McMaster University

Electrical and Computer Engineering Department

EE3EJ4 Electronic Devices and Circuits II - Fall 2025

Lab. 4 Feedback Circuits

Lab Report Due on Nov. 16, 2025

Objective: To design and characterize a negative feedback amplifier and an oscillator.

Attributes Evaluated: These are the attributes you need to demonstrate in your solutions.

- **Competence in specialized engineering knowledge to simulate circuit performance using a SPICE-based circuit simulator and conduct analog circuit debugging;**
- **Ability to obtain substantiated conclusions as a result of a problem solution, including recognizing the limitations of the approaches and solutions and**
- **Ability to assess the accuracy and precision of results.**

Test Equipment:

- Analog Discovery 2 (or 3) (AD2/3)
- WaveForms from Digilent Link and WaveForms Reference Manual
- Analog Discovery 2 Quick Start Series Videos

Components:

- Op-Amp: 1 × TLV2371
- Transistors: 6 × NPN-BJT 2N3904, 3 × PNP-BJT 2N3906
- Resistors: 3 × 8.06 kΩ resistor, 3 × 76.8 kΩ resistor, 3 × 57.6 kΩ resistor, 2 × 8.06 kΩ resistor, 2 × 8.25 kΩ resistor, 2 × 100 kΩ resistor, 1 × 240 kΩ resistor
- Capacitors: 2 × 1 nF (102) capacitor

Information of Components:

For a detailed description of these transistors, please check the following websites:
https://www.onsemi.com/pub/Collateral/2N3903-D.PDF
https://www.onsemi.com/pub/Collateral/2N3906-D.PDF

For the description of Op-Amp TLV2371 and its SPICE model, please check the following websites:
https://www.ti.com/product/TLV2371?dcmp=dsproject&hqs=sw&#design-development##design-tools-simulation

Reminder: Switch off the DC power suppliers first whenever you need to change the circuit configurations. Turn on the DC power only when you no longer change the circuit connection.

Part 1: Negative Feedback Amplifier

Description of the Negative Feedback Amplifier

In this lab, we design a negative feedback amplifier using a directly coupled (DC), multi-stage operational (Op) amplifier (Amp). We then construct a feedback amplifier by connecting the output of the Op-Amp. In this section, we first characterize the Op-Amp and then characterize the voltage gain and frequency response of the negative feedback amplifier using the feedback theory.

A. SPICE Simulation – Op-Amp Characterization

1.1 In PSpice, construct the Op-Amp, as shown in Fig. 1, using the common-emitter (CE) amplifier from Lab 2, the common-collector (CC) amplifier, and the differential amplifier with a current mirror load from Lab 3. Here, I_x is a current test source used to characterize the output resistance of the Op-Amp.

1.2 **Voltage Gain:** Set the DC values of V_1 and V_2 = 0 V and their AC amplitude 1 mV. Set the DC and AC values of the test current source I_x to zero. For the differential mode signal, set the phases of the AC signal V_1 and V_2 to be 0° and 180°, respectively, as shown in Fig. 1. In this setting, the differential-model signal $v_{id} = V_1 - V_2 = 1\text{ mV} - (-1\text{ mV}) = 2\text{ mV}$. Conduct AC analysis to obtain the base current i_{b1} of Q1, the voltages V_{o1}, V_{o2}, and V_o at the base of Q8, the base of Q9, and the emitter of Q9, respectively. Set the AC sweep type in Logarithmic with Start Frequency = 100 Hz, End Frequency = 100 kHz, and Points/Decade = 101. Record the simulated magnitude and phase of i_{b1}, V_{o1}, V_{o2}, and V_o in the sheet "Step 1.2" of the Excel file "Lab 4 – Feedback Circuit.xlsx".

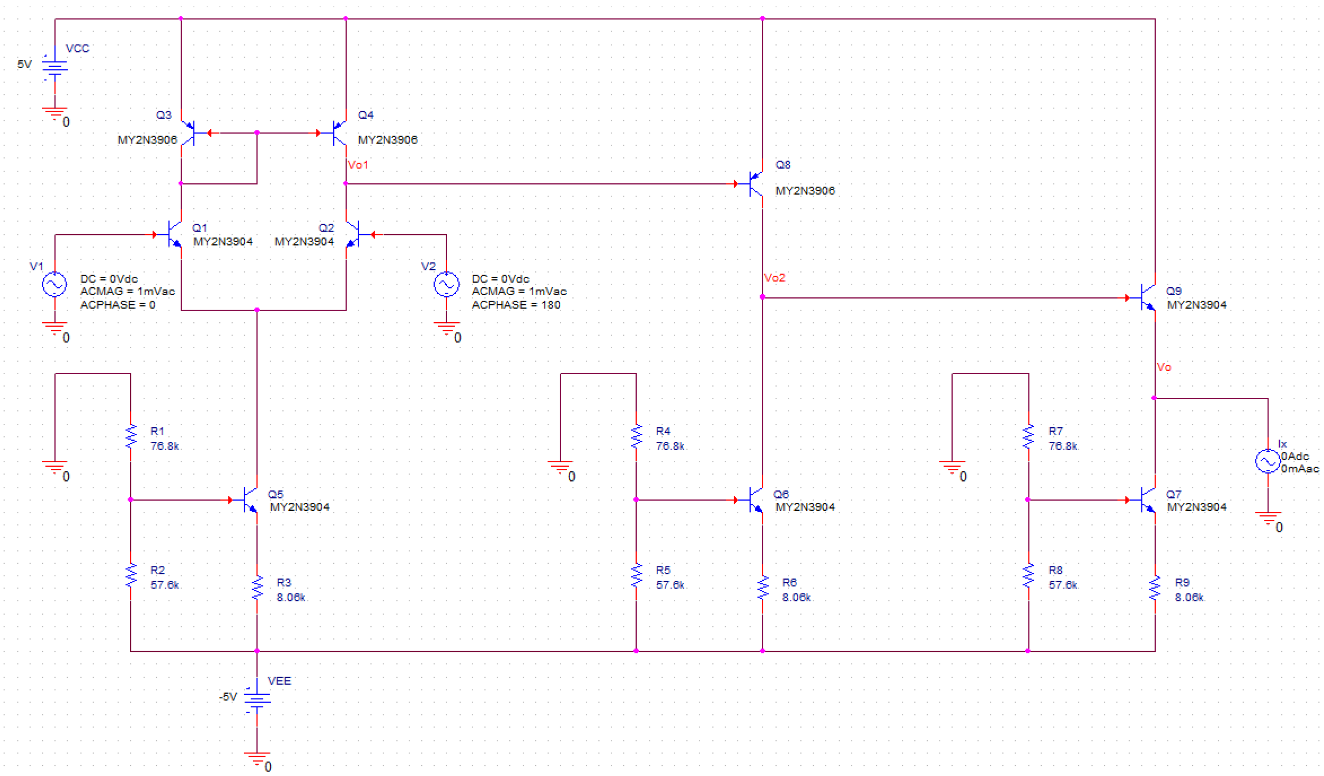


Fig. 1 Schematic diagram of the directly coupled (DC), multi-stage operational amplifier

1.3 **Frequency Response:** Set the DC and AC values of V_1 and $V_2 = 0$ V. Set the DC value of $I_x = 0$ A and AC amplitude of the test current source I_x to 1 mA. Since SPICE uses a passive sign convention, i.e., positive current flows from the (+) node of the current source to its (-) node, we set its AC phase of I_x to 180° for the current flowing into V_o, as required by the linear two-port network theory. Conduct AC analysis to obtain V_o at the emitter of Q9. Set the AC sweep type in Logarithmic with Start Frequency = 100 Hz, End Frequency = 100 kHz, and Points/Decade = 101. Record the simulated magnitude and phase of V_o in the sheet "Step 1.3" of the Excel file "Lab 4 – Feedback Circuit.xlsx".

B. SPICE Simulation – Negative Feedback Amplifier

1.4 Construct the negative feedback amplifier in PSpice using the circuit diagram shown in Fig. 2, with resistance values and supply voltages specified in the figure.

1.5 To place the sinusoidal voltage source V1, access the Modeling Application with Place -> PSpice Part -> Modeling Application, and select Sources -> Independent Sources. In the dialogue box, choose Sine -> Voltage -> Sine, and set the Offset value VOFF = 0 V, VAMPL = 1 mV, Frequency = 1 kHz, Phase = 0°, DC = 0 V, and AC = 1 mV, respectively.

1.6 **Voltage Gain:** Conduct Transient Response simulation for V_o at the emitter of Q9 with Run to Time TSTOP = 6 mS and Maximum Step Size = 25us. Record the output voltage V_o versus time in the sheet "Step 1.6" of the Excel file "Lab 4 – Feedback Circuit.xlsx".

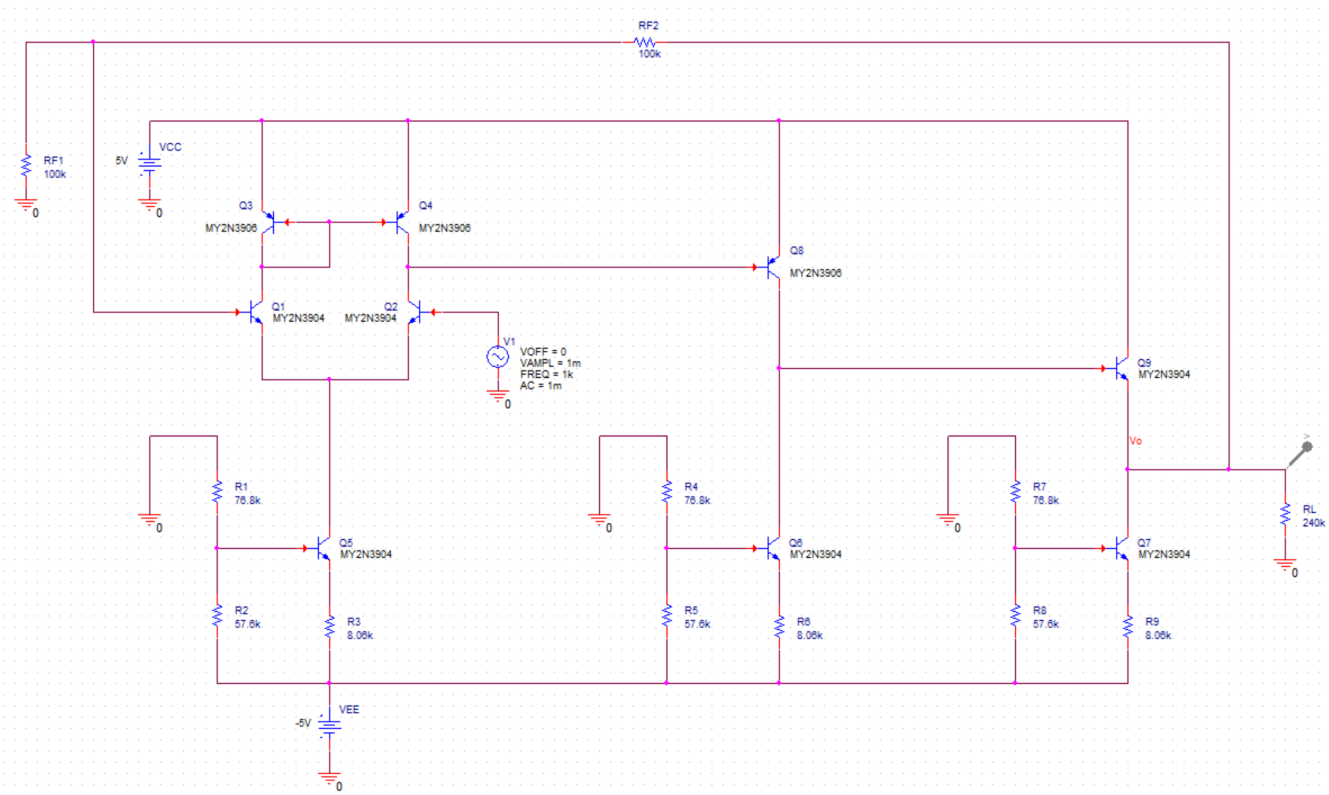


Fig. 2 Schematic diagram of the negative feedback amplifier

1.7 **Frequency Response:** Conduct an AC analysis to obtain V_o at the emitter of Q9. Set the AC sweep type in Logarithmic with Start Frequency = 1 kHz, End Frequency = 10 MegHz, and Points/Decade = 101. Record the simulated magnitude and phase of V_o in the sheet "Step 1.7" of the Excel file "Lab 4 – Feedback Circuit.xlsx".

C. AD2/3 Measurement

Because operational amplifiers (Op-Amps) are directly coupled devices with significant differential gains, they are prone to DC problems (e.g., mismatch), resulting in the DC offset voltage. Suppose the two input terminals of an Op-Amp are tied together and connected to the ground; even though $V_{id} = 0$, a finite DC voltage still exists at the output. The output will be at the positive or negative saturation level if the Op-Amp has a high differential-mode gain. The Op-Amp output can be brought back to its ideal value of 0 V by connecting a DC offset voltage with appropriate polarity and magnitude between the two input terminals of the Op-Amp. In this lab, we first characterize this DC offset voltage of the designed Op-Amp and then use it to design a feedback amplifier.

1.8 Use the port definition diagram of the AD2/3 shown in Fig. 3 when setting up your circuits.

1.9 Based on Fig. 1, construct the measurement setup for an operational amplifier using the common-emitter (CE) amplifier from Lab 2 and the common-collector (CC) and differential amplifiers from Lab 3.

1.10 Use V+ = 5V for V_{CC} and V- = -5V for V_{EE}. Connect Scope Ch. 1 Negative (1-), Scope Ch. 2 Negative (2-), GNDV+, GNDV-, GNDW1, and GNDW2 to a common ground line.

1.11 **DC Offset Voltage:** Based on Fig. 1, connect Waveform Generator Channel 1 (W1) to the input V_1 (the base of Q_1), Waveform Generator Channel 2 (W2) and Scope Ch. 1 Positive (1+) to the input V_2 (the base of Q_2), and Scope Ch. 2 Positive (2+) to V_o (the emitter of Q_9). In WaveForms, click Workspace, open the provided workspace script "Lab4_Step1.11.dwf3work" and press Run. This script will set the offset voltage of Wavegen 1 (W1) to 0 V and gradually change the offset voltage of Wavegen 2 (W2) until the output voltage of the Op-Amp switches around 0 V. Record the measured offset voltage for V+ (or Q_2) in the sheet "Step 1.11" of the Excel file "Lab 4 – Feedback Circuit.xlsx".

1.12 Based on Fig. 2, construct the measurement setup, as shown in Fig. 4, for the feedback amplifier using the operational amplifier and the offset voltage for Channel 2 (W2) found in Step 1.11.

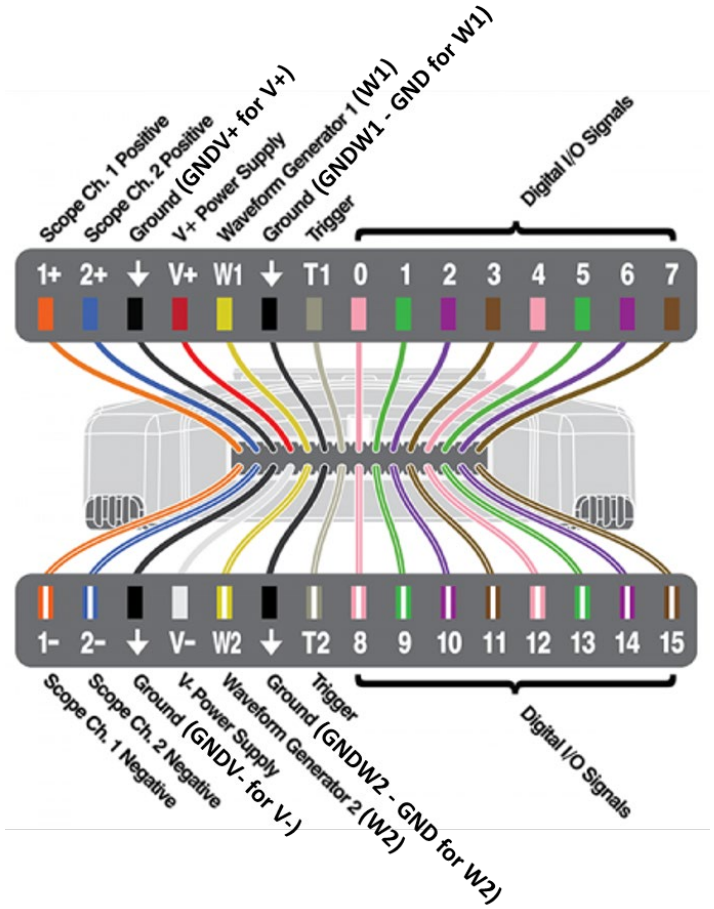


Fig. 3 Diagram for the port definition of an Analog Discovery 2 (or 3) (AD2/3)

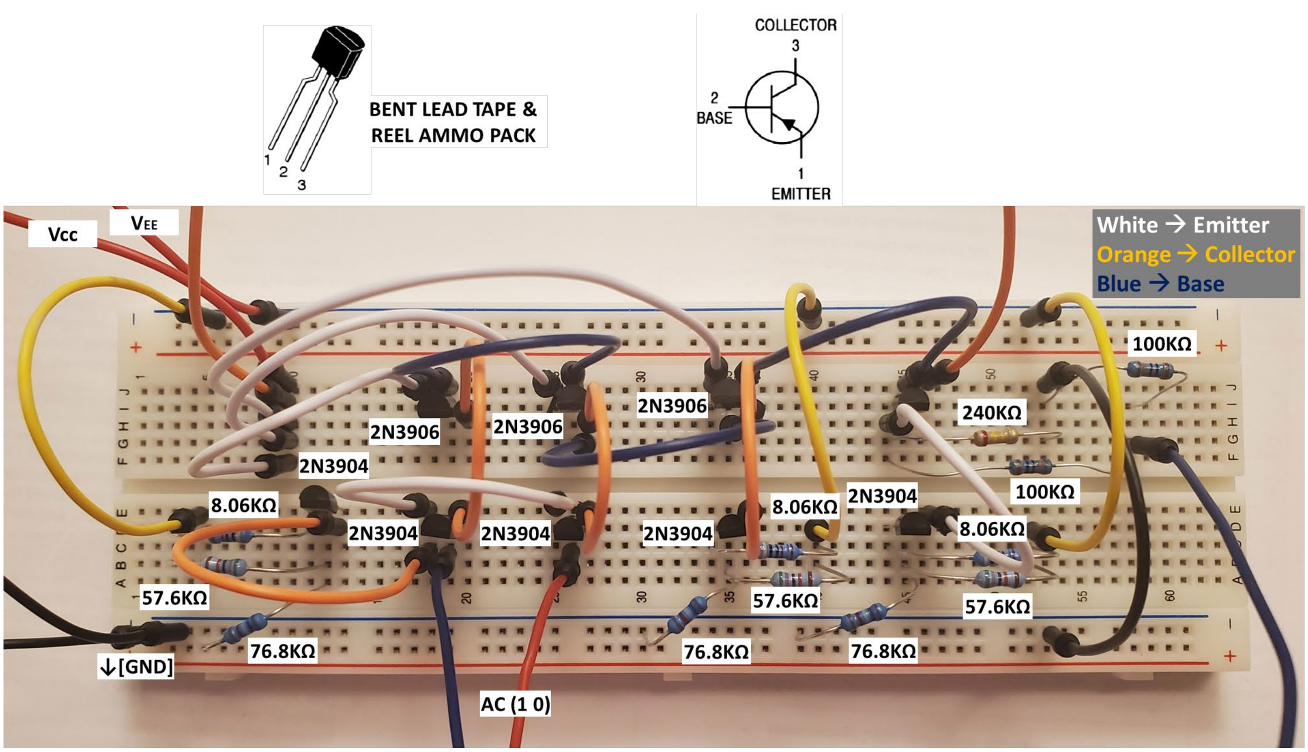


Fig. 4 Experimental setup for the negative feedback amplifier

1.13 **Voltage Gain:** Connect the Waveform Generator Channel 1 (W1) and Scope Ch. 1 Positive (1+) to the input V_1 (<u>the base of Q_2</u>) of the amplifier. Connect Scope Ch. 2 Positive (2+) to the output V_o of the amplifier at the emitter of Q_9. In the WaveForms software, open the Wavegen, Scope, and Supplies tools. In the Wavegen Channel 1 (W1), launch a sine wave with Frequency = 1 kHz, Amplitude = 1 V, and Offset = the offset voltage obtained in Step 1.11, and then press Run. (**Note:** Here, we use Amplitude = 1 V instead of 1 mV used in Steps 1.5 and 1.6 to demonstrate that the negative amplifier can amplify signals with an amplitude larger than $V_T/2$ = 12.5 mV). Next, in the Supplies tab, enable the voltages with V+ = 5 V and V- = -5 V, respectively. In the Scope tab, press Run to see the input (Ch.1) and output (Ch.2) signals. Set the Base (sec/div) and Range (V/div) to be the same as these in both channels to see the signals. Next, press Single to take a screenshot of the data. Finally, go to the upper toolbar and select Export to copy both input V_1 and output V_o voltage signals. Uncheck the Comments, Copy to Clipboard, and Paste these data into the sheet "Step 1.13" of the Excel file "Lab 4 – Feedback Circuit.xlsx". <u>Replace the screenshot with yours, and make sure to capture your date and time to avoid mark deduction.</u>

1.14 **Frequency Response:** Based on Fig. 2, with the same wire connections as those in Step 1.11, go to Waveforms software and, from the Supplies tab, enable the voltages V+ = 5 V and V- = -5 V, respectively. Next, open the Network Analyzer tool and press Run to see the input (Ch.1) and output (Ch.2) magnitude and output phase versus frequency characteristics. Set the start frequency = 100 Hz, stop frequency = 4 MHz, Scale = Logarithmic, and samples = 201/decade. Set the Wavegen Offset = the offset voltage found in Step 1.11, Amplitude = 1 V, and set Magnitude Units to gain(x), Top equal 3X, and Bottom equal 0 X. Also, set Channel 1 and Channel 2 Gain to 1 X and offset to zero. Then, press Single to take a screenshot of the data. Finally, select File, then Export to copy the input V_1, output V_o magnitude, and phase response versus frequency characteristics. Uncheck the Comments, Copy to Clipboard, and Paste these data into the sheet "Step 1.14" of the Excel file "Lab 4 – Feedback Circuit.xlsx". <u>Replace the screenshot with yours, and make sure to capture your date and time to avoid mark deduction.</u>

D. Questions for Part 1

For the non-inverting amplifier designed, answer the following questions with simulated and measured data and discuss any discrepancy between the simulation and measurement results.

Q1. (10 Points) (1) Based on the simulation data obtained in Step 1.2, what is the low-frequency (i.e., f = 100 Hz) voltage gain in dB for the first-stage differential amplifier A_{d1}, the second-stage CE amplifier A_{d2}, and the third-stage CC amplifier A_{d3}, respectively, for the differential-mode signal? (2) What is the overall voltage gain for the differential-mode signal? (3) Which input (V_1 or V_2) is the non-inverting input of the operational amplifier? (4) What is the upper 3-dB frequency f_H of the amplifier?

Q2. (5 Points) Compare the simulated differential-mode gain A_{d1} found in Q1 and the simulated gain A_d in Q5 of Lab 3. What causes these two gains to be so different from each other for the same differential amplifier?

Q3. (5 Points) Based on the simulated results obtained in Steps 1.2 and 1.3, what are the input resistance R_{in} and the output resistance R_o of the Op-Amp?

Q4. (10 Points) (1) Based on the simulated and measured results from Steps 1.6 and 1.13, plot the simulated and measured output voltages V_o vs. time characteristics at 1 kHz. (2) Calculate the simulated and measured peak-to-peak voltage V_{pp}, the AC amplitude V_p, and the dc voltage V_{dc} of V_o, and compare the simulation and measurement results.

Q5. (10 Points) (1) Based on the simulated and measured results from Steps 1.7 and 1.14, plot the simulated and measured voltage gain magnitude and phase vs. frequency characteristics. What is the low-frequency gain of this amplifier? (2) To operate this amplifier, what is its highest operating frequency to provide a constant gain as designed?

Q6. (5 Points) What kind of feedback configurations (e.g., shunt-shunt) is it for the amplifier in Fig. 2?

Q7. (10 Points) Find the beta network and the feedback components β, R_{11}, and R_{22}, respectively.

Q8. (15 Points) Use the feedback theory and simulation results to find the amplifier's voltage gain, input resistance, and output resistance, respectively.

<u>Part 2</u>: Positive Feedback Circuit – Oscillator

Description of the Positive Feedback Circuit – Oscillator

The oscillator circuit utilizing a positive feedback network consists of an Op-Amp connected in the non-inverting configuration, with a closed-loop gain of $1+R_2/R_1$. In the feedback path of this amplifier, an *RC* network is connected. We will design a positive feedback oscillator circuit in this lab using an Op-Amp (TLV2371). In order to ensure the oscillation start, the value of R_2/R_1 should be slightly greater than 2.

A. SPICE Simulation

2.1 In Avenue to Learn, under *Lab 4 Feedback Circuits*, use the "Op-Amp TLV2371.txt" model in the SPICE Model submodule and follow the tutorial in "Link TLV2371 PSpice Model to a Capture Symbol.pdf" in the OrCAD PSpice submodule to create an Op-Amp TLV2371 symbol.

2.2 Construct the oscillator circuit in Fig. 5 in PSpice with the specified resistance, capacitor values, and supply voltages.

2.3 **Time Domain (Transient) Response:** Conduct the Time Domain (Transient) simulation with Tun To Time = 2.5 ms and Maximum Step Size = 1 us. In addition, in the Simulation Settings window, click Options -> Analog Simulation -> Auto Converge, and check the box beside AutoConverge to use the auto converge function. Record the OP-AMP output voltage (V_{out}) in the sheet "Step 2.3" of the Excel file "Lab 4 – Feedback Circuit.xlsx". Find the settling time, which is the time required for the peak value of V_{out} to reach and remain at around 5 V.

2.4 **Settling Time:** Change the R_2 value to 220 kΩ, 240 kΩ, and 280 kΩ, respectively, repeat Step 2.3 and record their settling times in the sheet "Step 2.4" of the Excel file "Lab 4 – Feedback Circuit.xlsx".

2.5 With R_2 = 240 kΩ, change R_3 = R_4 from 8.25 kΩ to 4.02 kΩ, repeat Step 2.3, and record the output voltage (V_{out}) in the sheet "Step 2.5" of the Excel file "Lab 4 – Feedback Circuit.xlsx".

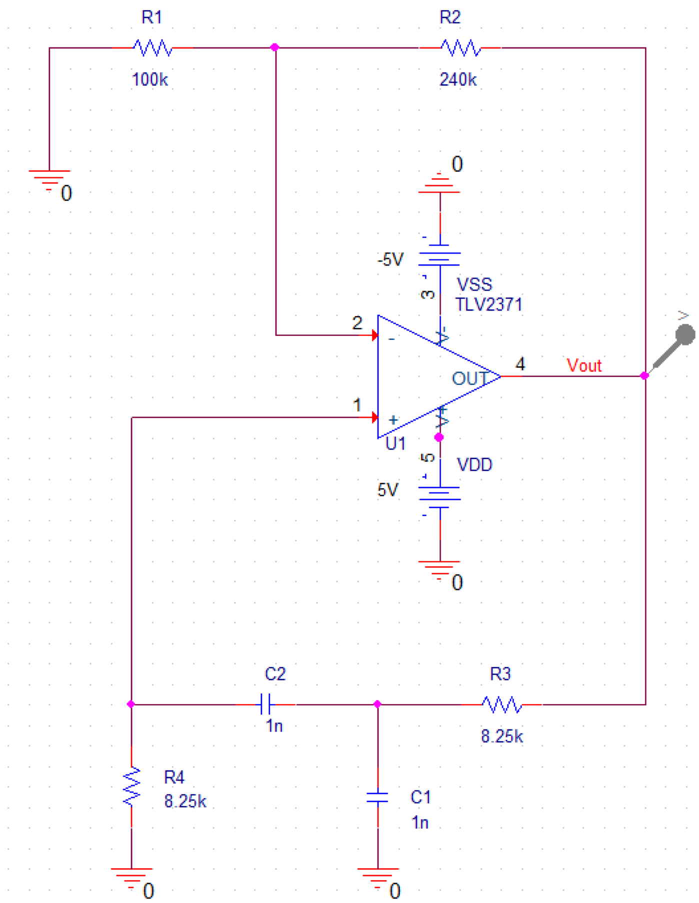


Fig. 5 Schematic diagram of the oscillator circuit

B. AD2/3 Measurement

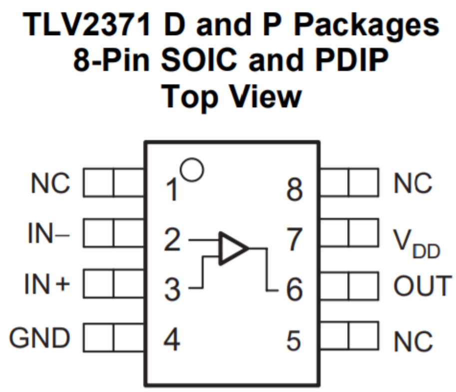
2.6 Based on the circuit diagram shown in Fig. 5, construct the measurement setup for the oscillator circuit, as shown in Fig. 6, using the same settings in Step 2.2 with $R_2 = 240$ kΩ.

2.7 Use V+ = 5V for V_{DD} and V- = -5V for V_{SS}. Connect Scope Ch. 1 Negative (1-), GNDV+, GNDV-, GNDW1, and GNDW2 to a common ground line.

2.8 **Transient Response and Oscillation Frequency:** Connect the Scope Ch. 1 Positive (1+) to the Op-Amp TLV2371 output. In WaveForms, click Workspace, open the provided workspace script "Lab4_Step2.8.dwf3work", and press Single in Scope 1. In Scope 1, click the 'data' tab, Export, Uncheck Comments, Copy to Clipboard, and Paste the data in the sheet "Step 2.8" of the Excel file "Lab 4 – Feedback Circuit.xlsx". Click the 'Measurements' tab, select the Frequency, Period, Amplitude, Maximum, and Minimum of Channel 1 (C1), copy (use Ctrl + C), and paste them into the sheet "Step 2.8" of the Excel file "Lab 4 – Feedback Circuit.xlsx". Replace the measurement screenshot in the sheet and <u>make sure to capture the date and time in the screenshot to avoid mark deduction.</u>

2.9 Based on Fig. 5, change $R_3 = R_4$ from 8.25 kΩ to 4.02 kΩ, repeat Step 2.8, and record the output voltage (V_{out}) versus time characteristics in the sheet "Step 2.9" of the Excel file "Lab 4 – Feedback Circuit.xlsx". <u>Make sure to capture the date and time in the screenshot to avoid mark</u>

deduction.



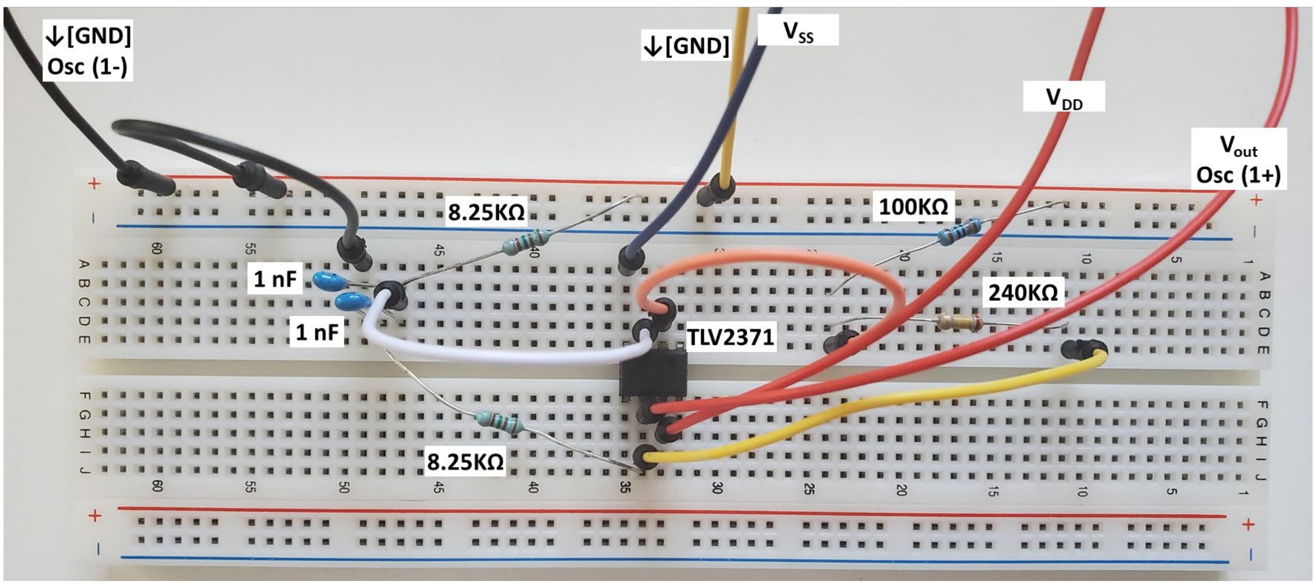


Fig. 6 Experimental setup of the oscillator

C. Questions for Part 2

For the oscillator designed in Part 2, answer the following questions with simulated and measured data and discuss any discrepancy between the simulation and measurement results.

Q9. (15 Points) For the oscillator circuit in Fig. 5, find its loop gain $L(s)$, the frequency for the zero loop phase, and R_2/R_1 for oscillation.

Q10. (5 Points) Based on the simulated results in Step 2.4, what are the settling times for R_2 = 220 kΩ, 240 kΩ, and 280 kΩ, respectively? What do you observe? Explain the observed trend.

Q11. (10 Points) (1) Based on the setup in Steps 2.3, 2.5, 2.8, and 2.9, plot the simulated and measured V_{out}. (2) Calculate the simulated and measured oscillation frequencies in each case. Compare and discuss them with the results from the theory.