**Experiment #10-Prelab ENEE2103**

***The Operational Amplifier***

***Prelab:***

You have to use PSPICE simulation of the circuits to be tested experimentally in the experiment.

***PROCEDURE:***

***I. Adding Application***

1. Enter the following Circuit in Pspice and perform bias point analysis for the the values of V1 and V2 shown in Table 10.1, fill the results in the table



|  |  |  |  |
| --- | --- | --- | --- |
| **Input voltage** | | **Output voltage** | |
| **V1** | **V2** | **VO** | **Calculated voltage** |
| **0.5** | **2** |  |  |
|  |  |  |  |
| **0.3** | **4** |  |  |
|  |  |  |  |
|  |  |  |  |
| **-1.5** | **6** |  |  |

Table 10.1

1. Write The expression relating Vo to V1 and V2 : i.e. Vo =XV1 + YV2 (Find X, Y )

***II. Voltage Follower Application***

1. Connect Circuit of Fig.(10-2). In Pspice



1. Perform DC sweep For DC voltage source Vi
2. Plot Vo and observe the relationship between Vo and Vi
3. Plot Io and observe its behavior?
4. Repeat 4 , but with the 220 ohm resistor replaced by 10 k

***III. Comparator Application***

1. Set up the circuit of Fig.(10-3).



1. Use 1 kHz triangular input signal 🡸 in Pspice you must use Vpulse to create a triangular wave form: V1=-1V, V2=1V, tr=0.5m, tf=0.5m,PW=10n,Period=1m
2. Perform transient analysis with suitable end time and withV1=0V, V1=0.98V, V1=-0.98 V

***IV. Integrator and Differentiator***

1. Set up the circuit of Fig.(10-4). 
2. Put an input signal with 10VP-P and a frequency of 100Hz ,and plot the voltage Vo and compare it to Vi.
3. Set up the circuit of Fig. (10-5).



1. Repeat steps 2 above

***V. To investigate the effect of adding hysteresis:***

1. Connect the Schmitt trigger circuit shown in Fig.(10-6).



1. Put Vi(t) = 15Vp-p sine wave of frequency 1 kHz.
2. Plot the Vo and Vi on same plot.
3. Indicate the levels of Vi(t) where Vo(t) changes its level from low to high and high to low