# Electronic Circuit Design Theory, Methods and Tools

Part I: Circuit theory (Moodle section Circuit theory)

The software used in the simulation exercises is LTSpice XVII by Analog Devices. LTSpice can be downloaded from

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

Each simulation exercise is based on a single LTSpice simulation schematic, including the MOSFET model or other model (operational amplifier), all necessary parameters, as well as simulation commands. These are marked with LTSpice comments (blue text) above each set of LTSpice directives (black text). An example schematic can be seen below:
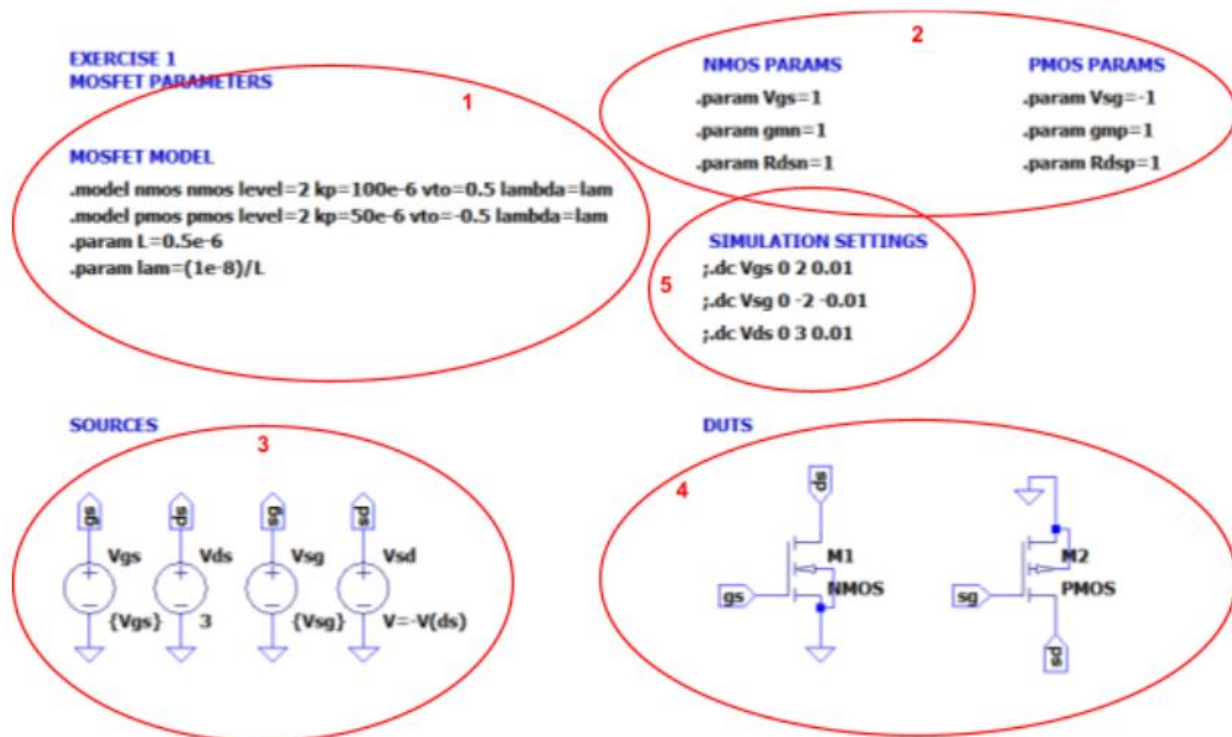

Here, you can see the model definition (1), which should not be changed. The parameters for the current simulation can be seen in (2), this is where the values will be filled for reference while writing the report. The voltage and current sources for the simulation exercise can be seen in (3), and the device under test, or DUT, can be seen in (4). In this current example, there are two DUTs, since we are looking at both n-type and p-type MOSFETs. The simulation commands can be seen in (5). The simulation commands are disabled by adding a semicolon in front, in the screenshot above all simulation commands are disabled. Right-clicking on a simulation command and removing the semicolon activated it.

A simulation is initiated by right-clicking on a blank area on the schematics, and then clicking on run in the mouse menu. This can also be done by clicking on the running person on the taskbar.

In order to change the values of the parameters, or values of components, a right click on the parameter or component opens up a dialog where the desired changes can be done.

In order to plot waveforms from a schematic you can just move your mouse over some node after simulation. To plot voltages the cursor of the mouse should look like a red probe as shown in Fig below (on the right) and to plot currents through some component the cursor should look like that black current probe as shown in Fig. below on the left.
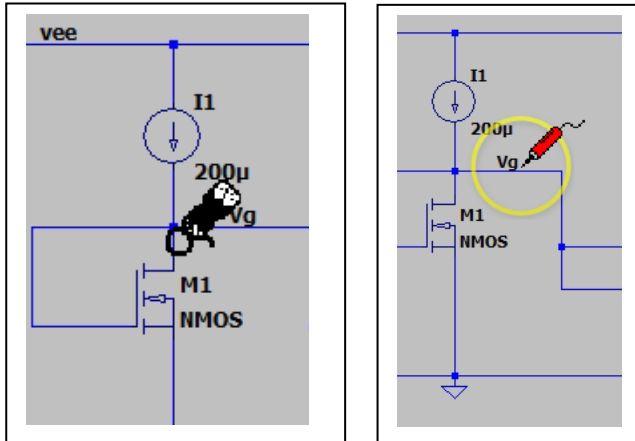
You can also add traces by clicking the mouse right button over the plotting window and choosing Add trace. You will get the list of all the nodes and all the currents through different components.
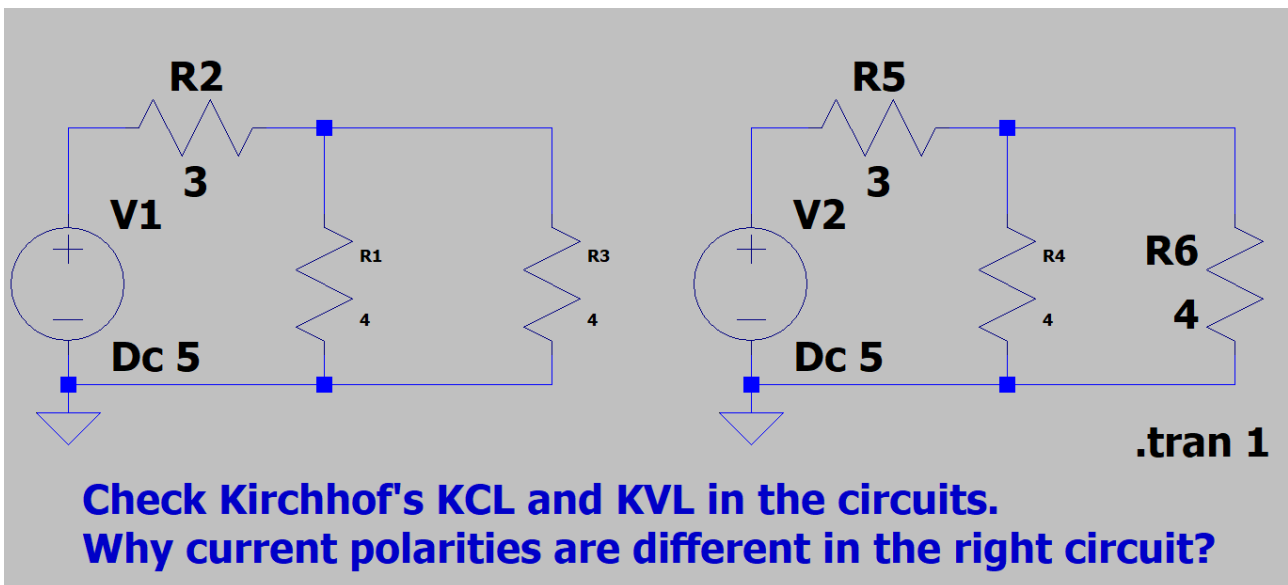
Task 1: Voltages and currents

In this simulation task, we will get to know the basic calculation of node voltages and currents.

1. Pre-material for this task can be found from Moodle on section => Pre-material for tasks
2. Simulation files for this task can be found from Moodle in directory => Task 1. Download all the files to your LTSpice directory.

a.　　　Kirchhof's laws.

The schematic of this exercise is shown below and is named Q1_1.asc.

Check Kirchhof's KCL and KVL in the circuits.
Why current polarities are different in the right circuit?

Calculate the node voltages and currents through resistances by using Kirchhof's KCL and KVL. Give the formulas and answers.

Answer:

Applying KCL in node 2, results in: $I2 = I1 + I3$

Applying KVL in outer loop results in: $V1 = I2 \cdot R2 + I3 \cdot R3$

Applying KVL in outer loop results in: $I1 \cdot R1 = I3 \cdot R3$

Therefore, $I1 = \frac{R3}{R1} \cdot I3 = \frac{4}{4} \cdot I3 \Rightarrow I1 = I3 \Rightarrow I2 = 2I1 = 2I3 \Rightarrow I1 = I3 = \frac{1}{2}I2$

Replacing, we get: $V1 = I2 \cdot R2 + \frac{1}{2}I2 \cdot R3 = I2\left(R2 + \frac{R3}{2}\right) \Rightarrow I2 = \frac{V1}{\left(R2 + \frac{R3}{2}\right)}$

$I2 = \dfrac{5}{\left(3 + \frac{4}{2}\right)} = 1\,A \Rightarrow I1 = \frac{1}{2} = 0.5\,A \Rightarrow I3 = 0.5\,A$

The node voltages are: $V1 = 5V$; $V2 = I2 \cdot R2 = I3 \cdot R3 = 0.5 \cdot 4 = 2\,V$
And for the second circuit: $V1 = 5V$; $V3 = V2 = 2V$ and $I5 = I2 = 1\,A$; $I4 = I6 = I1 = 0.5\,A$

Now run the transient simulation either by clicking the running person in the toolbar, or by right mouse-click → run. Then, check your calculations by probing currents through the resistances and node voltages. Why current polarities are different between circuits (This is related to orientation of the resistance)? Hint: when you are probing the current check the arrow showing the direction of current that you are plotting? Did you get the same results as calculated?
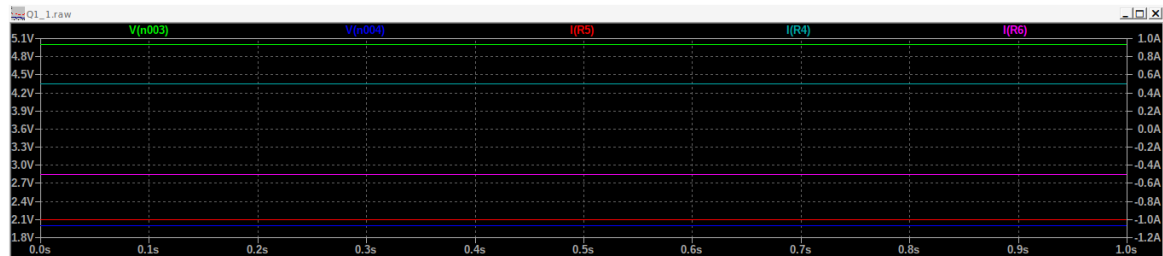
Answer:

current polarities are different between circuits because in the second circuit resistors are flipped and in spice software each pin has a designation and for each components the current should a certain direction related to that designation.

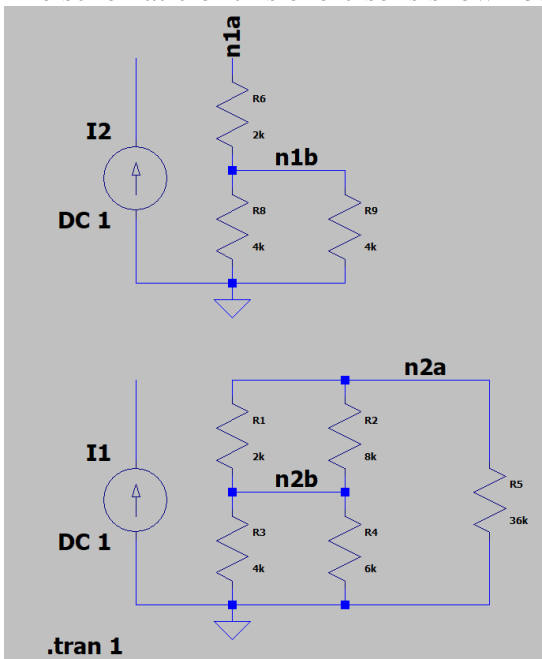For the first circuit the results were identical to the calculations.
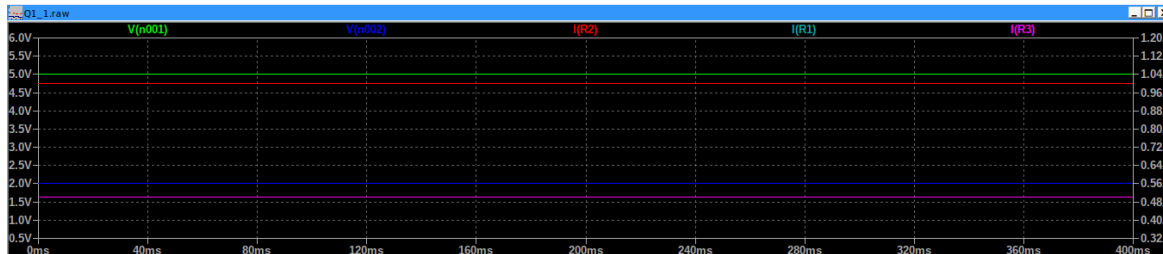
For the second circuit the currents $I5$ and $I6$ are in the opposite directions making their values negative as shown in the plot bellow.

b.   total resistance

The schematic of this exercise is shown below and is named Q1_2.asc.

Calculate the total resistance between n1a and ground node and between n2a and ground node. Give the formulas your derivation and give the results.

Answer:

For the first circuit, $R8$ and $R9$ are in parallel, therefore, the total resistance from node n1a to the ground is: $R_{n1a} = R6 + (R8 \parallel R9) = 2 + (4 \parallel 4) = 2 + 2 = 4\,k\Omega$

For the second circuit, $R1$ and $R2$ are in parallel, $R3$ and $R4$ are in parallel, and their series combination is in series with $R5$, therefore, the total resistance from node n2a to the ground is:

$R_{n2a} = R5 \parallel \{(R1 \parallel R2) + (R3 \parallel R4)\} = 36 \parallel \{(2 \parallel 8) + (4 \parallel 6)\}$
$R_{n2a} = 36 \parallel \{1.6 + 2.4\} = 36 \parallel 4 = 3.6\,k\Omega$

Now, make a correct connection to simulate these total resistance values. Give the simulation results. Did they match with your calculation?

Yes, the simulation results do math the calculation.

The voltage at node n1a is $4kV$ and for a current of $1A$ the resistance is: $\dfrac{4kV}{1A} = 4k\Omega$

The voltage at node n2a is $3.6kV$ and for a current of $1A$ the resistance is: $\dfrac{3.6kV}{1A} = 3.6k\Omega$
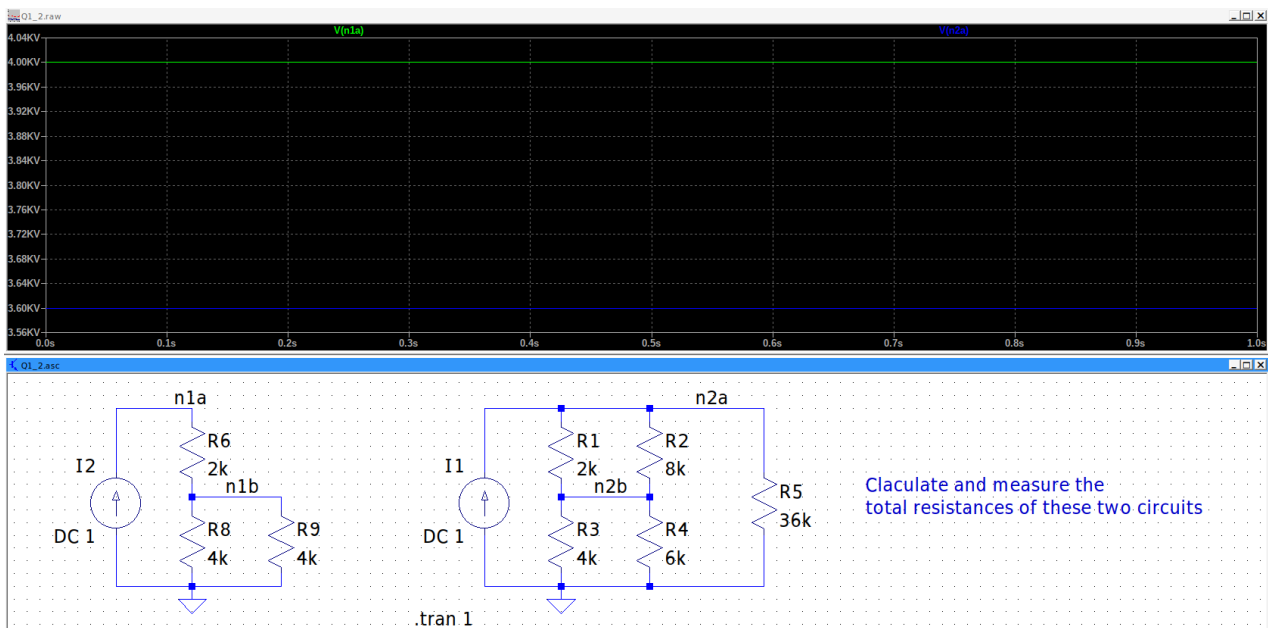
C. ## current-voltage pairs Theven and Norton.

The schematic of this exercise is shown below and is named Q1_3.asc.

**Measure the I-V pairs of the sources for r= 10, 100, 1000
Note & explain the polarities**
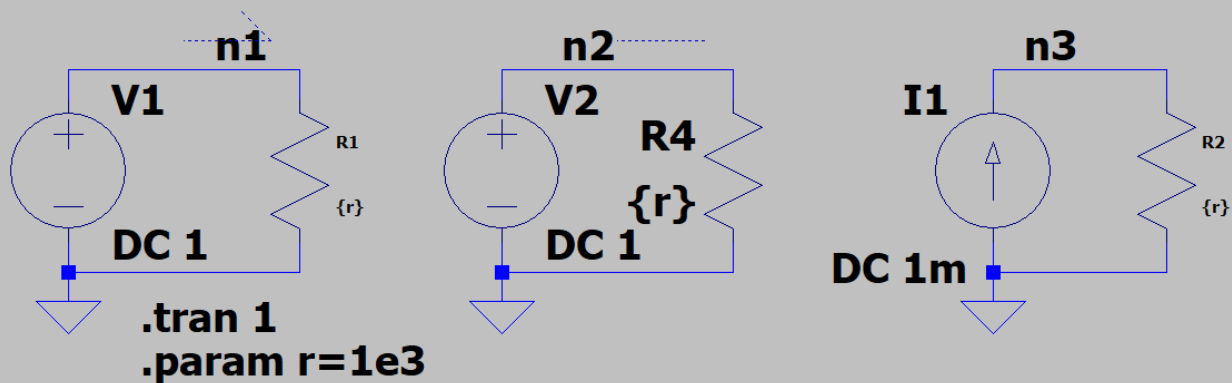
Give the values of currents through resistances and voltages over the resistances:

	n1		n2		n3	
r	I	V	I	V	I	V
10	100 mA	1 V	-100 mA	1 V	1 mA	10 mV
100	10 mA	1 V	-10 mA	1 V	1 mA	100 mV
1k	1 mA	1 V	-1 mA	1 V	1 mA	1 V

Are there differences between Norton's (current source) and Theven's (Voltage source) circuits?

Answer:
There are no differences between Norton's and Theven's circuits.

What is causing the polarity difference between n1 and n2 circuits. Again remember probing issue!
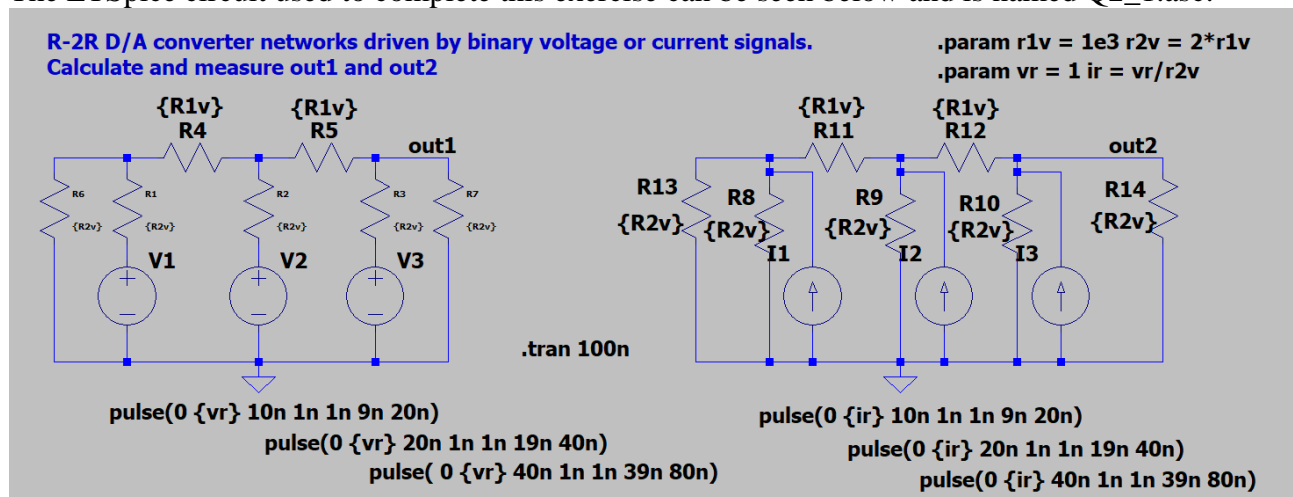
Answer:
The difference between n1 and n2 circuits is a result of the resistor R4 flipped so the direction of the current through it is from the ground to node n2. This issue results because for spice components the direction of current through any component is defined from a specific pin to another.

Task 2. Voltage and current division and impedances

1. Pre-material for this task can be found from Moodle on section => Pre-material for tasks
2. Simulation files for this task can be found from Moodle in directory => Task 2. Download all the files to your LTSpice directory.

a.　　　R-2R voltage

The LTSpice circuit used to complete this exercise can be seen below and is named Q2_1.asc:

R-2R D/A converter networks driven by binary voltage or current signals. Calculate and measure out1 and out2

.param r1v = 1e3 r2v = 2*r1v
.param vr = 1 ir = vr/r2v

pulse(0 {vr} 10n 1n 1n 9n 20n)
pulse(0 {vr} 20n 1n 1n 19n 40n)
pulse(0 {vr} 40n 1n 1n 39n 80n)

pulse(0 {ir} 10n 1n 1n 9n 20n)
pulse(0 {ir} 20n 1n 1n 19n 40n)
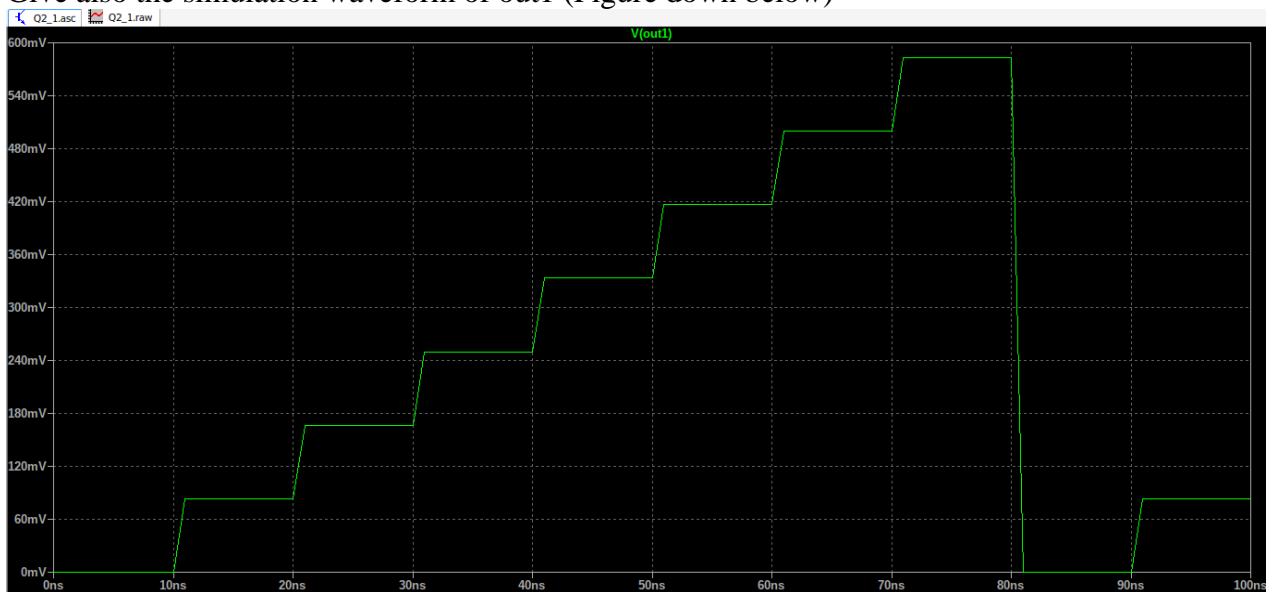pulse(0 {ir} 40n 1n 1n 39n 80n)

.tran 100n

Calculate out1 values when the V1, V2 and V3 have values shown Table below and add values to column Out1(cal.).

Simulate the transient simulation and give the simulated values. Check also V1, V2 and V3 nodes that they are giving this binary counter output.

|	V3[V]	V2[V]	V1[V]	Out1(cal.)	Out1(sim)
0	0	0	0	0 V	0 V
1	0	0	1	83.33 mV	83.333336 mV
2	0	1	0	166.67 mV	166.66667 mV
3	0	1	1	250 mV	250 mV
4	1	0	0	333.33 mV	333.33334 mV
5	1	0	1	416.67 mV	416.66666 mV
6	1	1	0	500 mV	500 mV
7	1	1	1	583.33 mV	583.33331 mV
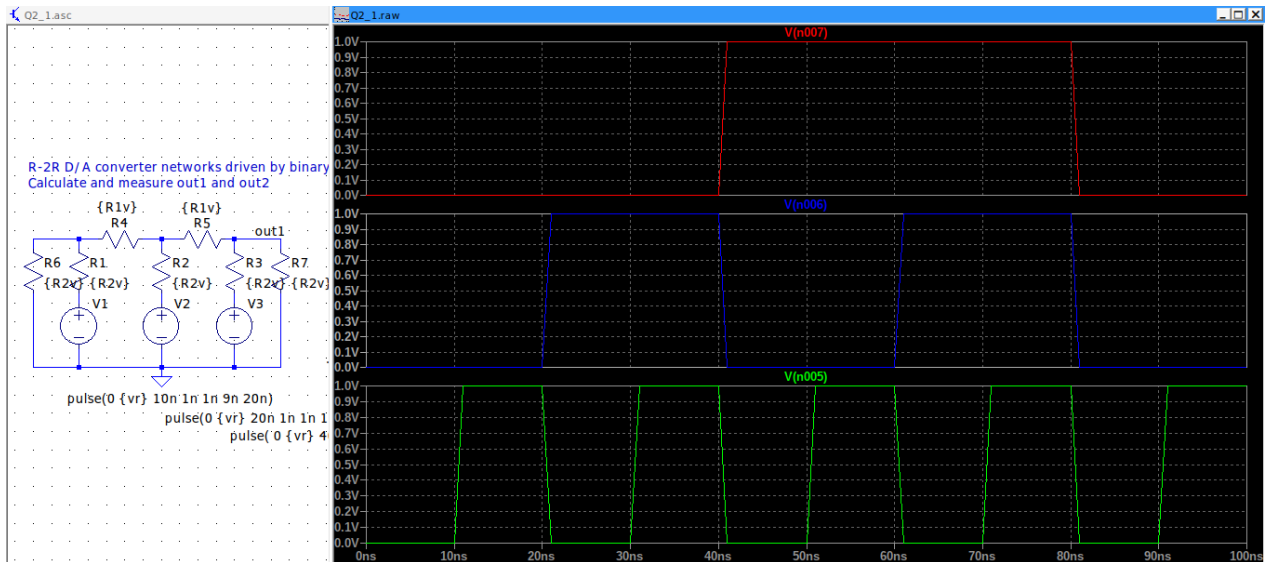
Give also the simulation waveform of out1 (Figure down below)

Calculate out2 values when the I1, I2 and I3 have values shown Table below and add values to column Out1(cal.). Simulate the transient simulation and give the simulated values. Check also I1, I2 and I3 nodes that they are giving this binary counter output.

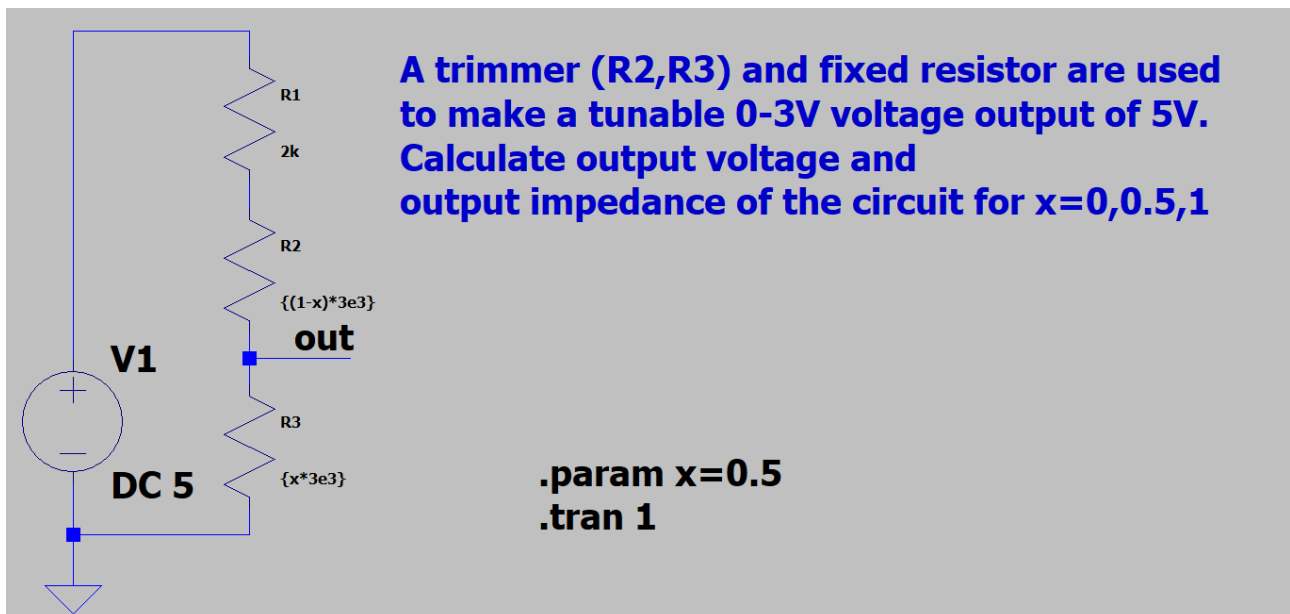|	I3[mA]	I2[mA]	I1[mA]	Out2(cal.)	Out2(Sim.)
0	0	0	0	0 V	0 V
1	0	0	0.5	83.33 mV	83.333336 mV
2	0	0.5	0	166.67 mV	166.66667 mV
3	0	0.5	0.5	250 mV	250 mV
4	0.5	0	0	333.33 mV	333.33334 mV
5	0.5	0	0.5	416.67 mV	416.66666 mV
6	0.5	0.5	0	500 mV	500 mV
7	0.5	0.5	0.5	583.33 mV	583.33331 mV
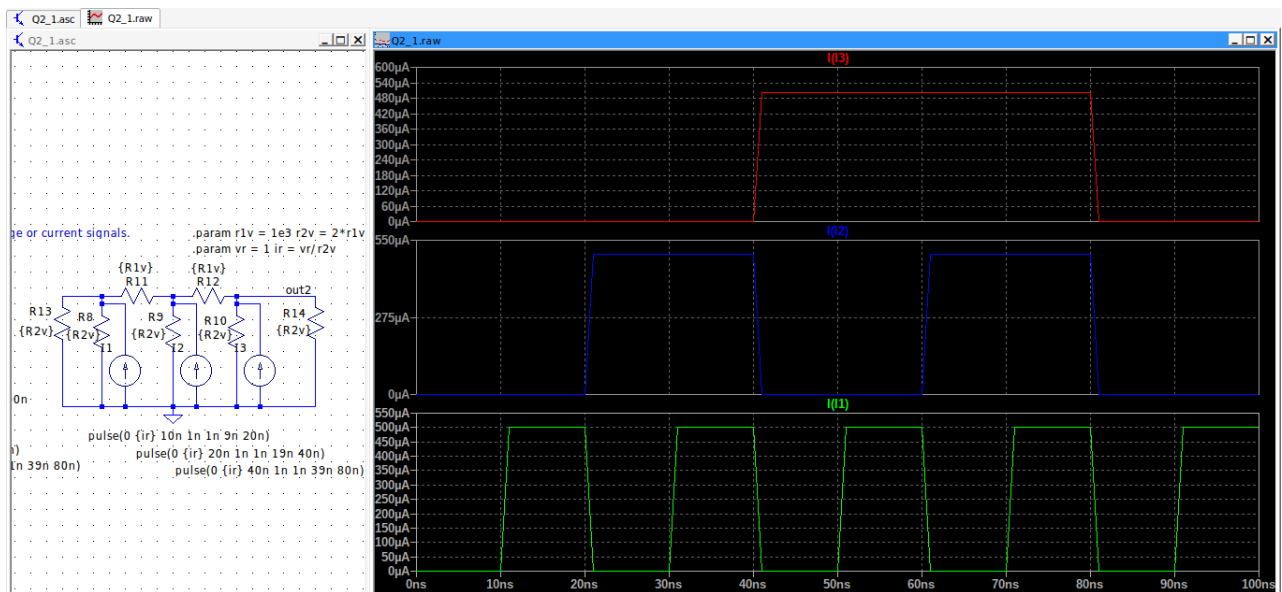
Give also the simulation waveform of out2.
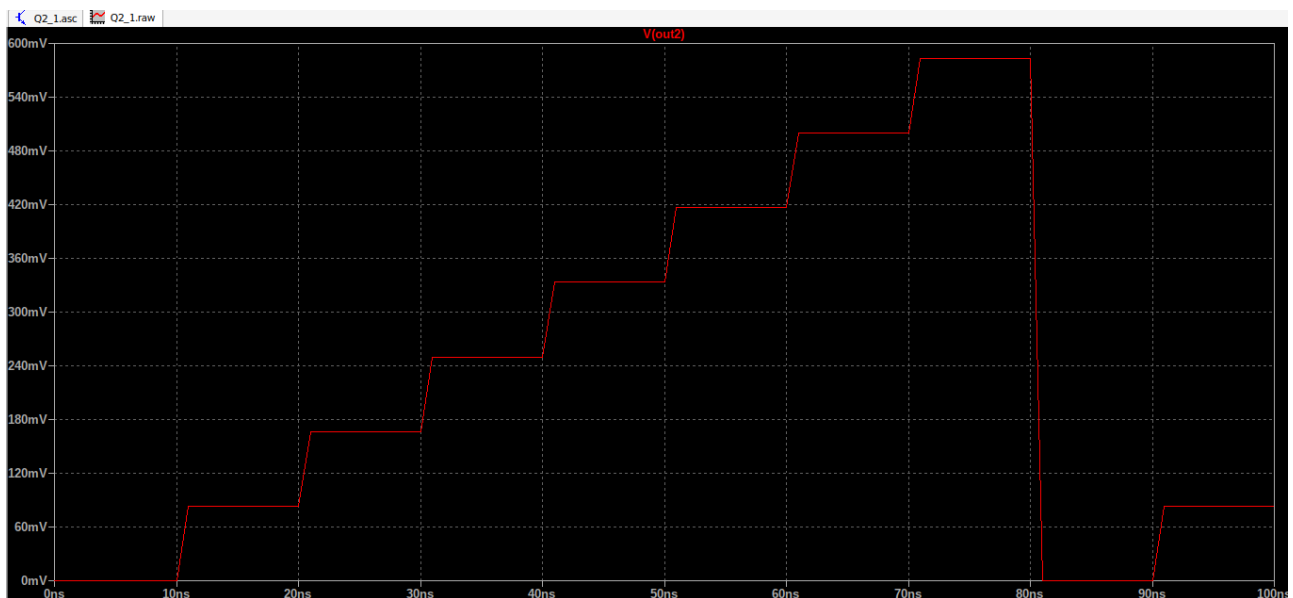
b.   Trimmer

The LTSpice circuit used to complete this exercise can be seen below and is named Q2_2.asc:

A trimmer (R2,R3) and fixed resistor are used to make a tunable 0-3V voltage output of 5V. Calculate output voltage and output impedance of the circuit for x=0,0.5,1

R1
2k

R2
{(1-x)*3e3}

out

V1

R3
{x*3e3}

DC 5

.param x=0.5
.tran 1

Calculate the output resistance Rout values when the x has values shown Table below and add values to column Out1(cal.) and Rout.

Simulate the transient simulation and give the simulated values Out(sim).

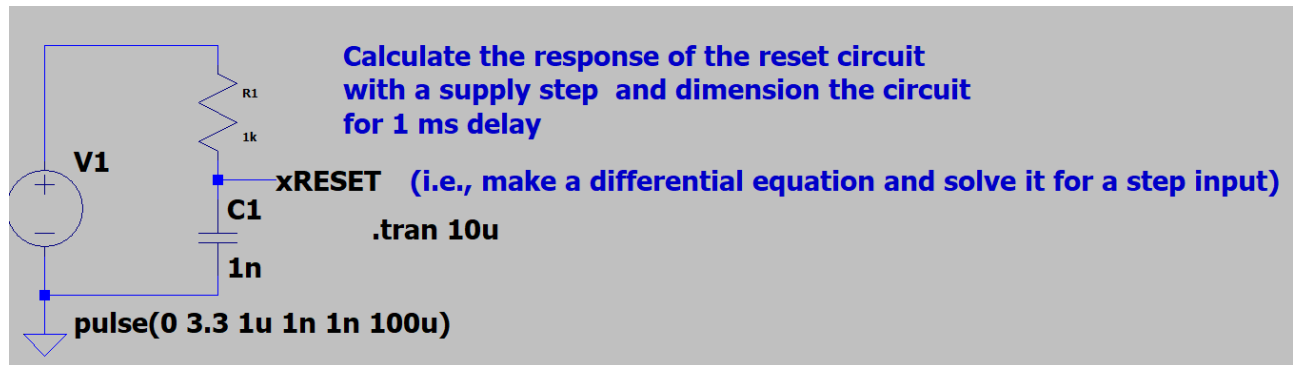(When setting $x = 0$ or $x = 1$, an error appears saying that the resistance value cannot be set to zero. We can avoid that by setting the approximation $x = 0.9999$ and $x = 0.0001$)

X	Out(cal.)	Out(sim)	Rout
0	0 V	0 V	0 Ω
0.5	1.5 V	1.5 V	1.05 kΩ
1	3 V	3 V	1.2 kΩ

Task 3. Step response

1. Pre-material for this task can be found from Moodle on section => Pre-material for task
2. Simulation files for this task can be found from Moodle in directory => Task 3. Download all the files to your LTSpice directory.

The LTSpice circuit used to complete this exercise can be seen below and is named Q3_1.asc:

Calculate the response of the reset circuit with a supply step and dimension the circuit for 1 ms delay

xRESET (i.e., make a differential equation and solve it for a step input)
.tran 10u

pulse(0 3.3 1u 1n 1n 100u)

Calculate the step response of the circuit making differential equation. Give the equation and give the the value of the rise time of output, when the output has reached the value of 63% of it's maximum?

Answer:

Applying KVL, results in: $R1 \cdot C \frac{dv_{xRESET}}{dt} + v_{xRESET} = V1 \Rightarrow \frac{dv_{xRESET}}{dt} + \frac{v_{xRESET}}{R1 \cdot C} = \frac{V1}{R1 \cdot C}$

This differential equation has the following solution: $v_{xRESET} = V1 \left(1 - e^{-\frac{t}{R1 \cdot C}}\right)$

Thus, $v_{xRESET} = 3.3 \left(1 - e^{-\frac{t}{1e-6}}\right)$

The maximum value of v_{xRESET} is , therefore, for the rise time t_{Rise}, we get:

$0.63 \cdot 3.3 = 3.3 \left(1 - e^{-\frac{t_{Rise}}{1e-6}}\right) \Rightarrow t_{Rise} = 0.9943e - 6sec \approx 1 \, \mu sec$

The output v_{xRESET} would reach 63% of its maximum value after $1 \mu sec$

Check that simulation is showing the same risetime. Give the simulation waveform with the value of risetime=>

The rise time in the simulation is time between cursor 1 and cursor 2, as highlighted in the cursor menu below.
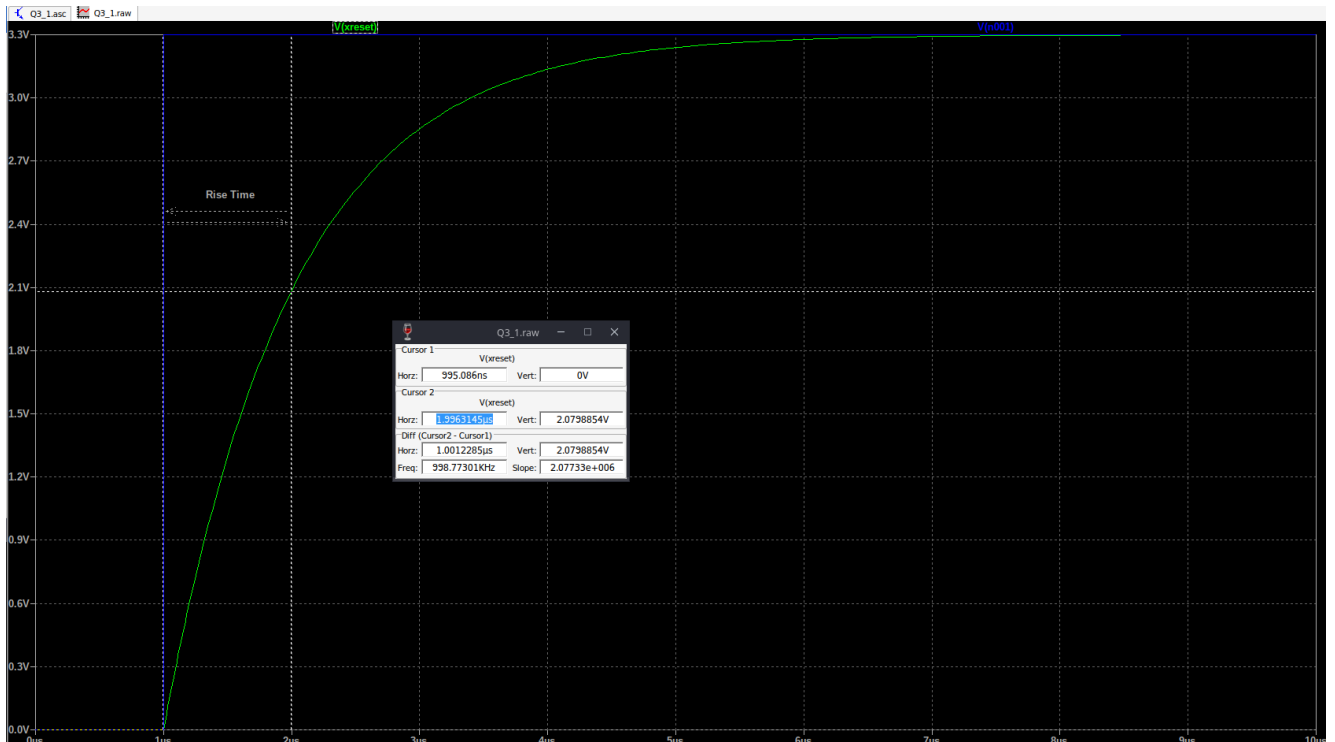
What is defining this 63% risetime?

Answer:
The 63% rise time is defined by the time constant of the circuit τ , and in this circuit $\tau = R1 \cdot C = 1\,\mu s$

Size the risetime for 1mS and give the simulation result=>

To size the rise time to 1ms, we change the capacitor value to be $1\mu F$ and adjust the pulse duration to keep supplying enrgy for a proper time to observe the rise time. We also change the transient analysis duration.
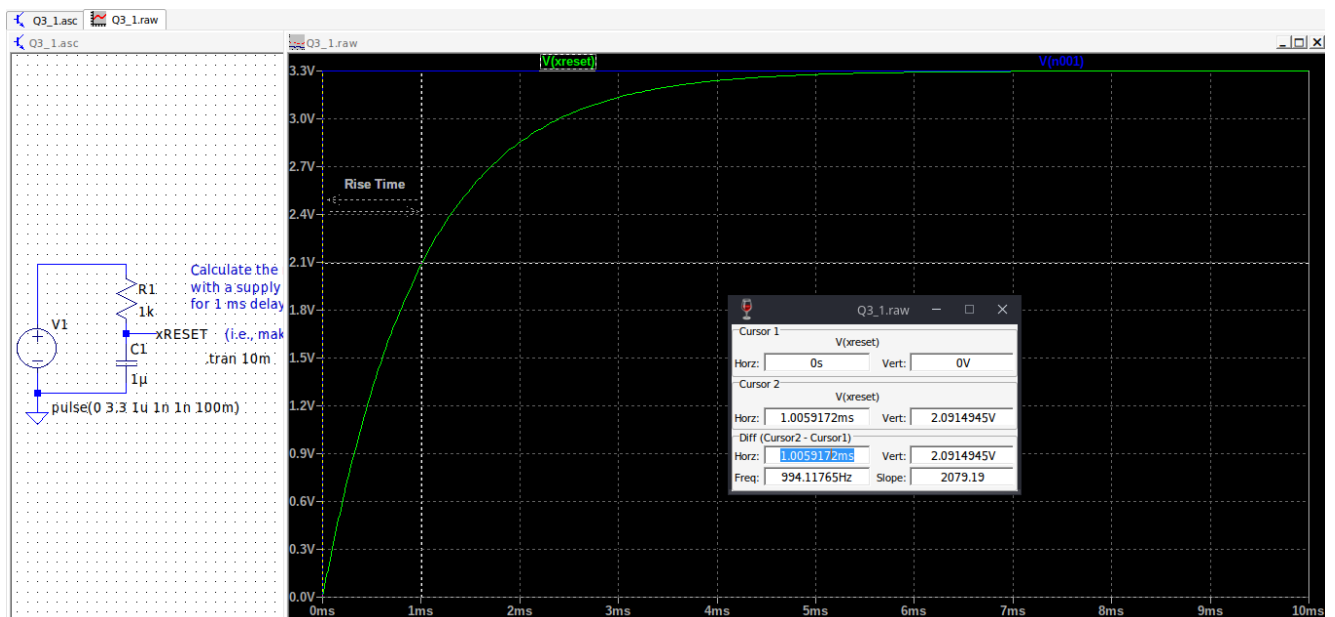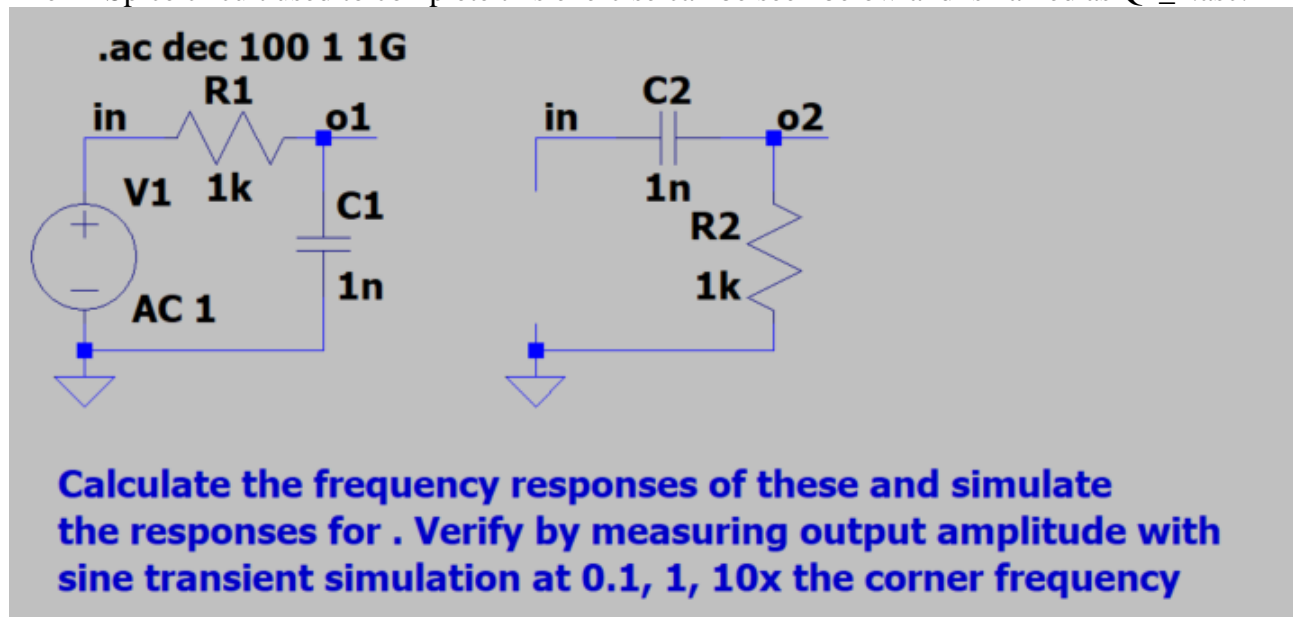
Task 4. Frequency response

Here we are simulating and calculating frequency responses of different circuits

1. Pre-material for this task can be found from Moodle on section => Pre-material for tasks
2. Simulation files for this task can be found from Moodle on section => Simulation files => Task 4. Download all files.

a.   RC and CR-responses

The LTSpice circuit used to complete this exercise can be seen below and is named as Q4_1.asc:

Calculate the frequency responses of these and simulate the responses for . Verify by measuring output amplitude with sine transient simulation at 0.1, 1, 10x the corner frequency

Calculate the frequency responses of both circuits (amplitude and phase). Hint: Impedance of C is $Z_c = 1/sC$.

Answer:

Applying the voltage divider rule in the first circuit with a Laplace domain, results in: $V_{o1} = \frac{Z_c}{Z_c + R1} V1 = \frac{1/sC}{1/sC + R1} V1 = \frac{1}{1 + sC \cdot R1} V1$

Setting $s = j\omega$, gives: $V_{o1} = \frac{1}{1 + j\omega R1C} V1$

Hence: $|V_{o1}| = \frac{1}{\sqrt{1 + \omega^2 R1^2 C^2}}$ and $\angle V_{o1} = -\tan^{-1}(\omega R1C)$

Applying the voltage divider rule in the second circuit with a Laplace domain results in: $V_{o2} = \frac{R2}{Z_c + R2} V1 = \frac{R2}{1/sC + R2} V1 = \frac{sCR2}{1 + sC \cdot R2} V1$

Setting $s = j\omega$, gives: $V_{o2} = \frac{j\omega R2C}{1 + j\omega R2C} V1$

Hence: $|V_{o2}| = \frac{\omega R2C}{\sqrt{1 + \omega^2 R2^2 C^2}}$ and $\angle V_{o2} = 90° - \tan^{-1}(\omega R1C)$

Simulate the circuits and define the pole/corner frequencies (give also the frequency response waveforms)

Answer:
The corner frequency is where the power at the output is equal to half of the power at the input (thus it's defined by the 3dB frequency/bandwidth of the circuit).

For the output voltage this means that the corner frequency $f_c : |V_o| = \frac{V1}{\sqrt{2}}$

For the first circuit, we get: $\omega_{c1} = \frac{1}{R1C} \Rightarrow f_{c1} = \frac{1}{2\pi R1C} = 159.15\ kHz$

For the second circuit, we get: $\omega_{c2} = \frac{1}{R2C} \Rightarrow f_{c2} = \frac{1}{2\pi R2C} = 159.15\ kHz$

From the simulation results below, both corner frequencies are $f_{c1} = f_{c2} \approx 160\ kHz$
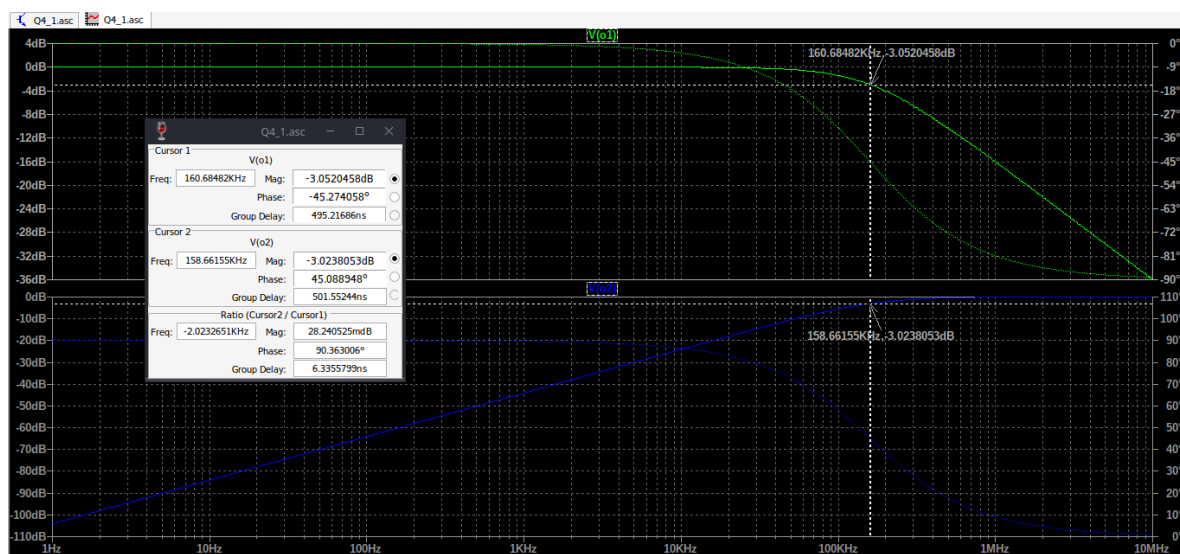
 Set the transient simulation with the proper stop time and use sinewave as a input source with 3 different frequencies 0.1*f_corner, f_corner and 10*f_corner and 1V amplitude. Give the out voltages o1 and o2 with all frequencies to table: (the output voltage here is the peak voltage)

	0.1*fcorner	fcorner	10*fcorner
o1	991.52907 mV	665.25481 mV	100.38972 mV
o2	99.543968 mV	692.58009 mV	967.42752 mV

b.   RLC-response

The LTSpice circuit used to complete this exercise can be seen below and is named as Q4_2.asc:
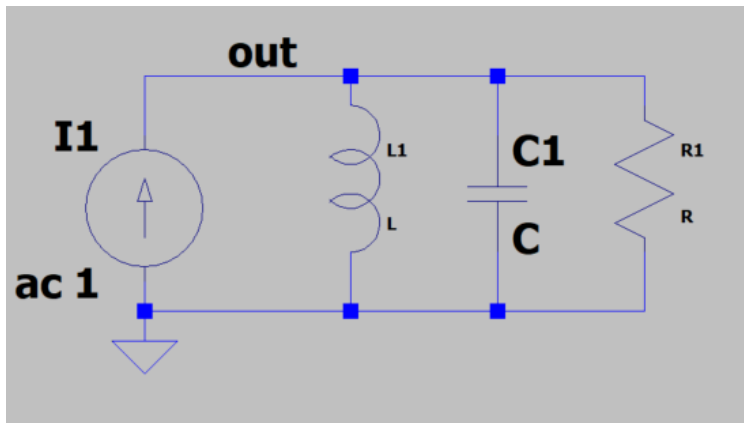
Dimension this for resonance freq of 100kHz (f_0) and Q of 10. Hint: Solve out/I1? Give the Transfer function and the values of R C and L to achieve $f_0 = 100$kHz and Q =10.

Answer:

$V_{out} = I1 \cdot Z; Z = R \parallel Z_C \parallel Z_L = R \parallel sL \parallel \frac{1}{sC} = \frac{1}{\frac{1}{R}+\frac{1}{sL}+sC} = \frac{sRL}{R+sL+s^2RLC}$ The transfer function is:

$TF(s) = \frac{V_{out}}{I1} = Z = \frac{sRL}{R+sL+s^2RLC}$

The resonance occurs when the imaginary impedances cancel out, i.e: $|Z_C| = |Z_L| \Rightarrow \frac{1}{\omega_0 C} = \omega_0 L \Rightarrow \omega_0 = \frac{1}{\sqrt{LC}}$
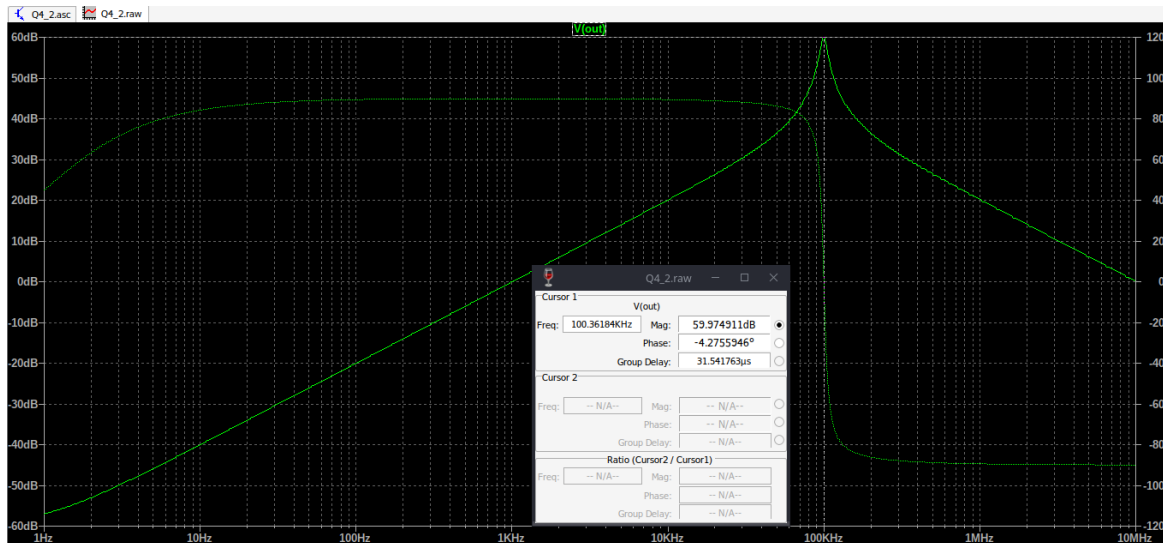
The Q factor of such a parallel circuit is: $Q = \frac{f_0}{BW} = \frac{R}{Z_{C0}} = \frac{R}{Z_{L0}} = RC\omega_0 = \frac{R}{\omega_0 L}$

Therefore, by fixing $R = 1k\Omega$, we get: $C = \frac{Q}{R\omega_0} = \frac{10}{2\pi 100 \cdot 10^3 \cdot 10^3} = 15.92 \, nF$ and $L = \frac{R}{Q\omega_0} = \frac{10^3}{2\pi 100 \cdot 10^3 \cdot 10} = 159.15 \, \mu H$
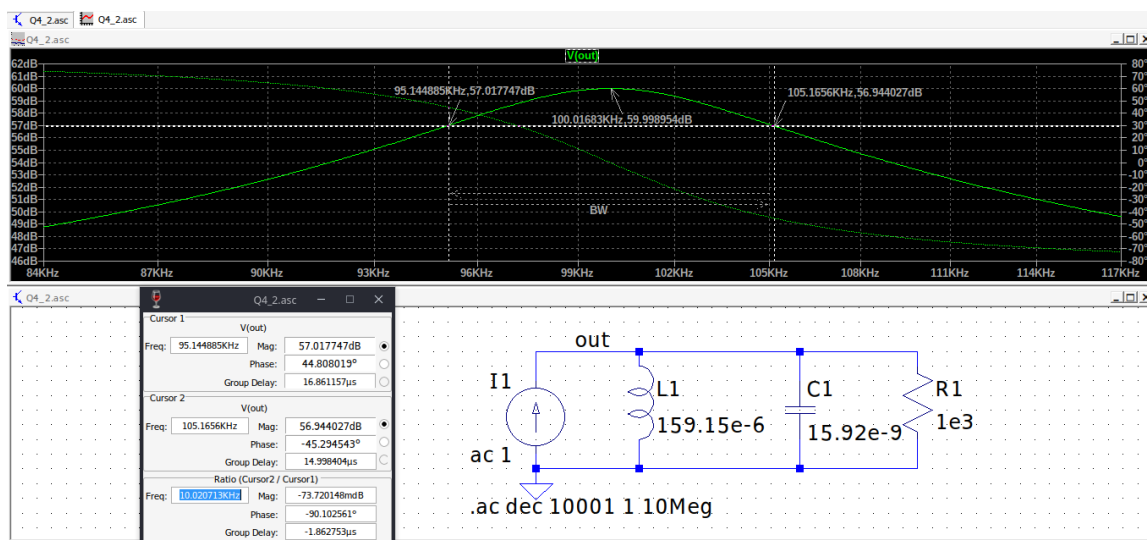
Verify by ac simulation measuring center frequency and -3dB bandwidth. Give the waveform showing the values. Use cursors=>

Answer:

Below is the waveform of the simulation.

This is a zoomed-in view of the waveform with cursor showing the upper and lower 3dB frequencies and the highlighted value in the cursor menu show the 3dB frequency of: $f_{3dB} = \frac{f_0}{Q} = \frac{100 \cdot 10^3}{10} = 10 \, kHz$


Excite the circuit in transient analysis by a 1us pulse, and count the oscillations cycles you see. Do they match the Q value?

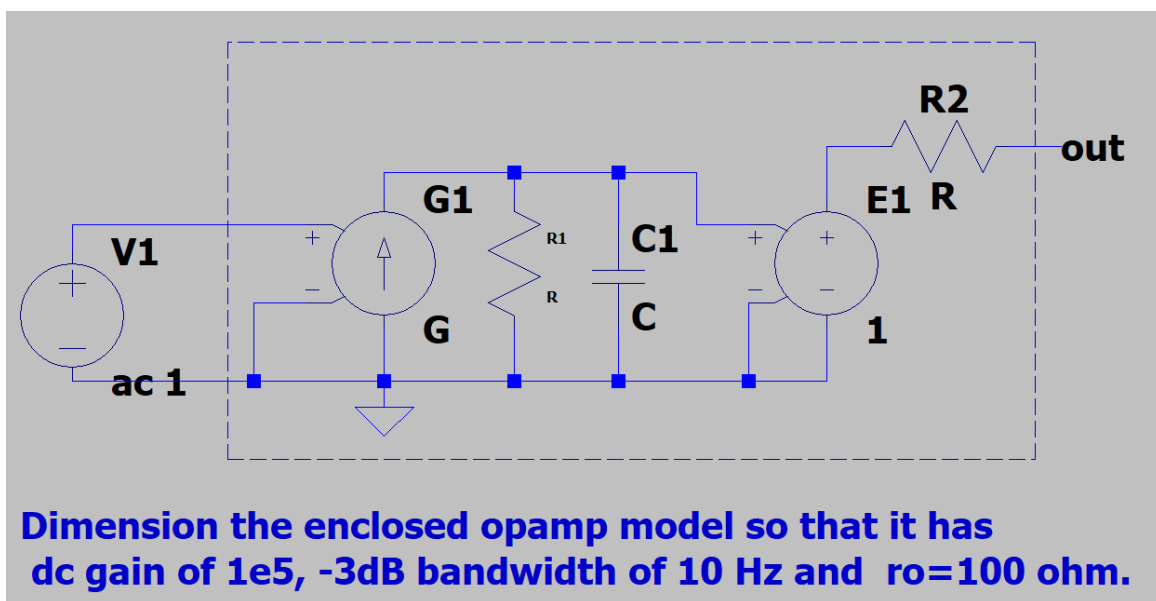**Answer:**
Yes, the oscillation cycles match the Q value.

In $100\mu \sec$, there are 10 cycles giving $10\mu \sec/cycle$ and thus a frequency of $100 \, kHz$

Dimension this for resonance freq of measuring center frequency and -3d
Excite the circuit in transient analys you see. Do they match the Q value?
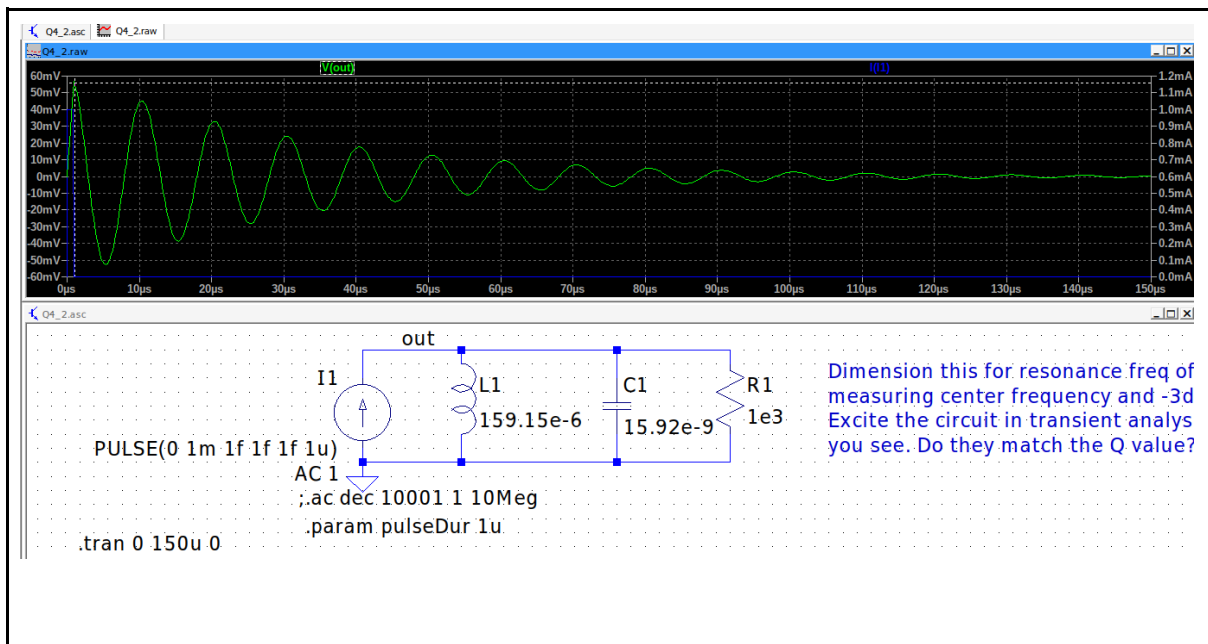
c.   Frequency response of an operational amplifier

The LTSpice circuit used to complete this exercise can be seen below and is named as Q4_3.asc:

Dimension the enclosed opamp model so that it has dc gain of 1e5, -3dB bandwidth of 10 Hz and ro=100 ohm.

Size the values R2, G, R1 and C1 to achieve the DC gain of OpAmp A_o = 100 000V/V, -3db bandwidth of 10Hz and output resistance of OpAmp of 100 Ohm. Give the values:

Answer:
In DC, the capacitor is an open circuit, this makes the DC gain solely determined by G and R1.

$$A_0 = \frac{E1}{V1} = \frac{G \cdot R1 \cdot V1}{V1} = G \cdot R1$$

Setting $R = 10 \, k\Omega$, gives: $G = \frac{A_0}{R1} = \frac{10^5}{10^3} = 100 \, \Omega^{-1}$

The lowpass filter (R1 and C1 combination) has a corner frequency of: $f_c = \frac{1}{2\pi R1 C}$

Therefore: $C = \frac{1}{2\pi f_c R1} = 15.91 \, \mu F$

The output impedance is solely defined by R2 since the output is driven by voltage dependent voltage source, hence, $R2 = R_{out} = 100 \, \Omega$

Simulate AC-simulation to verify your sizing. Give the frequency response waveform=>

Answer: