Lab Number 2
EEE 120L - Instrumentation Laboratory

Instrumentation Amplifiers and First Order Filters (Two weeks)

Purpose

1) Design a high voltage gain instrumentation amplifier using the Texas Instruments INA121CP. Then design a cascaded first order bandpass filter using two LM741 operational amplifiers;
2) Construct a prototype of the design and test it for overall ac voltage gain and CMRR, and examine its I/O impedance, and
3) Simulate this circuit.

Upon completion of this laboratory exercise, you should have a good understanding of instrumentation amplifier/filter design and the electrical characteristics of such circuits.

Prelab

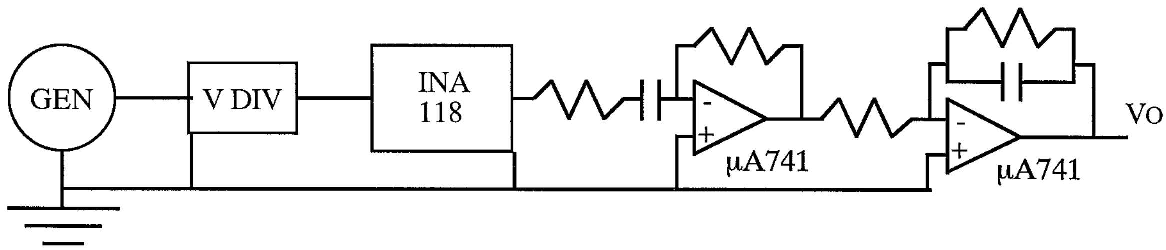
Inspect the data sheet for the Texas Instruments INA121CP on the course website. Note particularly the circuit design of the IC and how its differential gain is determined by selection of a single external circuit component.

Similarly, review the data sheet for the LM741 Op Amp.

Part I – Circuit Design and Verification Using LabVIEW

1. Determine appropriate external components for use with the INA121 (the figure shows a INA118 from past labs) to achieve an instrumentation amplifier with differential voltage gain of 1000 at a frequency of 100 Hz.

Amplifier/Bandpass Filter Block Diagram



2. Read the first-order low-pass and high-pass filter section starting on page 608 in the Nilsson and Riedel text. Section 15.3 of the text brings the discussion to a design solution with op-amps.

3. Determine appropriate external components for use with a LM741 op amp to construct a <u>high-pass</u> first order filter with a -3db cutoff frequency of 10 Hz and a voltage gain of 1 in the "high-pass" frequency region.

4. Similarly, determine appropriate external components for use with a second LM741 op amp to construct a <u>low-pass</u> first order filter with a -3dB cutoff frequency of 1000 Hz and a voltage gain of 1 in the 'low-pass" frequency region.

   Remember that these circuits will be using components from the laboratory stock, so choose available R and C values for your designs that will achieve the desired specifications.

5. Open LabVIEW on your workstation and navigate to Amp_Filter.vi in the EEE_120_LabView_VIs folder (I am trying to place one copy of the folder on the local computer and another copy on the network resource “voyager.”).

   Enter the data required for this VI to verify your design values at a frequency of 100 Hz.

   RUN the VI and compare both peak output voltage and gain to your predicted design values.

   Repeat this for frequencies of 10 Hz and 1000 Hz.

   If your design checks out ok, proceed to Part II below.
   If it doesn't check out, go back and re-analyze your design.

Part II – Instrumentation Amplifier/Bandpass Filter Prototype Construction and Testing

1. At one end of your protoboard, mount the INA121 and wire it according to your design.

Make sure both dc power supplies are connected as required.

Using a signal generator set to deliver a 20 Vp-p sinusoidal output signal at a frequency of 100 Hz, construct the resistive voltage divider circuit shown in Figure 1 below.

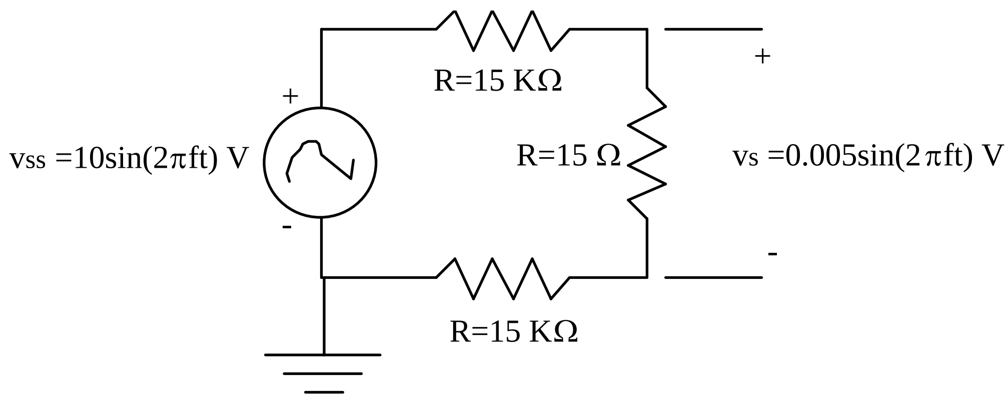


Figure 1. Signal Generator Voltage Divider Circuit

It should deliver a Thevenin equivalent sinusoidal voltage (v_s) of approximately 5 mV peak amplitude through an equivalent source resistance (R_S) of 15 Ω to the differential input pins of the INA121.

Note that this circuit is slightly different from the one used in the previous laboratory in that the voltage (v_s) is <u>not</u> referenced directly to "ground".

Make sure you have a common reference ground in your circuit.

The ground side of the Agilent signal generator, pin 5 (ref) of the INA121, and the ground terminal of your dc power supplies must all be connected together (common ground).

2. When you are satisfied with your wiring, connect the instrumentation amplifier inputs across the 15 Ω resistor of the voltage divider.

Using the oscilloscope, measure and record the voltage at the output pin of the IOA.

Calculate the ac voltage gain (A_V).

How does it compare with your design value?

3. Because this instrumentation amplifier has such a high differential input impedance, it is almost impossible to measure it using general-purpose test instruments such as we have in the laboratory.

In order to get an impression for how large this input impedance really is, insert a 51 kΩ resistor in series with the + input of the instrumentation amplifier.

Again, measure the voltage at the output pin of the IC.

Does the 51 kΩ resistor make any difference?

Consider the high input impedance of the INA121 circuit design and explain why.

4. Remove the 51 kΩ resistor and reconnect the + input of the instrumentation amplifier directly to the voltage divider.

Now place a 2.2 kΩ resistor between the output pin of the IOA and ground (output load resistor).

Measure the voltage across the resistor and compare it to the "open circuit" output voltage.
Is there a difference?
Try this again with a 1 kΩ resistor.
If there is still no difference, try a 510 Ω resistor.

What do these measurements indicate about the output current limit for your IOA?

What is the minimum load impedance you should design for if you expect to achieve the desired 10 V peak-to-peak output voltage swing?

5. Remove the load resistor.

Set the generator to produce a 5 V p-p sine wave.

Disconnect the signal generator from the voltage divider circuit and connect BOTH the + and - inputs of the instrumentation amplifier directly to the + output terminal of the signal generator to produce a “common mode” input signal.

Again, measure the voltage at the output pin of the IOA.
(Note you may have to edit the voltage scale after recording in order to magnify the signal and measure the small common mode output voltage. Ignore any "spike" noise that may be present and measure (estimate) the p-p amplitude of the 100 Hz sinusoidal signal.)

Determine the common mode gain for your amplifier.

Calculate the common mode rejection ratio in decibels (CMRR in dB).

When you are finished, return your input connections to the voltage divider as before, and reset your signal generator amplitude to 20 V p-p.

6. Next to the INA121 on your protoboard, mount one LM741 op amp and wire it according to your high-pass filter design.

Connect the filter input to the instrumentation amplifier output.

Using the oscilloscope, to measure the filter output voltage.

How does it compare to the instrumentation amplifier output voltage measured in part 2 above?

Now change the signal generator frequency to 10 Hz and measure the filter output voltage again.

Is the amplitude of the output voltage reduced by -3dB (a factor of .707) at this frequency as predicted by your design?

Explain.

7. Right next to the high-pass filter mount another LM741 op amp and wire it according to your low-pass filter design.

Measure the output voltage of the low-pass filter at 100 Hz.

Is it what you expected?

Change the signal generator to a frequency of 1000 Hz and measure the low-pass filter output voltage.

Is the amplitude reduced by -3dB at this frequency as predicted by your design?

Explain.

Measure the output voltage of your low-pass filter at 100 Hz. Nominally, it should be 5 V peak.

Is it greater or less than you expected?

Verify the phase at the upper and lower corner frequencies of the bandpass filter.

What instrumentation amplifier and/or filter circuit modifications could you make to achieve an output voltage closer to this design expectation?

For report documentation, show your completed circuit output at frequencies of 10 Hz, 100 Hz, and 1000 Hz.

Part III – Filter Simulation

Simulate your circuit. In Multisim, use the INA133 device model since this model exists already (rather than create a model) and is close to the INA121 parameters.

Create the circuit incrementally. First create the INA121 gain stage without the bandpass filter. Analyze the device for dc bias and then ac analysis between 1 Hz and 100 kHz.

Then separately create the bandpass filter section (only) of your circuit using LM741 op-amps from the parts library. Run a transient analysis for the first 100 ms. Then perform the ac analysis between 1 Hz and 10 kHz. Verify both stopband (upper and lower) and passband behavior .

Now create the complete amplifier and filter circuit and run the same dc bias and ac analysis.

Report:
The report will consist of a title page (as always) and a discussion of the hand analysis, simulation and measurement. The hand analysis should include the frequency behavior of the filter. Verify the phase at the upper and lower corner frequencies of the bandpass filter.

EEE 120L Virtual Instrument Documentation

Amp_Filter.vi

This VI is designed to allow the user to input specified parameters for design of an instrumentation amplifier and first-order bandpass filter. It uses a formula node to calculate the peak output voltage (Vop) and ac voltage gain (Av) for the design when the RUN button is pressed. It also displays a representation of the sinusoidal output voltage as a function of time.

Input parameters:
Vsp is the peak input source voltage;
frequency is the frequency of the input signal in Hz;
RG is the gain control resistor for the Texas Instruments INA 121 instrumentation amplifier,
RH1, RH2, and CH2 are the relevant circuit components for the first-order high-pass filter;
RL1, RL2, and CL1 are the relevant circuit components for the first-order low-pass filter.