McMaster University

Electrical and Computer Engineering Department

EE3EJ4 Electronic Devices and Circuits II - Fall 2023

Lab. 5 Active Filter Circuits

Lab Report Due on Dec. 3, 2023

Objective: To design and characterize the performance of active filters.

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

Components:

- Op-Amp: 1 × TLV2371
- Capacitors: 2 × 1 nF (102) capacitor   1 × 2.2 nF (222) capacitor
- Resistors: 4 × 100 kΩ resistor   1 × 200 kΩ resistor   2 × 240 kΩ resistor

Information of Components:

For the detailed 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. Switch on the DC power suppliers only when you no longer have to change the circuit connection.

<u>Part 1</u>: First-order Low-pass Filter

A. SPICE Simulation

1.1 In PSpice, construct the first-order low-pass filter (LPF), as shown in Fig. 1 and load the Library Op-Amp TLV2371 created in Lab 4.

1.2 For the input signal V_{in}, set its DC Voltage = 0 V, AC Magnitude = 100 mV, and AC phase = 0°, respectively.

1.3 **Frequency Response:** Conduct AC sweep for V_{out} at the output of Op-Amp TLV2371 in Logarithmic with Start Frequency = 1 Hz, End Frequency = 100 kHz, and Points/Decade = 101. In the simulator window, click the Add Trace icon , choose Plot Window Templates under Functions or Macros, select Bode Plot – separate(1), select V(Vout) in Simulation Output Variables, and press OK. Record the simulated magnitude and phase of V_{out} in the sheet "Step 1.3" of the Excel file "Lab 5 – Active Filter.xlsx".

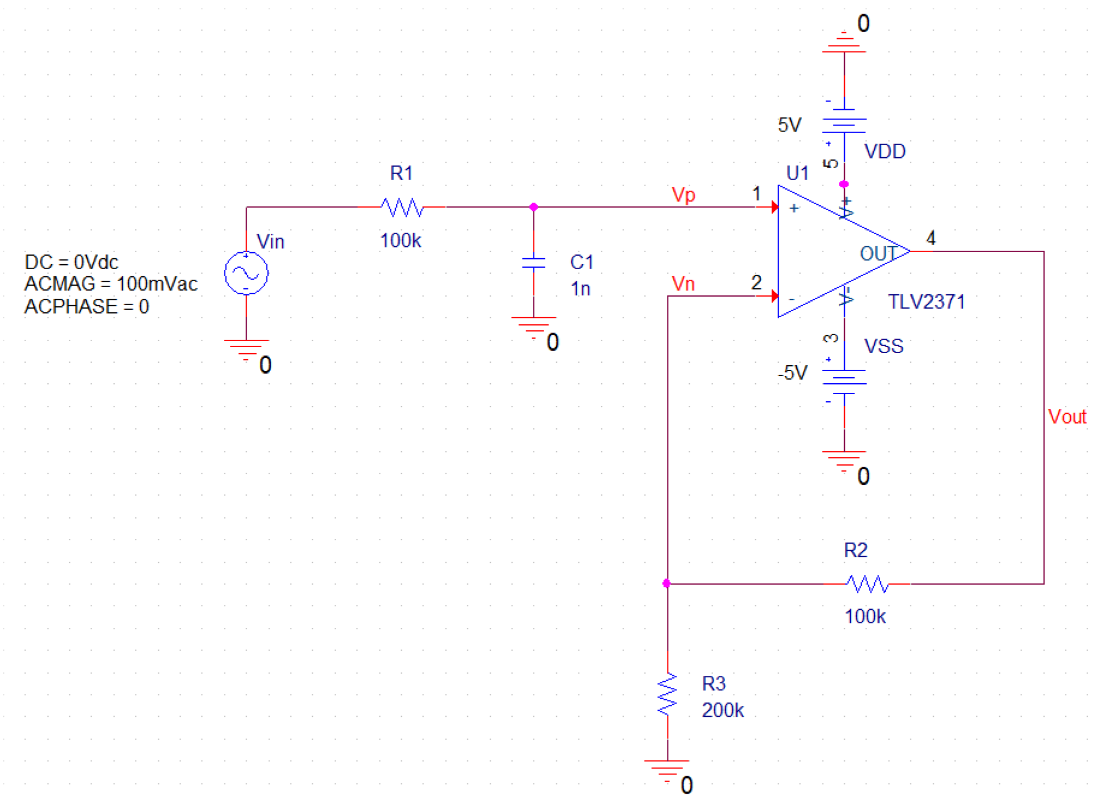

Fig. 1 Schematic diagram of the first-order low-pass filter

B. AD2 Measurement

1.4 Use the port definition diagram of the AD2 shown in Fig. 2 and the pin configuration of Op-Amp TLV2371 in Fig. 3 when setting up your circuits.

1.5 Based on Fig. 1, construct the measurement setup for the first-order LPF, as shown in Fig. 4.

1.6 In AD2, use V+ = 5V for V_{DD} and V- = -5V for V_{SS}. Connect Scope Ch. 1 Negative (1-), Scope Ch. 2 Negative (2-), GNDV+, GNDV-, GNDW1, and GNDW2 to a common ground line. For the Op-Amp, connect its GND (pin 4) to the V- (i.e., -5 V).

1.7 Connect the Waveform Generator (W1) and Scope Ch. 1 Positive (1+) to the input V_{in} of the LPF and connect Scope Ch. 2 Positive (2+) to the output (pin 6) of the TLV2371.

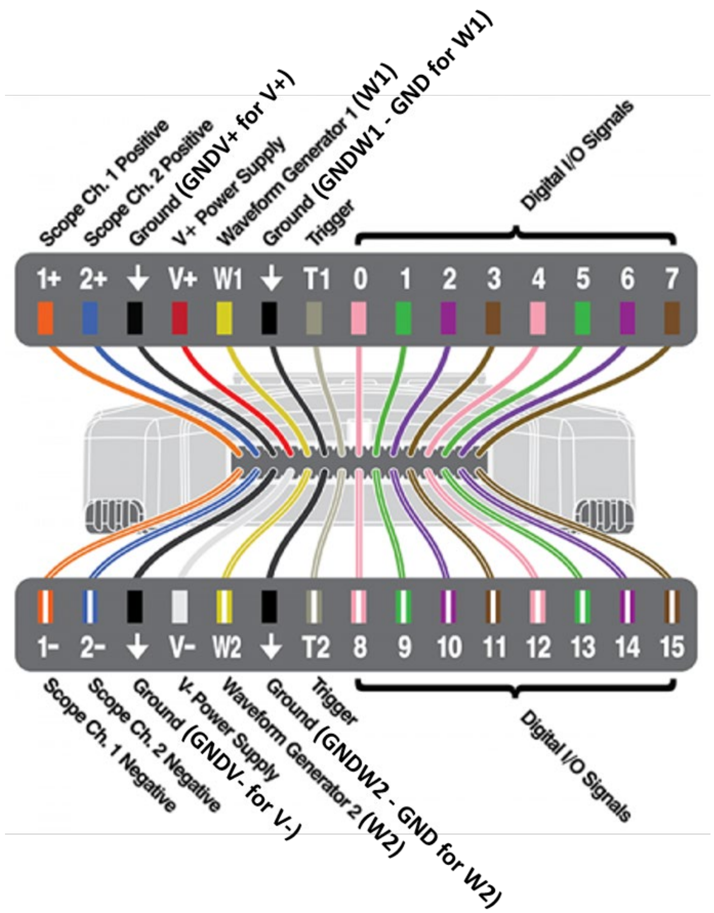

Fig. 2 Diagram for the port definition of an Analog Discovery 2 (AD2)

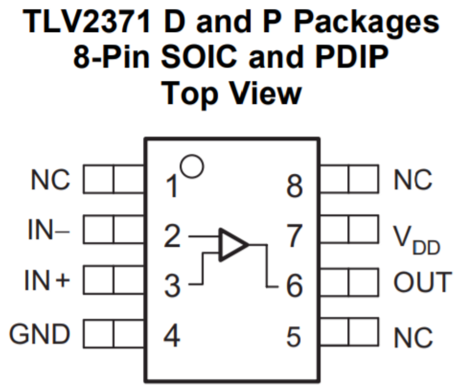

Fig. 3 Pin Configuration of Op-Amp TLV2371

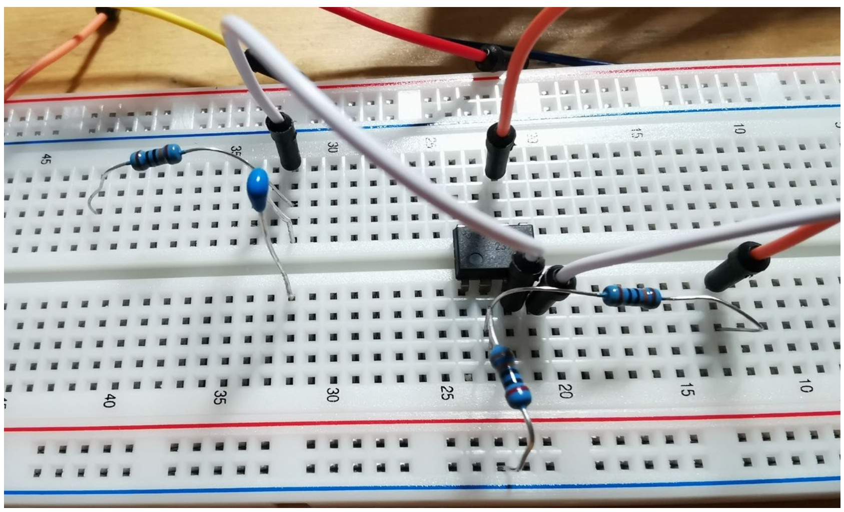

Fig. 4 Experimental setup for the first-order low-pass filter

1.8 **Frequency Response:** In Waveforms, use the Network Analyzer tool and set Wavegen (W1) to generate a sinewave with Amplitude = 100 mV and Offset = 0 V. For frequency, set Scale = Logrithimc, Start = 1 Hz, Stop = 100 kHz, Samples = 501. Measure and record the measured magnitude and phase versus frequency characteristics of the voltage gain in the sheet "Step 1.8" of the Excel file "Lab 5 – Active Filter.xlsx". Replace the screenshot in the sheet with yours, and make sure to capture your date and time to avoid mark deduction.

C. Questions for Part 1

For the first-order LPF designed, answer the following questions with simulated and measured data and discuss any discrepancy between the simulation and measurement results.

Q1. (20 Points) (1) Find the transfer function of the first-order LPF, its low-frequency gain, and its -3dB frequency f_c. (2) Compare the calculated low-frequency gain and the -3dB frequency f_c with the simulated data from Step 1.3 and the measured data from Step 1.8, respectively. Justify/discuss the observation and comparison.

Part 2: Second-order Low-pass Filter

A. SPICE Simulation

2.1 In PSpice, construct the second-order low pass filter (LPF) as shown in Fig. 5. using an operational amplifier TLV2371IP, four 100 kΩ resistors, one 1 nF capacitor, and one 2.2 nF capacitor, respectively. Here, V_{in} provides the AC signal input for this second-order LPF. The DC and AC values of the AC voltage source V_{in} are set to 0 V and 1 V, respectively.

2.2 **Frequency Response:** Conduct AC sweep for V_{out} at the output of Op-Amp TLV2371 in Logarithmic with Start Frequency = 1 Hz, End Frequency = 100 kHz, and Points/Decade = 101. In the simulator window, click the Add Trace icon, choose Plot Window Templates under Functions or Macros, select Bode Plot dB – separate(1), select V(Vout) in Simulation Output Variables, and press OK. Record the simulated magnitude and phase of V_{out} in the sheet "Step 2.2" of the Excel file "Lab 5 – Active Filter.xlsx".

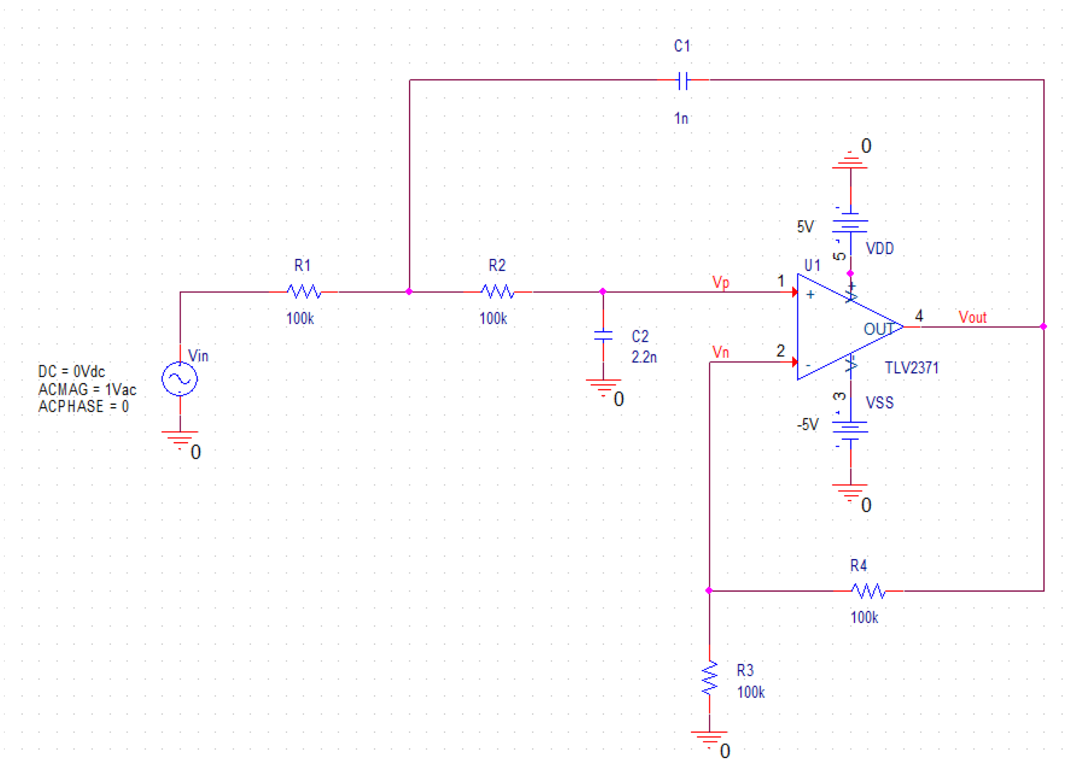

Fig. 5 Schematic diagram of the second-order low-pass filter

B. AD2 Measurement

2.3 Based on Fig. 5, construct the measurement setup for the second-order LPF.

2.4 For the AD2, Use V+ = 5V for V_{DD} and V- = -5V for V_{SS}. Connect Scope Ch. 1 Negative (1-), Scope Ch. 2 Negative (2-), GNDV+, GNDV-, GNDW1, and GNDW2 to a common ground line. For the Op-Amp, connect its GND (pin 4) to the V- (-5 V).

2.5 Connect the Waveform Generator (W1) and Scope Ch. 1 Positive (1+) to the input V_{in} of the LPF and connect Scope Ch. 2 Positive (2+) to the output (pin 6) of the TLV2371.

2.6 **Frequency Response:** In Waveforms, use the Network Analyzer tool and set Wavegen (W1) to generate a sine wave with Amplitude = 100 mV and Offset = 0 V. For frequency, set Scale = Logrithimc, Start = 1 Hz, Stop = 100 kHz, Samples = 501. Measure and record the magnitude and phase versus frequency characteristics of the voltage gain in the sheet "Step 2.6" of the Excel file "Lab 5 – Active Filter.xlsx".

C. Questions for Part 2

For the second-order low-pass filter designed, answer the following questions with simulated and measured data and discuss any discrepancy between the simulation and measurement results.

Q2. (20 Points) Derive the transfer function and calculate the low-frequency gain. Verify the calculated gain using the simulated data obtained in Step 2.2 and the measured data obtained in Step 2.6, respectively.

Q3. (20 Points) Calculate (1) the pole frequency f_o, (2) the cut-off frequency (or -3dB frequency) f_c, (3) the pole quality factor Q, (4) the peak value of the magnitude of the transfer function, and (5) the frequency f_{max} where the peak value of the magnitude of the transfer function happens. Verify the calculated results using the simulated data obtained in Step 2.2 and the measured data obtained in Step 2.6, respectively.

Part 3: **Second-order Bandpass Filter**

A. SPICE Simulation

3.1 In [PSpice](), construct the second-order bandpass filter (BPF) as shown in Fig. 6 using an operational amplifier TLV2371IP, two 240 kΩ resistors, and two 1 nF capacitors, respectively. Here, V_{in} provides the AC signal input for this second-order BPF. The DC and AC values of the AC voltage source V_{in} are set to 0 V and 1 V, respectively.

3.2 **Frequency Response:** Conduct AC sweep for V_{out} at the output of Op-Amp TLV2371 in Logarithmic with Start Frequency = 1 Hz, End Frequency = 100 kHz, and Points/Decade = 101. In the simulator window, click the Add Trace icon , choose Plot Window Templates under Functions or Macros, select Bode Plot dB – separate(1), select V(Vout) in Simulation Output Variables, and press OK. Record the simulated magnitude and phase of V_{out} in the sheet "Step 3.2" of the Excel file "Lab 5 – Active Filter.xlsx".

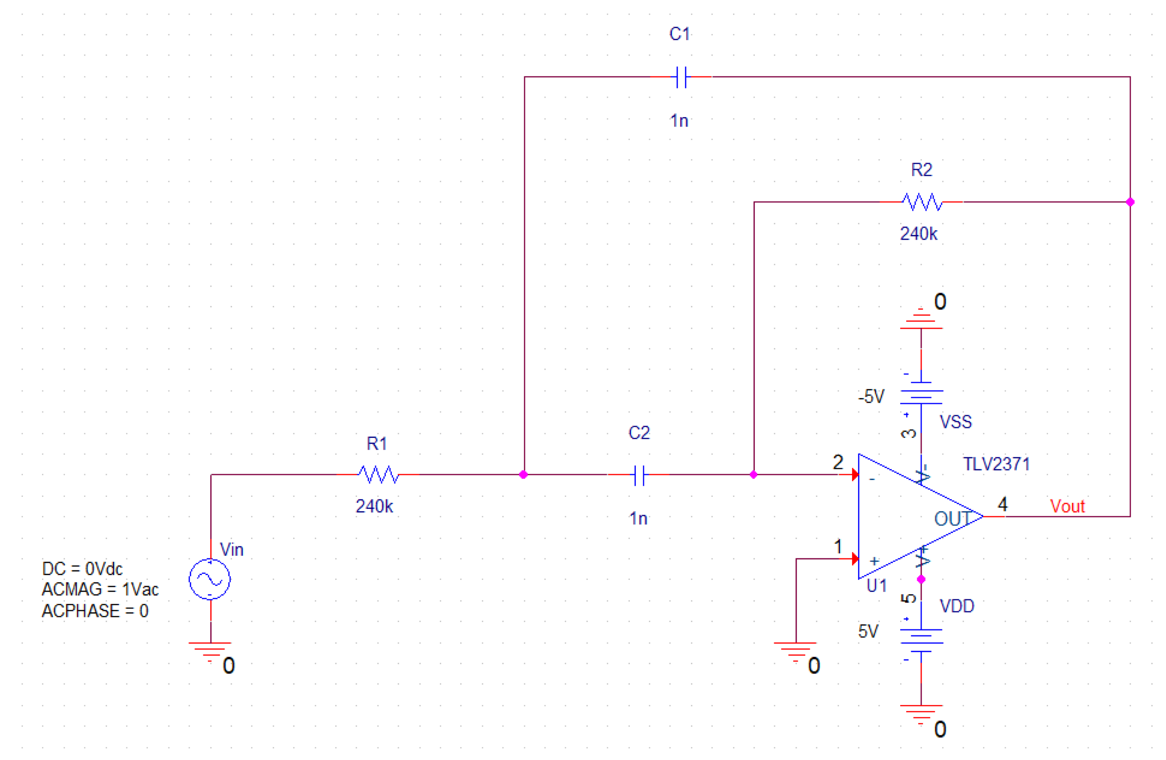

Fig. 6 Schematic diagram of the second-order bandpass filter

B. AD2 Measurement

3.3 Based on Fig. 6, construct the measurement setup for the second-order BPF.

3.4 For the AD2, Use V+ = 5V for V_{DD} and V- = -5V for V_{SS}. Connect Scope Ch. 1 Negative (1-), Scope Ch. 2 Negative (2-), GNDV+, GNDV-, GNDW1, and GNDW2 to a common ground line. For the Op-Amp, connect its GND (pin 4) to the V- (-5V).

3.5 Connect the Waveform Generator (W1) and Scope Ch. 1 Positive (1+) to the input V_{in} of the BPF and connect Scope Ch. 2 Positive (2+) to the output (pin 6) of the TLV2371.

3.6 **Frequency Response:** In Waveforms, use the Network Analyzer tool and set Wavegen (W1) to generate a sinewave with Amplitude = 100 mV and Offset = 0 V. For frequency, set Scale = Logrithimc, Start = 1 Hz, Stop = 100 kHz, Samples = 501. Measure and record the measured magnitude and phase versus frequency characteristics of the voltage gain in the sheet "Step 3.6" of the Excel file "Lab 5 – Active Filter.xlsx".

C. Questions for Part 3

For the second-order bandpass filter designed, answer the following questions with simulated and measured data and discuss any discrepancy between the simulation and measurement results.

Q4. (20 Points) Derive the transfer function and calculate the center frequency gain. Verify the calculated gain using the simulated data obtained in Step 3.2 and the measured data obtained in Step 3.6, respectively.

Q5. (20 Points) Calculate (1) the center frequency ω_o, (2) the pole quality factor Q, (3) the two -3dB frequencies ω_1 and ω_2, and (4) the 3-dB bandwidth $BW = \omega_2 - \omega_1$. Verify the calculated results using the simulated data obtained in Step 3.2 and the measured data obtained in Step 3.6, respectively.