



Basics of MicroCap

1. MicroCap

Based on the Electronics lectures the student learn the acquired knowledge in practice. For this the MicroCap simulation software will be used in the practical courses. The newest version of this software is available on: <http://www.spectrum-soft.com> webpage. In the classroom the MicroCap 9 demo version will be used. The software will be used for the simulation of analog circuits. The special circuit elements - which are needed during the classes – need to install from MC9_additions_install.zip file.

This guide is a basic description of the program – more information could be found in the “Help” menu in the software or on the manufacturer homepage (<http://www.spectrum-soft.com/download/rm.pdf>, <http://www.spectrum-soft.com/aboutus.shtm>, <http://www.spectrum-soft.com/download/ug.pdf>, http://www-cecs.evansville.edu/~hwang/f07-courses/ee210/microcap/microcap_transient_03.pdf)

2. User Interface

After the launching of the MicroCap 9 the following screen appears:

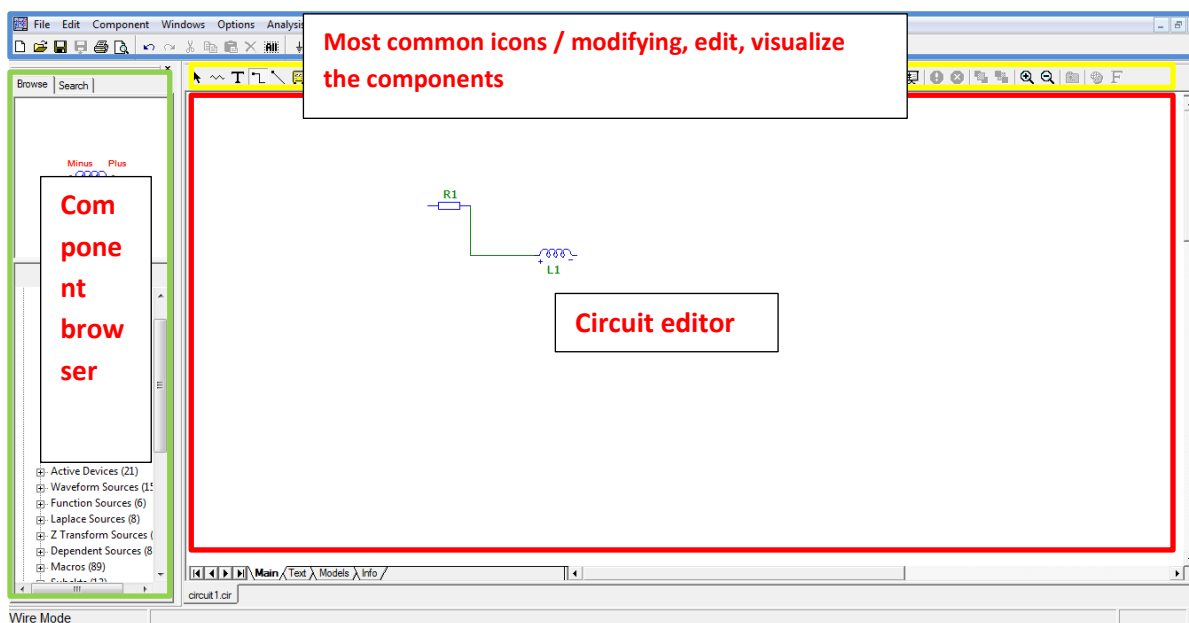


Figure 1.

3. New circuit design



To the simulation of a given circuit first we need to design the circuit in the Circuit editor surface with using different electronic objects. we referred to as object the different components, like R, L, C, power source, etc. During the simulation process the object are replaced by their own mathematical models. The parameters are changeable. The placed objects need to be connected each other – for this we used wires.

4. Placing components

The components are available from the Components menu which is shown in the left side of the graphical interface or the menu bar on the top of the page.

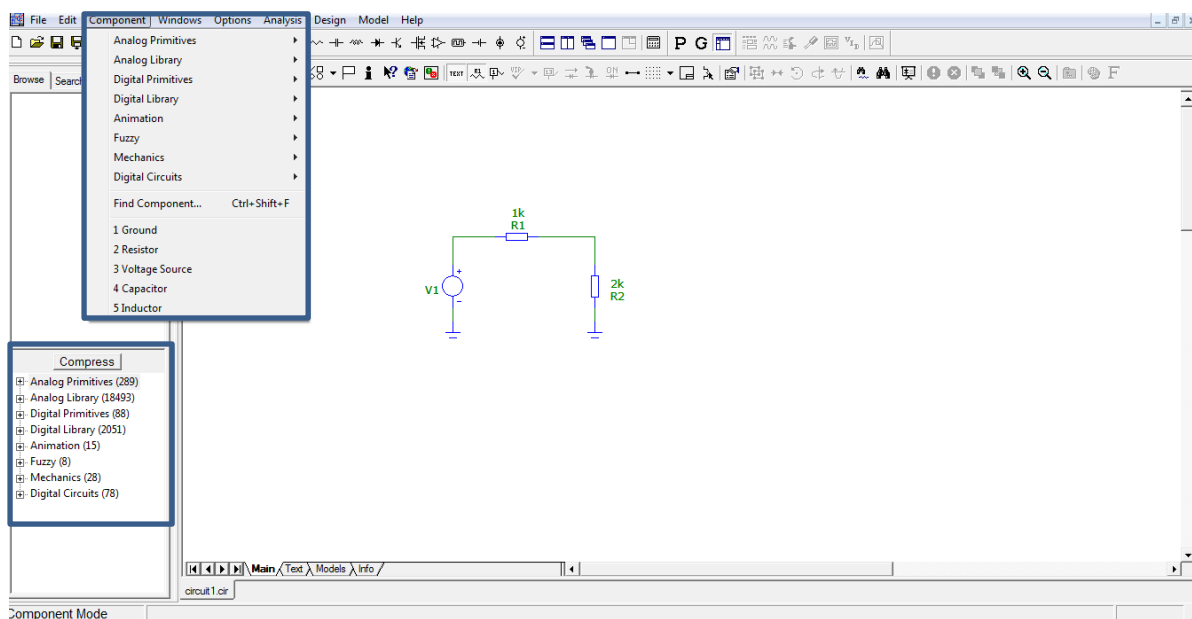


Figure 2.

The generally used components are placed under the menu bar (resistor, capacitor, inductor, ground, etc.).

5. Practical task

To learn the program usage the first practical task will be to realize and analyze a simple voltage divider.

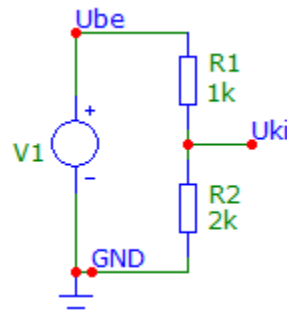


Figure 3.

$$U_{ki} (U_{out}) = U_{be} (U_{in}) \cdot \frac{R_2}{R_1 + R_2}$$

First we have to put down a voltage source (Component/Analog Primitives/Waveform Sources/Voltage Source). We have two ways to rotate the components: Left mouse button + Space or after the placing with Ctrl+R or Ctrl+L – if the component is selected with green marking.

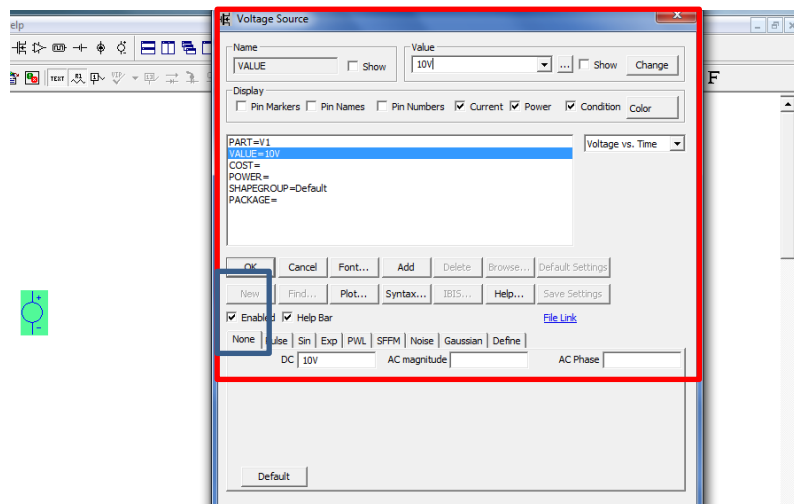


Figure 4.

The values of the Voltage source can be changed in the after the placing or after double clicking on the component in the property panel – this applies for the all components. Set the voltage to 10V.

Secondly, we have to put down two resistors. The values let the following: R1=1k (which means that the resistance of the component is 1 kiloOhm), R2=2k.

The software can handle the prefixes easily: m (milli), u (micro), n (nano), p (pico); k (kilo), Meg (mega), G (giga). For example, if we want to use a current source with 1 milliAmpere we can set this value as 1mA; resistor values could be: 1, 1m, 1u, 1k, 1Meg, etc.

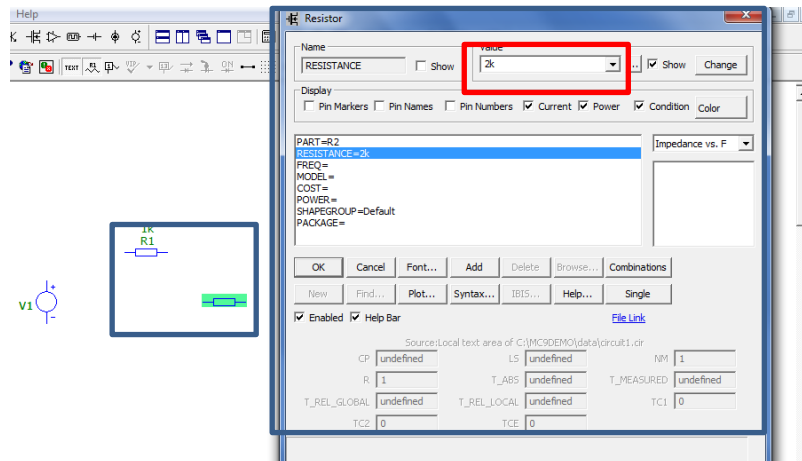


Figure 5.

A short summary to summarize the mentioned things: so with the objects/components we can simulate the electrical parts physical behavior during the simulation. The objects are described by mathematically. If we simulate a circuit the software translate the environment to differential equations and use numerical approximation to calculate the behaviors. The using of simple components is easy, we have to put down these and change the values or other aspects. **IF we using complex parts like Operational Amplifiers the parameters are given! Do not change these parameters after you chosen the appropriate OA (Fig. 6.).**

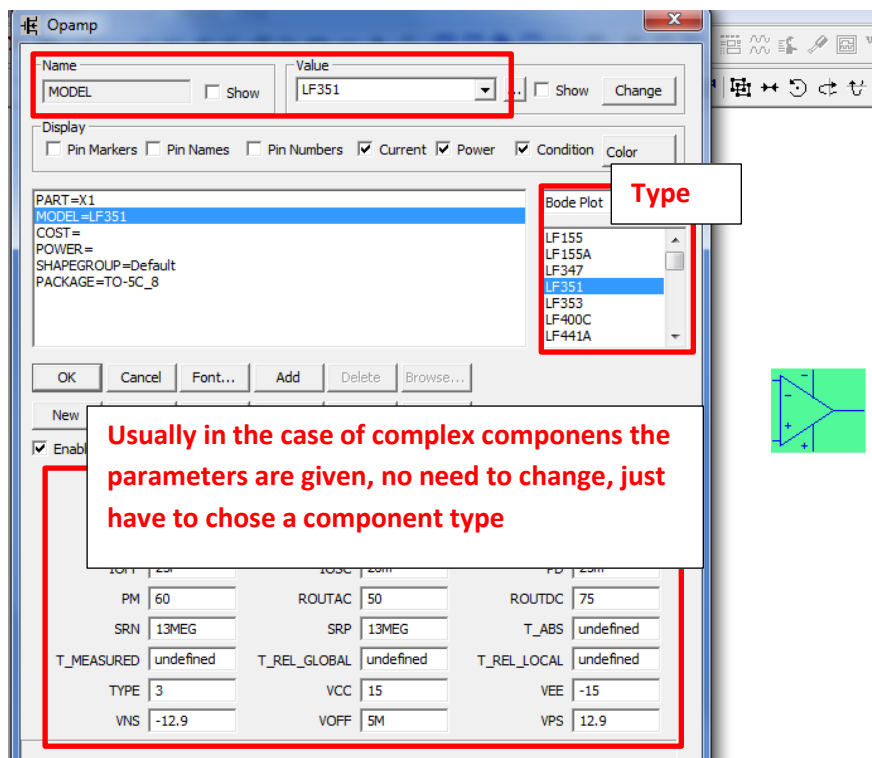


Figure 6.

After the placing of the resistors only one thing left: we need to put down two Grounds.

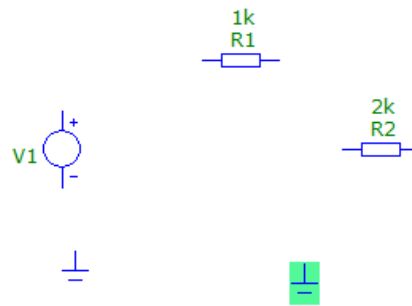



Figure 7.

To complete our circuit we have to connect the components with wires (Ctrl+W or  button).

The completed circuit can see on Fig. 8.

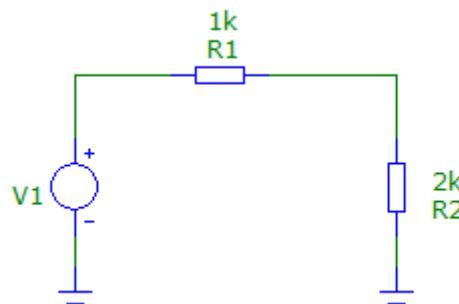


Figure 8.


6. Simulation

We have several options to analyze our circuits. The Transient simulation is give the time-domain signals so with this simulation we can analyze the changing of the signals in time. The AC analyses give the frequency responses of the circuit so with this we can analyze the frequency dependent properties of circuit when the input signal is a sinusoidal type changing signal. During the simulation the circuit is replaced by its linearized mathematical model (substitute image) which is valid in a given the analyzed working point (depends on the values of the elements). In this case we investigate the ratios of the voltage and current furthermore the phase difference. The AC analyzes using a frequency spectra from f_s (starting freq.) to f_e (ending freq.). The DC analyzes is suitable to examine the circuits transfer properties (the transfer effects from the inputs to the outputs).

The simulations provide the investigation of the electrical signals besides the modelling of the physical interactions between the different components. During the simulations the software takes into account the physical properties of the components (not just the values!), like temperature, catalogue data, wire resistance, etc. In logical circuit simulations the voltage and current variables could be 1; 0; and X.



Transient analysis

In doing the Transient analyzes we investigate the effect of the input signals on the output signals. During the simulations we can refer to the objects (electrical components) and the nodes of the wiring with their name (the name determines where is the object inside the circuit). There are different options for the measuring of the particular conditions of the elements or nodes. One on these is possible with the visualization of the *Node numbers* . In the case we can refer to the measurement places with the appeared numbers. It is possible to rename the wires with unique identifiers in complex systems – for this action is only need a double clicking over the wires and add a new name in the appearing window.

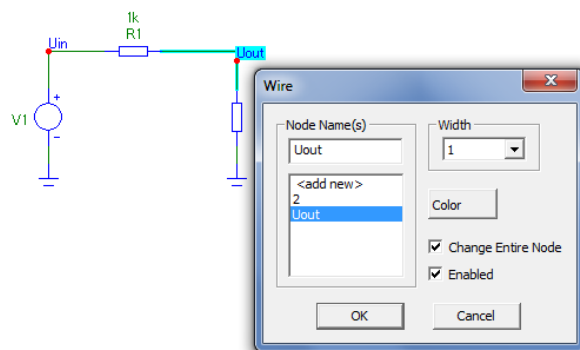


Figure 9.

To begin the Transient analysis select that from the menu bar (Analysis / Transient).

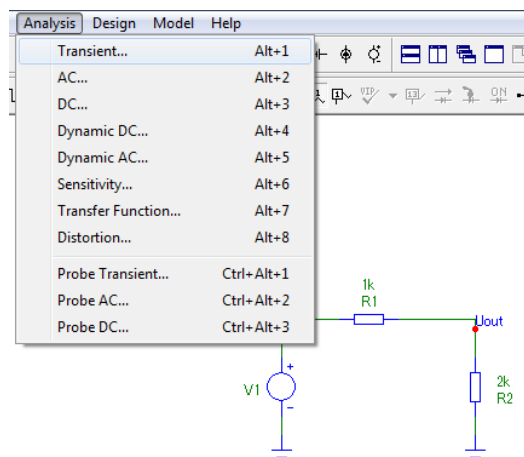


Figure 10.

The simulation parameters could be set in the opened window. First enter the simulation time into the Time Range box and set the Maximum Time Step minimum thousandths of the Time Range (this is the sampling frequency – the maximum time difference between two examination point).

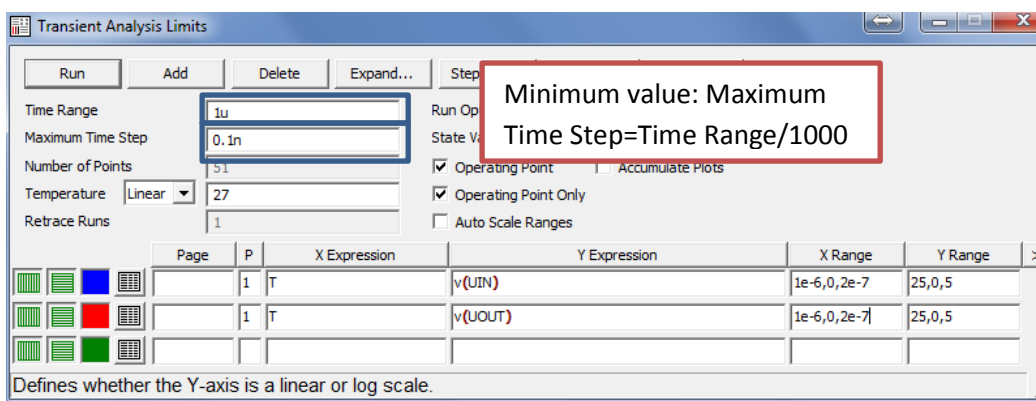



Figure 11.

If the Operating Point Only checkbox is active, then the setting of the intervals are not necessary because then the examination is limited to only one point in time.

The simulation is running by the clicking to run. Then if we turn on the  Node, voltage, current and power buttons the result of the simulation appears on the edit board (remember, that was a static investigation because we have clicked to the Operating point only).

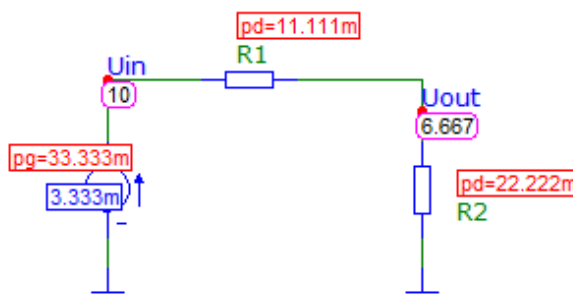


Figure 12.

Waveform Investigation

Try out the Transient analysis with the following options:

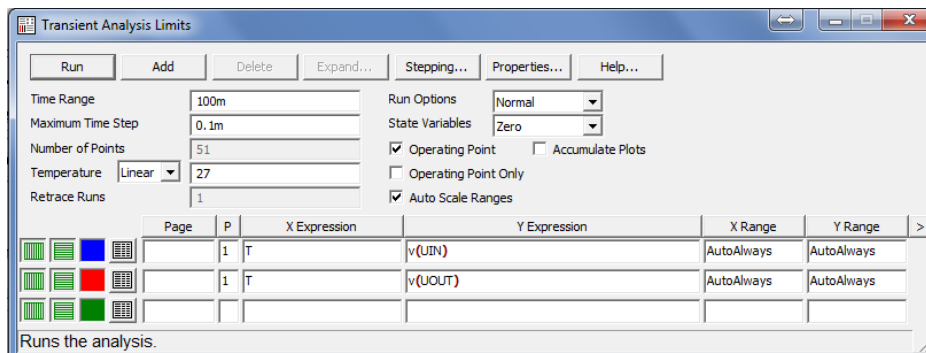


Figure 13.

We can see the result in Fig. 14. Later, if it necessary (for measuring report or other purposes) with Select Mode we can export to the clipboard the result.

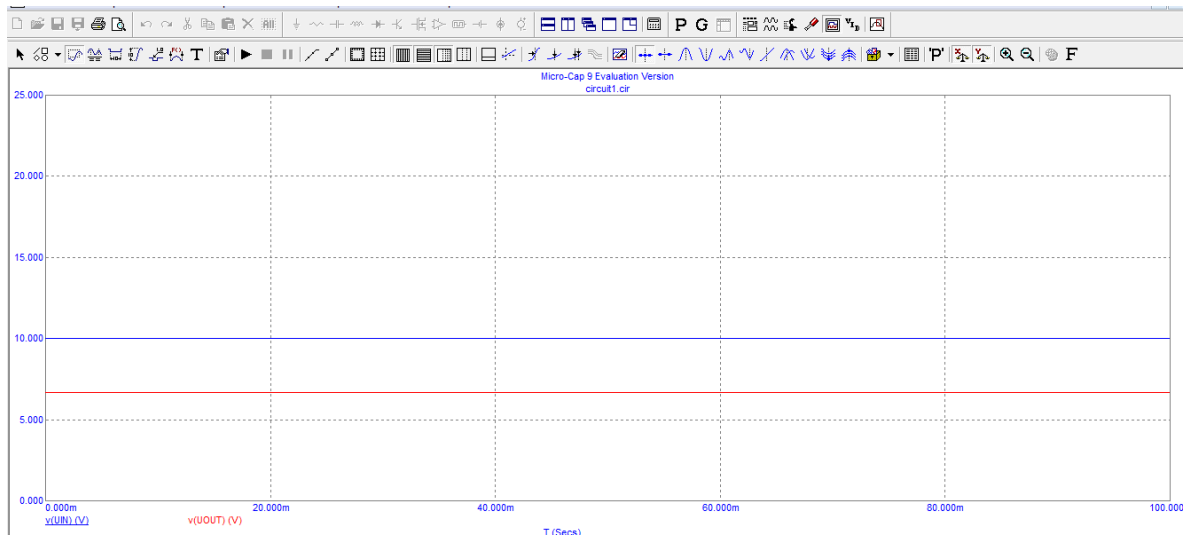


Figure 14.

We always should keep in mind that every electric elements (or a piece of an element in the case of complex elements) has the own electrical behavior, an effect on the dropping voltage and flowing current beyond the general purpose. That means that the voltage which is dropping on the given component and the current which is flowing through the given component are modified by that electrical component. Different component has different properties. For example, capacitor passes the input signal on lower frequency and inductor passes on higher frequency. Because of this we always can do a circuit which modifies the input signals such as it necessary to reach a determined goal. Actually, that is the substance of electrics: change the signals what we got in such a way that the outputs become usable for a given purpose.

7. Tasks

7.1 RC - Give the network voltage-transfer function ($A_u(j\omega)$).

Build up the following circuit. Set the resistor's resistance to 1k and the capacitor's capacitance to 1u (1 microFarad).

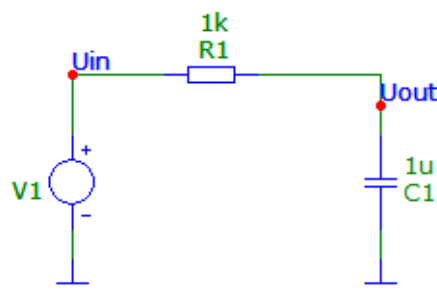


Figure 15. RC circuit



Change the DC voltage source to AC voltage source, with sinusoidal output signal. We can set different type of alternating signals – sinusoidal, triangle, square, etc. If we selected the sinusoidal output signal we can adjust the signal parameters. The most important properties is DC (DC transfer), VA (AC voltage value; if VA is 1, then the output is an 1V strength signal, on F0 frequency), F0 (frequency of the signal). Other properties may be important too if we want to specialize the output signal. **Do not forget that the output here is the output from the generator's viewpoint but it is the input from the circuit's viewpoint.**

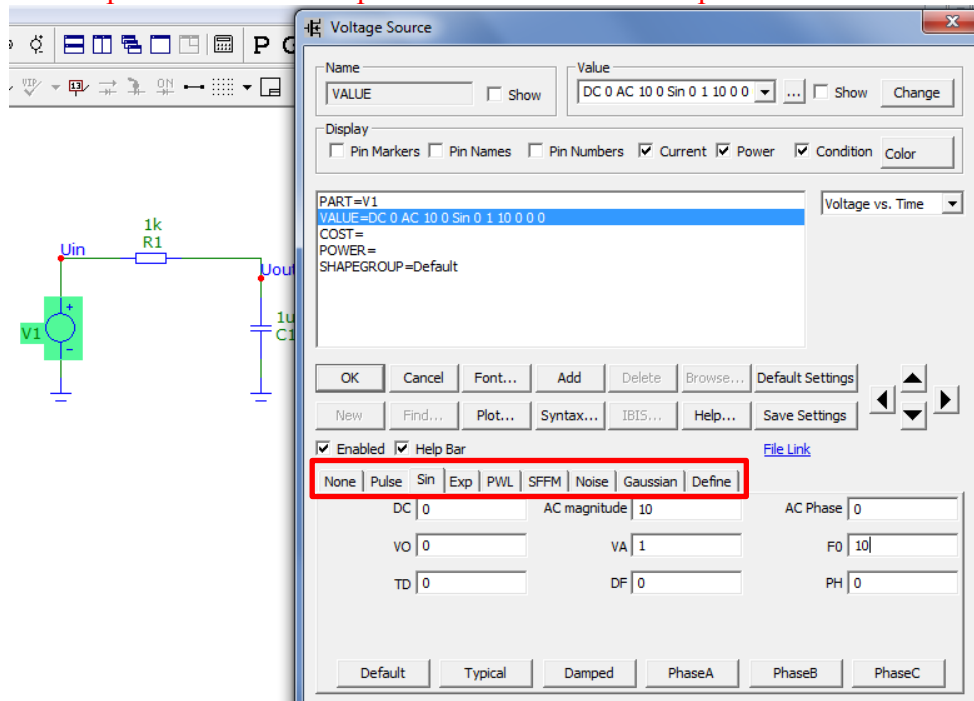


Figure 16. Voltage source properties

After that do some investigation on the circuit with Transient analysis. Set the Time Range to 1 sec, and the Maximum Time Step to 1msec. Details can be seen the figure below:

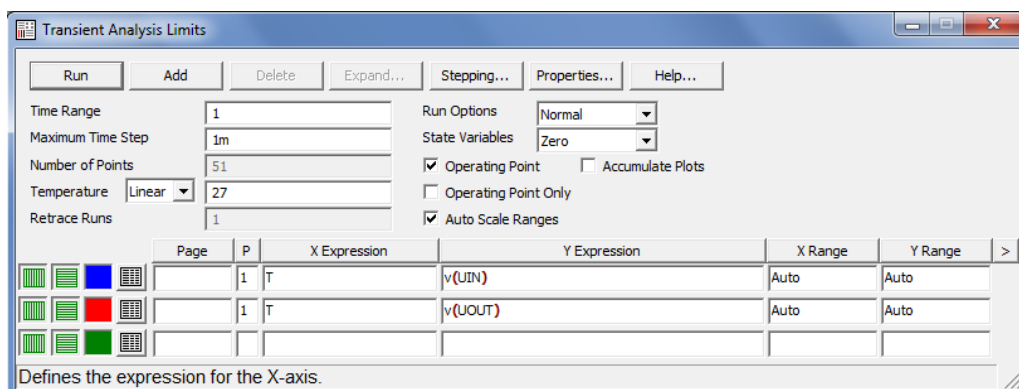


Figure 17.

As you can see, the result is almost overlapping signals. We can calculate the transfers with the formula below, but first we have to measure the amplitude of the input and output signals. Convenient solution is the using of Cursor mode (F8).

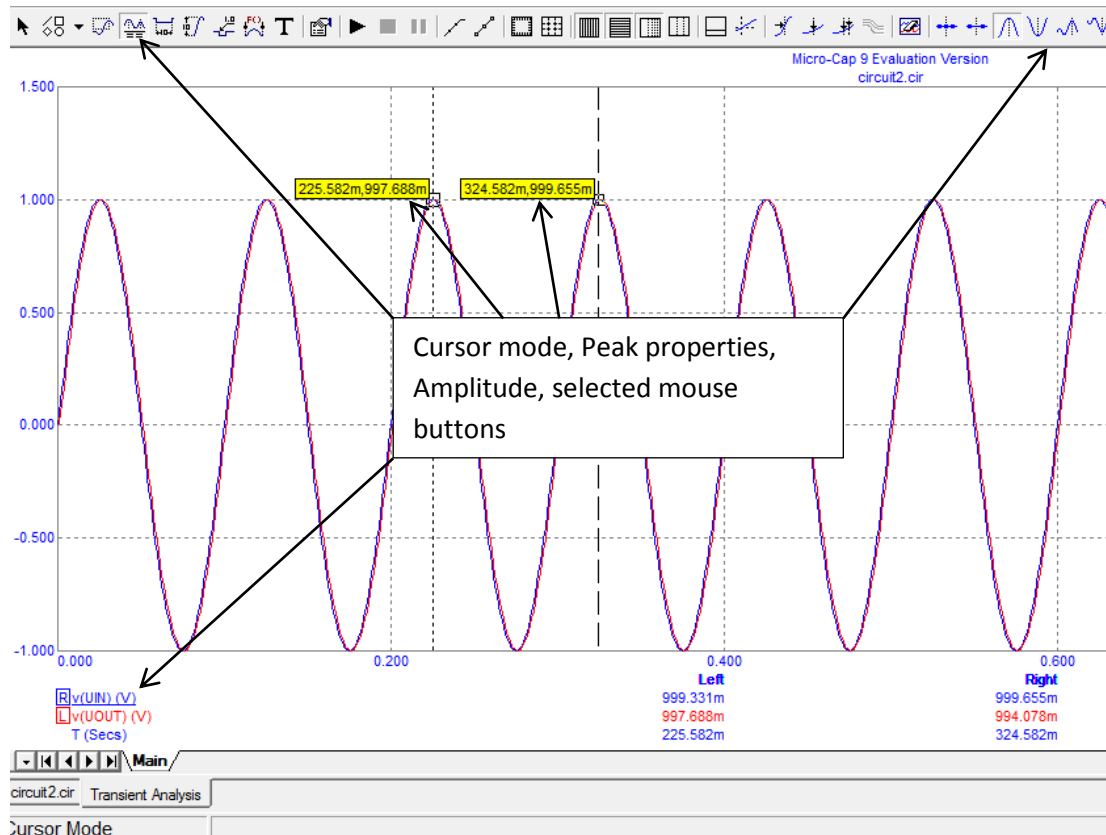


Figure 18. 10 Hz

The strengthening (or, like in this case weakening) of the circuit is:

$$Au = \frac{U_{out}}{U_{in}} = \frac{1}{1 + j\omega C}$$

$$Au[dB] = 20 \log Au$$

In this case the strengthening was 0.988 which is almost 1, in decibel's that is -0.0394. The absolute value of the strengthening, the phase rotation and the Cutoff frequency is (not calculated, because its simple value replacing):

$$|A| = \frac{1}{\sqrt{1 + j^2 \omega^2 C^2}}$$

$$\rho = -\arctg(\omega RC)$$

$$\rho_c = \frac{1}{2\pi RC}$$

Set the generator's frequency to 100, 1k and 5k Hz and look at the results. Then calculate the strengthening in these cases, in dB too.

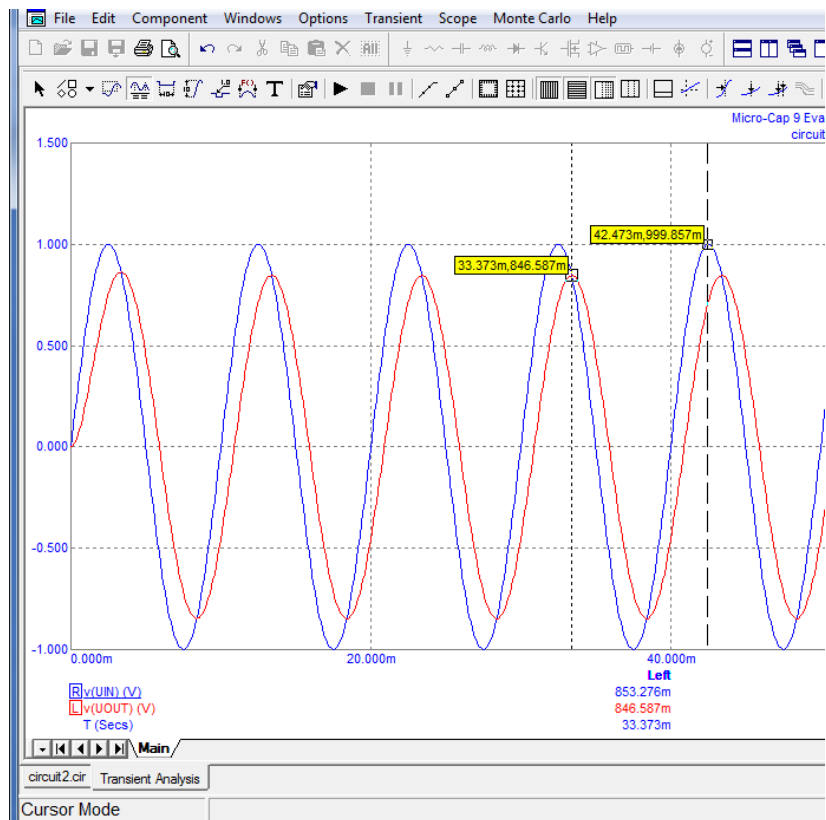


Figure 19. 100 Hz

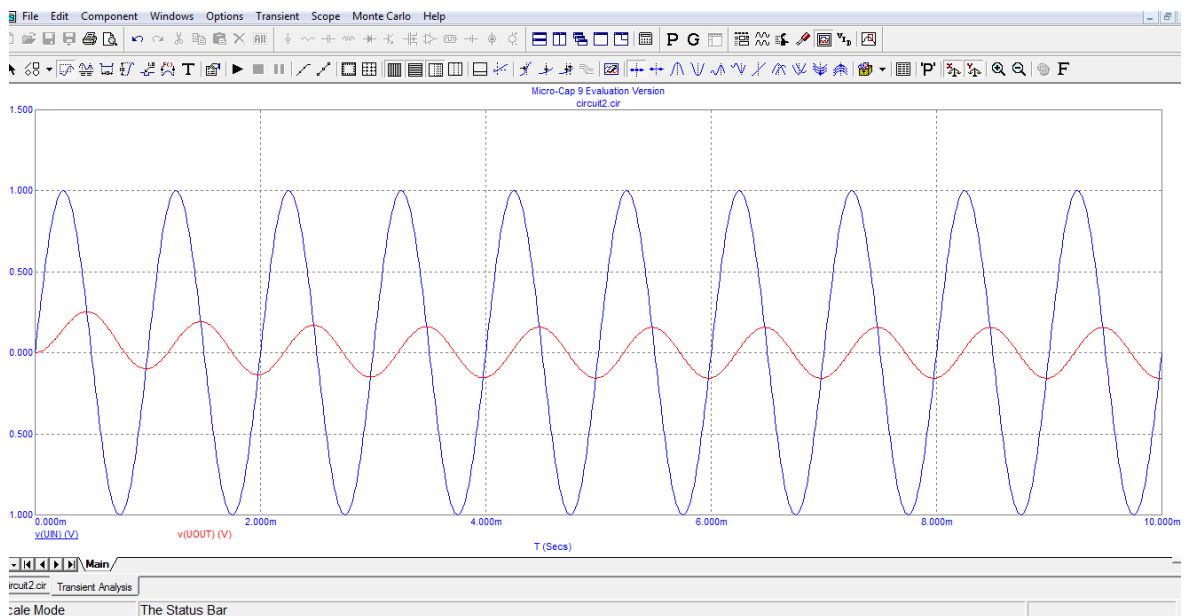


Figure 20. 1 kHz

f [Hz]	$A_u = U_{out}/U_{in}$	$au[dB] = 20 \log A_u$ [dB]
10 Hz		



100 Hz		
1 kHz		
5 kHz		

Help to the interpretation: during the operation, a simple RC circuit passes the input signal through itself such that it not passes the higher frequency components of the input signal. That means that this kind of RC circuit behaves as a filter – we called it Low pass filter (pass through those signal components which has low frequency). The RL circuit operating conversely to the RC circuit. It passes through the signal components which has high frequency.

7.2 RL - Give the network voltage-transfer function ($A_u(j\omega)$) - HOMEWORK

This will be the homework including the others: build up the following circuit with the inductor instead of the capacitor. Test it with different frequency settings. What happened? What was yours experiences?

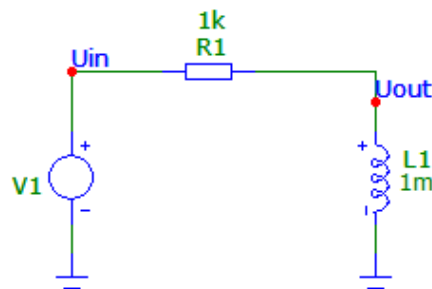


Figure 21.

f [Hz]	$A_u = U_{out}/U_{in}$	$au[dB] = 20 \log A_u$ [dB]
10 Hz		
100 Hz		
1 kHz		