Stage II Electrical and Electronic Engineering

EEE2008: PROJECT AND PROFESSIONAL ISSUES

Guidance Notes for the Engineering part: 25-26

Sensor Subsystem (Inductive Guidance)

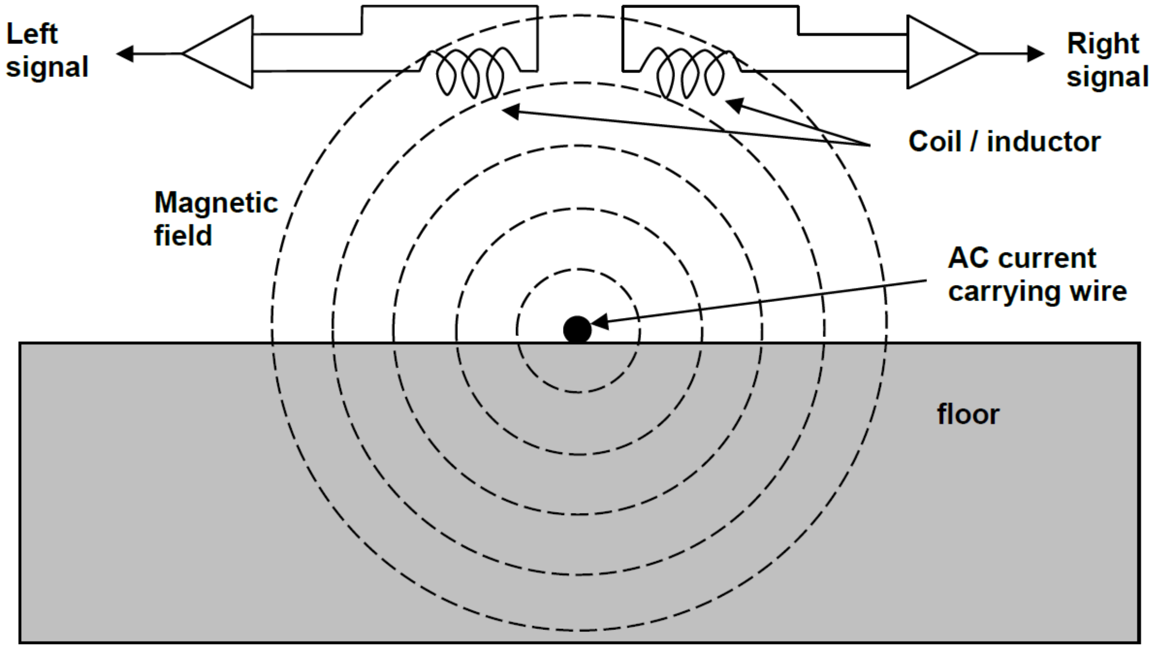

Figure 1: Inductive Signal

The diagram above (Fig. 1) illustrates the principle of inductive or "wire" guidance. A wire carrying an AC current induces a voltage in a nearby coil. If two coils are placed side by side along the same axis, then by comparing the amplitude of the induced voltages you can deduce which way you need to steer a vehicle to follow the track. You need to think carefully about both the location and orientation of the coils with respect to the magnetic field and the vehicle for best results.

NOTE: Induced voltage depends on the magnitude and direction of the magnetic field (i.e., the signal is weak if the field is perpendicular to the coil axis).

The basic theory behind this is electromagnetic induction.

The induced voltage is produced as a product of electromagnetic induction. Electromagnetic induction is the procedure of producing an emf (induced voltage) by exposing a conductor to a varying magnetic field. The induced voltage is described using Faraday's law of induction where the induced voltage of a closed-circuit is described as the rate of change of magnetic flux through that closed circuit.

The induced voltage formula can be written as:

$$e = -N\frac{d\phi}{dt} \tag{1}$$

and:

$$\phi = BA \tag{2}$$

Where:

e is the induced voltage.

N is the total number of turns of the loop.

ϕ is the (Magnetic flux).

B is the magnetic field.

A is the area of the loop.

t is the time.

You also need to be aware of possible interference sources that might superimpose signals on this induced voltage, such as motors. It is recommended that you use ferrite core inductors for your sensors (available in the workshop) since they are sensitive and low cost. A value of 330 µH is recommended.

LC Resonator Circuit

When an inductive coil is placed in parallel with a capacitor (Fig.2), the induced voltage from the varying magnetic field causes an induced current to flow, and this charges and discharges the capacitor at a certain frequency, called the resonant frequency.

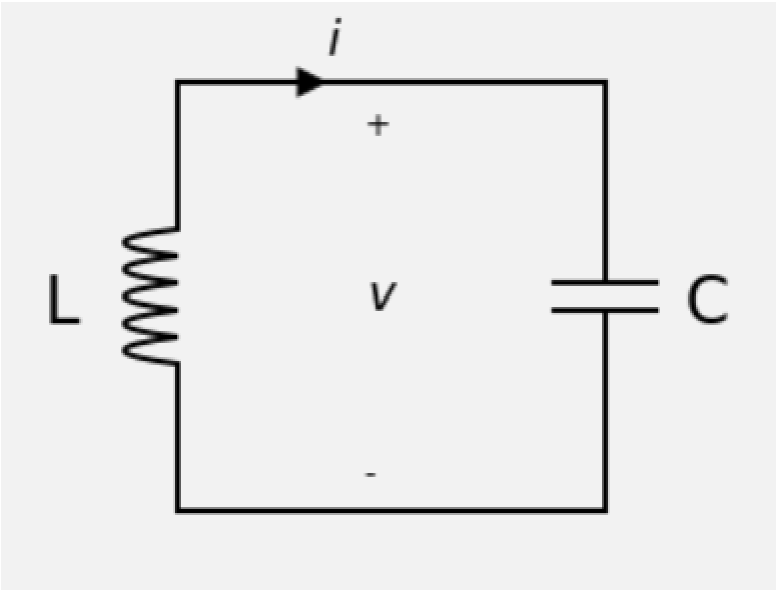

Figure 2: LC Resonator Circuit

The resonant frequency can be calculated from:

$$f = \frac{1}{2\pi\sqrt{LC}} \qquad (3)$$

Where:

L is the inductance of the coil.

C is the capacitor value.

We want this resonant frequency to be approximately the same as the 10 kHz signal in the wire in the floor.

You will need to construct analogue circuits to amplify, filter, and detect the amplitude of the induced signals in each sensor. To get you started, a recommended filter / amplifier circuit is provided below (Fig. 3), you can use as a baseline circuit to simulate and analyse and improve further as you see fit.

You must calculate appropriate values for the parallel resonant circuit (C2/3), the low pass cut-off (C5), high pass cut-off (C1), and the amplifier gain (R7) depending on your design needs. You will determine the gain of your sensors suitably to ensure a balance between high sensitivity (and as such low controllability of the motor) and low sensitivity (and as such low impact on the motor speed). R1 can be set as 1 kOhm.

This circuit should share the same power supply (+V) as the microprocessor (i.e. the PIC).

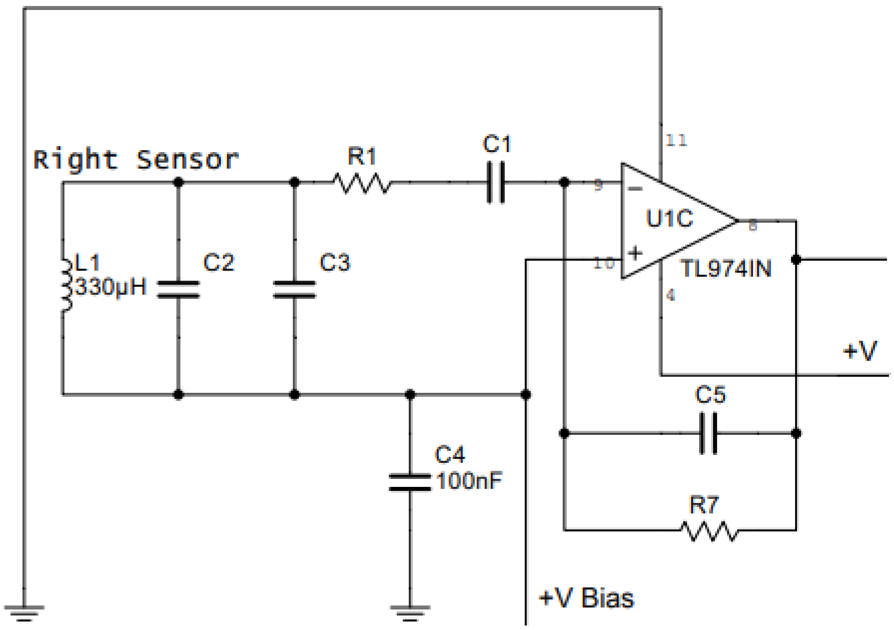

Figure 3: Filter / Amplifier Circuit

A summary of low and high pass filters, and the op-amp circuit are shown below.

Low Pass RC Filter

A low-pass filter is a filter that allows signals to pass which have a frequency lower than a specific cut-off frequency, and it attenuates (reduces) signals with frequencies higher than the cut-off frequency. The frequency response of the filter depends on many different options including the component values, and the order of the filter (first order, second order, etc…), with higher orders giving a better response but with a more complicated design.

For the project, a simple first order filter consisting of a resistor and capacitor can be used as show in Fig. 4 below:

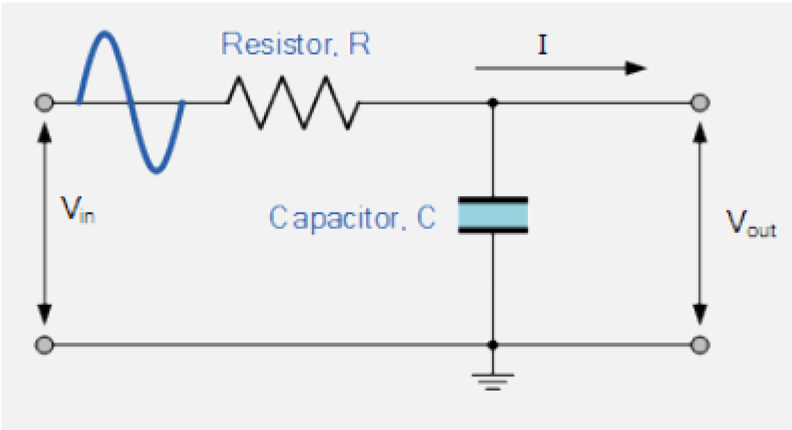

Figure 4: Low Pass RC Filter

The cut-off frequency for this filter can be calculated from:

$$f = \frac{1}{2\pi RC} \tag{4}$$

Where:

R is the resistance.

C is the capacitor value.

The standard response of the filter is shown below:

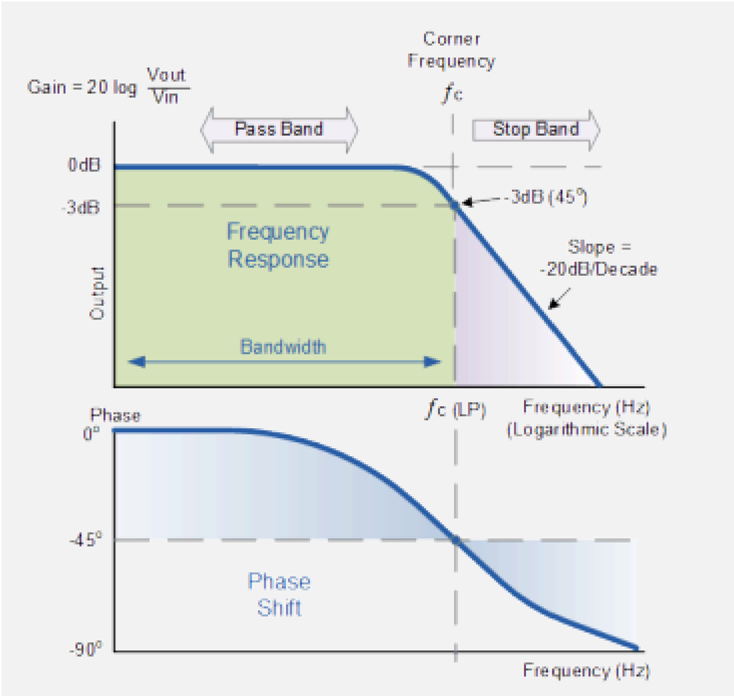

Figure 5: Low Pass Filter Response

High Pass RC Filter

A high-pass filter is the opposite to the low pass filter. This filter allows signals to pass which have a frequency higher than a specific cut-off frequency, and it attenuates (reduces) signals with frequencies lower than the cut-off frequency. As with the low-pass, the frequency response of the filter depends on many different options including the component values, and the order of the filter (first order, second order, etc…), with higher orders giving a better response but with a more complicated design.

For the project, a simple first order filter consisting of a resistor and capacitor can be used as show in Fig. 6 below:

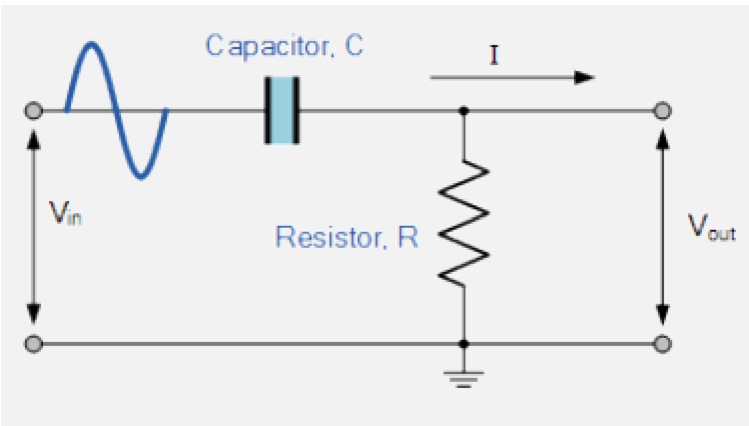

Figure 6: High Pass RC Filter

As with the low-pass, the cut-off frequency for this filter can be calculated from:

$$f = \frac{1}{2\pi RC} \tag{5}$$

Where:

R is the resistance.

C is the capacitor value.

The standard response of the filter is shown below:

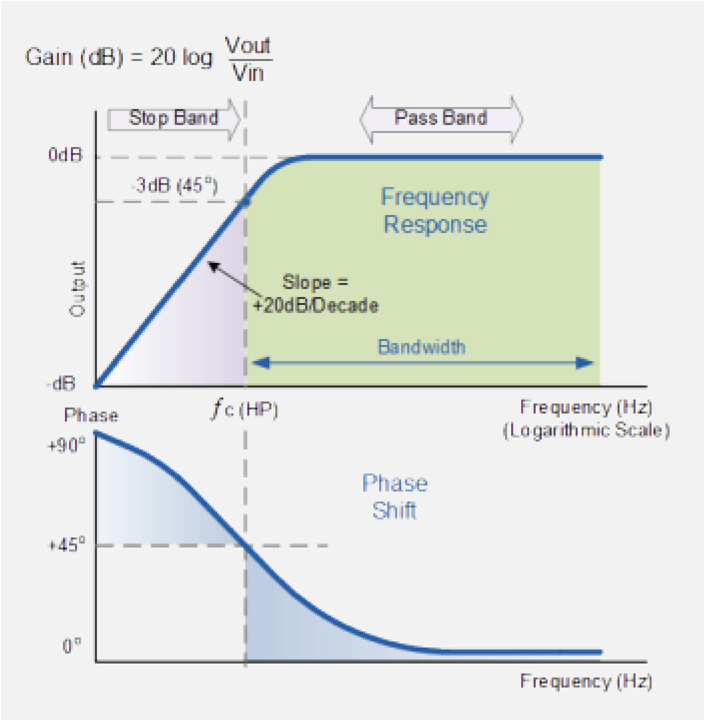

Figure 7: High Pass Filter Response

Inverting Amplifier

The inverting amplifier configuration (Fig. 8) is one of the most widely used analogue circuits, and the input stage of every op-amp is a differential amplifier. This circuit amplifies the difference between the two input signals.

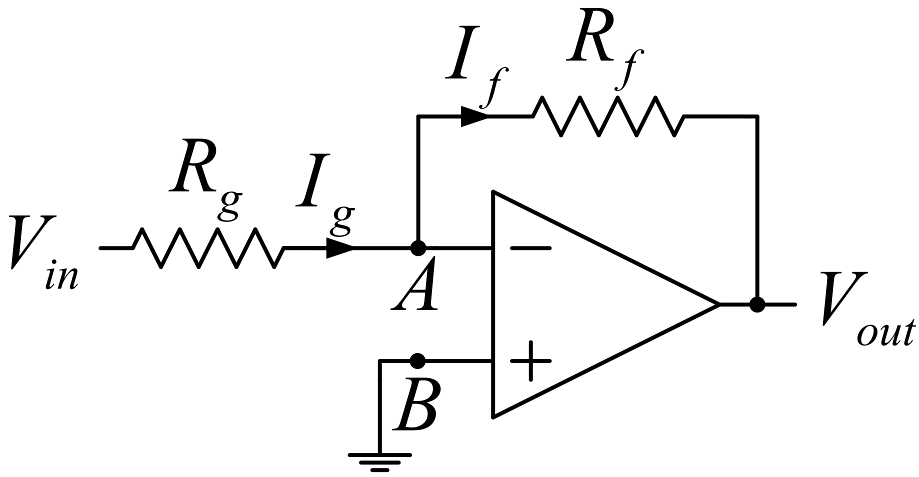

Figure 8: Inverting Amplifier

Op-amp terminals are considered equivalent to virtual grounds, and the circuit shown can be analysed to find the relationship between V_{out} and V_{in}.

$$\therefore A_{CL} = \frac{V_{out}}{V_{in}} = -\frac{R_f}{R_g} \tag{5}$$

As the op-amp is only being power from a single supply, a DC offset (+V bias) has to be added to positive input. This can be achieved using a resistive potential divider circuit – research the equation for this and think of a suitable voltage that can be applied to the positive input.

After designing the op amp circuits, you need to think about how you turn a sine wave output into a varying DC voltage (representing magnetic field strength) which can be fed into the control subsystem. A precision rectifier circuit is recommended. Note: the circuit in Fig. 3 is designed to operate from a single DC supply. Hence a resistor divider is used to set the DC operating point of the amplifier.

Precision Half Wave Rectifier (AC-DC)

The diodes in this circuit (Fig. 9) allow for precision half wave rectification, where the input will be a sine wave and the output will be half-sine pulses with positive values only as shown in Fig.10

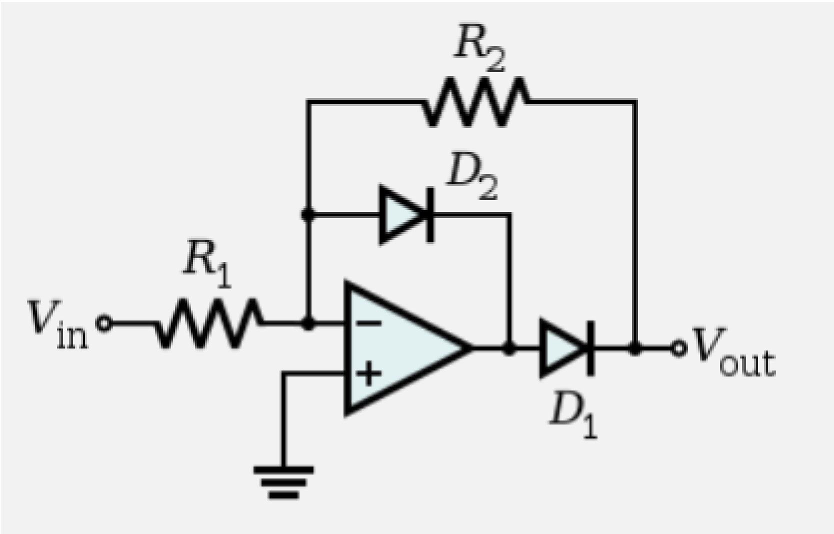

Figure 9: Precision Half Wave Rectifier

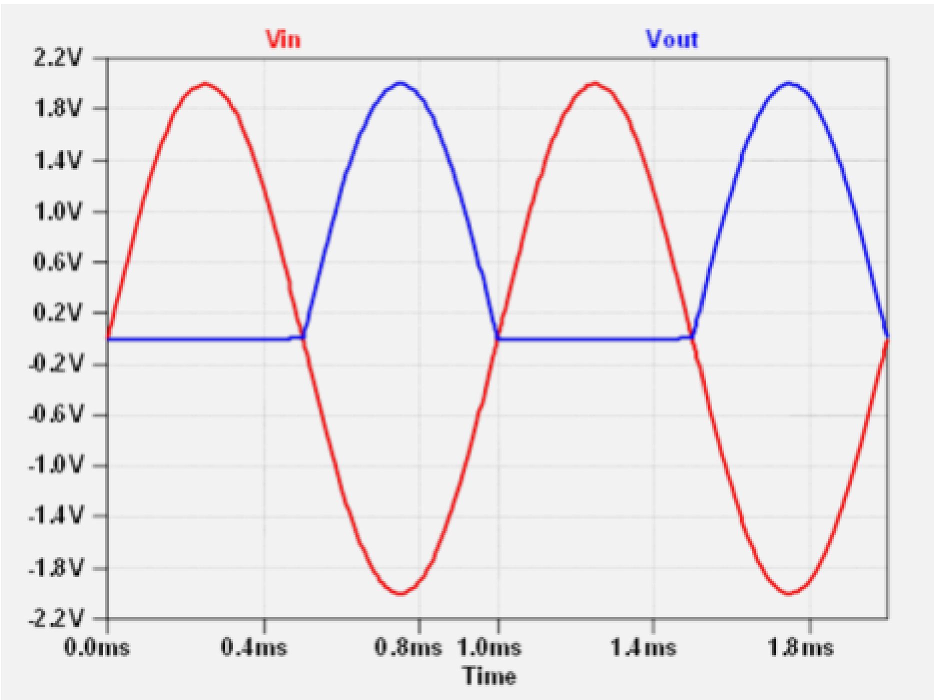

Figure 10: Rectified Signal

The circuit operates as follows. For a positive input voltage, diode D2 conducts and closes the negative feedback loop of the op-amp circuit. A virtual ground appears at the inverting input, and the op-amp output is clamped at a voltage equal to one diode drop below ground. The negative voltage will keep diode D1 off and no current will flow in the loop containing the feedback resistor R2. Therefore, the output voltage of the circuit will be zero. As the output voltage goes negative, the inverting terminal voltage will also go negative, causing the op-amp output voltage to go positive. This mean D2 will be reverse biased and turn off. The diode D1 will conduct through R2 creating a negative feedback loop around the op-amp and a virtual ground will appear at the inverting input. Therefore, the current through R2 will be equal to the current through R1. When R1=R2 the relationship between the input and output voltages is:

$$V_{out} = -V_{in}, \text{for } V_{in} \leq 0 \tag{6}$$

The values of R1 and R2 can alter the slope of the output voltage.

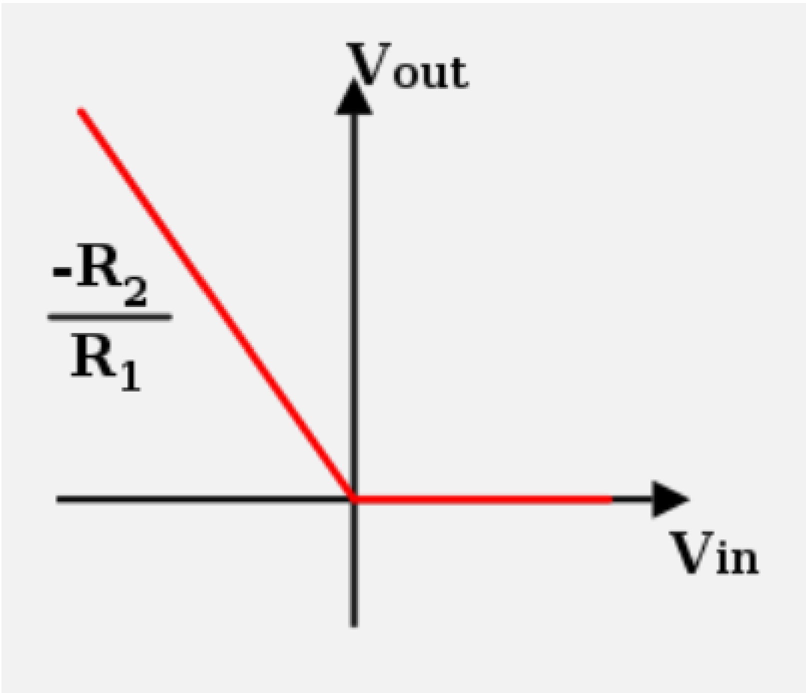

Figure 11: Input / Output voltage relationship

An example schematic of the full sensor system for both sides is shown in Fig. 12 below.

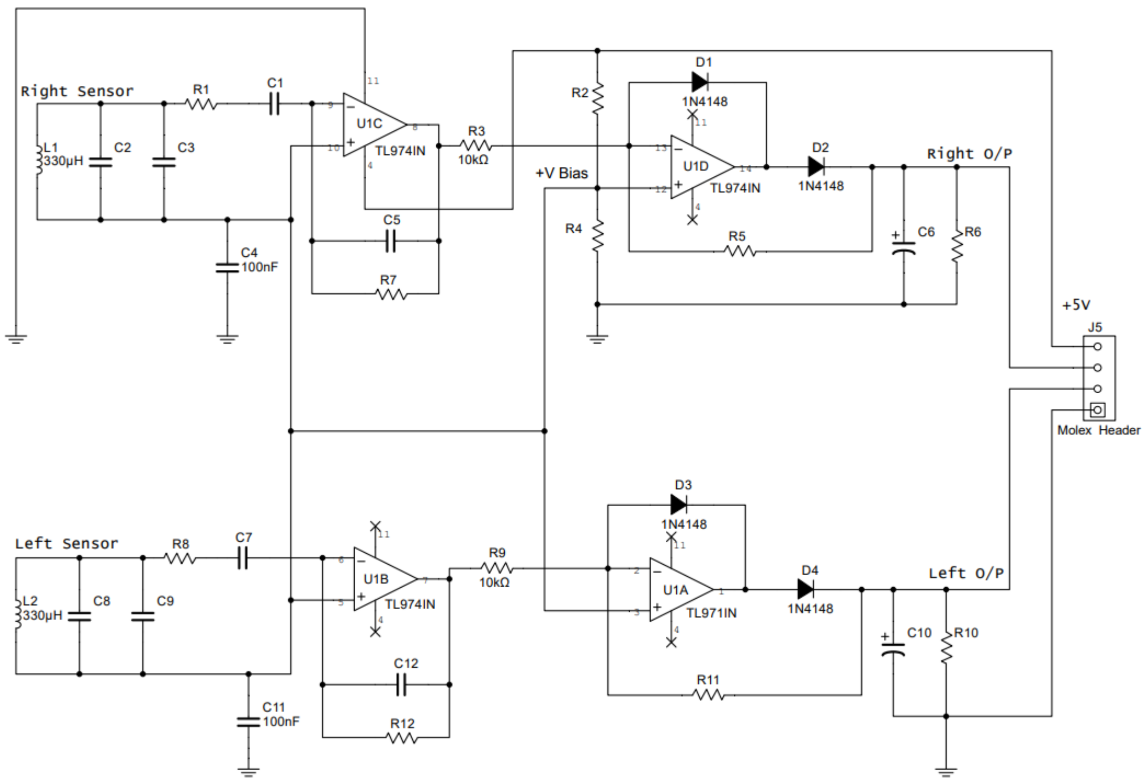

Figure 12: Example schematic of two sensor circuits – left and right – creating a differential DC voltage for position determination of the vehicle.

For the sensor subsystem it is suggested to carry out the following design and analysis tasks:

Sensor Subsystems To-do:

1) Study the different components of the circuits (using one sensor circuit first): which components contribute to what kind of behaviour for the circuit behaviour, e.g., low pass filter, high pass filter, amplifier, smoothing circuit, rectifier.
2) Build the circuit (first on LTSpice and then on Project board) with the components as suggested (replace the LC system by a small 20mVp-p AC source, sinusoidal and 10kHz frequency). If a component is not available, use the circuit components with similar electrical characteristics.
3) Study the following:
    a) transient behaviour to reason for component functionality.
    b) Steady-state behaviour for investigating the overall circuit behaviour.
    c) AC sweep of the input source to reason for performance of your filters as well as conditioning filters at the end. Also create a model for relating your DC outputs based on the AC inputs. This model will be crucial for your integration.
    d) DC sweep to investigate minimum DC operating points (especially when the battery power is low).
    e) Fault diagnostics capability with simulated open and short circuit faults at different points in the circuit.
    f) Can you measure the power consumption of the overall circuit?
4) Can you improve this circuit further?