Creator: Wojciech Rościszewski-Wojtanowski
Index No.: 140062

CAD LAB
Date of Exercise:
19 November 2020
Date of report:
19 November 2020

Exercise 5.

Statistical Analysis.

Table of Contents

List of Figures

Creator: Wojciech Rościszewski-Wojtanowski
Index No.: 140062

CAD LAB
Date of Exercise:
19 November 2020
Date of report:
19 November 2020

Abstract

The aims of this laboratory exercise were learning how to define statistical analyzes Monte Carlo (MC) and Worst Case (WC) in Ltspice and interpret simulated data into histogram.

Creator: Wojciech Rościszewski-Wojtanowski
Index No.: 140062

CAD LAB
Date of Exercise:
19 November 2020
Date of report:
19 November 2020

Part 1: Non-inverting Amplifier Analysis

For begin our exercises we will conduct an analysis of the amplifier circuit for nominal values. Therefore to start we must firstly create an amplifier circuit. In the figure below please see such circuit scheme, find the operational amplifier used being OP37. We have a sinusoidal input. Using the parameters we can measure the precise or ideal voltage gain from this circuit. Realistically these values will be a little different as each component as well as the conditions will affect the circuit.

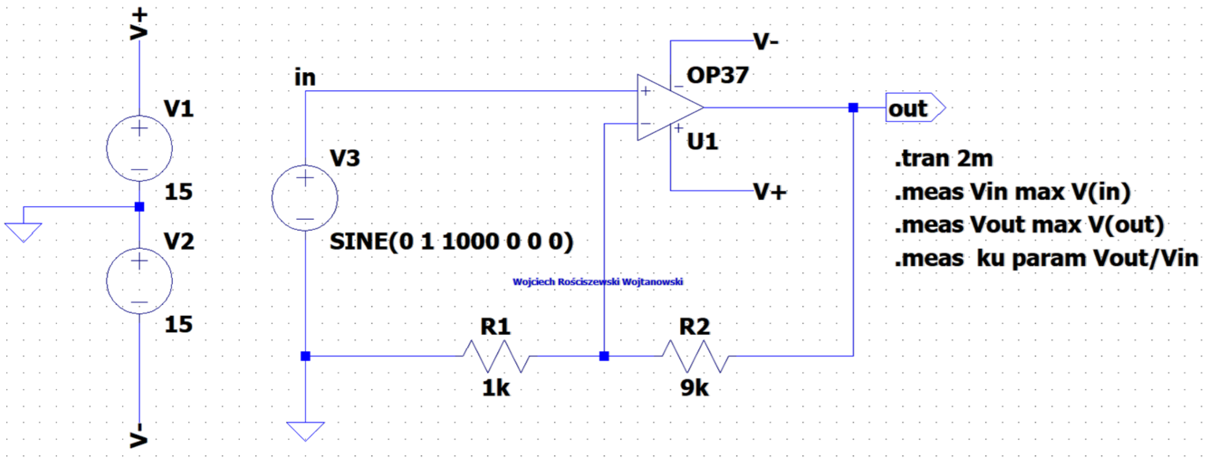

Figure 1 Diagram of the Tested Amplifier Circuit.

We performed transient analysis and measured all the minimal and maximal values from which we can calculate the gain coefficient ku. Our general voltage source is sinusoidal input, we determine the amplitude of 1v. This is for ideal situation, when we have 1k and 9k, so not taking into consideration other factors influencing this. The gain of this circuit should be 10V. In the below please see the plot characteristic, we see the blue line is our input signal from voltage source V3. The green line is our output characteristic. We clearly see our signal is amplified, but by how much?

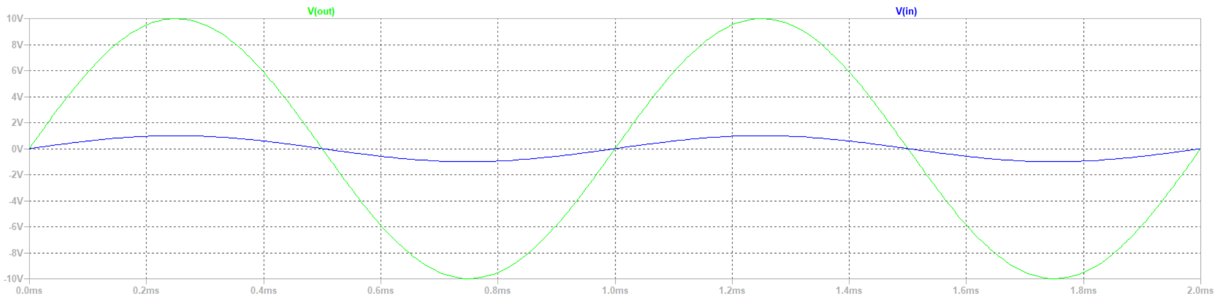

Figure 2 Plot Characteristic.

In the below please find a figure presenting the previously mentioned gain calculation. Please see we calculated this from comparing the Vin and Vout voltage values, all of this has been only done by the spice directives .meas so our calculations should be considered very precise as they come directly from the simulation. We see the gain value is almost close to 10V of original signal, which seems quite accurate.

```
vin: MAX(v(in))=0.998893 FROM 0 TO 0.002
vout: MAX(v(out))=9.98853 FROM 0 TO 0.002
ku: vout/vin=9.99959
```

Figure 3 .meas calculations.

For the next part of the exercise we will perform the Monte Carlo (MC) analysis of the amplifier circuit previously created. In the below please see the completed circuit. Please see that the tolerance of the resistive components has been set to 10%. We did this by setting a variable to 10% and then expanding it onto each resistive element requiring such tolerance assumption. Please see tol variable under resistors and tol parameter setting.

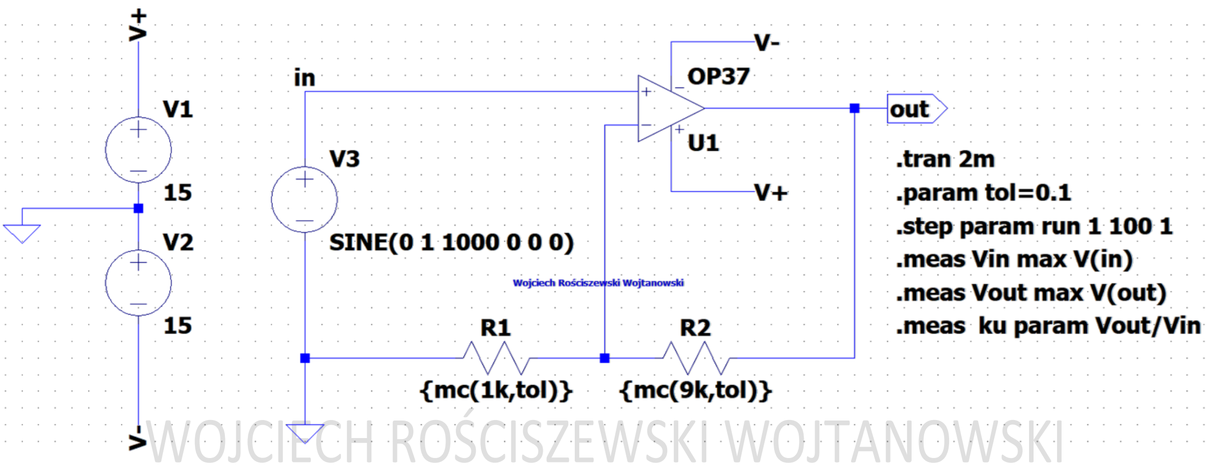

Figure 4 Scheme of the Tested Amplifier Circuit for MC analysis.

In the below please see the output. Circuit is working correctly. There are 100 steps analyzed in this simulation.

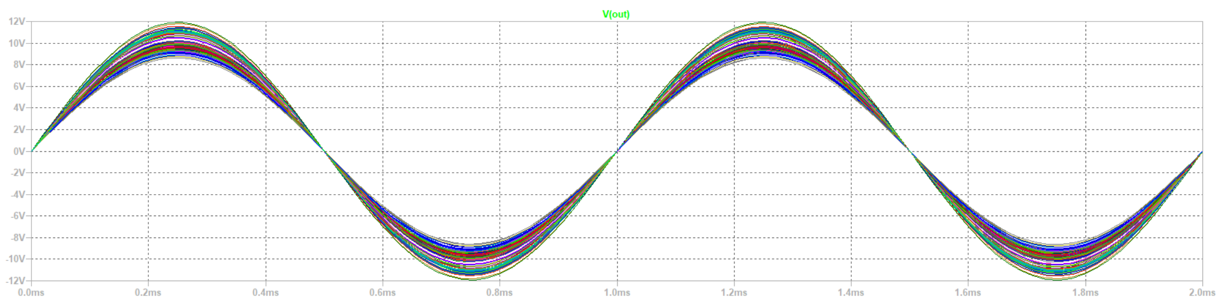

Figure 5 Plot Characteristic.

For the next part of the exercise we must determine the ranges of which the gain changes towards. Please see the measured corresponding values for this. Measurement of ku gain values:

STEP	VOUT/VIN
1	11.1239
2	11.1797
3	9.81352
4	11.0121
5	9.8707

6	11.3182
7	9.65939
8	9.45819
9	10.5348
10	10.0356
11	9.10949
12	9.77667
13	10.7361
14	10.0703
15	10.0091
16	10.9578
17	9.82024
18	11.0871
19	10.0324
20	9.6089
21	9.74966
22	9.94979
23	9.29053
24	9.58835
25	8.90424
26	10.1087
27	10.0043
28	9.61392
29	10.0583
30	8.62372
31	9.47682
32	8.9201
33	11.816
34	9.99396
35	10.1823
36	9.81224
37	9.49488
38	9.52718
39	10.8912
40	8.90791
41	9.62224
42	9.78291
43	9.62513
44	10.1419
45	10.6898
46	10.8478
47	9.3038
48	10.9409
49	9.22731
50	9.66218
51	11.5528
52	11.4801

53	9.32392
54	9.34897
55	9.53301
56	11.3999
57	9.59436
58	9.97377
59	9.42089
60	10.673
61	9.43812
62	10.5441
63	10.1274
64	9.57458
65	10.4761
66	8.65068
67	10.2614
68	9.41221
69	9.95341
70	9.63681
71	9.82447
72	9.10734
73	9.66509
74	8.87496
75	9.70429
76	11.083
77	9.25495
78	10.7001
79	11.9221
80	9.07332
81	9.39636
82	9.63529
83	9.19733
84	8.7886
85	11.2358
86	11.1718
87	10.0941
88	11.1494
89	9.78598
90	10.2589
91	9.79462
92	10.1183
93	10.0783
94	9.81814
95	9.03971
96	10.6643
97	9.88638
98	9.16169
99	9.57161

100	9.29661

Table 6 Gain Measurements.

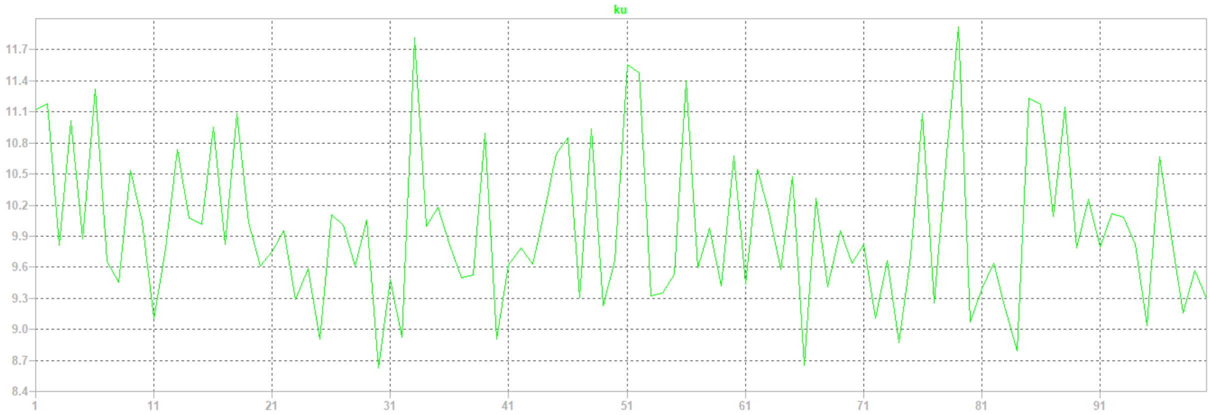

Figure 7 stepped measurements of gain.

Average gain value of our output port is 9.977006 [V].

Below please find the distribution of our gain of our produced amplifier.

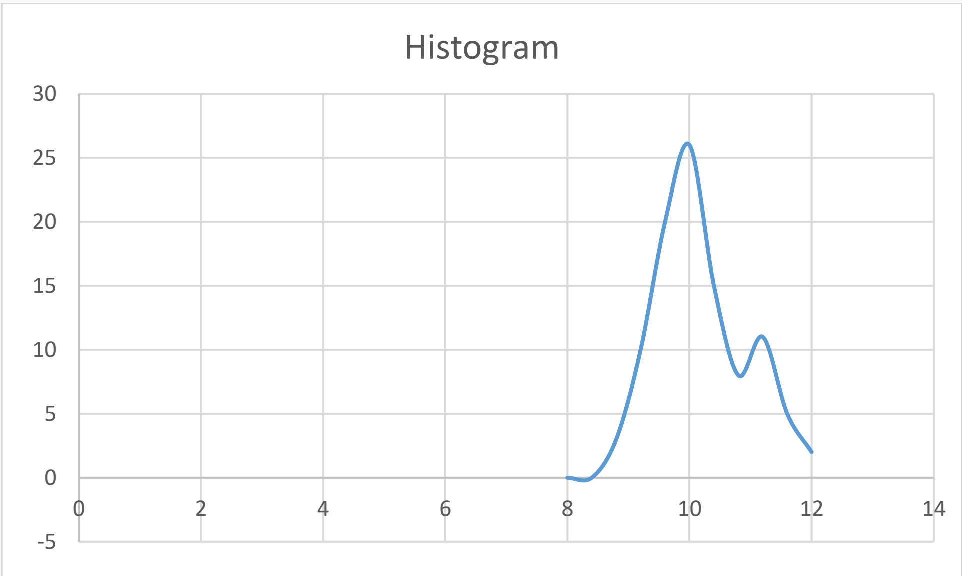

Figure 8 Graph of the histogram.

Below please find a closed up:

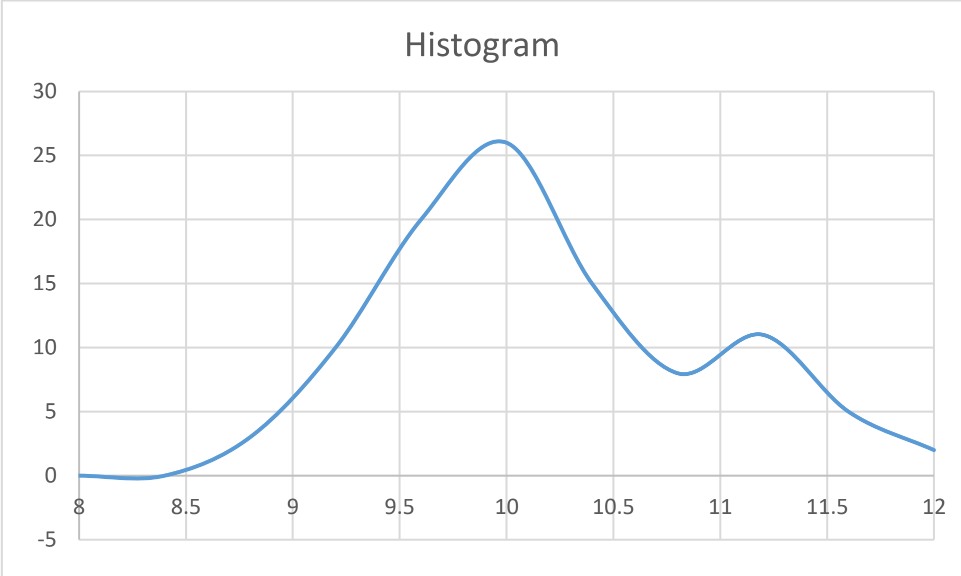

Figure 9 Graph of the histogram Close Up.

So, when performing the Monte Carlo simulation we change value of resistor, so nominal value for example can be 1kΩ. If we simulate with tolerance included, such as 10%. Our value of our resistor can change from values 0.9kΩ to 1.1kΩ. This is called gaussian distribution, in the below please find a figure presenting the described. We see the values such as 1kΩ appearing more often, whilst values such as 0.9kΩ/1.1kΩ occurs less often. In LTspice we cannot really control this, in other program we can control these by changing the type of the distribution, as there are more available. Generally, we use gaussian as it is somewhat a norm or default option even in other programs.

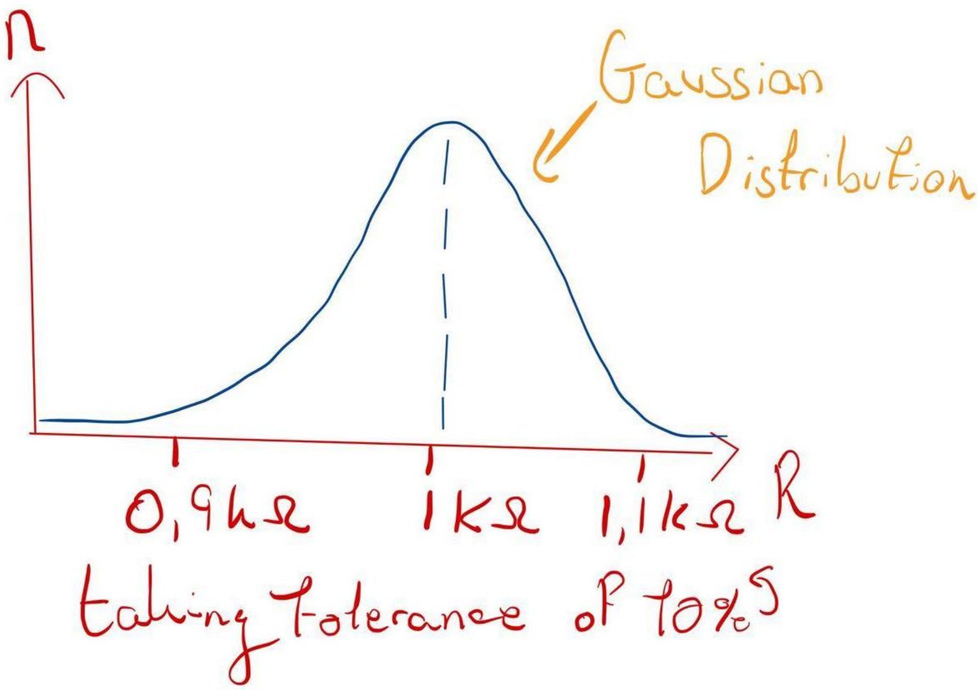

Figure 10 Gaussian Distribution in Tolerance

The next part of the exercise is to modify the previous circuit and perform the worst case (WC) analysis of the amplifier. Please find the figure below presenting the circuit diagram.

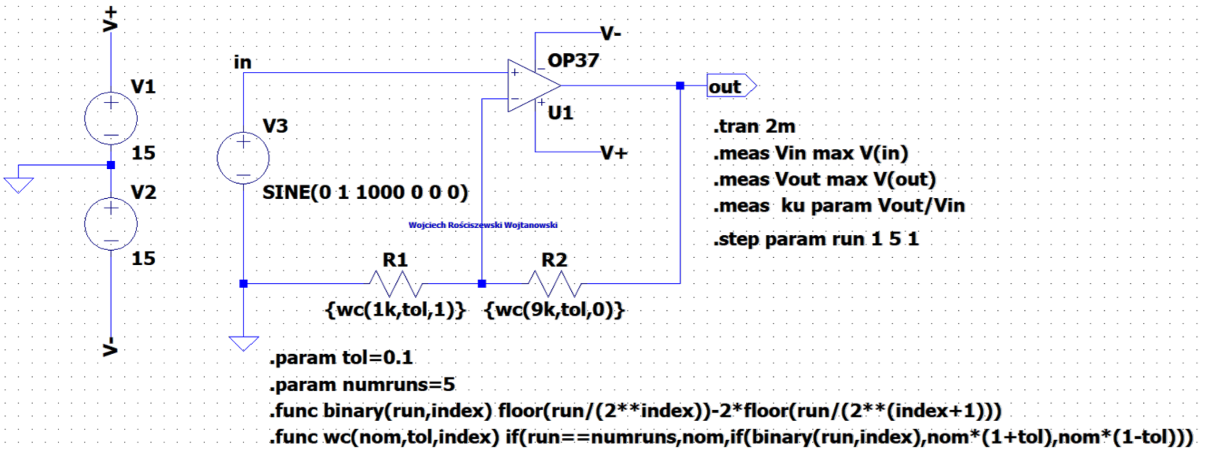

Figure 11 Diagram of the Tested Amplifier Circuit for WC analysis.

In the below please see the circuit output. We see our five steps, where only thee are visible.

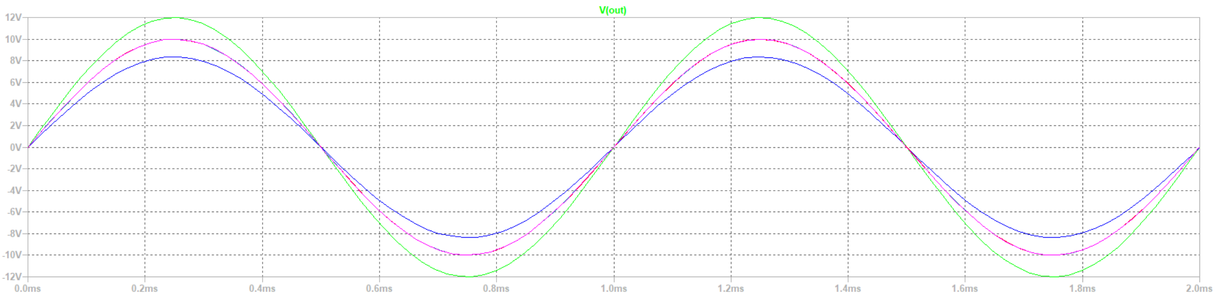

Figure 12 Plot Characteristic.

We must determine the gain characteristic of this therefore please see the output of the .meas directive. We see that we have a gain by comparing the input and output voltages.

STEP	VOUT/VIN
1	11.9996
2	8.36328
3	9.99957
4	9.99959
5	9.99959

Table 13 Gain Measurement

Average gain in our output is: 10.072326 [V].

Tolerance was 10%, tol=0.1. Gain is ku=8, till ku12. We run simulation for 100 steps.

For the next part we will change the tolerance of the resistors to 1%. In the below please see modified circuit where only the tolerance parameter was modified.

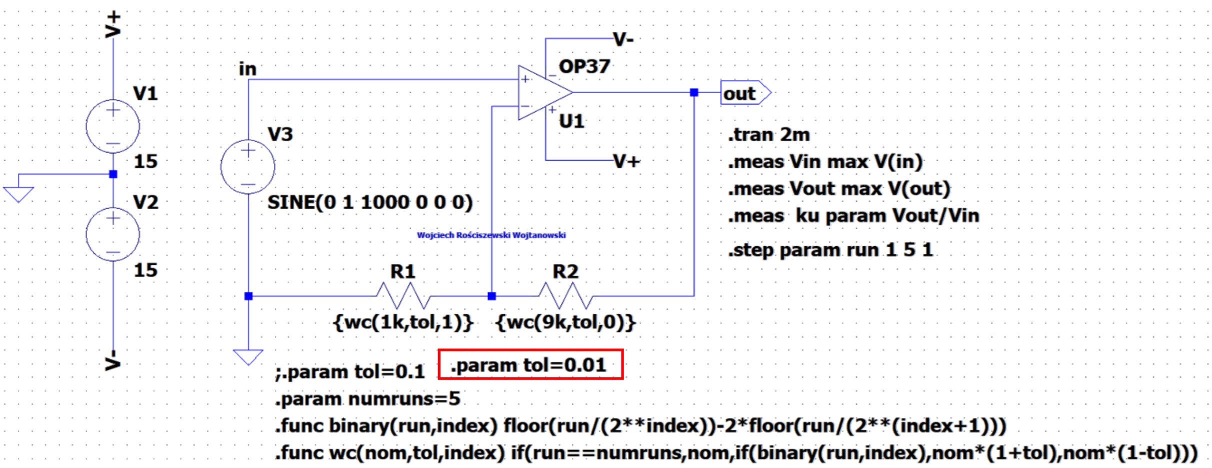

Figure 14 New Diagram of the Tested Amplifier Circuit for WC analysis.

In the below please see the output of the circuit, we see that the tolerance of the resistor affects the circuit state output greatly.

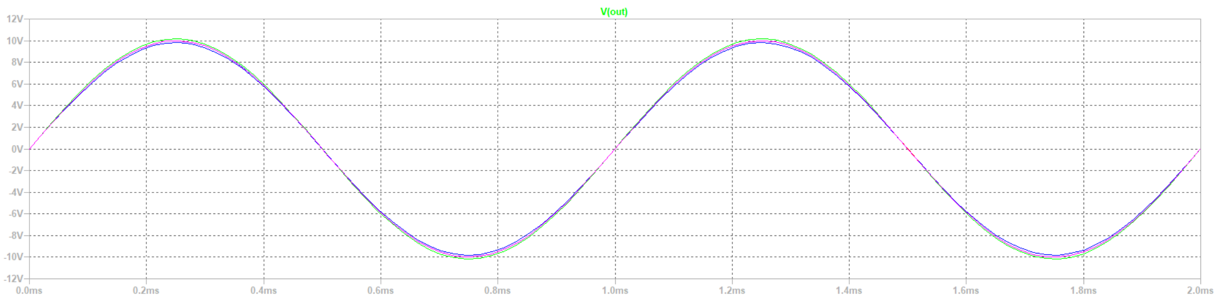

Figure 15 New Plot Characteristic.

Since out output waveform has changed, we know the gain has also changed therefore, please see the below table as well as calculated average.

STEP	VOUT/VIN
1	10.1814
2	9.82138
3	9.99959
4	9.99959
5	9.99959

Figure 16 New Gain Measurements

Average gain in our output is: 10.00031 [V]. Previously we had, 10.072326. Our system is more precise I assume, since we are getting closed to 10V!

For the next part of the exercise we will determine the worst-case analysis of our amplifier circuit but this time we will take into account the temperature limits at which the tested system (and elements) can operate in between the temperatures of -55°C and 125°C. In the below please see the modified circuit and directives that suit this amplifier circuit characteristics.

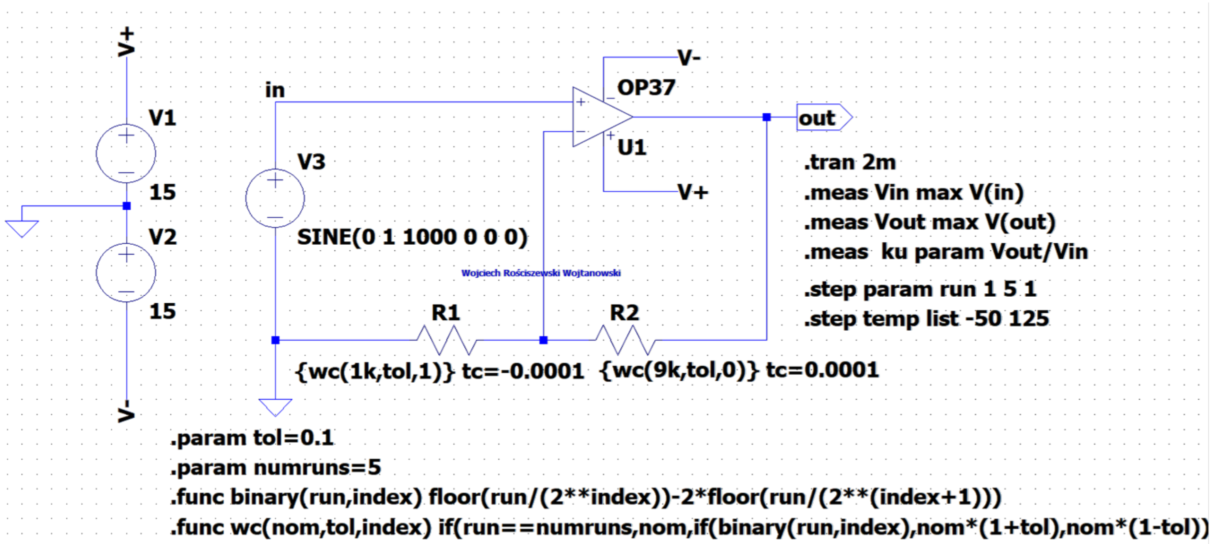

Figure 17 Diagram of The Tested Amplifier Circuit for WC Analysis Taking into Account Temperature Changes.

In the below please see the output of our circuit

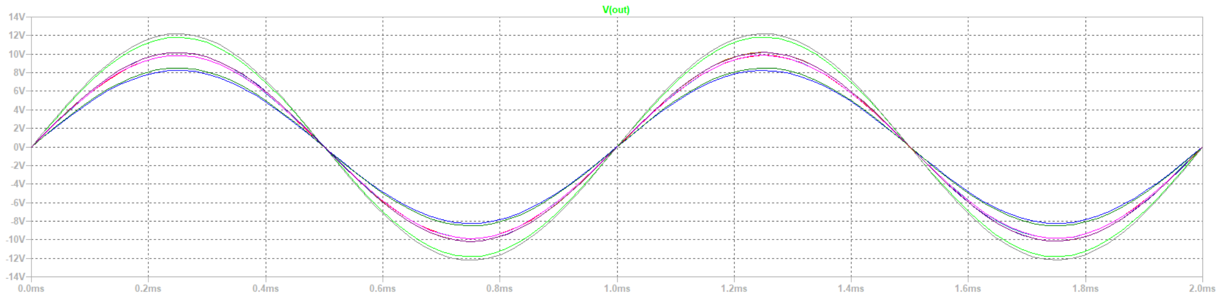

Figure 18 Plot Characteristic.

In the below please see the gain measurements of our output values from circuit along with the average gain value.

STEP	VOUT/VIN
1	11.8314
2	8.25075
3	9.86204
4	9.86206
5	9.86206
6	12.2173
7	8.50904
8	10.1777
9	10.1777

10	10.1777

Table 19 Gain Measurements

Average gain in our output is: 10.092775 [V]. From all of the information we can compare from previous circuits that the temperature has minimalistic effects on the results of given circuits. In the case of very sensitive electrical circuits such as radio heads the temperature can greatly affect these elements, as a radio receiver circuit will be out of tune and will change it’s tuning, for example in a famous polish radio amplifier Radmor 51XX. In this receiver I personally seen the radio escape from its true parameters because of a diode heating up. Temperature of a circuit is extremely important as it affects the scheme parameters, again implies for only really sensitive circuits.

Part 2: MC Analysis of the Comparator with Feedback

For this part of the exercise we will perform analysis on the comparator system for only nominal values. Therefore, we must create a comparator system with a feedback. In the below figure please see such circuit scheme presented.

Yes, I have created my own component with the information I have gathered from the internet that's why the LT1021-5 module has a different symbol. It is exactly the same as the original LT1021-5 component. This was created as I couldn't find the module in LTspice component search, so I decided to improvise. I have later found the module in the References folder. This clearly shows that the previous labs have been sufficient enough for me to improvise during such situations.

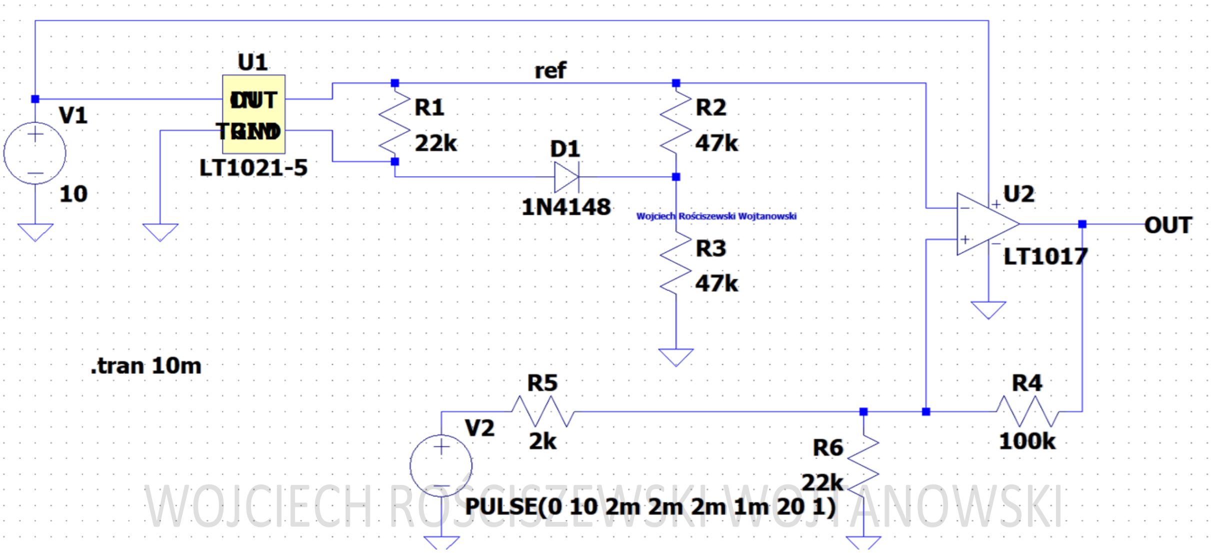

Figure 20 Diagram of the Tested Comparator System.

In the below please see the output of our circuit.

Figure 21 Plot Characteristic.

Please see the green and trapezoidal shape being our two inputs, with one being our steady voltage (V1) whilst the other being our pulse (V2). The square wave in blue is our output value characteristic. We must now calculate the hysteresis voltage. Hysteresis else known as the Schmitt trigger circuit. It means that once the output is triggered into the saturation phase there will be some change to the input signal before our output will be switched back at the original saturation phase point. We must calculate this difference here. We are working with a non-inverting operation amplifier. In more detail the LT1021-5 V(ref)=5V whilst the output signal of our V2 voltage source is equal to zero and then it rises for the first 2ms (pulse function), the voltage output of comparator depends on V2 and comparator switching voltage, so 0 -10V.

Below is the gain of this circuit: **vin:**
MAX(v(in))=9.18068 FROM 0 TO 0.01
vout: MAX(v(out))=9.94579 FROM 0 TO 0.01
ku: vout/vin=1.08334

From calculating the V_{UTP} (upper voltage trip point) and V_{LTP} (lower voltage trip point). We in fact don't calculate this but rather find the correct points on the graph. Since we are comparing the difference between the output and input, how the system changes the values! Find circled in blue exactly what is meant. $V_{HYS}=V_{UTP}-V_{LTP}$ = 5.046-4.769 = 0.277.

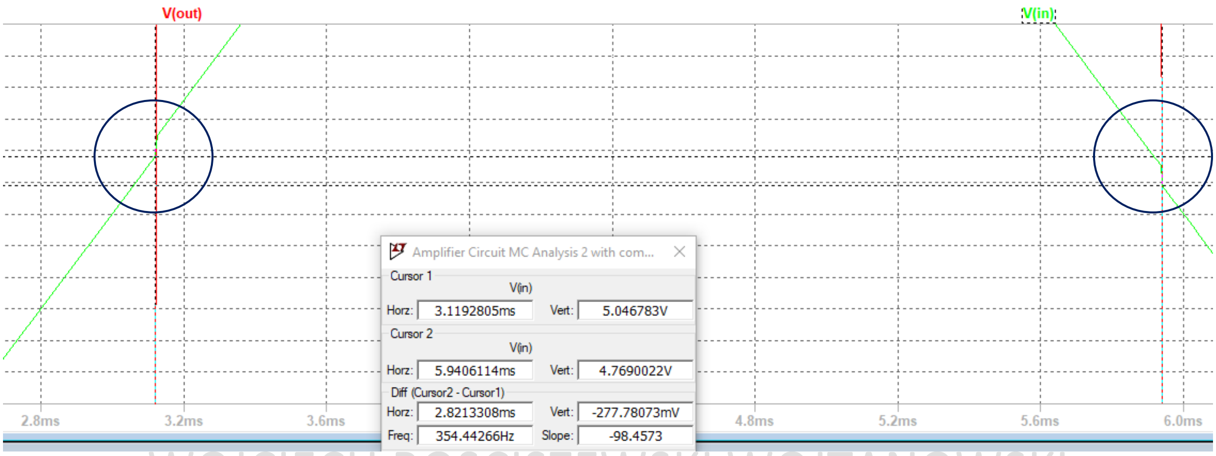

Figure 22 Hysterisis.

The next step of the exercise is to modify the circuit to run a parameter sweep, also change the resistor values to 10% and see how it affects the system. Please see the figure below presenting comparator system for MC analysis.

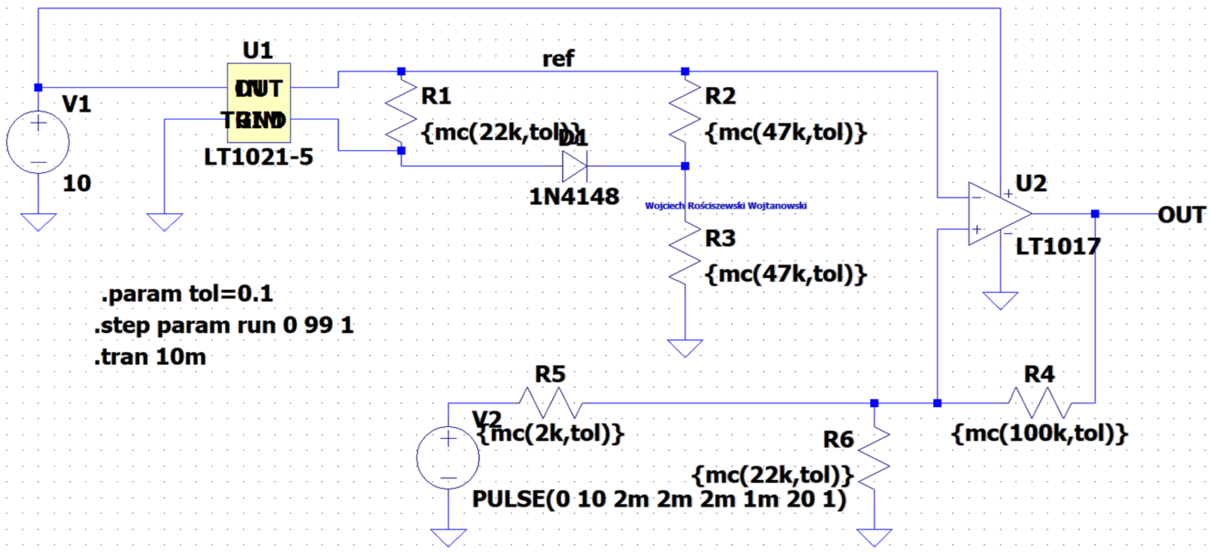

Figure 23 Diagram of the tested comparator system for MC analysis.

In the below please see the output of our circuit

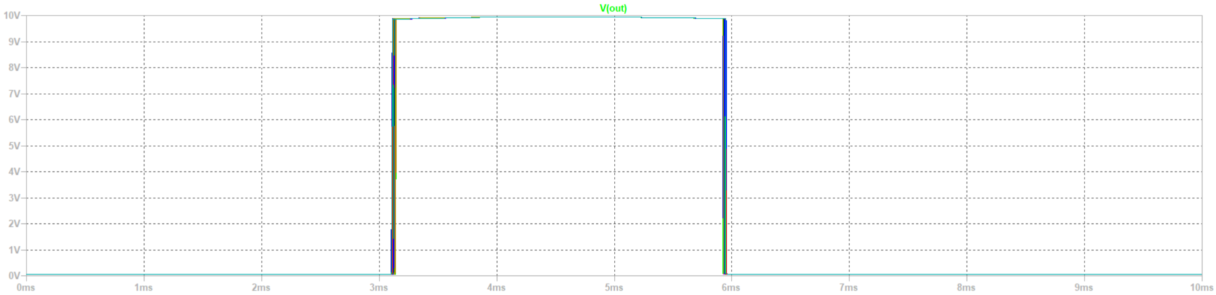

Figure 24 Plot Characteristic.

We must calculate this difference here. From calculating the V_{UTP} (upper voltage trip point) and V_{LTP} (lower voltage trip point). We in fact don't calculate this but rather find the correct points on the graph. Since we are comparing the difference between the output and input, how the system changes the values this time we have a tolerance change!

So since we are stepping originally throughout this simulation I will create another histogram presenting this. This isn't asked directly in the manual however I assume that this should be included. Therefore, I have written a simple LTspice directive to calculate this for us, below please find two figures: the table presenting our hysteresis and our histogram. Additionally, I will include the average HYS value.

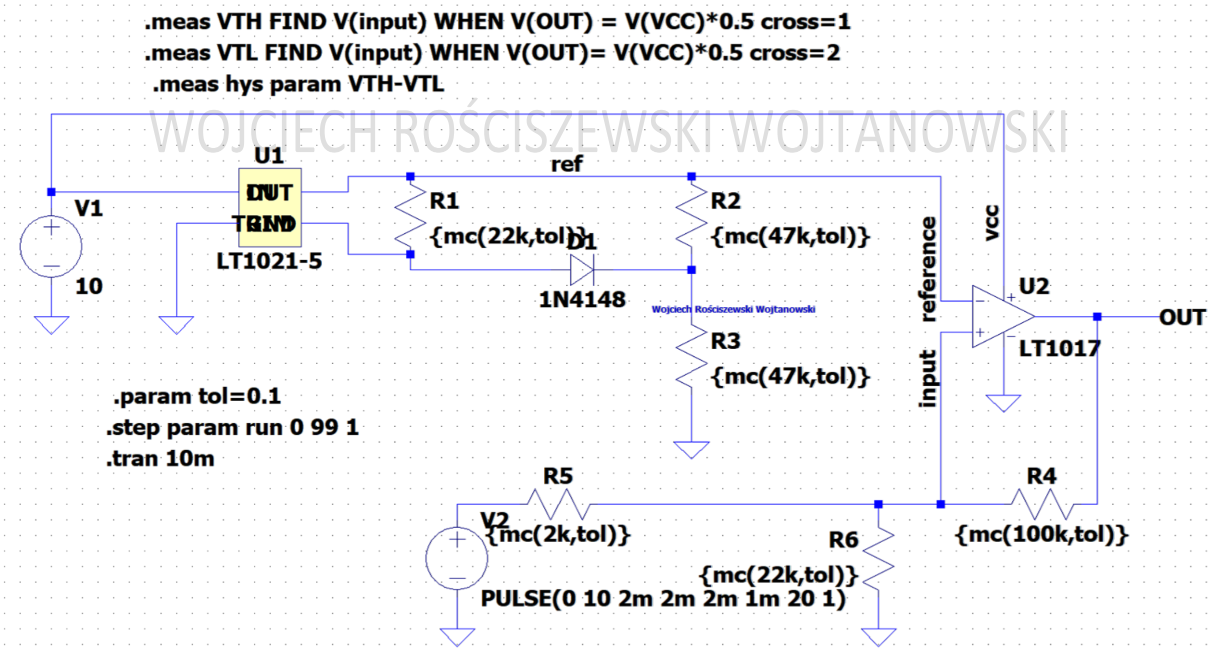

Figure 25 Modified Circuit.

STEP	VTH-VTL
1	0.273588
2	0.265754
3	0.284627
4	0.268473
5	0.282516
6	0.272701

7	0.301115
8	0.290674
9	0.286268
10	0.295619
11	0.282654
12	0.286096
13	0.28254
14	0.293057
15	0.289567
16	0.276301
17	0.271396
18	0.283554
19	0.278287
20	0.271672
21	0.285109
22	0.293704
23	0.313476
24	0.294985
25	0.296272
26	0.285363
27	0.297102
28	0.300539
29	0.261458
30	0.278303
31	0.308715
32	0.285021
33	0.292827
34	0.292751
35	0.273864
36	0.266467
37	0.293275
38	0.267612
39	0.28553
40	0.272909
41	0.260034
42	0.281906
43	0.281976
44	0.281167
45	0.271322
46	0.262311
47	0.269761
48	0.26941
49	0.278229

50	0.306378
51	0.272335
52	0.301464
53	0.285335
54	0.281333
55	0.286435
56	0.272193
57	0.275058
58	0.272685
59	0.286194
60	0.267475
61	0.282446
62	0.287336
63	0.278309
64	0.287562
65	0.288407
66	0.274462
67	0.292759
68	0.270445
69	0.30613
70	0.265433
71	0.275085
72	0.263268
73	0.266293
74	0.27198
75	0.270854
76	0.282543
77	0.280766
78	0.261999
79	0.309765
80	0.265268
81	0.308246
82	0.275439
83	0.260755
84	0.278635
85	0.259818
86	0.295068
87	0.296906
88	0.278311
89	0.285062
90	0.286613
91	0.272404
92	0.259337

93	0.260556
94	0.282317
95	0.284021
96	0.279038
97	0.265022
98	0.284822
99	0.278436
100	0.272122

Figure 26 HYS STEP Results

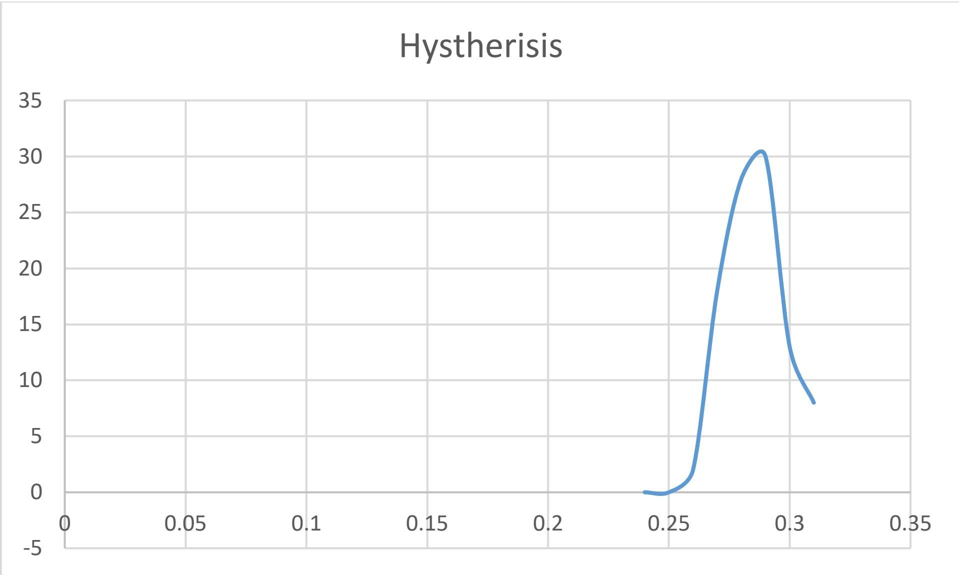

Figure 27 Histogram

Average Value: 0.280888 V.

So the switching voltages of upper threshold vary from 5.5V to 5.7V whilst the lower threshold voltage oscillates from 5.38V to 5.23V.

In the below please see the worst-case WC analysis, the circuit is modified for this as shown in the figure below.

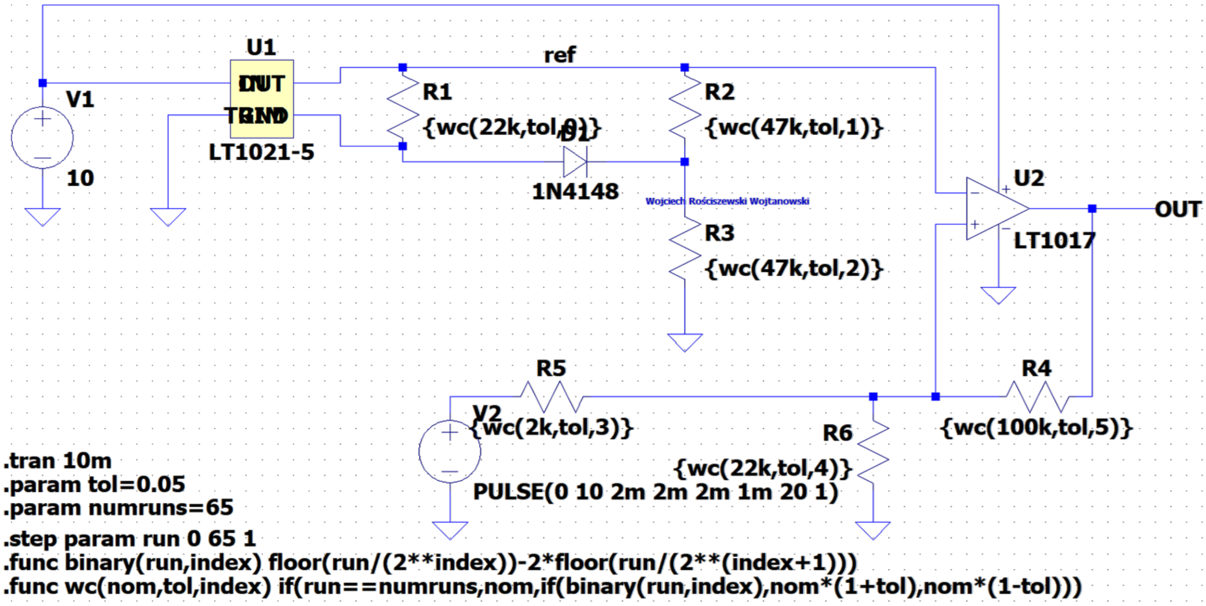

Figure 28 Diagram of The Tested Comparator System for WC Analysis.

In the below please see the output of our circuit

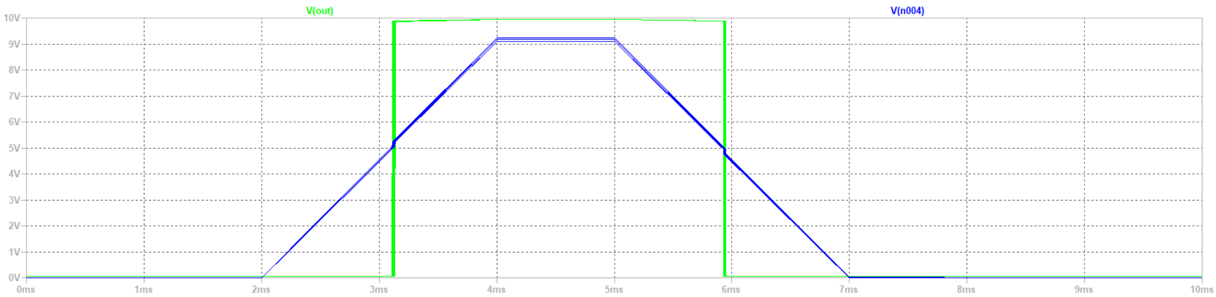

Figure 29 Plot Characteristic.

We see that the output of the system has also been affected since the upper threshold voltage varies from 5.5V to 5.6V whilst the lower threshold voltage varies from 5.3V to 5.2V.

Part 3: Analysis of Band-Pass filter

For the next part of the exercise we create a band-pass filter for nominals values. Below please find the filter design circuit scheme.

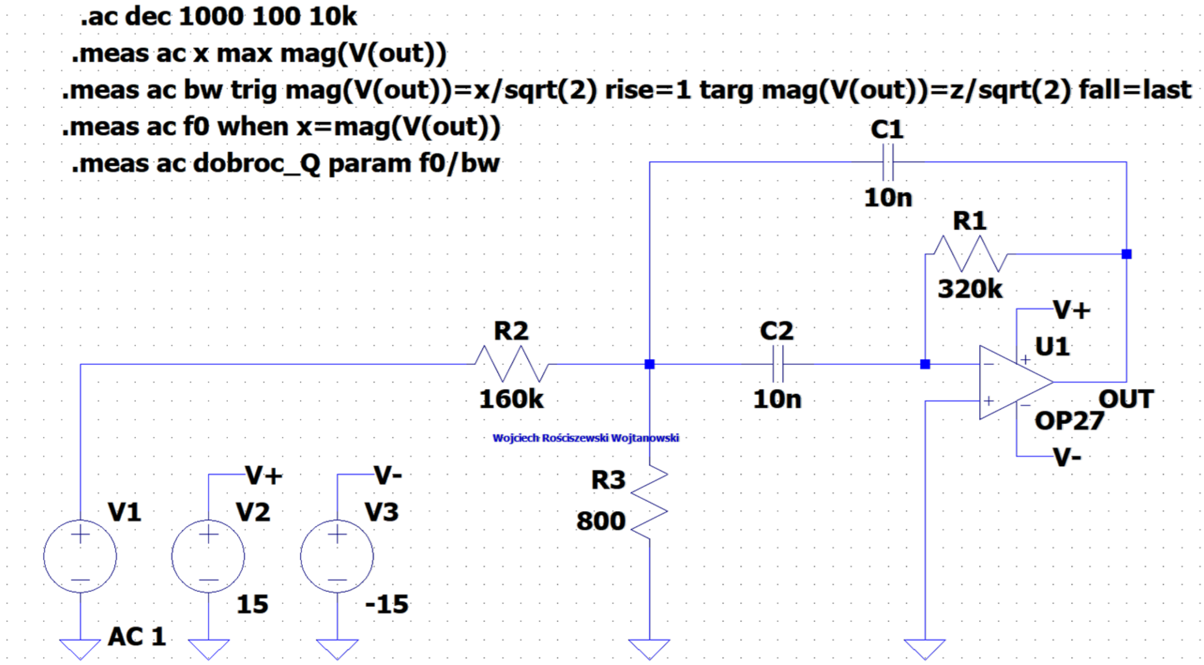

Figure 30 Diagram of the tested filter system.

In the below please see the output of our circuit

Figure 31 Plot Characteristic.

Below please find the quality filter Q characteristic according to the equation.

$$\boldsymbol{Q} = \frac{f_0}{f_g - f_d}$$

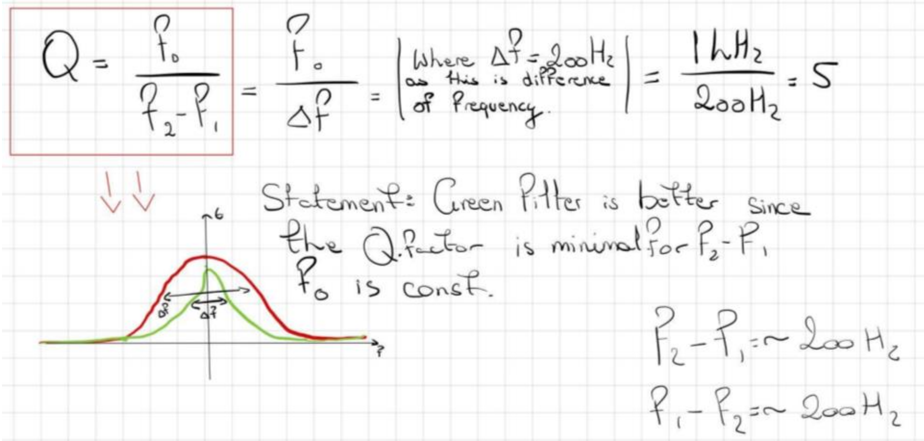

Figure 32 Quality Factor From Previous Exercise

So to calculate the quality, we know the following values.

f_0=995Hz ; f_g=1kHz ; f_d=946Hz

So: Q=9.65

For the next step of the exercise we must perform the MC analysis, taking into account the 10% tolerance capacitors and 5% tolerance for resistors.

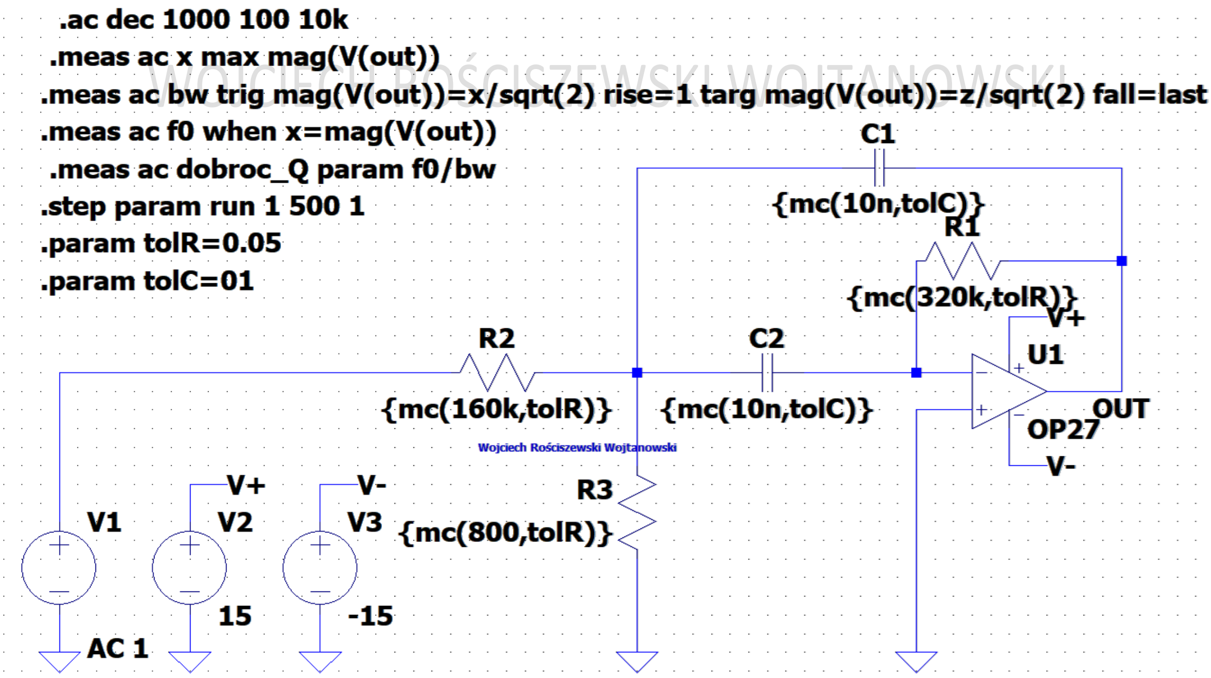

Figure 33 Diagram of the tested filter system for MC analysis.

In the below please see the output of our circuit

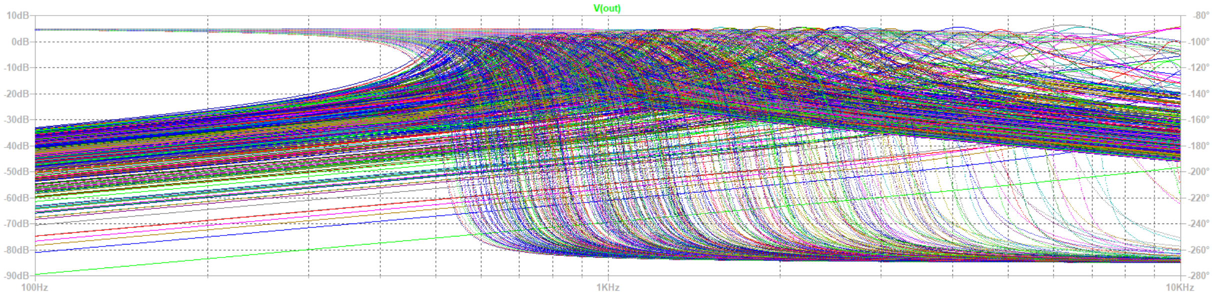

Figure 34 Plot Characteristic.

Max f_0=1127 Hz

Min f_0=881 Hz

Central Frequency:

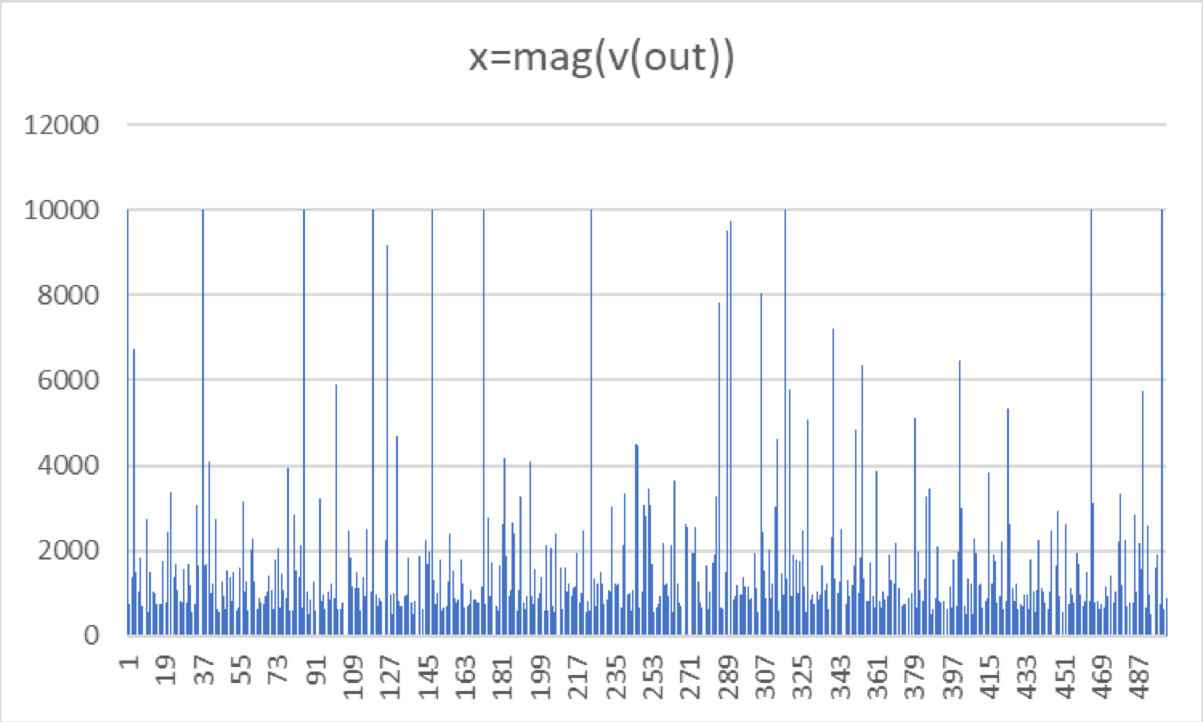

Figure 35 x=mag(vout)

For the next step of the exercise we must perform the WC analysis, taking into account the 10% tolerance capacitors and 5% tolerance for resistors.

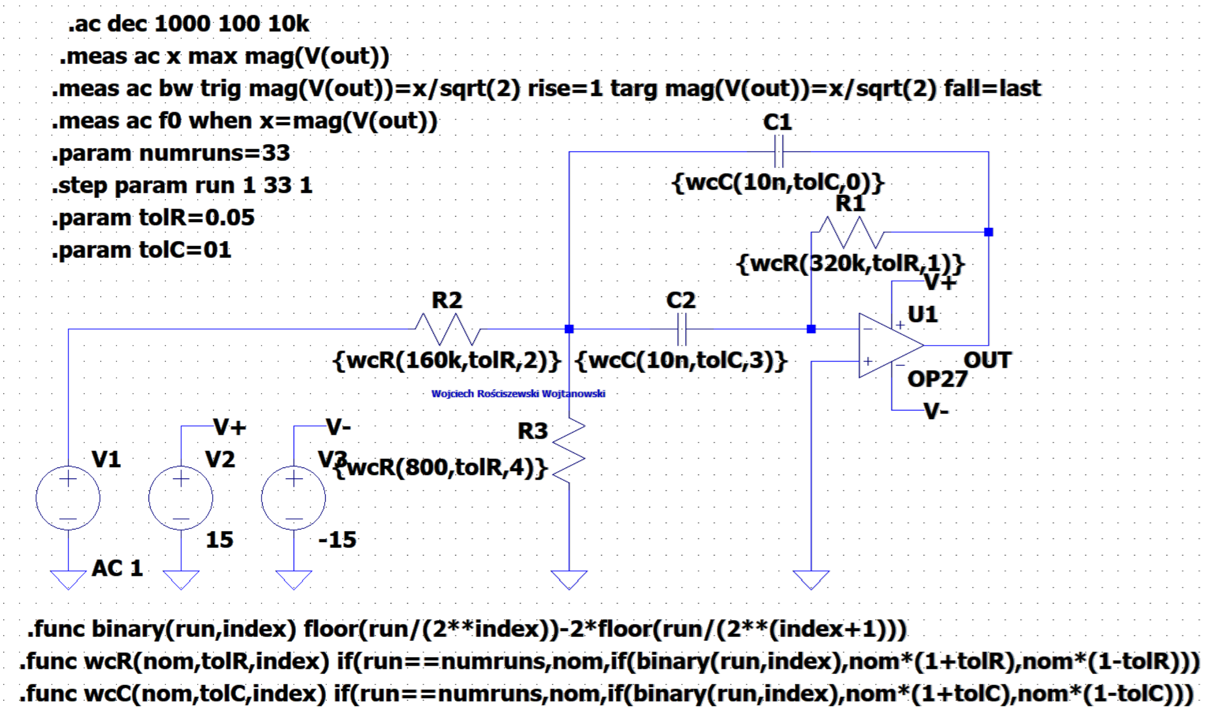

Figure 36 Scheme of the tested filter system for WC analysis.

In the below please see the output of our circuit

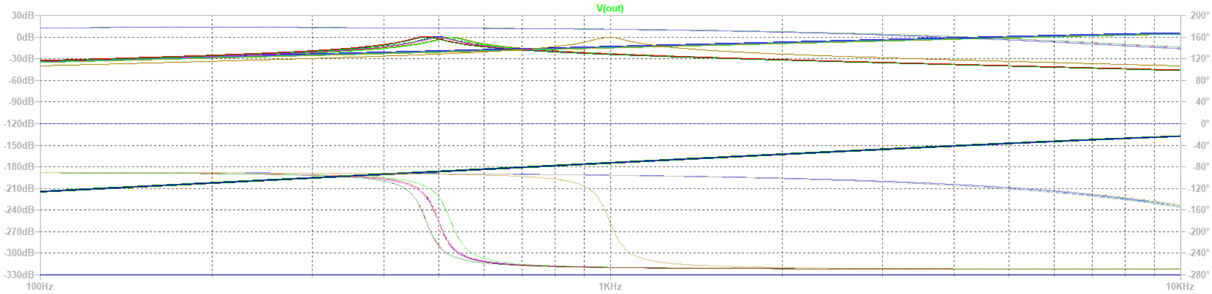

Figure 37 Plot Characteristic.

Central Frequency:

Max f_0=114 Hz

Min f_0=862 Hz

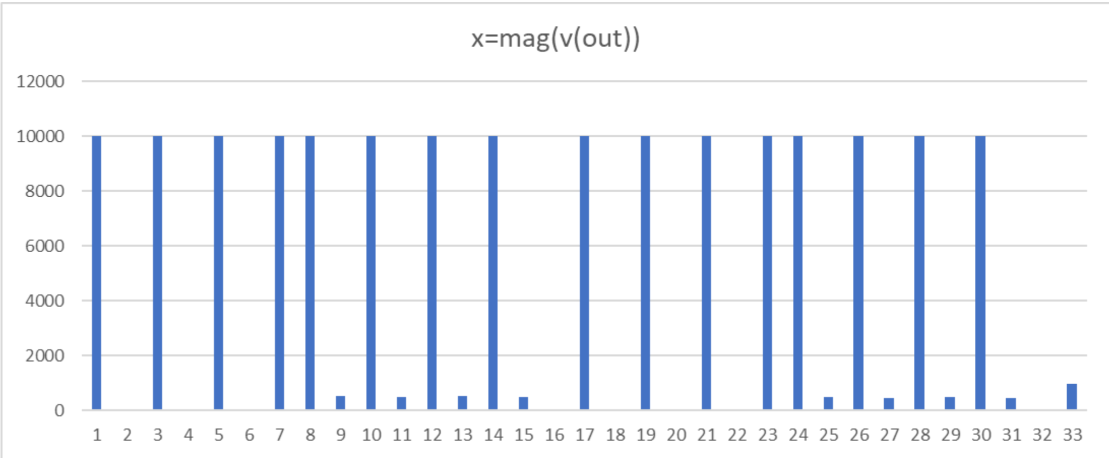

Figure 38 x=mag(vout)

Part 4: Homework

For the first part of the homework task we will develop a band pass filter as previously, identically. This time we will alter the values of the temperature for resistors and capacitors – we will lower it and see the effects it makes on the system performance.

Below please see the circuit that has been used for this part.

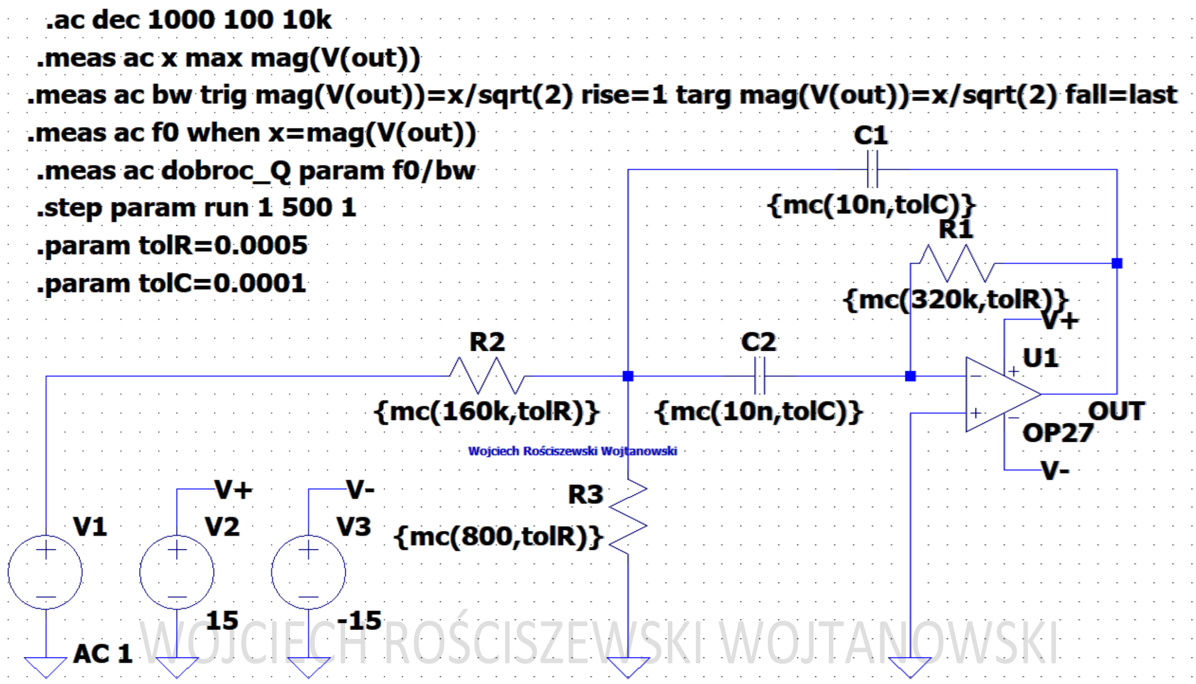

Figure 39 Band Pass Filter Circuit.

In the below please see the output of our circuit.

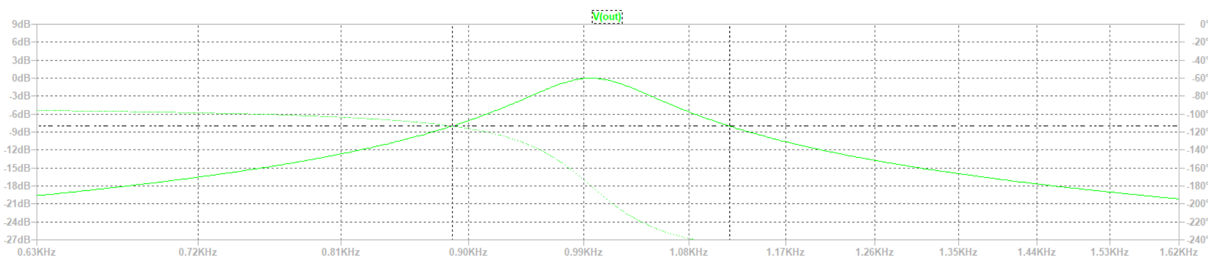

Figure 40 Plot Characteristic.

Below please find the quality filter Q characteristic according to the equation.

$$\boldsymbol{Q} = \frac{f_0}{f_g - f_d}$$

$$\boxed{Q = \frac{f_0}{f_2 - f_1}} = \frac{f_0}{\Delta f} = \left| \begin{array}{l} \text{Where } \Delta f = 200\,Hz \\ \text{as this is difference} \\ \text{of Frequency.} \end{array} \right| = \frac{1\,kHz}{228\,Hz} = 4{,}38$$

Figure 41 Quality Factor.

What did we observe? We observed that by changing the tolerance of our capacitors and resistors we have influence our systems behavior to change and the quality factor of your band pass filter has been lowered by roughly 0.62.

For the second part of the homework task we will develop a transistor type amplifier circuit system, and then we will perform similar tests as to the ones performed during exercises. Please note that for these systems I will be using a 10% value for resistor tolerance.

In the below please find the provided circuit from our course instruction manual.

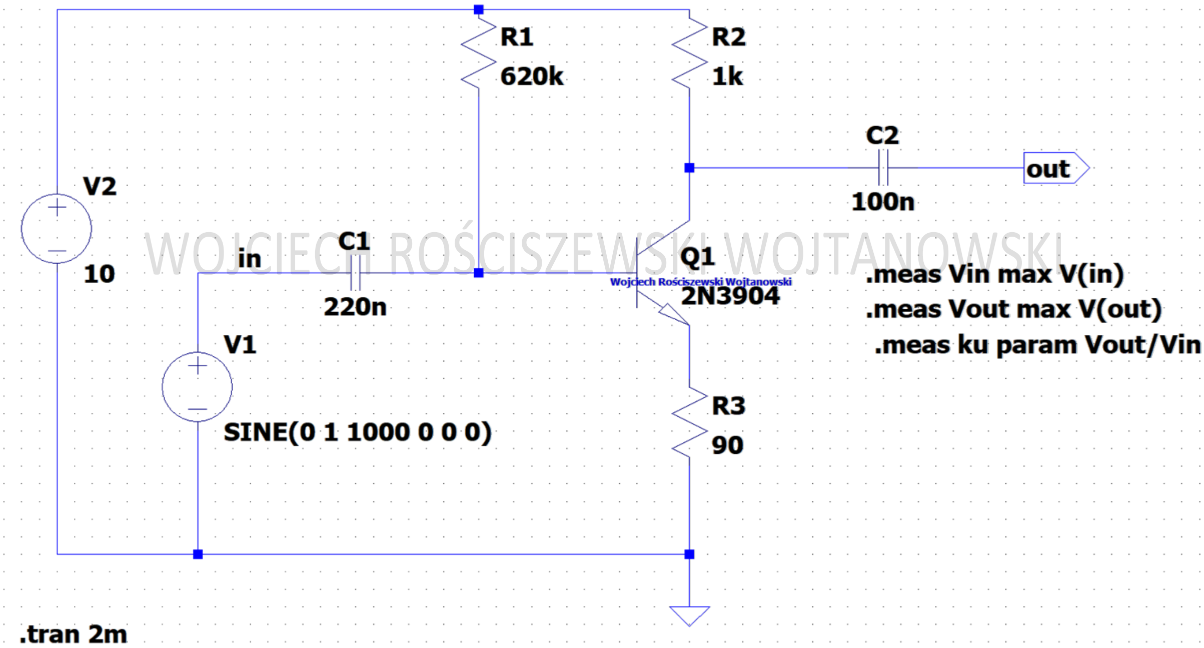

Figure 42 Diagram of a transistor amplifier

Please see in the figure below the output of this system. We see that the sinusoidal waveform is amplified.

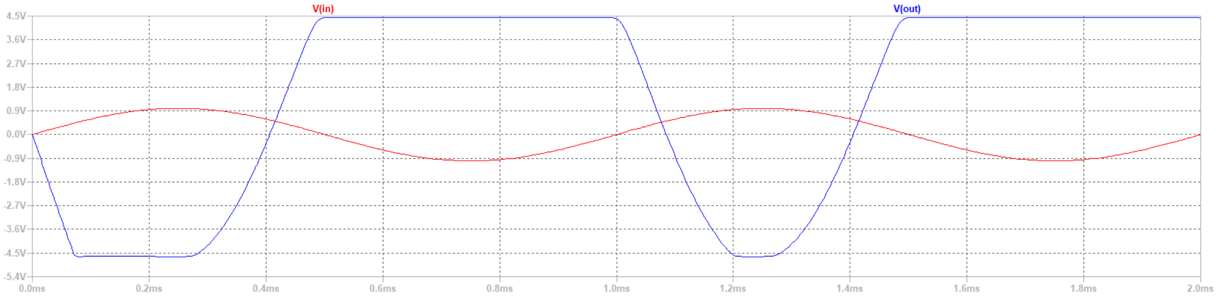

Figure 43 Plot Charactersistic

Please see the input waveform in red, and output in blue. We clearly see our amplification.

Below please see measurement results:

```
vin: MAX(v(in))=0.999989 FROM 0 TO 0.002
vout: MAX(v(out))=4.45289 FROM 0 TO 0.002
ku: vout/vin=4.45294
```

We see that this system amplifies input characteristic to 4.45 V.

Now we will perform the MC Analysis:

In the below please find our new system:

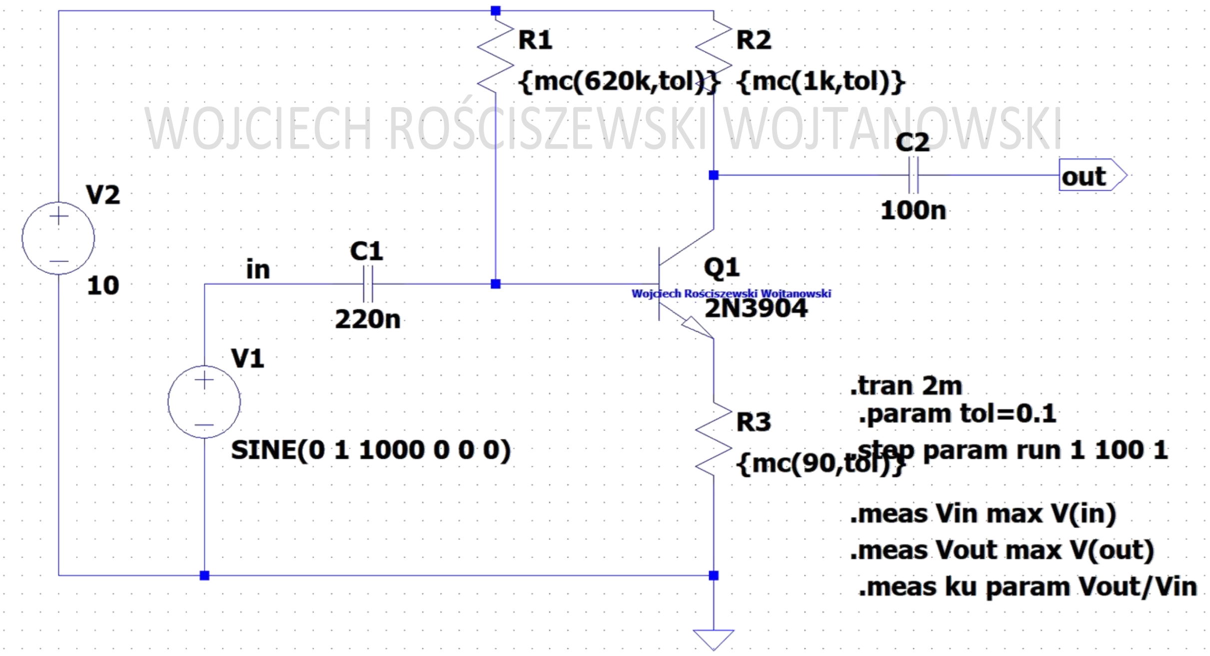

Figure 44 MC Analysis.

Please see in the figure below the output of this system. We see that the sinusoidal waveform is amplified.

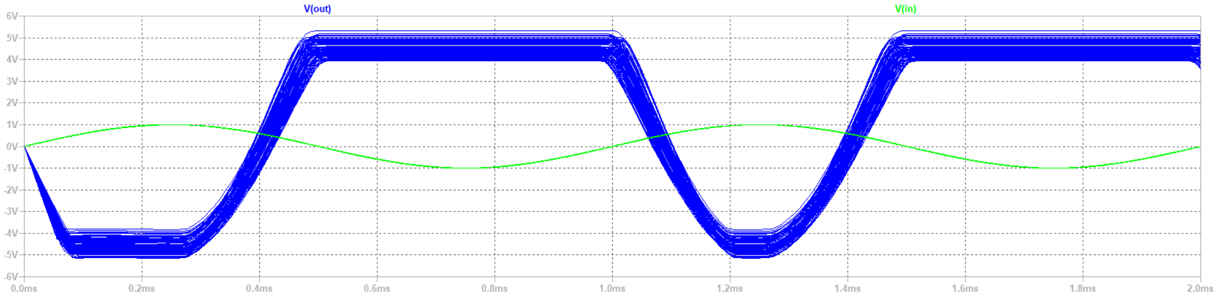

Figure 45 Plot Characteristic.

We see in the figure above how the tolerance of the resistive components in our system changes the entire operation of our system, maybe not greatly but as mentioned previously in the report sensitive systems such as radio receivers might show impact of this.

Now we will perform the WC Analysis:

In the below please find our new system:

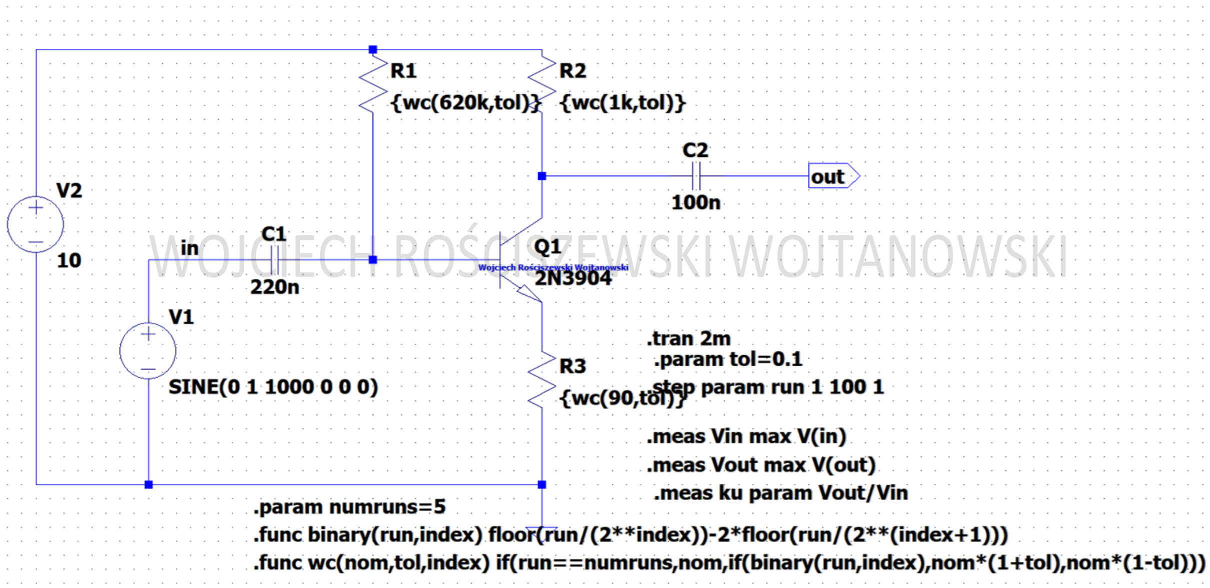

Figure 46 WC Analysis.

Please see in the figure below the output of this system. We see that the sinusoidal waveform is amplified.

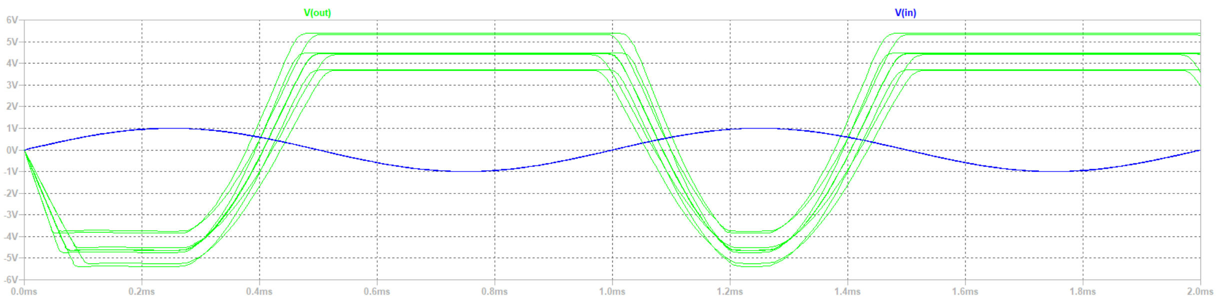

Figure 47 Plot Characteristic.

From the figure below we see that the signal of course is amplified but we see a strange event being carried out, as the amplitude of the signal is changed according to tolerance. Therefore, through this activity we see how greatly a system can be affected by the tolerance of the resistive elements. Out of curiosity I will do the WC analysis including the temperature coefficients. I will use the values used in the previous homework activity. I believe we will see a bigger impact of how temperature and tolerance of a resistor can affect the system behavior. I now understand why technicians before calibrating their equipment first let the device run and pre-heat for an hour or two so that all of the voltages stabilize.

In the below please find our new system:

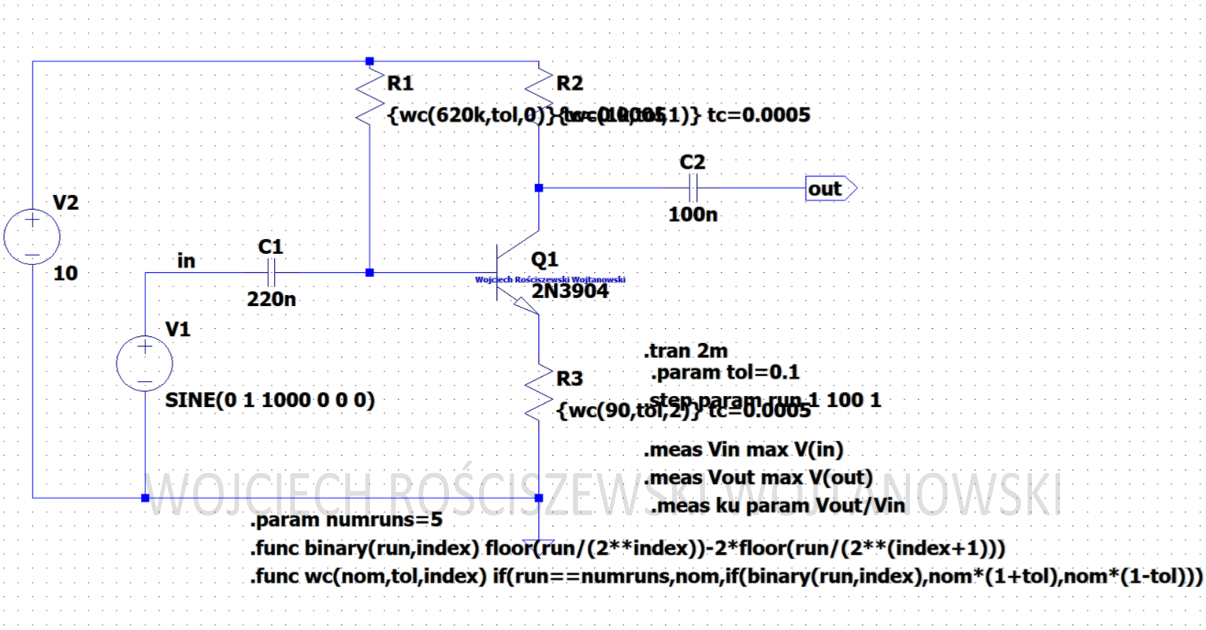

Figure 48 WC with TEMP.

Please see in the figure below the output of this system. We see that the sinusoidal waveform is amplified.

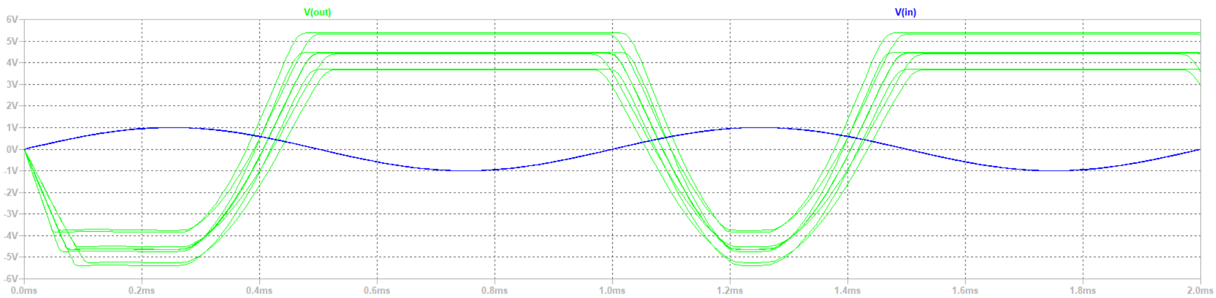

Figure 49 Plot Characteristic.

What we see is basically the same output, or similar. After inspection there are very small changes to the signal, therefore I will conclude with the fact that the used temperature coefficient did not effect the system greatly.