



Class #24: Transient Simulations and Voltage Follower Circuits

Purpose: In this experiment, we will use simulation software (LTspice) to study transient responses using ideal models. We will also introduce the amplifier circuits used for isolation (voltage follower circuits).

Background: Before doing this experiment, students should be able to

- Review online background materials.
- Build and operate simple circuits on a Protoboard.
- Analyze circuits consisting of combinations of resistors, capacitors and inductors.
- Do a transient (time dependent) simulation of circuits using LTspice
- Review the background for the previous experiments.

Learning Outcomes: Students will be able to

- Implement transient simulations on LTspice and extract waveform characteristics from measured results.
- Build a voltage follower amplifier circuit and verify that it will not affect circuit characteristics.

Resources Required:

- LTspice
- M1K board and Alice tools
- Parts kit

Helpful links for this experiment can be found on the course website under Class #24.

Part A – Transients in LTspice

Background

We have used LTspice to simulate the DC bias points of circuit (operating point, .op). In this experiment we will use LTspice to simulate circuits with voltage and currents that vary with time. This analysis is called transient simulation, which is a fundamental part of engineering modeling. In the last two experiments, we have used resistors, capacitors and inductors. In LTspice, we are familiar with the placing resistors on the schematic. Capacitors and inductors are also available as hotkeys, to the ‘right’ of the resistor hotkey, as seen in Figure A-1. Since our voltage source in the last two experiments also has transient characteristics (changes with time), we will also need to step up “Advanced” voltage sources. Once the circuit is complete, a transient simulation must be setup. The example below shows a transient simulation that runs from 0 to 5ms with a maximum step size of 1 μ s. Once the simulation is run, it displays an output similar to what you would see if you hooked the circuit to an oscilloscope. Figure A-2 contains a sample diagram.

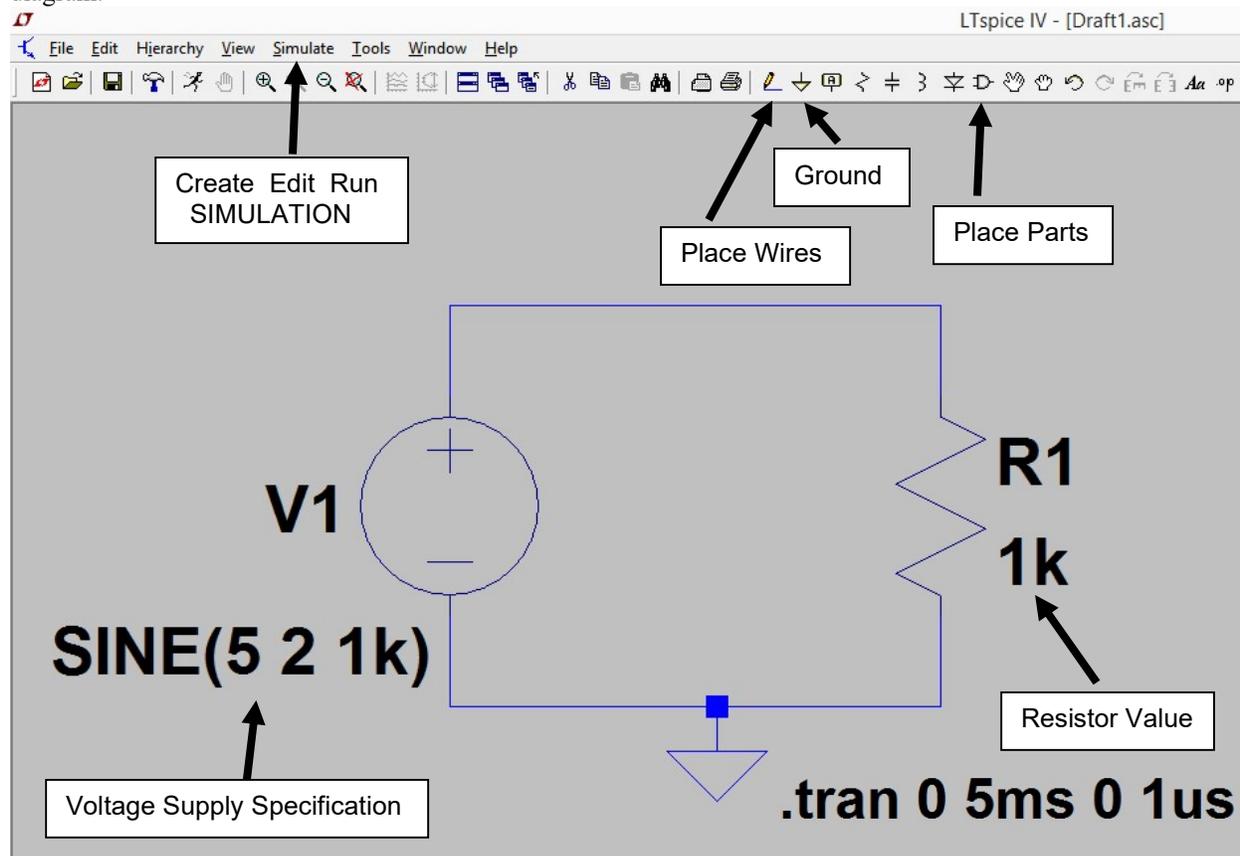


Figure A-1. Simple Circuit with Sinusoidal Voltage Source

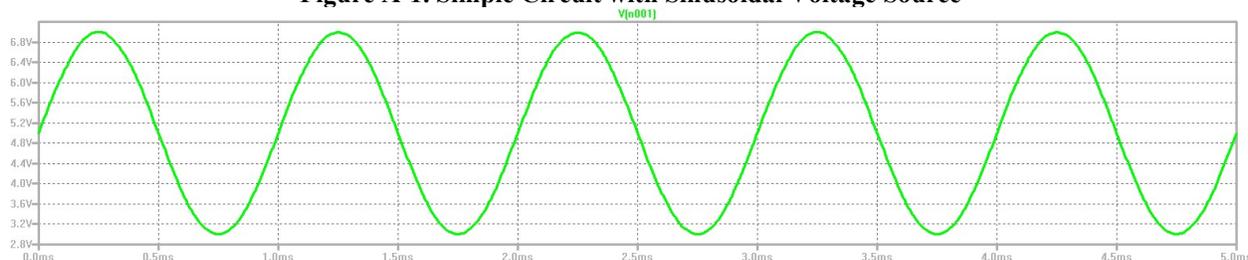


Figure A-2: Transient Voltage across the Resistors.

Experiment (Simulation)

Opening a New Simulation

In this part of the experiment, we will simulate a familiar voltage divider circuit.

Drawing the Circuit

Figure A-3 is a picture of the *LTspice IV* main screen with the circuit we will be drawing.

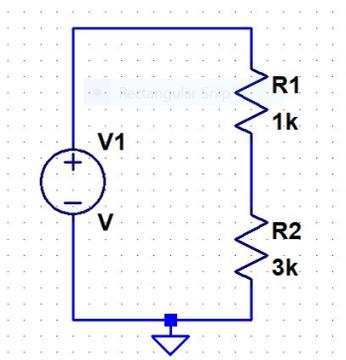


Figure A-3. Voltage Divider Circuit

- Simulate the circuit shown in Figure A-3, setting the values of the resistors and remembering to include the ground node.
- Right click on the voltage source. Rather than setting a DC value for the source, click the Advanced Button. A screen similar that shown in Figure A-4 will open.
- We will initially be studying sinusoidal sources, so click on the SINE radio button. More setting options will now be available. We will keep the configuration fairly simple and only set the amplitude (to 4V) and the frequency (to 1kHz). The DC offset will automatically be set to 0V.

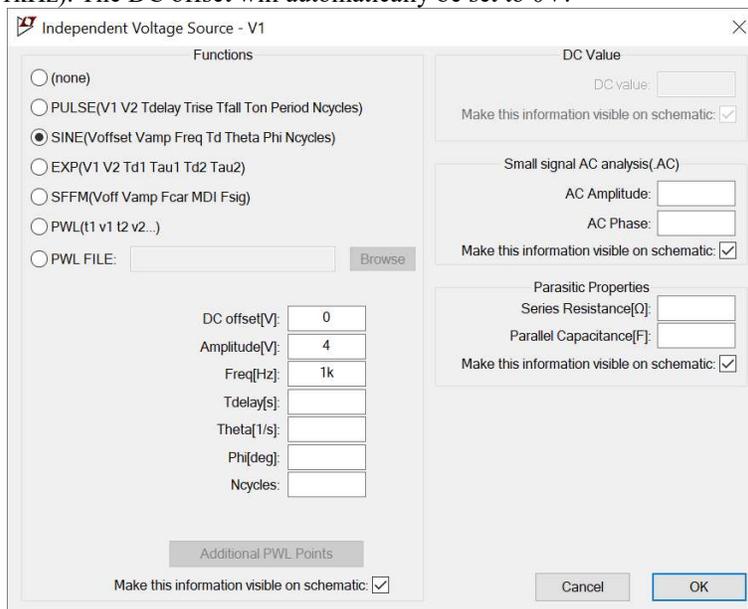


Figure A-4. Sinusoidal Voltage Source Options

The circuit will now appear as shown in Figure A-5

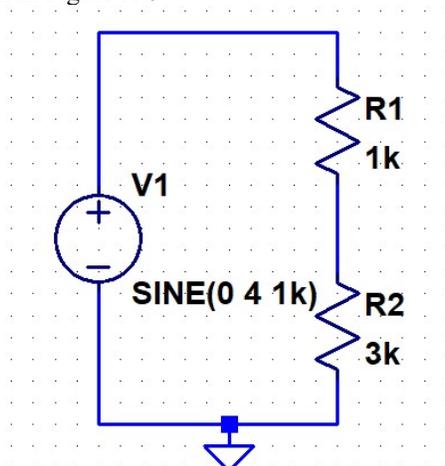


Figure A-5. Fully Implemented Circuit with Sinusoidal Source

Setting Up the Analysis

After we have defined all the components, wired them up and changed their values appropriately, we are ready to do some analysis.

- Find the *Simulation* drop-down menu and click and select *Edit Simulation Cmd*. You will get the window shown in Figure A-6. Select the tab for the type of analysis you wish to perform. In this case it is *Transient* since we are interested in output that is time-dependent like a scope. The simulation will begin at time $t = 0$, but we can choose to start saving data after that. Thus, we specify the *Stop Time* (the end time of the simulation), the *Time to Start Saving Data*, and the *Maximum Timestep* (the resolution of our simulation). **It is important to choose a maximum time step much less than the period of the signal.** Here we choose, 5ms, 0 and 1 μ s, respectively. Note that there is no μ so we have to use the letter u.

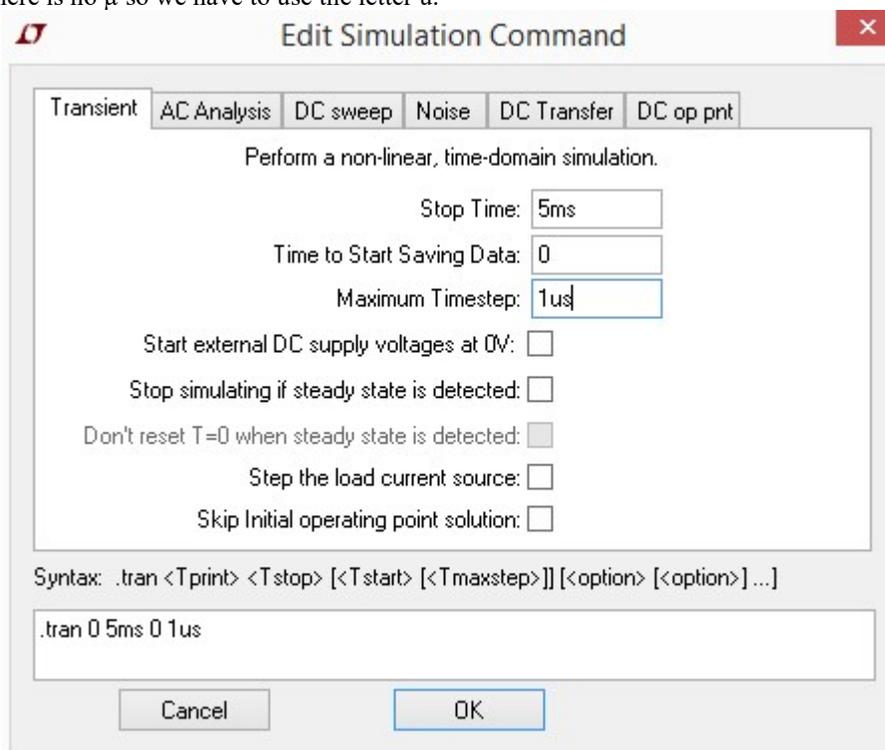


Figure A-6. Simulation Settings

The final time (*Stop Time*;) has been set at 5ms because the period of a 1kHz sine wave is 1msec. This allows us to see five periods. The *Maximum Timestep* needs to be set so that you get a reasonable representation of the output. A step size that is too small will take a long time to run, a step size that is too big will give you an under-sampled representation of the output. A step size between 1/100 to 1/1000 of the run time is reasonable. The analysis will begin saving data at 0 seconds. Note again that there should never be any spaces between the number and its units. Click on the *OK* button when you are finished putting in the numbers. You will be able to place the information on the simulation anywhere on your circuit. Choose a space where it is easy to read and does not block anything else.

- When you have finished, you can run the simulation either by pushing the button that looks like a stick person running or select *Run* from the *Simulation* drop-down menu.

Transient Analysis

You are now ready to do the simulation.

- Click on the *Run* button. A set of axes with no signals shown will appear. You can now add voltages wherever you wish by left-clicking with your mouse. You will see that whenever you hover over a wire with your mouse, a symbol like a voltage probe will appear. Click where you want to know the voltage. If you want to plot current, hover over a device symbol and your mouse symbol will change to a current probe. If you click on a device, the current through it will appear on the plot. In the plot shown in **Figure A-7**, the input and output voltages for an example voltage divider are displayed.
- You should get something like (but not identical to) results in Figure A-7. The voltage divider expression we used in the previous classes can be applied to sinusoidal signals as well. For sinusoidal signals, the amplitude of the sine wave changes based on the voltage divider ratio. We can then perform the same comparison between source voltage amplitude and resistor voltage amplitude as we did in experiment 7 where we compared the source DC voltage and the resistor DC voltage. For your voltage divider circuit, do we see the same ratio of voltage amplitudes as seen for DC voltages in experiment 7?
- It is useful to know how to copy plots and paste them into word. Click on (highlight) the window with the transient plots. Under the *Tools* menu, click on “copy to clipboard”. Now there is a bitmap in the clipboard that you can paste into any application. Open word and paste the bitmap in should see a plot of the voltages. You can do the same thing with the schematic window.
- In your report, include the schematic, source voltage and VR2 for your voltage divider circuit.

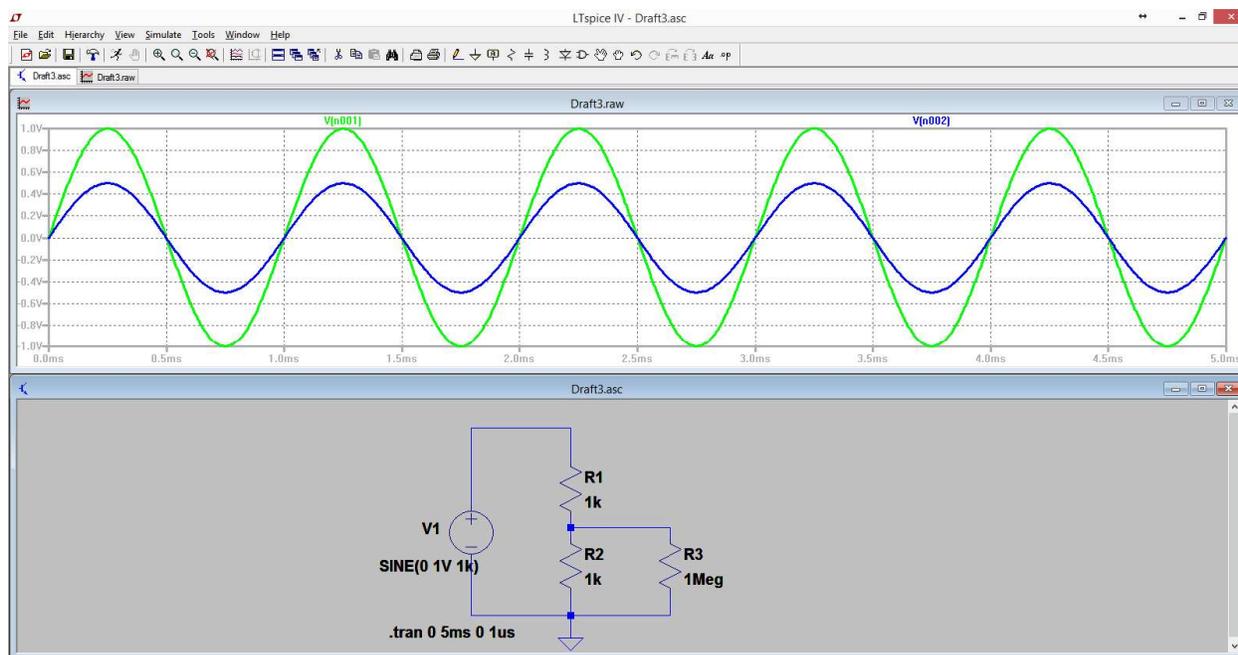


Figure A-7. Source Voltage and VR2 Voltage

- Right click on the voltage source, click the Advanced button, and change the type of source to a square wave using the PULSE radio button.
- We will generate a square wave pulse with a 1kHz frequency (1ms period), maximum voltage of 2V, minimum voltage of 0V and 50% duty cycle (half the time v_{max}, half the time v_{min}). To make LTspice behave well, we will also need to set up some of the other options. The full setup is shown in Figure A-8.

In the settings for the PULSE square wave, the boxes have the following application

Vinitial: Either the maximum or the minimum voltage (whichever V_{on} is not)

V_{on}: Either the maximum or the minimum voltage (whichever V_{initial} is not)

Tdelay: Time delay, very small for our purposes

Trise: Rise time, very small for our purposes

Tfall: Fall time, very small for our purposes

Ton: The amount of time the pulse is high, half the period for a 50% duty cycle

Tperiod: The total time for one cycle, 1/f

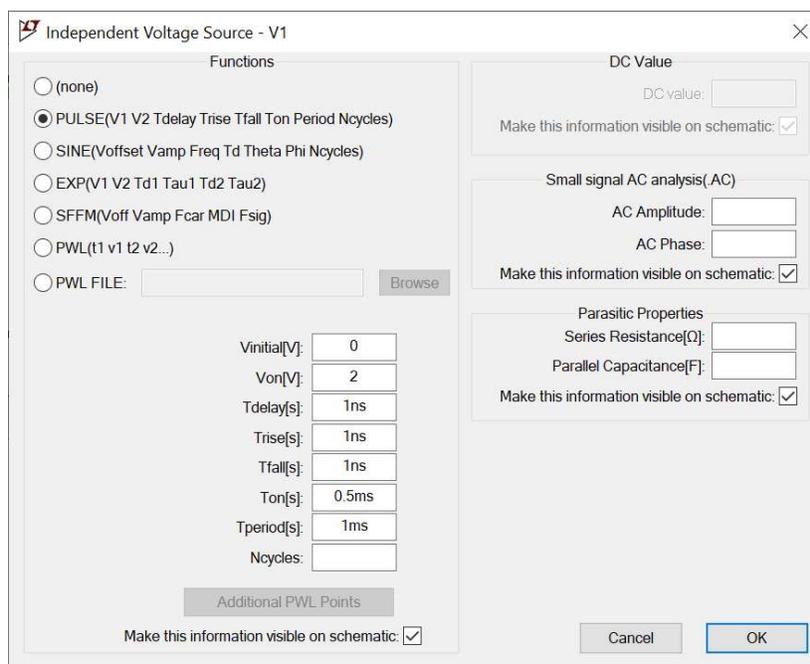


Figure A-8. Square Wave Voltage Source Options

- Using the same simulation profile, run the simulation with the square wave source. Is the output again consistent with the expected results for the voltage divider?
- In your report, include the schematic, source voltage and VR2 for your voltage divider circuit.



Part B – RC, RL and RLC circuits

- Using the PULSE setting for the voltage source, implement the RC circuit shown in Figure B-3 of experiment 22.
- Plot the voltage across the capacitor. Is the plot similar to that seen when using the M1K Board?
- Pick two points on the plot when the voltage is decaying to zero and estimate the time constant using the

formulat $\tau = -\frac{t_2 - t_1}{\ln\left(\frac{V_2}{V_1}\right)}$ (the inverse of the attenuation constant equation from the last experiment). Is the

estimated time constant close to the expected value?

- Repeat the above for the RL circuit from experiment 23 when $R = 470\Omega$. You will need to change your voltage source period and on-time based on your CH A function generator settings. You will also need to change the simulation settings stop time such that you can see five periods in your measurements. You should also modify your step size. In this case, measure the current through the inductor instead of the voltage across the resistor. Use the same transient calculation as above to estimate the time constant, replacing voltage values with current values. Are the results again consistent with the calculated time constant?
- Repeat the above for the RLC circuit from experiment 23 when $R = 10\Omega$. You will again need to modify your voltage source settings and simulation settings based on your experiment 23 CH A function generator settings. In this case, you should set the simulation settings such that you can clearly see the decaying sinusoid rather than seeing five cycles of the source frequency.
- Using the same process in experiment 23, estimate the oscillation frequency and attenuation constant using your plot results. How do those values compare with the experimental estimates? Why is it important that we have a thorough understanding of our components and equipment when we implement simulations?

Part C – Isolation Amplifiers (Voltage Dividers)

A new component we will introduce is called an amplifier circuit. There are many applications for amplifier circuits. We will implement a voltage divider circuit, which is used to ‘measure’ voltage without interfering with the regular operation of the circuit. An amplifier is an active device, which means it needs power (voltage sources) to work correctly. The circuit we will implement is shown in Figure C-1.

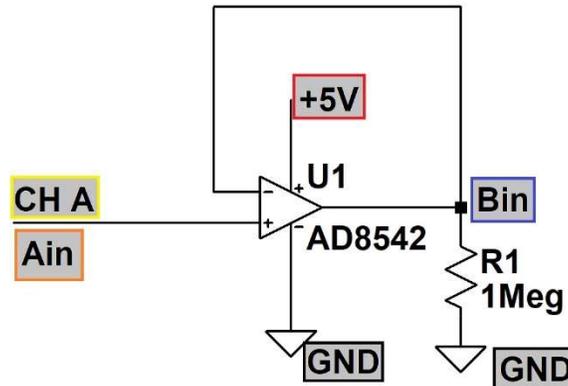
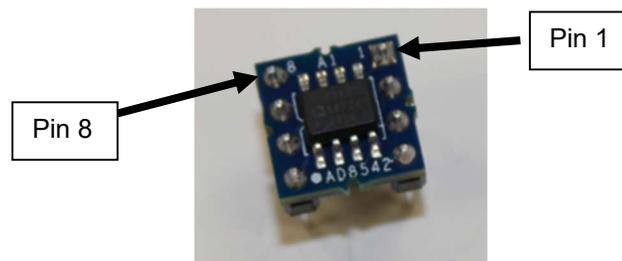


Figure C-1. Basic Voltage Follower Circuit

- An operational amplifier is a two input (shown on the ‘left’ flat side of the device), one output (shown on the right ‘triangle point’ of the device) component. In many operations, the circuit involves feedback from the output to the input. The feedback will always be connected to the negative input. In this circuit, the feedback consists of a wire connecting the input and the output.
- Find the AD8542 amplifier chip surface mounted on a small PCB in your parts kit and place it on your protoboard. Make sure the chip spans the chip gap such that the legs on each side are in separate 5-hole rows. There should be four legs on one side of the gap, each in a different row and four legs on the other side of the gap.



- The physical layout of the chip is shown in Figure C-2 below. Identify the pin numbers 1 to 8 on the AD8542 board by finding pin 1 and counting clock-wise all the way to pin 8. Pin 1 and 8 are marked on the board as shown above.
 - Connect **CH A** (function generator) from M1K to positive input pin 3 of the chip (labelled in Figure C-2).
 - Connect the output pin 1 to the negative input pin 2 of the chip using a wire. That is the feedback wire from the output to the input, shown in Figure C-1.
 - Connect the **+5V** voltage supply from M1K to pin 8 of the chip.
 - Connect the **GND** from M1K to pin 4 of the chip.
 - Connect **Ain** from M1K to pin 3 to measure voltage waveform supplied by CH A. CH A needs to be configured in Split I/O mode.
 - Connect **Bin** from M1K to pin 1 to measure output voltage waveform. Channel B needs to be configured in Split I/O mode.
 - Connect a 1Meg Ω resistor from the output pin 1 to ground. (This step is not really necessary since the M1K board voltmeter acts like a 1Meg Ω resistor.)

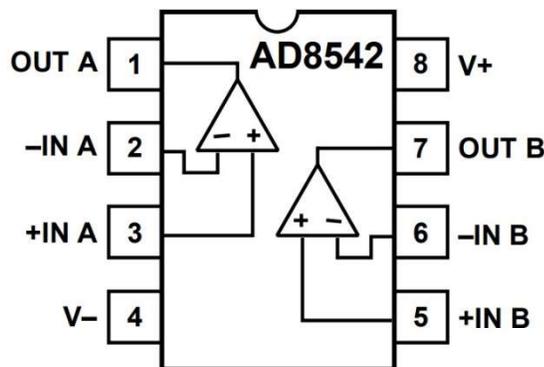


Figure C-2. AD8542 Chip Layout

- Configure CH A to generate a 1kHz, 1V amplitude, 1V offset sinusoidal wave. i.e. $\sin(2 \times \pi \times 1000 \times t) + 1$. In other words, Min = 0 V, and Max = 2 V. Frequency = 1000 Hz.
- Measure the function generator voltage using Ain and the amplifier output using Bin as described earlier. Display 2-3 cycles of both input and output voltage measurements on Alice desktop. Adjust V/div and ms/div for appropriate horizontal and vertical scaling of the oscilloscope view.
- If everything is working correctly, the input and output voltage waveforms should be the same.
- Capture the image of the voltage measurements and put it in your report.

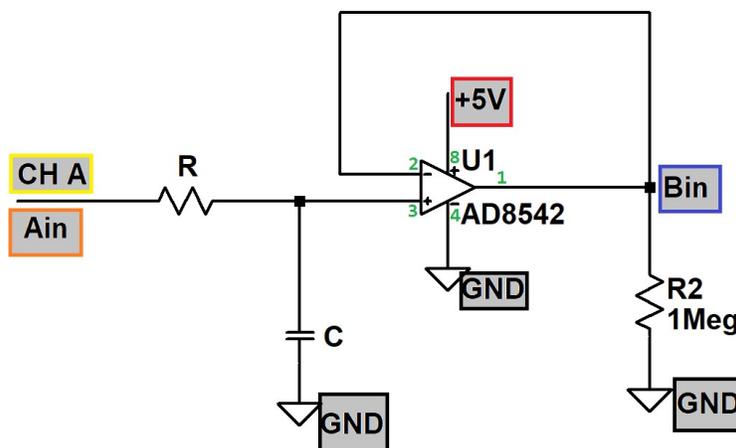


Figure C-3. Experiment 19 B-3 RC Circuit with Voltage Follower

- Add the resistor and capacitor values from experiment 22 B-3 circuit to the circuit you just built, as shown in Figure C-3. You will need to remove the CH A from the op-amp chip and connect it to the resistor as shown. The capacitor node is now connected to the positive input (pin 3 of the chip).
- Measure the function generator voltage using Ain and the amplifier output using Bin from M1K.
- The output voltage should be the same as you saw in experiment 22.
- Capture the image of the voltage measurements and put it in your report.