

# McMaster University

## Electrical and Computer Engineering Department

### EE3EJ4 Electronic Devices and Circuits II - Fall 2022

#### Lab. 4 Feedback Circuits

#### Lab Report Due on Nov. 13, 2022

**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 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 (AD2)
- [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/products/discretes-drivers/general-purpose-and-low-vcesat-transistors/2n3904>

or <https://www.onsemi.com/pub/Collateral/2N3903-D.PDF>

<https://www.onsemi.com/products/discretes-drivers/general-purpose-and-low-vcesat-transistors/2n3906>

or <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 do not change the circuit connection anymore.

## 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 – Characterization of an Op-Amp

- 1.1 In [PartSim](#) ([LTspice](#) or [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 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  $0^\circ$  and  $180^\circ$ , respectively, as shown in Fig. 1. In this setting, the differential-mode signal  $v_{id} = V_1 - V_2 = 1$  mV  $- (-1$  mV) = 2 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 Q3, the base of Q4, and the emitter of Q4, respectively. Set the sweep type in DEC with Start Frequency = 100 Hz, Stop Frequency = 100 kHz, and Total Points Per Decade = 101. Choose REAL for magnitude unit and degree (DEG) for phase unit. Record the simulated magnitudes and phases 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”.

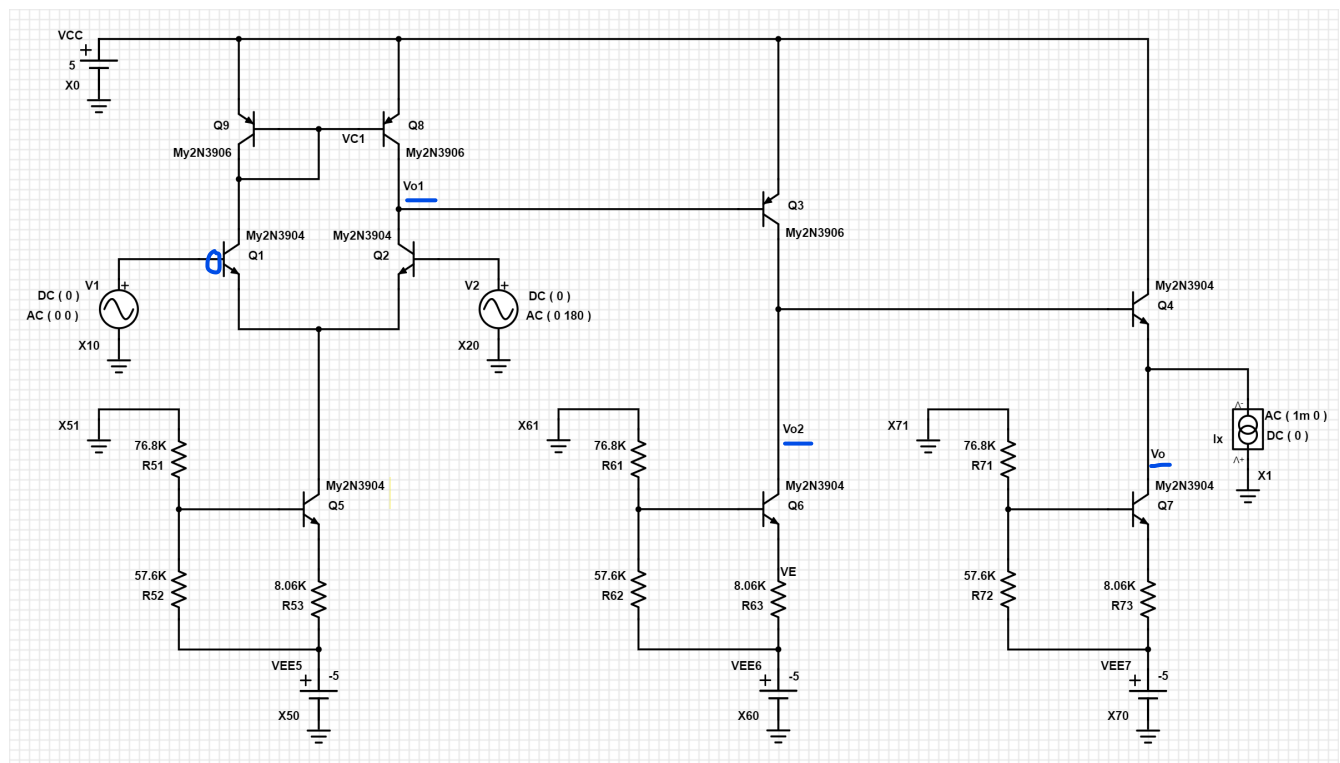
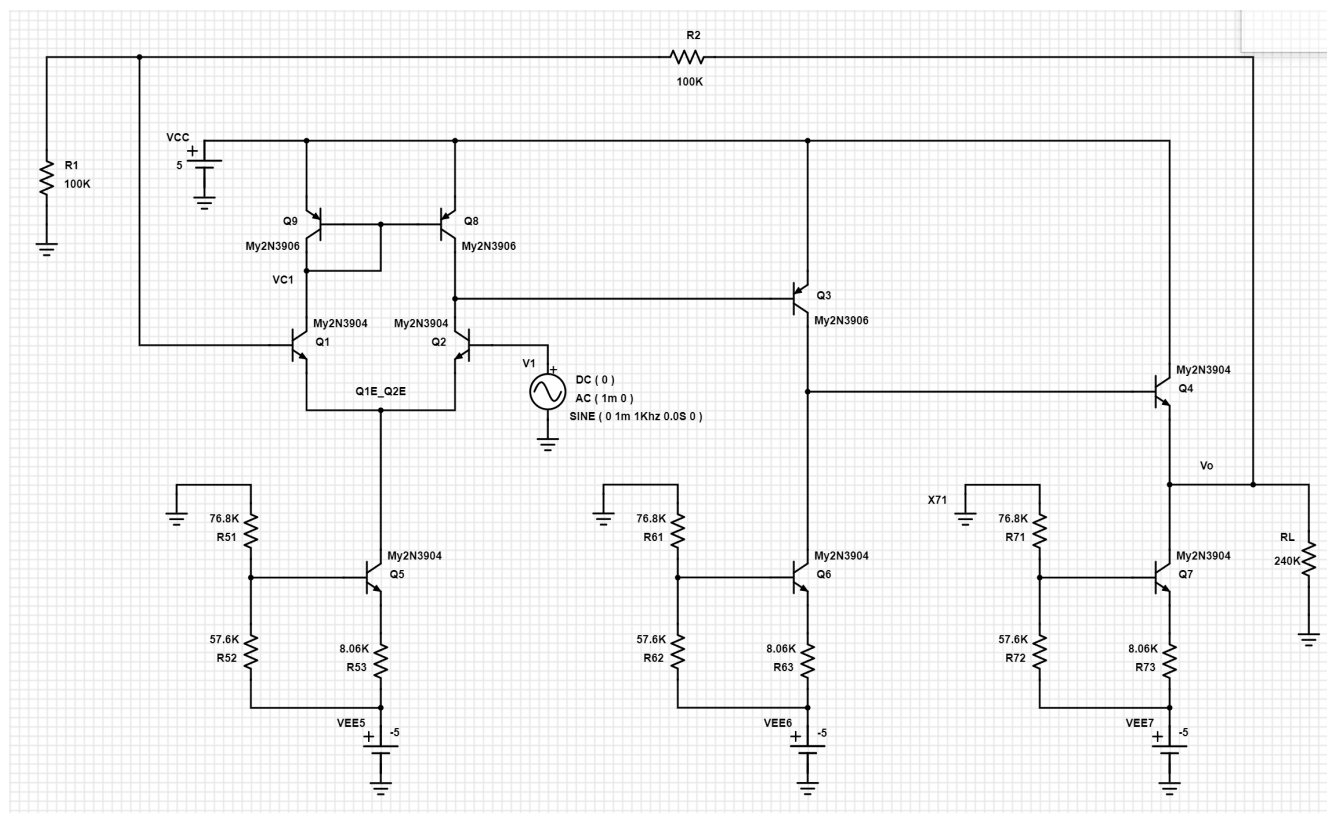


Fig. 1 Schematic diagram of the directly coupled (DC), multi-stage operational amplifier

- 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. Conduct AC analysis to obtain  $V_o$  at the emitter of Q4. Set the sweep type in DEC with Start Frequency = 100 Hz, Stop Frequency = 100 kHz, and Total Points Per Decade = 101. Choose REAL for magnitude unit and degree (DEG) for phase unit. 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 – Feedback Amplifier

- 1.4 In [PartSim](#) ([LTspice](#) or [PSpice](#)), construct the negative feedback amplifier using the circuit diagram shown in Fig. 2, with resistance values and supply voltages specified in the figure.
- 1.5 For the source V1, set its DC voltage to 0 V, its AC Magnitude to 1 mV, and its AC Phase to 0. In addition, enable its Transient Source using “Sine” with Offset = 0 V, Amplitude = 1 mV, and Frequency = 1 kHz, respectively.
- 1.6 **Voltage Gain:** Conduct **Transient Response simulation** for  $V_o$  at the emitter of Q4 with Start Time = 0, Stop Time = 6 mS, Time Step = 25us and Max 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”.
- 1.7 **Frequency Response:** Conduct AC analysis to obtain  $V_o$  at the emitter of Q4. Set the Sweep Type in DEC with Start Frequency = 1 kHz, Stop Frequency = 10 MHz, and Total Points Per Decade = 101. Choose REAL for magnitude unit and degree (DEG) for phase unit. Record the simulated magnitudes and phases of  $V_o$  in the sheet “Step 1.7” of the Excel file “Lab 4 – Feedback Circuit.xlsx”.

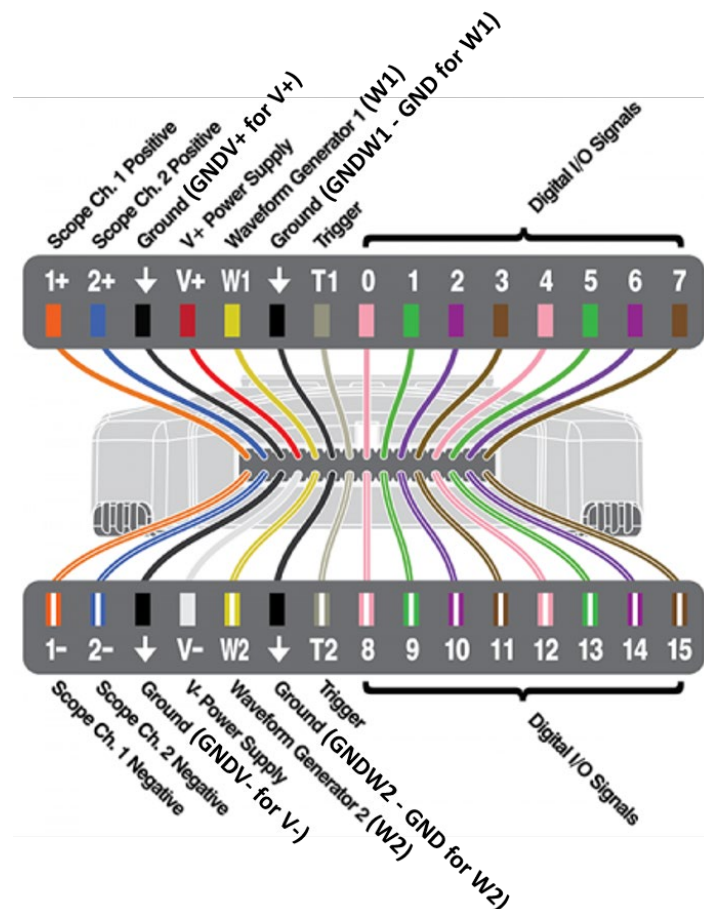


**Fig. 2** Schematic diagram of the negative feedback amplifier

### C. AD2 Measurement

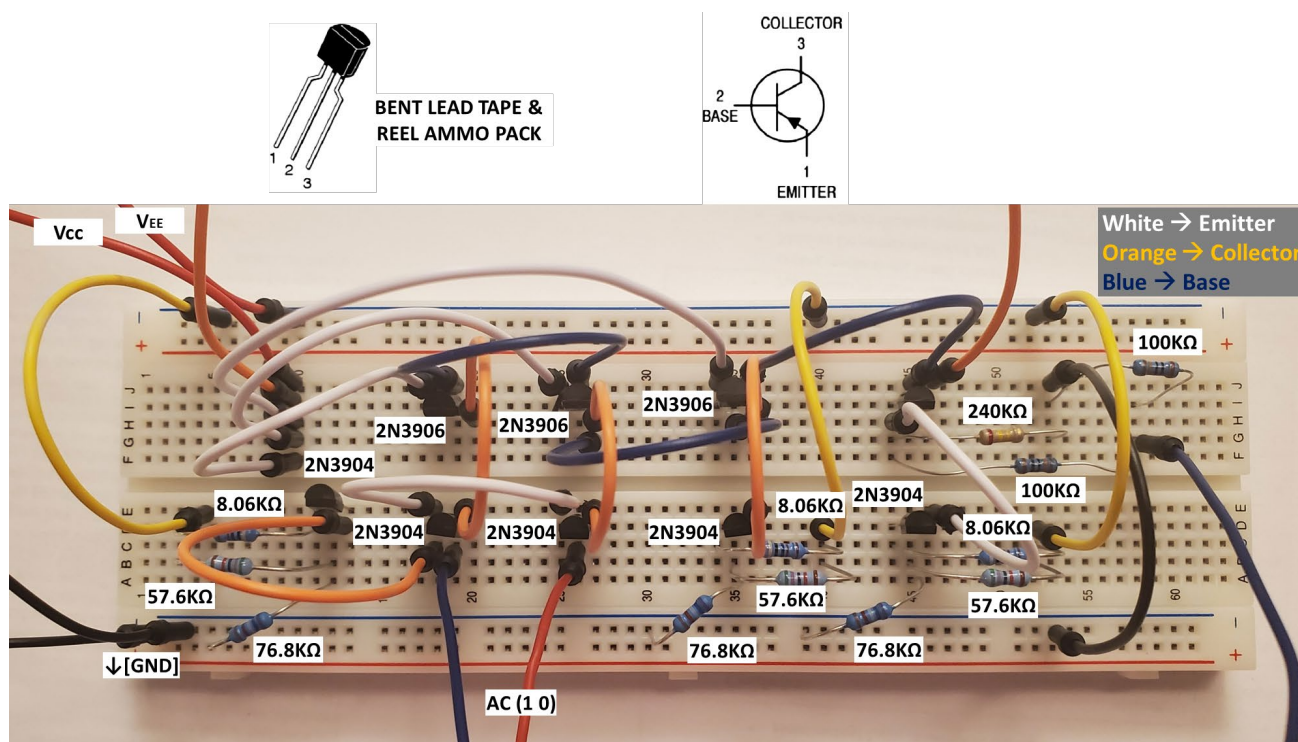
Because operational amplifiers (Op-Amps) are directly coupled devices with significant differential gains, they are prone to dc problems (e.g., mismatch), resulting in the dc offset voltage. Suppose the two input terminals of an Op-Amp are tied together and connected to the ground; even though  $V_{id} = 0$ , 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 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.



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

- 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  $Q_4$ ). 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. 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  $V_1$  (the base of  $Q_2$ ) of the amplifier. Connect Scope Ch. 2 Positive (2+) to the output  $V_o$  of the amplifier at the emitter of  $Q_4$ . 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. 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”. Replace the screenshot with yours and make sure to capture your date and time to avoid mark deduction.

**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  $V^+ = 5\text{ V}$  and  $V^- = -5\text{ V}$ , 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”. Replace the screenshot with yours and make sure to capture your date and time to avoid mark deduction.

#### 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.,  $f = 100\text{ Hz}$ ) 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 the 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 1.13, 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  $\beta$ ,  $R_{11}$ , and  $R_{22}$ , respectively.

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



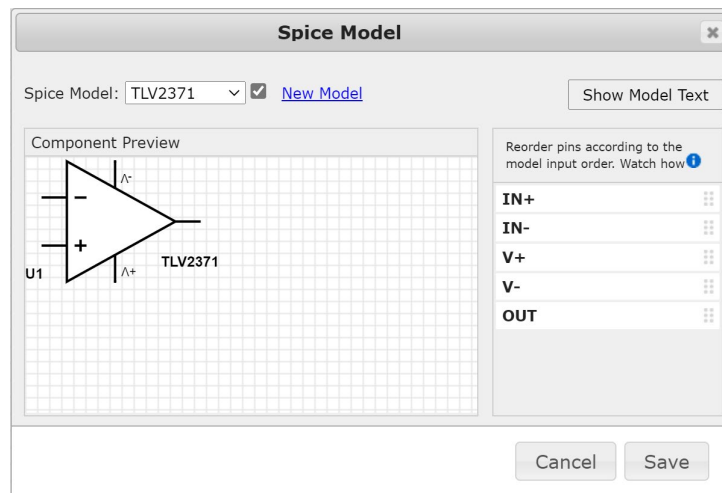
## **Part 2: Positive Feedback Circuit – Oscillator**

### **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.

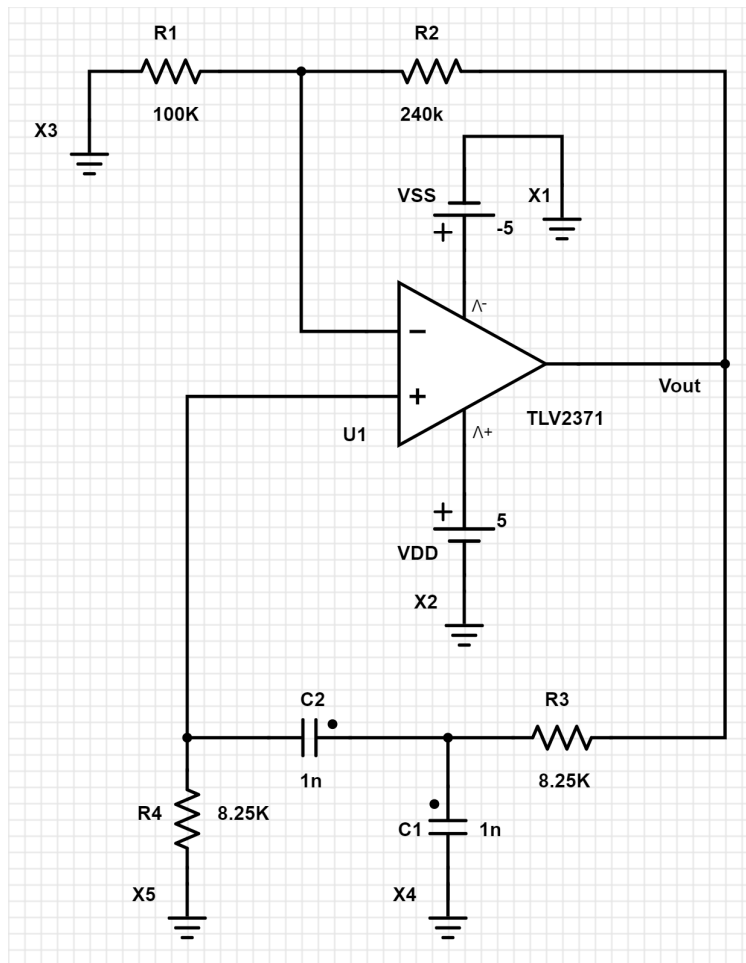
#### **A. SPICE Simulation**

- 2.1 In [PartSim](#) ([LTspice](#) or [PSpice](#)), add an OPAMP (Power Terminals) from Components - Generic Parts - Amplifiers. Download the Op-Amp TLV2371 Spice model from the link provided on Page 1. Copy and paste the Op-Amp model from the TLV2371.LIB file to the Spice Model of the Op-Amp in [PartSim](#) as done before with the BJT models. You can also use the “Op-Amp TLV2371.txt” model in the SPICE Model under the Lab 4 Module on Avenue to Learn.
- 2.2 Drag and drop the pin of Op-Amp TLV2371 in the Spice Model window to have the order: IN+, IN-, V+, V-, and OUT, as shown in Fig. 8.



**Fig. 5** Pin order of the schematic diagram for the Op-Amp TLV2371

- 2.3 Using the circuit diagram in Fig. 9, construct the oscillator circuits with the specified resistance, capacitor values, and supply voltages.
- 2.4 **Transient Response:** Conduct the Transient Response simulation with Start Time = 0 s, Stop Time = 5 ms, Time Step = 10 us, and Max Step Size = 20 us. Record the OP-AMP output voltage ( $V_{out}$ ) in the sheet “Step 2.4” 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.



**Fig. 6** Schematic diagram of the oscillator circuit

- 2.5 **Settling Time:** Change the  $R_2$  value to 220 k $\Omega$ , 240 k $\Omega$ , and 280 k $\Omega$ , respectively, repeat Step 2.4 and record their settling times in the sheet “Step 2.5” of the Excel file “Lab 4 – Feedback Circuit.xlsx”.
- 2.6 With  $R_2 = 240$  k $\Omega$ , change  $R_3 = R_4$  from 8.25 k $\Omega$  to 4.02 k $\Omega$ , repeat Step 2.4, and record the output voltage ( $V_{out}$ ) in the sheet “Step 2.6” of the Excel file “Lab 4 – Feedback Circuit.xlsx”.

## B. AD2 Measurement

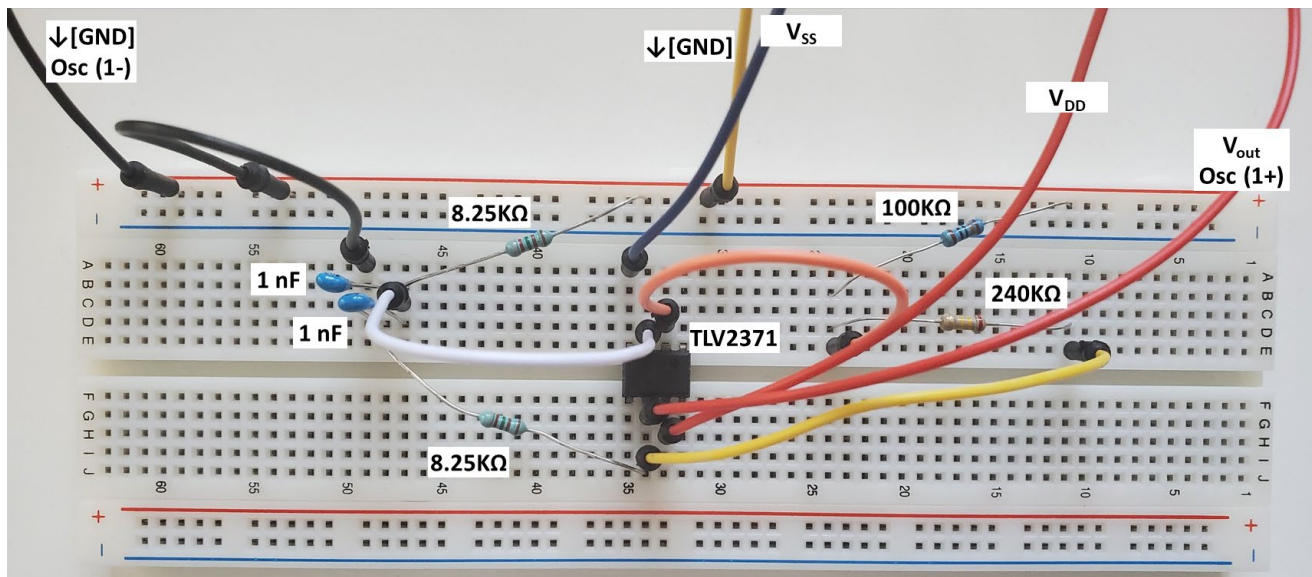
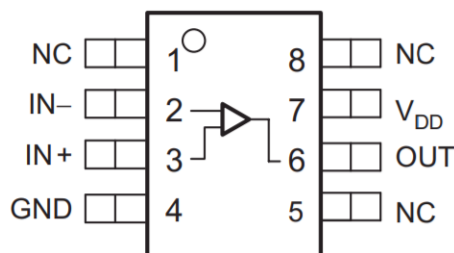
- 2.7 Based on the circuit diagram shown in Fig. 9, construct the measurement setup for the oscillator circuit, as shown in Fig. 10, using the same settings in Step 2.3 with  $R_2 = 240$  k $\Omega$ .
- 2.8 Use  $V_+ = 5V$  for  $V_{DD}$  and  $V_- = -5V$  for  $V_{SS}$ . Connect Scope Ch. 1 Negative (1-), GNDV+, GNDV-, GNDW1, and GNDW2 to a common ground line.
- 2.9 **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\_Step2.9.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 2.9” 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 2.9” of the Excel file “Lab 4 – Feedback Circuit.xlsx”. Replace the measurement screenshot in the sheet and make sure to capture the date and time in the screenshot to avoid mark deduction.

- 2.10 Based on Fig. 9, change  $R_3 = R_4$  from 8.25 k $\Omega$  to 4.02 k $\Omega$ , repeat Step 2.9, and record the output voltage ( $V_{out}$ ) versus time characteristics in the sheet “Step 2.10” of the Excel file “Lab 4 – Feedback Circuit.xlsx”. Make sure to capture the date and time in the screenshot to avoid mark deduction.

### TLV2371 D and P Packages 8-Pin SOIC and PDIP Top View



**Fig. 7** Experimental setup of the oscillator

### C. Questions for Part 2

For the oscillator designed, 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. 9, 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 2.5, what are the settling times for  $R_2 = 220\text{ k}\Omega$ ,  $240\text{ k}\Omega$ , and  $280\text{ k}\Omega$ , respectively? What do you observe? Explain the observed trend.

**Q11. (10 Points)** (1) Based on the setup in Steps 2.4, 2.6, 2.9, and 2.10, plot the simulated and measured  $V_o$ . (2) Calculate the simulated and measured oscillation frequencies in each case. Compare and discuss them with the results from the theory.