Analog Electronic Circuits Laboratory
Experiment 4
Transistor Feedback Amplifiers

## PRELIMINARY WORKS& THE THINGS THAT SHOULD BE THOUGHT BEFORE THE LAB

**Note:** It is an elective course and you chose this course. Thus, the grading, the credit of the course etc. are not important. The only important thing should be that you obtain some intuition and experience regarding the fundamental analog circuits. Please don't see this class as a compulsory class. We expect you to be interactive and curious during the lab.

- How many types feedback is employed in electronics?
- What are the advantages and disadvantages of negative feedbacks? When you consider both, does it worth to apply negative feedback?
- What type of negative feedback is applied in Fig.1? What is your method when you try to find out the type of the negative feedback?
- How much closed loop gain do you expect? Don't do any calculation, just look at the circuit and try to gain some insight.
- Regarding the circuit in Fig.1, why do we need the $C_f$ ?
- Why do we need 10 μF capacitor parallel to 3.9k resistor?
- How many high impedance nodes in the circuit in Fig.1?
- What is the purpose of 1 nF parallel capacitor in Fig.1? Why it is a terrible method to use this capacitor in that way?
- Please refresh your knowledge about the stability before starting the lab. How can you check the stability from open loop bode plot or loop gain plot.

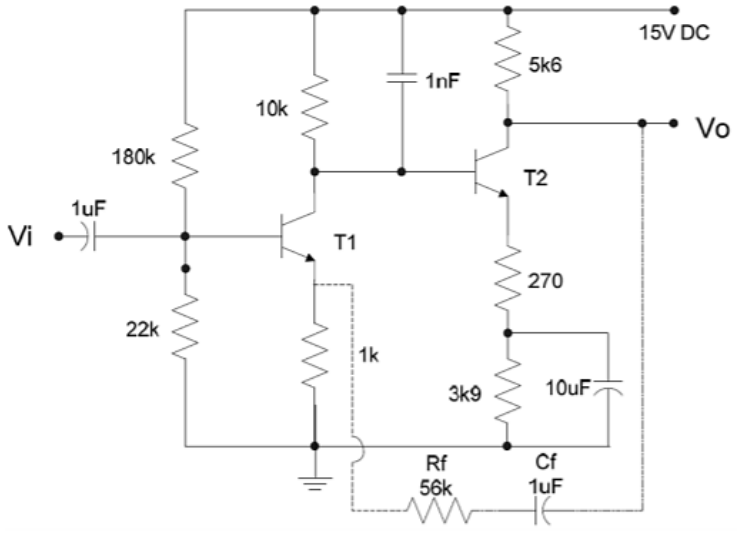


**Fig.1 :** The Circuit for the Lab

Analog Electronic Circuits Laboratory
Experiment 4
Transistor Feedback Amplifiers

## INSTRUCTIONS FOR THE EXPERIMENT

**Note 1** : Please download the cover via **https://www.elelab.itu.edu.tr/elegiris/kapak.doc**. You are going to fill this document during the lab.

**Note 2** : During the LTSpice simulations, please use these parameters for BC238C via https://www.elelab.itu.edu.tr/belgeler/pspice/bc238.txt. I assume you already know how to import 3rd party models to LTSpice. If you don't know, please learn it before you participate to the lab.

**Note 3** :Set two separate Spice schmetic, the one with non-feedback and the other one with feedback.

- Find the DC operating points for the circuit with and without feedback. Did you observe any change? **(15 mins at most with discussions, and setting the circuits in Spice)**
  Find the open loop midband gain, low and high frequency cut-off frequencies, input&output impedances by plotting the bode plot. Use only AC analysis for this part. Discuss the results. Suggested AC analysis command is given below. **(25 mins at most with discussions)** ***.ac dec 100 0.1 10MEG***
- Find the closed loop midband gain, low and high frequency cut-off frequencies, input&output impedances by plotting the bode plot. Use only AC analysis for this part. Discuss the results. Suggested AC analysis command is given below. **(25 mins at most with discussions)** ***.ac dec 100 0.1 10MEG***
- What did you observe on gain and corner frequencies? Discuss the comparison. Does feedback provide an accurate gain? **(10 mins at most with discussions)**
- Find the Gain-Bandwidth Product and phase margin for the open-loop configuration. Discuss the results. **(10 mins at most with discussions)**
- Find the phase margin from loop-gain. For this part, you need to break the loop from a convenient point and should observe the phase margin from returned voltage. **(10 mins at most with discussions)**
- For the closed loop configuration apply a 1kHz sinusoidal with 10 mV amplitude. After running the transient simulation, observe the $V_{out}$-8.275 (to see only AC part at the output, or you can connect a large load and coupling capacitor) and $V_{in}$. **(10 mins at most with discussions)**

- For the closed loop configuration apply a 10 kHz pulse with 10 mV amplitude, and %50 duty cycle. After running the transient simulation, observe the $V_{out}$-8.275 (to see only AC part at the output, or you can connect a large load and coupling capacitor) and $V_{in}$. **(10 mins at most with discussions)**

## REPORT

Preserving the circuit and parameters, remove only 1 nf capacitor. Instead choose an appropriate compensation capacitor and design the circuit for the stability with the same feedback conditions. Repeat the simulations for the circuit that you are going to design.