ES 330: Electronics II

Laboratory Experiment No. 4

Title: Special OpAmp Circuits

Objective: The main objectives of this lab session will be

1) Study Precision Rectifier
2) Study Schmitt trigger and Multivibrator

N. B. This lab experiment will be a part of your **project 2**. SO YOU MUST SAVE YOUR DESIGN.

Introduction:

Opamp is widely used for various types of circuits and system development. Other than signal amplification (linear circuit), opamp could be used to build some non-linear circuits. In this lab we will study the operation of three opamp based special circuits. These are

1. Precision rectifier
2. Schmitt trigger
3. Multivibrator

Precision Rectifier:

Precision rectifier is used to rectify a signal without a loss of signal amplitude. In a simple diode based rectifier circuit, signal amplitude must be higher than the turn on voltage of the diode. This is almost 0.7 V for half wave rectifier and 1.4 V for a diode based bridge rectifier. Using the property of opamp it is possible to develop a precision rectifier to rectify any ac signal without any signal attenuation.

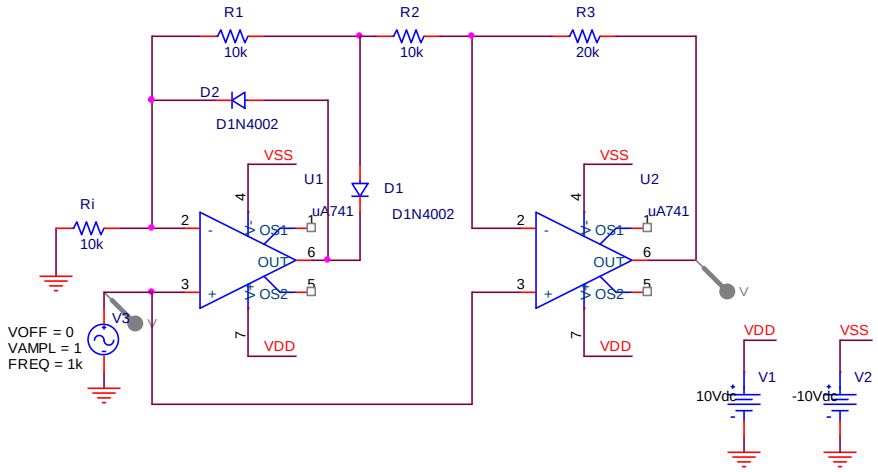

Fig 1: Schematic of a precision rectifier

Fig 2: Simulation output of precision rectifier

1. Draw the circuit as shown in Fig 1 using PSPICE
2. Do the transient simulation and you will get an output similar to as shown Fig 2.
3. Now change R3 to 10K and 30K and do the simulation again. Save the results and make your comments.
4. Set R3 value to 20K. Vary Ri to 5k, 10k, 15k and 20k. With each value of Ri do the same transient simulation. Save your results and make comments.
5. Now build the circuit in a protoboard. See the output on the oscilloscope.
6. Change R3 to 10K and 30K and observe the outputs. Compare your results with your simulation results.
7. Set R3 value to 20K. Vary Ri to 5k, 10k, 15k and 20k. With each value of Ri observe the outputs. Compare your measurement results with your simulation ones.
8. Now replace the sinusoidal source with a 1V dc source. Measure the voltage at each and every nodes of your circuit.
9. Change the value of dc source in step 8 to -1V and do the same. Based on step 8 and 9, try to explain the operation of the circuit.

Schmitt Trigger:

Schmitt trigger is generally used as a substitute of comparator. It uses hysteresis property to eliminate unwanted triggering from the input signal. Thus Schmitt trigger can help to reduce noise in the output signal. In this session we will study a simple opamp based Schmitt trigger.

1. Draw the circuit in PSPICE as shown in Fig 3.
2. Connect a VPWL source (10 KHz, 10 Vp-p) at the inverting input of the opamp.
3. DO the transient simulation and observe the voltages at different points.
4. Vary the value of R1 from 5K to 15K with 5K step. In each case observe the output.
5. Now build the circuit on the board. Apply a triangular pulse of 10KHz at the inverting input. Observe the output on the oscilloscope.
6. Vary the value of R1 as stated in step 4 and observe the output V_{out} and V_{R1} on the scope.

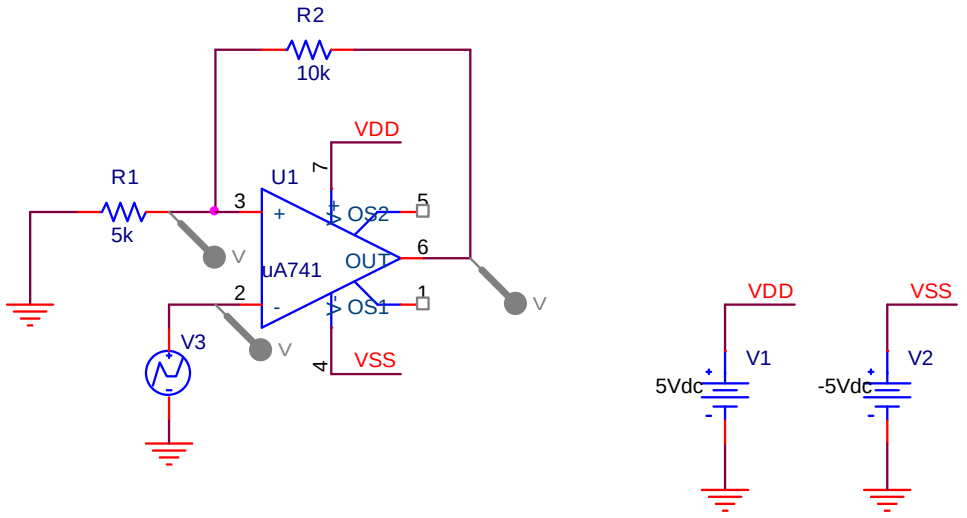

Fig 3: Schematic of Schmitt trigger

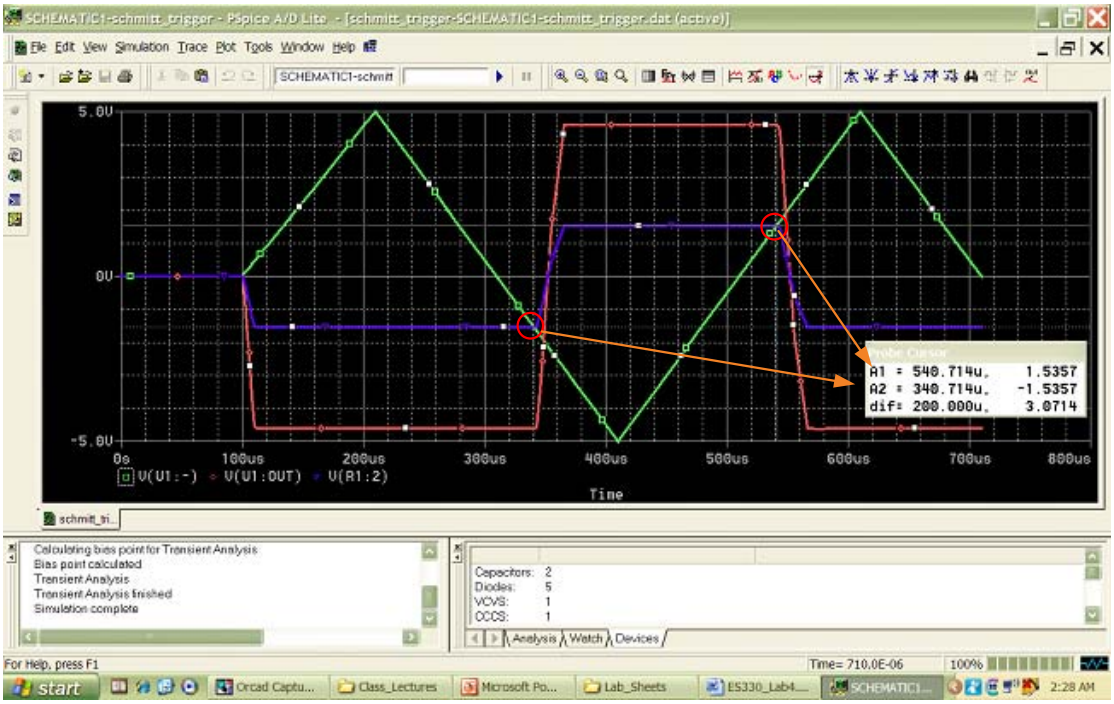

Fig 4: simulation output of Schmitt trigger

Multivibrator:

Multivibrator is used to generate an oscillating signal. It has lots of applications such as clock generation in digital circuit, carrier generation in communication system, variable frequency signal generation, etc. In this section a simple multivibrator will be studied.

1. Connect a R-C network with the Schmitt trigger and the resultant circuit will be as shown in fig 5.
2. Simulate the circuit in PSPICE. Measure the frequency of the output signal.

3. Vary R1 from 1k to 10k in step of 1k and measure the frequency (using cursors) of the output signal. Make a table of your simulation results.
4. Build the circuit on the protoboard.
5. Follow step 3 and make a plot of resistance vs output frequency.

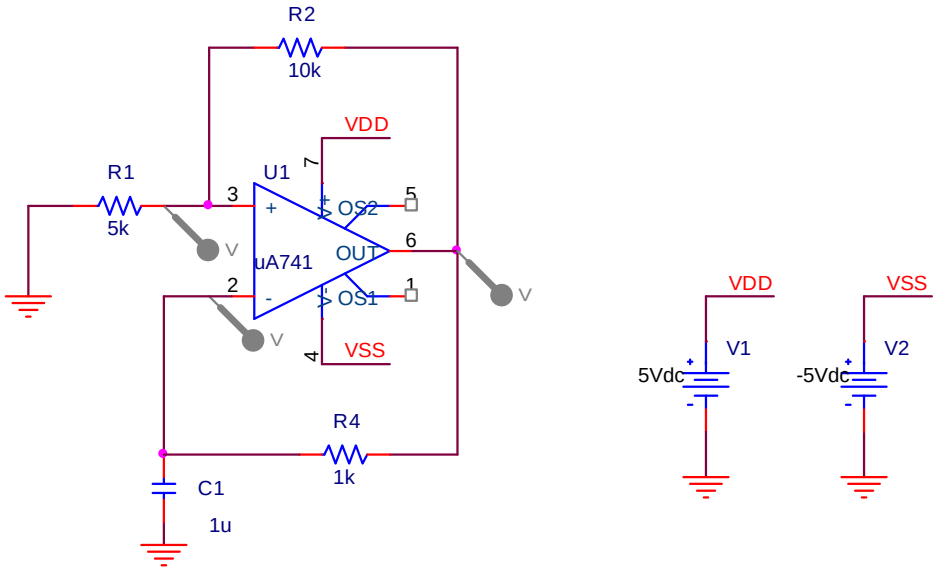

Fig 5: schematic of multivibrator

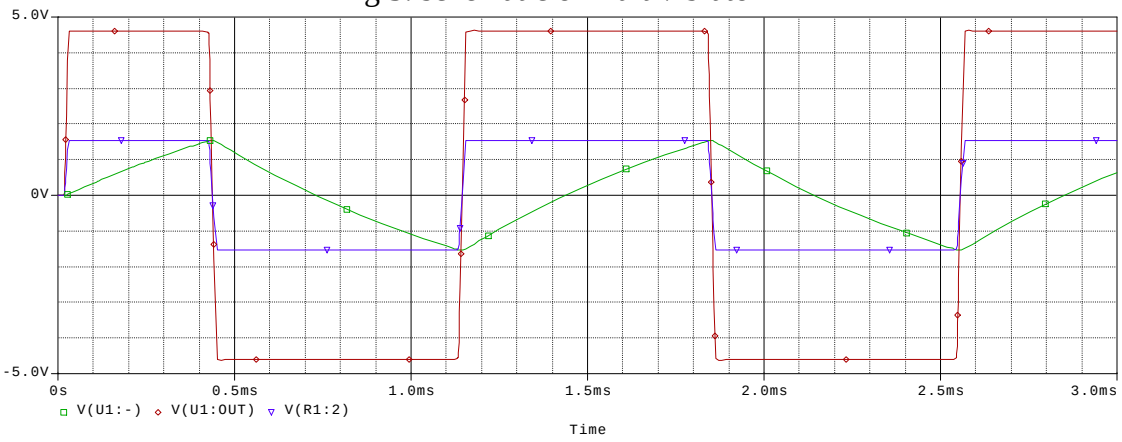

Fig 6: simulation output of multivibrator