**Class #25: Experiment**
**Transient Circuit Design**

**Purpose:** In this experiment we will apply our understanding of transient behavior in RC, RL and RLC to design circuits with specific transient characteristics.

Background: Before doing this experiment, students should be able to
- Review online background materials.
- Build and operate simple circuits on a Protoboard.
- Measure the voltages and determine the currents using a math channel in simple Protoboard circuits using Analog Discovery.
- Analyze the transient properties of simple circuits consisting of combinations of resistors, capacitors and/or 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
- Design a first order circuit with a specific time constant, $\tau$.
- Design a second order underdamped circuit with a specific oscillation frequency, $\beta$.

Resources Required:
- LTspice
- Analog Discovery and Parts Kit

Helpful links for this experiment can be found on the course website under Class #25.

## Background

In the last laboratory, we implemented a voltage follower circuit. In general, we can use the voltage follower for isolation, such that different parts of the circuit do not affect each other. There are several types of isolation circuits, many of which use amplifier circuits. Another example, in power electronics, a transformer is often used for isolation.

In Figure 1, we consider a multi-stage circuit, with an input stage, an isolation circuit, and an output stage. The isolation circuit means that we can analyze the performance of the input stage and the output stage separately, making the process much easier. In this laboratory, we want to build circuits with specific properties regarding timing, oscillation, and attenuation. If we connected the output stage directly to the input stage, those properties would change and we would need to redesign our circuit.
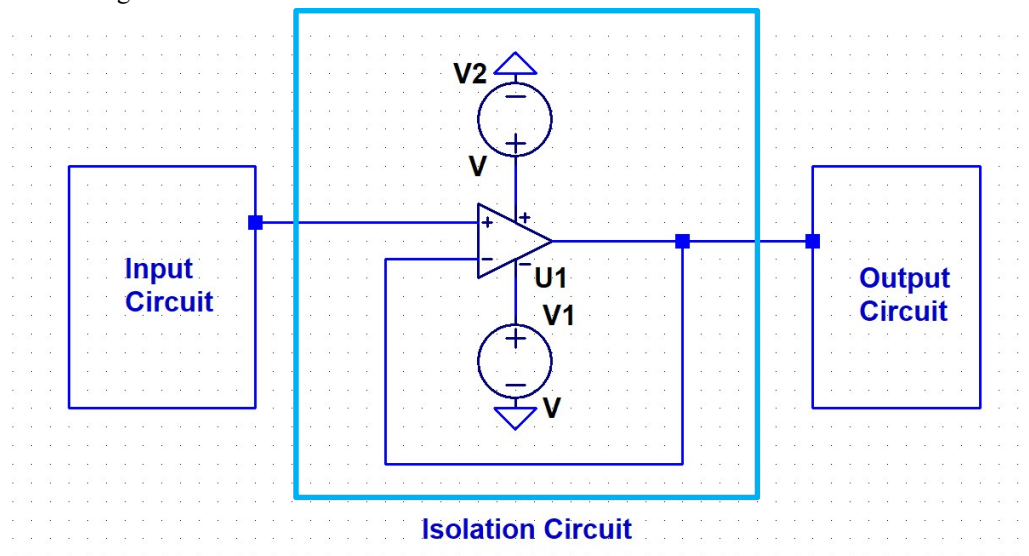


**Figure 1: Example of a Multi-Stage Circuit with Isolation**

From the last laboratory, the process for building the isolation circuit is
- Connect the function generator (Wavegen yellow wire) to positive input pin 3 of the chip (labelled in Figure C-2).
- Connect the output pin 6 to the negative input pin 2 of the chip. That is the feedback wire from the output to the input, shown in Figure C-1.
- Connect the Analog Discovery red wire (positive voltage supply) to pin 7 of the chip
- Connect the Analog Discovery white wire (negative voltage supply) to pin 4 of the chip
- Connect a 1MegΩ resistor from the output pin 6 to ground. (This step is not really necessary since the Discovery Board acts like a 1MegΩ resistor.)
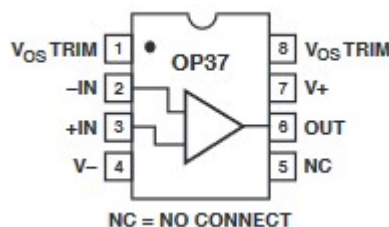


**Figure 2. OP37 Chip Layout**

## LTspice and opamps

The OP37 op-amp is a component in the LTspice library. You do need to pay attention to the power supply (voltages) connections, in that they are 'opposite' to that shown on the schematic of Figure 1. The supplies are circled in Figure 3, with the positive supply needing 5V and the negative supply needing -5V. The positive input is the connected to the voltage you are 'measuring' on the input circuit and the negative input is connected to the output, like the circuit in Figure 1.
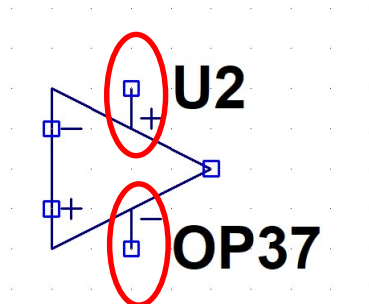


**Figure 3. OP37 Chip Layout**

## Part A – First Order Circuits

Design a circuit such that an LED stays on for ~3 seconds after power is turned off (your time constant will need to be longer than 3 seconds). In this case, the output circuit is an LED and resistor in series, similar to the LED circuits we have built previously. When you consider your design, remember that LEDs need a minimum voltage to be on and that minimum voltage depends on the color of the LED. You can search for those values online (LED on voltage). You will want the maximum voltage of your source to much larger (more than twice as large) as the LED on voltage. Your control circuit should be a first order circuit, an RC or RL circuit. Turning power off can be simulated by disconnecting the source wire, similar to the experiment in Laboratory 15.

In your report, include a schematic of the circuit and oscilloscope plots of both the capacitor voltage and the output voltage of the amplifier. Try to take pictures of the LED, capturing the dimming light over the ~3 seconds (3 or 4 pictures is enough). Simulate the circuit in LTspice, replacing the LED with a voltage source with voltage equal to the LED on voltage. Include plots of the simulation capacitor voltage and amplifier output voltage in your report.

Explain/document your process for designing the circuit and indicate any relevant parameters for the circuit you designed.

If you doubled the resistance in your circuit, explain how the performance should change. Double the resistance in your circuit and check to see if your expectations are correct.

## Part B – Second Order Circuits

Design a circuit such that the oscillation frequency is ~4kHz (4000 Hz). Your circuit should be an underdamped RLC series circuit. The source in the circuit is a square wave.

In your report, include a schematic of the circuit and Oscilloscope plots of the output voltage of the amplifier. Simulate the circuit in LTspice. Include a plot of the simulation amplifier output voltage in your report.

Explain/document your process for designing the circuit and indicate any relevant parameters for the circuit you designed.

Experiment with changing the resistance in the circuit and note how it affects the attenuation of the ringing (how the amplitude decreases with time).