboratory exercise

LTspice: Adding new parts to libraries

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

1. The aim of the exercise

- learning how to add new elements to element libraries in LTspice.

2. Adding new items to LTspice

Method 1. Adding a model to the standard library

Addition of bipolar transistors to the standard LTspice library

1. Make a copy of the **C:\Users\Student\Documents\LTspiceXVII\lib** directory (the name of the path could be different in your computer)
2. Find the Spice models of bipolar transistors BC107A, BC211 and BC313 in text format on the Internet and copy them.
3. Open the **standard.bjt** library of bipolar transistors in a text editor, eg *Notepad ++*:
C:\Users\student\Documents\LTspiceXVII\lib\cmp\standard.bjt.
4. Paste the transistor models at the end of the transistor list.
5. Save the modified file.
6. Start LTspice.
7. Draw the circuit diagram from Figure 1.
8. Right-click an NPN transistor, click the *Pick New Transistor* button and select **BC107A** from the displayed list.
9. Run the simulation, check if the system works properly.

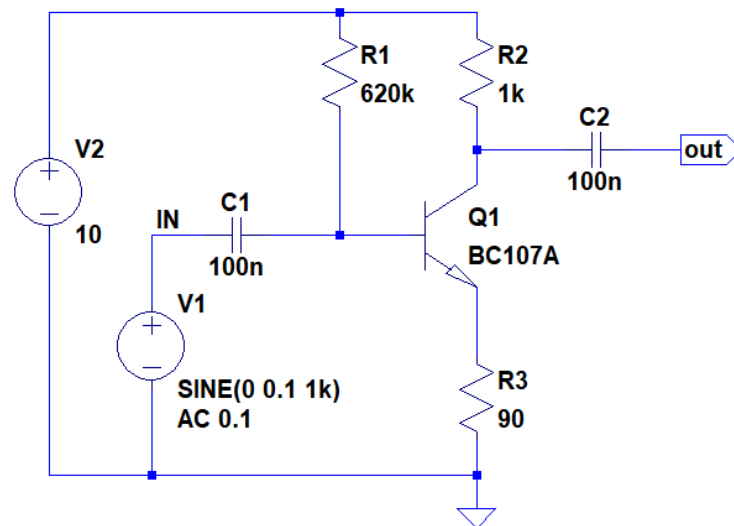


Fig. 1. Schema of a transistor amplifier.

10. Determine the gain and passband (3dB) of the amplifier.
11. Check the influence of the resistor R3 ($\pm 50\%$) on the operation of the circuit.
12. Check the effect of the capacitor C1 on the operation of the system (take 47n, 100n and 220n).
13. Replace the BC107A transistor with the **BC211**. How did this exchange of transistors affect the work of the system?

14. Draw the power amplifier circuit according to the diagram in Figure 2.

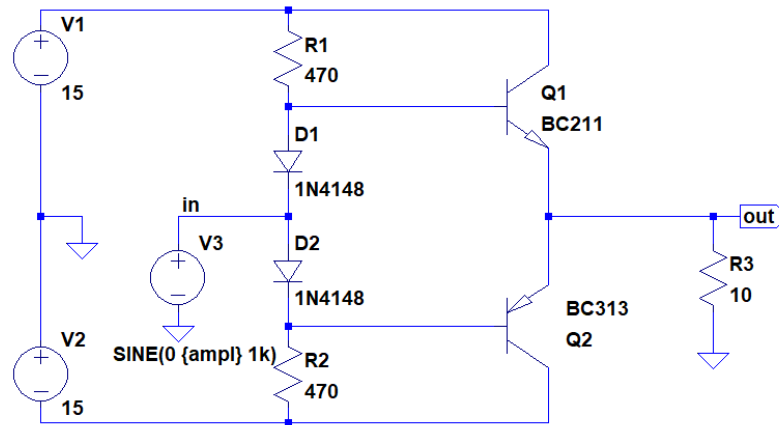


Fig 2. Schematic diagram of a power amplifier with complementary transistors.

15. Run the simulation, check if the amplifier works properly.

16. Determine the maximum value of the input voltage for which the output signal is not distorted. For example, use the FFT analysis.

Addition of the BF245C field effect transistor (njf) to standard LTspice library

1. Find the Spice model of the BF245C on the Internet and copy it.
2. Open the **standard.jft** FET library file in a text editor, eg *Notepad ++*:
C:\Users\student\Documents\LTspiceXVII\lib\cmp\standard.jft.
3. Paste the BF245C model at the end of the transistor list.
4. Save the modified file.
5. Draw the amplifier circuit according to Figure 3.

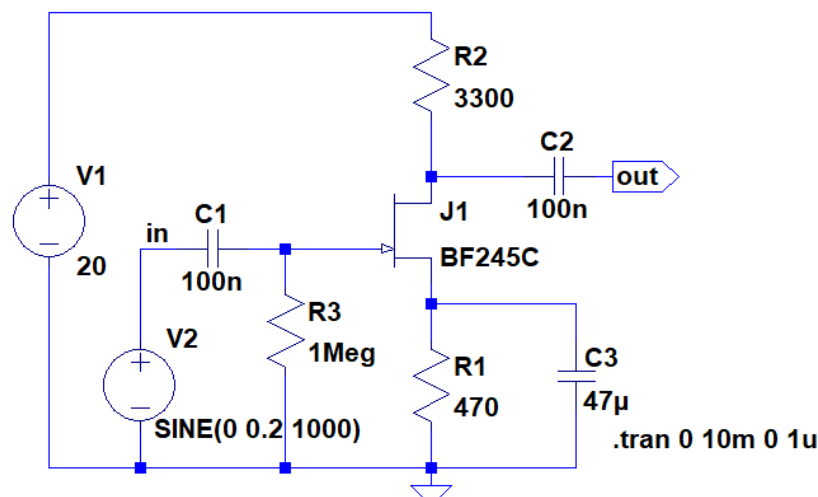


Fig. 3. Schematic diagram of an amplifier with a field effect transistor.

6. Run the simulation, check if the system works properly.
7. Determine the gain and passband (3dB) of the amplifier.
8. Determine the input voltage range for which the output signal is not distorted. For example, use the FFT analysis.

Method 2. Adding a library file, editing a symbol

Addition of the thyristor model 2N5171

1. Find a file with thyristor models **thyristor.lib** on the Internet. Save it (text file) in the directory:
2. *C:\Users\student\Documents\LTspiceXVII\lib\sub*
3. Give the name **thyristor.lib**.
4. Start LTspice and open the SCR.asy symbol (via menu **File > Open**) from the directory:
C:\Users\student\Documents\LTspiceXVII\lib\sym\Misc
5. Open the window for editing the attributes of the *Edit > Attributes > Edit Attributes* menu.
6. Set the attributes as shown in the Figure 4.

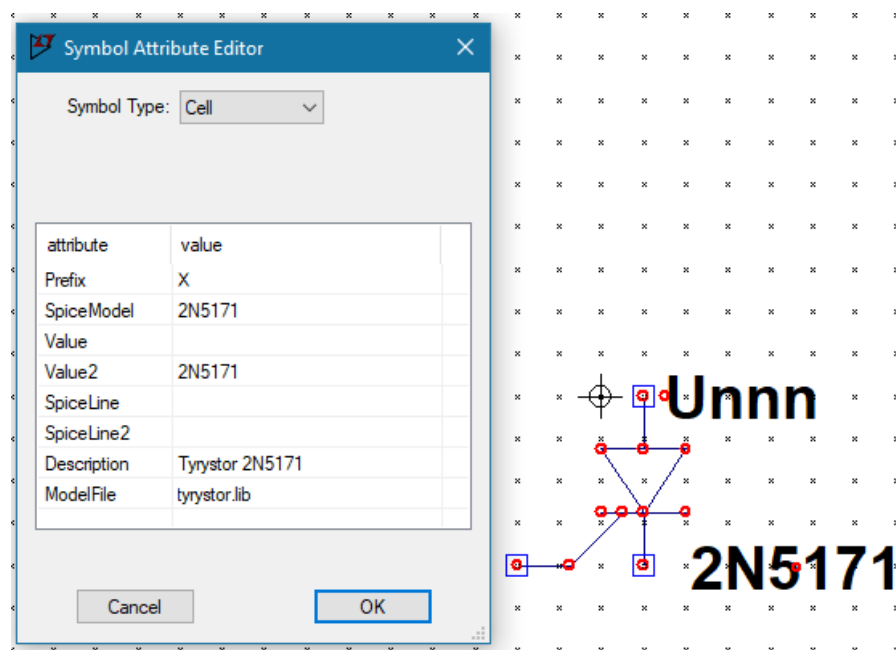


Fig. 4. Editing thyristor attributes for 2N5171.

7. In the *Attribute Window* (eg menu *Edit > Attributes > Attribute Window*), define which information about an element will be displayed in the circuit after its symbol is placed.
8. Using the **Save As** option, save the newly defined element under the name *2N5171.asy*, in the directory:
C:\Users\student\Documents\LTspiceXVII\lib\sym\Misc
9. Draw the circuit diagram in Figure 5.
10. Run the simulation, check if the circuit works properly.

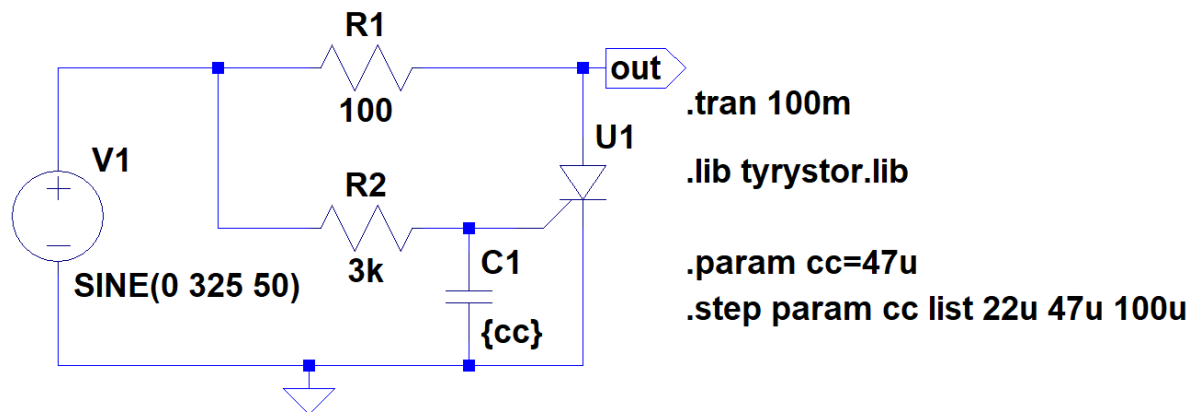


Fig. 5. Diagram of a thyristor circuit.

Method 3. Adding operational amplifiers

Texas Instruments OPA388 amplifier added

1. Visit **ti.com** to find information about the OPA388 amplifier. Download its model.
2. From the downloaded archive, save the **OPA388.LIB** library file of the OPA388 amplifier in the directory: *C:\Users\student\Documents\LTspiceXVII\lib\sub*
3. Start LTspice, open the OPA388.LIB library (via the **File > Open** menu).
4. Find the line:
.subckt OPA388 IN + IN- VCC VEE OUT
5. Right-click on the .subckt (in this line) and choose *Create Symbol*.
6. Save the **OPA388.asy** file in the directory:
C:\Users\student\Documents\LTspiceXVII\lib\sym\Opamps.
7. Draw the circuit diagram from Figure 6.

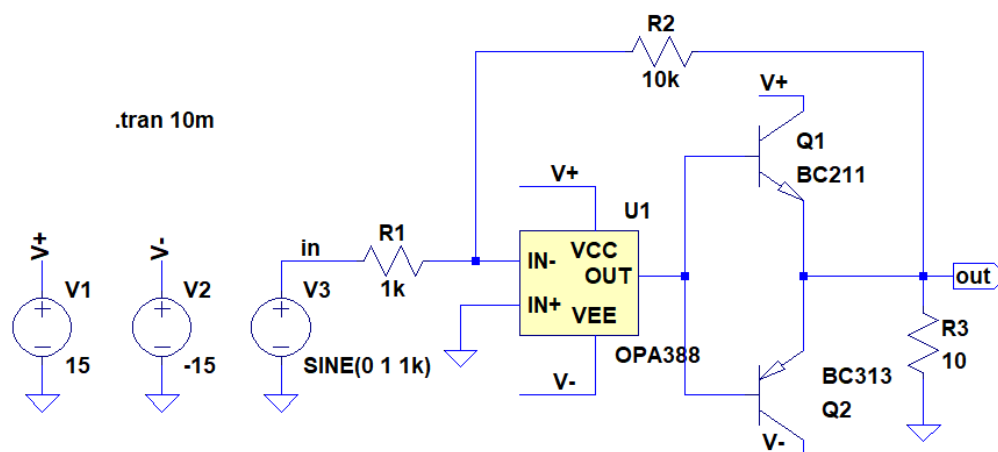


Fig. 6. Schematic diagram of the circuit with an operational amplifier.

8. Run the simulation, check if the system works properly.
9. Determine the gain and passband (3dB) of the amplifier.

3. Tasks for students to do homework (obligatory)

1. Edit the symbol *opamp2.asy*, assign the OPA388 amplifier model to it and save it under the name **OPA388s.asy** (see example with thyristor). Note the pin labels in the library and symbol.
2. Suggest an example circuit showing the operation of the OPA388 element (make simulation).
3. Find the Zener diode model: **BZX55C2V7** on the website of the electronic components manufacturer *www.vishay.com*. Add its model to the program and demonstrate how it works (make simulation).

4. Additional tasks

- Add **EPC2016C** FET transistor model to Micro-Cap 12 library.
- Draw the $I_D = f(V_{GS})$ characteristic (*Transfer Characteristics*), compare result with data sheet (see Figure 2 in data sheet):

https://epc-co.com/epc/Portals/0/epc/documents/datasheets/EPC2016C_datasheet.pdf

- Simulate the an example amplifier circuit (just like in Fig.3) with your EPC2016C device.

See an example: <https://www.youtube.com/watch?v=35DvfD-RAJ0>

5. Report

Should contain:

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