The 555 Timer

Prelab Exercises

1. This lab is all about a device known as a 555 timer. Questions:
   - In simple terms, what is a 555 timer? (Hint: Ask the Internet)
   - The model we will be using in this lab is a LM555CN made by Fairchild Semiconductor, how much does one cost when ordered from Digikey (ignore shipping costs)?
   - Using the datasheet available on Digikey, what are the pin outputs on the LM555CN? Draw the IC and label both the pin number and name for each pin.

   For those wanting to see how a 555 timer works, here is a great animation of the 555 timer operation. [Complements of Rensselaer Polytechnic Institute www.rpi.edu]

2. Using the general layout in Figure 1, design a circuit which will produce a square wave signal that has a frequency range from 1kHz to 10kHz, give or take 10% on each end.

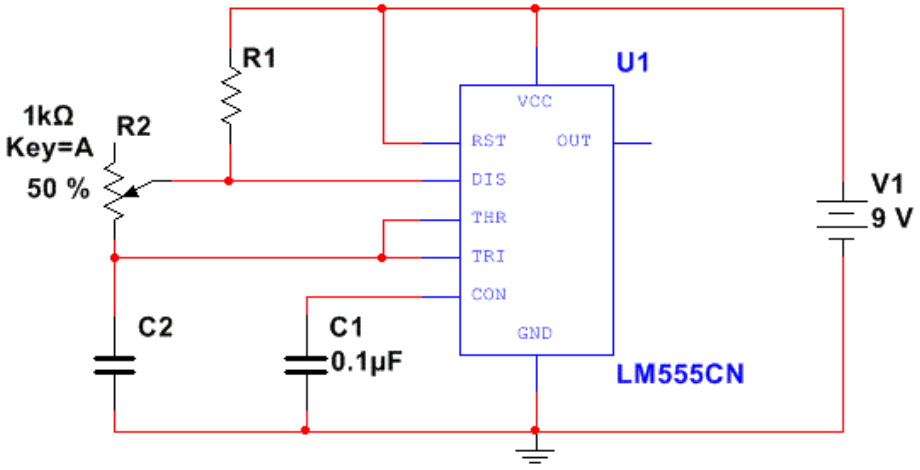

Figure 1: 555 timer circuit

Design Parameters

When determining the values for for R_1 and C_2, take note that:

- R_2 is a 1kΩ potentiometer, a variable resistor varying from 0Ω to 1000Ω.
- C_1 is a .1µF capacitor.
- Choose resistor values between 10Ω and 100kΩ, and capacitor values between .1nF and 1000µF.
- The TTL output signal has a frequency equal to

$$f = \frac{1.44}{(R_1 + 2R_2)C_2}\text{Hz} \tag{1}$$

*H*int: There are two ways to solve for both the unknowns in this equation, the brute force way and the clever way. Try to think through it and if you fail, simply plug and chug with a calculator.

Question: Once you have determined the values of R_1 and C_2, what is the output frequency for each case where $R_2 = 0\Omega$, .3kΩ, .6kΩ, and 1kΩ?

Your pre-lab should include a completed schematic with all circuit elements labeled (including pin numbers on ICs) along with the math you used to determine them.

Lab Exercises

For this lab, you will use Multisim to simulate your design.

1. Using the Multisim schematic capture program, enter the circuit that you designed in the pre-lab. Below are some items to help you define certain elements of your circuit.
   - The 555 timer chip is in the Mixed group, *TIMER family*. Choose LM555CN (be sure you get the exact match).
   - For the potentiometer (R_2) use the *Potentiometer family* and pick the proper value from the list.
2. Simulating your design: There are several ways to capture the output of the 555 timer in Multisim, and in this lab you will be using two: the transient analysis feature and one of oscilloscope (or o-scope). Start with transient analysis.
   - We will want to see 2 things on the simulation output, the voltage across C_2 and the voltage of the output pin. Since the transient analysis only measures net voltages, you will have to create a net to measure. Connect a 1kΩ resistor from the 555 timer output pin to ground.
   - Set R_2 to the minimum value and perform the transient analysis. Note that if you set it to 0Ω, that is basically shorting the DIS and TRI pins together, casuing the output to stop oscillating. You can get around this by only adjusting the potentiometer value down to 5%.
   - Run the simulation and measure one full period of the 555 timer output and then calculate the frequency. Be sure you do not inculde the first pulses, as the output can take a few cycles to stabilize.
   - Compare the measured frequency to what you calculated in the pre-lab. Calculate a percent difference (but you already knew that's what was meant by "compare," right?)
   - Repeat the above steps with $R_2 = .3k\Omega$, $R_2 = .6k\Omega$ and $R_2 = 1k\Omega$.

   Now repeat the above steps using one of the o-scopes. Choose an o-scope (I like the Tektronix model, but use whichever you like) and place it on your design. Wire the channel 1 input to the output pin on the 555 timer and the channel 2 input to the capacitor. Click "Run" (the little green play button) and view the o-scope by double clicking on it.

   Your lab needs to include a drawing of everything you complete in the lab including your circuit, the transient analysis output and the o-scope screen. Label all values and include the units.