

Laboratory 3. Building an electric schematic using OrCAD

Aim of the laboratory

Getting accustomed with the steps used in editing an electric circuit.

Necessary equipment

Work stations with OrCAD installed

Launch OrCAD from the menu Windows-> Start ->Program Files ->Orcad 9.2 ->Capture CIS. In the dialog window that appears fill in the name of the new projet, the type and give a path to where you want to save it. (create a folder with a distinctive name like STUDENT where you will save all your work during this semester)

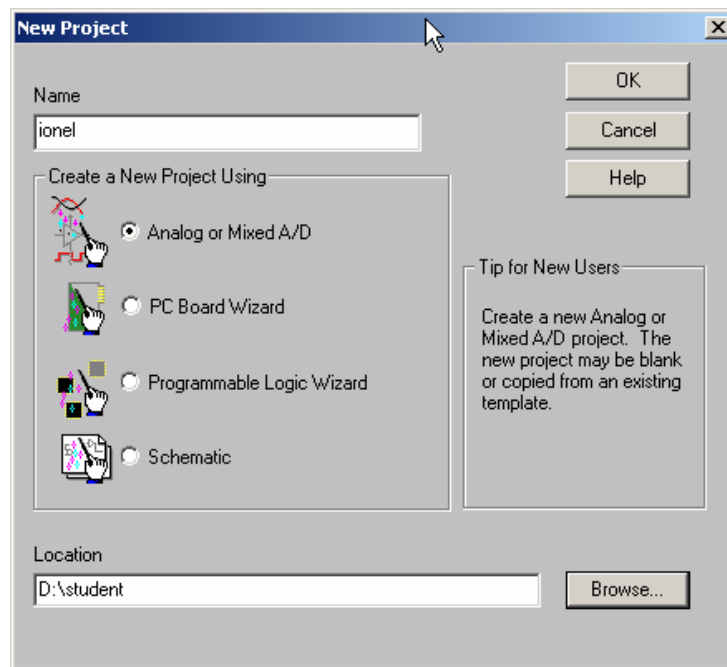


Fig1

Before starting, you must know the schematic that you want to built. For this laboratory you have bulid this circuit's schematic :

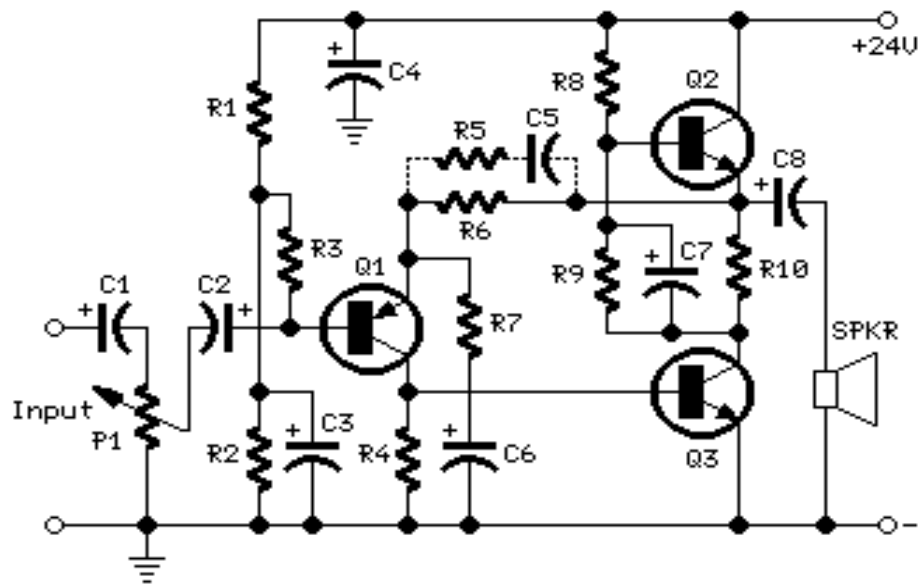


Fig2.

P1 - 47K,
 R1 - 100K, R2 - 12K, R3 - 47K,
 R4 - 8K2, R5 - 1K5, R6 - 2K7,
 R7,R9 - 100R, R8 - 560R, R10 - 1R,
 C1,C2 - 10 μ F 63V Electrolytic Capacitors,
 C3 - 47 μ F 25V Electrolytic Capacitor,
 C4 - 100 μ F 35V Electrolytic Capacitor,
 C5 - 150nF 63V Polyester Capacitor,
 C6,C7 - 220 μ F 25V Electrolytic Capacitors,
 C8 - 1000 μ F 25V Electrolytic Capacitor,
 Q1 - BC560C 45V 100mA Low noise High gain PNP Transistor,
 Q2,Q3 - BD439 60V 4A NPN Transistors.

You will create a new project like in the following:

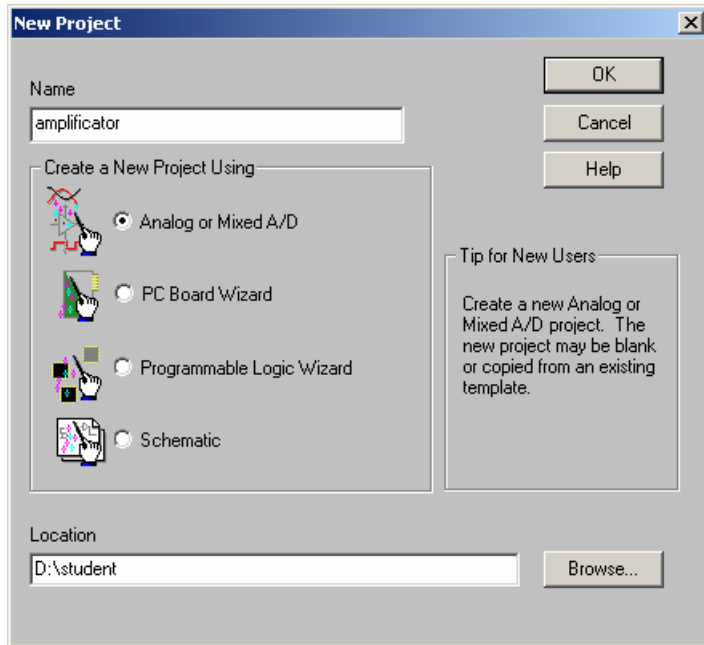


Fig3

As you can see, the name of the project is very suggestive and will help us find the project easily any time we want. After clicking OK the OrCAD work window will appear.

Before drawing the circuit we must know all its components and align them on the workspace. Just looking at Figure.2 we have:

- 10 resistors;
- 1 potentiometer;
- 7 electrolytic capacitors;
- 1 polyester capacitor;
- 1 transistor BC560C;
- 2 transistors BD439.

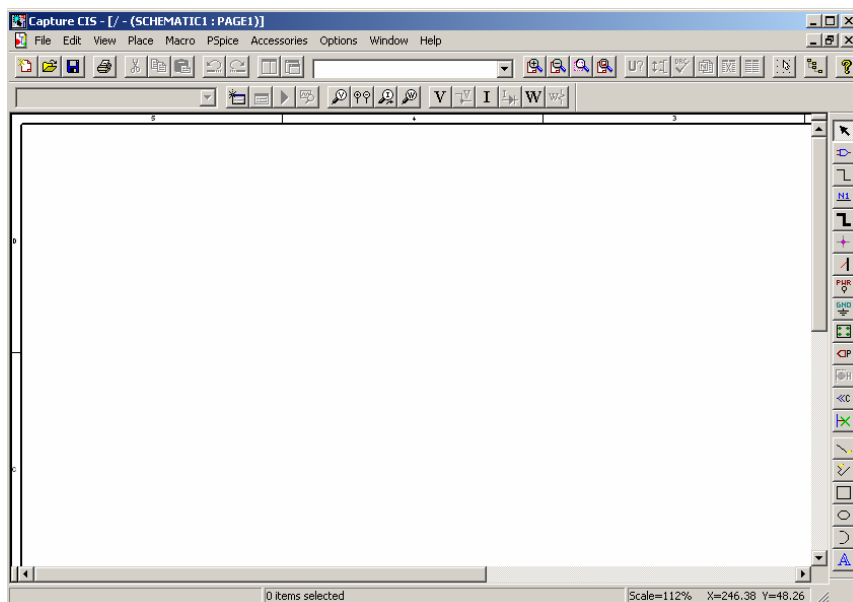


Fig4.

Those components will be selected from the menu Place->Part or by pressing the second button from the menu in the right (Figure.5).

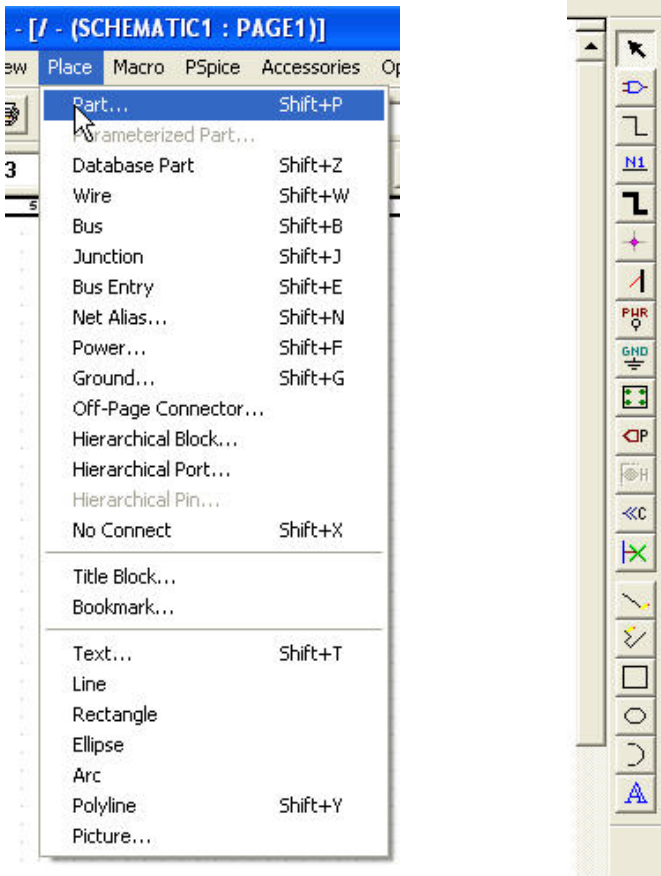


Fig5.

Next this window will pop-up:

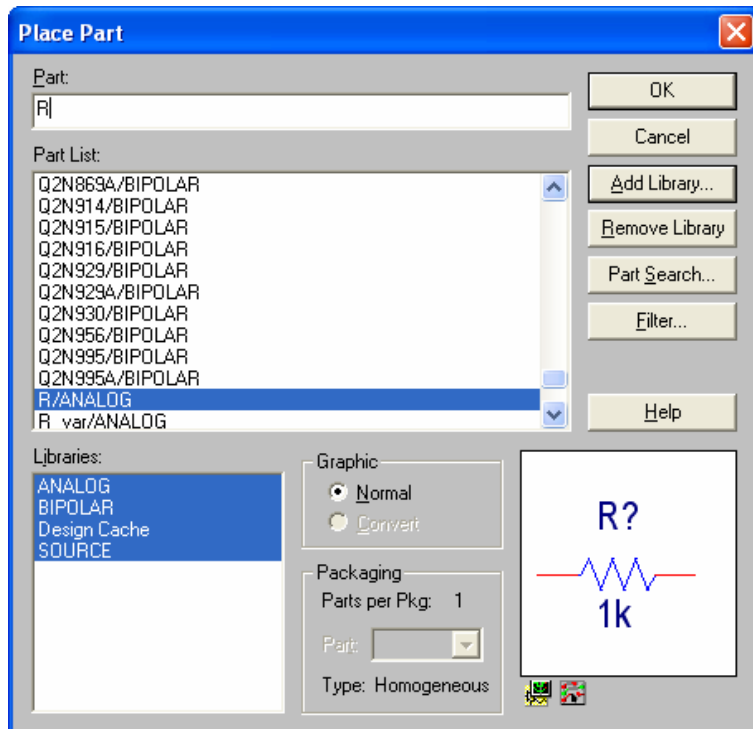


Fig6.

Choose the components and put them in the workspace (see figure 7). After that, we can modify every component's value. For example, R1 has 1K. Change the value to 100K by double clicking on the value of the resistance (see figure. 8).

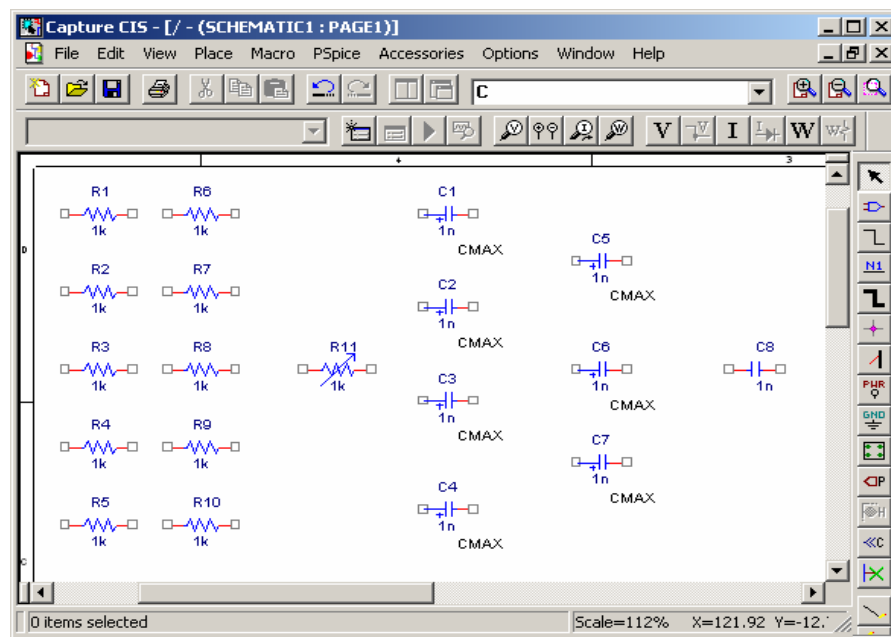


Fig7.

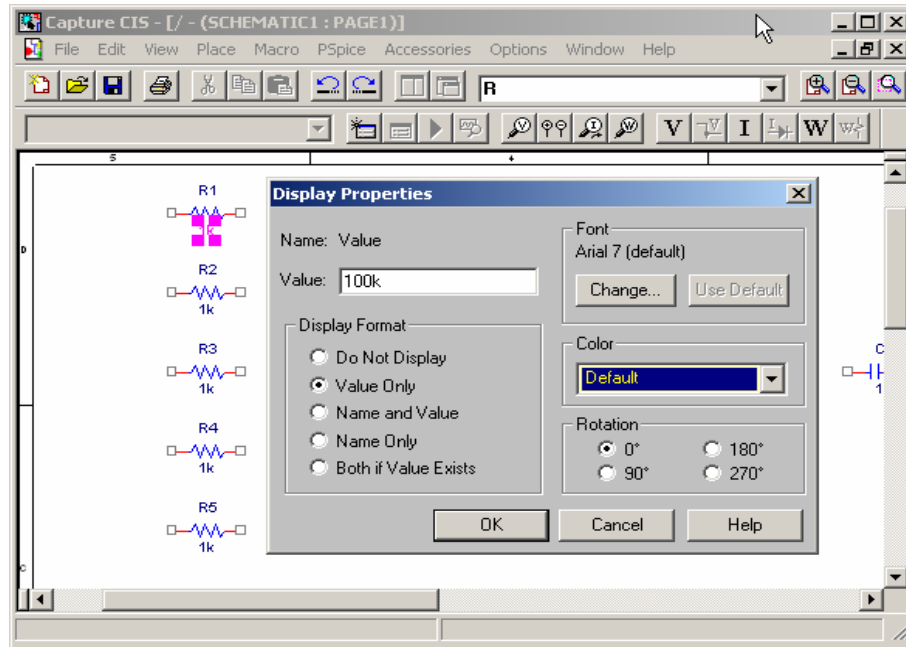


Fig8.

Arrange the circuit elements like in figure2 (in some cases we arrange circuit elements in the most optimal way).

Find the transistors using Part Search.



Fig9.

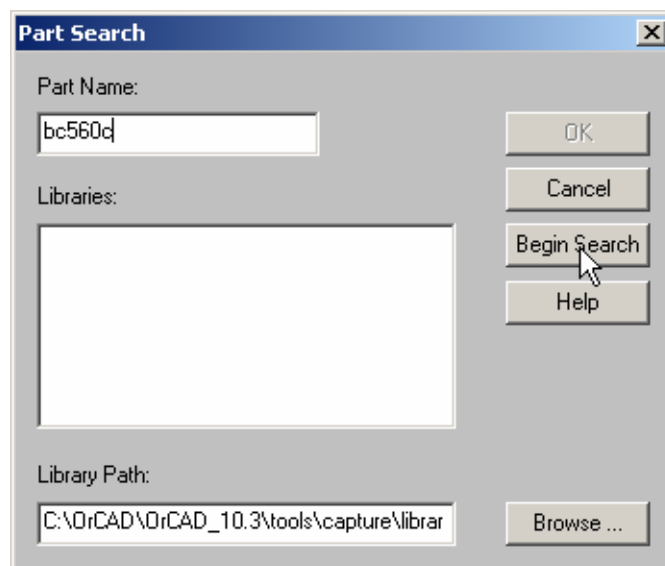


Fig10.

Next, draw the circuit lines between the elements using Place Wire button.

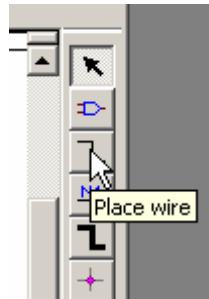


Fig11.

For signal income and outcome, we will use connectors. In this way, on the board won't be placed any loudspeaker, but we will connect the loudspeaker using two wires. The ground we will use is 0/SOURCE.

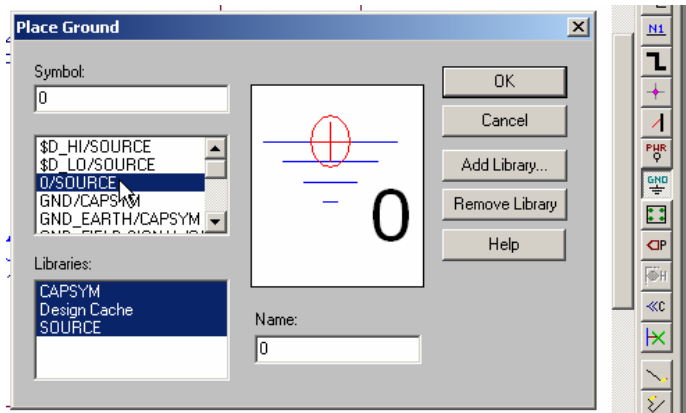


Fig12.

The outcome of this laboratory work is:

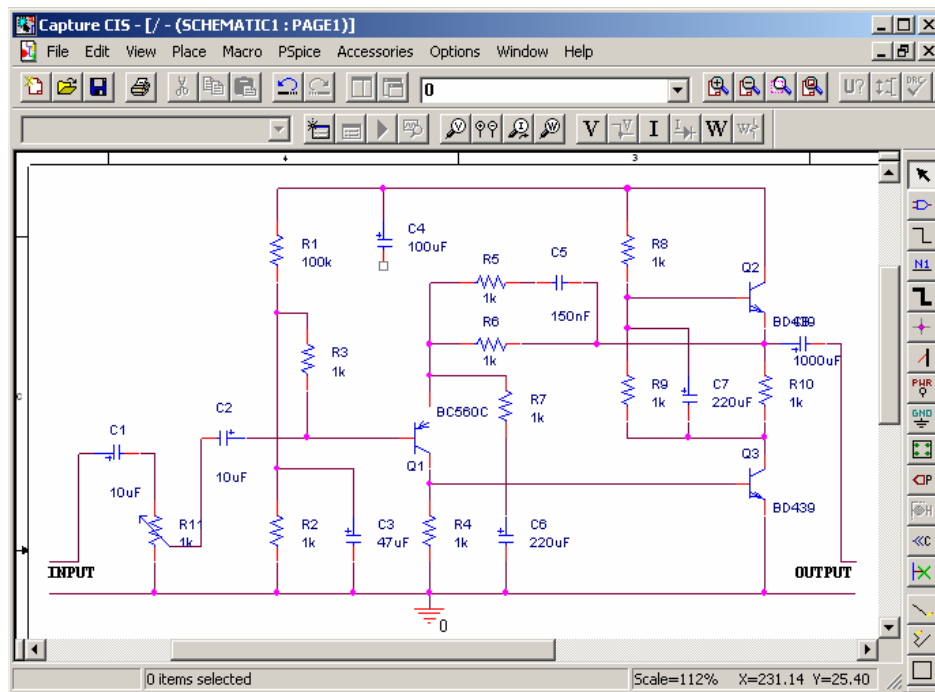
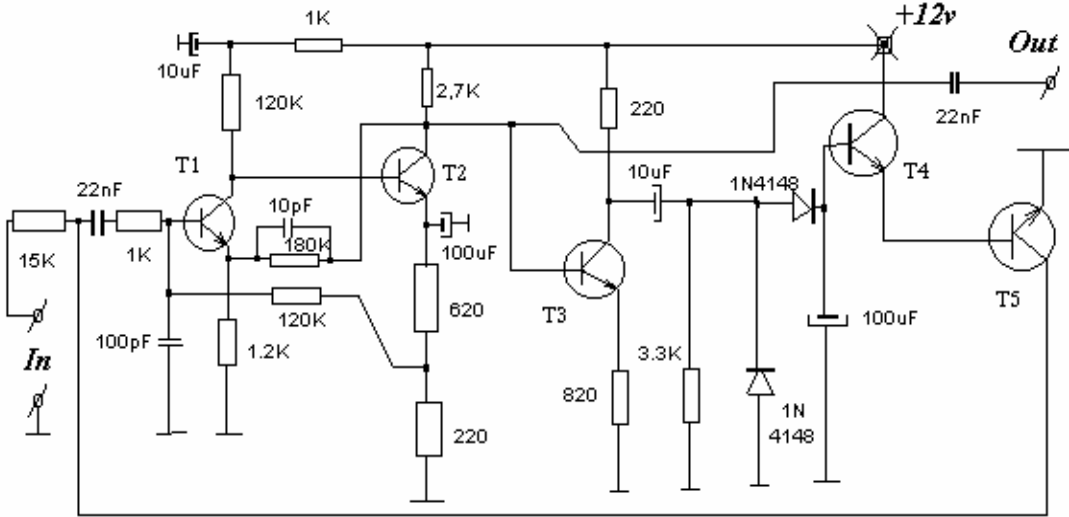


Fig13.

Homework:

Built the circuit using OrCAD:



The transistors are of type BC107.