ELECTRONICS

Practical work 3

COMPOSITION AND SIMULATION of ELECTRONIC CIRCUITS

Electronic Workbench – Multisim13

Control students access to features, instruments and analyses by locking them with **passwords**.

To start the analog module, go to the directory, in which Multisim13 is installed.
You probably recognize the components and instruments.

Components: Multisim13 includes a wide range of interactive, animated, virtual, rated and 3D components.
Placing a part is as simple as dragging and dropping the component into position on the workspace. You can move or rotate the component, change their values.
You connect components by dragging wires from their terminals, you can drop 2-terminal passives directly on a wire and Multisim13 will automatically make the connections for you.
A quick way to set a component`s value is to double-click the component with the left mouse button.

Virtual Instruments: Multisim13 is software to provide a suite of Virtual Instruments without needing expensive hardware.
Most needed Virtual Instruments are: voltmeter, multimeter, function generator, distortion analyzer, oscilloscope, spectrum analyzer, frequency counter, Bode plotter, etc.

Testing a Circuit: to place the instrument (voltmeter, multimeter, function generator, distortion analyzer, oscilloscope, spectrum analyzer, frequency counter, Bode plotter, etc.) on the workspace, point to its icon in the menu bar, press and hold the left mouse button, and drag it to a place above and to the right of the circuit.
To adjust a test instrument`s controls, it must be "zoomed open".
To open an instrument, select its icon and double-click the instrument icon.
Connect the instrument`s icon to the circuit.

Analyses:

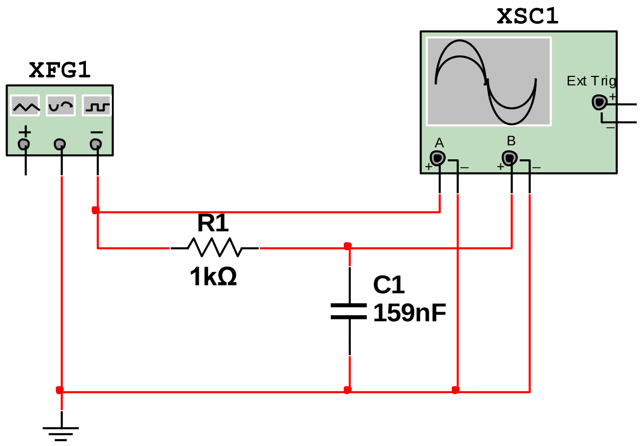
DC operating point, DC sweep, I-U analysis, Pole-Zero, AC frequency sweep, Tempretature sweep, Transfer function, Fourier, 3dB point, Monte Carlo etc.
To activate a circuit, click the power switch at the top right of the screen.

The grapher serves as a multi-purpose tool to view, save, print and export the results of a simulation analysis.

The Multisim RF module provides simulator enhancements and special functionality for higher frequencies enabling students and educators to build and experiments with RF circuits.

1. <u>Composition and simulation of analog circuits.</u>

1.1 Create a simple RC – integrator circuit as in Fig. 1.1.



2.

Fig. 1.1 Simulation circuit for RC – integrator

Simulate the RC-integrator and obtain the transient response for R1 = 1kΩ;
Sketch the transient responses for resistors 100 Ω and 10 kΩ. on Fig.1.2. and the freguency responses for resistors 1kΩ, 100 Ω and 10 kΩ. on Fig.1.3.
Calculate the periodic time: T =   ms.

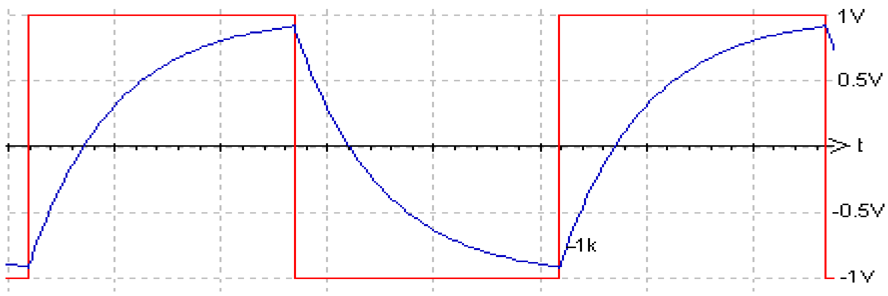


Fig.1.2. Transient response for RC – integrator.

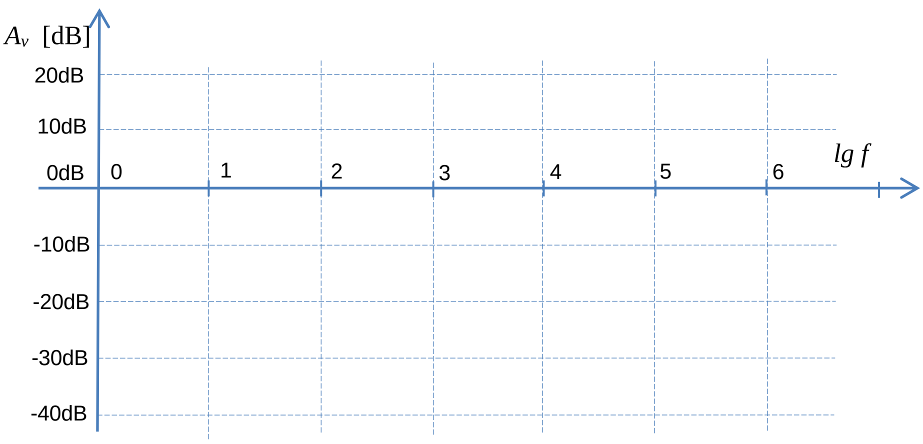


Fig.1.3. Gain-frequency response for RC-integrator.

Write the list of features for RC – integrator...

....

....

2. Create the simple RC – differentiator circuit as in Fig. 1.4

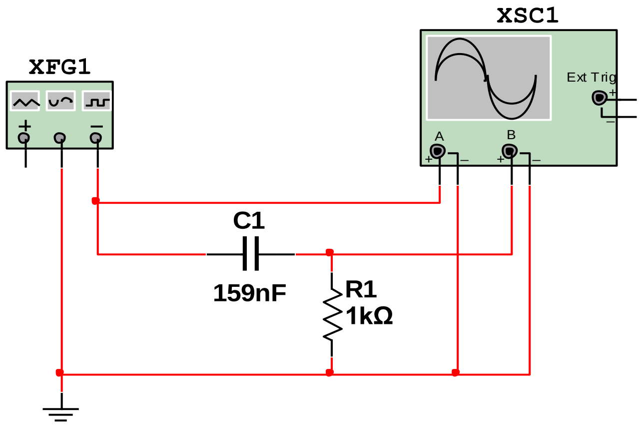


Fig. 1.4 Simulation circuit for RC – differentiator

Simulate the RC-integrator and obtain the transient response for R1 = 1kΩ;
Sketch the transient responses for resistors 100 Ω and 10 kΩ. on Fig.1.5. and the freguency responses for resistors 1kΩ, 100 Ω and 10 kΩ. on Fig.1.6.
Determine the V_{exm} = V for R1 = 1 kΩ

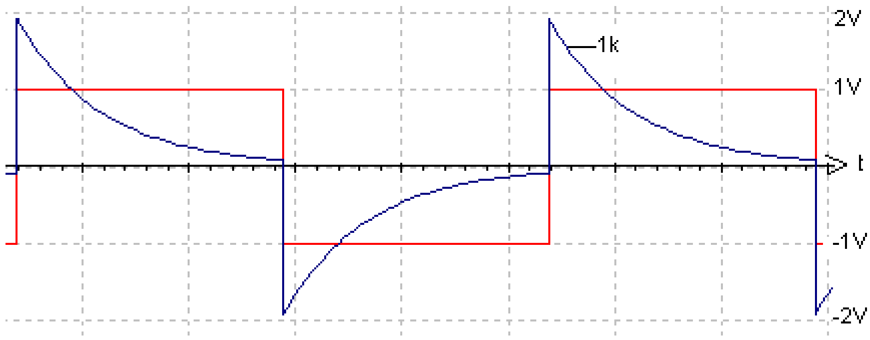


Fig. 1.5. Transient response for RC – differentiator.

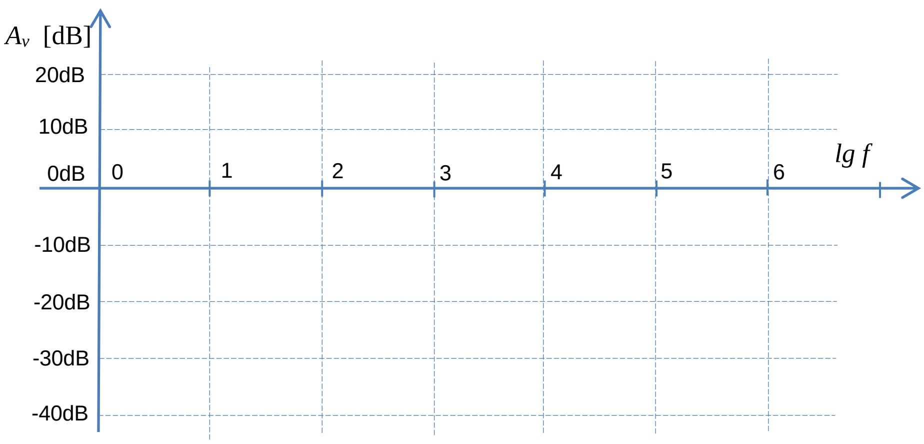


Fig.1.6. Gain-frequency response for RC-differentiator.

Write the list of features for RC –differentiator. ....

....

....

1.3. Create the non-inverting operational amplifier (OA) circuit as in Fig.1.7

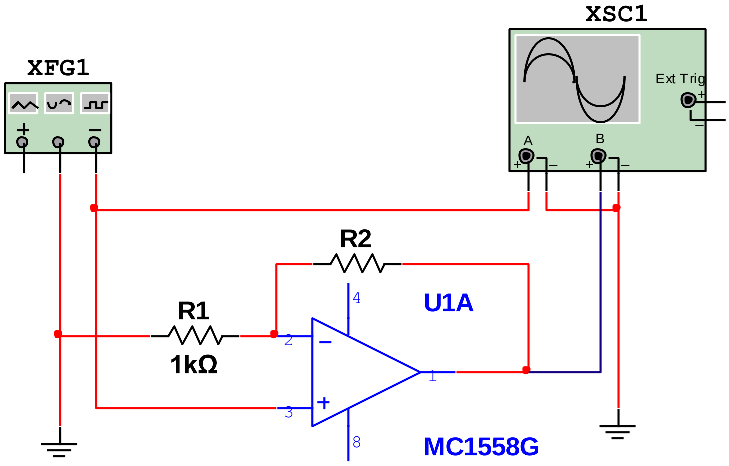


Fig. 1.7 Simulation circuit for non-inverting OA

Find the R2 value for non-inverting OA circuit at *Av* = 10:

Av = ; and R2 = = = *[Ω]*;

Simulate the non-inverting OA circuit and study the features of the amplifier.

1.4. Create the TT – bridge simulation circuit as shown in Fig. 1.8.

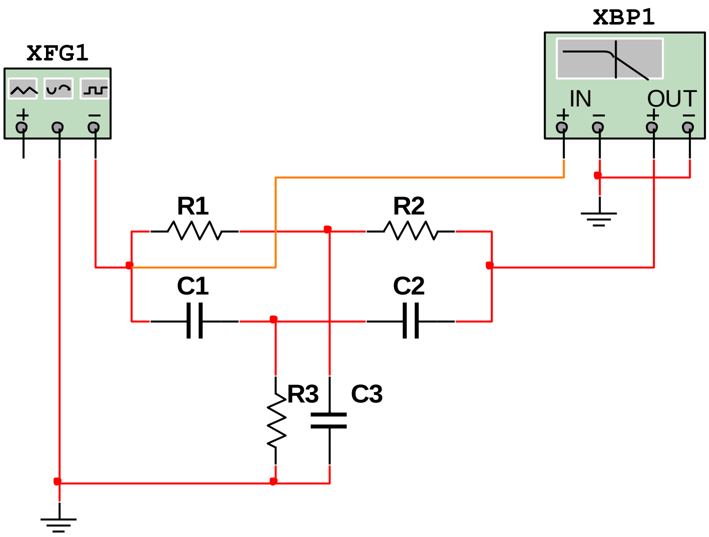


Fig. 1.8. Simulation circuit for TT – bridge

Simulate the TT - bridge and obtain the gain – frequency response.
Fix the rejection frequency f_0 = Hz;
and compare it to the theoretical rejection frequency $f_{0\,th}$ value:

f_{0th} = ---------------- = ---------------------- = = Hz;

Limit the Bode plotter`s vertical scale from – 80 dB to 10 dB;

Limit the Bode plotter`s horizontal scale $f_0 \pm 2$ decades ;

1.5. Create the integrator gain – frequency responses simulation circuit.

Simulate the integrator and obtain the gain – frequency responses for the integrators R1R10C1 and R1R10C2 at the next points:

- A_{0C1} and A_{0C2} at the frequency 10 Hz;
- nearest frequencies at $A_{C1} = A_{0C1}$ – 3dB , $A_{C2} = A_{0C2}$ – 3dB, 0 dB and -10 dB gain values accordingly using the next capacitors:

C1 = 1 nF ja C2 = 10 nF.

Record results in the Table 1 and place the points in Fig. 8 of the practical work nr.2.

Table 1

R1R10C1		R1R10C2	
f	A_{C1} [dB]	f	A_{C2} [dB]
10 Hz	A_{0C1} =	10 Hz	A_{0C2} =
f_{h1} =	$A_{C1} = A_{0C1}$ – 3dB ( )	f_{h2} =	$A_{C2} = A_{0C2}$ – 3dB ( )
f =	A_{C1} = 0dB ( )	f =	A_{C2} = 0dB ( )
f =	A_{C1} = -10dB ( )	f =	A_{C2} = - 10dB ( )

1.6. Create the integrator simulation circuit as shown in Fig.1.9.

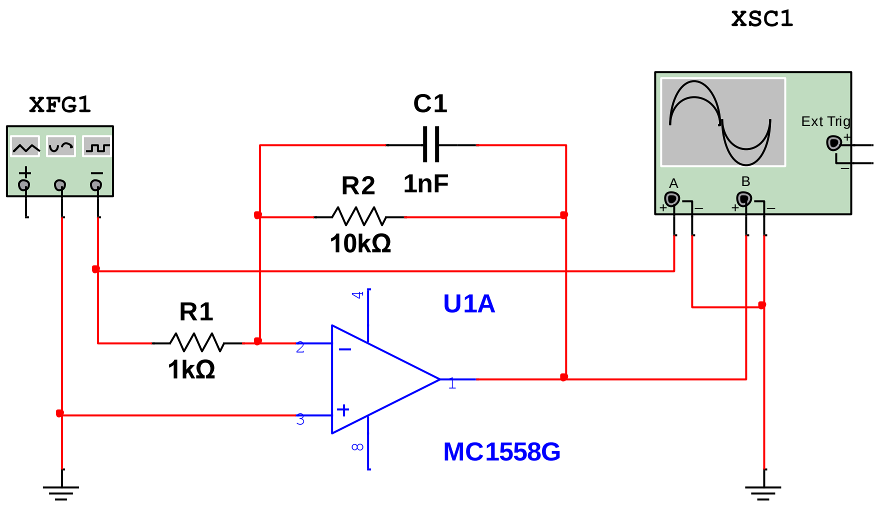


Fig. 1.9. Simulation circuit for integrator

Simulate the integrator and obtain the transient response for the integrator at square wave input of amplitude 2 V and of frequency 500 kHz.
Study the output response and fix the output peak to peak voltage Δv_{EX} = V;

Compare the fixed output peak to peak value to the calculated value Δv_{EXid} for ideal integrator:

$$\Delta v_{EXid} = \frac{1}{R1\,C1} \int v_{IN}\, dt = \frac{v_{IN} \cdot t}{R1\,C1} = \text{--------------------} = \quad ;$$

1.7. Create the CE configuration transistor amplifier circuit as in Fig. 1.10

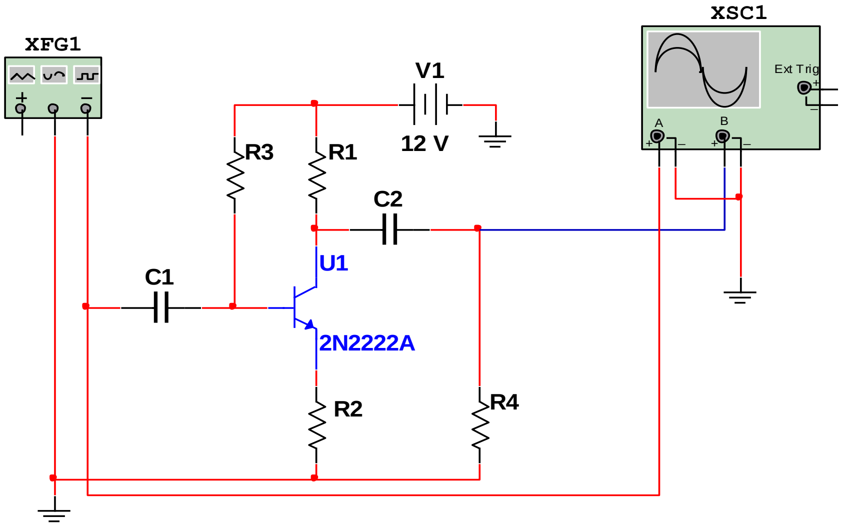


Fig. 1.10 Simulation circuit for CE configuration transistor amplifier

Choose resistors and capacitors for amplifier`s gain $A_v = \frac{V_{ex\,m}}{V_{in\,m}} = 10$ at 1kHz.

Input and output signal waveforms must be sinusoidal.
Input voltage V_{inm} = 200 mV;
Output voltage $V_{exm} = V_{inm} * A_v$ = 200mV*10 = 2 V;
Phase angle φ = 180º;

Record the resistors and capacitors values on the diagram in Fig.1.10.

Conclusions:………………………………………………………………………
…………………………………………………………………………………
…………………………………………………………………………………
…………………………………………………………………………………
………………………………………………………………………………..