

Lab 1: Voltage and Current Sensors

Introduction

Simulation is a really important tool in engineering and science. Essentially, rather than actually building something in the real world, we can model the system using mathematics, and then use a computer to evaluate those equations to see how different parameter changes affect the system behaviour.

The objective of this lab is to use LTspice to model the AC source-load setup used in the project, understand its behaviour and then with this information develop the voltage as well as current sensors needed for the project. LTspice is a [SPICE](#)-based simulation tool that can help us analyse the behaviour of complex electrical circuits. LTspice can be downloaded for free from the [Analog Devices](#) website and is a tool widely used in industry for verifying design ideas. As with any simulation tool, the results given by LTspice are only as reliable as the details captured in your circuit model. Students should appreciate that it is extremely difficult to model the behaviour of real-life circuits accurately, and therefore should exercise caution when interpreting simulation results. For this reason, students must first use a combination of theory and practice to predict the behaviour of the circuit under study so that we can confirm to ourselves that the simulation results are actually correct. To help students get used to this form of thinking, since you will eventually have to think this way as a 'real life design engineer', all questions in this lab require students to derive theoretically predicted behaviour of each circuit before simulating. The simulation tool is then used to validate your theoretical and practical understanding of the circuit analysed. If your theory doesn't match with simulation results, then chances are that you have done something wrong either in your theoretical analysis or the simulation.

Read the entire lab manual before you start. This lab should take you approximately **6 hours**. You should aim to complete Parts 1 and 2 during the 1st support session, and Parts 3 and 4 during the 2nd support session. To be able to do so, you will need to read this document carefully, before attending the 1st support session. The final answers should be written in this document, with supporting calculations shown in your logbook. Before starting the lab, pull the lab repository from GitHub as it contains the LTspice model(s). Remember to commit and push your saved work to the lab repository on GitHub after completing each task.

Part 1: Revising Basics

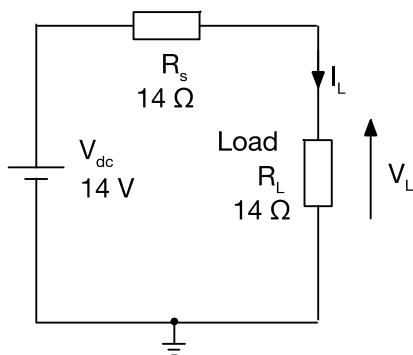
The first part of this lab intends to remind you of what you have learnt in ELECTENG 291. We will get you to use LTspice to do transient time simulation of a few very basic circuit models. Since you have already learnt how to use LTspice to draw a circuit model, we have provided you an LTspice file containing partially completed circuits that you could use for this lab. Here we use ideal device models as the devices you will be using are well within specifications and thus behave close to ideal devices. However, it should be noted that your simulation results are only as good as the underlying models used. As the device models used are not exact replicas of the real-world behaviour, simulation results always have errors and you must be mindful of these errors. If you must use detailed device models, LTspice contains a comprehensive library of device models. You can also

download 3rd party models or even create your own device models if required to improve the accuracy of your simulations.

As you learnt in the previous semester, before you can start simulating, you must first draw your circuit in the LTspice schematic editor window by adding components, wiring, and assigning values/part numbers. You must include a ground node, which is taken as the 0V reference node by the simulator. LTspice converts this graphical input given by you to something called a "Netlist", which is a text file that describes the interconnections between your components. SPICE then uses Nodal Analysis to work out all the voltages and currents in your circuit starting from 0s and stepping through the calculations in small "Timesteps" until the "Stop Time" you define under "Transient Analysis Command". Make sure to select a "Stop Time" that is sufficiently large so you can observe the steady-state behaviour of your circuit. But don't make the stop time very long as it will cause the simulation to run for a long time. Although LTspice tries to intelligently determine a suitable "Timestep", it may help in many occasions to define the "Maximum Timestep" to obtain accurate results. The coarser the "Timestep", the larger the error, but a really small "Timestep" can lead to a longer simulation time due to increased processing demand. As a designer, you must strike a balance and a good starting point is to select a "Timestep" that's about 1/10th to 1/100th of the period of the fastest frequency component in your circuit. In some cases, it also helps to define the initial conditions of your circuit to accelerate convergence of your simulation. LTspice by default tries to evaluate a suitable initial condition for your circuit, which in some cases may lead to inaccurate results.

After you "Run" the simulation, you can view voltage, current, instantaneous power, etc. waveforms in the waveform viewer by probing the circuit. You may also enter mathematical equations to format the data displayed in the waveform window. You can use cursors to measure salient features of your waveforms. You may also use the built-in capabilities of the waveform viewer to obtain RMS, Averages, and Integrals of the waveforms. However, note that these values are based on the waveforms displayed on the window and not having an exact number of cycles will lead to an incorrect reading. Remember to "Run" the simulation every time you make a change to the circuit. If you have forgotten how to use LTspice, then refer to the ELETENG 291 labs.

Q 1.1: We are going to analyse the simple circuit below to understand the basic functionality of LTspice:



First, using your theoretical understanding, determine the load current (I_L), load voltage (V_L), and power (P_L) delivered to the load resistor (R_L).

$I_L: 0.5 A$

$V_L: 7 V$

$P_L: 3.5 W$

Q 1.2: After predicting how the circuit should behave theoretically, you should now simulate the circuit in LTspice and validate your theoretical analysis. Use the LTspice model provided to you (labelled Part 1A).

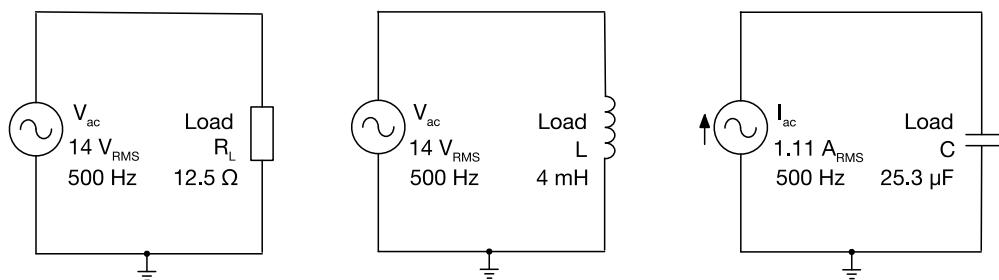
Parameter	Theoretical Value	Simulated Value
I_L	$0.5 A$	$0.5 A$
V_L	$7 V$	$7V$
P_L	$3.5 W$	$3.5 W$

Comments (state if results match/differ):

The theoretical and simulated results are match.

Hopefully, in the above case, the output of LTspice matches your theoretical analysis.

Q 1.3: We are now going to extend our analysis into the following 3 AC circuits to understand the importance of timestep, initial conditions, and settling time (note that the circuit with the capacitor is driven by an AC current source):



What would be an appropriate "Maximum Timestep" and "Stop Time" to be used for this simulation?

Time Step (assuming 1/20th of the period of 500 Hz): $0.1 ms$

Stop Time (assuming we'd like to capture 200 cycles): $0.4 s$

If you simulate these 3 circuits, in their ideal form, you will observe that both the inductive and capacitive circuits will not reach a steady-state within the "Stop Time" you defined (i.e. 200 cycles). As a result, your simulation results for the inductive and capacitive loads will not agree with circuit theory, which would assume these circuits would have reached steady-state. This is because the inductor and capacitor are modelled to be ideal (i.e. resistance of the circuit is 0Ω) and as a result the time constant of these two circuits are equal to infinity. This is a common problem encountered when trying to simulate the behaviour of a real circuit. Therefore, it is common practice to add small resistive components in some branches of the circuit to help the simulation settle down. These resistors help reduce the time constant of the circuit and enable the simulations to reach steady-state faster. It should be noted that adding a resistive component changes the circuit behaviour and thus you should be careful to keep it to a comparatively small value to minimise the errors introduced.

In real life, both inductors and capacitors have some resistive component associated with them. For example, the winding of an inductor has some resistance, which is often referred to as the Equivalent Series Resistance (ESR) of an inductor. These resistive components associated with real circuit elements, help a real circuit reach a steady-state.

Q 1.4: In the simulation model provided (labelled Part 1B-1D), add a 0.2Ω resistor in series with the inductor and a 790Ω resistor in parallel with the capacitor to help the two circuits reach steady-state conditions faster. What is the time-constant of each circuit? Will your simulation reach steady-state well before the "Stop Time" you have defined? (i.e., is the "Stop Time" larger than 5 times the time constant?)

Time constant of circuit with 4mH inductor: 0.02s

Time constant of circuit with $25.3\mu\text{F}$ capacitor: 0.019987s

Would simulation reach steady-state:

Both will reach steady-state before defined stop time, 0.4s

Q 1.5: Calculate the theoretical load current ($I_{L(\text{RMS})}$), load voltage ($V_{L(\text{RMS})}$), peak instantaneous power ($P_{L(t)}$), and average power delivered by the source (P_{in}) for the circuits shown above. When calculating, ignore the impact of the resistive components you added in to the inductive and capacitive circuit. This is because we have chosen them not to significantly impact the behaviour of the each circuit. Simulate these circuits in LTspice using the models provided (labelled Part 1B-1D) and compare with the theoretical results (remember to modify these models as per Q1.4 above).

Circuit with 12.5Ω Resistor

Parameter	Theoretical Value	Simulated Value
$I_{L(RMS)}$	1.12 A	1.1127 A
$V_{L(RMS)}$	14 V	13.908 V
Peak $P_{L(t)}$	31.36 W	31.1539 W
P_{in}	15.68 W	15.475 W

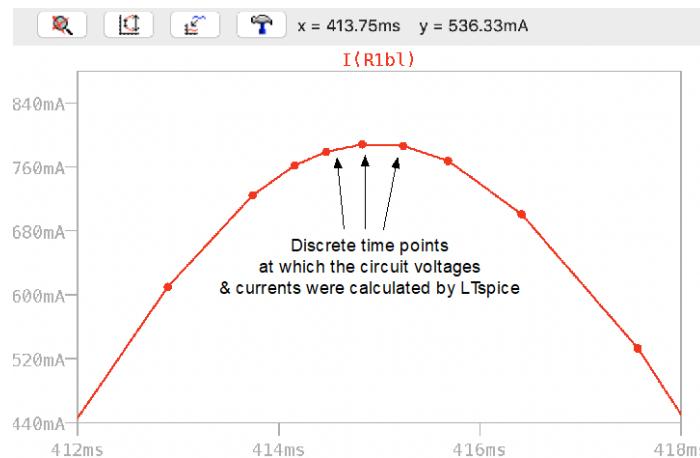
Circuit with 4mH Inductor

Parameter	Theoretical Value	Simulated Value
$I_{L(RMS)}$	1.11 A	1.0688
$V_{L(RMS)}$	14 V	13.914
Peak $P_{L(t)}$	15.54 W	14.9197 W
P_{in}	0	0

Circuit with $25.3\mu F$ Capacitor

Parameter	Theoretical Value	Simulated Value
$I_{L(RMS)}$	1.11 A	1.1032 A
$V_{L(RMS)}$	13.9654 V	13.898 V
Peak $P_{L(t)}$	15.502 W	14.675 W
P_{in}	0	0

Q 1.6: Your simulated results for the 12.5Ω resistive load will be somewhat different to what you predicted from circuit theory. Zoom to observe simulated waveforms near the peaks and you will see granularity of the waveform shape caused by the large timestep:



How can we minimise the granularity and improve the accuracy? What would be a drawback of doing so? Simulate the resistive load by setting the timestep to a finer value and state your observations.

Yes, I can minimise granularity and improve accuracy by using a smaller max timestep in the simulator. However the drawback of doing so is that it increases the simulation time because the computer has to perform more calculations.

When I simulate the resistive load with a smaller timestep I observe the waveform appears much smoother, especially when I zoom in on the peaks. The measured value is more closer to my theoretical calculation. I think it's because the simulator is now calculating points closer to the

Q 1.7: Modify the simulation by removing the resistive components you added as per Q1.4 and run it. Compare the simulation results you obtained from the modified circuits, which now represent an ideal inductive load and an ideal capacitive load, to show how the resistors help these circuits reach steady-state. State your observations.

I settled stop time to 40s, which is better to observe the trend of the waveform.

- Inductor: With removing the resistor, the waveform is pretty same. The current for the circuit without resistor shows an unstable waveform which fail to converge and never reach steady-state. In contrast, the current waveforms with resistor reaches steady-state after its transient decays.
- Capacitor: The waveform from the circuit with resistor shows the stable steady-state voltage after the transient decay. In comparison, the waveform from ideal circuit is unstable and distorted, also never reach steady state.

For both the inductor and capacitor circuits, the power waveforms from ideal circuits are unstable and never reach steady-state, and the circuits with resistor reached steady state after transient. It's obvious that the resistor provides the necessary damping for simulation to converge properly.

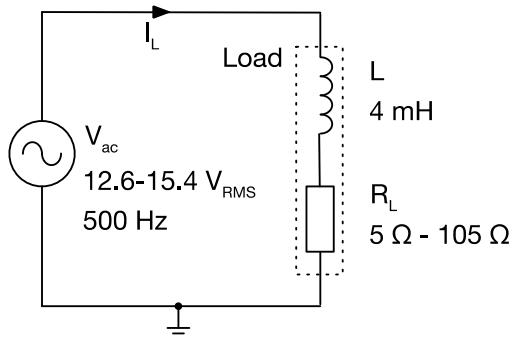
From the above tasks, you should realise that a simulation package may simulate even some of the simplest circuits incorrectly if not set up properly. Therefore, as a design engineer, you must learn to appreciate the limitations of your simulation tools and always use first-principles to understand the behaviour of your circuit (even if it is a very complex circuit) before simulating. Theoretical understanding of the circuit behaviour will help you choose a suitable simulator and set up appropriate simulation conditions.

Part 2: Modelling Your AC Load

Now that you have some familiarity with LTspice, you can use it to help speed up the development of your analogue circuitry required for the energy monitor. The first thing you should aim to do is to model the AC load you will be using in LTspice.

Having a model of the AC load will help you investigate how the load, as well as your analogue circuitry, behaves under all operating conditions.

The AC load we will use in the project compose of a series connected inductor and a variable resistor. The inductor has a nominal inductance of about 4mH. The inductor also has some internal series resistance (ESR), which is about 5Ω . Depending on the position of the viper, the variable resistor can have a nominal resistance between 0Ω and 100Ω . Thus, your nominal (or average) load can be modelled as a series combination of a 4mH inductor (L) and a variable resistance (R_L) between 5Ω and 105Ω , as shown by the circuit:



It should be noted that, as with any practical device, we should expect some variation in the inductance of the inductor as well the resistance range of the variable resistor. This can be due to, for example, manufacturing tolerances. This means that, we can expect each AC load to have a slightly different inductance and resistance range.

As indicated in the circuit above, the voltage of the AC source (V_{ac}) can change between $12.6V_{rms}$ and $15.4V_{rms}$. This comes from the design specifications given in the course outline, which states the supply voltage can change by $\pm 10\%$. Though the supply frequency can vary a little as per the specifications, we are going to ignore this, as the impact is somewhat negligible. So, we are going to assume that the frequency of the AC voltage is 500Hz.

When V_{ac} is a maximum ($15.4V_{rms}$) and R_L is a minimum (5Ω), the AC load will draw maximum VA, and you can calculate this as 17.5VA (taking the inductance as 4mH). When V_{ac} is a minimum ($12.6V_{rms}$) and R_L is a maximum (105Ω), the AC load will draw minimum VA, and you can calculate this as 1.5VA. However, the design specifications given to you state the maximum and minimum VA your design should be capable of measuring is limited to 7.5VA and 2.5VA, respectively. When we test your design, we will make sure to set the load to be within this range by setting R_L and V_{ac} to appropriate values. For example, we may set V_{ac} to $12.6V_{rms}$ and adjust R_L so that the AC load draws 7.5VA. Or we may set V_{ac} to $15.4V_{rms}$ and adjust R_L so that the AC load draws 2.5VA. Or we may set V_{ac} and R_L to any other combination resulting in 2.5VA-7.5VA. Also note, though we assumed the inductive part of the load is 4mH, this value can vary quite a bit due to manufacturing tolerances (as much as 50%). Hence the reason why your design specifications also state a range of power factors you can expect to see.

Q 2.1: As stated in your design specifications, the AC voltage supplied by the power supply across the AC load is going to be between $12.6V_{rms}$ and $15.4V_{rms}$. The AC load is going to be set to draw 2.5VA-7.5VA regardless of the AC voltage

supplied to it. What is the maximum and minimum current you expect to measure? What is the maximum and minimum AC voltage you expect to measure?

Maximum RMS load voltage (given by your design specs): $15.4V$

Minimum RMS load voltage (given by your design specs): $12.6V$

Maximum RMS load current: 595.238mA

Minimum RMS load current: 162.337mA

Q 2.2: Assume that the viper on the variable resistor is set somewhat closer to the lowest resistance position, and therefore R_L is 25Ω . We are also assuming that L is 4mH and V_{ac} is $14V_{rms}$ in this specific example. Note that these numbers are chosen as an example load setting we may test your circuit at. Using circuit theory, determine the RMS load current as well as the real, reactive, and apparent power consumed by the load. Verify answers using the LTspice model provided (labelled Part 2).

Parameter	Theoretical Value	Simulated Value
Load current ($I_{L(rms)}$)	500.3465mA	494.24mA
Real Power (W)	6.2587W	6.1174W
Reactive Power (VAR)	3.1459VAR	3.1023VAR
Apparent Power (VA)	7.0049VA	6.8586VA

Comment (state if results match/differ and why):

Slightly different, I think is due to the discrete timestep of the simulation.

If the peak right between the timestep points, the result can be slightly diff.

Q 2.3: Similarly, calculate and simulate the RMS load current, real, reactive, and apparent power if the viper position is changed to increase R_L to 75Ω (i.e. closer to the highest resistance position)? Again, these numbers are chosen as an example setting we may test your circuit at (note that L is 4mH and V_{ac} is $14V_{rms}$).

Parameter	Theoretical Value	Simulated Value
Load current ($I_{L(rms)}$)	184.1mA	182.35mA
Real Power (W)	2.542W	2.4948W
Reactive Power (VAR)	0.4259VAR	0.4212VAR
Apparent Power (VA)	2.5774VA	2.5301VA

Comments (state if results match/differ and why):

The result match very closely. The minor difference is likely due to the simulation's discrete timestep.

Part 3: Sensing AC Load Current

Your analogue circuitry should sense the current through the AC load as well as the voltage supplied by the AC source, and condition them before feeding them to the microcontroller. Now that you have a model of the load, you can proceed with designing, modelling, simulating, and verifying the analogue circuitry step-by-step. We will start with the current sensor required for your design. As discussed in the lectures, one way to measure current is to use a *shunt resistor*, which produces a voltage drop that is proportional to the current flowing through it. This is because the microcontroller ADC can only read analogue voltages and convert them to digital numbers. Therefore, we must convert the current to a corresponding voltage before it can be fed to an ADC channel through a signal conditioning circuit. Knowing the *shunt resistor* value and the ADC settings, the digital number derived from the ADC can be translated to the actual load current in your C program.

Q 3.1: Determine the value of the shunt resistor suitable to measure the current flowing through the AC load used in your project. Note that you have already calculated the minimum and maximum currents to be measured in Q2.1. Limit the maximum power dissipation in the shunt resistor to 200mW.

Shunt Resistor (R_s): $0.5645\ \Omega$

Q 3.2: Using the simulation model given (labelled Part 3 & 4), verify that you have calculated the correct shunt resistor value in Q3.1. You can validate the design by changing load resistance/supply voltage to vary the VA drawn by the AC load, while keeping it within 7.5VA to 2.5VA and observing peak amplitude of sensed voltage ($V_{is(pk)}$) as well as the power dissipated in the shunt resistor (P_{is}).

Summarise key findings (theoretical vs simulated) in table below. Here we are analysing circuit under 3 possible scenarios including the two extreme cases (i.e. minimum and maximum load current). We are assuming that L is exactly 4mH and R_L is changed to simulate varying load conditions.

Source VA	$V_{ac(rms)}$	R_L	$I_{L(rms)}$	$V_{is(pk)}$ Theo	$V_{is(pk)}$ Sim	P_{is} Theo	P_{is} Sim
7.5VA	12.6V	17.03445Ω	595.238mA	0.4752V	0.4719V	0.12W	0.11937W
7.5VA	15.4V	29.23065Ω	464.013mA	0.3864V	0.382V	0.1322W	0.1292W
2.5VA	15.4V	94.0285Ω	162.337mA	0.1296V	0.1283V	0.01488W	0.014587W

Comments (state if P_{is} is kept below 200mW under all conditions):

Yes, P_{is} is kept or below 200mW.

Q 3.3: Using table below, compare the advantages and disadvantages of using a bigger vs a smaller shunt resistor than the one in Q3.1? Based on this comparison,

what specific value will you use as the shunt resistor in your project? Will you use the value calculated in Q3.1 or a different value? Justify your selection.

Parameter	Bigger R_s Value	Smaller R_s Value
SNR	Lower	Higher
Dissipation (P_{is})	Lower	Higher
Size	Higher	Lower
Cost	Higher	Lower

Note: SNR stands for signal to noise ratio of the measured signal. Use "Low" or "High" to complete the table.

Shunt resistor (R_s) to be used in your design: 0.5Ω

Justification: My calculation showed $R_{s(\text{max})}$ is 0.5645Ω . Because the minimum resistor can be found in lab is 1Ω , so we use two 1Ω resistors to pair in parallel, which is less than 0.5645Ω . Additionally 1Ω is an E12 standard resistor.

Part 4: Sensing AC Source Voltage

The next step is to design your voltage sensing circuit. Since the ADC of your microcontroller can only measure voltages in the range from 0V to 5V, the voltage sensing circuitry should step-down the AC voltage to be within this range. As discussed in lectures, this can be achieved by using a *voltage divider* circuit. The *voltage divider* will translate the high voltage AC input to a lower and safer voltage before feeding it into an ADC channel through its own signal conditioning circuit. Knowing the *voltage divider* resistor values and the ADC settings, the digital number derived from the ADC can be converted to the actual input voltage in your C program. Note that you will be using one ADC channel to measure the current and another to measure the voltage. You will also have two separate signal conditioning circuits (one for signal from current sensor and another for signal from voltage divider) to condition the signals going into these 2 ADC channels.

Q 4.1: Determine the values of the voltage divider resistors suitable for measuring the input voltage supplied by the AC source. Note that you have already calculated the minimum and maximum input voltages to be measured in Q2.1. The voltage divider should step down the maximum load voltage to about $2V_{pk-pk}$ so that it can be processed by the signal conditioning circuit and fed to an ADC channel of the microcontroller. Note: Voltage measurement is taken across R_b (i.e., R_b is between ground and the midpoint of the voltage divider).

R_a / R_b Ratio: 20.7771

Voltage divider resistors (R_a and R_b): $R_a = 1\text{ k}\Omega$ and $R_b = 20\text{ k}\Omega$

Q 4.2: Using the simulation model given (labelled Part 3 & 4), verify that you have calculated the correct resistor values in Q4.1. You can validate the design by changing the load resistance/supply voltage to vary the VA drawn by the AC load, while keeping it within 7.5VA to 2.5VA and observing peak amplitude of sensed voltage ($V_{vs(pk)}$). Also check the total power dissipated in the resistors (P_{vs}).

Summarise the key findings (theoretical vs simulated) in table below. Here we are analysing the circuit under 3 possible scenarios including the two extreme cases (i.e. minimum and maximum input voltage). We are assuming that L is exactly 4mH and R_L is changed to simulate varying load conditions.

Source VA	$V_{ac(rms)}$	R_L	$I_{L(rms)}$	$V_{vs(pk)}$ Theo	$V_{vs(pk)}$ Sim	P_{vs} Theo	P_{vs} Sim
7.5VA	12.6V	17.0344Ω	595.2381mA	0.8485 V	0.8062 V	7.5600mW	7.0090mW
7.5VA	15.4V	29.2306Ω	484.013mA	1.0371 V	0.9863 V	11.2933mW	10.4602mW
2.5VA	15.4V	94.2085Ω	162.3377mA	1.0371 V	0.9859 V	11.2933mW	10.4780mW

Comments (state if $2V_{pk-pk}$ is met and P_{vs} is acceptably low):

Yes, the $V_{vs(pk)}$ in the three scenarios are very close to 1V, and P_{vs} are in milliwatts so I think it's acceptably low.

Q 4.3: Using table below, compare the advantages and disadvantages of using resistors for your voltage divider in the Ohms vs Kilo-Ohms vs Mega-Ohms range. Based on this comparison, what specific values will you use in the voltage divider? Will you use the values calculated in Q4.1 or different values? Justify your selection.

Parameter	Ohms	Kilo-Ohms	Mega-Ohms
SNR	High	High	Low
Dissipation (P_{vs})	High	Medium	Low
Sensitivity	Low	Medium	High

Note: SNR stands for signal to noise ratio of the measured signal. Sensitivity is a measure of how sensitive the design will be to external factors, like humidity, user touch, mechanical forces, etc. Use "Low", "Medium" or "High" to complete the table.

Voltage divider resistors (R_a and R_b) to be used in your design:

$$R_a = 20\text{ k}\Omega \text{ and } R_b = 1\text{ k}\Omega$$

Justification:

The kilo-ohm provides the best balance of performance. It kept power dissipation very low, which provide a high SNR for accurate measurement.

Optional Task: Practical Thoughts

There are many practical aspects that need to be thought off when designing even something simple as a shunt resistor or a voltage divider. You explored some of these practical aspects in Q3.3 and Q4.3 that helped you determine suitable values for R_s , R_a and R_b . However, when it comes to buying resistors with these values we should look at a number of other important parameters in addition to the value of the resistor. This information is usually found in the datasheet. For the work we are doing here, two of the most important parameters we should check are the tolerance and the temperature coefficient of the resistors. Tolerance refer to how much the resistance of a resistor can differ from its specified value. Temperature coefficient refers to how much the resistance will change with temperature. Finding a resistor with a smaller tolerance and a temperature coefficient will help improve the accuracy of our measurement circuitry.

Q O.1: Assume that you have decided to use two 1Ω [ROYALOHM MF006JJ](#) series metal film resistors in parallel as the shunt resistor (R_s). As shown in the datasheet, the resistors have a tolerance of $\pm 5\%$ and a temperature coefficient of $\pm 200\text{ppm}/^\circ\text{C}$ (measured from 20°C). What are the worst-case minimum and maximum values of your current shunt? Assume that your energy monitor will be designed to operate in a 10°C to 40°C ambient temperature. State all other assumptions you made. Also, comment on the expected measurement error based on the minimum and maximum shunt resistance values calculated.

Minimum R_s : (observed when)

Maximum R_s : (observed when)

Assumptions:

Expected Error:

It should be noted that finding an optimum solution can be quite complex as there are many practical factors to consider. There are many tools that help do this and engineers often relies on advanced techniques like Monte Carlo analysis.

To get signed off for the lab:

- Record all the workings in hardcopy or in digital format so we can give you marks for workings if final answers are not correct
- Modify the simulation models provided as per the lab tasks and save
- Commit and push your saved work to the lab repository on GitHub after completing each task and make sure it is up to date
- Complete this document summarising your final answers, commit and push to lab repository on GitHub
- Update logbook indicating yours and teams progress, meeting notes, etc.
- Go to your assigned interview session to check-in with a TA who will check your solutions and ask a few questions
- Report on your weekly progress

To get signed off for the lab:

- Record all the workings in hardcopy or in digital format so we can give you marks for workings if final answers are not correct
- Modify the simulation models provided as per the lab tasks and save
- Commit and push your saved work to the lab repository on GitHub after completing each task and make sure it is up to date
- Complete this document summarising your final answers, commit and push to lab repository on GitHub
- Update logbook indicating yours and teams progress, meeting notes, etc.
- Go to your assigned interview session to check-in with a TA who will check your solutions and ask a few questions
- Report on your weekly progress